MIKE 21 Toolbox

User Guide

PLEASE NOTE

COPYRIGHT

This document refers to proprietary computer software which is protected by copyright. All rights are reserved. Copying or other reproduction of this manual or the related programs is prohibited without prior written consent of DHI. For details please refer to your 'DHI Software Licence Agreement'.

LIMITED LIABILITY

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

'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 OUT OF THE USE OF OR THE INABILITY TO USE THIS DHI SOFTWARE PRODUCT, 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 COUNTRIES OR STATES 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 TO YOU. BY YOUR OPENING OF THIS SEALED PACKAGE OR INSTALLING OR USING THE SOFTWARE, YOU HAVE ACCEPTED THAT THE ABOVE LIMITATIONS OR THE MAXIMUM LEGALLY APPLICABLE SUBSET OF THESE LIMITATIONS APPLY TO YOUR PURCHASE OF THIS SOFTWARE.'

CONTENTS

M21 TOOLS: ENVIRONMENTAL

The ENVIRONMENTEL Tools in the M21 Toolbox include the facilities:

- AD transfer boundary data

1 AD transfer boundary data

Environmental Transfer Boundary is a MIKE 21 Flow Model AD tool for preparation of AD transfer boundaries. Boundary data are extracted from results of a previous AD simulation. A MIKE 21 AD simulation output data file (a matrix time series with fields of component concentrations) is used to generate the new boundary data (a line time series data file of component concentrations along the boundary line).

For all specifications, with the exception of I/O file names, default values are supplied. You can change these specifications by modifying the corresponding dialog entries.

1.1 Select Input File

On this page you specify the AD simulation output file to be used as the basis for transfer boundary generation.

1.2 Boundary Position and Length

Here you specify:

- the origin, and length, of the boundary line for which data should be extracted. The origin must be specified in "old grid" coordinates.

- the angle - or line orientation - clockwise from the boundary line to the south boundary of the old model

- the desired grid spacing along and across the boundary line

- whether the boundary in the new model is a West, North, East or South boundary (this affects the way the boundary data is calculated

The new boundary line has to lie within the old model area, not on a boundary.

1.2.1 Detailed description

You specify the line in the old model from where values for boundary data generation are to be taken. This is done by giving the start coordinate, the line orientation, number of points along line and the grid spacing in the new model, both along line and perpendicular to line (see Figure 1.1).

It is necessary to specify whether the boundary line is to be model south, model north, model east or model west in the new model, as this affects the boundary data calculations.

You can only prepare boundary data for one boundary per run.
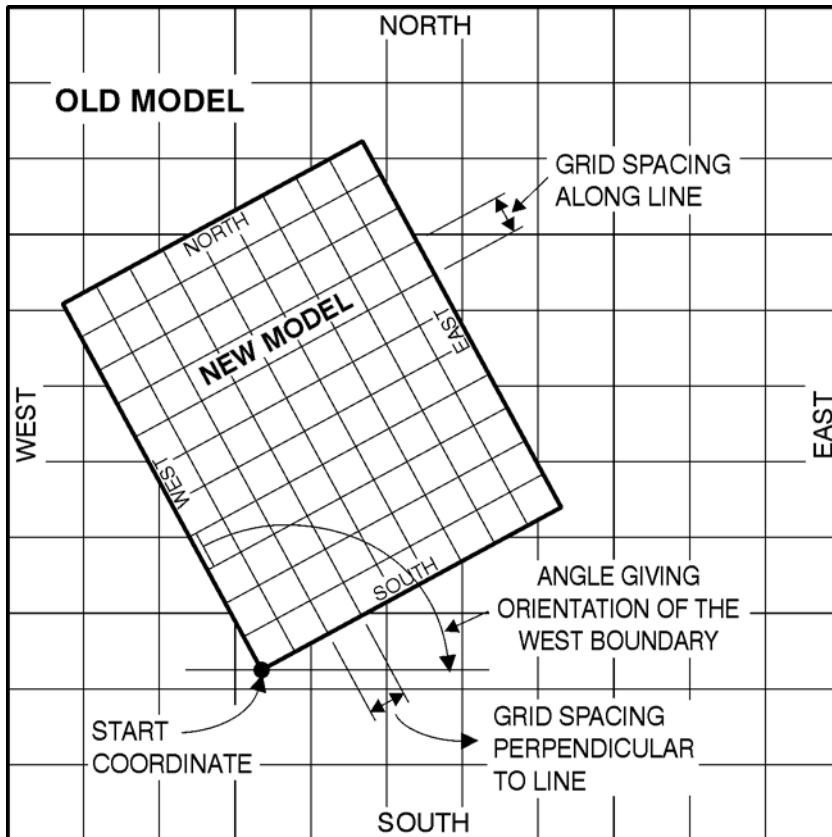
Figure 1.1   Example of a west boundary line for calculation of AD transfer boundary data

1.3   Subseries Selection

Here you may specify the time steps you want to process

1.4   AD Transfer Data Output

Here you specify a name for the transfer boundary output data file. You may also specify a title (data description).

1.5   Overview

The overview page presents you with a summary of all the input parameters you have specified. You should inspect the parameters to check they are correct, - and if not go back and change them.

From the page you can either *Finish* the set-up, which will transfer your set-up definition to the current toolbox file, or execute the set-up.

1.6   Remarks and Hints

The line must be fully inside the old model, and not along a boundary. At least one grid point in all directions around the line should be available in the old model.

See also the description of HD transfer boundary data.

M21 TOOLS: HYDRODYNAMICS

The HYDRODYNAMIC Tools in the M21 Toolbox include the facilities:

- Apparent Bed Roughness
- Border Adjustments
- Discharge Calculations
- MIKE 21 Hotstart File Generator
- HD Transfer Boundary Data

2 Apparent Bed Roughness

Welcome to the MIKE 21 Tool for generation of bed resistance and bed shear stress maps.

The tool uses standard output files from MIKE 21 HD and MIKE 21 PMS as input and calculates either a bed resistance (Manning) map for later use in a MIKE 21 HD simulation or a bed shear stress map, e.g. for the design of a scour protection.

In both cases, the apparent increase in bed resistance felt by the current due to the presence of the waves is taken into account.

2.1 Simulation Type

In this page you have to define whether you want to calculate a bed resistance (Manning) map for later use as input in a MIKE 21 HD simulation, or whether you want to obtain a map showing the maximum dimensionless bed shear stress (Shields parameter) over the wave period at every node of your simulation grid.

In both cases, the parameters are calculated by application of Fredsøe's (1984) model for the wave boundary layer in combined wave-current motion.

You have two alternatives to generate the Manning map:

- To perform a direct calculation, i.e. to apply the wave boundary layer model at every node of the calculation grid
- To use a parameterised version of the model

Use of the parameterised option will speed up the simulation. In either case, only the value of the Manning number M will be written to the output file.

Direct application of the wave boundary layer model takes place when calculating the bed shear stress map; in this case values of the Shields parameter, the Manning number and the apparent roughness k_N (in mm) will be written to the output file.

2.2 Select HD Input File

The path and the name of the HD data file where the results from the hydrodynamic simulation with MIKE 21 HD have been saved may be typed directly or found by browsing the disk by left-clicking with the mouse on the button to the right of the Data File entry in the menu.

Once the HD data file has been selected, the tool will check whether the specified file is a valid one, i.e. that it is output from a MIKE 21 HD simulation, and that it contains three items (h, P and Q).

2.2.1 Remarks and hints

- The simulation will only be performed in the area where wave and current data exist, i.e. in the region which is common to the hydrodynamic and wave model grids. Therefore, the user should carefully check that the area of interest for the computations is covered by both grids.

- The grid spacing(s) used in the simulations is/are identical to the grid spacing(s) that was/were used in the hydrodynamic simulation with MIKE 21 HD and are obtained from the HD data file.

- The subarea and sub series for the simulation are specified relative to the model grid used in the hydrodynamic simulation and the time step used to save the results to the HD output file. The grid spacing in the output data file will be the same as in the HD data file. If time series of Manning numbers and/or bed shear stress values are saved, the time step in the data file will be equal to or an integer number of times the time step in the HD data file.

2.3 Select Sub Area

The extension of the area where the apparent bed roughness simulations will be performed is specified in grid coordinates and relative to the grid where the HD input data has been saved.

2.4 Sub Series

The simulation period is specified by selecting the time steps in the HD input data file which define the start and end date(s) and time(s) for the calculation of the apparent bed roughness.

The time step used in the calculations may be selected as equal to or an integer number of times the time step in the HD data file.

Note that the start and end values are increased or decreased according to the value that has been specified in the Time steps field when the up/down arrows to the right of the Star and End fields are pressed.

2.5 Wave Input Data

The path and the name of the wave data file may be typed directly or found by browsing the disk by clicking the mouse on the button to the right of the Data File entry in the menu. The wave data may have been generated as the output from a wave simulation with MIKE 21 PMS.

The origin of the wave grid may not be coincident with the origin of the hydro-dynamic grid, as long as it is coincident with one of the nodes of this grid. If this is the case, the position of the origin of the wave grid relative to the origin of the HD grid must be specified (in HD grid-point coordinates) at the Origin of Wave Grid field.

After the wave data file has been selected, you must indicate which items in it correspond to the wave height, the direction of wave propagation and the wave period.

You must also specify whether the wave heights in the data file are significant wave heights H_{m0} or root-mean-square wave heights H_{rms}, and whether the wave period (as read from the data file or user-specified) is a mean wave period T_m or a peak wave period T_p.

2.5.1 Remarks and hints

- IMPORTANT: wave heights and periods are internally converted to root-mean-square wave heights H_{rms} and peak wave period T_p by the tool before using them as input for the wave boundary layer model. If the "bed shear stress map" option has been selected for the calculation, then the wave parameters input by the user will be assumed to be H_{rms} and T_p, meaning that the values in the data file will be used directly, without any type of scaling.

- The wave data file must contain at least two items (wave height and direction of propagation) or more than the three items (wave height, period and direction) required for the calculations. In either case it is pos-sible to choose a user-specified wave period rather than reading it from the wave data file.

- The Manning numbers and/or Shields parameter will only be calculated in the area where wave and current data exist, i.e. in the region which is common to the hydrodynamic and wave model grids. Therefore, the user should carefully check that the area of interest for the transport computa-tions is covered by both grids.

- The grid spacing(s) in the transport simulations is/are identical to the grid spacing(s) that was/were used in the hydrodynamic simulation with MIKE 21 HD, and are obtained from the HD data file.

- The grid spacing(s) used in the wave simulation does/do not need to be equal to the spacing(s) used for the hydrodynamic model grid. However, the grid spacing in the hydrodynamic data must be an integer number of times the spacing in the wave grid, both in the x- and the y-directions. The proportionality factors need not to be equal.

- $\Delta x_{HD} = n.\Delta x_{WV}$ with $n \geq 1$

- $\Delta y_{HD} = m.\Delta y_{WV}$ with $m \geq 1$

2.6 Sediment Data

The sediment data to be used in the computations may be specified as having constant or spatially varying size over the model area. In either case, the bed material is characterised through the median grain size, d_{50}. d_{50} is the particle size for which 50% of the sediment by weight is finer.

If the sediment properties are chosen as constant over the model area, you only have to select one value for d_{50} (in mm).

If spatially varying sediment properties are chosen, a data file containing one item (d_{50} in mm) must be specified. You may do this by typing the path and name of the data file (a .dt2 file in the old format and a .dfs2 file in the new format) or by left-clicking with the mouse on the button to the right of the Data file entry. The grid spacing and extent of the sediment data file must be the same as in the HD data file.

In both cases, the grain size is converted to geometric bed roughness k_N at every grid node by multiplying the value of d_{50} by 2.5. This value of k_N then enters the wave boundary layer model together with the wave and current parameters.

Furthermore, you have to specify the density of the bed material relative to that of water.

2.7 Output Specification

In this menu you specify the name and the path of the output data file where the parameters obtained from the calculation will be saved. You can choose to type the required information directly or to define the path and the name by browsing your disk. You may also provide a title for the output file

Only one item (the Manning number M) will be written to the output file if the Manning map calculation has been selected, regardless of whether the direct or parameterised version of the wave boundary layer model has been used. Two additional items will be saved in the output file if generation of a bed shear stress map was selected: the Shields parameter and the apparent roughness k_N.

2.8 Setup Status

The status page presents you with a summary of all the input parameters that you have specified. You should inspect the parameters to check that all of them are correct, and if not go back and change them

From this page you can either finish the setup, which will transfer your setup definition to the current toolbox file, or execute the setup

3 Border Adjustments

In MIKE 21 terminology a border is not the same as a boundary. A boundary is external, or "open". It is somewhere along the outer perimeter of a model.

A border is internal to a nested model. It is the region where grids (bathymetries) of different resolution meet.

You use the program 'border' to adjust nested bathymetries at model borders.

The input is a coarse grid and a fine grid bathymetry to be used by MIKE 21 Flow Model. Based on information about the position of the fine grid within the coarse grid, 'border' adjusts the bathymetries to ascertain compatibility between grids at all borders.

Changes are introduced within narrow sections along the position of borders. Changes are made in both the fine and the coarse grid. Common grid points are copied from the fine grid to the coarse grid.

When a nested model set-up comprises more than two bathymetries or levels of resolution, border adjustments must be made consecutively for overlapping pairs of bathymetries, starting from the inside moving out towards the (open) boundaries.

3.1 Bathymetry Input

On this page you specify the two bathymetry files to be adjusted.

You must also specify the origin of the fine grid within the coarse grid, in coarse grid coordinates.

NB: No changes are made to these original bathymetry input files.

3.2 Bathymetry Output

On this page you specify the names of the two adjusted bathymetry files to be created.

The names of output files must be different from the bathymetry input file names.

3.3 Overview

The overview page presents you with a summary of all the input parameters you have specified. You should inspect the parameters to check they are correct, - and if not go back and change them.

From the page you can either *Finish* the set-up, which will transfer your set-up definition to the current toolbox file, or execute the set-up.

3.4 Remarks and Hints

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 GridEdit tool.

If you plan to apply the Flow Model with more than one sub-level, you should apply the border adjustments 'inside-out'. That is, considering the example sketched in Figure 3.1, 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.
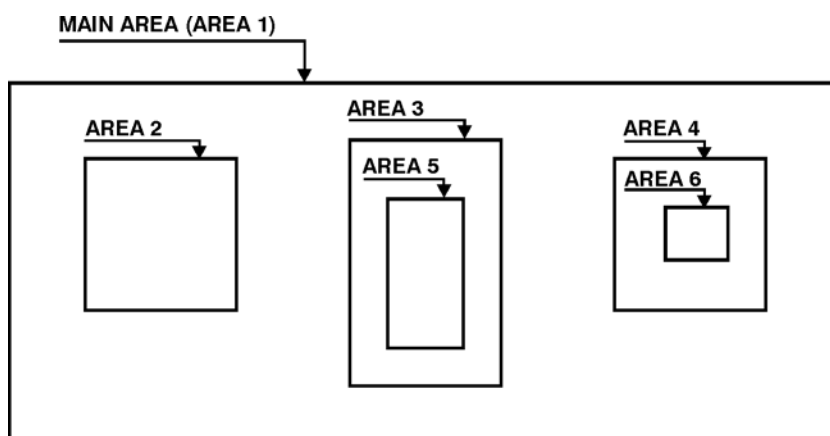
Figure 3.1   Sketch showing possible nesting of bathymetries

4 Discharge Calculations

M21DIS is a tool for calculating the discharge through a cross section, the cumulated discharge, and the cross sectional area. Instantaneous discharge and cumulated discharge are calculated separately for flow in each direction across the specified cross section as well as for net flow. They are all calculated for specified time steps.

Calculations are performed on the basis of data found in a user specified HD result file containing water depth and fluxes from a hydrodynamic simulation. Discharge results are stored in a time series file.

For all specifications, with the exception of I/O file names, default values are supplied. You can change these specifications by modifying the corresponding dialog entries.

4.1 Select Input File

On this page you specify the HD result file to be used as the basis for discharge calculations. The input data has to be a standard MIKE 21 HD dfs2 output file containing the items H, P and Q only, i.e. not other variables, and with Data type=1.

If your data file contains additional variables, or the Data type is different from 1 you will need to remove the additional variables and/or set the Data type to 1.
(This can be done by using the Grid Editor and the Data Utility Editor: please see 'MIKE Zero Pre- and Postprocessing, Generic Editors and Viewers, User Guide' from the MIKE Zero General Documentation Index.)

4.2 Cross Section Definition

Here you specify points defining the cross section.

The points must be given in grid coordinates.

You can make only one discharge calculation per execution of this tool, but you may let the cross section definition pass over model land areas.

The sign convention is: When looking along the section in the order of specification of points, the discharge will be counted positive when going from left to the right, and vice versa.

4.2.1 Detailed description

The cross section is specified in grid coordinates.

You can make only one discharge calculation per run, but the cross section may cross-land areas., see Figure 4.1.

The discharge will be taken as positive from left to right across the section.

You can specify at which time steps discharge is to be calculated.

At each time step 3 quantities are calculated: discharge through the cross section, accumulative discharge from time zero and the cross section area. Discharge and accumulated discharge are calculated for the two flow directions as well as for the net flow (positive plus negative flow directions).
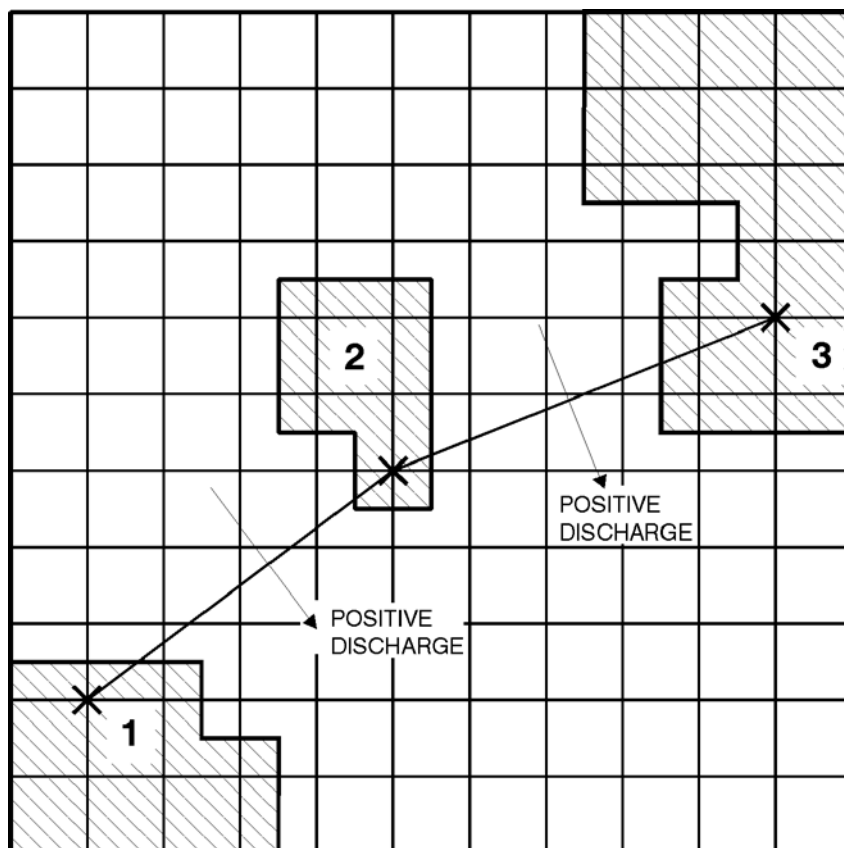
Figure 4.1   Definition of a cross-section

4.3   Select Subseries

Here you may specify the time steps you want to process.

4.4 Discharge Data Output

Here you specify a name for the discharge output data file. You may also specify a title (data description).

The Discharge calculation tool outputs a dfs0 file with 7 items:

1. Negative Discharge through the defined cross section
2. Positive Discharge through the defined cross section
3. Net Discharge through the defined cross section,
   this corresponds to the sum of 1. and 2.
4. Negative cumulative flux volume through the defined cross section
5. Positive cumulative flux volume through the defined cross section
6. Net cumulative flux volume through the defined cross section,
   this corresponds to the sum of 4. and 5.
7. Total defined cross section area.

4.5 Overview

The overview page presents you with a summary of all the input parameters you have specified. You should inspect the parameters to check they are correct, - and if not go back and change them.

From the page you can either *Finish* the set-up, which will transfer your set-up definition to the current toolbox file, or execute the set-up.

5 MIKE 21 Hotstart File Generator

Welcome to the MIKE 21 Tool for generating a hotstart file for MIKE 21 Cartesian grid simulations. The tool enables the creation of hot start files to be used as initial conditions for MIKE 21 simulations and particularly useful when hotstart files have not been created as part of a MIKE 21 simulation.

The numerical scheme of MIKE 21 (Cartesian grid version) requires an additional amount of information due to its staggered nature, and consequently, a hot start can not be performed from a standard result file.

The tool uses a standard dfs2 result file from an original MIKE 21 simulation as input and generates a hotstart file for use in subsequent simulations.

The opening page enable you to define a name for the actual hot start file generation setup. The name of toolbox setups will appear in the toolbox tree view once all specifications has been defined and the tool setup completed

5.1 Select Input Result Files

In this page you must specify definitions for the input results as well as basic definitions for generating the output hot start file.

- Number of areas:
  Specify the number of areas in the setup. Default value is one and should be only increased in the hot start tool is used in connection with nested MIKE 21 simulations.

- Time step:
  The simulation time step used in the original simulation generating the input result file (*.dfs2).

- Table definitions:

  – Result
    Input result files from previous simulation. Use the '...' browse button to select the input dfs2-files.
  – Origin
    The origin of the nested area defined as (j,k)-coordinates.
    Origin coordinates shall not be defined for area 1.
    Note: Only used in connection with nested simulations.
  – Enclosing Area
    Define the enclosing area of the actual input result file. The value is an integer pointing to the line-number in the table where the overall enclosing area is defined.
    Note: Only used in connection with nested simulations.

– Output hot start files
File name of the output hot start file generated from the tool. Use the '...' button to browse for a location and file name definition for the dfs2 output file.
I case of nested simulations, please make sure to define different output file names.

● Hot start time:
The hot start time is the time from which you wish to extract results from the input result file.
The 'Start time' and 'End time' fields will present the start and end time of simulation in the input result file for area 1.

5.2 Flooding and Drying Values

In this page you must specify whether Flooding and Drying was enabled in the simulation that generated the input result file. If so, you must enable the check-box in this page.

It is required to define the 'Flood depth' value manually as this parameter is not stored in the input result file. The value should be defined identical to the flooding depth defined in the simulation generating the input result file.

The value of the drying depth parameter in a MIKE 21 simulation is stored in the standard result file and hence, it is possible to extract this value and write this to the 'Drying depth' field. Consequently, the 'Drying depth' value is not editable in this page.

5.3 Setup Status

The status page presents you with a summary of all the input parameters specified. You should inspect the parameters to check that all of them are correct and if not, click Back to change them.

If you want to create the hot start file, click Execute, and finally, you can click Finish to complete and transfer the setup definition to the current toolbox file.

5.4 Remarks on Toolbox Assumptions and Limitations

Parameter values in hot start files generated as part of a MIKE 21 simulation are determined using some engine state variables that are not saved in a standard MIKE 21 result file. Consequently, this tool for generating a hot start file from a result file of a previous simulation will need to include a number of assumptions to accommodate for the missing information available in a simulation.

These basic assumptions and to some extent limitations in the generation of hot start files are listed below:

- The drying depth are used to evaluate whether cells at the specified hot start time are wet or dry. If the water depth in the input result file are smaller than the drying depth the tool assign a default value of 0.0002 m as a surface elevation (depth) value.
  The 0.0002 m is an internal default value of MIKE 21 to make cells in a simulation prepared for becoming active (parameter named 'epsf' in MIKE 21)

- The above listed assumptions imply that hot start files generated from this tool will in most cases be slightly different from hot start files generated as part of a MIKE 21 simulation.
  Therefore, initial conditions for a MIKE 21 simulation using hot start files created from this tool and from a previous MIKE 21 simulation can vary and result files from such two simulations most likely not completely identical.

6 HD Transfer Boundary Data

Transfer Boundary is a MIKE 21 Flow Model HD tool for preparation of HD transfer boundaries. Boundary data are extracted from results of a previous HD simulation. A MIKE 21 HD simulation output data file (a matrix time series with fields of H, P and Q) is used to generate the new boundary data (a line time series data file of water levels and fluxes across and along the boundary line).

For all specifications, with the exception of I/O file names, default values are supplied. You can change these specifications by modifying the corresponding dialog entries.

6.1 Select Input File

On this page you specify the HD result file to be used as the basis for transfer boundary generation.

6.2 Boundary Position and Length

Here you specify:

- the origin, and length, of the boundary line for which data should be extracted. The origin must be specified in "old grid" coordinates.

- the angle - or line orientation - clockwise from the boundary line to the south boundary of the old model

- the desired grid spacing along and across the boundary line

- whether the boundary in the new model is a West, North, East or South boundary (this affects the way the boundary data is calculated

The new boundary line has to lie within the old model area, not on a boundary.

6.2.1 Detailed description

The input data has to be a type 2 output file from a MIKE 21 HD simulation containing the three items H, P and Q.

You specify the line in the old model from where values for boundary data generation are to be taken. This is done by giving the *start coordinate*, the line *orientation*, *number of points* along the line and the *grid spacing* in the new model, both along line and perpendicular to line. Please see Figure 6.1 for an example.

The *start coordinate* is given in grid coordinates refering to the old model grid.

The angle giving the *orientation* of the boundary line is measured from the boundary line in the new grid to the south boundary in the old grid, positive clockwise.

The *number of points* along the line must match the definition of the corresponding boundary in the new model.

The *grid spacing* along and across the boundary line in the new model are assumed to be less than or equal to the grid spacings of the old model. The grid spacings in the new model must be specified in order to enable transferring of data from the old staggered grid to the new staggered grid.

It is necessary to specify whether the boundary line is to be model south, model north, model east or model west in the new model, as this affects the boundary data calculations, i.e. interpolation to the spatially staggered positions in the new model.

You can only prepare boundary data for one boundary per run. If your new model requires more than one transfer boundary data file, then you must apply.
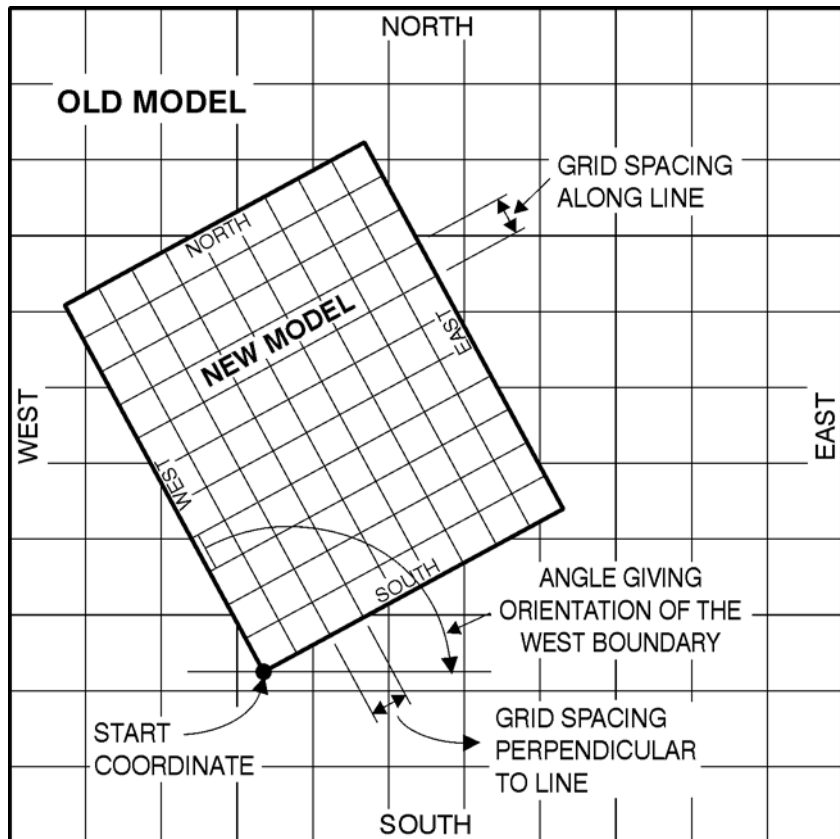
Figure 6.1   Example of a west boundary line for calculation of HD transfer boundary data

6.3   Subseries Selection

Here you may specify the time steps you want to process.

6.4   Transfer Data Output

Here you specify a name for the transfer boundary output data file. You may also specify a title (data description).

6.5   Overview

The overview page presents you with a summary of all the input parameters you have specified. You should inspect the parameters to check they are correct, - and if not go back and change them.

From the page you can either *Finish* the set-up, which will transfer your set-up definition to the current toolbox file, or execute the set-up.

6.6 Remarks and Hints

The position of the line must be fully inside the old model, and not along a boundary. At least one grid point in all directions around the line should be available in the old grid. Please make sure that this is the case.

The concept of applying transfer data relies heavily on the assumption that the water depth along the boundary line in the new model is the same as the water depth at corresponding locations in the old model. Care should be taken when preparing the bathymetries – for the old and for the new model – in order to fulfil this requirement.

In cases where flooding and drying occurs along the boundary line in the new model, the transfer data might be inadequate due to the interpolation from the old, coarser grid to the new, finer grid. In order to avoid model artifacts created by interpolation between flooded and dried points, it is recommended that the flooding and drying depths in the new model are set higher, say 5 % higher, than in the old model.

M21 TOOLS: MORPHOLOGY

The MORPHOLOGY Tools in the M21 Toolbox include the facilities:

- Curvilinear Discharge Calculations
- Scour Calculator

7 Curvilinear Discharge Calculations

This is a tool for calculating discharges through cross sections aligned with the curvilinear grid, used in the MIKE 21 C model.

The calculated discharges are obtained by integration along the specified line(s) of the instantaneous flux components perpendicular to the cell faces of interest. The discharge is calculated for the specified time steps.

Calculations are performed on the basis of data found in a user specified HD result file containing water depth and fluxes from a hydrodynamic simulation or an ST result file containing information about bed load, suspended load and/or total load. Discharge results are stored in a time series file.

7.1 Select Input File

On this page you specify the HD or ST result file and the curvilinear grid file to be used as the basis for discharge calculations.

In the case where a ST file is applied information about the j-component of the vector field of interest need to be specified, i.e. Bed load x, Suspended load x, or Total load x.

7.2 Cross Section Definition

Here you specify the number of cross sections from which you want information about the instantaneous discharge and the points defining the cross section.

The points must be given in grid coordinates aligned with either the j-axis or k-axis.

The sign convention is: Positive discharge means flux in the same direction as the j-axis (or k-axis)

7.2.1 Detailed description

First you specify which data to use. The input data has to be a type 2 HD simulation output file containing H, P and Q or a ST simulation output file containing information about either bed load, suspended load or total load. Furthermore, the corresponding grid file has to be specified.

The cross section is specified in grid coordinates aligned with either the j-axis or the k-axis.

You can make up to 15 discharge calculation per run. The cross sections are allowed to cross land areas.

The discharge is defined positive in the direction of the j-axis and the k-axis

You can specify at which time steps discharge is to be calculated.

7.3   Select Subseries

Here you may specify the time steps you want to process.

7.4   Discharge Data Output

Here you specify a name for the discharge output data file. You may also specify a title (data description).

7.5   Overview

The overview page presents you with a summary of all the input parameters you have specified. You should inspect the parameters to check they are correct, - and if not go back and change them.

From the page you can either *Finish* the set-up, which will transfer your set-up definition to the current toolbox file, or execute the set-up.

8   Scour Calculator

This tool creates a time series file reflecting the temporal evolution of a scour hole around a monopile due to combined wave and current motion. The scour hole will be represented by a scour depth and a collection of scour widths.
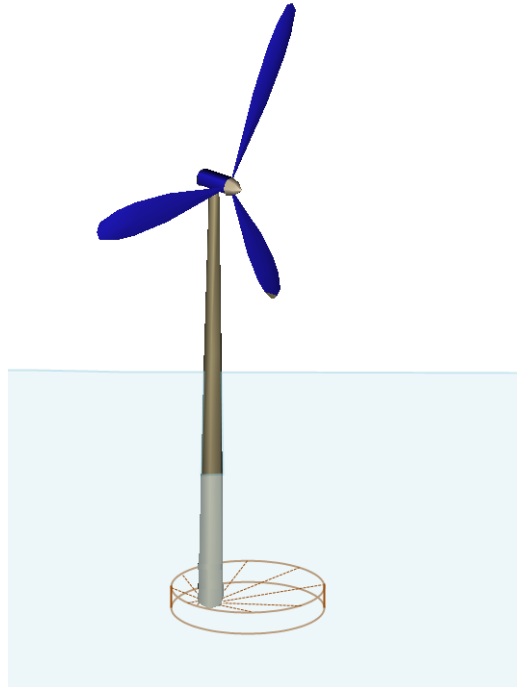
Figure 8.1   Simplified description of scour hole around monopile

8.1   Simulation Time

You have to specify the period for which the scour is to be calculated.

The scour period is defined by the simulation start date and end date, including the time step for the predicted values.

8.2   Wave and Current Input Data

You have to specify the wave and current conditions to be used in the calculation.

The wave input file must be a time series file (.dfs0) which contains items for the wave height, the wave period and the wave direction.

The current input file must be a time series file (.dfs0) which contains an item for the surface elevation as well as two items for the current conditions, specified by one of two different current formats:

- (u,v) components
- Current speed and current direction

The data must cover the complete simulation period. The time steps of the wave and current input data can be mutually different and neither do the time steps in the input data files have to be the same as the time step for the calculation. A linear interpolation will be applied if the time steps from the input files differ from the time step for the calculation.

NOTE: The definition of the North direction must be the same for the waves and the currents.
E.g. Using the vector component output (u,v) from a MIKE 21 HD FM simulation may give a minor error due to a missing influence of the convergence angle in the internal conversion of the (u,v) components into speed and direction.

8.3 Bed Shear Stress method and Site Parameters

8.3.1 Bed shear stress method

You can select one of three types of bed shear stress methods:

- Fredsøe (1984)
- Grant and Madsen (1979)
- Soulsby's Data 13

For more information please see Bed shear stress *(p. 53)*.

8.3.2 Site parameters

You have to specify a number of site parameters.

Still water depth
Still water depth at the location of the monopile.
The resulting depth is obtained by adding the surface elevation value.

Pile diameter
Diameter of the monopile structure.

Initial scour depth
Initial depth of scour hole at the beginning of the simulation

Scour depth/pile diameter ratio

This ratio express the non-dimensional equilibrium scour depth in current alone. The default value is S/D = 1.3.

8.4 Sediment and Flow Properties

8.4.1 Sediment parameters

You have to specify a number of parameters relating to the transport of sediment.

Mean sediment diameter

Average grain diameter, d_{50}, for sediment material on the sea bed.

Relative density of sediment

The default value is 2.65, corresponding to quartz sand. The value should be changed if the bed consist of heavy minerals.

Angle of repose

The slope failure mechanism will come into effect if the local bed slope becomes steeper than the specified angle of repose. The default value is 32 degrees.

8.4.2 Flow parameters

You have to specify two parameters relating to the flow.

Density of water

The default value is 1025 kg/m^3.

Viscosity

The default value is be 10^{-6} m^2/s.

8.5 Specify Output Data

8.5.1 Time series output

On this page you specify the settings for the default time series output file. Furthermore you can enable the Grid series output from here.

The time series output is defined by a file name and an output frequency. The output will contain the following items for each output time step (as illustrated in Figure 8.2):

1. Scour depth
2. The angle where maximum scour width occurs

3. Maximum width of scour

4. Width of scour at location 90 degree clockwise of maximum width

5. Width of scour at location 180 degree clockwise of maximum width

6. Width of scour at location 270 degree clockwise of maximum width

NOTE: The scour depth next to the monopile is assumed to be spatially uniform all around the monopile.
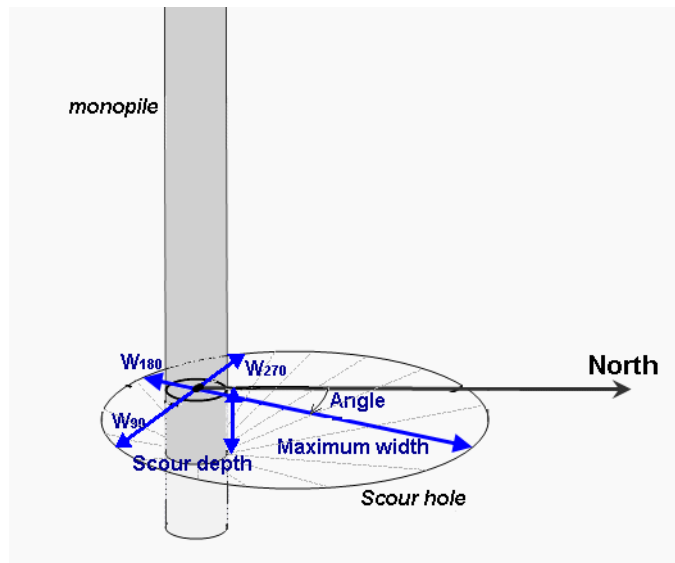
Figure 8.2   Illustration of definition of output items

8.5.2   Grid series output

On this page you specify the settings for the grid series output file. The grid series output is defined by a file name, an output frequency as well as some required grid properties.

The output grid series will contain an overall representation of the time-varying bathymetry around the monopile.

NOTE: If you are running a long Scour simulation and specify a grid with a high resolution and extent, the size of the output grid series may become extremely huge with an output frequency of 1. In such a case it is adviced to increase the output frequency, thus reducing the size of the grid series output file to an appropriate level.

Grid properties

You have to define the grid axis for the dfs2 output file. With the default values the resulting grid will support full representation of the modelled scour hole up to 10 meters away from the center axis of the monopile.

Grid spacing

The Grid spacing represents the degree of resolution in both the X and Y directions of the grid. The default value is 0.1 m.

Grid extent

The Grid extent is to be interpreted as the number of grid cells from the center axis of the monopile to the boundary of the grid, see Figure 8.3. The default value is 100.
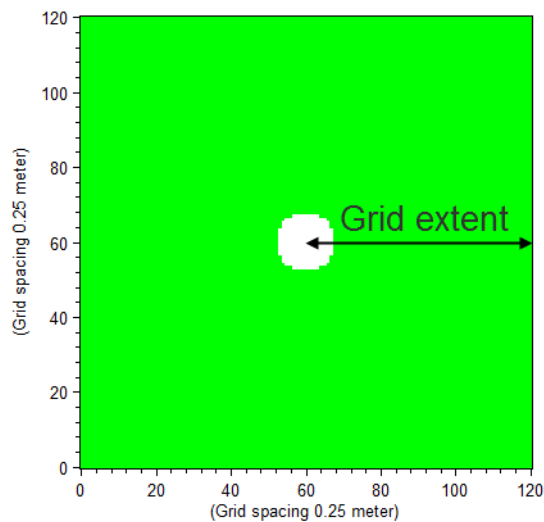
Figure 8.3   Example of 2D output for an area covering 30x30 m
Grid spacing = 0.25 m, Grid extent = 60, Pile diameter = 4 m.
Number of grid points along axis = 121.

8.6   Status

The status page presents you with a summary of all the input parameters you have specified. You should inspect the parameters to check they are correct, - and if not go back and change them.

From the page you can either *Finish* the set-up, which will transfer your set-up definition to the current toolbox file, or execute the set-up.

8.7 Scientific Background

Today industry standard for calculating the scour depth around a circular pile is using a fixed number (often 1.3 is used) times the diameter of the pile, corresponding to the equilibrium scour in pure current. However, it has been shown by model tests that the equilibrium scour depth in combined waves and current and in waves alone is significantly smaller than 1.3 times the pile diameter. This is especially the case when waves are dominating the near bed conditions.

This model is based on the assumption that the seabed near the pile always develops towards an equilibrium scour depth determined by the actual environmental conditions.

8.7.1 Handling of directions

The various directions in the calculation are defined as illustrated in the figures below.
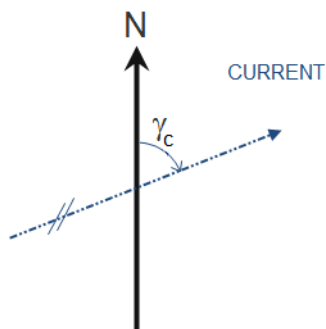
Figure 8.4   Definition of current direction
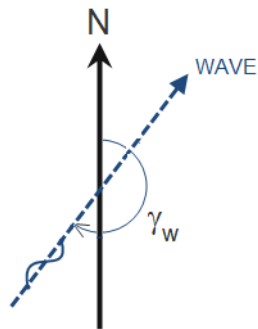
Figure 8.5   Definition of wave direction
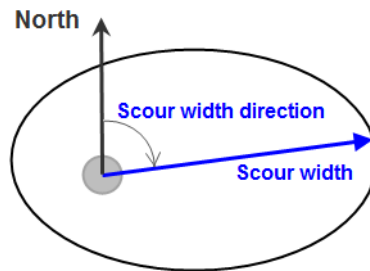
Figure 8.6   Definition of scour width direction for maximum scour width

8.7.2   Bed shear stress

The mean bed shear stress, τ_{mean}, is a function of the bed shear stress for currents, τ_c, the bed shear stress for waves, τ_w, and a coefficient, Ψ, that depends on the choice of bed shear stress method and the internal angle between waves and current /5/.

$$\tau_{mean} = \Psi(\tau_c + \tau_w)$$ (8.1)

The bed shear stress for currents only, τ_c, is related to the depth-averaged current speed, V, through the drag coefficient, C_D, by

$$\tau_c = \rho C_D V^2$$ (8.2)

The drag coefficient C_D is expressed by

$$C_D = \left[\frac{0{,}4}{1 + \ln(z_0/h)}\right]^2 \tag{8.3}$$

where h is the water depth, $z_0 = k_s/30$ and $k_s = 2.5*d_{50}$.

