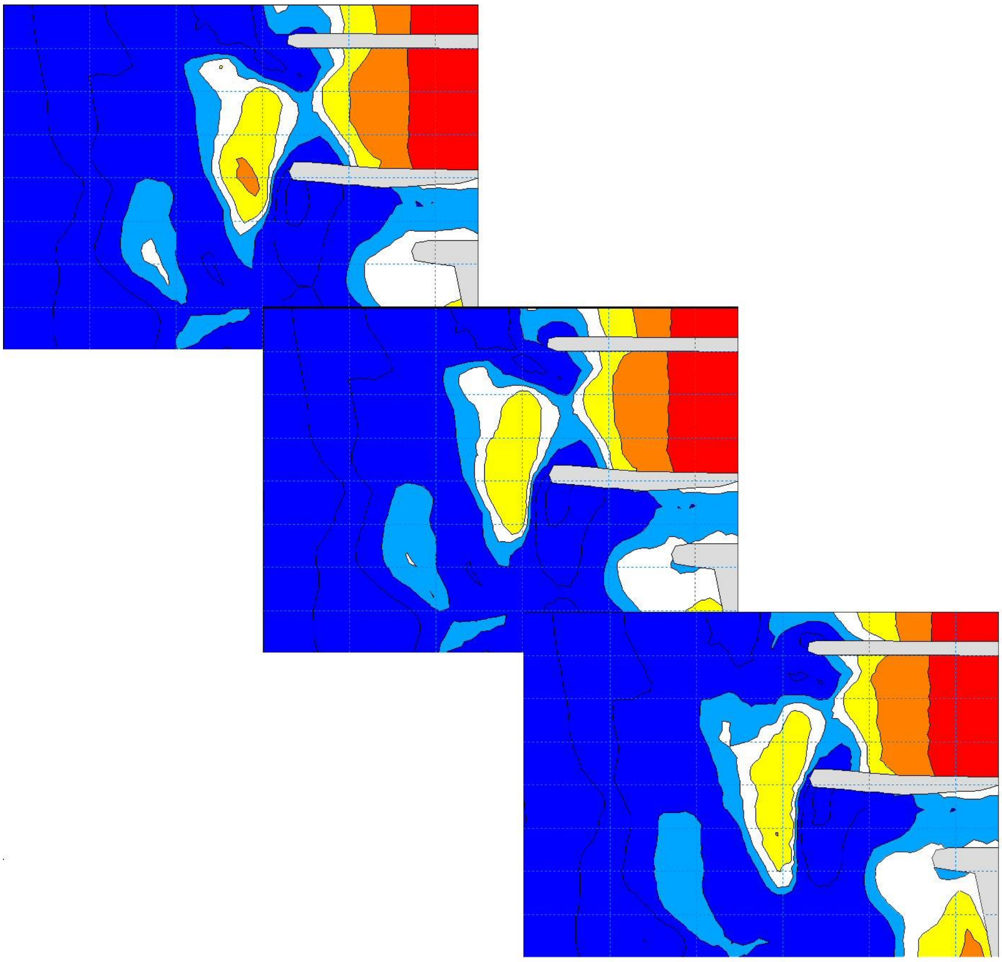

MIKE 21/3 Coupled Model FM

Step-by-step training guide: Coastal application

DHI A/S headquarters
Agern Allé 5
DK-2970 Hørsholm
Denmark

+45 4516 9200 Telephone
+45 4516 9333 Support
+45 4516 9292 Telefax

mike@dhigroup.com
www.mikepoweredbydhi.com

CONTENTS

MIKE 21/3 Coupled Model FM
Step-by-step training guide: Coastal application

1 Introduction

This Step-by-step training guide relates to Torsminde Harbour on the west coast of Jutland, Denmark, see Figure 1.1 and Figure 1.2.

Figure 1.1 Torsminde Harbour, Denmark. View towards north

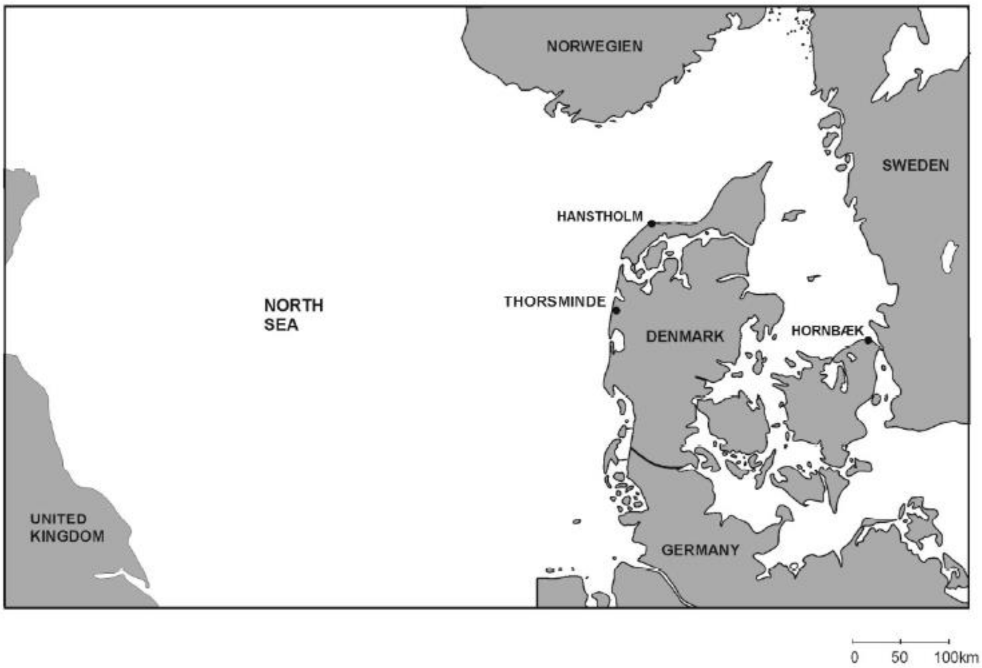

Figure 1.2 Map of the North Sea area. Torsminde is located in the western part of Jutland, Denmark