The bed shear stress for waves only, τ_w, is expressed by

$$\tau_w = 0{,}5\rho f_w U_w^{\,2} \tag{8.4}$$

where f_w is the friction factor for the wave and U_w is the velocity at the sea-bed.

The friction factor for waves, f_w, is found by

$$f_w = 1{,}39\left(\frac{A}{z_0}\right)^{-0{,}52} \tag{8.5}$$

where $A = U_w T/(2\pi)$.

The velocity U_w is found by

$$U_w = \frac{\pi H_s}{T_p \sinh(kh)} \tag{8.6}$$

The choice of bed shear stress method determines the coefficient Ψ as well as other parameters.

Fredsøe (1984)

The maximum shear stress is found by

$$\tau_{max} = Z \cdot (\tau_c + \tau_w) \tag{8.7}$$

where Z is a coefficient based on the choice of bed shear stress method and the internal angle between waves and current.

For more information see ref. /1/.

Grant and Madsen (1979)

The maximum shear stress is found by

$$\tau_{max} = Z \cdot (\tau_c + \tau_w) \tag{8.8}$$

where Z is a coefficient based on the choice of bed shear stress method and the angle between waves and currents.

For more information see ref. /2/.

Soulsby's Data 13

The maximum shear stress is found by

$$\tau_{max} = \sqrt{(\tau_{mean} + \tau_w\cos\gamma)^2 + (\tau_w\sin\gamma)^2} \qquad (8.9)$$

where γ is the angle between waves and currents.

For more information see ref. /4/.

8.7.3   Scour depth

The development of the scour depth for constant conditions in time is calculated by

$$S(t) = S_{eq}\left(1 - \exp\left(-\frac{t}{T}\right)\right) \qquad (8.10)$$

where $S(t)$ is the scour depth for time t, S_{eq} is the Equilbrium scour depth and T is the time scale for the erosion process /6/.

Equilbrium scour depth

The equilibrium scour depth S_{eq} depends on the wave and flow regime, specified by the Keulegan-Carpenter number KC and the velocity ratio, U_{cw} /6/.

$$U_{cw} = \frac{U_c}{U_c + U_m} \qquad (8.11)$$

where U_c is current velocity D/2 above the seabed and U_m is maximum wave velocity at seabed.

- For cases with $KC>100$ or $U_{cw}>0.8$ the model for steady current is applied, and the equilibrium scour depth is given by the user defined ratio S/D.

- If $U_{cw}<0.7$ and $6<KC<100$, the criteria is fulfilled for the combined waves and current model /6/ and the equilibrium scour depth is given by eq. (8.12).

- In the interval 90<*KC*<100 or 0.7<U_{cw}<0.8 a linear transition from the combined wave and current scour depth to the steady current scour depth is applied.

$$\frac{S_{wc,\,eq}}{D} = \frac{S_{c,\,eq}}{D}[1 - \exp\{-A(KC - B)\}] \qquad (8.12)$$

where

$$A = 0{,}03 + 0{,}75\,U_{cw}^{2,6} \qquad (8.13)$$

$$B = 6\exp(-4{,}7\,U_{cw}) \qquad (8.14)$$

If *B* > *KC* then *B* is set equal to *KC*.

8.7.4 Scour width

Together with the scour depth, the scour is characterised by the scour hole width. The equilibrium scour hole is characterised by four widths: Two widths in-line of the resulting flow (upstream and downstream of the pile) and two widths perpendicular to the resulting flow, as shown in Figure 8.7. The equilibrium scour width is assumed to vary smoothly between those four values.
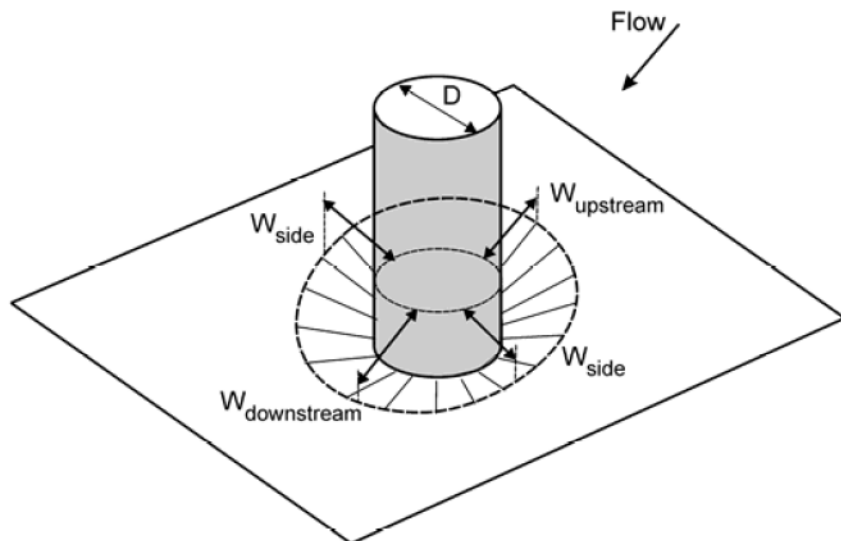
Figure 8.7   Definition of the width of the scour hole /3/.

The widths of the scour hole are calculated by use of eq. (8.15).

$$W_n(\alpha) = W_{eq,n}(\alpha) + (W_{n-1}(\alpha) - W_{eq,n}(\alpha))\exp\left(-\frac{dt}{T}\right) \tag{8.15}$$

where W is the scour width, W_{eq} is the equilibrium scour width, dt is the time frame between time step *n-1* and time step *n* and *T* is a time scale. It is assumed that the inclination, α, of the side of the scour hole is constant for a specific position relative to the flow, but will change depending on the position relative to the flow direction. Figure 8.8 shows the development of the scour width over a time step dt.
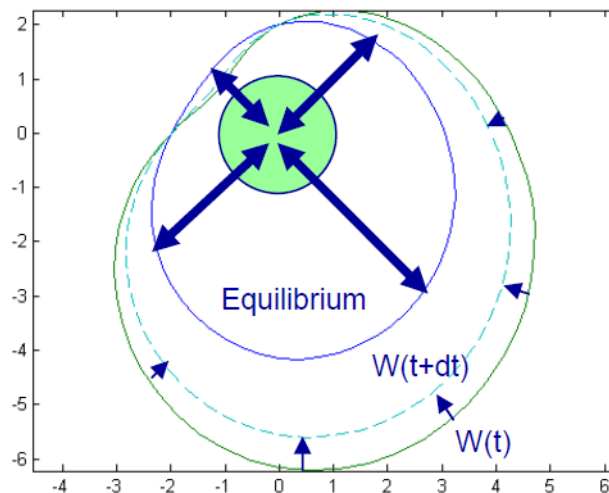
Figure 8.8   Equilbrium and time variation of scour width /3/

8.7.5   References

/1/   Fredsøe, J. (1984) *Turbulent boundary layer in wave-current motion.* J. hydraul. ENG. ASCE, 110,1103-20.

/2/   Grant, W.D. and Madsen, O. S. (1979). *Combined wave and current interaction with a rough bottom*. J. Geophys. Res. 84 (C4), 1798-1808.

/3/   Nielsen, A.W. and Hansen, E.A. (2007) *Time-varying wave and current-induced scour around offshore wind turbines*. Proceedings of 26th International Conference on Offshore Mechanics and Arctic Engineering, OMEA 2007. (OMAE2007-29028)

/4/   Soulsby, R. L. (1995). *Bed shear stress due to combined waves and currents in Advances in Coastal Morphodynamics*, ed. M. J. F.

Stive, H. J. de Vriend, J. Fredsøe, L. Hamm, R. L. Soulsby, C. Teisson and J. C. Winterwerp, pp.4-20 to 4-23. Delft Hydraulics, Netherlands

/5/   Soulsby, R. (1997) *Dynamics of marine sands: A manual for practical applications.* Thomas Telford Publications.

/6/   Sumer, B.M. and Fredsøe, J. (2001a). *Scour around a pile in combined waves and current.* J. Hydraulic Engineering, ASCE, Vol.127, No. 5, 403-411

8.8 Example

The purpose of this example is to illustrate how to generate a scour hole around a pile and to review the influence of the choice of Bed shear stress method.

8.8.1 Setup

The monopile is located in an area subject to tide. The simulation time covers a few months where the wave, water level and current data are varying in time as indicated in Figure 8.9.

The monopile is located at 5.0 m still water depth and has a diameter of 4.0 m. The scour depth to diameter ratio is set to 1.3 and initially no scour has occurred.

In three different setups the Bed shear stress method is selected as Fredsøe (1984), Grant and Madsen (1979) and Soulsby's Data 13, respectively. The setup using Soulsby's Data 13 also outputs the results in a Grid Series file.

For the remaining parameters the default parameters are used, e.g. the material on the bed has a mean sediment diameter of 0.2 mm.
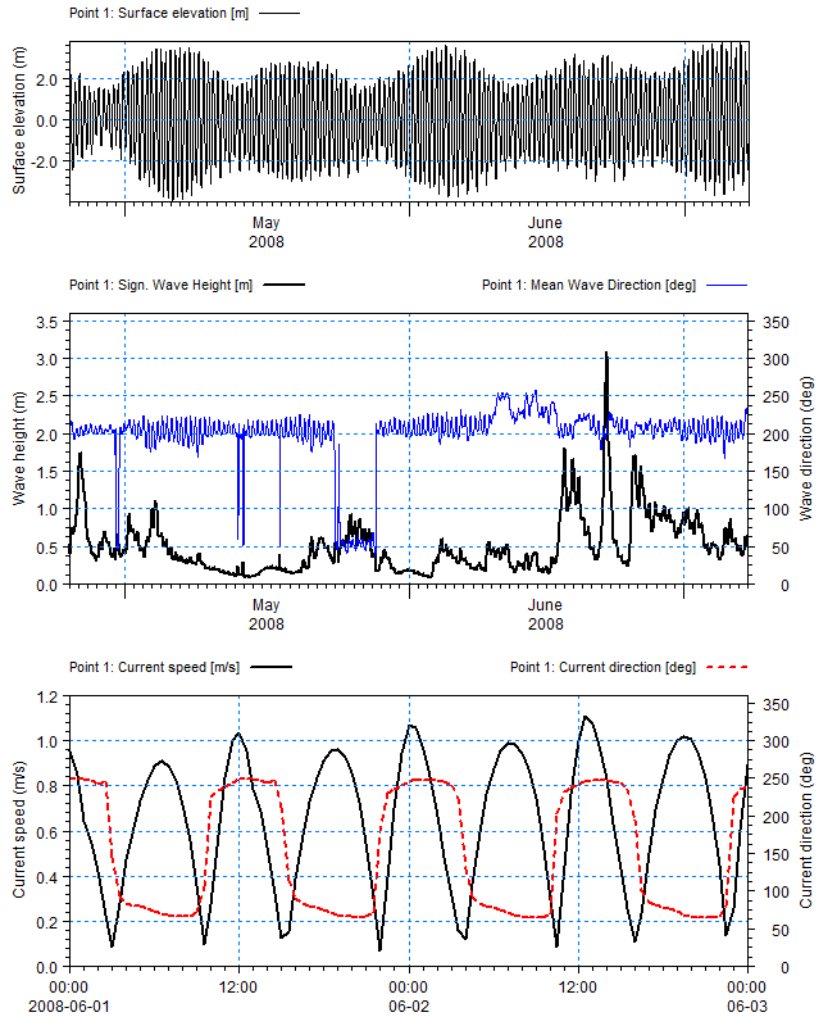
Figure 8.9   Time varying wave and current data
             Upper: Water surface
             Middle: Height and direction of waves
             Lower: Detail of speed and direction of tidal currents

8.8.2   Results

The resulting scour depth is shown in Figure 8.10 and the resulting maximum scour width (and associated direction) is shown in Figure 8.11.

Figure 8.10 shows that in general the use of Grant and Madsen (1979) will give a more rapid morphological response to changed conditions compared to Fredsøe (1984), whereas using Soulsby's Data 13 gives a slower response. The slower response time could however lead to slightly higher values of the maximum scour width as indicated in Figure 8.11.

Scour Calculator
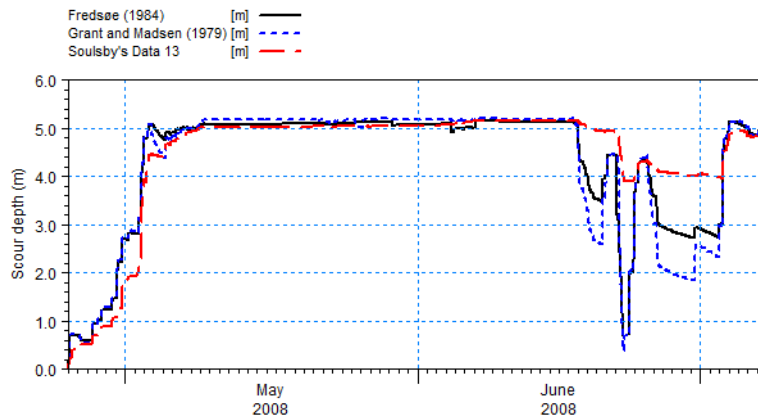
Figure 8.10   Resulting scour depth during simulation time
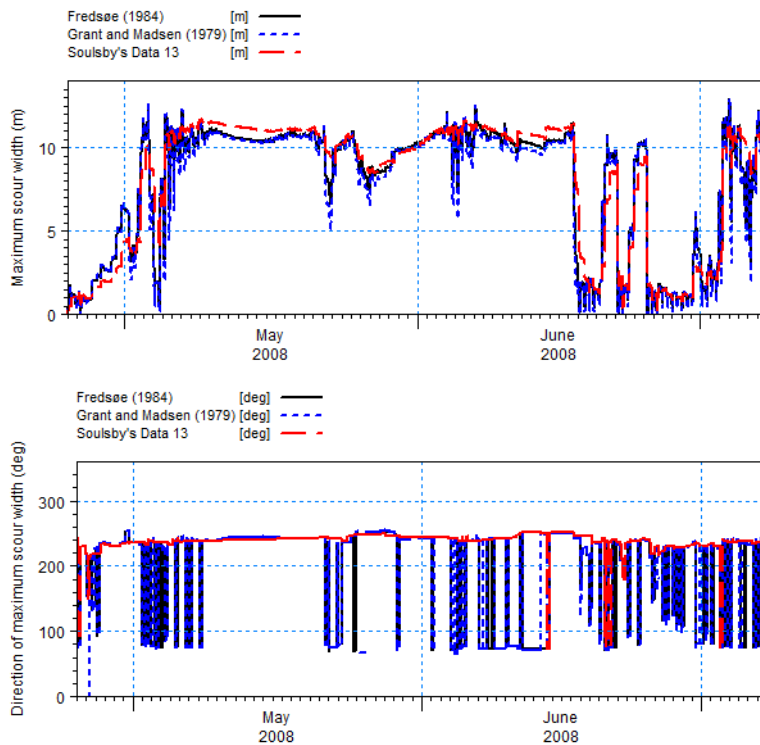
Figure 8.11   Resulting maximum scour width during simulation time
                    Upper: maximum scour width
                    Lower: direction of maximum scour width

The setup using Soulsby's Data 13 also outputs the scour depth by a dfs2 grid series file. This enables presentation in the spatial dimension, using e.g. Plot Composer or MIKE Animator Plus.

60   MIKE 21 Toolbox - © DHI
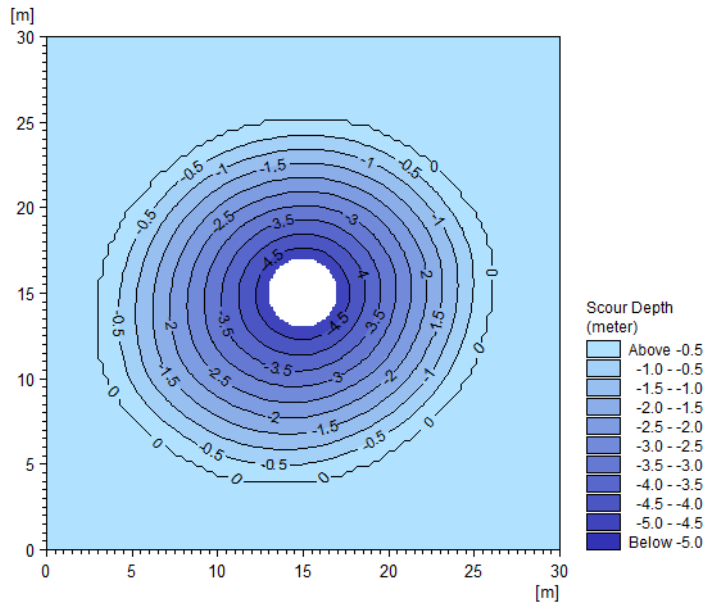
Figure 8.12   Example: Resulting scour bathymetry using Plot Composer
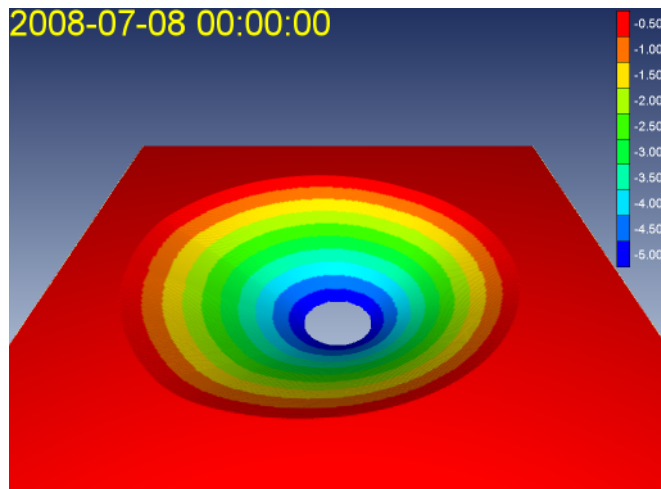
Figure 8.13   Example: Resulting scour bathymetry using MIKE Animator Plus

M21 TOOLS: SEDIMENTS

The SEDIMENTS Tools in the M21 Toolbox include the facilities:

- Generation of Sediment Transport Tables
- Generation of Q3D Sediment Transport Tables
- Extension of Sediment Transport Tables
- Extension of Q3D Sediment Transport Tables
- Integration of Storms
- Calculation of Sediment Discharge

9　Extension of Sediment Transport Tables

Welcome to the MIKE 21 Tool for extension of sediment transport tables created previously by the MIKE 21 Tool for the generation of sediment transport tables.

Extra values for the hydrodynamic and/or sediment transport parameters may be added along any of the axes of the existing transport table by use of this tool.

The transport rates that will be added to the tables are determined using the sediment transport program STP of LITPACK. It means that the effects of combined waves and current, arbitrary angle between the waves and the current, breaking/non-breaking waves, plane or ripple-covered bed, uniform or graded sediment, among others, on the sediment transport rates are accounted for.

The transport rates in the direction of the current will be saved in an ASCII data file with extension ".dat". If the effect of the bed slope on the sediment transport rates has been accounted for in the existing tables, then the bed-load transport rates in a direction normal to that of the current are saved in an ASCII data file with extension ".bsl".

9.1　Select Input File

In this menu you specify the path and the name for the existing sediment transport tables. You can either type the requested information, or browse your disk to select the path and the name of the file.

The tool founds out whether the effect of the bed slope on sediment transport has been accounted for or not in the existing tables by inspecting the heading of the ASCII file having the specified file name and extension ".dat". If the existing transport table includes the effect of the bed slope on the transport rates, then the table in the ASCII file with the selected name and extension ".bsl" will also be extended.

If any of the selected input files does not exist, a warning message will be issued.

9.2　Extended Axis Specification

In this menu you specify the number of points to add along the axes of the existing sediment transport table.

For all axes, you can either insert points before the existing one(s), or append after them, or both. If you attempt to insert too many points along the wave height, the wave period, the sediment grading or the wave height-to-water depth axes, in such a way that the First Value becomes zero or negative, a

warning message will be issued. In order to perform this check, the relevant information is obtained from the heading of the existing transport table(s).

A warning message will also be issued if the First Value for the Grain Size axis becomes smaller than 0.060 mm because an attempt to insert too many points before the existing one(s) has been done.

Note that extra points can only be appended along the axis corresponding to the current speed, as the First Value along this axis is always smaller than or equal to the spacing.

9.3   Select Output File

In this menu you specify the path and the name for the ASCII output data file(s) where the extended transport tables will be saved. You can either type the requested information, or browse your computer to select the path and the name of the file(s).

Note that an output file with the selected name and extension ".dat" will be created if the effect of the bed slope on sediment transport has not been accounted for, whereas two ASCII files having the same name and extensions ".dat" and ".bsl" will be created if the effect has been included in the calculations. The extension(s) will be added automatically by the tool for extension of existing transport tables.

If the selected output file(s) already exist(s), you will be prompted whether you want to overwrite it/them or not.

Note
No warning message will be issued if the tool is executed in batch mode.

9.4   Overview

The overview page presents you with a summary of all the input parameters you have specified. You should inspect the parameters to check they are correct, - and if not go back and change them.

From the page you can either Finish the set-up, which will transfer your set-up definition to the current toolbox file, or execute the set-up.

10 Extension of Q3D Sediment Transport Tables

Welcome to the MIKE 21 Tool for extension of quasi-3D (Q3D) sediment transport tables created previously by the MIKE 21 Tool for the generation of Q3D sediment transport tables.

Extra values for the hydrodynamic and/or sediment transport parameters may be added along any of the axes of the existing transport table by use of this tool.

The transport rates that will be added to the tables are calculated using STPQ3D, the deterministic intra-wave sediment transport program of DHI. For additional details about STPQ3D, reference is made to the scientific description available via the MIKE 21 or the MIKE 3 Documentation Index from *MIKE 21 & MIKE 3 Flow Model FM, Sand Transport Module, Scientific Documentation*.

The Q3D description of hydrodynamics and sediment transport included in STP_Q3 means that transport components will be calculated both in the direction of the mean current and perpendicular to it. STP_Q3 accounts for - among others - the effects of combined waves and current, arbitrary angle between the waves and the current, breaking/non-breaking waves, plane or ripple covered bed, uniform or graded sediment, when calculating the sediment transport rates.

The transport rates in the direction of the current will be saved in an ASCII data file with extension ".lon", whereas the bed-load transport rates in the direction normal to the current will be stored in an ASCII data file with extension ".crs".

10.1 Select Input File

In this menu you specify the path and the name for the existing sediment transport tables. You can either type the requested information, or browse your disk to select the path and the name of the file.

The tool reads the general parameters, the axis definitions and whether the effect of bed slope on sediment transport has been accounted for or not in the existing tables by inspecting the heading of the ASCII file having the specified file name and extension ".lon".

The tool also looks for an ASCII file having the selected name and extension ".crs" in the same directory, as the transport table for the cross-shore transport rates will also be extended. If any of the two files does not exist, a warning message will be issued.

10.2 Extended Axis Specification

In this menu you specify the number of points to add along the axes of the existing sediment transport table.

For all axes, you can either insert points before the existing one(s), or append after them, or both.

NOTE

- If you attempt to insert too many points along the wave height, the wave period, the sediment grading or the wave height-to-water depth axes, in such a way that the First Value becomes zero or negative, an error message will be written to the log-file and the computation will stop.

- An error message will also be issued if the First Value for the Grain Size axis becomes smaller than 0.060 mm because an attempt to insert too many points before the existing one(s) has been done.

- Extra points can only be appended along the axis corresponding to the current speed, as the First Value along this axis is always smaller than or equal to the spacing.

- Values may only be added to the Bed slope, current direction and Bed slope, normal to current axes if the effect of bed slope on the calculated transport rates was accounted for in the transport tables being extended. The tool checks this by reading the heading of the existing tables.

- Values may only be added to the Centrifugal Acceleration if the effect of centrifugal acceleration on the calculated transport rates was accounted for in the transport tables being extended.
  Extra points can only be appended along the axis corresponding to the centrifugal acceleration, as the First Value along this axis is always 0.

10.3 Select Output File

In this menu you specify the path and the name for the ASCII output data file(s) where the extended transport tables will be saved. You can either type the requested information, or browse your computer to select the path and the name of the file(s).

Note that two output files will be created. The first one will have the selected name and extension ".lon", where the calculated sediment transport rates in longshore direction will be saved. The second file will be named as the first one but will have extension ".crs" and will store the transport rates in cross-shore direction. The extensions will be added automatically by the tool for extension of existing transport tables.

If one or both of the selected output files already exists, you will be prompted whether you want to overwrite it/them or not.

Note

No warning message will be issued if the tool is executed in batch mode (i.e. from a DOS prompt).

10.4 Overview

The overview page presents you with a summary of all the input parameters you have specified. You should inspect the parameters to check they are correct, - and if not go back and change them.

From the page you can either Finish the set-up, which will transfer your set-up definition to the current toolbox file, or execute the set-up.

NOTE: Due to parallelisation in the model execution, the progress bar is only updated after a major block of data has been calculated, corresponding to all combinations of input velocity, H/D, wave height and wave period.

11 Generation of Sediment Transport Tables

Welcome to the MIKE 21 Tool for generation of simple sediment transport tables for the classic MIKE 21 ST module. The transport tables contain sediment transport rates calculated under the combined effect of waves and current for user-selected ranges of variation of the hydrodynamic and non-cohesive sediment transport parameters.

The effects of combined waves and current, arbitrary angle between the waves and the current, breaking/non-breaking waves, plane or ripple-covered bed, uniform or graded sediment, among others, on the sediment transport rates are accounted for.

It is assumed that the flow is mainly described by wave-driven currents, i.e. the angle between the wave and current direction is between 0 and 90 degrees.

The transport rates saved in the ASCII data files with extension ".dat" correspond to the magnitude of the sediment transport in the direction of the current. If the effect of the bed slope on the sediment transport rates has been accounted for, then the bed-load transport rates in a direction normal to that of the current are saved in an ASCII data file with extension ".bsl".

11.1 General Parameters

On this page you have to specify the values of the general parameters for the calculation of the sediment transport rates. These parameters are the tolerance in calculation of concentration of suspended sediment, the maximum number of wave periods the STP program will iterate over in the search for a periodic solution for the suspended concentration, the density of the sediment relative to that of water, the critical value of the Shields parameter for initiation of motion and the water temperature in degrees Celsius.

A value between 10^{-6} and 10^{-4} is expected for the tolerance. The smaller the value, the longer the time needed to generate the transport table.

The maximum number of wave periods may be selected between 70 and 300. A larger value will result in more time being required for the generation of the transport table.

The relative density of the sediment may be chosen between 1.10 and 3.65, with 2.65 the usual value for sand. The critical value of the Shields parameter may be between 0.045 and 0.060, while the water temperature may lie between 5 and 50 degrees Celsius.

In this menu you also have to select whether the influence on the sediment transport rates of wave-generated ripples and the bed slope are to be accounted for or not. You can also choose whether the bed concentration of suspended sediment is to be calculated according to the deterministic model

of Engelund & Fredsøe (1976) or by use of the empirical approach of Zyserman & Fredsøe (1994).

11.2 Sediment Transport Table Axes

The values of the parameters Gamma1 and Gamma2 which control the process of wave breaking according to the model of Battjes & Janssen have to be specified in this menu. Note that the values specified for both parameters should be identical to those used in the wave modelling. Values of Gamma1 between 0.5 and 2.0 and of Gamma2 between 0.3 and 2.0 will be accepted by the menu.

The range of variation of the relevant parameters for sediment transport has to be specified through the first value, the spacing and the number of points along each of the axes of the sediment transport table. This has to be done for the root-mean-square wave height H_{rms}, the depth-averaged current speed V, the peak wave period T_p, the wave height-to-water depth ratio H_{rms}/D , the median grain size d_{50} and the sediment grading, defined as $(d_{84}/d_{16})^{0.5}$.

If the effect of the bed slope on the sediment transport has been included in the calculations, then two additional axes have to be defined, the first one corresponding to the bed slope in current direction and the second one to the bed slope perpendicular to the direction of the current.

Along the grain size axis, the Next Value is found as
Previous Value * Spacing; along all other axes the second value is found as Previous Value + Spacing.

At least 1 point must exist along every axis. The spacing must be always > 0, except for the Grain Size axis, for which it has to be > 1. The First Value along the current speed axis must be less than or equal to the spacing.

11.3 Output File

In this menu you specify the path and the name for the ASCII output data file(s) where the calculated transport rates will be saved. You can either type the requested information, or browse your computer to select the path and the name of the file(s).

Note that an output file with the selected name and extension ".dat" will be created if the effect of the bed slope on sediment transport has not been accounted for, whereas two ASCII files having the same name and extensions ".dat" and ".bsl" will be created if the effect has been included in the calculations. The extension(s) will be added automatically by the tool for generation of transport tables.

If the selected output file(s) already exist(s), you will be prompted whether you want to overwrite it/them or not.

11.4 Overview

The overview page presents you with a summary of all the input parameters you have specified. You should inspect the parameters to check they are correct, - and if not go back and change them.

From the page you can either *Finish* the set-up, which will transfer your set-up definition to the current toolbox file, or execute the set-up.

12 Generation of Q3D Sediment Transport Tables

Welcome to the MIKE 21 Tool for generation of Q3D sediment transport tables. The transport tables generated by this tool contain sediment transport rates calculated under the combined effect of waves and current for user-selected ranges of variation of the hydrodynamic and non-cohesive sediment transport parameters.

The tabulated transport rates are calculated using the sediment transport program STPQ3D.

STPQ3D is a detailed intra-wave sediment transport program that uses a Q3D description of the hydrodynamics and the sediment transport to calculate the total sediment transport rates in two perpendicular directions: long-shore and cross-shore. STPQ3D accounts for the effects of combined waves and current, arbitrary angle between the waves and the current, breaking/non-breaking waves, plane or ripple covered bed, uniform or graded sediment, when calculating the sediment transport rates.

For additional details about STPQ3D, reference is made to the scientific description available via the MIKE 21 or the MIKE 3 Documentation Index from *MIKE 21 & MIKE 3 Flow Model FM, Sand Transport Module, Scientific Documentation*.

The sediment transport rates are tabulated in two ASCII data files. The first one has extension ".lon" and stores the sediment transport rates in current direction, which is assumed to be coincident with the longshore direction. The other transport table has extension ".crs" and stores the calculated sediment transport rates in cross-shore direction, i.e. the direction perpendicular to the mean current.

Transport rates in the tables are given in $m^3/s/m$ **solid volume**, i.e. without accounting for the porosity of the bed material.

12.1 General Parameters

On this page you have to specify the values of the general parameters for the calculation of the sediment transport rates. These parameters are the tolerance in calculation of concentration of suspended sediment, the maximum number of wave periods the STP program will iterate over in the search for a periodic solution for the suspended concentration, the number of time steps per wave period to be used in the intra-wave calculation of sediment transport, the density of the sediment relative to that of water, the critical value of the Shields parameter for initiation of motion and the water temperature in degrees Celsius.

The expected range of variation for these parameters is given in what follows.

A value between 10^{-6} and 10^{-4} is expected for the tolerance. The smaller the value, the longer the time needed to generate the transport table.

The maximum number of wave periods must exceed 70, but preferably be above 300. Note that a larger value may result in more time being required for the generation of the transport table, but also in increased accuracy.

The number of time steps per wave period should be kept between 100 and 200, with larger values resulting in longer time required for the table to be generated.

The relative density of the sediment may be chosen between 1.10 and 3.65, with 2.65 the usual value for sand.

The critical value of the Shields parameter may be between 0.045 and 0.060 when using the deterministic approach for calculation of bed concentration. When using the empirical approach the model use a hard-coded value of 0.045.

The water temperature may lie between 5 and 50 degrees Celsius.

12.2   Additional Parameters

This menu is used to define which effects should be taken into account by STPQ3D when calculating the sediment transport rates in the tables.

You have to define whether the influence on the sediment transport rates of wave-generated ripples, streaming in the wave boundary layer and bed slope are to be accounted for or not. You can also choose whether the bed concentration of suspended sediment is to be calculated according to the deterministic model of Engelund & Fredsøe (1976) or by use of the empirical approach of Zyserman & Fredsøe (1994). For some cases it is preferable to consider only the sediment transport in the direction of the mean current. For this purpose it is possible to omit the transport component perpendicular to the direction of the mean current.

Wave-generated ripples influence the bed roughness, the bed concentration of suspended sediment and the turbulence levels close to the bed.

The bed slope affects the calculated rates of bed-load transport. Both longitudinal and transverse slopes may be included in the simulations. However, the correction of the bed transport rates due to bed slop effects is normally very small, and therefore only relevant in connection with morphological simulations.

In the case of progressive waves, a small net shear stress is generated in the direction of wave propagation due to the non-uniformity of the wave boundary layer in this direction. This effect is called streaming in the wave boundary layer and is a second-order one. Streaming is important when calculating

cross-shore transport rates outside the surf zone, especially in connection with morphological simulations.

12.2.1 Advanced

It is possible to include or exclude the effects of more parameters than is accessible by the user interface. These parameters and can only be changed by modifying the input file manually in an ascii editor. The parameters are written as follows in the input file:

- Include_Undertow
  Include (1) or exclude (0) the shear stress associated with wave break-ing.
  Per default Include_Undertow = 1.

- Include_advective_terms
  Include (1) or exclude (0) the sediment transport due to the stokes drift, i.e. the mass transport between wave trough or wave crest (Eul)
  Per default Include_advective_tems = Include_Streaming_Terms

- Include_return_current
  Include (1) or exclude (0) the wave induced mass transport, the dis-charge in the surface rollers and the wave induced streaming in the cal-culation of the mean flow. The gradients in the water surface elevations in the x and y directions are not known beforehand and are found through iteration to obtain the prescribed fluxes.
  Per default Include_return_current = 1

12.3 Wave Parameters

In this section you have to specify the wave theory describing the wave-induced near-bed velocities above the wave boundary layer. Use of non-lin-ear wave theories is important to resolve wave asymmetry and skewness, which are important for sediment transport outside the surf zone.

As no wave theory covering all hydrodynamic conditions satisfactorily exists today, a number of the existing ones have been included in STPQ3D. The available theories and their area of application are:

- Stokes 1st, 3rd and 5th order wave theory. Deep water, non-breaking waves.

- Cnoidal 1st, 3rd and 5th order wave theory. Shallow water, non-breaking waves.

- Vocoidal theory. All water depths, non-breaking waves.

- Isobe and Horikawa semi-empirical theory. All water depths, breaking and non-breaking waves.

- Doering and Bowen semi-empirical theory. All water depths, breaking and non-breaking waves.

The program will internally check whether the selected wave theory is valid for the given conditions and, if not, choose an alternative theory.

The values of the parameters Gamma1 and Gamma2 which control the process of wave breaking according to the model of Battjes & Janssen have to be specified in this menu. Note that the values specified for both parameters should be identical to those used in the wave modelling. Values of Gamma1 between 0.5 and 2.0 and of Gamma2 between 0.3 and 2.0 will be accepted by the menu.

12.4 Sediment Transport Table Axes

The range of variation of every parameter relevant for sediment transport has to be specified through the first value, the spacing and the number of points along each of the axes of the sediment transport table. This has to be done for the root-mean-square wave height H_{rms}, the depth-averaged current speed V, the peak wave period T_p, the wave height-to-water depth ratio H_{rms}/D , the angle between the current and the waves, the median grain size d_{50} and the sediment grading, defined as $(d_{84}/d_{16})^{0.5}$.

If the effect of bed slope on the sediment transport has been included in the calculations, then two additional axes have to be defined, the first one corresponding to the bed slope in current direction and the second one to the bed slope perpendicular to the direction of the current.

Along the grain size axis, the Next Value is found as
Previous Value * Spacing; along all other axes the second value is found as
Previous Value + Spacing.

At least 1 point must exist along every axis. The spacing must be always > 0, except for the Grain Size axis, for which it has to be > 1. The First Value along the current speed axis must be less than or equal to the spacing.

If the effect of centrifugal acceleration (V^2/R) on the sediment transport has been included in the calculations, then an additional axis has to be defined. The first value value is per default set to 0 (defining no centrifugal acceleration), and the remaining values is found by adding the Spacing (m/s^2).
The definition of the first value in this dialog page is the minimum helical radius (R) of the flow. This value should not be lower than the minimum grid size in the bathymetry you want to use later.

12.5 Output File

In this menu you specify the path and the name for the ASCII output data files where the Q3D sediment transport tables will be saved. You can either type

the requested information, or browse your computer to select the path and the name of the files.

Note that two output files will be created. The first one will have the selected name and extension ".lon", where the calculated sediment transport rates in longshore direction will be saved. The second file will be named as the first one but will have extension ".crs" and will store the transport rates in cross-shore direction. The extensions will be added automatically by the tool for generation of Q3D sediment tables.

If one or both of the selected output files already exists, you will be prompted whether you want to overwrite it/them or not. **Note that no warning message will be issued if the tool is executed in batch mode** (i.e. from a DOS prompt).

12.5.1 Example of output

Both output data files consist of a Header that starts with a listing of the parameters used and a description of the dimension of the transport table. The Body with the actual transport table values follows next in a number of blocks.

Header

Figure 12.1   Example of header in table

Body
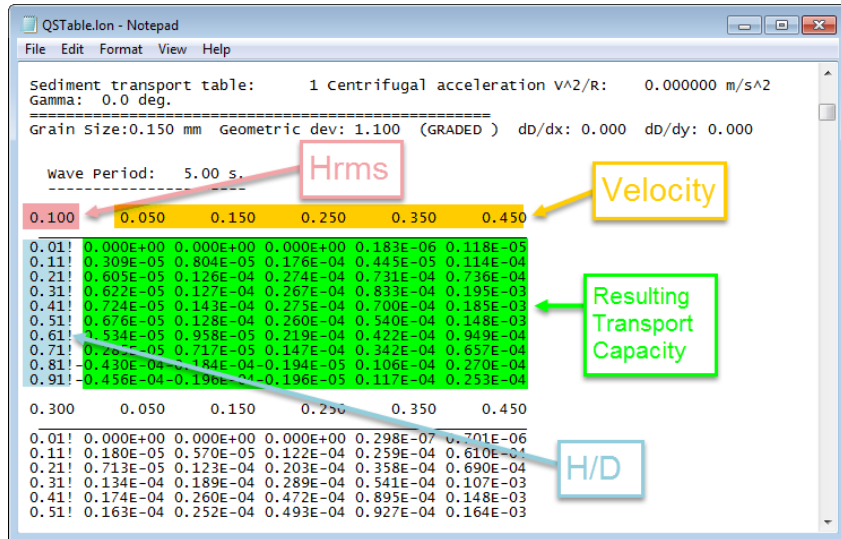
Figure 12.2   Example of detail of body in table

12.6   Overview

The overview page presents you with a summary of all the input parameters you have specified. You should inspect the parameters to check they are correct, - and if not go back and change them.

From the page you can either *Finish* the set-up, which will transfer your set-up definition to the current toolbox file, or execute the set-up.

NOTE: Due to parallelisation in the model execution, the progress bar is only updated after a major block of data has been calculated, corresponding to all combinations of input velocity, H/D, wave height and wave period.

13 Integration of Storms

Welcome to the MIKE 21 Tool for integration of sediment transport results corresponding to various storms. With this tool you can obtain the weighted sediment transport field and the weighted rates of bed level change on the basis of the results found by use of the non-cohesive sediment transport module of MIKE 21 or the Sand Transport module of MIKE 21 Flow Model FM for a number of hydrodynamic events or storms.

The results are integrated by assigning weights to the different events for which sediment transport calculations have been performed. For example, a yearly-averaged sediment transport field can be calculated in this way.

You can integrate sediment transport results calculated for combined current and waves or for pure current only.

13.1 Specification of Events to Integrate

In this page you specify the names of the data files to be integrated. You can either type the path and the file name or browse your disk by left-clicking on the button to the right of the Name field.

You may select either dfs2 files or dfsu files. The files to be integrated must have only one time step each and the size of the grid must be the same. When using dfsu files, the number of items in each dfsu file must be the same.

For each data file you must specify the three items that holds the values of the sand transport in the two main directions, ps and qs, and the bed level update rate, dz/dt, respectively. Per default the values are set to 1, 2 and 3, corresponding to the standard output file from the MIKE 21 ST model.

At least 2 and a maximum of 12 events can be processed. If you want to specify more than 2 events, just press the Tab key to open additional lines in this menu. If you opened too many new lines, you can get rid of the unneeded ones by left-clicking with the mouse on the button with the line number to highlight the empty line and then pressing Delete.

13.2 Storm Weights

The weights to be attached to each event have to be specified in this menu. The weights can be given in percent (e.g. yearly duration), through the actual number of days per year each event is happening, or even in a relative fashion (if Event 1 takes place twice as many times as Event 2 and three times the duration of the calm period, you can specify weights of 6, 3 and 2, respectively, or even 12, 6 and 4).

In any case, the weights are normalised in such a way that the sum of all the weights (including the calm period) yield 1.

The calm period corresponds to situations for which no or negligible sediment transport takes place.

You can choose to multiply the integrated results by the factor indicated at the bottom of this menu in case that the integrated transport rates are too small or too large.

13.3 Output File

In this menu you specify the path and the name of the file where the integrated results will be saved. You can either type the requested information or browse your disk by left-clicking with your mouse on the button to the right of the Name field. A title can also be included in the output file.