1.1 Background

Torsminde fishery harbour is located at a tidal inlet on the west coast of Jutland, Denmark, on one of the narrow tidal barriers which divide coastal lagoons from the sea. The port is located at the entrance to the coastal lagoon Nissum Fjord. Sluices regulate the water exchange between the lagoon and the sea.

Up until the 1980's, i.e. for about 100 years, critical parts of the coast were protected by traditional structures such as groynes. A comprehensive nourishment scheme was established in the mid-eighties. Nourishment of about 3 million m^3/year along a stretch of approximately 115 km has now stabilised the beach at critical stretches, while other stretches have been left to retreat as part of an overall shoreline management plan prepared and controlled by the Danish Coastal Authority.

Torsminde harbour is located in the central part of a very exposed stretch, where the net littoral drift is southward with an order of magnitude of 0.4 million m^3/year, but where the gross transport is several times larger.

As a result severe sedimentation and shoaling problems affected the harbour entrance and a need for alternative layout of the harbour made it necessary to make preliminary investigations of the sediment transport pattern in the area.

1.2 Objective

The objective of this Step-by-step training guide is to set up a model simulating the morphology for the Torsminde harbour and the adjacent coastal areas from scratch and to calibrate the model to a satisfactory level.

The coupled model setup involves the following MIKE 21 modules:
- MIKE 21 Flow Model FM, Hydrodynamic Module
- MIKE 21 Spectral Wave Module
- MIKE 21 Flow Model FM, Sand Transport Module

Attempts have been made to make this exercise as realistic as possible although some short cuts have been made with respect to the data input. This mainly relates to quality assurance and pre-processing of raw data to bring it into a format suitable for the MIKE Zero software. Depending on the amount and quality of the data sets this can be a tedious, but indispensable process. For this example the 'raw' data has been provided as standard ASCII text files.

All files used in this Step-by-step training guide are a part of the installation. You can install the examples from the MIKE Zero start page.

Please note that all future references made in this Step-by-step guide to files in the examples are made relative to the main folders holding the examples.

User Guides and Manuals can be accessed via the MIKE Zero Documentation Index in the start menu.

If you are familiar with importing data into DHI's file system, you do not need to generate all the MIKE Zero input parameters yourself from the included raw data. All the MIKE Zero input parameter files required to run the example are included and the simulation can be initiated right away.

2 Creating the Computational Mesh

Creation of the mesh file is a very important task in the modelling process.

The mesh file couples water depths with different positions and contains the following information:

1. Computational grid
2. Water depths
3. Boundary information

Creation of the mesh requires an xyz file(s) for positions of land-boundaries and an xyz file(s) that couples geographical positions with water depths. A data file including a bathymetry can also be used as an alternative to an xyz file with water depths.

Creation of the computational mesh typically requires numerous modifications of the data set, so instead of explaining every mouse click in this phase, the main considerations and methods are explained in the MIKE FM HD document:

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

which can be accessed via the MIKE Zero Documentation Index in the start menu.

When creating the computational mesh you should work with the mdf file, which contains information about settings for the mesh. Focus in this example is on Sand Transport Module (ST), so both the necessary xyz files and the mdf file and a mesh file are already supplied with the example, so that the remaining time can be spent on specific ST subjects. Two mesh files are supplied with the example: a coarse mesh and a fine mesh. Using the coarse mesh will reduce simulation time as fewer calculations are needed and the time step may be increased.

An outline of the properties for the two bathymetries is given in Table 2.1.

Table 2.1 Mesh bathymetry properties

	Fine mesh	Coarse mesh
No. of elements	7876	3674
No. of nodes	4211	1997
Minimum element length	4.8 m	11.8 m
Maximum element length	214.8 m	241.0 m
Minimum element area	13 m^2	107 m^2
Maximum element area	21980 m^2	28648 m^2

Figure 2.1 shows the fine mesh, which is used in this example, and Figure 2.2 shows a graphical presentation of the mesh of the Torsminde area as it can be displayed with MIKE Animator.

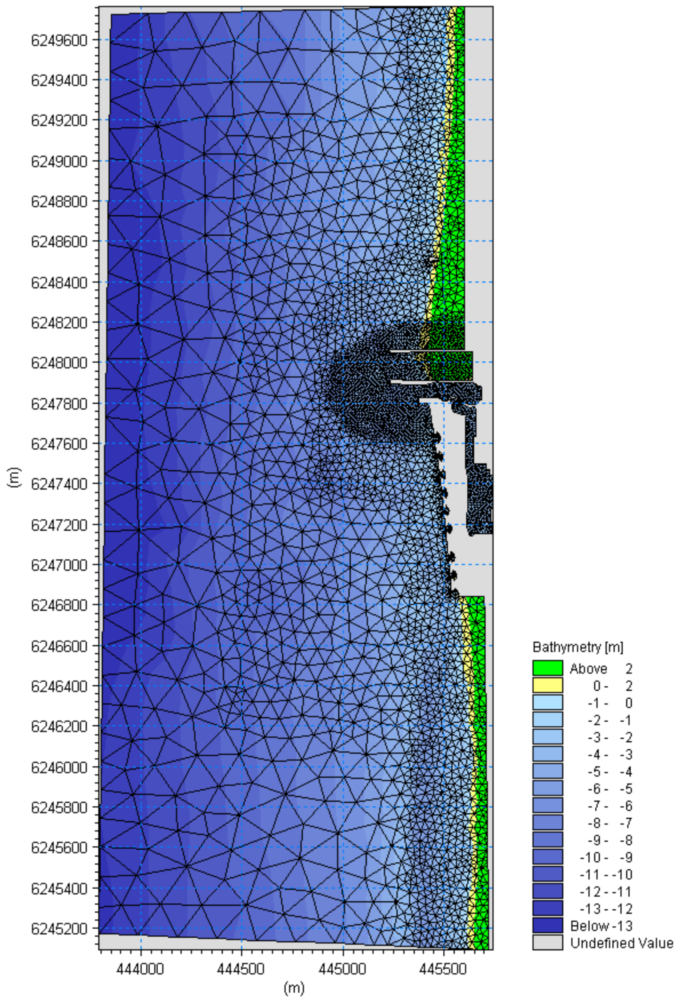

Figure 2.1 The Torsminde area computational mesh and bathymetry as it can be presented with the Data Viewer tool

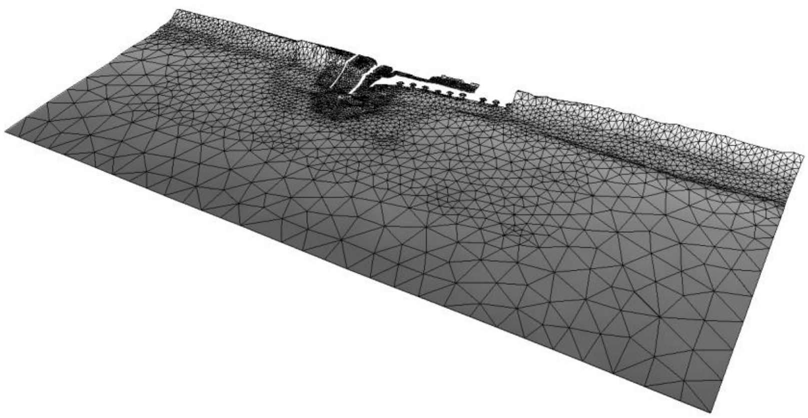

Figure 2.2 The Torsminde area computational mesh and bathymetry as it can be presented with the MIKE Animator tool

3 Creating Various Input Parameters to the MIKE 21/3 Coupled Model FM

Before the MIKE 21/3 Coupled Model FM complex can be set up, the input data must be generated from the measurements.

The following measurement data are used:

1. Water levels at the northern jetty in the period from 1997 to 1999
2. Wave model results from the North Sea
3. Measured pre- and post storm bathymetric survey data in the harbour entrance. The surveys were carried out on 16 October 1997 and 27 October 1997, respectively
4. Measured survey lines along the west coast from 1998
5. Measured local survey lines from 1999

The bathymetric survey data in the harbour entrance form the basis of calibrating the ST model. Thus it is chosen to start the simulation period at 16/10 1997 00:00 and end it at 27/10 1997 00:00:00 (i.e. 11 days).

The measured data is included in the MIKE Zero Examples folder:

.\MIKE_21-3_Integrated_Models\CoupledModel_FM\Torsminde\data

Preparation of input data is made by using various tools in MIKE Zero. Therefore reference is also made to the MIKE Zero User Guide, which can be accessed via the MIKE Zero Documentation Index in the start menu:

MIKE Zero Pre- and Postprocessing, Generic Editors and Viewers, User Guide

3.1 Water Level Boundary Conditions

Measured water level recordings from Torsminde harbour are available.

The Torsminde model is forced with water level boundaries and wave radiation stress. Unfortunately there are no measurements of water levels at the boundary, so the correct variation along the boundary is not known. Therefore the boundary is specified as a time series (dfs0 type file) constant along the boundary. Because of the open area and relatively small domain of the model, the water levels measured at Torsminde harbour are selected as boundary conditions for the entire model.

Please note that the time series must have equidistant time steps. That means that if the raw data have time gaps without measurements, the gaps in the raw data must be filled before importing it, or the time series must be interpolated from a non-equidistant time series after importing it. In this case we will do the latter.

3.1.1 Importing measures water levels to time series file

The file 'Torsminde_levels.txt' containing measured water levels can be found in the folder:

.\MIKE_21-3_Integrated_Models\CoupledModel_FM\Torsminde\Data\Boundary_Conditions

Note that the measured values are stored in unit cm, see Figure 3.1.