The output file will contain three items containing the integrated values of item1, item 2 and item 3. The item names will be created based on the item names given in input data file number 1.

13.4 Overview

The overview page presents you with a summary of all the input parameters you have specified. You should inspect the parameters to check they are correct, - and if not go back and change them.

From the page you can either *Finish* the set-up, which will transfer your set-up definition to the current toolbox file, or execute the set-up.

14   Calculation of Sediment Discharge

Welcome to the MIKE 21 tool for calculation of sediment discharges.

With this tool you can calculate the instantaneous and cumulative total transport through a cross section at user-specified time intervals using a MIKE 21 ST result file as input.

The result file from the non-cohesive sediment transport simulation may contain either a time series of sediment transport rates or transport rates averaged over the simulation period.

14.1   Input File

In this page you specify the name of the data file where the MIKE 21 ST results have been saved. You can either type the path and the file name or browse your disk by left-clicking on the button to the right of the Name field.

The input data file may contain instantaneous sediment transport components per unit width P_s and Q_s, i.e. 2 items, or time-averaged transport components per unit width (P_s and Q_s) and initial rates of bed level change dz/dt, i.e. 3 items.

14.2   Definition of Cross Section

The cross section through which the total sediment transport rate will be calculated is defined by at least 2 and at most 8 points.

The position of the points that define the section must be specified in grid coordinates with the points along it in the right sequence. The section may extend over land areas, see.

The total transport will be taken as positive when it crosses the selected section from left to right.

If the net supply of sediment into a closed area is to be assessed, and the coordinates of the points defining the area are specified in clockwise sequence, then positive transport values will correspond to net deposition taking place over the area and conversely.
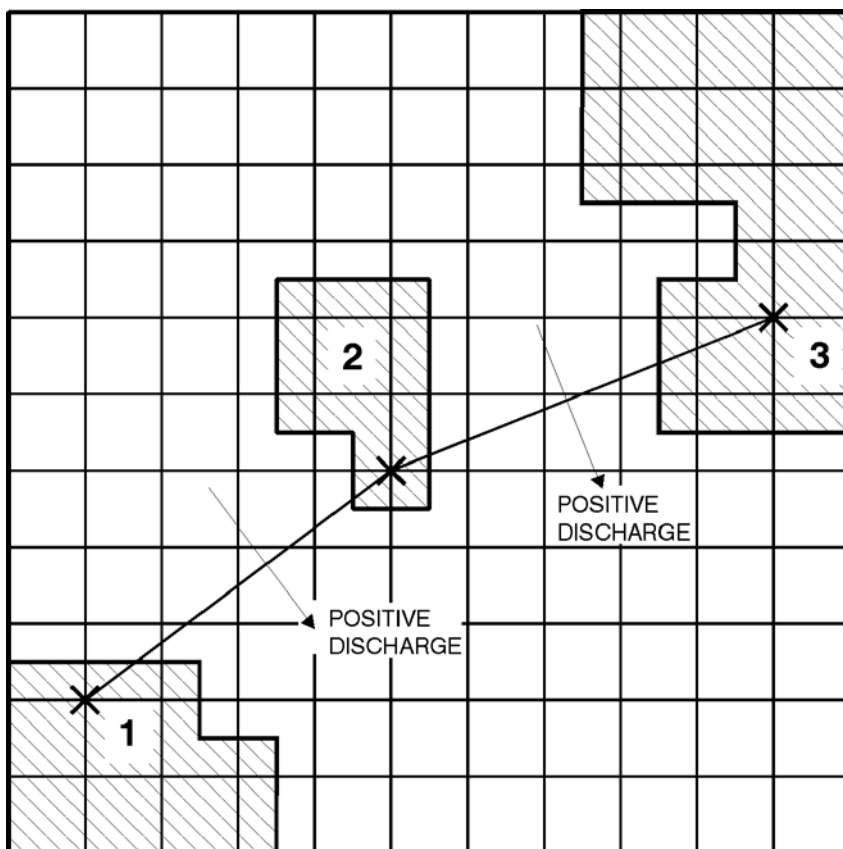
Figure 14.1   Definition of a cross-section

14.3   Subseries Selection

The first and last time steps in the data file that will be included in the calculation have to be specified in this menu, as well as the time interval (in number of time steps) between consecutive integrations of the transport.

14.4   Output File

In this menu you specify the path and the name of the time series file where the total transport rates will be saved. You can either type the requested information or browse your disk by left-clicking with your mouse on the button to the right of the Name field. A title can also be included in the output file.

The output file will contain two items: the total sediment transport across the section (in m^3/s or m^3/year) and the cumulative sediment discharge (in m^3).

14.5 Overview

The overview page presents you with a summary of all the input parameters you have specified. You should inspect the parameters to check they are correct, - and if not go back and change them.

From the page you can either *Finish* the set-up, which will transfer your set-up definition to the current toolbox file, or execute the set-up.

M21 TOOLS: SEISMOLOGY

The SEISMOLOGY Tools in the M21 Toolbox include the facility:

- Earthquake Bathymetry Adjustment

15 Earthquake Bathymetry Adjustment

The Earthquake Bathymetry Adjustment tool provides a user-friendly way of generating bathymetries or initial water surface levels based on vertical displacements due to earthquakes with the help of an input file, earthquake parameters and Lamé coefficients.

15.1 General Introduction

The Earthquake Bathymetry Adjustment tool generates vertical displacements in the bathymetry or in the water surface elevation caused by earthquakes, which can be used to model tsunamis.

There are a number of methods available to calculate such deformations. Widely used is Okada's Double couple method (Okada, 1985 /9/). Here, a homogeneous isotropic elastic earth's crust is assumed. Its advantage is that all required parameters are available within minutes in the Internet. Furthermore, different scaling laws have been implemented to account for different types of earthquakes. They are used to calculate fault dimensions, which are used within Okada's method.
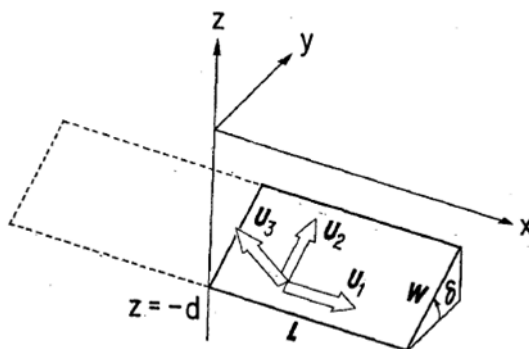
Figure 15.1   General scheme of earth quake dynamic (Okada, 1985 /9/)

15.2 Input Data

On this page you specify the input file. The input file item may be one of three item types:

- Bathymetry
- Water Level
- Surface Elevation

In case that the item type is "Bathymetry", the resulting adjusted file consists of two time steps. The first represents the input bathymetry; the second represents the adjusted bathymetry. It can be used as time-varying bathymetry together with the land-slide option in MIKE 21. In this way it is possible to easily set up a simulation in which the earthquake occurs later than at the beginning.

In case that the item type is "Water level" or "Surface Elevation" the resulting adjusted file consists of one time step and can be used as initial surface elevation in the hydrodynamic model.

15.3 Earthquake Parameters

For characterising an earthquake, there exist numerous parameters which are distributed in the Internet. On this page the first set of input parameters for the scaling laws and the double-couple method (Okada, 1985 /9/) have to be specified.

The parameters can be found at the websites of various geophysical institutes, e.g. USGS (http://earthquake.usgs.gov/earthquakes/) or GFZ (http://geofon.gfz-potsdam.de/eqinfo/list.php)....

15.3.1 Rupture time

Here the initial rupture time at which the earthquake occurred has to be specified. This is important especially in case of more than one earthquake during the hydrodynamic simulation, when one uses time-variable bathymetries.

15.3.2 General parameters
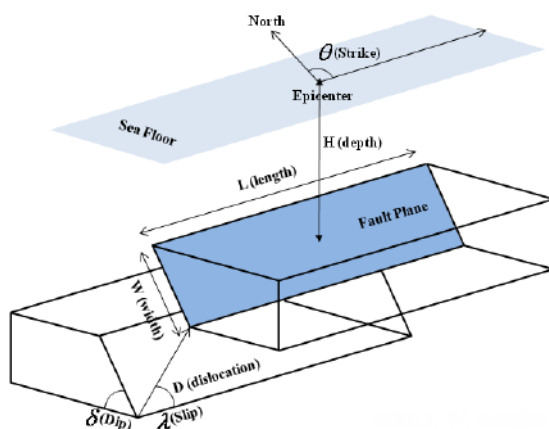
Figure 15.2   Diagram of earthquake fault parameters (Kongko, 2011)

Dip angle δ

δ is the angle of descent of one plate relative to the horizontal plane, measured in degrees (°). The angle lies in a vertical plane which is orientated 90° to the strike angle. The angle is 0° in horizontal direction and increases clockwise.

Slip angle λ

λ is the angle of the dislocation of one plate relative to the other in degrees (°), lying in the contact plane of both plates. The angle is 0° in horizontal direction, pointing into the strike direction. It increases anticlockwise.

Strike angle θ

θ is the horizontal angle of the contact plane between both plates. The angle is zero when pointing Northwards and increases clockwise (Eastwards).

Depth H

H is the vertical position of the rupture relative to the epicentre in kilometres (km). It is 0 km at the earth's crust surface and increases downwards.

Latitude and Longitude

This gives the horizontal position of the epicentre in World Geodetic System (WGS1984) in degrees (°).

15.3.3   Mw/Fault dimensions

You may choose between four different fault mechanisms in the drop-down list:

- Crustal strike-slip (Wells and Coppersmith, 1994 /17/),
- Crustal dip-slip and (Wells and Coppersmith, 1994 /17/),
- Subduction zone dip-slip (Papazachos et al., 2004 /10/)
- No Mw.

The first mechanisms describe ruptures within a single plate, the latter describes ruptures between two plates.

If in case of crustal earthquakes there exist uncertainties about the fault mechanism, the crustal dip-slip mechanism should be selected (Wells and Coppersmith, 1994 /17/). Based on the here taken decision the fault dimensions Length L, Width W and vertical Slip will be determined using the appropriate scaling law, as cited. In such a case, the user is required to give the moment magnitude MW of the earthquake as well.

In case that different scaling laws than the implemented ones are preferred, "No Mw" might be selected. The user can then give L, W and u independently to specify the Dimension of fault directly. MW does not have to be given, because it is only used to calculate the fault dimensions.

Dimension of fault

Three parameters define the dimension of the fault.

Length L
The Length describes the length of the fault surface in the horizontal.

Width W
The width describes the width of the fault surface moving.

Slip
The slip gives the mean vertical displacement on the fault area.
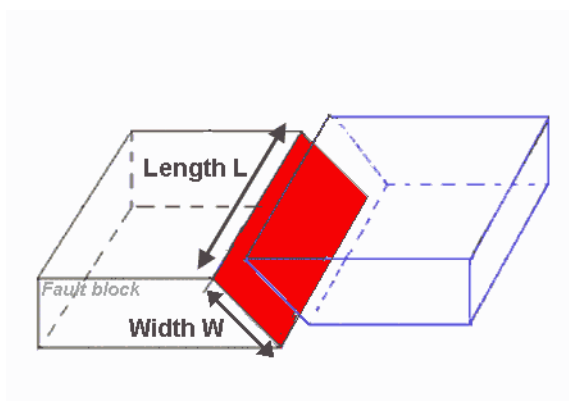
Figure 15.3   Diagram of fault dimension

15.4 Lamé Coefficients

The two Lamé coefficients describe physical characteristics of the earth's crust.

Shear Modulus µ
µ or rigidity is the measure of how an elastic body deforms due to forces parallel to one of its surfaces. It is specified in Giga Pascal (GPa).

Lambda λ
λ has no physical meaning, but can be derived from shear stress and Poisson rati o. It is given in Giga Pascal (GPa).

The values for shear modulus and lambda depend on the depth of the rupture and can be taken from the four-layer half-space model of the earth (Wang et al., 2003 /16/), given in Table 15.1.

Table 15.1   Four-layers half-space model (Wang et al., 2003)

Depth [km]	Shear modulus μ [GPa]	Lambda λ [GPa]
0-1	15.6	23.6
1-13	29.4	25.9
13-30	39.7	32.1
>=30	69.8	66.4

15.5   Output Surface Elevation

On this page you specify the name and location of the resulting file with the adjusted surface elevation for a single fault.

In case that the input file item is of item type "Water level" or "Surface Elevation", the result file will contain vertical deformations saved as the same item type in one time step.

15.6   Output Bathymetry

On this page you specify the name and outline of the resulting file with the time-varying bathymetry file.

In case the input file item type is "Bathymetry", the result will contain vertical deformations saved as the same item type in two time steps. The first time step contains the original bathymetry, while the second comprises of the resulting bathymetry, which includes the vertical deformations. Furthermore, there is a file generated which contains the difference between the values of the input and the result file.

15.7   Overview

The overview page presents you with a summary of all the input parameters you have specified. You should inspect the parameters to check they are correct, - and if not go back and change them.

You have the option to specify the location of an eventual log-file from the conversion. If the conversion is successful no log-file is created, otherwise a logfile with a list of the files not matching the template is saved.

From the page you can either **Finish** the set-up, which will transfer your set-up definition to the current toolbox file, or **Execute** the set-up.

15.8   References

/7/   Kongko, W. (2011). South Java Tsunami Model Using Highly Resolved Data and Probable Tsunamigenic Sources, Leibnitz-Universität Hannover, Hanover, 314 pp.'

/8/   Nettles, M. and Ekstrom, G., (2006). Magnitude 7.1 Taiwan Region Haward Moment Tensor Solution. USGS

/9/   Okada, Y. (1985). Surface deformation due to shear and tensile faults in a half-space. Bulletin of the Seismological Society of America, 75(4): 1135-1154.

/10/   Papazachos, B.C., Scordilis, E.M., Panagiotopoulos, D.G., Papazachos, C.B. and Karakaisis, G.F. (2004). Global relations between seismic fault parameters and moment magnitude of earthquakes. Bulletin of the Geological Society of Greece, 36: 1482-1489.

/11/   USGS, 2006. Magnitude 7.1 Taiwan Region Earthquake summary. USGS, http://earthquake.usgs.gov/earthquakes/event-page/usp000f114#executive.

/12/   USGS, 2009a. Magnitude 8.1 Samoa Island Region USGS Centroid Moment Tensor Solution, USGS

/13/   USGS, 2009b. Magnitude 8.1 Samoa Islands Region Earthquake Summary. USGS, http://earthquake.usgs.gov/earthquakes/event-page/usp000h1ys#executive.

/14/   USGS, 2011a. Magnitude 9.0 Near the East Coast of Honshu, Japan Earthquake Summary. USGS, http://earthquake.usgs.gov/earthquakes/eventpage/official20110311054624120_30#executive.

/15/   USGS, 2011b. Magnitude 9.0 Near the East Coast Of Honshu, Japan USGS Centroid Moment Tensor Solution. USGS

/16/   Wang, R., Martín, F.L. and Roth, F. (2003). Computation of deformation induced by earthquakes in a multi-layered elastic crust - FORTRAN programs EDGRN/EDCMP. Computers & Geosciences, 29: 195-207.

/17/   Wells, D.L. and Coppersmith, K.J. (1994). New empirical relationships among magnitude, rupture length, rupture width, rupture area, and surface displacement. Bull. Seism. Soc. Am., 84(4): 974-1002.

15.9 Examples

Tsunami modeling is an interdisciplinary topic, in which the first task in the chain is earthquake modeling. There exist a wide range of methods which can be applied. This is often difficult, especially when missing special knowledge in geology or seismology. In order to facilitate it, three examples are provided and explained.

Among available methods, the easiest approach to deal with it is to assuming that the earthquake causes one single rupture. Further, one assumes that the earth consists of an isotropic elastic body. Its mechanical behavior is described with the help of two parameters: shear modulus or rigidity and lambda. Their values depend on the depth of the earthquake (Wang et al., 2003). They are given in the help files. Furthermore, parameters to describe particular earthquake events have to be specified. This is briefly described for the following earthquakes in the next sections.

- 26 December 2006 Taiwan Region, MW=7.1
- 29 September 2009 Samoa Islands Region, MW=8.1
- 11 March 2011 Near the East Coast of Honshu, Japan, MW=9.0.

15.9.1 26 December 2006 Taiwan Region, MW=7.1

Description

The earthquake occurred 90km SSE of Kao-hsiung, Taiwan and 795km N of Manila, Philippines at 8:26am local time in Taiwan. At least one person was killed and three injured at P'ing-tung. The earthquake was felt throughout Taiwan and along the coast of southeastern China from Shantou in the south to Fuzho in the north. According to USGS webpage, the normal faulting focal mechanism suggests that this shock occurred as a result of intraplate stresses within the subducting Eurasian plate (USGS, 2006 /11/). One option to model it is to use the scaling laws valid for crustal dip-slip faults.

Parameters

The parameters to model this earthquake have been taken from the Havard Moment Tensor Solution (Nettles and Ekstrom, 2006 /8/) and are summarised in Table 1 and are included in the setup file 2006Tw.t21. The setup file uses the grid file 2006TwUtm51N.dfs2, which contains the item "surface elevation".

.

Table 15.2   Parameters for the 26 December 2006 Taiwan Earthquake (Nettles and Ekstrom, 2006 /8/)

Parameter	Value
Initial rupture time [hh:mm:ss]	12:26:27 UTC
Dip Angle [°]	148
Slip Angle [°]	86
Strike Angle [°]	335
Depth [km]	22.5
Latitude [°]	21.38
Longitude [°]	120.39
Fault mechanism	Crustal dip-slip
Mw [-]	7.0
Shear Modulus [GPa]	39.7
Lambda [GPa]	32.1

For longitude and latitude of the earthquake, the centroid location has been used, the dip, slip and strike of nodal plane 1 (NP1) have been selected. The vertical deformations due to this earthquake have been calculated to be very small.

Result

The resulting disruption of the initial water surface is shown in Figure 15.4.
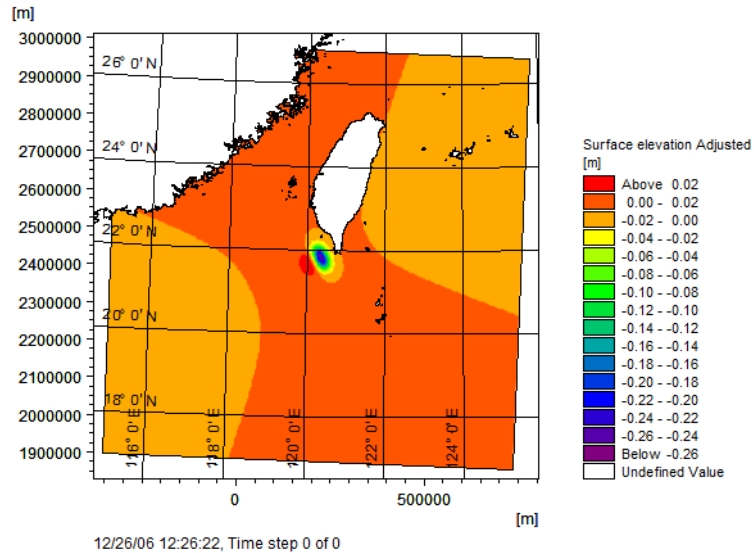
Figure 15.4   Resulting water surface disruption in Taiwan region.
Land areas shown for reference.

15.9.2 29 September 2009 Samoa Islands Region, MW=8.1

Description

The earthquake occurred 185km NEN of Hihifo, Tonga and 190km S of Apia, Samoa at 6:48am local time in Samoa. 192 people were killed in Samoa, American Samoa and Tonga. It was felt in Apia, Samoa, at Ili'Ili and Tafuna, American Samoa, northern Tonga and Wallis and Futuna Islands. According to the USGS webpage, it has been inferred that the earthquake occurred as a normal fault rupture on or near the outer rise of the subducting Pacific Plate (USGS, 2009b /13/). Therefore a scaling law is used in this example which is valid for subduction zone dip-slip faults.

Parameters

The parameters to model this earthquake have been taken from the USGS Centroid Moment Tensor Solution (USGS, 2009a /12/) and are summarised in Table 15.3 and are included in the setup file 2009Samoa.t21. The setup file uses the grid file 2006SamoaUtm60S.dfs2, which contains the item "Bathymetry", see Figure 15.5.

Table 15.3   Parameters for the 29 September 2009 Samoa Earthquake (USGS, 2009a /12/)

Parameter	Value
Initial rupture time [hh:mm:ss]	12:48:52 UTC
Dip Angle [°]	52
Slip Angle [°]	-61
Strike Angle [°]	345
Depth [km]	10
Latitude [°]	-15.466
Longitude [°]	-172.49
Fault mechanism	Subduction dip-slip
Mw [-]	8.0
Shear Modulus [GPa]	29.4
Lambda [GPa]	25.9

For longitude and latitude of the earthquake, the centroid location has been used, the dip, slip and strike of nodal plane 1 (NP1) have been selected.

Figure 15.5   Initial bathymetry for Samoa region

Result

The resulting disruption of the bathymetry is shown in Figure 15.6.

09/29/09 10:38:48, Time step **1** of **1**
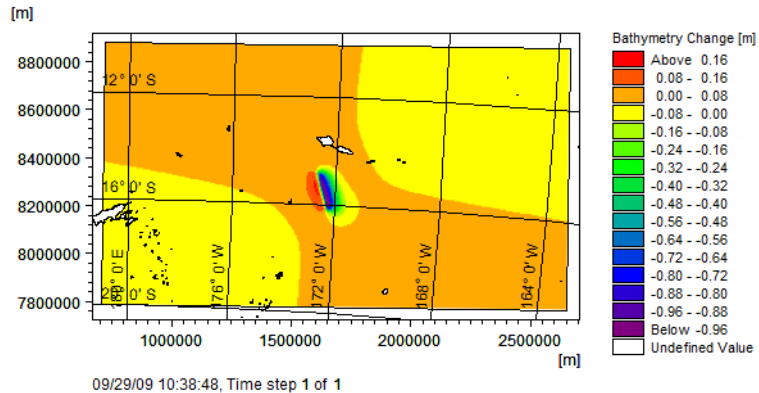
Figure 15.6   Resulting bathymetry change in Samoa region.
Land areas shown for reference.

15.9.3   11 March 2011 Near the East Coast of Honshu, Japan, MW=9.0

Description

The earthquake occurred 128km E of Sendai, Japan and 373km NE of Tokyo, Japan March 11th 2011 at 2:46pm local time in Japan. 15,703 people were killed, 4,647 people are missing, 5,317 injured and 130,927 displaced. The earthquake was felt in Japan from Hokkaido to Kyushu, northern Mariana Islands, North Korea, South Korea; Taiwan, northeastern China and south-eastern Russia as far as Kaohsiung, Beijing and Petropavlovsk-Kamchatskiy. Seiches were observed at Leikanger, Norway and water fluctuations were observed in a well in Newfoundland, Canada. The tsunami had maximum runup heights of 29.6m in Ofunato, 18.4m at Onagawa and 9.4m at Iwaki. According to the USGS webpage, the earthquake resulted from thrust faulting on or near the subduction zone plate boundary between the Pacific and the North American plates (USGS, 2011a /14/). Therefore a scaling law which is valid for subduction zone dip-slip faults is used in this example.

Parameters

The parameters to model this earthquake have been taken from the USGS Centroid Moment Tensor Solution (USGS, 2011b /15/). They are summarised in Table 15.4 and are included in the setup file 2011Jpn.t21. The setup file uses the grid file 2011JpUtm54N.dfs2, which contains the item "surface elevation".

.

Table 15.4   arameters for the 11 March 2011 Japan Earthquake (USGS, 2011b /15/)

Parameter	Value
Initial rupture time [hh:mm:ss]	05:47:47
Dip Angle [°]	77
Slip Angle [°]	95
Strike Angle [°]	29
Depth [km]	10
Latitude [°]	38.486
Latitude [°]	142.597
Fault mechanism	Subduction dip-slip
MW [-]	9.0
Shear Modulus [GPa]	29.4
Lambda [GPa]	25.9

For longitude and latitude of the earthquake, the centroid location has been used, the dip, slip and strike of nodal plane 1 (NP1) have been selected.
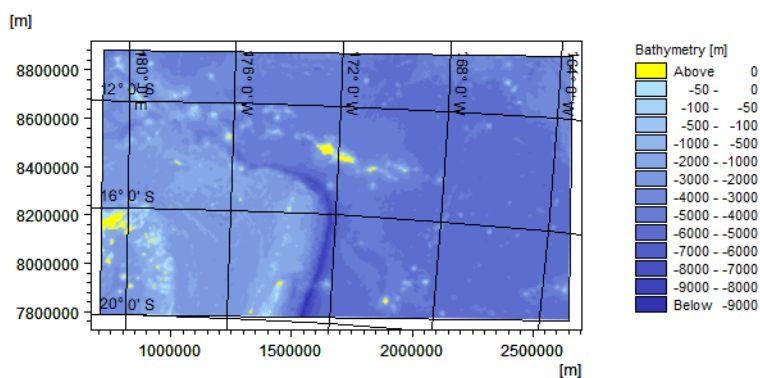
Result

The resulting disruption of the initial water surface is shown in Figure 15.7.
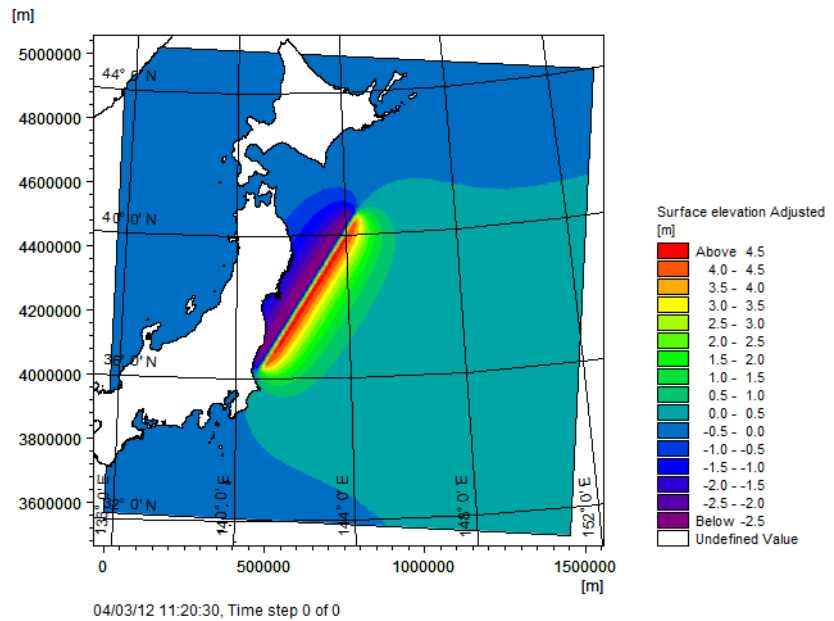
Figure 15.7   Resulting water surface disruption near coast of Japan.
Land areas shown for reference.

M21 TOOLS: TIDAL

The Tidal Tools in the M21 Toolbox include the facilities:

- Tidal Analysis of Currents
- Tidal Analysis of Heights
- Tidal Prediction of Currents
- Tidal Prediction of Heights

16 Tidal Tools

16.1 General Introduction

The MIKE 21 Toolbox comprises the following four programs for the analysis and prediction of tidal heights and currents:

- Tidal Analysis of Heights
- Tidal Analysis of Currents
- Tidal Prediction of Heights
- Tidal Prediction of Currents

These programs constitute a necessary tool when dealing with coastal engineering problems, in particular regarding the definition of boundary conditions and the calibration and validation of hydrodynamic models, as well as the long term prediction of tidal levels and currents. They are also fundamental for the interpretation of large scale circulation processes, through the calculation of cotidal and corange lines and the analysis of the individual propagation of tidal constituents across the study zones.

The programs provide the user with the option of prediction/analysis based on either the Admiralty Method or the IOS method.

A Constituent Data Package with standard constituent input data files needed for the execution of the program are included in the installation. The file contains information on frequencies, Rayleigh comparison pairs, astronomical variables, Doodson numbers and satellite constants, and data on shallow water constituents.

16.2 The Origin of Tide

The short explanation to the origin of tide is, that the tide is effect of gravity from forces the moon (and the sun) on the water in the oceans. This is, however, only half of the explanation as it only explains one high tide a day, one for each passage of the moon.

The full explanation is that Tide is caused by the forces involved in the moon's orbit around the earth.

16.2.1 Influence of the moon
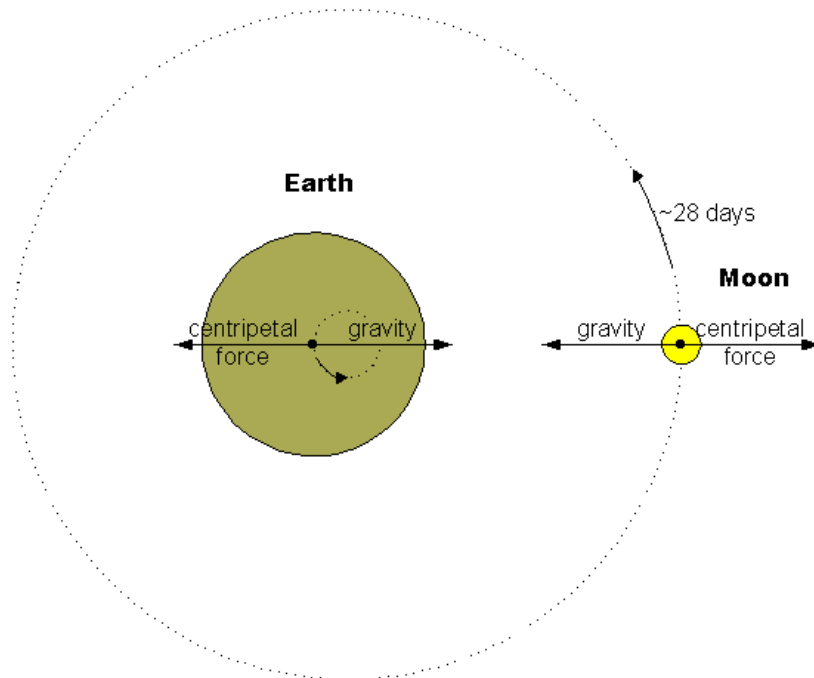
Figure 16.1   Equilibrium of forces in the moons 28 day orbit around the earth

As the moon orbits around the earth, there is equilibrium between the gravity from the earth and the centripetal "force" (which the equivalent to the acceleration involved in constantly changing direction of the velocity of the moon).

The same equilibrium exist for the forces on the earth: although as a first approximation the earth is stationary while the moon orbits around it, this is not possible as then no forces exist to balance out the gravity force from the moon. In reality the earth also orbits, and the gravity from the moon is balanced out by the centripetal force in the earth's small orbit "around the moon", or more specifically: around the centre of the moons orbit.

This balance of the total forces on the earth is the net sum of the forces on the individual parts of the earth, but for each part these forces do not balance:

- In areas closest to the moon and also closest to the centre of the earth's orbit, gravity forces from the moon dominate.

- In areas furthest away from the moon and also furthest away from the centre of the earth's orbit, centripetal forces dominate.

Although gravity from the earth takes care of the equilibrium of forces on each individual volume (so that no part escapes the earth towards the moon and

away from the moon), this imbalance in gravity and centripetal forces results in higher water level in areas closest to the moon and away from the moon.
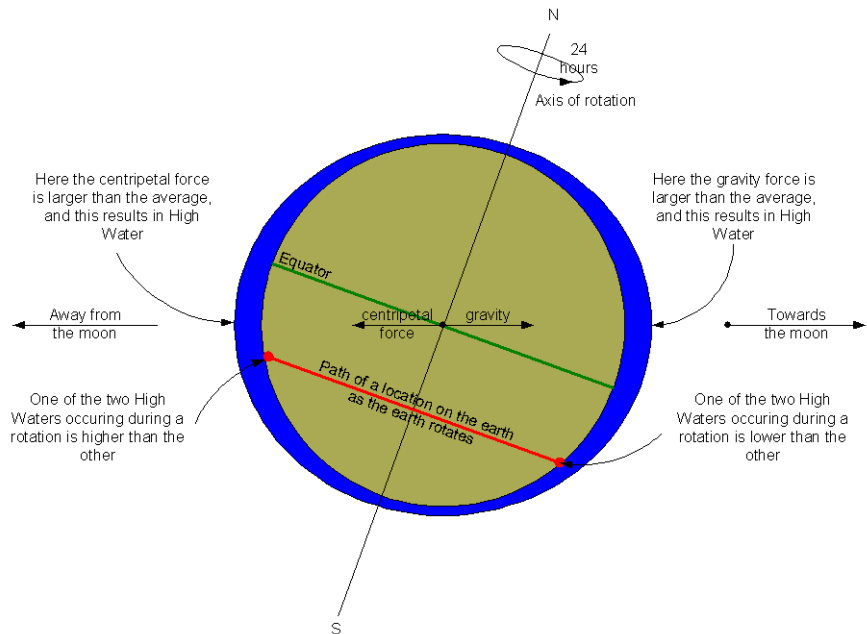
Figure 16.2   Semi-diurnal tide as the earth rotate.

In Figure 16.2 it is seen that during one rotation of the earth (~ 24 h 50 min.) a particular point on earth moves through two High-waters. It is also seen that, because the axis of rotation is tilted relative to the plane in which the moon orbits, one of the two High-waters is typically higher.

16.2.2   Influence of the sun

In addition to the moon, the gravity from the sun (and centripetal forces in the earth's orbit around the sun) contributes to the tide.
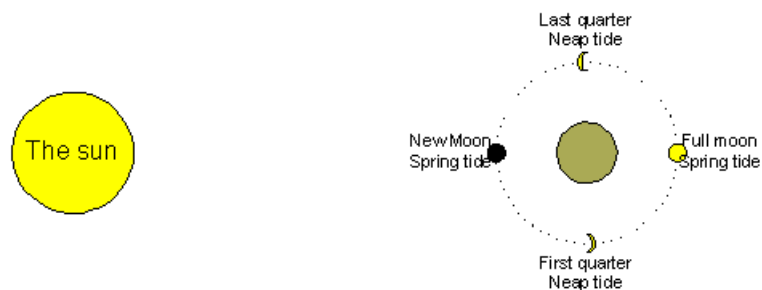
Figure 16.3   Phases of the moon corresponds to the Spring tide Neap Tide cycle.

Twice per lunar orbit (~28 days) the sun and the moon and the earth are in line and the tidal effect of the sun and the moon adds up, creating Spring tide:

- At New Moon (which rises at dawn and sets at dusk)
- At Full Moon (which rises at dusk and sets at dawn)

When the moon is in first and last quarter the tidal effect do not add up, and the tide is relatively low (Neap Tide).

16.2.3 Characteristics of tide

- These simple characteristics of the orbit of the moon and the sun describe, in connection with the rotation of the earth, the main characteristics of the tide:

- Two high/low water cycles per lunar day

- Of which one typically is larger than the other (daily inequality)

- High tidal range every fortnight (spring tide).

- A more detailed description of the tide also takes the following into account:

- Eccentricity of the moon's orbit (27.55 day periodicity). Lower tides occur at apogee (407, 000 km) and higher tides at perigee (357, 000 km).

- Tilt of the Earth in relation to the sun affects the magnitude of the spring tides. The greatest difference in bulges occurs when N. hemisphere is inclined toward or directly away from the sun.

- Equinoxes (Sept. 21 and March 21) spring tides are highest because the moon-earth-Sun lie along a straight line.

- Solstices (June 21 and Dec. 21) spring tides are lower because the Moon-Earth-Sun do not lie along a straight line.

- Proximity to the sun: the sun is closer (perihelion:148,500,000 km) during our northern winter and furthest (aphelion: 152,200,000 km) during our Summer.

The tide is mainly generated in the large masses of water in the deep oceans, although the effect (in terms of water level variations) in deep water is small.

The local tide at the coast and in the shallower waters near the coast is the effect of the amplification of the small tidal wave generated on deep water as it progresses towards shallower areas, and the degree of resonance that the local conditions afford. The tide on shallower water is therefore, in addition to the position of the sun and the moon, depending on the local conditions (sea

bottom topography) experienced by the tidal wave as it progresses. The tide therefore varies very much from location to location.

16.3   Harmonic Description of Tide

16.3.1   Periodicities

The motion of the sun and the moon relative to earth is characterised by the following periodicities:

- Rotation of the Earth (lunar day, 24.8h);

- The orbit of the moon around the Earth (lunar month, 27 days);

- The orbit of the Earth around the Sun (tropical year);

- The periodicity in the orientation of axes the lunar orbit ellipse (i.e. the location of lunar perigee) (8.85 years);

- The periodicity of the lunar orbital tilt (18.6 years);

- The periodicity in the orientation of axes the earths orbit ellipse (i.e. in the location of perihelion) (~21,000 years)

As the tidal response to the motion of the planets mainly contains frequencies present in the forcing, this suggests a description of the tide as a sum of harmonic constituents, the frequencies of which are integer multiples of these 6 fundamental frequencies. The set of 6 signed integers used for description of particular constituent is called the Doodson's numbers of the constituent.

In a harmonic description of tide, the tidal water level variations is thus described as a sum of harmonic constituents, each characterised by a frequency (or period) as follows:

$$\eta(t) = \sum_{i=1}^{N} A_u(t) \; a_i \; \cos(\omega_i t + f_i + F_i(t)) \qquad (16.1)$$

where

$\eta(t)$ is the water level,

ω_i is the frequency of constituent i,

a_i is the amplitude of constituent i,

f_i is the phase of constituent i,

$A_i(t)$ is the amplitude correction to constituent i and

$F_i(t)$ is the phase correction to constituent i.

16.3.2 Constituents

The constituents can be categorised by their period into:

- Diurnal and semi-diurnal constituents (period around 12 and 24 hours)
- Sub-harmonic constituents (period >> 24 hours)
- Super-harmonic constituents (period << 24 hours)
- Seasonal variations.

Main constituents are the Diurnal and Semi-diurnal constituents such as

- M_2 period 12.4 h
- S_2 period 12 h
- K_1 period 23.9 h
- O_1 period 25.8 h

Normally semi-diurnal constituents dominate, and in such locations the effect of diurnal components is that of the two tidal cycles occurring every day one is larger and the other is smaller. In some locations, however, the tide is diurnal with only one high tide and one low tide a day.

The effect of two (or more) constituents with almost the same period is a tide with that period, but with slowly varying amplitude. For example, M_2 and S_2 the result is as semi-diurnal tide with a tidal range varying between a_{M2}–a_{S2} and a_{M2}+a_{S2}.

The period of tidal range variation is

$$\frac{1}{1/12 - 1/12,4} = 253 \text{ hours} = 14,7 \text{ days} \tag{16.2}$$

the so-called spring-neap variation of the tidal range.

Thus it is seen that tide with a varying range can be described either as

1. a sum of harmonic constituents with almost equal phase
2. one harmonic constituent with a slowly varying amplitude (and phase) correction.

A harmonic description of tide is, in practice, is always a combination of the two.

Sub- and super-harmonic constituents are shallow water effects on the tide.

Sub-harmonic constituents (with period 6 h, 4h, 3h, etc.) describe the shape of the (diurnal or semi-diurnal) tidal variation. Normally (on deep water) the

variation is sinusoidal. As the tide propagate to shallower water the shape can change, so that e.g. the difference between Mean Water Level and high tide is larger (or smaller) than that of low tide. Another such effect could be that the tide rises faster (or slower) that it recedes. Sub-harmonic tidal constituents describe such effects.

Sub-harmonic constituents (with period around 14 days) describe "tidal pumping". If a high tidal range (such as during spring tide) results in a general rise in the mean water level (and correspondingly a low tidal range results in a lowering of the mean water level) then the mean water level varies as the (diurnal or semi-diurnal) tidal range varies.

Finally the water level (and in fact also the tide) can vary over the year, due to climatic changes (wind, temperature etc.), so-called seasonal variations.

16.4   Harmonic Tidal Description Methods.

The basis for a harmonic description of tide is the assumption that tidal response to the motion of the planets only contains frequencies present in the forcing. However, sometimes the tide is delayed (or advanced) slightly by meteorological effects, so in practice tide is not strictly harmonic. A harmonic description of tide cannot take such effects into account, neither in tidal prediction of course, nor (more importantly) in tidal analysis. In the extreme, exaggerated case where the half-daily tidal cycle is delayed/advanced randomly ±6 hours, harmonic analysis is not able to identify any trace of tide, although measurements clearly contain 12.4h tidal variations.

This is a characteristic of any harmonic description of tide.

A "pure" harmonic description of tide (i.e. a harmonic description where amplitude and phase of each constituent is constant, and thus no "correction" function is applied) requires quite a number of constituents to describe the tide decently. Often 30 or more constituents can be identified in water level measurements.

Although the tide varies from location to location, there is limits to how much the character of the tide changes between locations in the same area.

In terms of tidal constituents this means that there is a relationship between the amplitude and phase of the constituents (say 30 in number) describing the tide in two different locations.

A drawback by the pure harmonic description of tide is that this relationship cannot be expressed in terms of some constituents being the same within in an area and other constituents vary from place to place in that area.