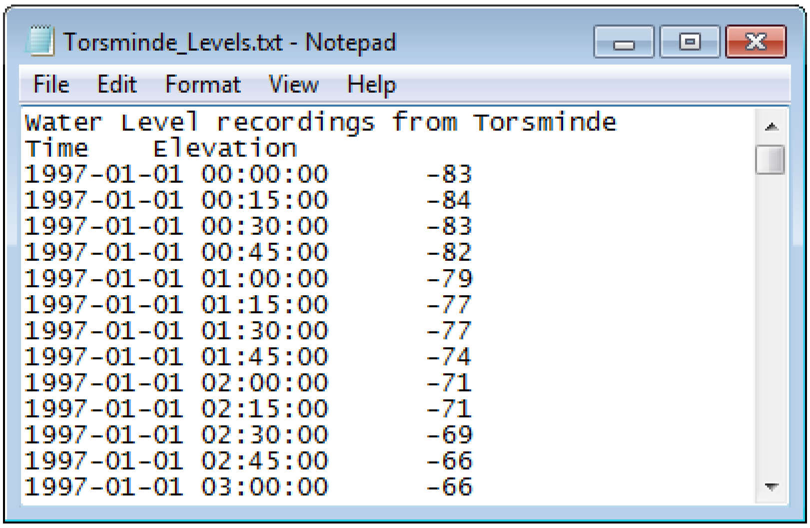

Figure 3.1 ASCII file with water level recordings from Torsminde

Open the Time Series Editor in MIKE Zero (File→New→File), see Figure 3.2.

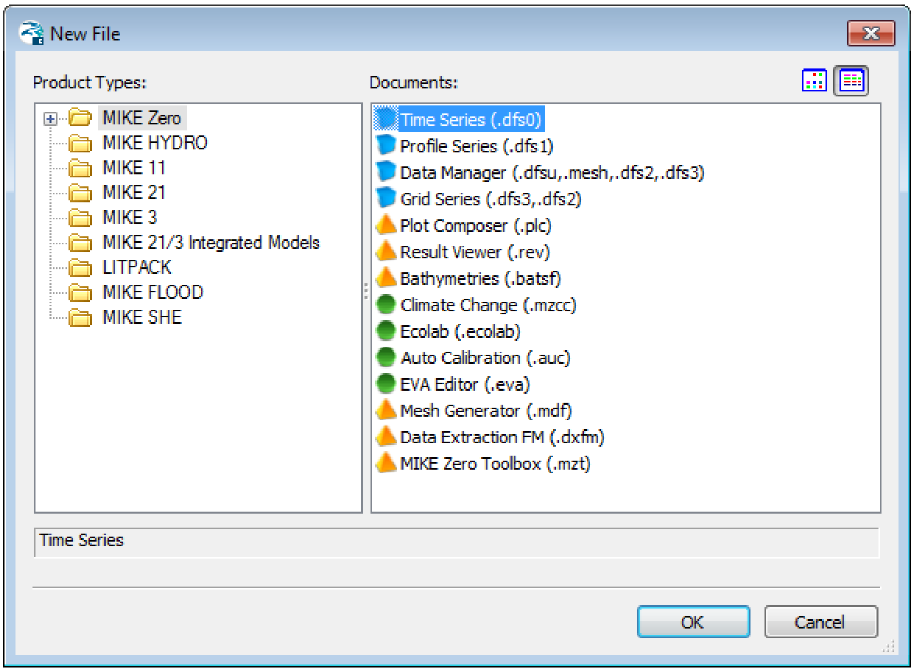

Figure 3.2 Starting the Time Series Editor in MIKE Zero

Select the ASCII template. Open the text file 'Torsminde_levels.txt'. Change the time description to 'Non-Equidistant Calendar Axis' – uncheck the 'Delimiter between time and first item' box and press OK, see Figure 3.3. Then right click on the generated data and select properties change the type to 'Water Level', see Figure 3.4.

Save the data as 'WaterLevels_NonEq.dfs0'.