A simple example: the combination of M_2 and S_2 describe the spring-neap variation of semi-diurnal tide. Both amplitude and phase of the semi-diurnal tide can vary a lot even between nearby locations, but it is unlikely that the

that the timing of spring tide and neap tide, or the ratio of spring time tidal range to neap tide tidal range differs too much. The similarity of the tide in the two locations cannot be expressed in terms of e.g. S_2 being the same but M_2 being different. Both S_2 and M_2 vary (in amplitude and phase). Instead the similarity can be expressed in terms of the amplitude ratio between M_2 and S_2, and the phase difference between M_2 and S_2. Another way of expressing the similarity is to describe the tide by one semi-diurnal constituent only (e.g. M_2) and to describe the spring-neap variation of the semi-diurnal tide as correction (in amplitude and phase) of the M_2 tide. With such a description of the tide the constituent (M_2) varies from location to location in an area, but the correction is the same all over the area.

Harmonic Tidal analysis methods differ in the degree to which variations in the daily or half daily tidal range is described by amplitude and phase corrections or by sums of individual constituents.

Another drawback by the harmonic description of tide is in the description of 2nd Order effects (shallow water effects). These effects result in

1. either a distortion from the sinusoidal shape of the semi-diurnal (or diurnal) tide that in harmonic terms is described by super-harmonic constituents,

2. or in a variation in the mean water level that is related to the tidal range (described by sub-harmonic constituents).

These effects are "bounded" in phase to the tidal variation, so that a super-harmonic is introduced to describe that the tidal shape e.g.has higher high waters and less deep low waters (compared to the symmetric sinusoidal shape) must follow (in phase) the variation to which it is a super-harmonic.

Typically such effects (being 2nd Order of nature) are also proportional to the square of the tidal range.

For tide described by a single harmonic constituent, such second order effects can be described by a single super-harmonic. However, for a tide described by n harmonic constituents, it requires n^2 super-harmonic constituents to describe the 2nd order effect correctly.

Harmonic Tidal analysis methods differ in the degree to which shallow water effects are described by harmonic constituents.

16.5 IOS Method

16.5.1 General introduction

In the traditional harmonic method for the description of tide, such as the one implemented in the IOS method (by G Foreman), the tidal variation is

described by harmonic constituents, except for the nineteen yearly variation of the tide caused by periodic changes in the lunar orbital tilt.

Variations caused by the nineteen yearly variation in the lunar orbital tilt is described by amplitude and phase corrections to tidal constituents (denoted nodal corrections). Doodson tidal constituents describe sub and super-harmonics as well as seasonal variations.

This method is potentially the most detailed description of the tide at a specific location, and is therefore typically used for locations where the tide is monitored continuously through several years.

The Rayleigh criterion is used for the selection of constituents from a standard data package composed by 69 constituents. The standard constituents include 45 astronomical main constituents and 24 shallow water constituents. They are derived only from the largest main constituents, M_2, S_2, N_2, K_2, K_1 and O_1, using the lowest types of possible interaction.

Additionally, 77 shallow water constituents can be included if explicitly requested by the user. The shallow water constituents that can be included in the modulation of tidal time series, are derived from the remainder main constituents considering higher types of interaction.

The amplitudes and phases are calculated via a least squares method, which enables the treatment of records with gaps. For the calculation of frequencies, nodal factors and astronomical arguments, the program is based on Doodson's tidal potential development and uses the reference time origin of January 1, 1976 for the computation of astronomical variables.

16.5.2 Time series representation

The general representation of a tidal time series is made according to the harmonic development

$$x(t) \ = \ \sum_{j=1}^{N} f_j(t) \ a_j \ \cos\left(V_j(t) + u_f(t) - g_j\right) \tag{16.3}$$

where
a_j, g_j are the amplitude and Greenwich phase lag,
$f_j(t)$, $u_j(t)$ are the nodal modulation amplitude and phase correction factors,
and $V_j(t)$ is the astronomical argument, for constituent j.

The astronomical argument $V_j(t)$ is calculated as

$$V_j(t) \ = \ V_j(t_0) + (t - t_0)\omega_j \tag{16.4}$$

where t_0 is the reference time origin.

The first stage in the analysis of tidal records is done via a least squares fit method for the computation of the amplitudes A_j and phases f_j representative of the combined effects of the main constituents and respective satellites, according to

$$x(t) = \sum_{j=1}^{N} A_j \cos(\omega_j t - \phi_j) \qquad (16.5)$$

The time series to be analysed have to be recorded with a 1 hour interval, and gaps can be automatically handled by the program. In order to reduce the computational time, the time origin is taken as the central hour of the record.

For the purpose of forecasting, the values of the amplitudes and Greenwich phase lags of the main constituents, as well as the corresponding time dependent correction factors that account for satellite interaction, are calculated via nodal modulation.

The following relationships are used:

$$a_j = A_j / f_j(t_0) \qquad (16.6a)$$

$$g_j = V_j(t_0) + u_j(t_0) + \phi_j \qquad (16.6b)$$

where t_0 is the central hour of the tidal record.

The program calculates the correction factors for all satellites that have the same first three Doodson numbers, which implies that the modulation is only fully effective for records with one year of duration.

16.5.3 Rayleigh criterion

The Rayleigh criterion is applied for the choice of main constituents, the synodic period being considered as default (Rayleigh number equal to 1.0). For records shorter than 1 year, some main constituents cannot be analysed, which implies the falsification of the amplitudes and phases of the neighbouring constituents. It should be kept in mind that the programs cannot compensate for the associated errors, because the nodal modulation supposes that all the main constituents have been included in the analysis. Nevertheless a correction can be made for the nearest component in frequency, with simultaneous inference of the amplitude and phase of the non-analysed constituent, when requested by the user.

When applying equation (16.3) for prediction, the nodal correction factors $f(t)$ and $u(t)$ are approximated by a constant value throughout the period of a month, calculated at 00 hr. of the 16th day of the month. This approximation is

currently used due to the small variations experienced by those factors over the period of a month. It should be kept in mind that differences between analysed and predicted records may arise due to the fact that for analysis the nodal correction factor are calculated only for the central hour of the whole record. In any case the calculation of *f, u* and *V* is based on the values of the astronomical variables *s, h, p, N'* and *p'* at the reference time January 1, 1976, and on the first term in their Taylor expansion, e.g.

$$s(t) \; = \; s(t_r) + (t - t_r)\frac{\partial s(t_r)}{\partial t} \tag{16.7}$$

The rationale behind it is as follows:

Suppose the tide is dominated by the constituents M_2 (with a period of 12.42 hours) and S_2 (with a period of 12.0 hours).

Both are Semidiurnal constituents (with period ~12 hours) and the resulting tide is semidiurnal (one high water and one low water every ~12 hours).

However, once every ~14 days the tidal amplitude is relatively high (spring tide) and once every ~14 days the amplitude is relatively low (neap tide).

In "constituent terms" this corresponds to the fact that once every 1/(1/12-1/12.42)hours (=354 hours =14.8 days) M2 and S2 are in phase (and amplify each other) and once they are out of phase (and counteract each other).

That means that if the tide is only measured for a period of 2-3 days, it is impossible to determine whether the measurement period represent spring tide , neap tide or something in between.

In "constituent terms" this corresponds to the fact that it is impossible to derive BOTH M_2 and S_2 from the measurements.

In order to be able to determine BOTH M_2 AND S_2 from measurements, a measuring period of at least R*354 hours is required, where R is the Rayleigh criterion. In other words R is the number of full spring-neap tide cycles required to include BOTH M_2 AND S_2 in a tidal analysis.

If the vater level variations are completely dominated by regular tide then R=1 full spring-neap tide cycles is sufficient. If other (non tidal) effects influence the water level (e.g. storm surge) then R=2..3 spring-neap tide cycles are required to determine a reliable estimate of both M_2 and S_2.

16.5.4 Step-by-step calculation

The steps performed by the analysis programs are as follows:

1. Data input comprising:
   – control and decision parameters

      –   time series (levels or north/south and east/west current compo-
nents)

2. Calculation of the middle hour t_c of the analysis period.

3. Calculation at t_c, of the nodal modulation correction factors f and u and of the astronomical arguments V, for all the constituents in the constituent data package.

4. Determination via the Rayleigh criterion, of the constituents to be used in the least squares fits.

5. Construction and solution relative to time t_c, of the least squares matrices (for levels or for each of the x (east/west) and y (north/south) current components). Details of the fit, such as error estimates, average, standard deviation, matrix condition number and root mean square residual error values are given, together with the parameters C_j and S_j from which the amplitudes and phases of the constituents can be computed, according to the relationships given in e.g. (16.8) and (16.9).

6. Inference for the requested constituents not included in the least squares fit, and adjustment of the constituents used for inference.

7. Nodal modulations of the analysed and inferred constituents, using the previously computed nodal correction factors f and u. Greenwich phase lags and constituent amplitudes are then obtained (for currents the amplitudes of the major and minor semi-axis lengths).

8. Data output.

$$A_j = \sqrt{C_j^2 + S_j^2} \tag{16.8}$$

$$\phi_j = \operatorname{atan}(S_j / C_j) \tag{16.9}$$

16.6 Admiralty Method

In the Admiralty method only the four main constituents M_2, S_2, 0_1 and K_1 are explicitly considered, and corrective factors are allowed to take into account the effects of a number of astronomical and shallow water generated constituents.

This method should be used especially when the harmonic constituents are obtained from the Admiralty Tide Tables.

The Admiralty Method constitutes a simplified and expeditious way for the prediction and analysis of tides. Only the four main tidal constituents are explicitly considered (M_2, S_2, 0_1 and K_1). These constituents are amended, via corrective factors for the amplitudes and phases, in order to take into account the effects of the following constituents:

- Semi-diurnal - $2N_2$, μ_2, N_2, n_2, l_2, L_2, T_2, K_2

- Diurnal- $2Q_1$, s_1, Q_1, r_1, $\_1$, F_1, J_1

The method assumes that the tide response functions (i.e. the ratio of amplitude and phase differences between the generating forces and the observed tide) are the same for tidal constituents of close frequencies.

The application of this method is of practical interest in connection for example with the simulation of tidal propagation over large areas. In this case it is not usually possible to rely on a comprehensive set of data measurements for the specification of boundary conditions and for the calibration/validation of the models. On the other hand historical data have been compiled for a large number of stations around the world and are presented in the Admiralty Tide Tables in the form of tables containing the four main tidal constituents in accordance with the Admiralty Method previously described. It is therefore common practice to use the available information on tides as background for coastal engineering studies, and complement it with ad hoc measurements, when necessary, for clarification of particular phenomena along the local areas of main interest.

The routines used in the present programmes are based on the principles described in "The Admiralty Manual of Tides" (Doodson, 1941) and the "NP159 Admiralty Method of Tidal Prediction".

16.7 Constituent Data Package

A Constituent Data Package with standard constituent input data files needed for the execution of the program are included in the installation. The two files, **anapack.dat** for the analysis programs and **prepack.dat** for the prediction programs, are situated in the MIKEZero/bin directory and automatically read by the program.

The contents of the files containing the constituent information is presented below. The data is organised in four main groups as follows:

1. A list of all the possible constituents to be included in the analysis, their frequencies in cycles/hour and the constituent with which they should be compared according to the Rayleigh criterion. A blank data field for the comparison constituent implies that the corresponding constituent is not being included in the analysis unless specifically designated in the input

files.

2. The values for the astronomical arguments *s, p, h, N'* and *p'* and their respective rates of change over a 365 day period at the reference time origin (January 1, 1976).

3. The Doodson numbers for all the main tidal constituents and information on the satellite constituents. The first line for each constituent contains the following information:

    – The constituent name
    – The six Doodson numbers of the constituent
    – The phase correction for the constituent
    – The number of satellite constituents (N_J)

    If $N_J>O$, information on the satellite constituents follows, three satellites per line. For each satellite the values read are:

    – The last three Doodson numbers of the main constituent subtracted from the last three Doodson numbers of the satellite constituent.
    – The phase correction of the satellite constituent relative to the phase of the main constituent.
    – The amplitude ratio of the satellite tidal potential to that of the main constituent
    – R1, if the amplitude ratio has to be multiplied by the latitude correction factor for diurnal constituents,
    – R2, if the amplitude ratio has to be multiplied by the latitude correction factor for semi-diurnal constituents

4. List of shallow water constituents and of the main tidal constituents from which they are derived. The respective variables are:

    – The name of the shallow water constituent. The number of main constituents from which it is derived.
    – Combination number and name of these main constituents.

The constituent data package for the prediction programs does not include the first group of data. Therefore, in the prediction programs the constituent frequencies are calculated via the derivatives of the astronomical variables and the constituent Doodson numbers.

17 Tidal Analysis of Currents

With the TIDCAC tool you can analyse time series of hourly measurements of currents (e.g. current meter recordings) for a specified period of time.

17.1 Select Input File

Here you select the data file with North and East current component that you want to analyse. You can select either a standard time series or a MIKE 21 HD result file.

NOTE: The analysis is based on input data using hourly time steps (3600 s). In case the time step in the input data file is less than one hour, the calculation will be based on the hourly value, e.g. for a time series with a time step of 600 s, only every 6th value will be used in the analysis.

17.2 Subseries Selection

Here you specify the data period you want to analyse.

17.3 General Parameters

For the analysis of currents you first have to specify which method to use:

- The IOS method should be chosen when constituents should be inferred directly from measurement data by use of Rayleigh criterion

- The Admiralty method should be chosen when the four main constituents (M_2, S_2, K_1 and O_1) and their satellites distribute the main energy

17.3.1 IOS method

For the IOS method, you define the latitude of where the data is valid and the Rayleigh criterion. Per default this criterion is 1.0.

Furthermore, the number of inferred constituents must be given, and for each inferred constituent, the name and frequency and the parent constituent (name and frequency) must be specified. Additional constituents can be included in the analysis. These can be chosen among the constituents from the constituent data package, which does not have a specified Rayleigh comparison constituent.

A more detailed description of the method can be found in IOS Method *(p. 112)*.

17.3.2 Admiralty method

For the Admiralty method only two amplitude factors and two phase difference values must be specified. They are all derived from the four main constituents (M_2, S_2, K_1 and O_1).

A more detailed description of the method can be found in Admiralty Method *(p. 116)*.

17.4 Inferred Constituents

Here you select the number of constituents to be inferred by the analysis when using the IOS method.

For each constituent you must specify:

- name and frequency (degrees/hour) of the constituent to be inferred
- name and frequency of the parent constituent to be used for the inference
- amplitude ratio between the constituents
- phase difference (the Greenwich phase lag of the inferred constituent subtracted from the Greenwich phase lag of the analysed constituent)

17.5 Additional Constituents

Additional constituents to be included in the analysis may be chosen from amongst the constituents in the Constituent Data Package with no specified Rayleigh comparison constituent.

For each constituent you must specify

- the constituent name
- the Rayleigh comparison constituent (that has to be found amongst the constituents of the list included in the **anapack** file, that don't have a defined Rayleigh comparison constituent)

17.6 Select Output File

Here you must specify (path and) name of the output file(s).

17.6.1 Constituent output file

The standard output file contains the final results of the analysis. This includes the calculated tidal constants, namely the name, amplitudes and

Greenwich phase lags for each of the constituents, after interference and nodal modulation have been applied.

This file has the extension .con and may later be used as input file for the prediction program.

17.6.2 Residual output

If requested, a file with predicted and residual time series of the specified period of analysis is also produced (file extension is dfs0 or dfs2, depending on the input file).

17.7 Status - Overview

The overview page presents you with a summary of all the input parameters you have specified. You should inspect the parameters to check they are correct, - and if not go back and change them.

From the page you can either *Finish* the set-up, which will transfer your set-up definition to the current toolbox file, or execute the set-up.

18 Tidal Analysis of Heights

With the TIDHAC tool you can analyse time series of hourly measurements of water levels (e.g. tide gauge measurements) for a specified period of time.

18.1 Input Data

Here you select the data you want to analyse for tidal constituents. You can select either a standard time series or a MIKE 21 HD result file.

You can select one item for the analysis.

NOTE: The analysis is based on input data using hourly time steps (3600 s). In case the time step in the input data file is less than one hour, the calculation will be based on the hourly value, e.g. for a time series with a time step of 600 s, only every 6th value will be used in the analysis.

18.2 Subseries Selection

Here you specify the data period you want to analyse.

18.3 General Parameters

For the analysis of heights you first have to specify which method to use:

- The IOS method should be chosen when constituents should be inferred directly from measurement data by use of Rayleigh criterion
- The Admiralty method should be chosen when the four main constituents (M_2, S_2, K_1 and O_1) and their satellites distribute the main energy

18.3.1 IOS method

For the IOS method, you define the latitude of where the data is valid and the Rayleigh criterion. Per default this criterion is 1.0.

Furthermore, the number of inferred constituents must be given, and for each inferred constituent, the name and frequency and the parent constituent (name and frequency) must be specified. Additional constituents can be included in the analysis. These can be chosen among the constituents from the constituent data package, which does not have a specified Rayleigh comparison constituent.

A more detailed description of the method can be found in IOS Method *(p. 112)*.

18.3.2 Admiralty method

For the Admiralty method only two amplitude factors and two phase difference values must be specified. They are all derived from the four main constituents (M_2, S_2, K_1 and O_1).

A more detailed description of the method can be found in Admiralty Method *(p. 116)*.

18.4 Inferred Constituents

Here you select the number of constituents to be inferred by the analysis when using the IOS method.

For each constituent you must specify:

- name and frequency (degrees/hour) of the constituent to be inferred
- name and frequency of the parent constituent to be used for the inference
- amplitude ratio between the constituents
- phase difference (the Greenwich phase lag of the inferred constituent subtracted from the Greenwich phase lag of the analysed constituent)

18.5 Additional Constituents

Additional constituents to be included in the analysis may be chosen from amongst the constituents in the Constituent Data Package with no specified Rayleigh comparison constituent.

For each constituent you must specify

- the constituent name
- the Rayleigh comparison constituent (that has to be found amongst the constituents of the list included in the **anapack** file, that don't have a defined Rayleigh comparison constituent)

18.6 Select Output File

Here you must specify (path and) name of the output file(s).

18.6.1 Constituent output file

The standard output file contains the final results of the analysis. This includes the calculated tidal constants, namely the name, amplitudes and

Greenwich phase lags for each of the constituents, after interference and nodal modulation have been applied.

This file has the extension .con and may later be used as input file for the prediction program.

18.6.2 Residual output

If requested, a file with predicted and residual time series of the specified period of analysis is also produced (file extension is dfs0 or dfs2, depending on the input file).

18.7 Overview

The overview page presents you with a summary of all the input parameters you have specified. You should inspect the parameters to check they are correct, - and if not go back and change them.

From the page you can either *Finish* the set-up, which will transfer your set-up definition to the current toolbox file, or execute the set-up.

19 Examples - Tidal Analysis

19.1 Analysis of Tidal Height using the Admiralty Method

The following figures describe how to analyse a time series of water level in order to calculate the tidal constituents using the Admiralty Method.

The same method is applicable using tidal analysis of currents.
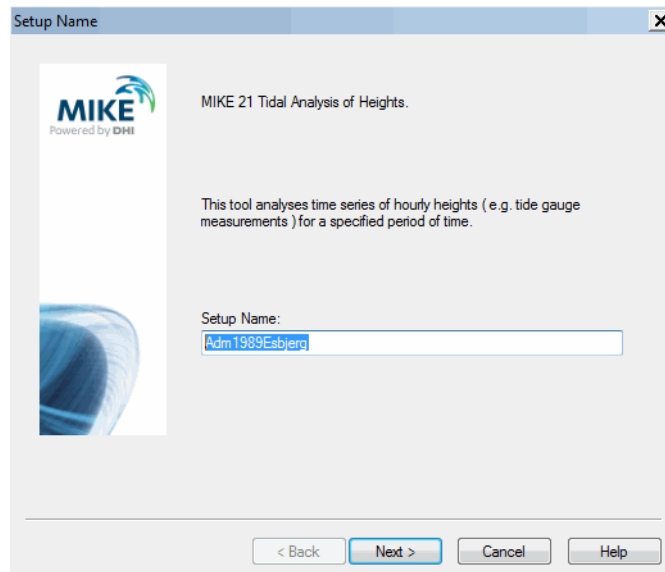
Figure 19.1 Specify a Name in the Tidal Analysis of Height from the Tidal Tools in MIKE 21 Toolbox
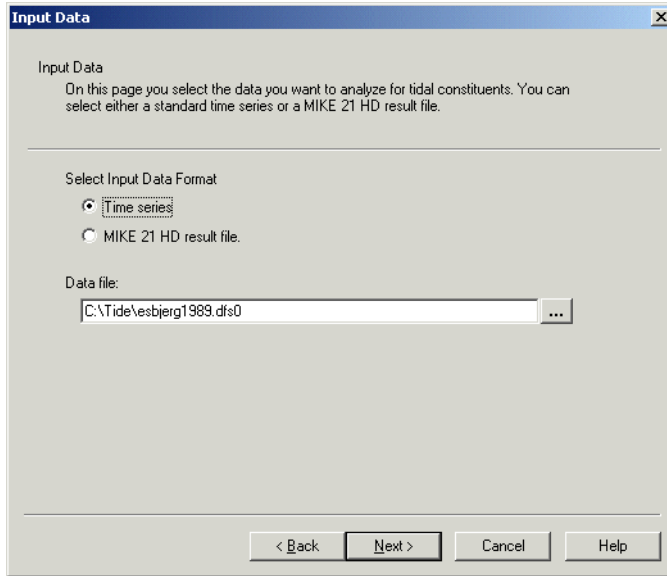
Figure 19.2   Specify Data Format of Time series to be analysed
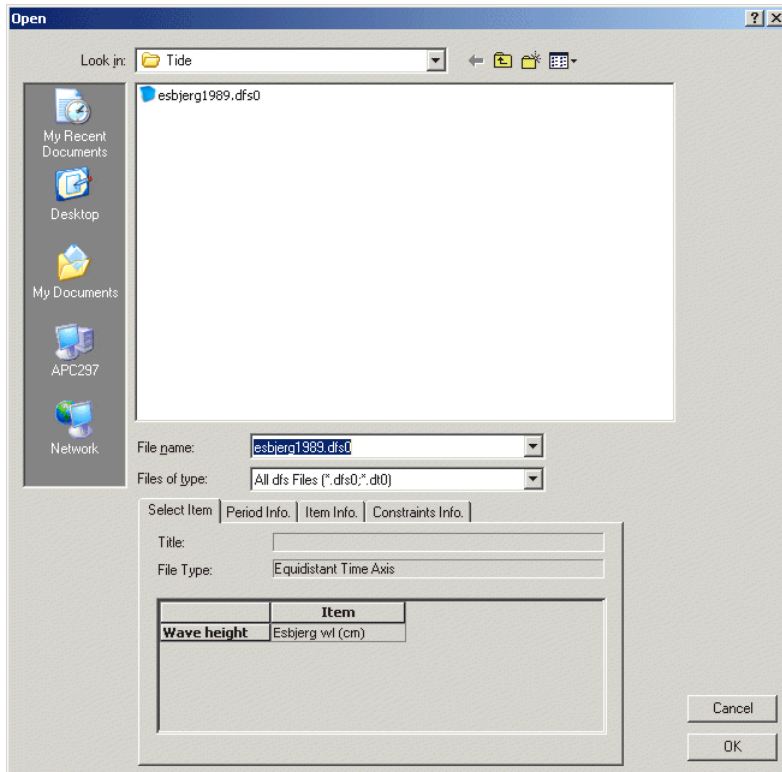
Figure 19.3   Select the time series to be analysed

Figure 19.4   Specify the period to be analysed

Figure 19.5   Select the Analysis Method. For short-range time series optional specify the Phase difference and Amplitude factor between M2-S2 and K1-O1
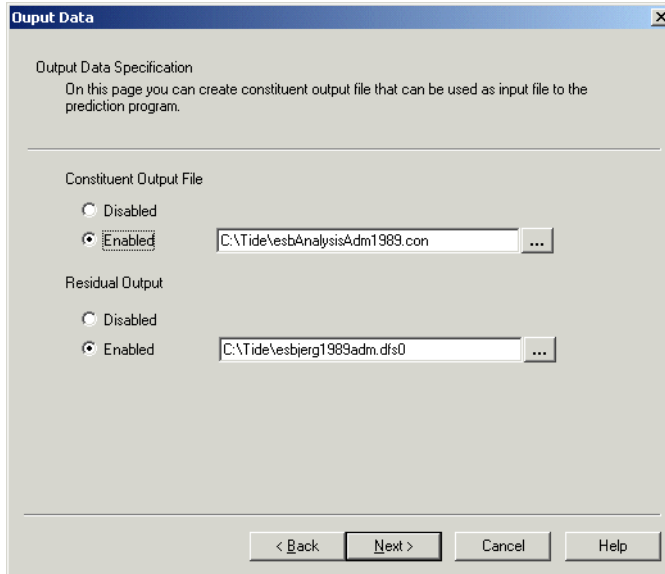
Figure 19.6   If enabled, specify the name of Constituent output file and Residual output file (see output examples)
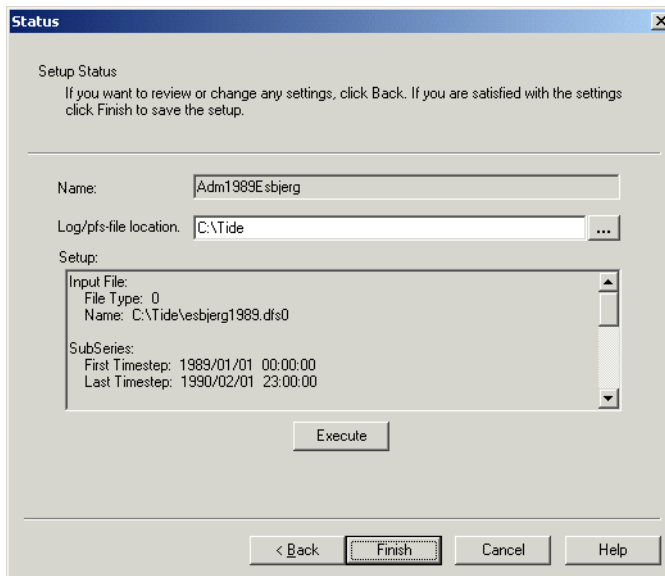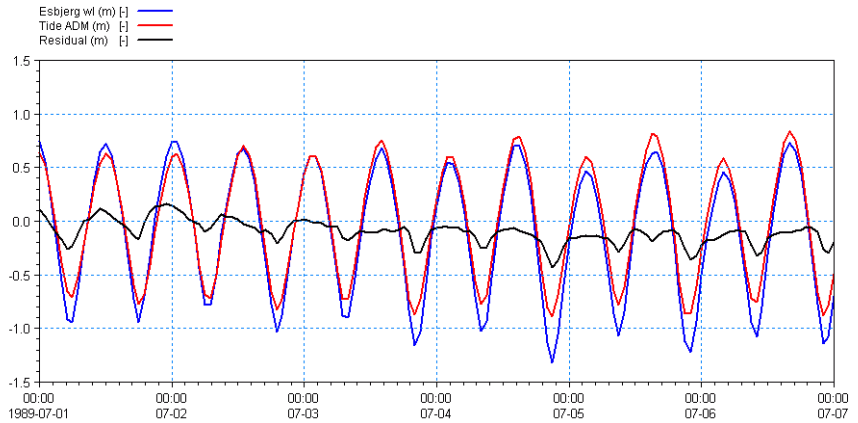

Figure 19.7   Specify location of log file and execute the program

Figure 19.8   Example of Residual output file. The file includes the measured values, the calculated tide and the residual (the difference between the calculated and the measured tide)
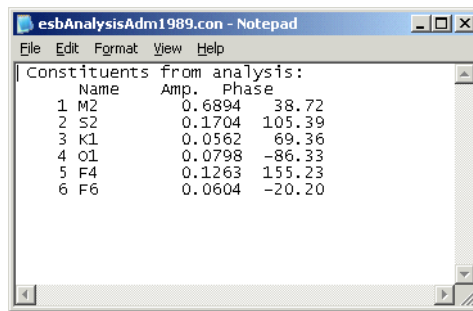

Figure 19.9   As an optional output, you can get the actual calculated constituent - an example is given above

19.2   Analysis of Tidal Height using the IOS Method

The following figures describe how to analyse a time series of water level in order to calculate the tidal constituents using the IOS Method.

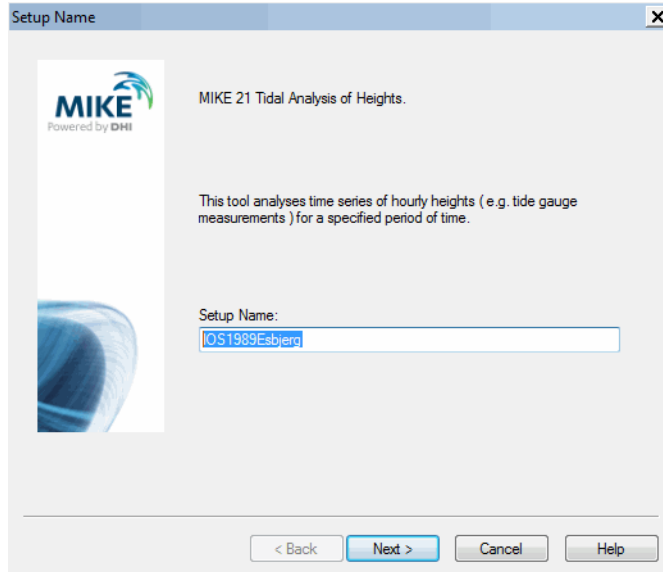The same method is applicable using tidal analysis of currents.

Figure 19.10   Specify a Name in the Tidal Analysis of Height from the Tidal Tools in MIKE 21 Toolbox
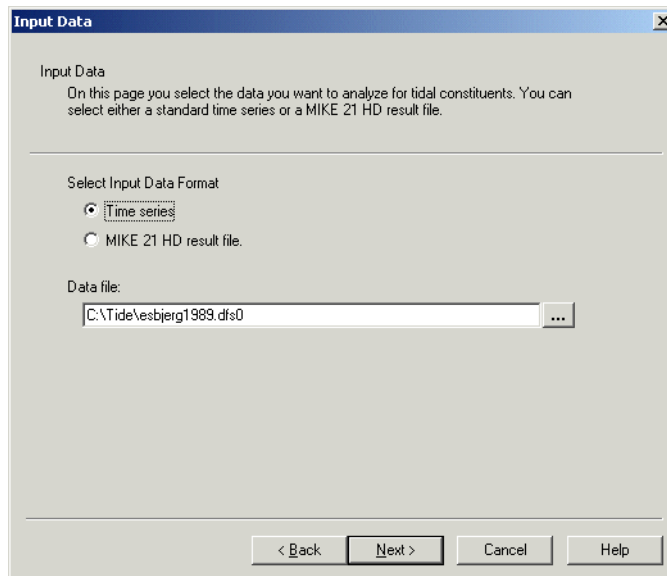

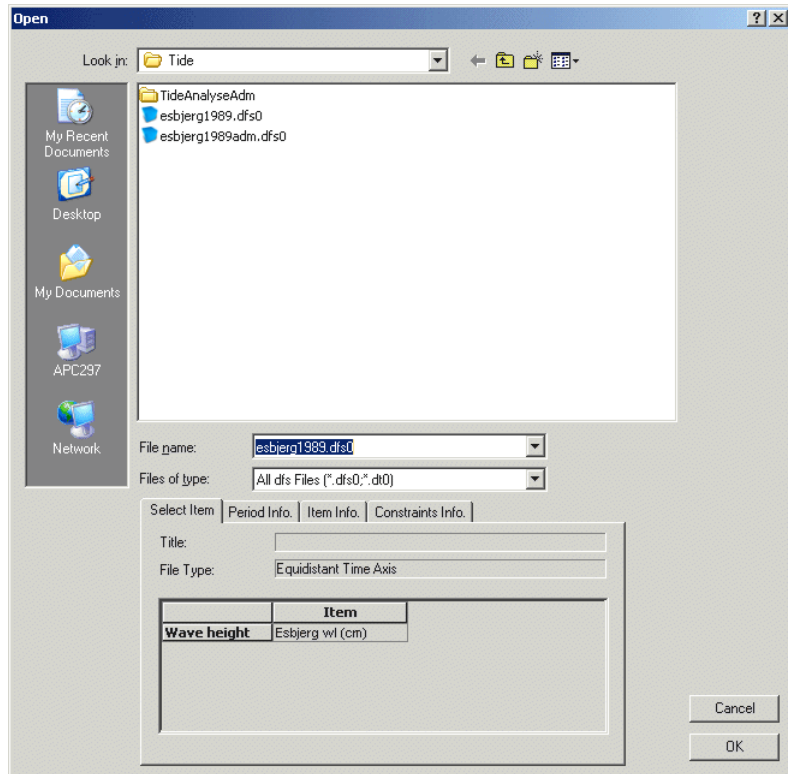Figure 19.11   Specify Data Format of Time series to be analysed

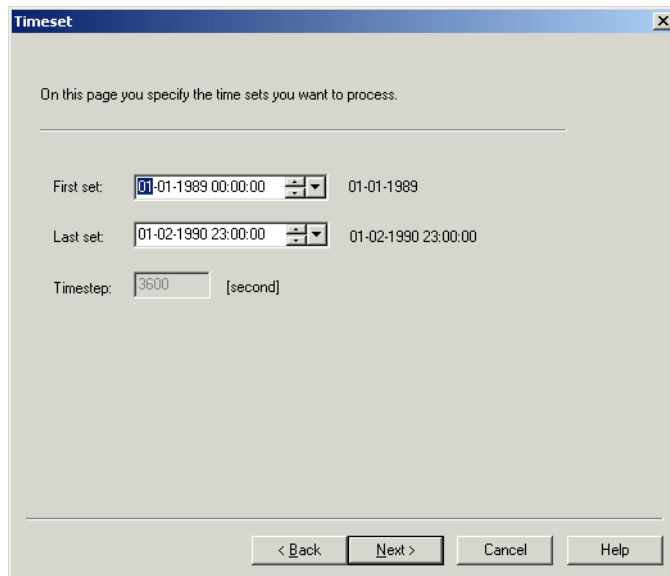Figure 19.12   Select the time series to be analysed

Figure 19.13   Specify the period to be analysed

Figure 19.14   Select the Analysis Method. For the IOS method specify the Latitude and Rayleigh Criteria.

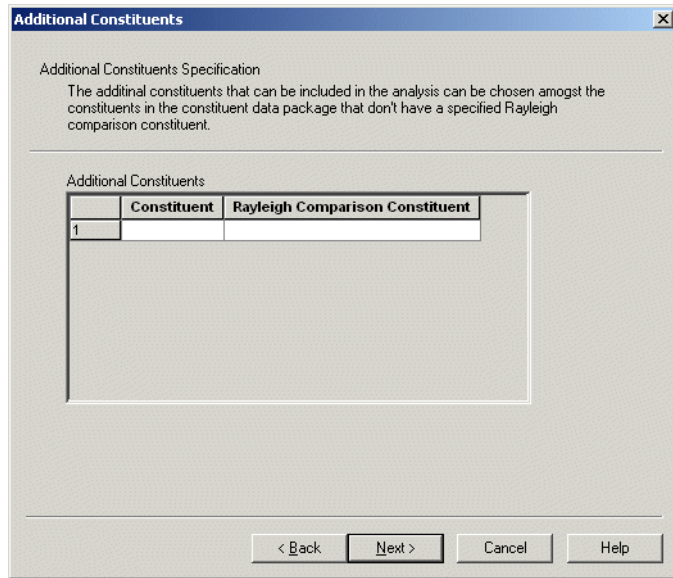Figure 19.15   Specify any interfered constituents

Figure 19.16   Specify any additional Constituents, which do not have a specified Rayleigh comparison constituent
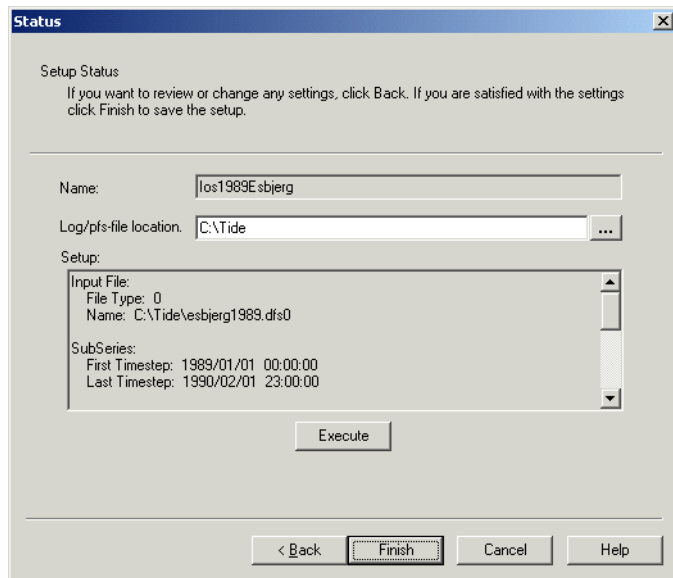

Figure 19.17   Specify location of log file and execute the program.
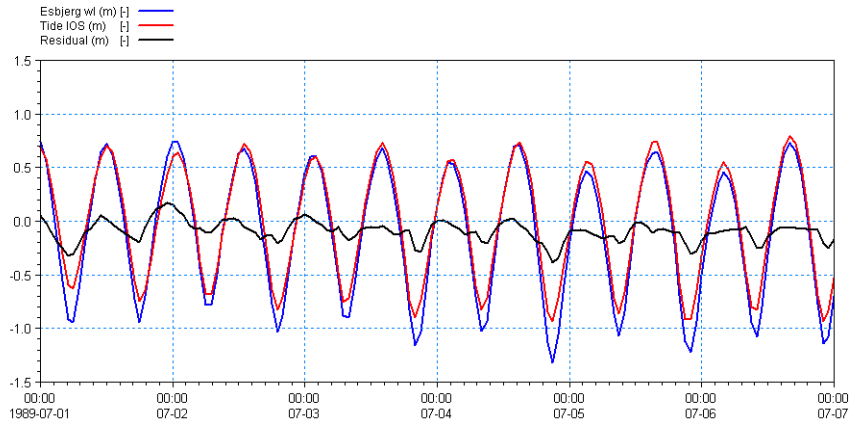
Figure 19.18   Example of Residual output file. The file include the measured values the calculated tide and the residual, the difference between the calculated and the measured tide

```
esbAnalysisios1989.con - Notepad
File  Edit  Format  View  Help
Constituents from analysis:
      Name     Amp.   Phase
   1 Z0        0.2019    0.00
   2 SA        0.2010  347.23
   3 SSA       0.1095  277.89
   4 MSM       0.0544  129.25
   5 MM        0.0636  115.07
   6 MSF       0.0429  220.07
   7 MF        0.0533   85.44
   8 ALP1      0.0124   42.36
   9 2Q1       0.0070  234.85
  10 SIG1      0.0027   49.65
  11 Q1        0.0290  229.65
  12 RHO1      0.0071  216.53
  13 O1        0.0785  277.06
  14 TAU1      0.0088  101.76
  15 BET1      0.0009  172.46
  16 NO1       0.0018    8.30
  17 CHI1      0.0039  257.68
  18 PI1       0.0082  117.43
  19 P1        0.0205   78.32
  20 S1        0.0098  249.81
  21 K1        0.0557   68.58
  22 PSI1      0.0089   57.28
  23 PHI1      0.0019   89.39
  24 THE1      0.0064  104.69
  25 J1        0.0061  120.00
  26 SO1       0.0053  326.64
  27 OO1       0.0020  327.80
  28 UPS1      0.0037  196.51
  29 OQ2       0.0097  121.13
  30 EPS2      0.0204  116.21
  31 2N2       0.0188  188.43
  32 MU2       0.0629  140.17
  33 N2        0.1053   18.63
  34 NU2       0.0463  347.16
  35 H1        0.0180  192.10
  36 M2        0.6889   39.13
  37 H2        0.0302  229.06
  38 MKS2      0.0195  184.53
  39 LDA2      0.0170   41.47
  40 L2        0.0780   39.05
  41 T2        0.0173   79.95
  42 S2        0.1718  105.15
  43 R2        0.0064  148.09
  44 K2        0.0442  109.29
  45 MSN2      0.0105  300.30
  46 ETA2      0.0028  308.45
  47 MO3       0.0059  177.59
  48 M3        0.0016   16.69
  49 SO3       0.0028  273.53
  50 MK3       0.0047  354.34
  51 SK3       0.0019   31.64
  52 MN4       0.0228  204.33
  53 M4        0.0638  234.92
  54 SN4       0.0025  222.71
  55 MS4       0.0296  307.20
  56 MK4       0.0048  320.36
  57 S4        0.0009   61.43
  58 SK4       0.0006  227.57
  59 2MK5      0.0031  153.73
  60 2SK5      0.0007  155.14
  61 2MN6      0.0101   75.92
  62 M6        0.0204  100.91
  63 2MS6      0.0187  165.23
  64 2MK6      0.0038  172.18
  65 2SM6      0.0033  259.85
  66 MSK6      0.0019  267.20
  67 3MK7      0.0007   55.65
  68 M8        0.0039  317.77
```

Figure 19.19   As an optional output, you can get the actual calculated constituent, see the example above

20 Tidal Prediction of Currents

With the TIDCPC tool you can produce a time series of tidal current values for a specified period of time.

20.1 General Parameters

Here you specify various general information about the tide station and the algorithm to be used for tidal prediction (.IOS method or Admiralty method).

Prediction period specifications are also included on this page, including the time step for the predicted values.

You may choose to create a tide table for the prediction period. The tide table will be saved in one or several files (.txt), each covering a 4 month period. The tide table will be saved in the same directory as the output file. Additional information will be written to the log file.

20.2 Tidal Constituents

Here you specify tidal constituents to be used as the basis for predictions. The number of constituents is dependent on the choice of algoritm on the previous dialog.