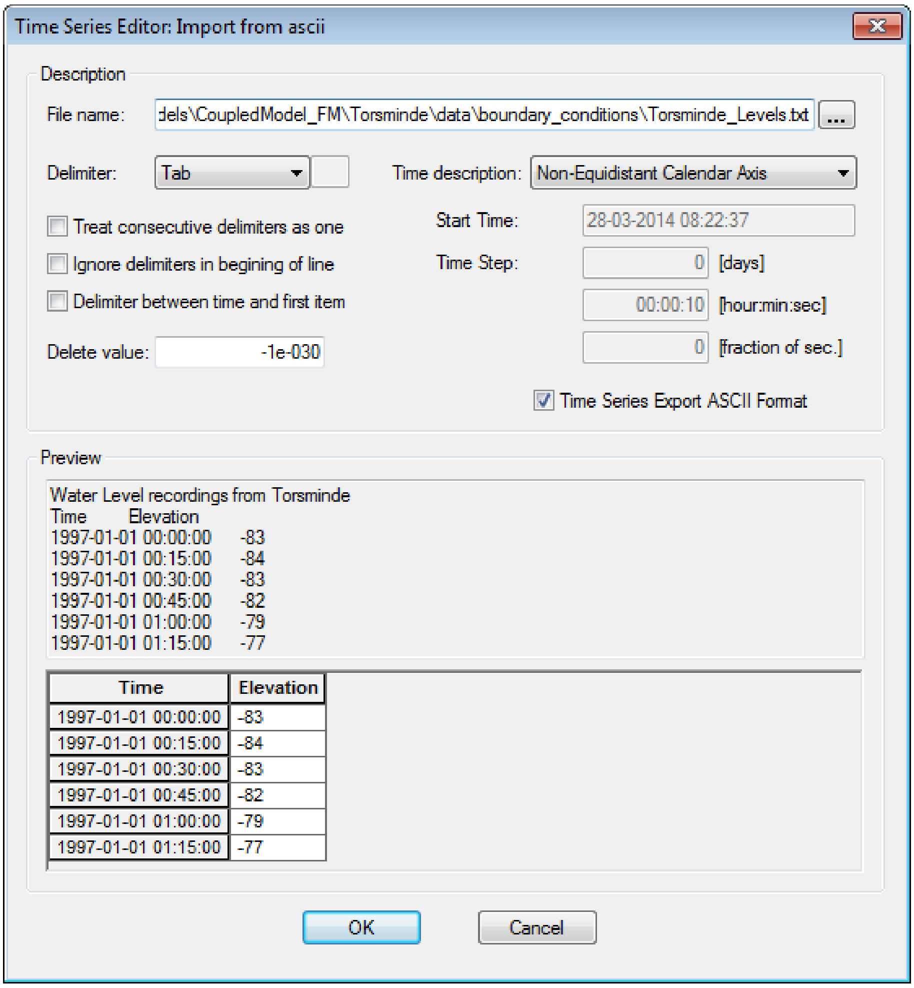

Figure 3.3 Time Series Editor Import Template

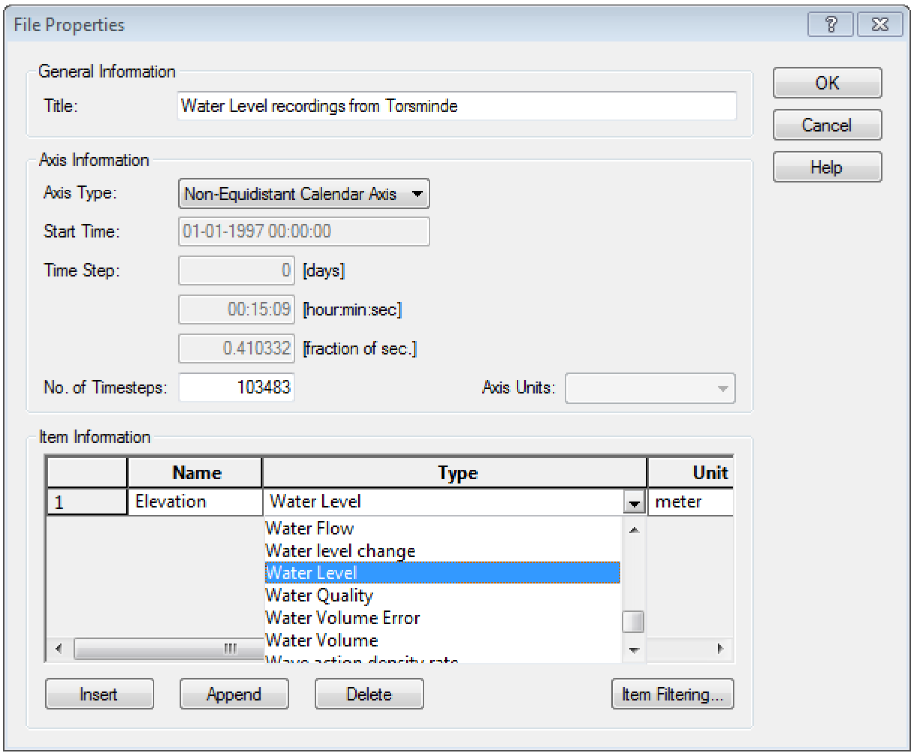

Figure 3.4 Time Series Properties

3.1.2 Create equidistant time axis

The time series file created has a number of gaps in the time series. For example, on 1/7 1997 the recordings stop at 17:30 and are first resumed at 09:00 the next day.

To obtain an equidistant time series for the simulation you should use the MIKE Zero Tool *Interpolate Time Series*.

Open the MIKE Zero Toolbox in MIKE Zero (File→New→MIKE Zero Toolbox). Select *Interpolate Time Series* in the *Time Series* group, see Figure 3.5.

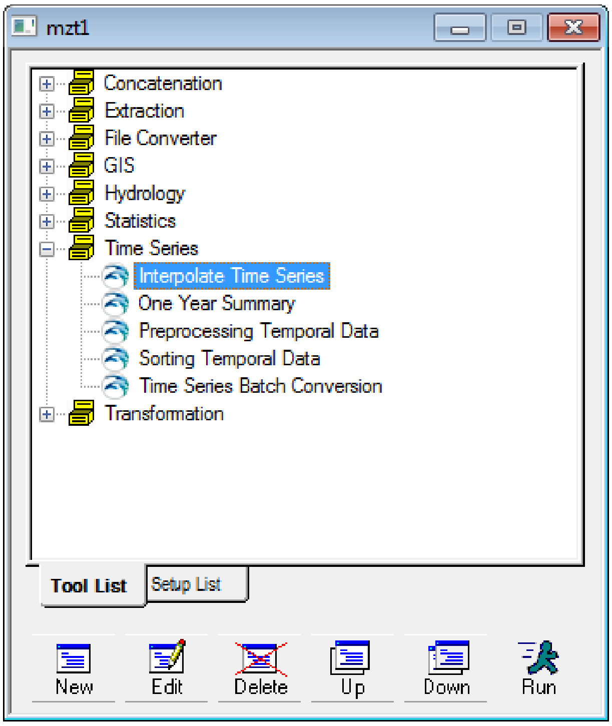

Figure 3.5 Selecting the Interpolate Time Series tool in MIKE Zero Toolbox

Press the *New* button and follow the dialogues.

Select the file 'WaterLevels_NonEq.dfs0' as the input file. In the next dialogue box, please define the interpolation parameters as shown in Figure 3.6. The time step is set to 900 s (i.e. 15 minutes).