You have two options for choosing the algorithm to be used:

- .IOS method
- Admiralty method

20.2.1 .IOS method

The IOS method should be chosen when height (water level) time series should be inferred directly from the constituents.

You may either specify the number and type of constituents yourself or import a constituent output file (.con) previously generated by the tide analysis programs. In the latter case the constituent types and values are inserted automatically into the dialog.

A more detailed description of the method can be found in IOS Method *(p. 112)*.

20.2.2 Admiralty method

The Admiralty method should be chosen when only the four main constituents (M_2, S_2, K_1 and O_1) and their satellites distribute the main energy.

You must specify a number of constituents for the prediction of heights.

A more detailed description of the method can be found in Admiralty Method *(p. 116)*.

20.3   Select Output File

Here you must specify a (full path) name and, optionally, a title for your output file.

The output is a time series data file, which contains time series of current velocities.

20.4   Status - Overview

The overview page presents you with a summary of all the input parameters you have specified. You should inspect the parameters to check they are correct, - and if not go back and change them.

From the page you can either *Finish* the set-up, which will transfer your set-up definition to the current toolbox file, or execute the set-up.

21 Tidal Prediction of Heights

This tool is applied for prediction of time series of tidal heights (or water levels). The prediction can be based on either user defined tidal constituents or constituents derived from a global tide model.

The output is a time series data file (dfs0) or line series data file (dfs1), which contains time series of heights (water levels) in a point or along a line, respectively. The data is often used as input to models within the MIKE 21, MIKE 3 and LITPACK modelling systems.

21.1 Type of Prediction

Following two tide prediction types are available:

- Prediction based on User Defined Constituents
- Prediction based on Global Tide Model Data

If you choose the latter one, you have to specify the path and name of the constituent data file and the prepack file.

The default path is C:\Program Files (x86)\DHI\2016\MIKE Zero\Application Data\Tide Constituents\ and the default names for the two files are global_tide_constituents_0.25deg.dfs2 and prepack.dat, respectively. An updated version of the Global Tide Model Data file in resolution 0.125° × 0.125° is available from the MIKE Powered by DHI download page.

Note that the tidal data will be generated with respect to the UTC/GMT time system.

21.2 User Defined Constituents

21.2.1 General parameters

You must define the location and name of the station for which you want to make the prediction. The prediction period is defined by the start date and stop date, including the time step for the predicted values.

You have two options for choosing the algorithm to be used:

- IOS method
- Admiralty method

You may choose to create a tide table for the prediction period. The tide table will be saved in one or several files (.txt), each covering a 4 month period. The tide table will be saved in the same directory as the log-file. Additional information will be written to the log file.

21.2.2 Tidal constituents

You have to specify the tidal constituents to be used as the basis for predictions. The number of constituents is dependent on the choice of algoritm on the General parameters dialog.

21.2.3 IOS method

The IOS method should be chosen when height (water level) time series should be inferred directly from the constituents.

You may either specify the number and type of constituents yourself or import a constituent output file (.con) previously generated by the tide analysis programs. In the latter case the constituent types and values are inserted automatically into the dialog.

A more detailed description of the method can be found in IOS Method *(p. 112)*.

21.2.4 Admiralty method

The Admiralty method should be chosen when only the four main constituents (M_2, S_2, K_1 and O_1) and their satellites distribute the main energy.

You must specify a number of constituents for the prediction of heights.

A more detailed description of the method can be found in Admiralty Method *(p. 116)*.

21.2.5 Output specification

You must specify a (full path) name and, optionally, a title for your output file.

The output is a time series data file (dfs0), which contains time series of heights (water levels).

If the option for tide table has been included, the tide table is written into a number of a ASCII .txt files and the .log file will contain the occurence of high and low water levels.

21.3 Global Tide Model Data

The Global Tide Model data represents the major diurnal (K1, O1, P1 and Q1) and semidiurnal tidal constituents (M2, S2, N2 and K2) with a spatial resolution of 0.25° × 0.25° based on TOPEX/POSEIDON altimetry data.

The Global Tide Model data use UTC/GMT as the time system.

For more information see e.g. Ole Baltazar Andersen (1995), Global ocean tides from ERS 1 and TOPEX/POSEIDON altimetry, J. of Geophys. Res., 100, C12, p. 25249-25260.

Please note that the data is mainly applicable in relatively deep water, say, depths greater than 20 meter.

21.3.1 General parameters

You define the type of output and the tide prediction period.

You may choose between two types of output:

- Time series output
- Line series output

The latter option is particularly useful when generating water level boundary conditions for flow models in MIKE 21 and MIKE 3.

The prediction period is defined by the start date and stop date, including the time step interval for the predicted values.
Please note that the time step interval must be above 0.1 hours to ensure accuracy of the predicted values.

21.3.2 Time series output

For time series output you must specify the name and geographical position for each of the points at which you want to make a tidal prediction.

The results are written into a dfs0 file, each point is given by an item. You must specify a (full path) name and, optionally, a title for your result file.

21.3.3 Line series output

For each line at which you want to make a tidal prediction you must specify the geographical position (start and end), the number of points on the line, a (full path) name and, optionally, a title for the output file.

You can specify the location of a line in one of three ways:

- User specified
- Bathymetry file (.dfs2)
- Mesh file (.mesh)

You may exclude a detected boundary by leaving the "Include" check box unticked.

User specified

By user specified input you can define any number of lines. You must specify the start point and end point for each line in geographical coordinates (longitude, latitude).

Bathymetry file

When specifying a bathymetry file (.dfs2), the dialog automatically scans the bathymetry file, detects the open boundaries and updates the number of lines and location accordingly. The number of points on the line is initially set to the actual number of grid points along the boundary given by the resolution of the bathymetry file.

You cannot modify the location of a line but you have the option to show the start and end position of the line by either the grid coordinates in the bathymetry file or as geographical coordinates.

Mesh file

When specifying a mesh file (.mesh), the dialog automatically scans the mesh file, detects the boundaries (code values different from 0 and 1) and updates the number of lines and location accordingly. The number of points on the line is initially set to the number of node points along the boundary given by the resolution of the mesh file.

NOTE: Each line is defined by the start and end point only, and the height values are found at positions interpolated linearly in between the two points. Thus curved boundaries in the mesh might produce inaccurate results.

You cannot modify the location of a line but if the mesh has its geographical information given as a projection you have the option to show the start and end position of the line by either the projection coordinate system defined in the mesh file or as geographical coordinates.

21.4 Status - Overview

The overview page presents you with a summary of all the input parameters you have specified. You should inspect the parameters to check they are correct, - and if not go back and change them.

From the page you can either *Finish* the set-up, which will transfer your set-up definition to the current toolbox file, or execute the set-up.

22 Examples - Tidal Prediction

22.1 Prediction of Tidal Height using the Admiralty Method

The following figures describe how to predict a time series of water level based on tidal constituents using the Admiralty Method.

NOTE: The resulting setup file for this example is part of the installation

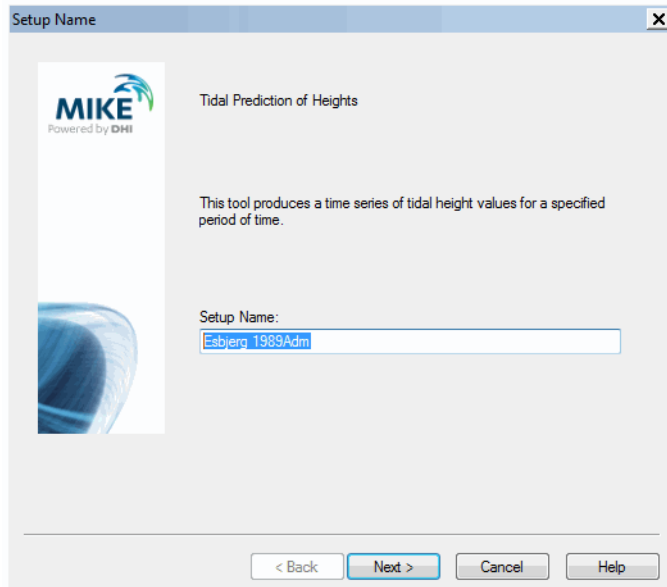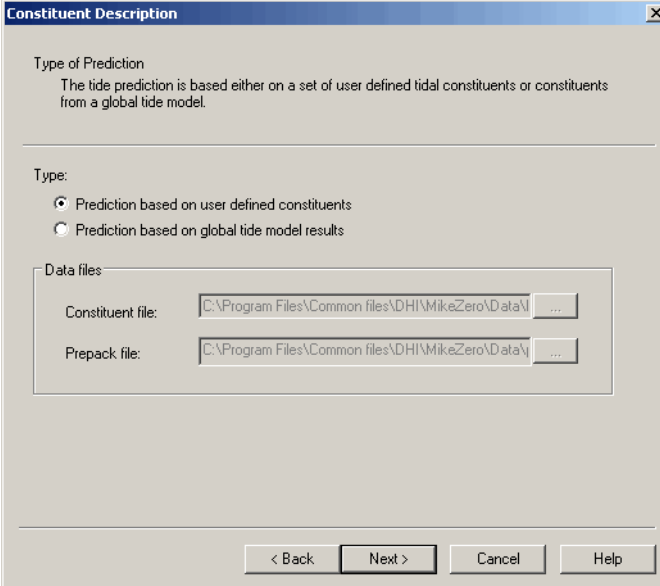The same method is applicable using tidal prediction of current.

Figure 22.1   Specify a Name in the Tidal Prediction of Height from the Tidal Tools in MIKE 21 Toolbox

Figure 22.2   Specify the type of prediction to be using user defined constituents

Figure 22.3   Specify the Station Name and Longitude and Latitude of the station and the wanted prediction period

Figure 22.4   Specify the Amplitude and Phase for each individual Constituent

Figure 22.5   Specify the output file name

Figure 22.6   Specify location of log file and execute the program
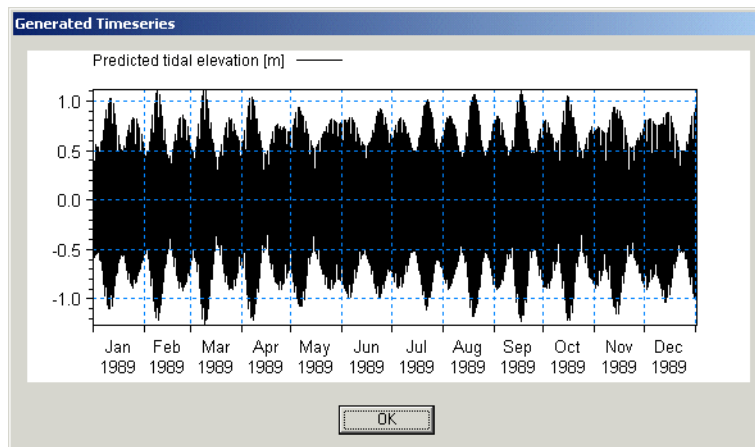
Figure 22.7   A graphical file showing the predicted tide will signify the termination of the program
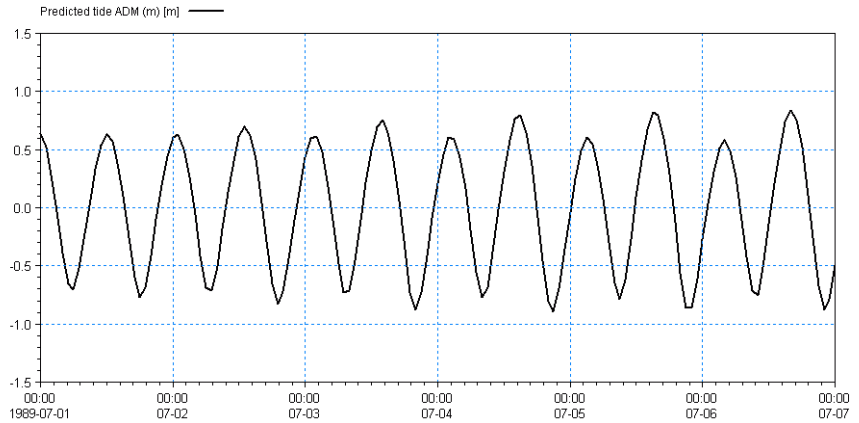
Figure 22.8   Example of generated tidal data

22.2   Prediction of Tidal Height using the IOS Method

The following figures describe how to predict a time series of water level based on tidal constituents using the IOS Method.

The same method is applicable using tidal prediction of current.
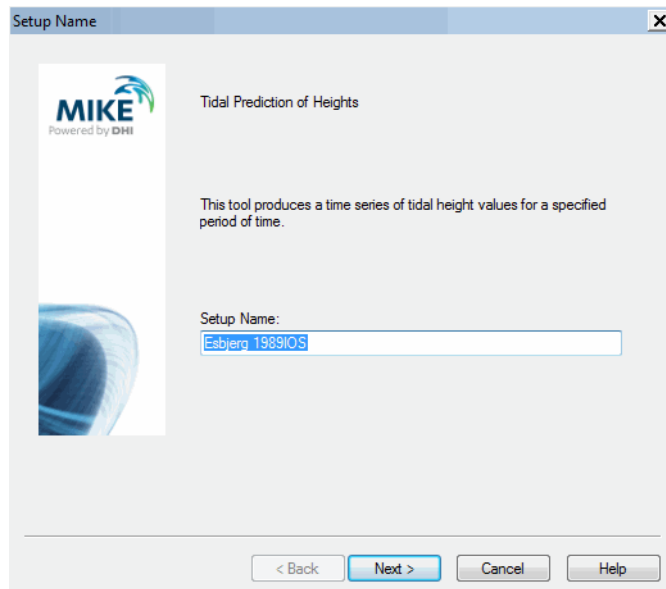
Figure 22.9   Specify a Name in the Tidal Prediction of Height from the Tidal Tools in MIKE 21 Toolbox
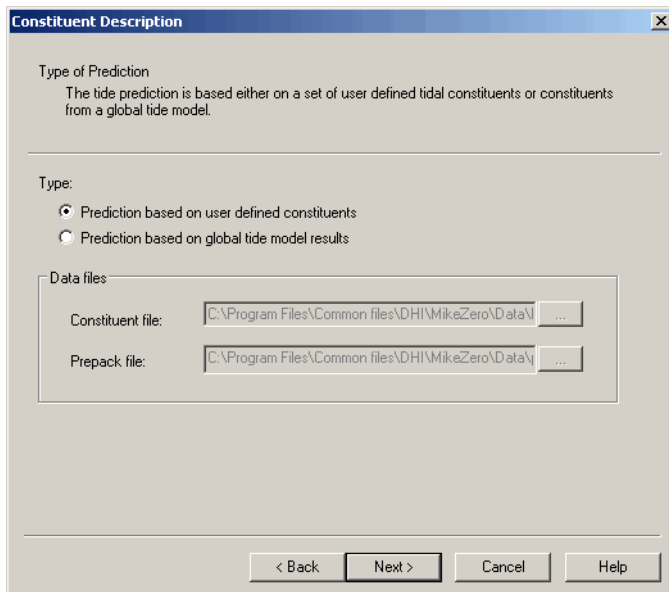
Figure 22.10   Specify the type of prediction to be using user defined constituents
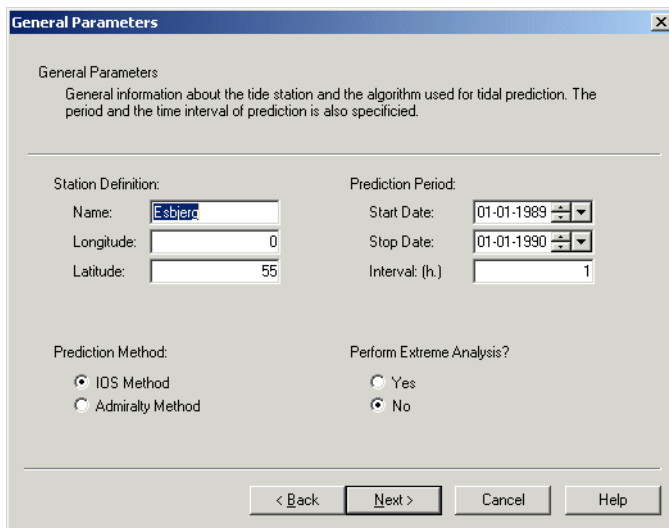
Figure 22.11   Specify the Station Name and Longitude and Latitude of the station and the wanted prediction period

Figure 22.12   Select the number of constituents and specify the Amplitude and Phase for each individual Constituent, or select a pre-made constituents file

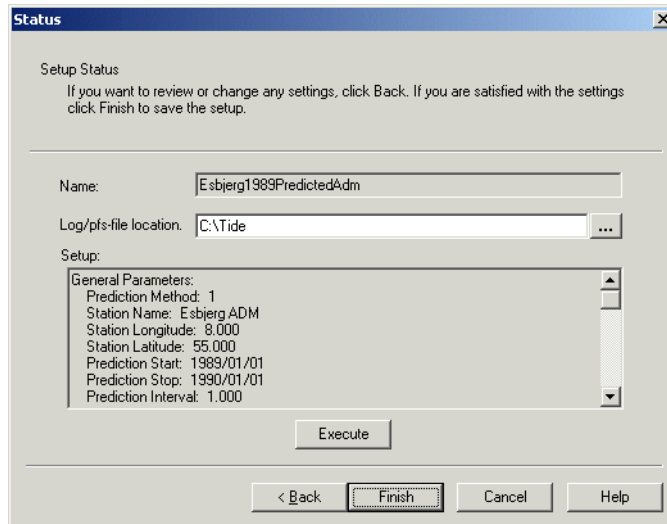Figure 22.13   Specify the output file name
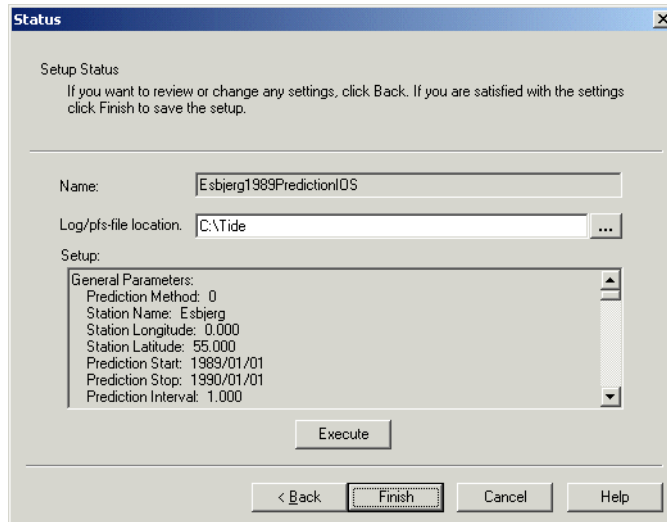
Figure 22.14   Specify location of log file and execute the program
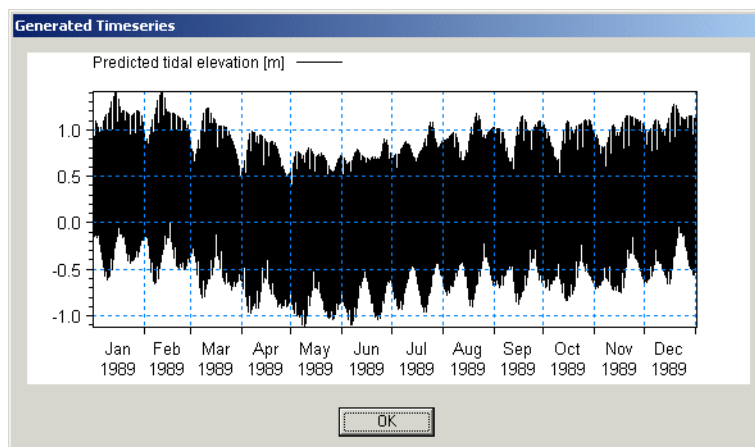
Figure 22.15   A graphical file showing the predicted tide will signify the termination of
the program. Please note, the seasonal variation, which is due to the
fact that more than one year's data has been analysed
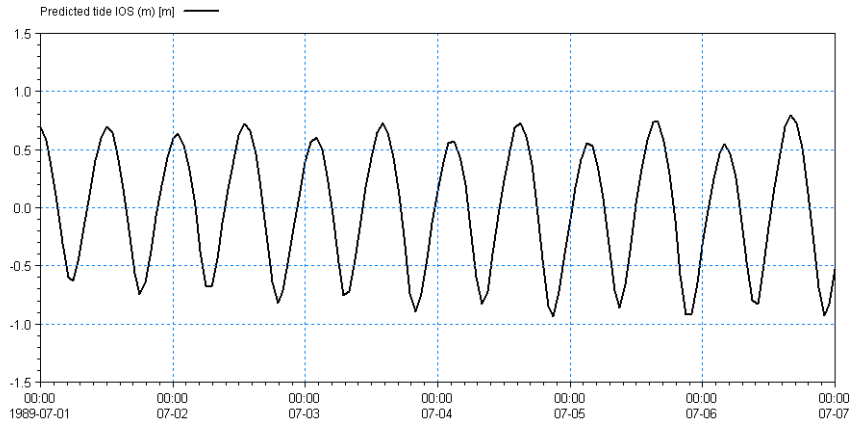
Figure 22.16   Example of generated tide data

M21 TOOLS: WAVES

The WAVES Tools in the M21 Toolbox include the facilities:

- Calculation of Reflection Coefficients
- Highpass and Lowpass Filtering of Time Series
- Generate Sponge and Porosity Maps
- Generate Wave Energy Spectrum
- Mesh Generation for MIKE 21 BW 1DH
- Generation of Random Waves
- Generation of Regular Waves
- Wave Generated Current and Setup
- Wave Spectra Converter

23 Calculation of Reflection Coefficients

Modelling of partial reflection of waves from e.g. porous rubble mound break-waters is specified through a porosity layer in the Boussinesq Wave Module, MIKE21 BW, and through a friction factor describing the frictional loss in the Elliptic Mild-Slope Wave Module, MIKE21 EMS.
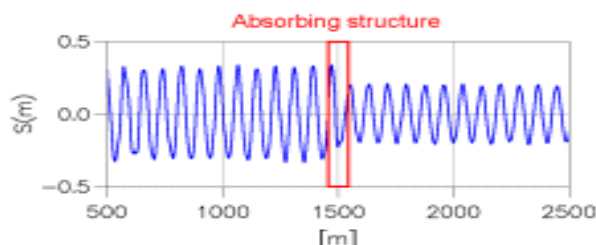
Figure 23.1   Reflection and transmission of regular waves in a numerical wave flume modelled by MIKE21 BW. Conditions: Incoming wave height = 1 m, wave period= 9 s and water depth= 10 m.

This tool can help you determine which porosity (or friction factor) to use in your wave simulation to obtain the desired reflection and/or transmission coefficient.

23.1 Specify Wave Parameters

For Boussinesq wave simulations you have to specify the expected wave height and wave period in front of the porous structure. You also have to give the water depth. In case you are using irregular waves as input to your MIKE21 BW simulation, you should take the wave height as the significant wave height. The wave period should be taken as the spectral peak wave period specified at the open boundary or at the internal generation line.

The expected significant wave height can be estimated on the basis of diffraction diagrams, see Further References *(p. 161)*. Typically, the wave period can be taken as the period specified at the boundary.

In case you want to calculate the friction factor corresponding to a MIKE21 EMS simulation you only have to specify the water depth and wave period.

23.2 Specify the Wave Absorber

The width of the absorber should usually correspond to the thickness of the porosity/friction layer used in your wave model set-up. The width should not be less than 25% of the characteristic wave length in order to be efficient. In practical short periodic harbour wave disturbance studies 3-5 porosity layers are typically used.

The various values of the porous flow parameters (diameters of stones, laminar and turbulent resistance coefficients) are not that important. Unless you have a specific reason for using custom values, the parameters can usually be left at the default values, i.e.

- Stone diameter: 0.2 m
- Laminar resistance parameter: 1000
- Turbulent resistance parameter: 2.8

Make sure that you use the same parameters as in MIKE21 BW.

In case of a MIKE21 EMS wave simulation, you only have to specify the width of the absorber.

If the core material of the absorber is permeable, the transmission coefficient will also be calculated.

23.3   Select the Porosity/Friction Factor Range

The range for the porosity should be, typically, 0.2-1 with an increment of 0.01

The range for the friction factor should be, typically, 0-50 with an increment of 0.1

23.4   Output Specification

MIKE21 BW theory
In case Boussinesq wave theory has been used for the calculation, the output file will contain the reflection coefficient (and the transmission coefficient in case of a permeable core). Also, the non-linear friction factor corresponding to the specified porosity is included in the output file.

MIKE21 EMS theory
If you have used linear theory (corresponding to the MIKE21 EMS) for the calculation, the output file will contain the reflection coefficient (and the transmission coefficient in case of a permeable core).

23.5   Overview

The overview page presents you with a summary of all the input parameters you have specified. You should inspect the parameters to check they are correct, - and if not go back and change them.

From the page you can either *Finish* the set-up, which will transfer your set-up definition to the current toolbox file, or execute the set-up.

23.6   Remarks and Hints

How to find the porosity/friction value?

As the reflective characteristics of an absorbing structure is mainly a function of the wave height, wave period and water depth, you often have to apply this tool for a number of different combinations of these parameters. Typical examples of the output of this tool is presented below.

23.6.1 How to find the friction factor to be used in MIKE 21 EMS

A typical example of the output of this tool is presented below.
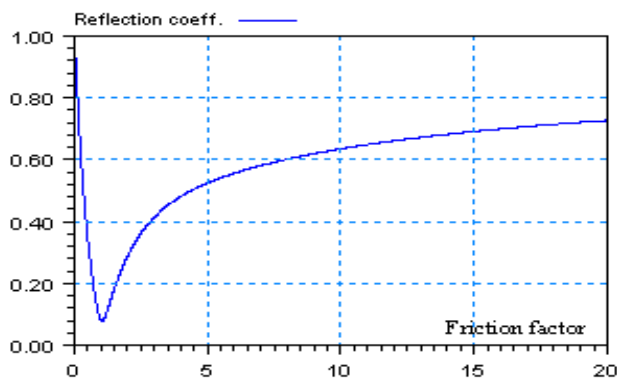
Figure 23.2   Reflection coefficient versus friction factor. Conditions: wave height= one unit (linear model), wave period= 8 s and water depth= 10 m. The width of the absorber is 20 m and the core is assumed impermeable.

As seen from Figure 23.2 two different friction values may yield the same reflection coefficient. You should usually select the friction value corresponding to the second part of the curve, i.e. use a friction value of 2.86 obtain 40 % reflection instead of 0.44 in Figure 23.2.

23.6.2 How to find the porosity to be used in MIKE 21 BW

A typical example of the output of this tool is presented below:
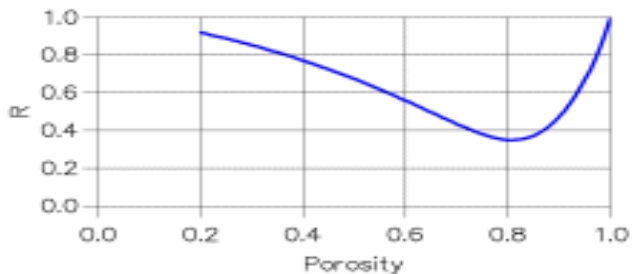
Figure 23.3   Reflection coefficient versus porosity. Conditions: Significant wave height = 1 m, spectral peak wave period= 9 s and water depth= 10 m. The width of the absorber is 15 m and the core is impermeable.

As seen from Figure 23.3 two different porosities may yield the same reflection coefficient. You should usually select the porosity which corresponds to the part of the curve with the smallest slope (e.g. use a porosity of 0.73 to obtain 40 % reflection instead of 0.87 in Figure 23.3). If, however, the desired reflection coefficient requires a porosity less than about 0.4, it is recommended you consider the steepest part of the curve.

23.6.3 What is a typical value of the reflection coefficient?

Typically, the reflection coefficient for a rubble mound structure is 30-50 %. More information about reflection, absorption and transmission characteristics can be found in the paper by Thompson et al (1996), see Further References *(p. 161)*.

Figure 23.4 shows typical reflection coefficient ranges for short period waves.
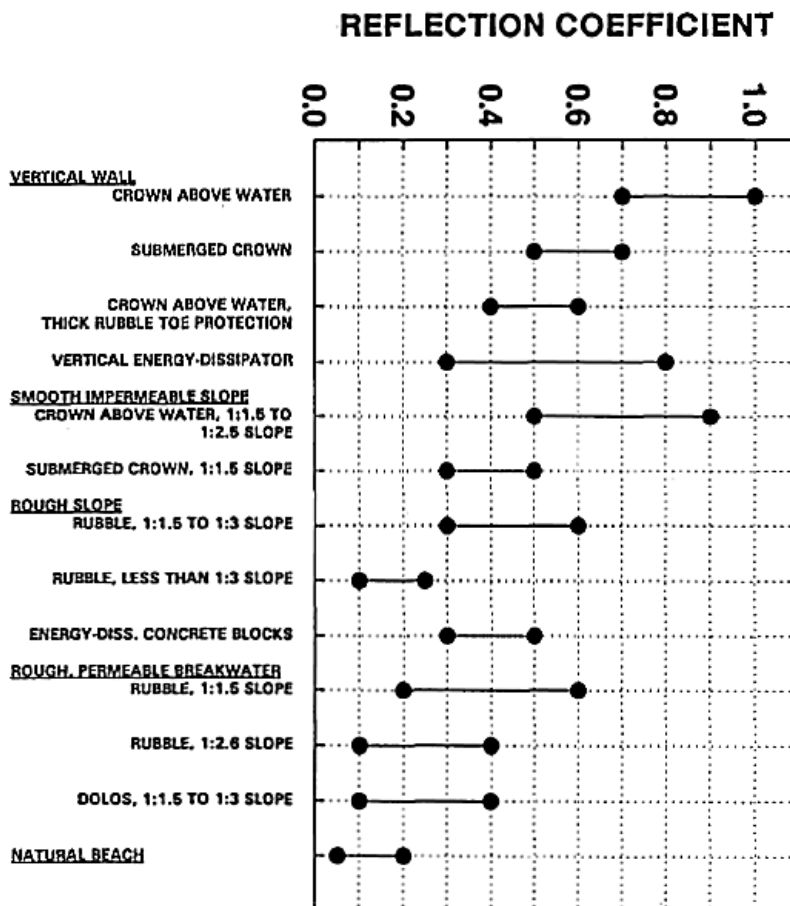
Figure 23.4   Typical reflection coefficient ranges for short period waves (T<20s). From Thompson et al (1996), see Further References *(p. 161)*.

23.7   Scientific Background

The scientific background of this tool is given in the papers listed below:

- Madsen, P.A., 1983: Wave reflection from a vertical permeable wave absorber. Coastal Engineering, 7. 381-396.

- Madsen, P.A. and I.R. Warren, 1984: Performance of a short wave model. Coastal Engineering, 8. 73-93.

23.8   Further References

The references listed below provide you with more basic information pertaining to this tool:

- Thompson, E. F., H. S. Chen and L. L. Hadley, 1996: Validation of numerical model for wind waves and swell in harbours. *J. of Waterway, Port, Coastal and Ocean Engineering*, **122**,5. 245-257.

- Shore Protection Manual, 1984: 2 volumes, 4th edition, US Army Corps of Engineers, Waterways Experiment Station, Vicksburg, MS, USA.

24 Highpass and Lowpass Filtering of Time Series

With this tool you can read in time series of eg. surface elevation, decompose it into its Fourier components, cut off energy on frequencies below (highpass filter) and above (lowpass filter) specified limits, and regenerate the time series with waves on frequencies in the specific range. The resulting raw spectrum can also be saved. An example is illustrated in Figure 24.1.
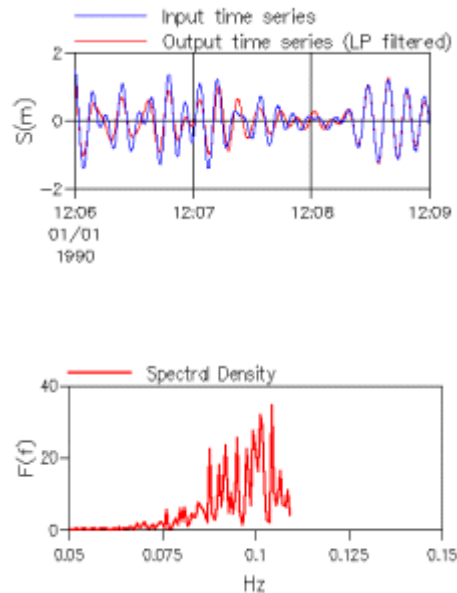
Figure 24.1   Figure 1 Input time series of surface elevation, filtered time series and the resulting raw spectrum.The cut-off frequency is 0.11 Hz.

The frequency domain analysis is based on the Fast Fourier Transform (FFT), see eg. Scientific Background for general treatment of frequency domain analysis.

24.1 Selecting the Input Data

The input data may an extracted time series of surface elevation from a Boussinesq wave simulation. If the input data file contains more than one item you have to specify which item you would like to process.

24.2 Specification of the Frequency Range

Here you specify the lower and upper frequencies of interest.

If the resulting wave spectrum (for the processed signal) should be saved, a name and title must be specified.

Note that the spectrum is the raw spectrum of the time series. The output data file contain the spectral density (m^2/Hz) and the corresponding frequency (Hz).

24.3 Specification of the Output Subseries

Here you specify the subseries for the filtered time series. Most often the subseries is identical to input time series.

24.4 Specification of the Output Data

The output data file contains the time series with wave frequencies in the specified range.

24.5 Overview

The overview page presents you with a summary of all the input parameters you have specified. You should inspect the parameters to check they are correct, - and if not go back and change them.

From the page you can either *Finish* the set-up, which will transfer your set-up definition to the current toolbox file, or execute the set-up.

24.6 Scientific Background

The scientific background of this tool is given in the textbook listed below:

- Bendat, J.L. and A.G.Piersol, 1986: Random Data. Analysis and Measurements Procedures. John Wiley & Sons, New York

24.7 Remarks and Hints

Beside the application of this tool for removal of long and short periodic waves from a simulated or measured time series, the tool can also be used for an intial assessment of possible long periodic oscillations in ports and harbour bassins. In that case the input data should be extracted from the results data file from a Boussinesq wave simulation.

25 Generate Sponge and Porosity Maps

This tool is a MIKE 21 Tool to generate sponge and porosity maps for application in MIKE 21 BW Boussinesq Wave Module and MIKE 21 EMS Elliptic Mild-Slope Wave Module.

Based on the bathymetry file the tool can be used to generate initial maps containing either sponge layer coefficients or porosity values (friction coefficients in case of MIKE 21 EMS). Afterwards you can adjust the generated maps to fulfil the overall modelling requirements.

25.1 Bathymetry Input

The bathymetry is specified as a 2D map, ie. file format dfs2 or dt2.

NOTE: All bathymetry values besides the land values should be negative as positive values are considered to be code values that may affect the generation of the map.

MIKE 21 BW 1DH
Please notice that this tool can also be used to generate sponge layer files (type dfs1) for application in MIKE 21 BW 1DH. The procedure is the same as for 2D maps (type dfs2).

25.2 Parameter Settings

On this page you specify the various parameters. A definition is given below.

Select area
You specify the area where you want to generate the map. Typically the range would be identical to the range of the bathymetry.

Add values along bathymetry value
You specify the bathymetry value along which you want to generate a sponge or porosity/friction layer. The value may be different from the land value.

Include corners in search
For a proper generation of sponge layers in connecting corners this option should be set. In case of generation of porosity layers the 'corner search' may result in too many layers in front of your porous structure if they have an oblique orientation relative to the bathymetry grid. Therefore you should exclude this option in that case.

Number of layers
Here you specify the number of layers of sponge or porosity/friction coefficients.

In case of sponge layers you will usually apply 10-20 layers for short wave applications. For simulation of long periodic waves (particularly for MIKE21 EMS and oblique incident waves) you may need to use more layers to absorb the waves.

In case of porosity/friction layers you will usually apply 3-5 layers of constant values as a minimum. It is recommended to create a total layer thickness corresponding to 25% of the wave length.

Sponge layer generation

For generation of sponge layers you should always use this option. The sponge layer coefficients is calculated using the formula:

$$C_{sponge} = a^{r^{i-1}} \quad , \quad i = 1, N_{sponge} \qquad (25.1)$$

where a is the base value, r is the power value and N_{sponge} is the number of sponge lines (counting from the closed boundary towards the first grid line).

Depending on the number of layers you may use the values listed in Table 25.1.

Table 25.1 Recommended values for sponge layer coefficients

N_{sponge}	a	r
10	5	0.5
20	7	0.7
50	10	0.85
100	10	0.92
200	10	0.95

Porosity layer generation.

You specify the value for the porosity/friction factor.

Background value

The background value is typically set to one for calculation of sponge layers and porosity layers. In case of friction layers (for application in MIKE 21 EMS) the background value is typically zero.

25.3 Output Specification

The generated map is saved as a 2D file, ie. file format dfs2 or dt2.

25.4 Overview

The overview page presents you with a summary of all the input parameters you have specified. You should inspect the parameters to check they are correct, - and if not go back and change them.

From the page you can either *Finish* the set-up, which will transfer your set-up definition to the current toolbox file, or execute the set-up.

25.5 Examples

25.5.1 Example of generation sponge layer maps

The purpose of this example is to illustrate how to generate a sponge layer map. The generated map may be used in connection with modelling of the phenomenon of diffraction. Diffraction is important in regions that are sheltered by barriers such asbreakwaters, jetties or inlets.

Defining the bathymetry
The bathymery is shown in .Figure 25.1

Figure 25.1   Model bathymetry. The blue area indicates a constant water depth of 40 m (bathymetry = -40) and the grey area land having bathymetry= 10. The red area has a bathymetry value of 9 and a sponge layer is to be calculated in front of this.

Parameters
The selected area is the entire bathymetry (ie. x= 0:120 and y=0:70), the background value is 1, number of layers is 10 and corners are included in the search.

We use 'Sponge Layer Generation' setting the 'Add values along bathymetry value' to 9. The value of 9 is edited in the bathymetry along the boundaries where we want to have sponge layers. The base value is 0.5 and power value 5.

Results

The generated sponge layer map is shown on Figure 25.2 below. The sponge layer is indicated by green colour. In this case the map can be used directly in MIKE 21 BW or MIKE 21 EMS wave simulations.

Figure 25.2   Generated sponge layer map. The sponge layer area is indicated by the green colour.

25.5.2   Example of generation porosity/friction layer maps

The purpose of this example is to illustrate how to generate a porosity/friction layer map. The generated map may be used in connection with modelling of the partial refelction or transmisson in the wave models MIKE 21 BW or MIKE 21 EMS.

Defining the bathymetry

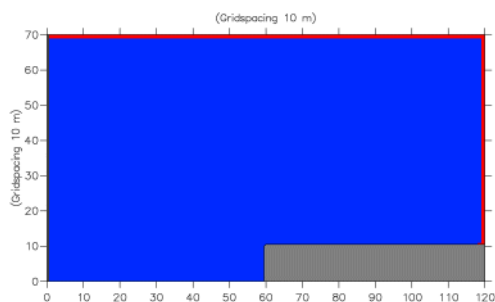The bathymery is shown in Figure 25.3.
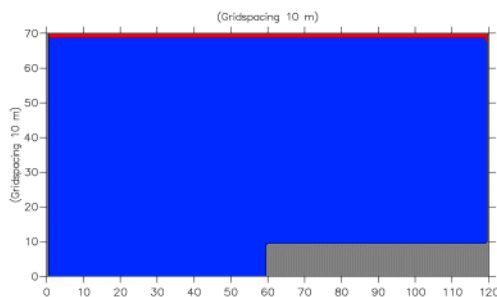
Figure 25.3   Model bathymetry. The blue area indicates a constant water depth of 40 m (bathymetry = -40) and the grey area land having bathymetry= 10. The red area has a bathymetry value of 9 and a sponge layer is to be calculated in front of this.

Parameters

The selected area is the entire bathymetry (ie. x= 0:120 and y=0:70), the background value is 1, number of layers is 3 and corners are included in the

search. We 'Porosity Layer Generation' setting the 'Add values along bathymetry value' 9 (red value). The value of 9 is edited in the bathymetry along the boundaries where we want to have porosity/friction layers. The constant value is 0.76.

Results

The generated porosity layer map is shown on Figure 25.4 below. The porosity layer is indicated by green colour. In this case the map can be used directly in MIKE 21 BW or MIKE 21 EMS wave simulations.
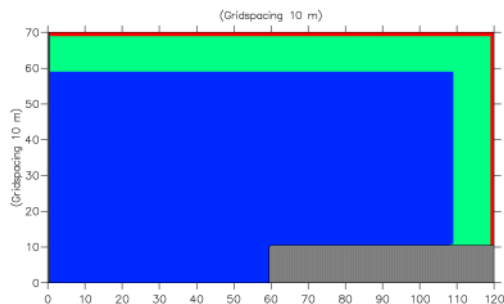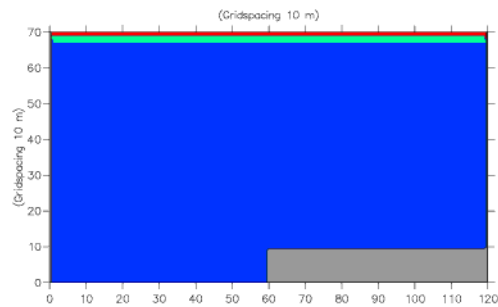

Figure 25.4   Generated porosity layer map. The porosity layer area is indicated by the green colour.

26 Generate Wave Energy Spectrum

With the MIKE 21 Tool Generate Wave Energy Spectrum, you can generate a discrete two-dimensional directional frequency spectrum which corresponds to a specified directional spreading function. The generated 2D spectrum can then be used as input along the offshore boundary in a MIKE 21 PMS (Parabolic Mild-Slope module) simulation. A number of different wave spectra and directional distribution functions can be specified.

26.1 Simulation Period

The MIKE 21 Tool Generate Wave Energy Spectrum can be run in two modes: *Constant in Time* and *Varying in Time*. Constant in Time is used when you want to create a spectrum corresponding to a single wave event, and Varying in Time is used when you want to create a (time-varying) spectrum corresponding to a time series of wave events.

Please note that these wave events are mutually independent.

The simulation period must be specified using the start date, the end date and the interval (in seconds) between each time step, and should cover the entire simulation period in the subsequent MIKE 21 PMS simulation.

26.2 Frequency Spectrum

You may specify the frequency spectrum from which the directional frequency spectrum is generated in one of the following ways.

- A Pierson-Moskowitz spectrum
- A JONSWAP spectrum
- A TMA spectrum
- Monochromatic spectrum
- Time series of surface elevations

A type 0 data file is required for the wave climate if the simulation is to be made with a time-varying input wave data at the offshore boundary.

26.2.1 Type of frequency spectrum

1. A Pierson-Moskowitz spectrum specified through the following parameters:
   - Significant wave height, H_{mo}
   - Peak wave period, T_p

2. A JONSWAP spectrum is specified through the following parameters:

- – Significant wave height, H_{mo}
- – Peak wave period, T_p
- – Shape parameters: σ_a, σ_b, and γ

3. A TMA spectrum is specified through the following parameters:

   - – Significant wave height, H_{mo}
   - – Peak wave period, T_p
   - – Shape parameters: σ_a, σ_b, and γ
   - – Average water depth (m).

   The average water depth specified here must correspond to the average water depth along the offshore boundary of the MIKE 21 PMS model.

4. Monochromatic waves can also be used in generating the directional frequency spectrum. In this case, all the energy will be concentrated in one frequency and distributed with directions according to the specified directional spreading function. Monochromatic waves are specified through:

   - – Wave height, H
   - – Wave period, T

26.2.2 Type of frequency discretisation

Next, you will need to specify the type of frequency discretisation. Here you can select a constant frequency interval in which case the frequency spectrum will be divided into energy bands with constant frequency interval. This would lead to a varying amount of energy in each discrete frequency. Alternatively, you can specify discretisation with constant energy interval. This implies that each discrete frequency will have the same energy content associated with it. This discretisation leads to a varying interval between the discrete frequencies.

You will also have to specify the minimum and maximum frequency cutoff (f_{min}, f_{max}) for the spectrum, plus the number of discrete frequencies. Note that the energy below f_{min} and above f_{max} is folded back onto the discretised spectrum, so that the total energy in the discretised spectrum is the same as in the continuous spectrum.

26.3 Directional Distribution

Next, you will have to specify the directional distribution of wave energy. You may specify this distribution in one of the following ways.

- • A $\cos^n(\theta-\theta_o)$ - distribution
- • A $\cos^{2s}[\frac{1}{2}(\theta-\theta_o)]$ - distribution
- • DHI distribution
- • MGS distribution

- Gaussian distribution
- Unidirectional waves

A type 0 data file is required for the wave climate if the simulation is to be made with a time-varying input wave data at the offshore boundary.

26.3.1 Type of directional distribution

1. A $\cos^n (\theta-\theta_o)$ where θ_o is the mean wave direction and n is the directional spreading index. This distribution is specified through:

   – Mean wave direction, θ_o
   – Maximum deviation from θ_o, $|\theta-\theta_o|_{max}$
   – Spreading index, n

2. A $\cos^{2s} [\frac{1}{2}(\theta-\theta_o)]$ where s is a user specified shape parameter. This distribution is specified through:

   – Mean wave direction, θ_o
   – Maximum deviation from θ_o, $|\theta-\theta_o|_{max}$
   – Shape parameter, s

3. DHI distribution. This is a $\cos^{2s} [\frac{1}{2}(\theta-\theta_o)]$ distribution where s is a frequency dependent shape parameter. This distribution is specified through:

   – Mean wave direction, θ_o
   – Maximum deviation from θ_o, $|\theta-\theta_o|_{max}$

4. MGS distribution. This is a $\cos^{2s} [\frac{1}{2}(\theta-\theta_o)]$ distribution, where s is a frequency dependent shape parameter. The difference between the MGS and DHI distributions is in the functional form of s as a function of frequency. This distribution is specified through:

   – Mean wave direction, θ_o
   – Maximum deviation from θ_o, $|\theta-\theta_o|_{max}$
   – Maximum spreading index, s_{max}

5. Gaussian (normal) distribution. This distribution is specified through:

   – Mean wave direction, θ_o
   – Maximum deviation from θ_o, $|\theta-\theta_o|_{max}$
   – Direction spreading, σ

6. Unidirectional waves can also be used in conjuction with a frequency spectrum to generate the directional frequency spectrum. In this case all the energy will be concentrated in one direction and distributed with frequency according to the specified frequency spectrum. Unidirectional waves are specified through:

   – Wave direction.

26.3.2 Type of directional discretisation

You will need to specify the discretisation of the energy spectrum with direction. As in the frequency discretisation, you can select a constant interval between the discrete directions, or a constant energy in each discrete direction. The latter will lead to a varying interval between the discrete directions, while the former gives a varying amount of energy in each discrete direction.

You will also need to specify the number of discrete directions that will be used. The minimum and maximum directions are calculated from the max. deviation from mean wave direction $|\theta-\theta_0|_{max}$ and the mean wave direction θ_0. Note that the energy below θ_{min} and that above θ_{max} is folded back onto the discretised spectrum, so that the total energy is conserved in the discretised spectrum.

26.4 Output

Three types of output data files can be obtained from the MIKE 21 Tool Generate Wave Energy Spectrum. These are:

- The directional frequency spectrum, $E=E(f,\theta)$. This is the main output file to be used in MIKE 21 PMS. The file (a type 2 file) contains the energy, the discrete frequencies and the discrete directions.

- The frequency spectrum, $E=E(f)$. This type of output file is specified **only** for the stationary case and for random (non-monochromatic) waves. The file (a type 0 file) contains the energy and the discrete frequencies.

- The directional distribution, $E=E(\theta)$. This type of output file is specified **only** for the stationary case and for directional waves. The file (a type 0 file) contains the energy and the discrete directions.

26.5 Overview

The overview page presents you with a summary of all the input parameters you have specified. You should inspect the parameters to check they are correct, - and if not go back and change them.

From the page you can either *Finish* the set-up, which will transfer your set-up definition to the current toolbox file, or execute the set-up.

26.6 Scientific Background

26.6.1 Frequency spectrum

Pierson-Moskowitz spectrum

A Pierson-Moskowitz spectrum is defined as:

$$S(f) = A \cdot f^{-5} \exp(-B \cdot f^{-4}) \tag{26.1}$$

where

$$A = \frac{5}{16} \frac{H_{m0}^2}{T_p^4} \tag{26.2}$$

$$B = \frac{5}{4} \frac{1}{T_p^4}. \tag{26.3}$$

Jonswap spectrum

A JONSWAP spectrum is defined as:

$$S(f) = S_{PM}(f) \cdot \gamma^a \tag{26.4}$$

where

$S_{PM}(f)$ = Pierson-Moskowitz spectrum defined above

γ = user-selected shape parameter. A value of 3.3 corresponds to standard JONSWAP spectrum

$$a = \exp\left[-\frac{1}{2}\left(\frac{f - f_p}{\sigma f_p}\right)\right] \tag{26.5}$$

$$\sigma = \begin{cases} \sigma_a & \text{for } f \leq f_p \\ \sigma_b & \text{for } f > f_p \end{cases} \tag{26.6}$$

Values corresponding to standard JONSWAP spectrum are σ_a = 0.07 and σ_b = 0.09.

TMA spectrum

A TMA spectrum is defined as:

$$S(f) = S_{JONSWAP}(f) \cdot \phi(\omega_h) \tag{26.7}$$

where

$S_{JONSWAP}(f)$ = JONSWAP spectrum defined above

$$\phi(\omega_h) = \begin{cases} \dfrac{1}{2}\omega_h^2 & \text{for } \omega_h \leq 1 \\[2mm] 1 - \dfrac{1}{2}(2 - \omega_h) & \text{for } 1 < \omega_h \leq 2 \\[2mm] 1 & \text{for } \omega_h > 2 \end{cases}$$

(26.8)

$$\omega_h = 2\pi f \sqrt{h/g}$$

(26.9)

and h = water depth

26.6.2 Directional discretisation

The equations for the directional distributions are given below:

1. A $\cos(\theta - \theta_0)$ distribution is defined as

$$D(\theta) = \cos^n(\theta - \theta_0)$$

(26.10)

where n is a user-specified spreading index.

2. A $\cos^{2s}\left(\dfrac{1}{2}(\theta - \theta_0)\right)$ distribution is defined as

$$D(\theta) = \cos^{2s}\left(\frac{1}{2}(\theta - \theta_0)\right)$$

(26.11)

where s is a user-specified spreading index.

3. The DHI distribution is defined as:

$$D(\theta) = \cos^{2s}\left(\frac{1}{2}(\theta - \theta_0)\right)$$

(26.12)

where

$$s = \frac{2}{\sigma} - \frac{1}{2} \tag{26.13}$$

$$\sigma = 0{,}24733\left(\frac{f}{f_p}\right)^{2{,}5} + 0{,}07153\left(\frac{f_p}{f}\right)^{2{,}5} - 0{,}1377 \tag{26.14}$$

$$f_p = \frac{1}{T_p} \tag{26.15}$$

4. The MGS distribution is defined as:

$$D(\theta) = \cos^{2s_{max}}\left(\frac{1}{2}(\theta - \theta_0)\right) \tag{26.16}$$

where

s_{max} = user-specified maximum spreading index.

5. The Gaussian (normal) distribution is defined as:

$$D(\theta) = \frac{1}{\sigma\sqrt{2\pi}}\exp\left[-\frac{(\theta - \theta_0)^2}{2\sigma^2}\right] \tag{26.17}$$

where

σ = user-specified spreading index.

27 Mesh Generation for MIKE 21 BW 1DH

This tool is a MIKE 21 tool to generate depth-adaptive and unstructured mesh for application in the MIKE 21 BW Boussinesq Wave Model 1DH. The MIKE 21 BW 1DH model is typically used for calculation of wave transformation from offshore to the beach for studying surf zone and swash zone dynamics.

Based on a coastal profile bathymetry (dfs1 type data file) this tool can be used to generate a refined depth-adaptive and unstructured mesh of your bathymetry, which subsequently can be used in the Boussinesq Wave Model MIKE 21 BW 1D.
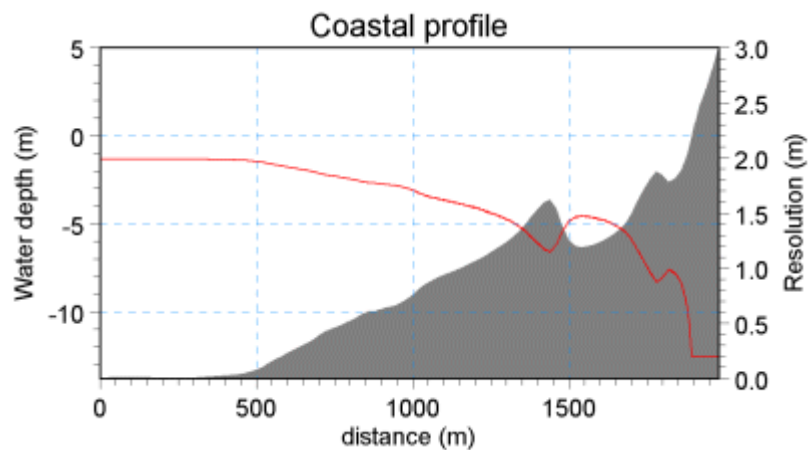

Figure 27.1   Example of spatial resolution for a barred coastal profile.
Conditions: Spectral peak period 8 s, 40 nodes per wave length and minimum mesh spacing 0.2 m.

27.1 Input File Specification

The bathymetry is specified as a dfs1 type data file.

Usually your data file has its origin from an encompassed 2D bathymetry and the item type is 'bathymetry'.

If the item type is not 'bathymetry' you have to re-defined the item type before you can use this tool.

You can edit and re-define the item information using the MIKE Zero Data Utility Tool (select 'properties') available in the Line Series Editor (double click on the dfs1 data file).

27.2 Parameter Specification

On this page you specify the mesh generation parameters.

You can control the mesh generation by either specifying the maximum number of nodes or the number of nodes per wave length.

27.2.1 Maximum number of nodes

This is the default generation method where the maximum number of nodes is taken from the input dfs1 data file.

The maximum number of nodes is distributed relatively to the local wave length, which depends on the characteristic wave period and water depth. By increasing the maximum number of nodes the number of nodes per wave length can be increased.

27.2.2 Number of nodes per wave length

A very small spatial resolution is required to model wave breaking and wave run-up using MIKE 21 BW. In practical applications it has been found that the minimum wave length should be resolved by at least 30-40 grid points in the surf and swash zone.

27.2.3 Characteristic wave period

It is recommended to use the spectral peak wave period as a characteristic wave period.

27.2.4 Minimum mesh spacing

It is recommended to use a minimum mesh spacing corresponding to 40 nodes per wave length.

27.2.5 Information on the generated mesh

Information on the generated mesh is also available from this dialog. By clicking on the 'View Mesh' button you are able to evaluate the generated mesh. If you are not satisfied with the mesh you may alter some of the mesh generation parameters and generate a new mesh.

27.2.6 Wave theory

The linear dispersion relationship (Stokes first order theory) is used to calculate the wave length.

27.3 Output File Specification

The generated non-equidistant mesh is saved as a profile file, i.e. dfs1 type data file.

The dfs1 type data file includes two items:

1. co-ordinate (in meter) of the node in the mesh
2. the corresponding water depth in the node

27.4 Overview

The overview page presents you with a summary of all the input parameters you have specified. You should inspect the parameters to check they are correct, - and if not go back and change them.

From the page you can either *Finish* the set-up, which will transfer your set-up definition to the current toolbox file, or execute the set-up.

28 Generation of Random Waves

Based on a specified frequency spectrum and a directional distribution this tool generates a line or time series of surface elevation, flux density and derived quantities. An example of a directional frequency spectrum is shown in Figure 28.1.
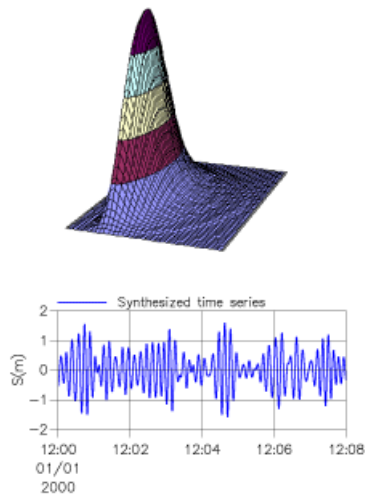

Figure 28.1   Directional wave spectrum (upper) and generated time series of surface elevation (lower). Conditions: JONSWAP spectrum with standard shape parameters, cos^2-directional distribution, H_{m0}= 2.5 m, T_p= 10 s and water depth= 12 m.

The generated data can be used as open boundary condition or as input at an internal generation line in the Boussinesq Wave Module, MIKE21 BW, for simulation of directional irregular waves. An example of a generated time series is shown in Figure 28.1, while the results of a Boussinesq wave simulation using that input is illustrated in Figure 28.2.
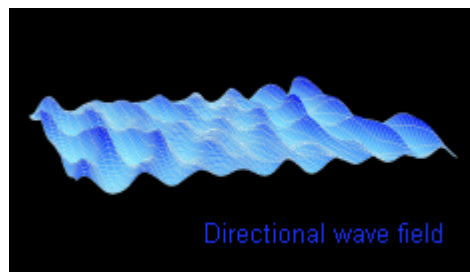

Figure 28.2   Directional wave field simulated by MIKE21 BW.

28.1 Specification of Frequency Spectrum

Here you specify the type of frequency wave spectrum from which the time or line series is to be generated:

- Pierson-Moskowitz spectrum
- JONSWAP spectrum
- TMA spectrum
- Monochromatic waves
- Bichromatic waves
- Time series of surface elevation
- User defined frequency spectrum (1D)
- User defined directional spectrum (2D)

28.1.1 Pierson-Moskowitz spectrum

The Pierson-Moskowitz spectrum (valid for fully developed sea states) can be specified through the following parameters:

- Wind speed 19.5 m above the sea surface, $U_{19.5}$
- Significant wave height, H_{m0}
- Significant wave height, H_{m0}, and mean wave period, T_{02}

The definition of the Pierson-Moskowitz spectrum and related information can be found in Definition of Pierson-Moskowitz spectrum *(p. 196)*.

28.1.2 JONSWAP spectrum

The JONSWAP spectrum (valid for fetch limited sea states) is specified through the following parameters:

- Wind speed 10 m above the sea surface, U_{10}, and effective fetch length, F
- Significant wave height, H_{m0}, peak wave period, T_p, and the shape parameters σ_a, σ_b and γ

The definition of the JONSWAP spectrum and related information can be found in Definition of JONSWAP spectrum *(p. 197)*.

28.1.3 TMA spectrum

The TMA spectrum is specified through the following parameters:

- Significant wave height, H_{m0}

- Peak wave period, T_p
- Shape parameters: σ_a, σ_b, and γ
- Average water depth

The average water depth is specified along the open boundary on the following page. The depth must correspond to the value at the internal generation line of the MIKE21 BW model if used in connection with the latter.

The definition of the TMA spectrum and related information can be found in Definition of the TMA spectrum *(p. 198)*.

28.1.4 Monochromatic waves

Monochromatic waves can also be used in synthesising the time or line series of random waves.

In this case, all the energy will be concentrated on one frequency and distributed with directions according to the specified directional distribution function. Monochromatic waves are specified through:

- Wave height, H
- Wave period, T

28.1.5 Bichromatic waves

Bichromatic waves can be synthesised in the same way as monochromatic waves. All the energy is here concentrated on two frequencies. Bichromatic waves are specified by

- Wave height of first wave, H
- Wave period of first wave, T
- Wave height of second wave, H
- Wave period of second wave, T

28.1.6 Time series of surface elevation

In case you want to synthesise BC's (Boundary Conditions) for MIKE 21 BW (time or line series) based on a time series of e.g. measured surface elevation, you specify the name of that dfs0 data file on this dialog.

From the specified file you have to select one item:

- Item for the measured/calculated surface elevation

Recommendations

The FFT-transformation is based on linear wave theory. It means that you should not use a time series of surface elevation measured in very shallow water (say, water depths less than 5 m) where nonlinear effects make the wave profile asymmetric vertically as well as horizontally. From a time series plot it is quite easy to identify whether the waves are linear or not.

Always make a comparison between the input time series and the output (surface elevation).

28.1.7 User defined frequency spectrum (1D)

In case you want to synthesise the time or line series based on e.g. a measured frequency spectrum or a spectrum extracted from a spectral wave model, here you must specify the name of that data file (of time series type, i.e. a .dt0 or .dfs0 data file).

From the specified file you have to select the two items:

- Item for the frequency (unit Hz)
- Item for the corresponding spectral density (unit m²/Hz)

28.1.8 User defined directional spectrum (2D)

A directional sea can synthesised according to a user supplied directional frequency spectrum in dfs2-format. Such a spectrum could be obtained as a point spectrum output from a MIKE 21 SW model or from other sources (see also MIKE 21 Toolbox: Wave Spectra Converter). This alternative to frequency spectrum combined with a prescribed directional distribution function allows for synthesis of multimodal seastates with e.g. swell and wind-sea coming from different directions.

From the specified file, which should have frequency (unit Hz) as first spatial axis and directions of waves (unit degree) as second spatial axis, you have to select:

- Item for wave energy spectrum (unit m²/Hz/rad)

28.2 Wave Generation

Type of waves

Which type of wave data you would like to generate depends on the purpose of your Boussinesq wave simulation. You can choose between following types:

- One-dimensional waves
- Unidirectional waves

- Directional waves

Auxililary parameters

In order to synthesise the required time or line series of data you have to specify a number of parameters:

- Random seed

- Water depth

- Minimum wave period

- Re-scaling truncated spectrum

Specification of line for internal generation

If you have selected unidirectional or directional waves you also have to specify the Position of generation line.

Synthesis

The wave generation is stochastic based on the specified spectrum, inverse FFT method and random phases and directions. For more information, see eg. Bendat and Pierson (1986) /25/.

28.2.1 One-dimensional waves

'One-dimensional waves' is selected if you would like to generate wave data for modelling of normally incident irregular long-crested waves, i.e. waves having an angle of 90 deg. relative to the generation line. An example is shown Figure 28.3.
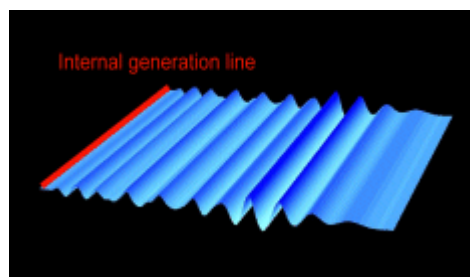

Figure 28.3   Boussinesq wave simulation using one-dimensional wave generation input data (time series data).

28.2.2 Unidirectional waves

'Unidirectional waves' is selected if you would like to generate wave data for modelling of obliquely incident irregular (or regular) long-crested waves. An example is shown Figure 28.4.
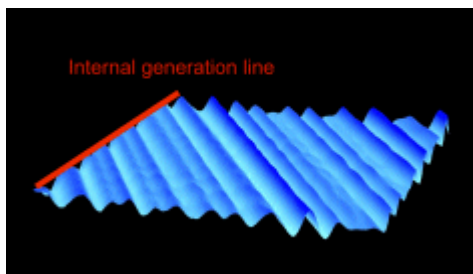
Figure 28.4 Boussinesq wave simulation using unidirectional wave generation input data (line series). The angle between the normal to the generation line and the wave direction is in this case 30 deg.

28.2.3 Directional waves

'Directional waves' is selected if you would like to generate wave data for modelling of short crested irregular waves. An example is shown Figure 28.5.
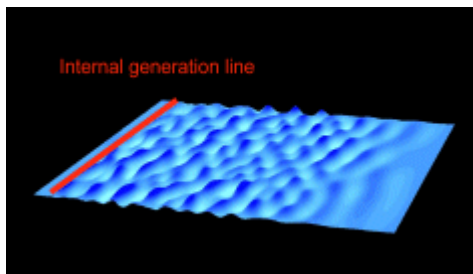

Figure 28.5 Boussinesq wave simulation using directional wave generation input data (line series).

28.2.4 Random seed

The seed is used for the calculation of the random phases in the Fourier description, and for the calculation of the random wave directions.

Thus, you can reproduce a random time or line series by specifying the same seed (and frequency spectrum, directional spreading etc.).

28.2.5 Water depth

For the calculation of the flux density along an open boundary or internal generation line, here you should specify a representative value for the water depth.

Note that the water depth along the offshore boundary in your MIKE21 BW model set-up should be nearly constant.

28.2.6 Minimum wave period

Here you have to specify the smallest wave period you would like to resolve in your Boussinesq wave simulation.

This wave period should not be less than the minimum period, which you have determined during the selection of your MIKE21 BW model parameters, i.e. time step and grid spacing.

28.2.7 Re-scaling truncated spectrum

As the high frequency part ($f>f_{cut-off}$) of the original spectrum is removed, the resulting integral parameters such as the significant wave height and mean wave periods are altered relative to the input specifications.

If you would like to maintain the significant wave height as specified, then you can choose to re-scale the spectrum. Thus, the total energy in the truncated spectrum is the same as in the original specified spectrum, see Figure 28.6.
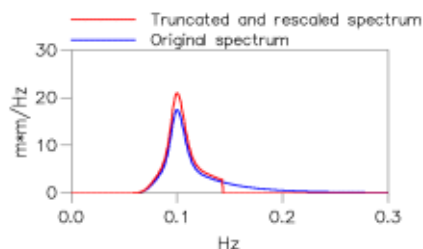

Figure 28.6   Example of the original and truncated spectrum. The spectrum is a standard JONSWAP spectrum (i.e. having the shape parameters σ_a= 0.07, σ_b= 0.09, and γ= 3.3). H_{m0}= 3 m and T_p= 10 s. The cut-off wave period is 7 s (corresponding to a frequency of 0.143 Hz).

28.2.8 Position of generation line

Definition The (internal) generation line should be parallel to either the x-axis or y-axis of your bathymetry used in MIKE 21 BW. As illustrated in Figure 28.7, the waves are always sent off from the left hand side of the line as seen from the starting point, see also Figure 28.8. You should be careful only to send off waves to one side of the generation line, or in other words $\Delta\theta_{max}$ should be less than θ_{main}.

Important: The Main wave direction should always be related to the general (or global) x-axis of your model bathymetry and not the internal generation line itself.

Two or more generation lines In Figure 28.9 it is shown how to join internal generation lines in a corner.

Grid spacing You must also specify the grid spacing in the x- and y-directions corresponding to the bathymetry used for the Boussinesq wave simulation.
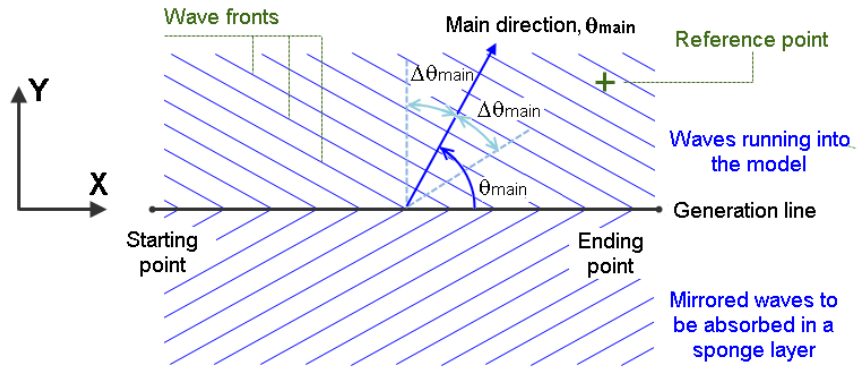

Figure 28.7   Definition of main wave direction θ_{main} (BW) and maximum deviation from main wave direction $\Delta\theta_{max}$. Please note that the co-ordinate system refers to the general (or global) co-ordinate system.
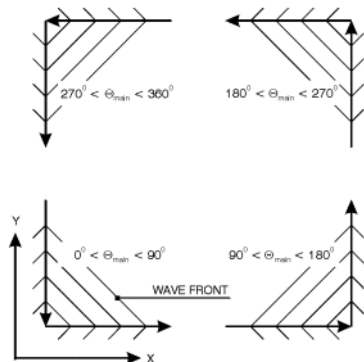

Figure 28.8   Definition of generation lines and main wave directions θ_{main} (BW), when using two generation lines to generate a wave field.
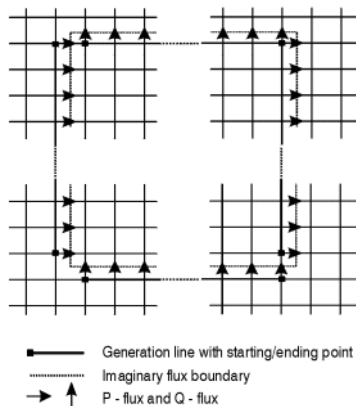
Figure 28.9   How to join internal generation lines in a corner.

28.3   Second Order Wave Generation

The default wave generation is of first order--meaning that the synthesised wave record can be considered as a simple sum of sinusoidal wave components. This is an adequate description if the wave generation line in MIKE 21 BW is positioned in sufficiently deep waters compared to the wave height. If this, however, is not the case waves will be non-linear and so should the applied wave series at the generation line be. If linear waves are applied where non-linear effects are significant, spurious (non-physical) waves will be released potentially compromising the quality of the MIKE 21 BW results.

Second order corrections to the linear wave generation can be enabled here. This means that bound sub- and superharmonic components will added to the primary wave components. Please consult section Second order wave generation *(p. 200)* for the scientific background.

Note: longer computation time should be expected as transfer of energy from all bichromatic pairs to the bound sub- and super-harmonic components must be considered.

28.4   Specification of the Historical Date

Here you specify the start time (historical date of the first time step), the total number of time steps, and - finally - you specify the time step.

It is recommended that the specified time step is the same as in your Boussinesq wave simulation. Otherwise a linear interpolation is used.

28.5 Description of Directional Distribution

Next, you will have to specify the directional distribution of wave energy.

28.5.1 Convention of angles

First choose which convention of angle to use when defining the main wave direction, θ_{main}. You have two options:

- Scientific / BW
- Meteorological

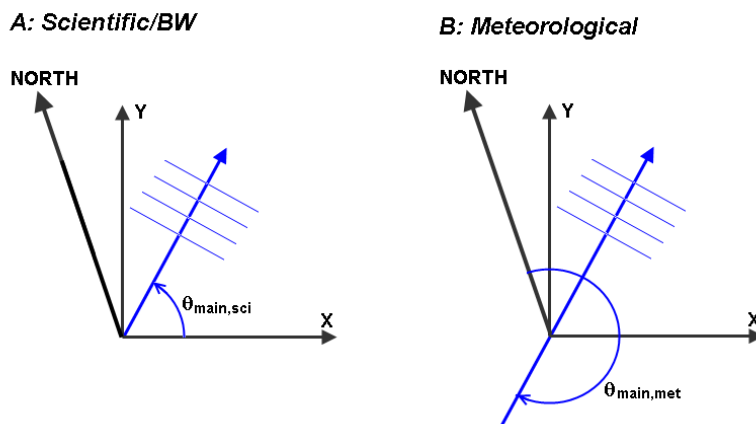**A: Scientific/BW**

B: Meteorological

Figure 28.10   Definition of the two types of wave angle conventions

The Scientific / BW convention relates the wave direction to the general (or global) x-axis of your MIKE 21 BW model and express the direction the waves are going towards. This is shown to the left in Figure 28.10.

The Meteorological convention relates the wave direction to North and express the wave direction as 'coming from' in a clockwise direction relative to North. Waves travelling from North to South have therefore a direction of 0 degrees. This is shown to the right in Figure 28.10.
Note that when choosing the meteorological convention you have to define the grid orientation of the MIKE 21 BW model as well.

Conversion between conventions

The scientific / BW wave direction, $\theta_{main,sci}$, can be expressed as

$$\theta_{main,sci} = -\theta_{main,met} + \alpha_{ori} + 270 \tag{28.1}$$

where $\theta_{main,met}$ is the meteorological wave direction and α_{ori} is the North to Y-axis orientation of the grid bathymetry, measured clockwise (NYC).

In the example illustrated in Figure 28.7 the scientific wave direction is found to be 235 degrees to the x-axis, based on a meteorological wave direction of 350 degrees N and a NYC rotation of 315 degree N.
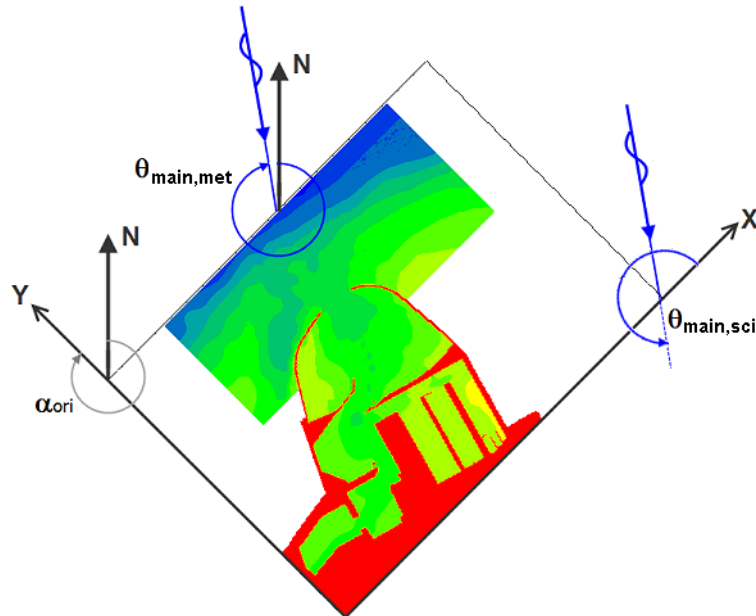

Figure 28.11   Example of main wave direction for a given grid orientation and meteorological wave direction.

28.5.2   Directional distribution settings

You may specify this distribution in one of the following ways.

Frequency independent distributions

- A $\mathbf{cos^n}$ $(\theta - \theta_{main})$ where θ_{main} is the Main wave direction and n is the directional spreading index. This distribution is specified through:
    - Main wave direction, θ_{main}
    - Maximum deviation from main wave direction, $\Delta\theta_{max}$
    - Spreading index, n

- **Gaussian distribution**. This distribution is specified through:
    - Main wave direction, θ_{main}
    - Maximum deviation from main direction, $\Delta\theta_{max}$
    - Directional variance, σ^2

- **Uniform distribution**. This distribution is specified through:

  - Main wave direction, θ_{main}
  - Maximum deviation from main direction $\Delta\theta_{max}$
    In this case, in principle, the waves do not have a main wave direction. However, θ_{main} has to be given as the maximum deviation specified relative to it.

Frequency dependent distributions

- A **cos^{2s} [½(θ-θ_{main})]** where s is a frequency dependent shape parameter. This distribution is specified through:

  - Main wave direction, θ_{main}
  - Maximum deviation from main direction, $\Delta\theta_{max}$
  - Frequency shape parameter, s

- User defined directional spectrum
  In this case you have to defined the Grid Orientation.

Note: All directional distributions are re-scaled so that the area under the truncated distribution function is (or becomes) one.

28.6 Reference Directional Spectrum Specification

If directional waves was chosen as type of wave generation, it is possible to output a grid file (dfs2) with desired target directional spectrum and the resulting directional spectrum. The target and result will be different since the wave generation is stochastic based with a single direction -- randomly picked from a directional distribution -- per discrete frequency. The number of discrete frequencies (and thereby the number of directions) may be increased by increasing the length of the generated time series which in turn reduces the directional variability. Examining the reference directional spectrum can be used to assess the obtained directional variability and in the case of specified directional distribution also to check that orientation of the grid and the main direction has been chosen correctly.

28.7 Specification of Reference Points

In order to check the wave field in front of the generation line, you may define one or more reference points (see Figure 28.7), in which the surface elevation is calculated (assuming constant depth).

After giving the coordinates, you are requested to specify a name of the time series data file (data type .dfs0 or .dt0).

28.8 Specification of Output Data

The type of output depends on the selected wave type:

- One-dimensional waves
- Unidirectional waves
- Directional waves

28.8.1 One-dimensional waves

The output file is a .dfs0 (or .dt0) type data file (time series) containing the four items:

- Surface elevation [m]
- Slope of surface elevation [-]
- Curvature of surface elevation [m^{-1}]
- Flux density [m^3/m/s]

The output data file can be used for open boundary conditions and internal wave generation in MIKE21 BW.

See output example in Figure 28.3.

28.8.2 Unidirectional waves

The output file is a .dfs1 (or .dt1) type data file (line series) containing the two items:

- Flux density [m^3/m/s]
- Slope of surface elevation [-]

The output data file can be used for internal wave generation in MIKE21 BW.

See output example in Figure 28.4.

28.8.3 Directional waves

The output file is a .dfs1 (or .dt1) type data file (line series) containing the two items:

- Flux density [m^3/m/s]
- Slope of surface elevation [-]

The output data file can be used for internal wave generation in MIKE21 BW.

See output example in Figure 28.5.

28.9 Overview

The overview page presents you with a summary of all the input parameters you have specified. You should inspect the parameters to check they are correct, - and if not go back and change them.

From the page you can either *Finish* the set-up, which will transfer your set-up definition to the current toolbox file, or execute the set-up.

28.10 Scientific Background

The scientific background of this tool can be found in a number of textbooks and scientific papers, pls see References *(p. 202)*.

28.10.1 Definition of Pierson-Moskowitz spectrum

The Pierson-Moskowitz spectrum is defined as:

$$S(f) = A \cdot f^{-5} \exp(-B \cdot f^{-4}) \tag{28.2}$$

where S(f) is the spectral density and f the frequency.

The parameters A and B have the following values:

Known wind speed 19.5 m above the sea surface, $U_{19.5}$:

$$A = \frac{\alpha g^2}{(2\pi)^4} \tag{28.3}$$

$$B = 0{,}74 \left(\frac{g}{2\pi U_{19,5}} \right)^4 \tag{28.4}$$

where $\alpha = 0.0081$ is Phillips' "constant", see e.g. Komen et al (1994)/23/ p. 103, and g= 9.81 m/s^2 is the gravitational acceleration.

Known significant wave height, H_{m0}:

$$A = \frac{\alpha g^2}{(2\pi)^4} \tag{28.5}$$

$$B = \frac{4A}{H_{m0}^2} \tag{28.6}$$

Known significant wave height, H_{m0}, and mean wave period, T_{02}:

$$A = \frac{1}{4\pi} \frac{H_{m0}^2}{T_{02}^4}$$ (28.7)

$$B = \frac{1}{\pi T_{02}^4}$$ (28.8)

28.10.2 Definition of JONSWAP spectrum

The JONSWAP spectrum is defined as:

$$S(\mathrm{f}) = S_{PM}(\mathrm{f}) \cdot \gamma^a$$ (28.9)

where $S_{PM}(f)$ = Pierson-Moskowitz spectrum (see equation (28.2)).

The shape parameters are defined as

$$a = \exp\left[-\frac{1}{2}\left(\frac{\mathrm{f} - \mathrm{f}_p}{\sigma \mathrm{f}_p}\right)^2\right]$$ (28.10)

$$\sigma = \begin{cases} \sigma_a & \text{for } \mathrm{f} \leq \mathrm{f}_p \\ \sigma_b & \text{for } \mathrm{f} > \mathrm{f}_p \end{cases}$$ (28.11)

where f_p denotes the peak frequency.

γ = user-defined shape parameter. A value of 3.3 corresponds to a standard JONSWAP spectrum.

Values corresponding to a standard JONSWAP spectrum are σ_a = 0.07 and σ_b = 0.09.

Known wind speed 10 m above the sea surface, U_{10}, and effective fetch length, X:

$$A = \frac{\alpha g^2}{(2\pi)^4}$$ (28.12)

where α is Phillips' "constant" given by

$$\alpha = 0{,}076 \tilde{X}^{-0{,}22}$$ (28.13)

see e.g. Komen et al (1994)/23/ p. 187.

The dimensionless fetch is defined by

$$\tilde{X} = \frac{gX}{U_{10}^2} \tag{28.14}$$

where X and U_{10} is the fetch and wind speed (10 m above sea level), respectively.

$$B = \frac{5}{4}\frac{1}{T_p^4} \tag{28.15}$$

where

$$f_p = 3{,}5\frac{g}{u_{10}}X^{-0{,}33} \tag{28.16}$$

$$f_p = \frac{1}{T_p} \tag{28.17}$$

In this case the peak enhancement factor $\gamma = 3.3$ and the shape parameters $\sigma_a = 0.07$, $\sigma_b = 0.09$.

Known significant wave height, H_{m0}, peak wave period, T_p, and the shape parameters σ_a, σ_b, and γ

$$A = \frac{5}{16}\frac{H_{m0}^2}{T_p^4} \tag{28.18}$$

$$B = \frac{5}{4}\frac{1}{T_p^4} \tag{28.19}$$

Please note that the proportional value (5/16 and 5/4) used in the expression for A and B is only valid for small deviations of the shape parameters σ_a, σ_b, and γ from the standard values (0.07, 0.09 and 3.3).

28.10.3 Definition of the TMA spectrum

The TMA spectrum is defined as:

$$S(\mathrm{f}) = S_{JONSWAP}(\mathrm{f}) \cdot \phi(\omega_h) \tag{28.20}$$

where $S_{JONSWAP}(f)$ = JONSWAP spectrum as defined in equation (28.9).

$$\phi(\omega_h) = \begin{cases} \dfrac{1}{2}\omega_h^2 & \text{for } \omega_h \leq 1 \\[2mm] 1 - \dfrac{1}{2}(2-\omega_h) & \text{for } 1 < \omega_h \leq 2 \\[2mm] 1 & \text{for } \omega_h > 2 \end{cases} \qquad (28.21)$$

$$\omega_h = 2\pi f \sqrt{h/g} \qquad (28.22)$$

and h = water depth.

The TMA spectrum was originally suggested by Bouws et al (1983)/19/.

28.10.4 Definition of directional distributions

The equations for the directional distributions are given below:

1. A $\cos^n(\theta - \theta_{main})$ distribution is defined as

$$D(\theta) = \cos^n(\theta - \theta_{main}) \qquad (28.23)$$

where θ_{main} denotes the main wave direction and n is a user-specified spreading index.

2. The Gaussian (normal) distribution is defined as:

$$D(\theta) = \frac{1}{\sigma\sqrt{2\pi}}\exp\left[-\frac{(\theta-\theta_0)^2}{2\sigma^2}\right] \qquad (28.24)$$

where θ_{main} denotes the main wave direction and σ is user-specified spreading index.

3. The uniform distribution is defined as:

$$D(\theta) = \frac{1}{2\Delta\theta_{max}} \qquad (28.25)$$

where $\Delta\theta_{max}$ is the maximum deviation from the main wave direction.