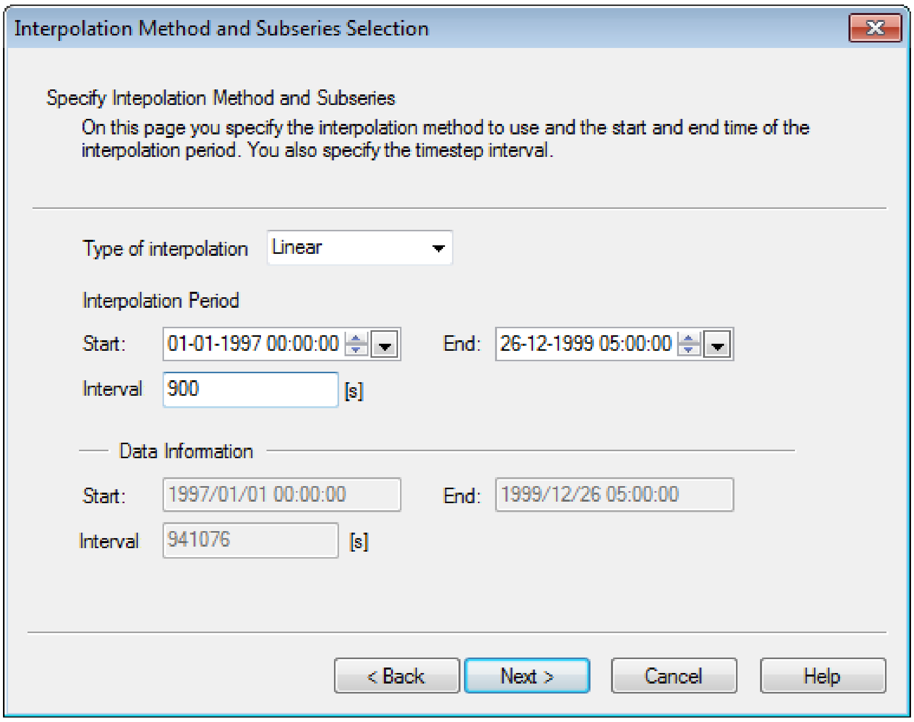

Figure 3.6 Specifying interpolation parameters in Interpolate Time Series tool

Save the data as 'WaterLevels_Eq.dfs0'. Press execute and finish to interpolate the data.

Open the file in the *Time Series Editor* and use the Calculator to multiply the data values with 0.01 to specify the data as unit metres, see Figure 3.7.

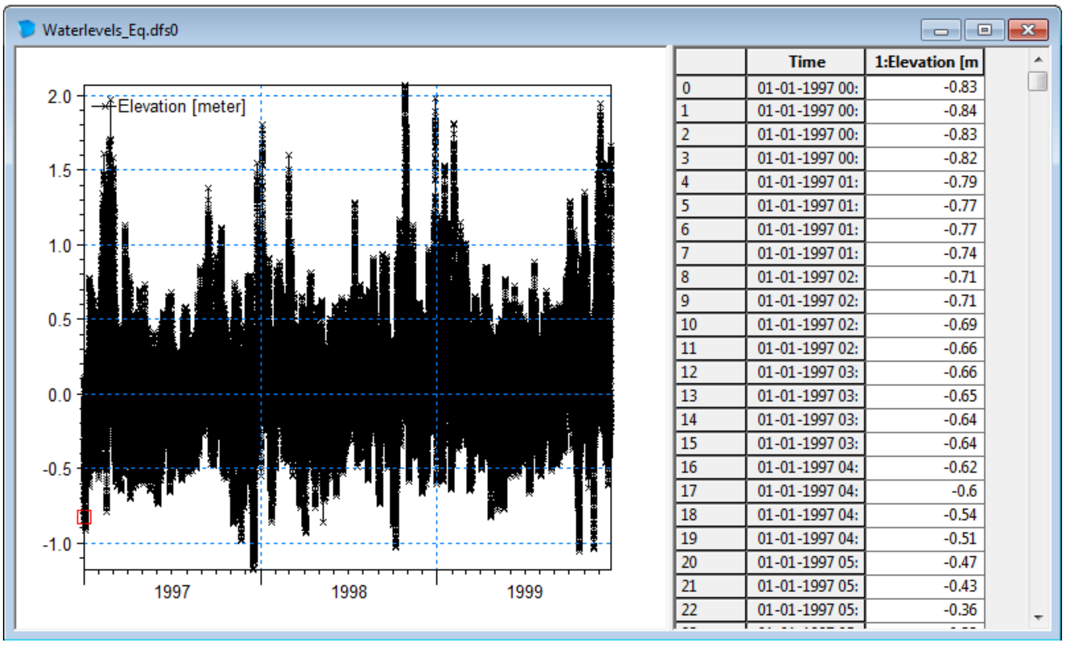

Figure 3.7 Time Series Editor with imported Water Levels from Torsminde

3.1.3 Filtering time series

To obtain a smooth variation of water levels, the measured water level recordings from Torsminde must be low pass filtered. If you do not filter the time series, the numerical simulation may become unstable.

To filter the time series use the MIKE 21 Tool *FFT Filtering of Time Series*.

Open the MIKE 21 Toolbox in MIKE Zero (File→New→File), see Figure 3.8.

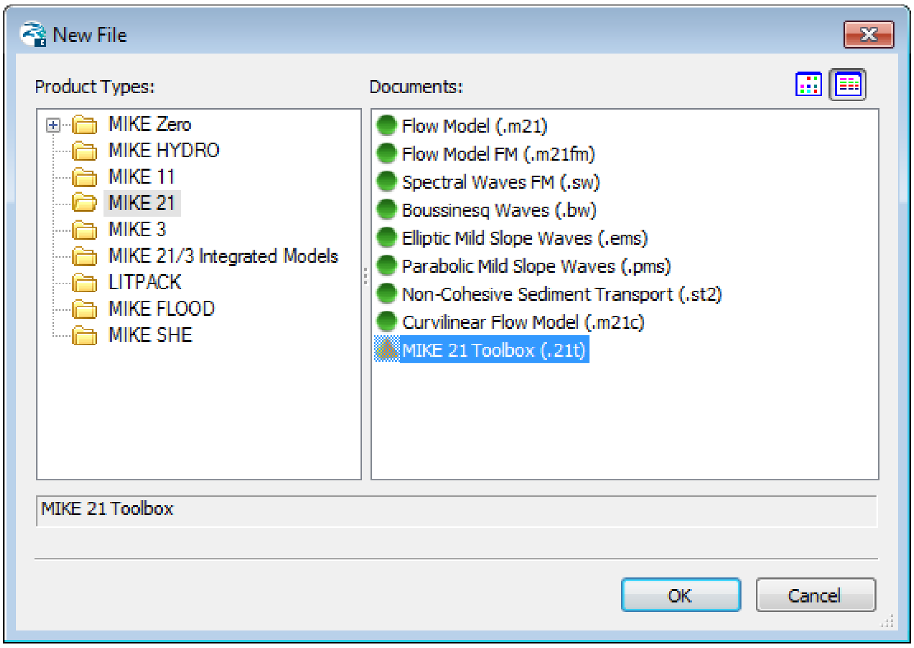

Figure 3.8 Selecting the MIKE 21 Toolbox in MIKE Zero

Select the tool *FFT Filtering of Time series* in the *Waves* group, see Figure 3.9.

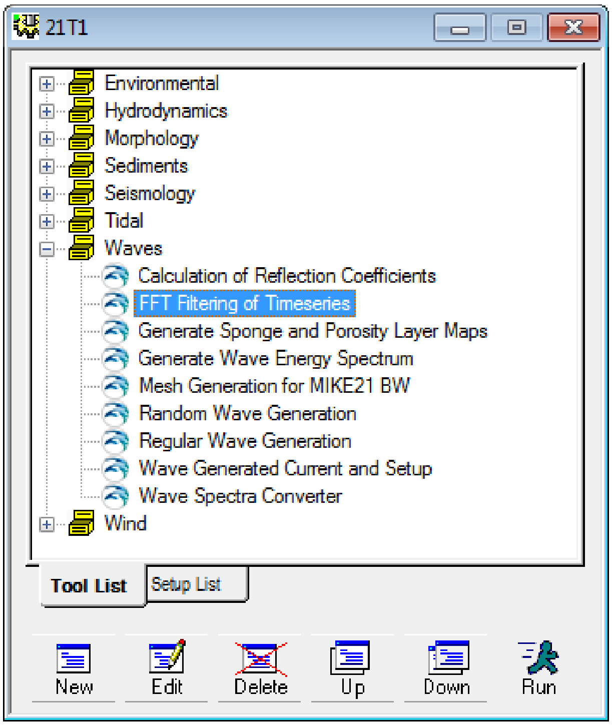

Figure 3.9 Selecting the FFT Filtering of Time Series tool in MIKE 21 Toolbox

Press the *New* button and follow the dialogues. Select the file 'WaterLevels_Eq.dfs0' as the input file. In the next dialogue box, define the frequency range as shown in Figure 3.10. The maximum frequency of interest is set to 5e-005 Hz, corresponding to 5.5 hours.

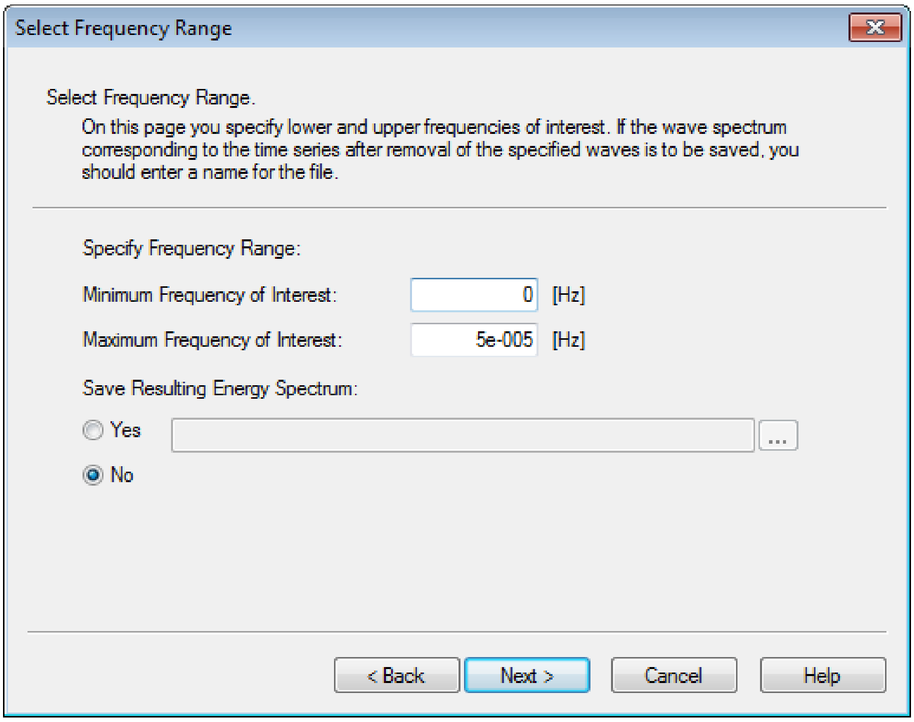

Figure 3.10 Specifying frequency range in FFT Filtering of Time Series tool

Save the data as 'Waterlevels_Torsminde.dfs0'[1]. Press ‘Execute’ and then finish.