4. The DHI distribution is defined as:

$$D(\theta) = \cos^{2s}\left(\frac{1}{2}(\theta - \theta_0)\right) \tag{28.26}$$

where

$$s = \frac{2}{\sigma} - \frac{1}{2} \tag{28.27}$$

$$\sigma = 0{,}24733\left(\frac{f}{f_{1,-1}}\right)^{2,5} + 0{,}07153\left(\frac{f_{1,-1}}{f}\right)^{2,5} - 0{,}1377 \tag{28.28}$$

Here $f_{1,-1} = (m_1/m_{-1})^{0.5}$, where m_i is the i'th moment of the frequency spectrum. $\Delta\theta_{max}$ is the maximum deviation from the main wave direction.

28.10.5 Second order wave generation

Second order wavemaker theory based on second order Stokes theory obtained by classical perturbation method was first developed for general irregular seas by Dean and Sharma in /27/. See also the corresponding theory for Boussinesq theory in the paper by Madsen and Sørensen /28/.

28.11 Remarks and Hints

CPU and RAM The computational time and internal memory requirement is usually small for generation of signals for one-dimensional waves (time series, .dfs0/.dt0 data files). For unidirectional and directional waves (line series, .dfs1/.dt1 data files) it may take longer time. If you have a generation line with a very large number of grid points (e.g. 1000) and the length of generated line series should be of e.g. 1 hour, then it might take a while to generate that series. Of course it depends of your hardware.

- **Internal time step** The time step used internally in this tool (for unidirectional and directional waves) is half of what you specify as input.

- **Log file** For the log output ASCII file, which has the same name as the input file, but with ".log" as suffix. This file contains a list of all the specifications used for the execution, information about the input data and the generated data. The log file also includes the spectral density and the corresponding frequency of the selected spectrum.

- **Number of time steps** The number of time steps specified should at least be 2 time steps more than used in your Boussinesq wave simulation.

- **Replacement of RANDWA and RANDIR** This tool replace the tools for random wave generation used in previous versions of MIKE21 PP, i.e. RANDWA (for generation of unidirectional normal incident waves) and RANDIR (for generation of directional waves).

- **Sponge layers** An internal generation line should always be located in front of a sponge layer. There should be one layer (with sponge coefficient 1) between the first sponge layer and the generation line. This is necessary, as the internal generation of waves in MIKE21 BW is performed by adding the flux of the incident wave field along the generation line, or rather more correctly by implicitly introducing two internal flux boundaries back to back.Thus, it is important to note that the waves will propagate symmetrically with respect to the generation line and that waves passing through the generation line will not be distorted.

- **Statistical variability** The spectral/statistical parameters of the generated time or line series might differ slightly from the ones you have specified in the input. This is most often the case if the length of your generated time series is too short.

- **Water depth** If the water depth varies significantly along the generation line and it is difficult to find a representative average depth, then you may have to re-consider your MIKE21 BW model set-up. Alternatively you can split the generation line into a number of sections and generate the signal separately for each of the sections. Make sure that you join the minor generation lines/sections correctly.

- **Wave propagation direction** In order to make sure that you have generated the correct signal (unidirectional or directional waves) for application in MIKE21 BW, it is always a good idea to test this by simulating the wave field in a smaller model area. After the simulation is finished you can plot the surface elevation pattern (e.g. wave crests) and easily see whether your generated data looks correct and as expected.

- **Which type of frequency spectrum?** If you do not have any information on the type of the frequency spectrum, - which one should you then select? Most often you you will select a JONSWAP or Pierson-Moskowitz type spectrum. For typical studies of short periodic wave disturbance in ports and harbours, the type of the frequency spectrum is usually not that important. The directionality of the wave field is generally of more importance.

- **Which type of directional distribution?** If you do not have any specific information of the directionality of the waves, - which one should you then select? Most often you you will select a $\cos^n$-type distribution. The spreading index n may be taken as 2 or 4 for wind waves and 10 or larger for swell. It is always a good idea to simulate the wave field for unidirectional as well directional incident waves. This is particularly relevant in case you do not have any detailed information about the directionality of the waves.

A relation between the directional spreading index, n, and the directional standard deviation, DSD, is shown below in Table 28.1 below.

Table 28.1   The relationship between the directional spreading index, n, and the directional standard deviation, DSD.

Directional spreading index	Directional standard deviation (deg)
1	39.15
2	32.52
3	28.36
4	25.45
5	23.28
6	21.58
7	20.20
8	19.05
9	18.08
10	17.24
11	16.52
12	15.87
13	15.30
14	14.78
15	14.31
16	13.88
17	13.49
18	13.13
19	12.80
20	12.49
40	8.94
60	7.33
80	6.36
100	5.70

28.12 References

The references listed below provide you with more basic information applicable for this tool :

About wave spectra

/18/   Battjes, J. A., 1978: Probabilistic aspects of ocean waves. Report No 77-2, 2nd printing, Lab. of Fluid Mech., Dept. of Civil Eng., Delft Univ. of Tech. 52pp.

/19/   Bouws, E., H. Gunther, W. Rosenthal and C.L. Vincent, 1985: Simi-

larity of wind wave spectrum in finite depth of water, 1. Spectral Form. *Journal of Geophysical Research*, **90**, C1.975-986.

/20/   Hasselmann, K. et al., 1973: Measurements of wind-wave growth and swell decay during the Joint North Sea Wave Project (JONSWAP). *Deutsche Hydrographische* Zeitschrift, Reihe A (8), **12**.

/21/   Huang, N. E., C.-C. Tung and S. R. Long, 1990: Wave spectra, Chapter 6 of "The Sea", **9**, Part A: Ocean Engineering Science (eds. B. Le MÈhautÈ and D. N. Hanes), John Wiley and Sons, Inc., New York.197-237.

/22/   Isherwood, R. M., 1987: Technical Note: A revised parameterisation of the JONSWAP spectrum. *Applied Ocean Research*, **9**, 47-50.

/23/   Komen, G.J., L. Cavaleri, M. Donelan, K. Hasselmann, S. Hasselmann and P.A.E.M. Janssen, 1994: Dynamics and modelling of ocean waves. Cambridge University Press. 532 pp.

/24/   Sand, S.E., 1979: Three-dimensional Deterministic structure of ocean waves. Institute of Hydrodynamics and Hydraulic Engr., Tech. Univ. of Denmark, Series Paper No 24.

About wave generation and synthesising

/25/   Bendat, J.L. and A.G.Piersol, 1986: Random Data. Analysis and Measurements Procedures. John Wiley & Sons.

/26/   Sand, S.E., 1987: Directional wave generation and analysis. AIRH-Seminar-IAHR Wave analysis and generation in laboratory basins, Lausanne. 203-235.

About second order wave generation

/27/   Sharma, J. N. and Dean, R.G., 1981: Second-Order Directional Seas and Associated Wave Forces, SPE Journal, Vol. 21, pp. 129-140.

/28/   Madsen, P. A. and Sørensen, O. R, 1993: Bound waves and triad interactions in shallow water, Ocean Engineering, Vol. 20, No 4, pp 359-388.

/29/   H. Schäffer, H. A., 1995: Second-order wavemaker theory for irregular waves, Ocean Engineering, vol. 23, no. 1, pp. 47-88.

/30/   Schäffer, H. A. and Steenberg, C.M., 2003: Second-order wavemaker theory for multidirectional waves, Ocean Engineering, vol. 30, pp. 1203-1231.

29 Generation of Regular Waves

Based on a specified wave theory this tool generates a time series of surface elevation, flux density and derived quantities. The generated time series can be used to fill in model open boundary conditions or as input at an internal generation line in the Boussinesq Wave Module, MIKE 21 BW, for simulation of regular long crested waves. An example is shown in Figure 1.
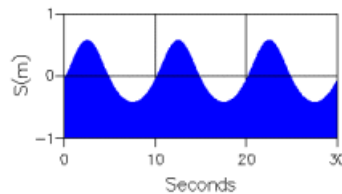

Figure 29.1   Generated time series of surface elevation. Conditions: 3. order Fourier series to the Boussinesq equations, wave height = 1m, wave period= 10 s and water depth= 8 m.

29.1 Specify Wave Theory and Parameters

First you select a wave theory from among the three different theories available:

- Stokes wave theory
- Boussinesq wave theory
- Cnoidal wave theory

For Stokes and Boussinesq wave theories you should also specify the order of the theory.

If you select Fourier series for the Boussinesq theory the so-called dispersion parameter, B, must also be given. B= 0 for the classical equations and B= 1/15 is recommended for the enhanced equations (incl. deep-water terms).

29.2 Specification of the Historical Date

Here you specify the start time (historical date of the first time step), the total number of time steps and finally you specify the time step.

It is recommended that the time step is the same as in your Boussinesq wave simulation. Otherwise a linear interpolation is used.

29.3 Specification of the Output Data

In case you have used Stokes or Boussinesq wave theories the output data file always contains the following four items:

- Surface elevation [m]
- Slope of surface elevation [-]
- Curvature of surface elevation [m^{-1}]
- Flux density [m^3/m/s]

The second and third items are required in your Boussinesq wave simulation when the deep-water terms are included.

29.4 Overview

The overview page presents you with a summary of all the input parameters you have specified. You should inspect the parameters to check they are correct, - and if not go back and change them.

From the page you can either *Finish* the set-up, which will transfer your set-up definition to the current toolbox file, or execute the set-up.

29.5 Scientific Background

You can find the theoretical background for the different types of waves in numerous textbooks and papers on wave theories. A number of suggestions for further reading are listed below:

/31/   Mei, C.C.,1983: The applied dynamics of ocean surface waves. John Wiley & Sons, New York.

/32/   Svendsen, I.A. and I.G. Jonsson, 1980: Hydrodynamics of coastal regions. Technical University of Denmark.

/33/   Dean, R.G. and R.A. Dalrymple, 1991: Water wave mechanics for engineers and scientists. Advanced Series on Ocean Engineering - Volume 2, World Scientific, Singapore.

/34/   Fenton, J.D., 1985: A fifth-order Stokes theory for steady waves. J. Waterway, Port, Coast and Oc. Eng., ASCE, **111**,2. 216-234.

/35/   Madsen, P.A. and O.R.S̄rensen, 1993: Bound waves and triad wave interactions in shallow water. Ocean Engineering, **2**,4. 359-

388.

29.6   Remarks and Hints

Which wave theory?

As input to Boussinesq wave simulations you typically need to use 1. order Stokes or 1. order Boussinesq wave theory. If the nonlinearity is high the 3. order Boussinesq theory may be used.

Consistency?

Make sure that the specified parameters (dispersion parameter and water depth) correspond to your Boussinesq wave model setup. If the water depth varies along the internal generation line (or at the open boundary) you should apply a representative average water depth.

If you select Fourier series to the Boussinesq theory the so-called dispersion parameter must also be given. This parameter must be identical to the value used in your Boussinesq wave simulation, when the deep-water terms (enhanced Boussinesq equations) are included.

30 Wave Generated Current and Setup

With the MIKE 21 Tool **Wave Generated Current and Setup** you can calculate the wave generated current and wave setup along the open boundaries of the model area. The results can be used as boundary data to a MIKE 21 Flow Model simulation including wave radiation stresses.

The tool can also be used to combine the boundary data for wave generated current (calculated by the tool) with the boundary data extracted from a Flow Model simulation including other forces (e.g. tides, storm surges). The method of simple linear superposition is used to combine the water levels and fluxes from both sources.

Only one boundary can be treated per run.

30.1 Select Wave Data File

A type 2 data file containing the three components of radiation stress. This file can be generated directly by one of the MIKE 21 wave modules: MIKE 21 PMS (Parabolic Mild Slope Waves) or MIKE 21 EMS (Elliptic Mild Slope Waves).

30.2 Surface Elevation

The surface elevations (or water levels) can be specified as constant, as time varying (a type 0 file) or as spatial varying (a type 2 file). The surface elevation must be exactly the same as used in the wave simulation. If you are specifying a type 0 or type 2 file, remember also to specify the number of the surface elevation item in the file (use the browse facility to choose the file).

The surface elevation is measured positive upwards from the datum used.

30.3 Boundary Line

Here you specify the open boundary where you want to calculate the wave generated current and setup. The position of the open boundary line is specified by two sets of grid point coordinates, the first and the last *water* point along the boundary.

If you want the current direction to be different from 90º to the boundary line, an alternative angle a (typically parallel to the coast) can be specified, see Figure 30.1.
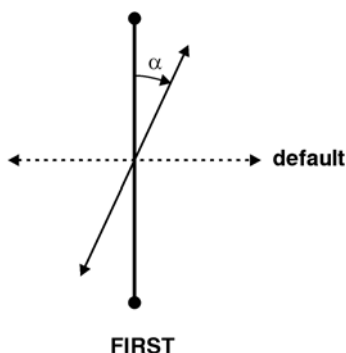
Figure 30.1   Definition of current direction, ($0 \leq a < 180$).

The bathymetry used in the calculations is the same as the one used in the MIKE 21 wave model. In addition, if a file with time varying or spatial varying surface elevations (water levels) has been used in the wave simulation, this file **must** also be specified for this simulation. This is done on the surface elevation dialog.

Finally, you should specify the value of true land points in the bathymetry (normally equal to +10.0).

30.4 Bed Resistance

The bed resistance can be specified as constant or variable along the boundary. The value can be specified as a Manning number or as a Chezy number. For variable bed resistance along the boundary, a type 2 data file is specified, from which the boundary resistance values are extracted.

The bed resistance file must be the exactly same file as will be used in the subsequent Flow Model simulations. Remember to specify the number of the bed resistance item in the file (use the browse facility to choose the file).

30.5 Eddy Viscosity

The eddy viscosity, if included at all in the simulations, can be specified as constant or variable along the boundary. It can also be specified using the Smagorinsky formulation for the eddy viscosity.

The eddy viscosity must be the exactly same as will be used in the subsequent Flow Model simulations. In the case of a variable eddy viscosity along the boundary (using a type 2 file), remember to specify the number of the eddy viscosity item in the file (use the browse facility to choose the file).

30.6   Miscellaneous Parameters

If the output boundary data should contain level and/or flux distributions from other sources (e.g. tide), a type 1 file containing these data on the boundary must be specified. This type 1 file is generated from an encompassing Flow Model using the MIKE 21 Tool Transfer Boundary and will be superposed on the wave generated current and setup.

The simulation period, specified by the Time Series properties, for the subsequent Flow Model simulation must be covered by boundary data file. The time interval is the interval in seconds between time steps.

If the radiation stresses should be gradually increased from zero to the value calculated by the wave model over a period of time, a soft start can be specified. Since the wave generated currents (or wave setup) can be relatively strong, the soft start facility is recommended to avoid generation of shock waves propagating from the boundary into the model area.

30.7   Output Specification

The output is given as a type 1 data file which can be used as transfer boundary data for a Flow Model simulation.

You can specify if the output should be levels, levels and fluxes, or levels and velocities along the boundary. The estimated wave setup (levels) will be given relative to the bathymetry datum. The possibility of selecting velocities as output is only for checking purposes. Before using the boundary data in an Flow Model simulation, the tool should be re-run with specification of fluxes as output.

30.8   Overview

The overview page presents you with a summary of all the input parameters you have specified. You should inspect the parameters to check they are correct, - and if not go back and change them.

From the page you can either *Finish* the set-up, which will transfer your set-up definition to the current toolbox file, or execute the set-up.

30.9   Example

An example of calculated wave induced setup/setdown and longshore currents is illustrated in the figure below in Figure 30.2, along with the corresponding bed profile in Figure 30.3.
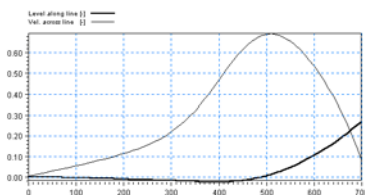
Figure 30.2   Wave setup/setdown and longshore currents
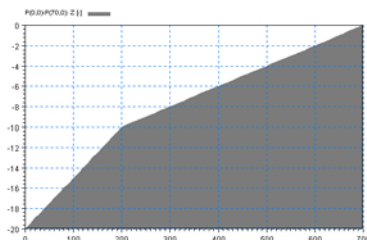
Figure 30.3   Bottom profile

30.10 Scientific Background

30.10.1 Main equations

The 2D equations of conservation of mass and momentum can be written as:

Continuity Eq.:

$$\frac{\partial s}{\partial t} + \frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} = 0$$

x-momentum:

$$\frac{u}{t} + u\frac{\partial u}{\partial x} + v\frac{\partial u}{\partial y} = -g\frac{\partial s}{\partial x} - \frac{\tau_{bx}}{\rho h} - \frac{1}{\rho h}\left[\frac{\partial S_{xx}}{\partial x} + \frac{\partial S_{xy}}{\partial y}\right] + \frac{1}{h}\left[\frac{\partial}{\partial x}\left(Eh\frac{\partial u}{\partial x}\right) + \frac{\partial}{\partial y}\left(Eh\frac{\partial u}{\partial y}\right)\right]$$

y-momentum:

$$\frac{v}{t} + u\frac{\partial v}{\partial x} + v\frac{\partial v}{\partial y} = -g\frac{\partial s}{\partial y} - \frac{\tau_{by}}{\rho h} - \frac{1}{\rho h}\left[\frac{\partial S_{xy}}{\partial x} + \frac{\partial S_{yy}}{\partial y}\right] + \frac{1}{h}\left[\frac{\partial}{\partial x}\left(Eh\frac{\partial v}{\partial x}\right) + \frac{\partial}{\partial y}\left(Eh\frac{\partial v}{\partial y}\right)\right]$$

The following assumptions are used to simplify the above equations:

Assume uniform flow conditions in the longshore direction (y-axis). This implies

$$\frac{\partial v}{\partial y} = \frac{\partial s}{\partial y} = 0$$

Assume steady state conditions. This implies

$$\frac{\partial s}{\partial t} = \frac{\partial u}{\partial t} = \frac{\partial v}{\partial t} = 0$$

Assume longshore uniformity in radiation stresses. This implies

$$\frac{\partial S_{xy}}{\partial y} = \frac{\partial S_{yy}}{\partial y} = 0$$

Using these assumptions, we obtain the following simplified equations

Continuity Eq.:

$$\frac{\partial u}{\partial x} = 0 \Rightarrow u = 0$$

since u = 0 on shoreline

x-momentum:

$$g\frac{\partial s}{\partial x} = -\frac{1}{\rho h}\left[\frac{\partial S_{xx}}{\partial x}\right]$$

y-momentum:

$$\frac{\tau_{by}}{\rho} = -\frac{1}{\rho}\left[\frac{\partial S_{xy}}{\partial x}\right] + \left[\frac{\partial}{\partial x}\left(Eh\frac{\partial v}{\partial x}\right)\right]$$

where:

$$\tau_{by} = \frac{\rho g v |v|}{M^2 h^{1/3}}$$

These x- and y-momentum equations are the simplified equations solved in the tool for estimating setup/setdown and wave-induced longshore currents.

The x-momentum equation is a balance between changes in water level variation and the radiation "force", while the y-momentum equation expresses the balance between the bottom friction, wave radiation "force" and the eddy viscosity term.

Note that in cases where $\alpha \neq 90^o$, the simplifications 1,2,3 above are applied assuming the longshore direction is along the line specified by α. Thus, the calculated radiation stresses are rotated from the x-y (grid on which the radiation stress is computed) to x'y' grid (grid defined such that one of the axes is parallel to longshore direction, and the other perpendicular to it). The computations are internally made on this grid, but the results are transformed back to the x-y grid before being output to files.

30.10.2 Numerical solution

The x-momentum equation is solved for the setup/setdown using central differences for the right hand side, and forward differences for the left hand side. The solution is initiated at the offshore boundary with an s = 0 boundary condition.

The y-momentum equation is solved for the longshore currents v. The radiation stress term is discretised using central differences. The eddy viscosity

term is discretised as follows:

$$\frac{\partial}{\partial x}(Eh) = \frac{1}{\Delta x}\left[\frac{v_{i+1} - v_i}{\Delta x}\left(\frac{E_{i+1}h_{i+1} + E_i h_i}{2}\right) - \frac{v_i - v_{i-1}}{\Delta x}\left(\frac{E_i h_i + E_{i-1}h_{i-1}}{2}\right)\right]$$

The friction term (nonlinear in v) is linearised using:

$$v|v| = |v_{old}|v$$

where v_{old} is a previous guess of the longshore current.

After discretising the terms in the y-momentum equation and linearising the friction term, a system of linear equations (tridiagonal) is obtained. The system of equations is solved using the double sweep method. The boundary conditions are:

$$v(-1) = v(n+1) = 0$$

The following algorithm is used in the solution of the y-momentum equation:

1. Guess v_{old} using E = 0

2. Solve the y-momentum equation for new values of v

3. Check vector norm $\sum |v_{i,new} - v_{i,old}| < \varepsilon$

4. If tolerance is not satisfied, continue iteration, otherwise, exit.

5. Update old guess: $v_{i,old} = (1 - coeff) \cdot v_{i,new} + coeff \cdot v_{i,old}$

6. Repeat the above calculations from step (2).

The following iteration parameters are used:

- Tolerance value = ε = 10^{-4}

- Relaxation parameter = coeff = 0.25

- Max. number of iterations = niter = 300

31 Wave Spectra Converter

The MIKE 21 Wave Spectra Converter tool allows conversion of 2D directional wave spectra data into MIKE Powered by DHI Dfs format. The resulting files can be used for example as boundary conditions in MIKE 21 Spectral Waves FM model.
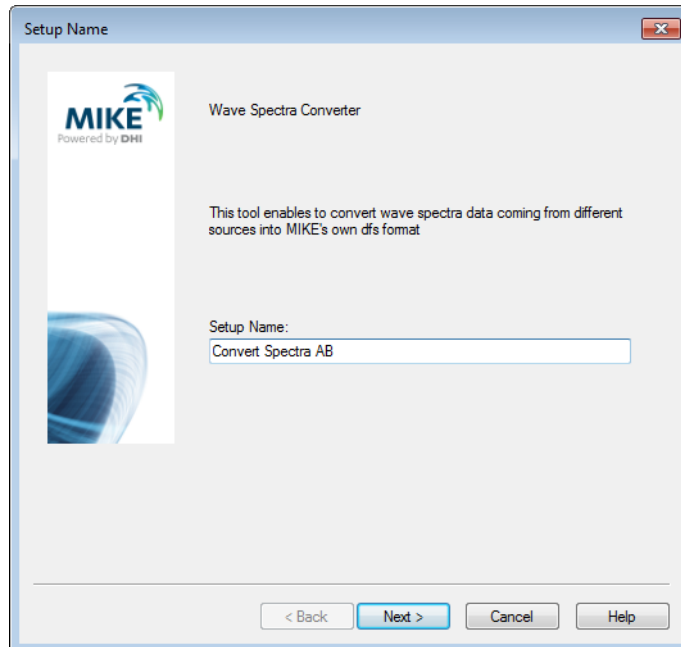
Figure 31.1   Setup of Wave Spectra Converter

The use of the Wave Spectra Converter tool can be decomposed in four basic steps:

- Select 2D wave spectra input format
- Select files to convert
- Set outputs
- Execute the Wave Spectra Converter process

Formats

Three formats of 2D wave spectra data can be used as input data by the Wave Spectra Converter tool:

- DHI ASCII
- DHI Dfs2/Dfsu
- Oceanweather

Input

Multiple 2D wave spectra input data files can be converted in one single instance of the Wave Spectra Converter tool but these files have to share the same input format and the same discretisation (temporal, frequency and direction). By converting several files simultaneously it is possible to get output data formatted as Line Series in the DHI Dfs format. These specific files can then be used as "varying in time and along line" boundary conditions in MIKE 21 Spectral Waves FM model allowing more realistic boundary conditions in the SW model.

Output

The Wave Spectra Converter gives access to the following information as output data in the DHI Dfs format:

- Wave parameters (like significant wave height, peak wave period…)
- 1D wave integrated spectra (direction spectrum and frequency spectrum)
- 2D wave spectrum

31.1 Wave Spectra Format

The first task to do is to select the format of the wave spectra data.
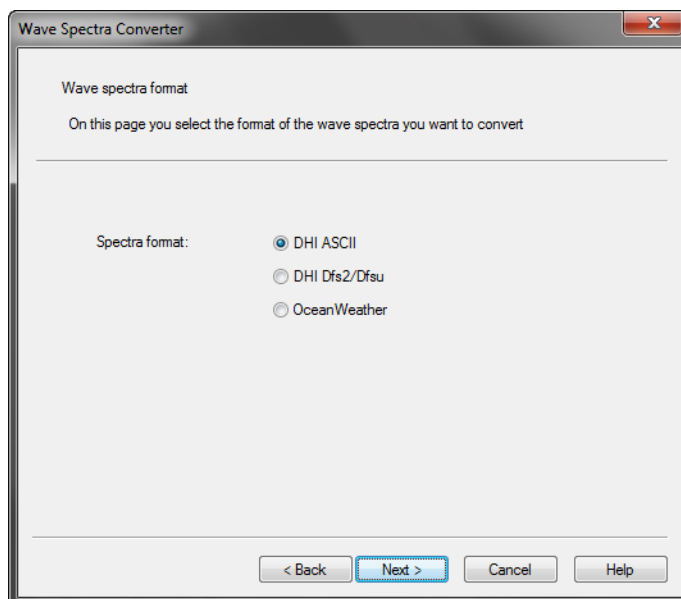
Figure 31.2   Define Wave spectra format

Three main formats of 2D wave spectra data can be used independently as input data by the Wave Spectra Converter tool:

- DHI ASCII
- DHI Dfs2/Dfsu
- Oceanweather

31.1.1 DHI ASCII

2D wave spectra data can be presented in various formats depending on the provider. In order to facilitate the conversion process DHI has developed a simple ASCII format which can be used as an entry point to the Wave Spectra Converter tool. Other 2D wave spectra data can be formatted by the user to the DHI ASCII format for later use as input data in the Wave Spectra Converter.

Examples of 2D wave spectra data in DHI ASCII format are presented below and with the Wave Spectra Converter example in the default installation folder:
\MIKE_21\Toolbox\Waves\WaveSpectraConverter\DHI_Ascii

Figure 31.3   Example of spectral data in DHI format.
(Values in this example is specified as the variance component.)

Format description

An example of spectral data in DHI format is shown in Figure 31.3.

Header information
The first part of the ASCII File must contain the header information.

The first line of the header information is a character string containing the title. The second line contains the data and time followed by the number of time steps and the time step interval. The date and time format follows the

ISO standard 8601, which is YYYY-MM-DD HH:MM:SS. The third line contains the number of points. Note, that ONLY one point is presently allowed.

The next two lines contain the x- and y-coordinates for the points. The coordinate must be given in spherical co-ordinates (lat/long).

The next line contains the type of frequencies discretisation and the number of discrete frequencies. Two types of frequency discretisation can be applied:

1. Equidistant distribution, where the minimum frequency and the frequency interval are specified.

2. Logarithmic distribution, where minimum frequency and the frequency factor are specified.

The next line contains the type for directional discretisation and the number of discrete directions. Two types of directional discretisation can be applied:

1. 360 degree rose, where the minimum direction is specified.

2. Directional sector, where minimum direction and the directional interval are specified.

Data

Data for the variance component or the discrete energy density follows after the header information.

The data is arranged in blocks for each time step. The first line in the block is a character string containing the time information for the actual time step (presently not used).The spectra data for each point and each time step consist of a number of lines corresponding to the number of discrete directions and each line consist of a number of values corresponding to the number of discrete frequencies

Which type of spectra data that is used is not saved in the ASCII file but must be specified by an additional input parameter. The relation between the discrete energy density, E_{ij} (specified in unit m²s/rad) and the variance component, V_{ij} (specified in unit m²) is as follows

$$E_{ij} = \frac{V_{ij}}{\Delta f \cdot \Delta \theta}$$
(31.1)

where Δf is the constant frequency interval and $\Delta \theta$ is the constant directional interval in radians.

Read statements

The ASCII data file are read using the fortran statements shown in Figure 31.4. The variables therein are described in Figure 31.5.

```
!- Local variables -----------------------------------------------
character(255) :: title
integer        :: np, ns, i, j
integer        :: YYYY, MM, DD, HH, MI, SS
integer        :: nsteps
real(8)        :: dt
integer        :: npoints
real(8)        :: x(:), y(:)
integer        :: frequency_type, nfrequencies
real(8)        :: f_min, f_factor, df_constant
integer        :: direction_type, ndirections
real(8)        :: theta_min, dtheta_constant
character(255) :: text
character(40)  :: string
character(255) :: time

!----------------------------------------------------------------
read (lun,100) title
read (lun,200) YYYY, MM, DD, HH, MI, SS, nsteps, dt
read (lun,300) npoints
read (lun,310) (x(np), np=1,npoints)
read (lun,310) (y(np), np=1,npoints)
read (lun,400) frequency_type, nfrequencies
if (frequency_type == 1) then
  read (lun,410) f_min, df_constant
end if
if (frequency_type == 2) then
  read (lun,410) f_min, f_factor
end if
read (lun,500) direction_type, ndirections
if (direction_type == 1) then
  read (lun,510) theta_min
end if
if (direction_type == 2) then
  read (lun,510) theta_min, dtheta_constant
end if
do ns = 1, nsteps
  do np = 1, npoints
      ........
      read (lun,*) string
      read (lun,*) ((e(i,j), i=1,nfrequencies), j=1,ndirections)
      ........
  end do
end do
100 format (a)
200 format(12x, i4, 1x, i2, 1x, i2, 1x, i2, 1x, i2, 1x, i2, i10, e14.6)
300 format (12x, i10)
310 format (5e14.6)
400 format (12x, i10, i10)
410 format (5e14.6)
500 format (12x, i10, i10)
510 format (5e14.6)
500 format (12x, i10, i10)
510 format (5e14.6)
```

Figure 31.4   Fortran read routine for DHI ASCII file

Variable definitions:

title	Character string containing the title for the file
YYYY	Year
MM	Month
DD	Day
HH	Hours
MI	Minutes
SS	Seconds
nsteps	Number of time steps
dt	Time step interval (in seconds)
npoints	Number of points
x(*)	x-coordinates for the points (longitude in degrees)
y(*)	y-coordinates for the points (latitude in degrees)
frequency_type	Type of frequency discretization
nfrequencies	Number of frequencies
f_min	Minimum frequency (in Hz)
df_constant	Frequency interval (in Hz)
f_factor	Frequency factor
directional_type	Type of directional discretization
ndirections	Number of directions
theta_min	Minimum direction (in degree)
dtheta_constant	Directional interval (in degree)
time	Character string (not used)
e(*,*)	Variance component or spectral density

Figure 31.5   Description of variables used in code in Figure 31.4.

31.1.2 DHI Dfs2/Dfsu

2D wave energy spectra data in MIKE software is saved either as a .dfsu or a .dfs2 file (the latter format being used by the Polar Plot control of the Plot Composer).

31.1.3 Oceanweather

Oceanweather provides waves hindcast products around the world. Wave spectra are delivered as ASCII files in Oceanweather's standard format which has been implemented in the Wave Spectra Converter tool.

An example of wave spectra data in Oceanweather ASCII format are presented below.

Figure 31.6   Example of spectral data table in Oceanweather format

Read more about Oceanweather at their website:

http://www.oceanweather.com/metocean/

Format description

An example of a Table section containing spectral data in Oceanweather format is shown in Figure 31.6.

The top of the file must indicate the number of frequencies and directions per table as well as the number of time steps and the number of grid points per time step within the file (not shown in Figure 31.6). It is followed by one blank line and a number of table sections for each time step and grid point.

Table section
Every table is 32 lines long, with 5 header lines (the fourth one is blank), 24 data lines, and 3 footer lines (the last one is blank).

The first line of each table includes the latitude, longitude, angle, and water depth. The second line is a character string indicating which variables are located in the third line and include wind speed, wind direction (measured from which), peak period, sea and swell partitions as well as significant wave height and is followed by a blank line. The next line gives the nominal frequencies of each frequency bin.

Within the 24 data lines, directional bands are identified at the left. The 552 element array contains the variance components (NOT spectral densities) for 23 frequencies and 24 directions. The 24 directional bins, each 15 degrees wide, are numbered clockwise from north; the first bin, with a nominal direction 7.5 degrees, extends from 0 to 15 degrees. The sum over all the frequencies per one direction band is given in the right most column.

The first footer line contains the sum of the energy per frequency (over all directions). See Frequency bins *(p. 225)* for a more detailed description. The second footer line contains the "density" as explained in Density *(p. 225)*.

Read statements

The Oceanweather data file are read using the fortran statements shown in Figure 31.7. The variables therein are described in Figure 31.8.

```
REAL THETA(24), SPEC(23,24), FREQ(23), ANGSPEC(24)
REAL FSPEC(23), ETOT, DENS(23)
INTEGER CYM, DHM, GP
REAL LAT, LONG, ANGLE, DEPTH
REAL WD, WS, TP, VMD, ETOTSE, TPSE,
& VMDSE, ETOTSW, TPSW, VMDSW, MO1, MO2, HSIG, DOMDR,
& ANGSPR, INLINE, TAU
10 format (9x, f9.4, 11x, f10.4, 7x, f10.4, 7x, f10.4)
11 format (f7.2, 24f7.4)
12 format (7x, 23f7.4, f7.3)
13 format (7x, 23f7.2)
14 format (3i7, f6.1, f7.3, 3(2f7.3,f7.1), 3f7.3,
& f6.1, 2f7.4, f7.2)
15 format (7x, 23f7.4)
READ (10,*) !summary line at top of file
READ (10,*)!blank line at top of file
20 READ (10,10,END=30) LAT, LONG, ANGLE, DEPTH !Looped for each tau
READ (10,*)
READ (10,14)CYM,DHM,GP,WD,WS,ETOT,TP,VMD,ETOTSE,TPSE,
& VMDSE,ETOTSW,TPSW,VMDSW,MO1,MO2,HSIG,DOMDR,ANGSPR,
& INLINE,TAU
READ (10,*) !blank header line
READ (10,15) (FREQ(I), I=1,23)
READ (10,11) (THETA(J), (SPEC(I,J), I=1,23), ANGSPEC(J), J=1,24)
READ (10,12) (FSPEC(I), I=1,23), ETOT
READ (10,13) (DENS(I), I=1,23)
READ (10,*)!blank footer line
! ****** [insert processing code here]
GO TO 20 !read next table
30 CONTINUE !end of file during read
```

Figure 31.7   Fortran read routine for Oceanweather ASCII file

Variable definitions:

CYM	Year and month in the format CCYYMM
DHM	Day, hour and minute (gmt) in the format DDHHmm
GP	Grid point
LAT	Latitude of grid point
LONG	Longitude of grid point
DEPTH	Depth of grid point
WS	Wind speed (m/s) at grid point
WD	Wind direction (from which, clockwise from true north)
TP	Peak period (sec)
VMD	Vector mean direction (to which)
"SE"	Sea partitions of TP, ETOT, and VMD
"SW"	Swell partitions of TP, ETOT, and VMD
HS	Significant wave height (m)
MO1, MO2	First and second spectral moments
DOMDR	Dominant direction (to which)
ANGSPR	Angular Spreading function
INLINE	In-line variance ratio
TAU	Always=0 for hindcast purposes
FREQ(I)	Geometric mean of the lower and upper ends of the bandwidth. The frequency bands are given below.
THETA(J)	Mean direction of angular bin, to which waves are traveling, clockwise from true north. The bin extends +/- 7.5 degrees from the center (the value displayed in the table).
SPEC(I,J)	Variance component (not spectral density), in m^2, in frequency band I and angular band J.
ANSPEC(J)	Variance summed over all frequencies per direction (the right most column).
FSPEC(I)	Variance summed over all directions per frequency (the first footer line).
ETOT	Total variance located at the end of the first footer line.
DENS(I)	Frequency spectrum represented as density in units of m^2 Hz (second footer line).

Frequency bands:

```
nom. freq left end right end bandwidth
1 0.0390000 0.0371742 0.0409155 0.0037413
2 0.0429251 0.0409155 0.0450333 0.0041178
3 0.0472451 0.0450333 0.0495656 0.0045323
4 0.0520000 0.0495656 0.0545540 0.0049884
5 0.0572334 0.0545540 0.0600444 0.0054904
6 0.0629935 0.0600444 0.0660874 0.0060430
7 0.0693333 0.0660874 0.0727386 0.0066512
8 0.0763112 0.0727386 0.0800592 0.0073206
9 0.0839914 0.0800592 0.0881166 0.0080574
10 0.0924444 0.0881166 0.0969849 0.0088683
11 0.1017483 0.0969849 0.1067457 0.0097608
12 0.1119885 0.1067457 0.1174888 0.0107431
13 0.1232593 0.1174888 0.1293131 0.0118244
14 0.1356644 0.1293132 0.1423275 0.0130144
15 0.1493180 0.1423275 0.1566517 0.0143242
16 0.1643457 0.1566517 0.1724175 0.0157658
17 0.1808858 0.1724175 0.1897700 0.0173525
18 0.1990906 0.1897700 0.2088690 0.0190989
19 0.2191276 0.2088690 0.2298900 0.0210211
20 0.2411811 0.2298900 0.2530267 0.0231367
21 0.2654541 0.2530267 0.2784919 0.0254652
22 0.2921701 0.2784919 0.3065200 0.0280281
23 0.3215748 0.3065200 2.5274134
```

Figure 31.8   Description of variables used in code in Figure 31.7.

Details

Frequency bins

Frequency bins are spaced in geometric progression (to facilitate the computation of interactions); the nominal frequency is the geometric mean of the two ends (see table below format explanation). The frequency ratio is $0.75^{(-1./3.)}$, i.e. 1.100642416; this ratio was chosen in preference to the 1.1000 of official WAM to simplify interaction formulas.

The first 22 bins are straightforward; the last requires explanation. The 23rd frequency band is an integrated band comprising what would be bins 23 through 44 (continuing the geometric progression) of a fully discrete bin system. To model the cascade of wave energy from high to low frequencies endorsed by non-linear interactions, we compute interactions involving bins out to 44. This requires a parametric assumption about the spectral density between 0.30652 and 2.52741 Hz; and the customary assumption is that density is proportional to $\omega^{(-x)}$, where x is a disposable parameter. We are using x = 4.5 for the following reasons:

1. There are quasi-physical arguments supporting the exponents 4 & 5. The exponent 5 is germane to a Pierson-Moskowitz spectrum.

2. A crude energy balance computation in the tail, with wind input scaled as ω^2 and interactions scaled as ω^{11}, shows that 4.5 is the only exponent capable of yielding an equilibrium spectrum in the tail.

Density

To compute a "density" at 0.32157 Hz, we compute what fraction of the integrated band belongs to the bin from 0.30652 to 0.33737 Hz. Sparing a few details, the result is:

dens = (variance component)*rbw

where rbw (dimensions seconds) is a function of the exponent as follows:

x	rbw
4.0	8.11849
4.5	9.24794
5.0	10.32933

31.2   Wave Spectra File

The second task is to set the number of files and their location.
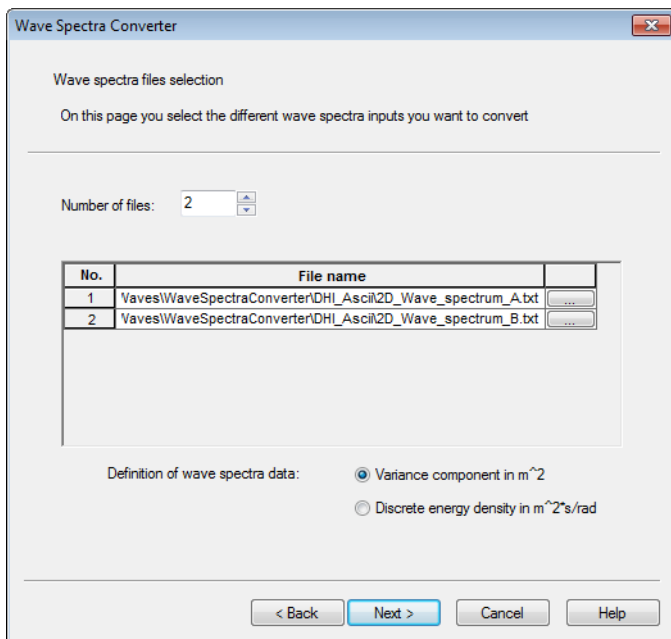
Figure 31.9   Selection of wave spectra input files

Multiple 2D wave spectra input data files can be converted in one single instance of the Wave Spectra Converter tool but these files have to share the same input format and the same discretisation (temporal, frequency and direction). Validation on discretisation is made during the execution of the tool and not at the graphical interface level. Check the log file in case of problem for further information.

When converting multiple files in a same setup the order of these files determines the structure of the line series in the outputs (from first file to last file).

In case the wave spectra data is specified in the format DHI ASCII you get the option to specify whether the spectral data is given by the variance component or the discrete energy density.

31.3   Output

The last task is to set the different outputs of the conversion's process. In addition to the expected conversion of 2D wave spectra into DHI Dfs format some post processed outputs can also be added to the setup.

The Wave Spectra Converter possible outputs are presented in three successive outputs pages based on the output type:

● Wave parameters

● 1D wave integrated spectra

- 2D wave spectrum

When converting multiple wave spectra files in a same setup the output data can be presented as line series in addition to the standard point series output. The line series files can be used as "varying in time and along line" boundary conditions in MIKE 21 Spectral Waves FM model.
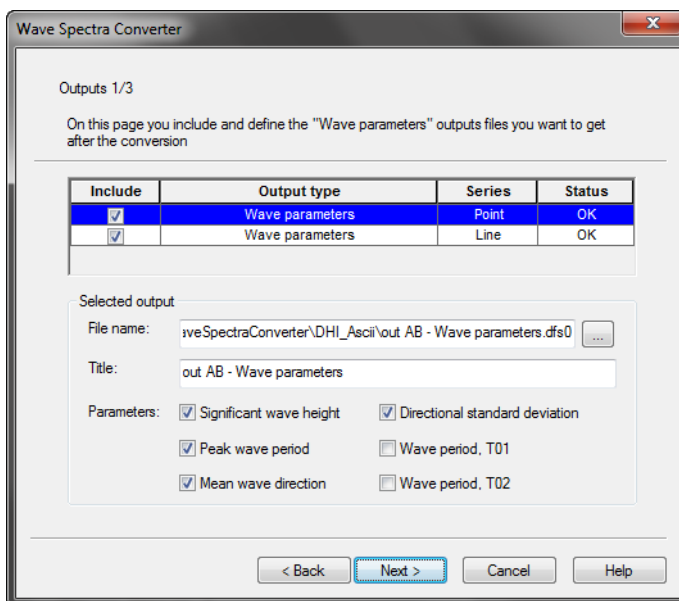
Figure 31.10   Specifying Wave parameters for output

Each individual output can be included in the conversion setup by checking the corresponding checkbox in the grid list.

If included in the setup the output selected in the grid list (highlighted in blue) needs to be individually set in the lower part of the output page (file name, title and individual parameters).

The status column is an indicator regarding the settings of the corresponding output. Status is not set if the corresponding file is not included in the setup. Status is automatically set to "OK" if the corresponding output is correctly set. Status is set to "Error" if:

- The file name is not set

- The file extension is incompatible with the nature of the corresponding output

- No parameter is checked for output

31.3.1 Wave parameters

The first output type is the so-called "Wave parameters" one. These parameters are obtained by post processing the 2D wave energy spectrum in a similar way to the MIKE 21 Spectral Waves FM model. The following parameters can be computed by the Wave Spectra Converter tool:

- Significant wave height

- Peak wave period

- Mean wave direction

- Directional standard deviation

- Wave period, T01

- Wave period, T02

The first four parameters (checked by default) can be used as input data conditions to a boundary of type "Wave parameters (version 2)" in MIKE 21 Spectral Waves FM model. When the "Wave parameters" output of this tool is formatted as line series (multiple wave spectra required as input) a "varying in time and along boundary" boundary condition can be used in MIKE 21 SW.

See MIKE 21 Spectral Waves FM scientific documentation for the definition of these parameters.

31.3.2 1D wave integrated spectra

The outputs of type "Wave integrated spectrum" are post processed results of the 2D wave energy spectrum:

- The direction energy/action spectrum is obtained by integration over the discretised frequencies

- The frequency energy/action spectrum is obtained by integration over the discretised directions

For these different spectrum outputs it is possible to include the wave energy and/or wave action as output data parameters. See MIKE 21 Spectral Waves FM scientific documentation for the definition of these parameters.
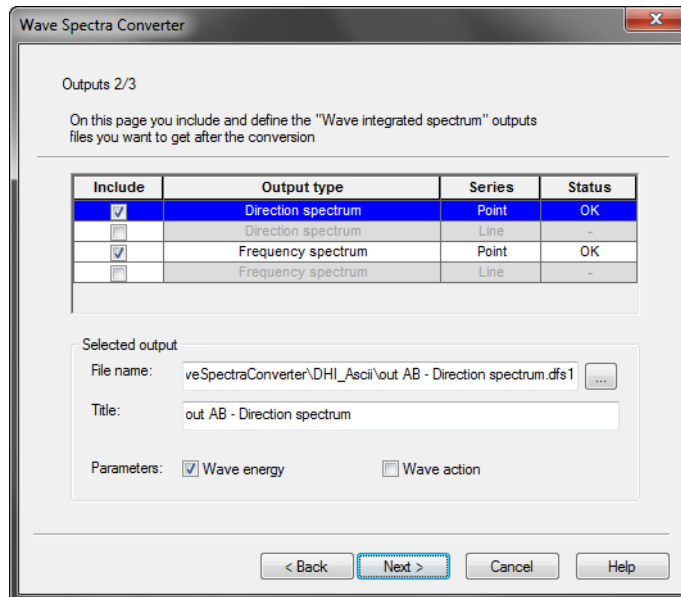
Figure 31.11   Specifying Wave integrated spectrum as output

It is possible to format "Wave integrated spectra" outputs data as line series when multiple input spectra are selected as input to this tool. However these outputs as line series are of no practical use so far with MIKE software and are therefore not included by default in a new setup of the Wave Spectra Converter tool.

31.3.3   2D wave spectrum

The outputs of type "2D wave spectrum" are the natural outputs of the Wave Spectra conversion process without any kind of post processing involved.

2D wave spectrum outputs as Point series can be saved in DHI Dfs format either as dfsu or dfs2 files. Both files are accepted as boundary conditions to the MIKE 21 Spectral Waves model but the dfs2 one can also be used by the Polar Plot control of the Plot Composer to produce graphics and/or animations of the 2D wave energy spectra.
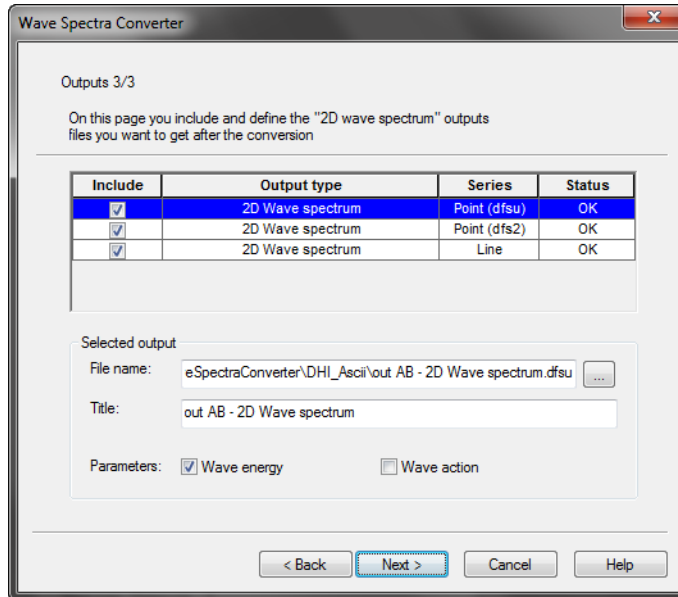
Figure 31.12   Specifying 2D wave spectrum as output

When converting multiple wave spectrum files in a same setup the output data can be presented as line series in addition to the standard point series output but only as a dfsu file. The line series files can be used as "varying in time and along line" boundary conditions in MIKE 21 Spectral Waves FM model.

For these different spectrum outputs it is possible to include the wave energy and/or wave action as output data parameters. See MIKE 21 Spectral Waves FM scientific documentation for the definition of these parameters. Both parameters are accepted as boundary conditions to MIKE 21 SW.

31.4 Status
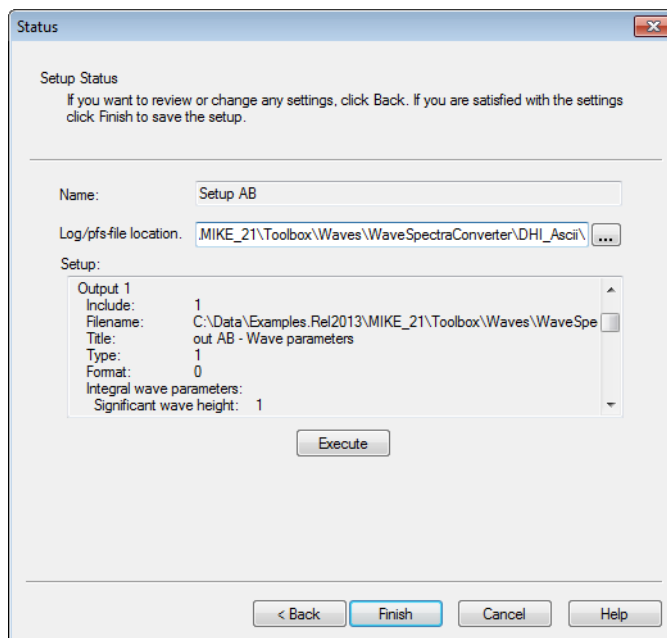
Figure 31.13 Overview page

The status page presents you with a summary of all the input parameters you have specified in the present Wave Spectra Converter setup. You can inspect the parameters to check if they are correct, - and if not go back and change them.

From the page you can execute the setup or finish it, which will transfer your set-up definition to the current toolbox file.

The log/pfs location indicates where the log journal and intermediate pfs file are located. In case of problem during the wave spectra conversion process it is recommended to check the log file for additional information.

31.5 Example

The following figures describe how to convert two ascii files with directional wave spectra data into files of the DHI Dfs format.

The input files are located in the default installation folder at

.\Examples\MIKE_21\Toolbox\Waves\WaveSpectraConverter.
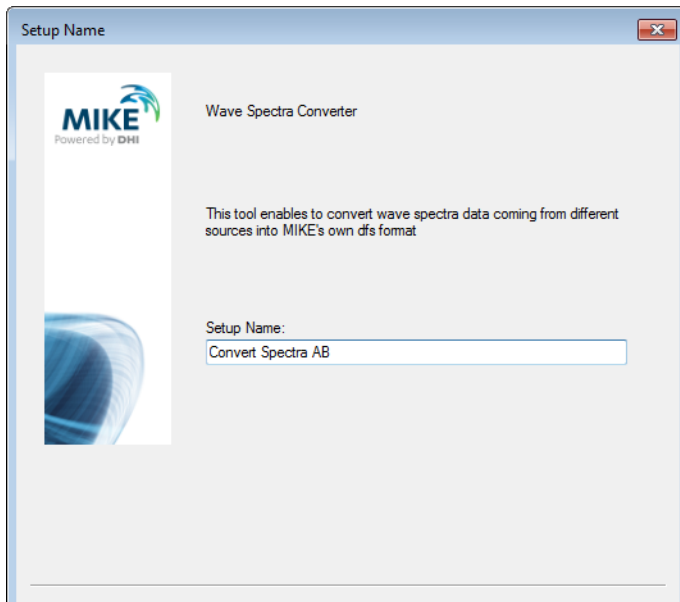
Figure 31.14   Specify a setup name
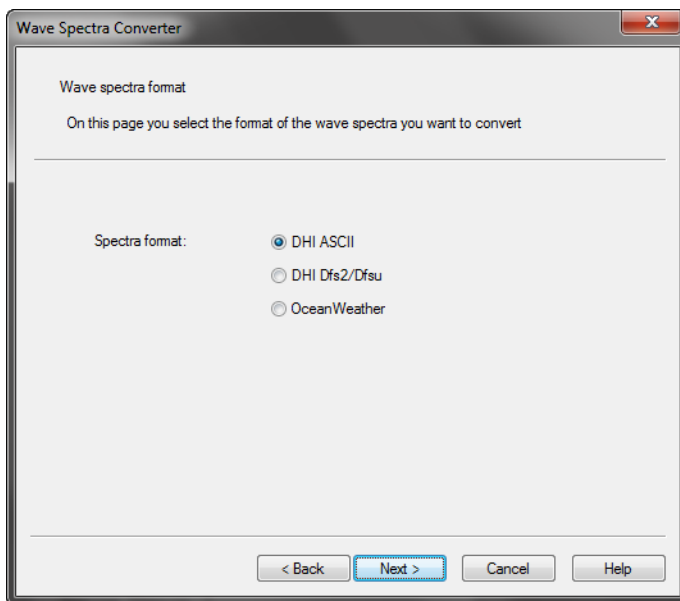
Figure 31.15   Specify the format of the spectra to be converted
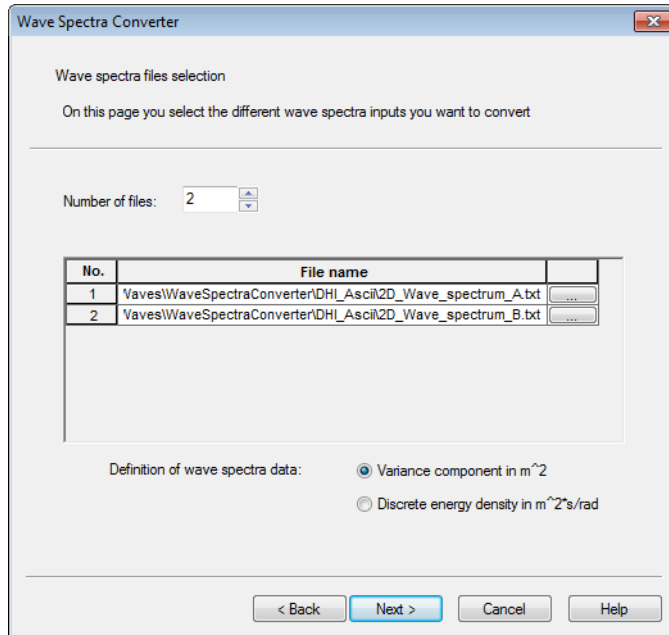
Figure 31.16   Specify the input data files and type of wave spectra data
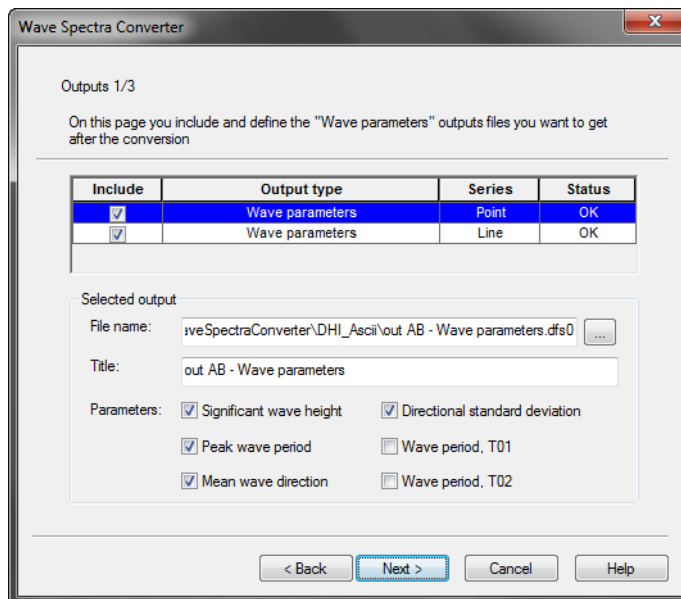
Figure 31.17   Specify the name and type of output wave parameters
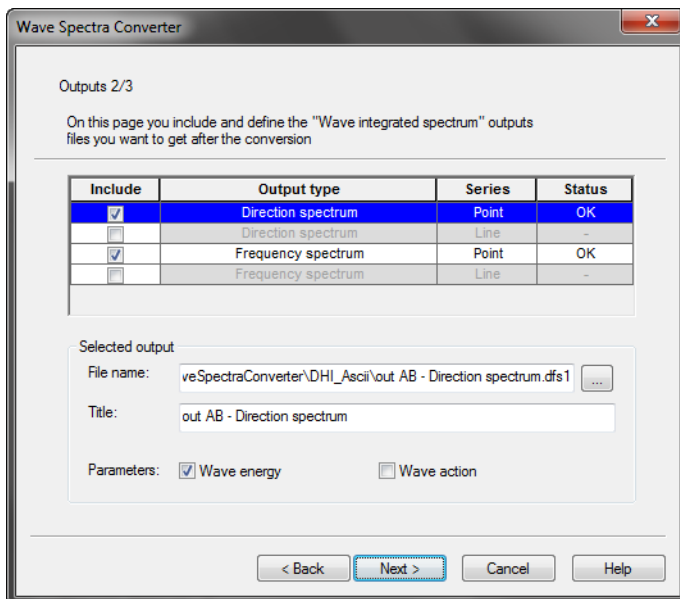
Figure 31.18   Specify name and type of integrated wave spectrum output
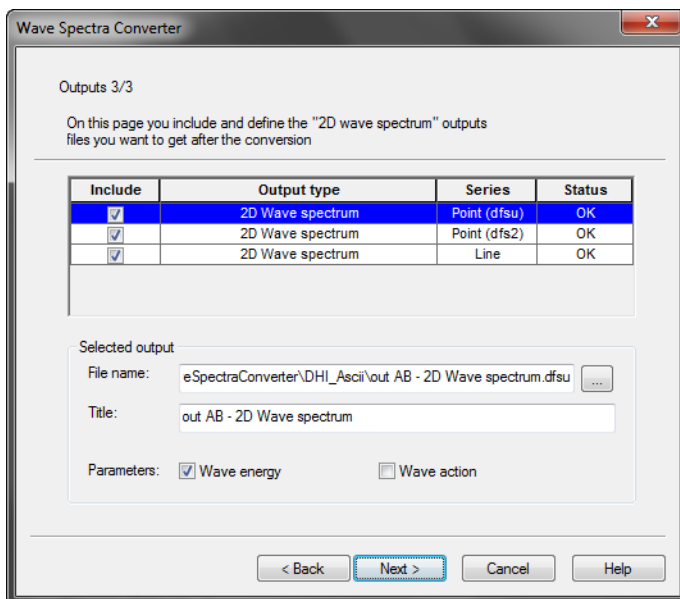
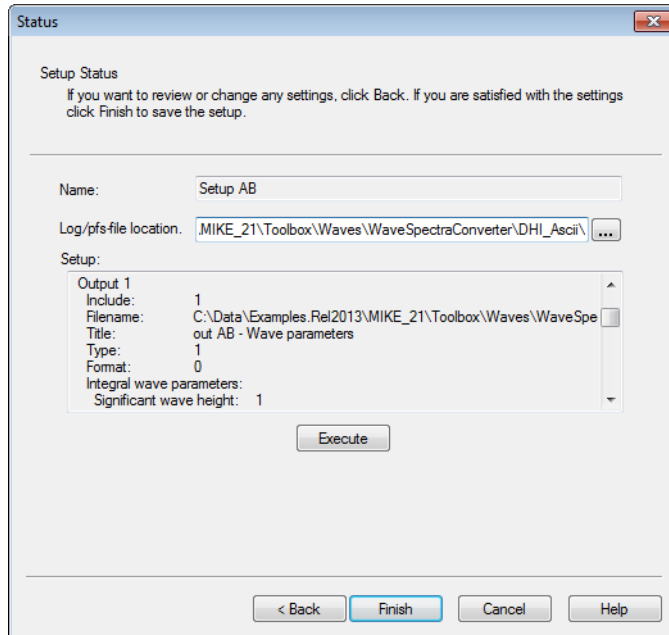Figure 31.19   Specify name and type of 2D wave spectrum output

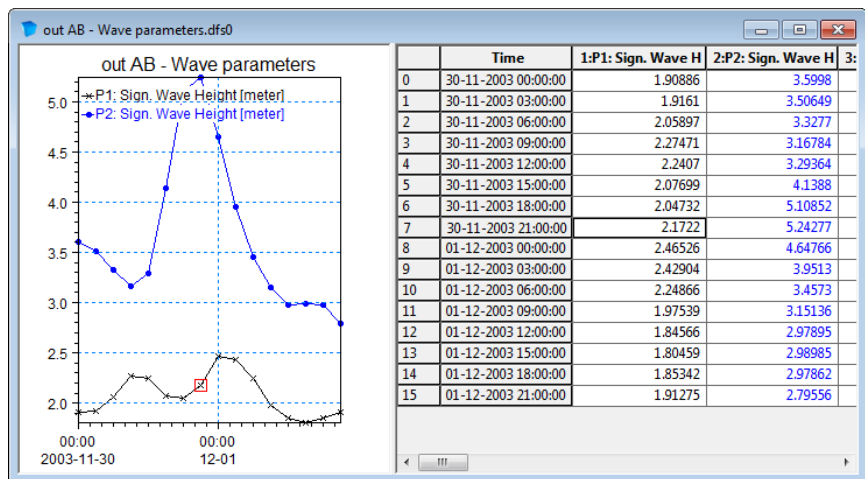Figure 31.20   Specify location of log file and execute the program

Figure 31.21   Result:. Significant wave height from wave parameters output

M21 TOOLS: WIND

The WIND Tools in the M21 Toolbox include the facilities:

- Cyclone Wind Generation
- Generating Wind Fields from Pressure Maps

32 Cyclone Wind Generation

The MIKE 21 Cyclone Wind Generation tool allows users to compute wind and pressure data due to a tropical cyclone (hurricane or typhoon). Wind and pressure data generated by a tropical cyclone can often be described by simple parametric models based on few parameters like position of the cyclone's eye, radius of the maximum winds, etc. Most of these parameters are generally available from meteorological publications as Best Track data table for each cyclone. Other parameters may need to be derived by the user or adjusted based on wind or pressure measurements.

Several cyclone parametric models are included in the present tool. In addition to that different kind of wind corrections can be applied by the user to reflect the cyclonic wind structure. Wind and pressure data are then computed for different formats of output files allowing the user to easily investigate the cyclonic forcings. Among them space and time varying wind and pressure fields can be obtained for later use in a MIKE hydrodynamic or wind wave generation simulation. In that way cyclonic surges or waves can be computed.

32.1 Setup name

On this introduction's page the user can specify a name for the setup.

32.2 Model and Best Track Data

In order to compute the wind and pressure data the user must first choose one of the parametric models. Four parametric models are available to compute wind and pressure data due to tropical cyclones:

- Young and Sobey
- Holland - single vortex
- Holland - double vortex
- Rankine

Each of these models requires specific time varying parameters of the cyclone, some of these being common to different parametric models (like the position of the eye for example) while others remain specific to fewer models. Based on the selected parametric model some of the cells of the Best Track data table are grayed out to indicate parameters which are irrelevant with the user model's choice. These grayed cells may therefore be left empty (validation check is made only on the non grayed cells).

The track data information is defined relative to the start time of the cyclone period.

You may either specify the number of points in the track yourself or load in an ASCII file (.txt) including the table values. The ASCII data has to be in comma or space separated form. The ASCII file is inserted automatically into the dialog.

32.2.1 Best Track data table

12 parameters (or columns) are defined in a Best Track data table common to all parametric models and need to be filled in by the user to a certain extent. A short explanation of each of these parameters is given here, specific uses of these parameters is available in the scientific documentation or the related references:

1. Time in hours from the start of the cyclone period (which is specified on the start date above the table). You must specify one line for each point in time (chronologically) when the cyclone parameters are known. The interval in time between the lines does not have to be constant and does not have to correspond to the time interval specified for the output data. The first line must specify the conditions for time = 0.

2. Longitude of the centre of the cyclone (E positive and W negative)

3. Latitude of the centre of the cyclone (N positive and S negative)

4. Radius to maximum wind speed R_{max} (or $R_{max,1}$, given in km)

5. Maximum wind speed V_{max} (in m/s)

6. Central pressure P_c in hPa

7. Neutral pressure Pn (the pressure outside the area influenced by the cyclone, in hPa)

8. Holland parameter B (or B_1)

9. Pressure deficit ΔP_2

10. Holland parameter B_2 (second vortex)

11. Second Radius to maximum wind speed $R_{max,2}$ (in km)

12. Rankine parameter X

Based on the selected parametric model some of the cells of the Best Track data table are irrelevant with the user model's choice. Figure 32.1 gives an overview of the requested information for each model.

	C1	C2	C3	C4	C5	C6	C7	C8	C9	C10	C11	C12
Parametric models	Time	LONG.	LAT.	Rmax	Vmax	Pc	Pn	B	DP2	B2	Rmax2	X
Young and Sobey												
Holland – single vortex												
Holland – double vortex				= Rmax1				= B1				
Rankine												

Figure 32.1   Necessary parameters from the Best Track data table
Grey cells correspond to irrelevant data for the given model

Additional information regarding the equations and requested parameters for each of these parametric models can be found in the scientific documentation which is accessible from the MIKE 21 Documentation Index.

Example of cyclone track

An example of a set of cyclone parameters for the Young and Sobey model is shown in Table 32.1.

Table 32.1   Cyclone Parameters for hurricane in Gulf of Mexico

Time (hours)	Longitude (degrees)	Latitude (degrees)	Radius (km)	V_max (m/s)	$P_{central}$ (hPa)	$P_{neutral}$ (hPa)
0	-72.8	15	45.6	67.056	923	1013
12	-74.7	16.3	45.6	64.8208	934	1013
24	-76.5	17.3	45.6	65.9384	926	1013
36	-78.4	17.7	45.6	67.056	923	1013
48	-79.7	18.2	45.6	73.7616	912	1013
60	-81.2	18.8	45.6	69.2912	919	1013
72	-82.8	19.5	45.6	67.056	915	1013
84	-84.1	20.4	45.6	71.5264	919	1013
96	-85.1	21.6	45.6	71.5264	914	1013
108	-86.1	23.1	45.6	71.5264	925	1013
120	-87	24.7	42.596	61.7328	928	1013
132	-89.9	26.7	27.78	58.127	939	1013

Table 32.1   Cyclone Parameters for hurricane in Gulf of Mexico

Time (hours)	Longitude (degrees)	Latitude (degrees)	Radius (km)	V_max (m/s)	P_central (hPa)	P_neutral (hPa)
144	-88.19	28.85	38.892	47.843	931	1013
156	-87.7	31.4	72.228	30.864	965	1013

Setup examples of cyclone wind generation for the northern and southern hemisphere, respectively, can be found in the default installation folder at .\MIKE Zero\Examples\MIKE_21\Toolbox\Wind\Cyclone.

32.3   Wind Correction

32.3.1   Geostrophic correction

The different parametric models usually provide wind information at the geostrophic or gradient wind level above the influence of the planetary boundary layer. This gradient wind speed *Vg* may be reduced to the standard surface reference level *Vm* by considering the effects of the boundary layers:

$$V_m = K_m \cdot V_g \tag{32.1}$$

Three options are available with the current tool:

- No correction

- Constant correction. In that case the user must also specify the constant boundary layer coefficient *Km*.

- Harper et al. In that case *Km* is dependent of *Vg* through a set of equations derived by Harper et al.
  (see scientific documentation for more details)

32.3.2   Forward motion asymmetry

The movement of the cyclone creates an asymmetry of the wind which can be accounted for.

Two options are available with the current tool:

- No correction

- Harper et al. In that case the user must specify the proportion of the correction (dfm factor) and the angle of maximum winds Omax
  (see scientific documentation for more details)

32.3.3 Inflow Angle

Friction effects between water and air cause a deflection of the wind direction towards the centre of the cyclone.

Two options are available with the current tool:

- No correction

- Sobey et al. The deflection is characterised by the angle is of the order of 25° but decreases towards the storm centre
  (see scientific documentation for a complete set of equations)

32.4 Area Series Output

This output type provide space and time varying wind and pressure fields which can be used in a MIKE hydrodynamic or wind wave generation simulation. The MIKE 21 Cyclone Wind Generation tool makes computations of wind and pressure data for each point of a grid based on a user defined domain extent (a dfs2 file usually representing e.g. the bathymetry).

Several 2D outputs can be defined and included or not in the final computation, enabling the user to assess the cyclone climatology over several domains within one single setup. Adding, deleting and including area outputs are controlled by the upper left list box. Navigating between the different 2D outputs definition is achieved by clicking on the number of the output in the same control.

For each 2D output included in the final computation the user must provide:

- A file name and a title for this output (see Output name section)

- Information on the time of the output data (see Period section)

- A domain file (dfs2 file) which extent and resolution will be used as template for the current output (see Domain name section).

32.5 Time Series Output

This output type provide time series files with wind and pressure information at specific user-defined points. This can for example be useful for comparison with local measurements.

Several time series outputs can be defined and included or not in the final computation domains within one single setup. Adding, deleting and including time series outputs are controlled by the upper left list box. Navigating between the different time series outputs definition is achieved by clicking on the number of the output in the same control.

For each time series output included in the final computation the user must provide:

- A file name and a title for this output (see Output name section)

- Information on the time of the output data (see Period section)

- Information on the points location (see Points information section)

32.6   Radial Series Output

Wind and pressure information along different radials away from the centre of the cyclone can be computed using the radial series output. This output is particularly useful to assess the structure of the cyclone through characteristic profiles.

If this output is included the user must provide:

- A file name and a title for this output (see Output name section)

- Information on the time of the output data (see Period section)

- Information on the radials (see Radials definition).
  Four predefined radials are set by default to the North, East, South and West direction but can be edited by the user. The user must also specify the number of points and the step interval (in km) valid for all profiles

You may either specify the number of cyclone lines yourself or import an ASCII file (.txt) including the cyclone data. The ASCII data has to be in comma or space separated form. The ASCII file is inserted automatically into the dialog.

32.7   Output Items

The output items are common for all three types of output. The output will contain the following five items for each time step:

- Atmospheric Surface Pressure [hPa]

- X-component of wind velocity [m/s]

- Y-component of wind velocity [m/s]

- Wind speed [m/s]

- Wind direction [deg.N]

32.8   Status

The overview page presents you with a summary of all the input parameters you have specified. You should inspect the parameters to check they are correct, - and if not go back and change them.

From the page you can either *Finish* the set-up, which will transfer your set-up definition to the current toolbox file, or execute the set-up.

33 Generating Wind Fields from Pressure Maps

If wind fields for a hydrodynamic simulation or an offshore wave simulation is not available from a meteorological model, or if the wind fields cannot be described through a parametric model like the one used for cyclonic wind fields, a well tested way of generating the wind fields is to generate these from charts of atmospheric surface pressure. And that is what this tool can be used for.

An example is shown in Figure 33.1.
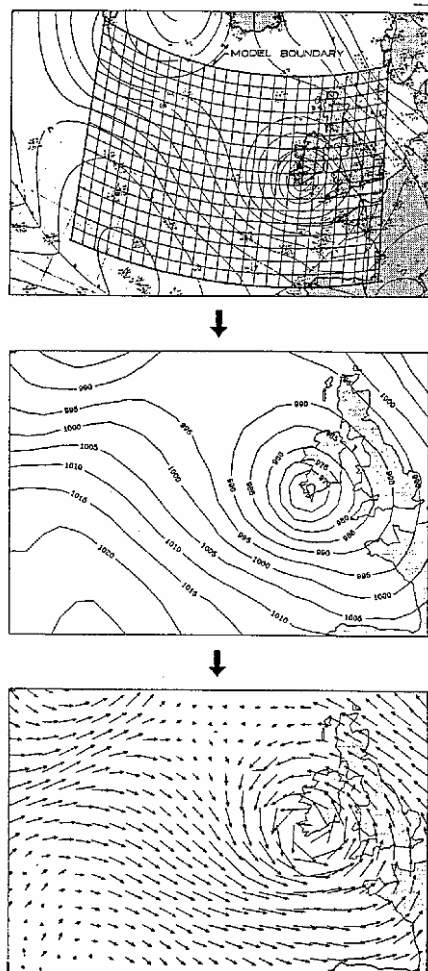
Figure 33.1   Atmospheric pressure chart with model grid (top), digitised pressure field (middle), and resulting wind field at 12 UTC, 12 March 1976. (from Ref. /36/)

33.1   Select Pressure Maps

Before you start using this tool you must prepare a type 2 data file containing the atmospheric pressure (in hPa) in each grid point of your model with a certain time interval (e.g. 3 hours).

The type 2 pressure data might be created by digitising the atmospheric pressure charts and then reading them into the data file using the import tool.

The temporal and spatial extent/resolution of this atmospheric pressure type 2 data file will, by using this tool, be inherited to the generated output-file (also type 2), which can be used as input file to a hydrodynamic simulation (it will contain wind and pressure data).

The time interval in this atmospheric pressure type 2 data file does not have to equal the one which is to be used in the hydrodynamic or offshore wave simulation. Nor does the grid spacing of the atmospheric pressure grid have to correspond to the grid spacing used in the simulation. However, if the grid spacing of this atmospheric pressure type 2 data file at this stage does not match the grid spacing used in the simulation, then you will need to use the Rotate Grid tool from the MIKE Zero Toolbox on the output-file (from this tool) to generate a type 2 data file with compatible grid spacing, before running the simulation.

33.2   Select Type of Wind Field

You can choose between following type of wind data:

- 10 m wind data
  I.e. wind speed taken 10 m above ground/sea

- Geostrophic wind data
  I.e. the theoretical wind which results from the equilibrium between horizontal components of the pressure gradient force and the Coriolis force (deviating force) above the friction layer. Only these two forces (no frictional force) are supposed to act on the moving air. It blows parallel to straight isobars or contours.

See also Scientific Background, p. 252.

33.3   Corrections Parameters

The wind fields determined on the basis of pressure differences can be corrected in three ways:

- The air-sea temperature difference, $\Delta T_{a\text{-}s}$
- The wind speed scaling factor

- The wind direction correction

You may specify the corrections as:

- Constant corrections for all grid points and for all time steps
- Time dependent corrections but constant for all grid points
- Time and space dependent corrections (only wind speed scaling and wind direction corrections)

33.3.1   The air-sea temperature difference, ΔT_{a-s}

The difference between the temperature of the air and the sea (ΔT_{a-s}) is an important parameter in the transformation of the geostrophic wind (through the boundary layer) into the 10 m wind.

ΔT_{a-s} can be specified in two ways:

- as constant for all time steps and all grid points. If the wind speed scaling factor and wind direction correction is also constant in time as well as in space you specify ΔT_{a-s} on the menu. If the wind speed scaling factor and wind direction correction vary in time, however, you must specify ΔT_{a-s} as item 1 of the type 0 data file, as described for a time varying ΔT_{a-s} (see Time and space dependent corrections *(p. 250)*).

- as varying in time but the same value for all grid points at each time step. In this case you must prepare a type 0 data file before starting to use this tool. The file must have 3 or 15 items (depending on whether the wind speed scaling factor and wind direction correction are constant in space or not, see Time and space dependent corrections *(p. 250)*) of which ΔT_{a-s} should be the first one. The starting date, the time interval and the number of time step in the file should be the same as for the file containing the pressure maps.

33.3.2   Wind field corrections

Although the wind fields calculated from the pressure fields are good approximations to the real wind fields, corrections to the wind speeds as well as the wind directions are sometimes required (e.g. after comparing the computed wind fields to synoptic weather charts). These wind speed scaling factors and wind direction corrections can be specified in three ways:

1. Constant corrections (Specifying ΔT_{a-s}, wind speed scaling factor and wind direction correction as constants on a menu)

2. Time dependent corrections (Specifying ΔT_{a-s}, wind speed scaling factor and wind direction correction as items 1, 2 and 3 of a type 0 file)

3. Time and space dependent corrections (Specifying a type 2 file (containing 2 items) with correction codes (1 through 7) and a type 0 file containing ΔT_{a-s} and 7 sets of wind speed scaling factors and wind direction corrections)

33.3.3 Constant corrections

If ΔT_{a-s} is also constant in time as well as in space you specify the wind speed scaling factor (a value of 1.0 corresponds to no scaling) and the wind direction correction (in degrees clockwise) on the menu.

If ΔT_{a-s} varies in time, however, you must specify the wind speed scaling factor and the wind direction correction as item 2 and 3, respectively, in the type 0 data file containing the ΔT_{a-s} values (see above).

33.3.4 Time dependent corrections

In this case you must prepare a type 0 data file (before starting to use this tool) as described for a time varying ΔT_{a-s}. The wind speed scaling factor and the wind direction correction should be specified as item 2 and 3, respectively.

33.3.5 Time and space dependent corrections

For ΔT_{a-s}, wind speed and wind direction corrections **varying in time** you must prepare a type 0 file which for each time step i n'th pressure file contains the following items. Note that only the first item is required if space dependent corrections is not applied.

1. ΔT_{a-s}
2. Wind Speed Scaling Factor for areas with correction code 1
3. Wind Direction Correction for areas with correction code 1
4. As item 2 but for areas with correction code 2
5. As item 3 but for areas with correction code 2
6. As item 2 but for areas with correction code 3
7. As item 3 but for areas with correction code 3
8. As item 2 but for areas with correction code 4
9. As item 3 but for areas with correction code 4
10. As item 2 but for areas with correction code 5
11. As item 3 but for areas with correction code 5
12. As item 2 but for areas with correction code 6

13. As item 3 but for areas with correction code 6

14. As item 2 but for areas with correction code 7

15. As item 3 but for areas with correction code 7

For $\Delta T_{a\text{-}s}$, wind speed and wind direction corrections **varying in space** you must prepare two maps of correction codes (i.e. a type 2 data file with 1 time step and 2 items) with the same spacial dimensions as the pressure maps. The first map (= item) should contain the correction codes for the wind speed velocity scales while the second map should contain the codes for the wind direction cor-rection. Each grid point must be given a value (the correction code) between 1 and 7 (including both 1 and 7). These codes corresponds to items 2 to 15 in the type 0 file described above.

33.4 Select Type of Latitude

The geostrophic wind speed (and thus also the 10 m wind speed) depends on the latitude. It should therefore be specified for each grid point, which can be done in two ways:

- as the latitude of the grid origin (i.e. point (0,0)) and the model orientation. The latitude for each grid point will then be computed by assuming a linear variation in space

- as a map (i.e. a type 2 file with one time step and one item) having the same dimensions as the pressure maps giving the latitude in each grid point (in degrees, not degrees, minutes and seconds). This possibility is useful e.g. for models close to the poles.

Please note that latitudes on the southern hemisphere should be specified as negative values.

33.5 Select Output File

On this page you specify the output file. It will contain the following items for each time step:

1. Atmospheric Surface Pressure [hPa]

2. X-component of wind velocity [m/s]

3. Y-component of wind velocity [m/s]

33.6   Overview

The overview page presents you with a summary of all the input parameters you have specified. You should inspect the parameters to check they are correct, - and if not go back and change them.

From the page you can either *Finish* the set-up, which will transfer your set-up definition to the current toolbox file, or execute the set-up.

33.7   Scientific Background

The geostrophic wind speed, U_g, is defined as

$$U_g = \frac{1}{f \cdot \rho_a} \frac{dP}{dn}$$

(33.1)

where

$$f = 2 \cdot \Omega \cdot \sin \theta$$

ρ_a = 1.247 kg/m³ (the density of the air)

$\dfrac{dP}{dn}$ is the horizontal pressure gradient

Ω = 7.292 · 10^{-5} rad/s (the angular velocity of the earth) and

θ is the latitude

The horizontal pressure gradient is calculated from the digitised pressure map as

$$\left[\frac{dP}{dn}\right]_{j,k} = \sqrt{\left(\frac{P_{j+1,k} - P_{j-1,k}}{2\Delta x}\right)^2 + \left(\frac{P_{j,k+1} - P_{j,k-1}}{2\Delta y}\right)^2}$$

(33.2)

where P is the pressure in (Pa), j and k are grid numbers in the x and y direction and Δx and Δy are the grid spacings. For points at the model boundaries forward/backward differences are used instead of the central differences given above.

The wind speed 10 m above sea surface, U_{10} , is calculated in accordance with Ref. /37/:

$$U_{10} = 0{,}7\,U_g \qquad\qquad \text{for} \quad U_g \le 15 \text{ m/s}$$
$$U_{10} = a\sqrt{b U_g + c} + d \quad \text{for} \quad U_g > 15 \text{ m/s}$$

(33.3)

where

a = 6.82
b = 0.54 - 0.012·$\Delta T_{a\text{-}s}$
c = 1.68 - 0.105·$\Delta T_{a\text{-}s}$
d = -11

and where $\Delta T_{a\text{-}s}$ is the air-sea temperature difference.

The wind direction for the geostrophic wind is parallel to the isobars while a deflection of 15 degrees towards the lower pressure is applied for the 10 m wind (see Figure 33.2).
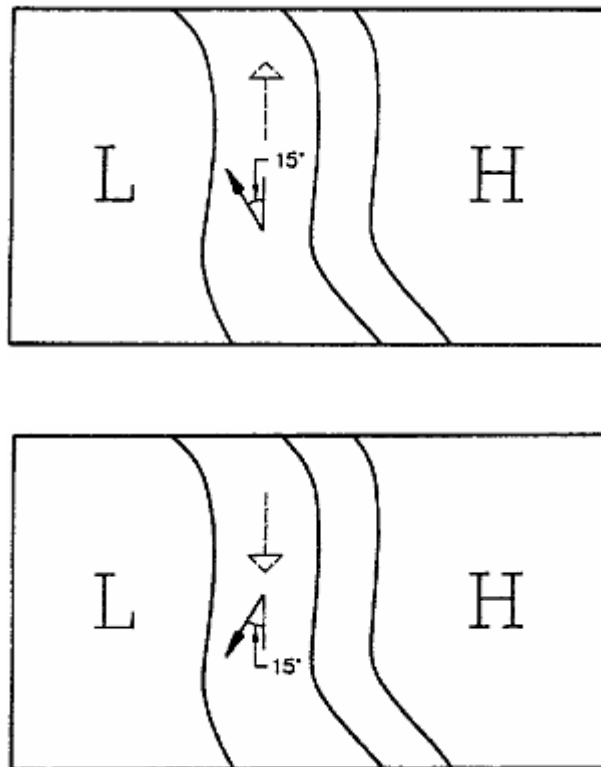
Figure 33.2 Wind direction for geostrophic wind (dashed arrow) and 10 m wind (full line arrow with a deflection of 15°).
Upper: Northern Hemisphere
Lower: Southern Hemisphere

33.8   References

/36/   Rugbjerg, M. *Numerical Modelling of Large, Medium and Small Scale Wind-Wave Fields for Use in Harbour* Design, Baltic Sea Conf., Norrköping, Sweden, 1987.

/37/   Duun-Christensen, J.T. *the Representation of the Surface Pressure*

Field in a Two-Dimensional Hydrodynamic Model for the north Sea, The Skagerrak and the Kattegat, Deutsche Hydrographische Zeitschrift, Vol. 28, No. 3, pp. 97-116, 1975.

INDEX