Now you are ready to use this file as boundary conditions for the flow simulation.

3.1.4 Plot time series file

To make a plot of the water level time series, please open the Plot Composer in MIKE Zero, see Figure 3.11.

[1] Please note that data files supplied with the installation are write-protected. This means that these files must be renamed or saved to a different folder.

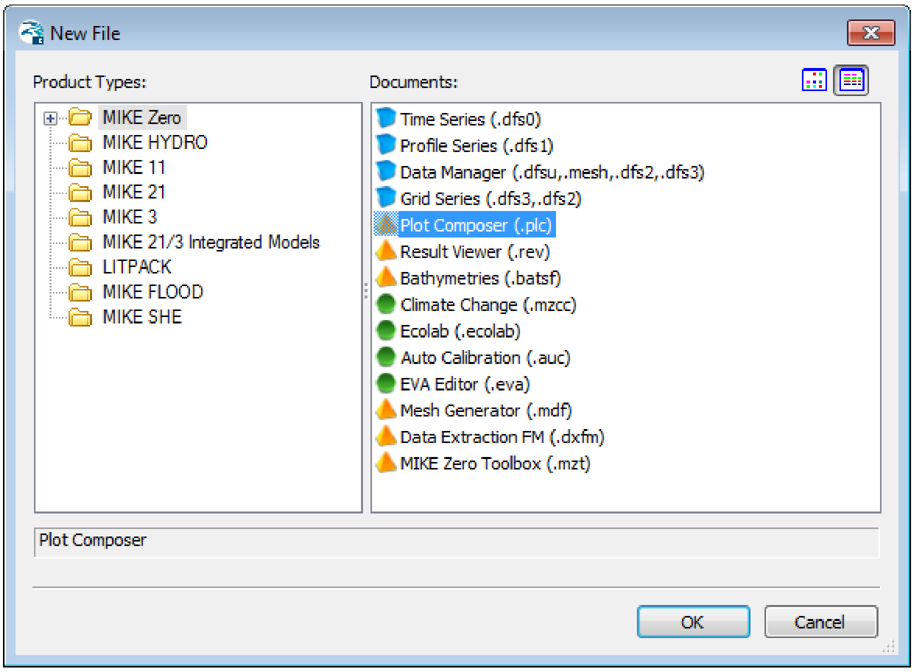

Figure 3.11 Starting the Plot Composer in MIKE Zero

Select 'plot' → 'insert a new plot object' and select 'Time Series Plot', see Figure 3.12. Right click on the plot area and select properties, if the GUI is not opened automatically.

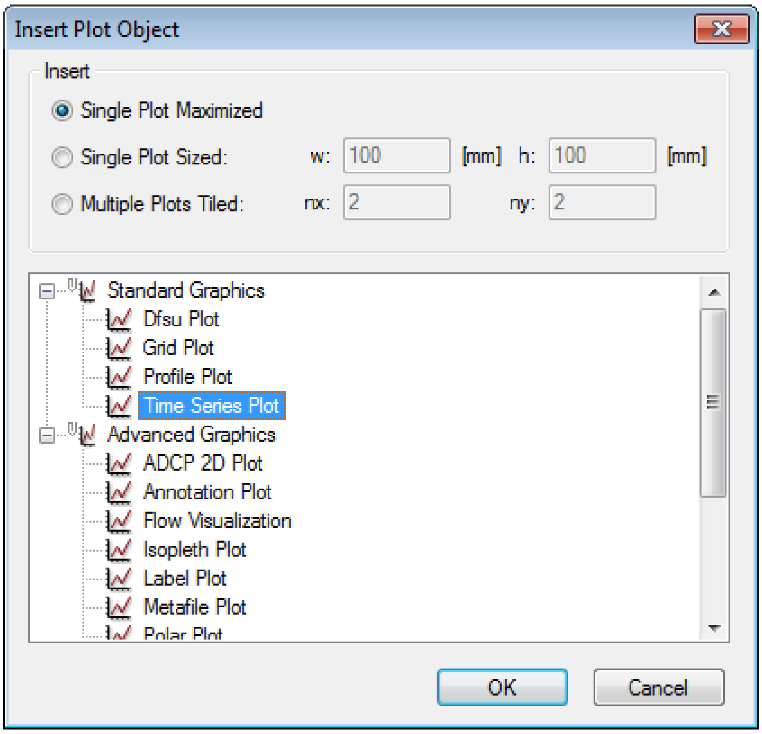

Figure 3.12 Insert a new Plot Object as Time Series in Plot Composer

Add the actual time series file to the Plot Composer by clicking [icon] and selecting the file. It is possible to add more than one time series to the same plot. In the 'Time Series Plot

Properties' dialogue it is possible to change some of the properties for the plot such a item names, colours of curves, etc. (see Figure 3.13).

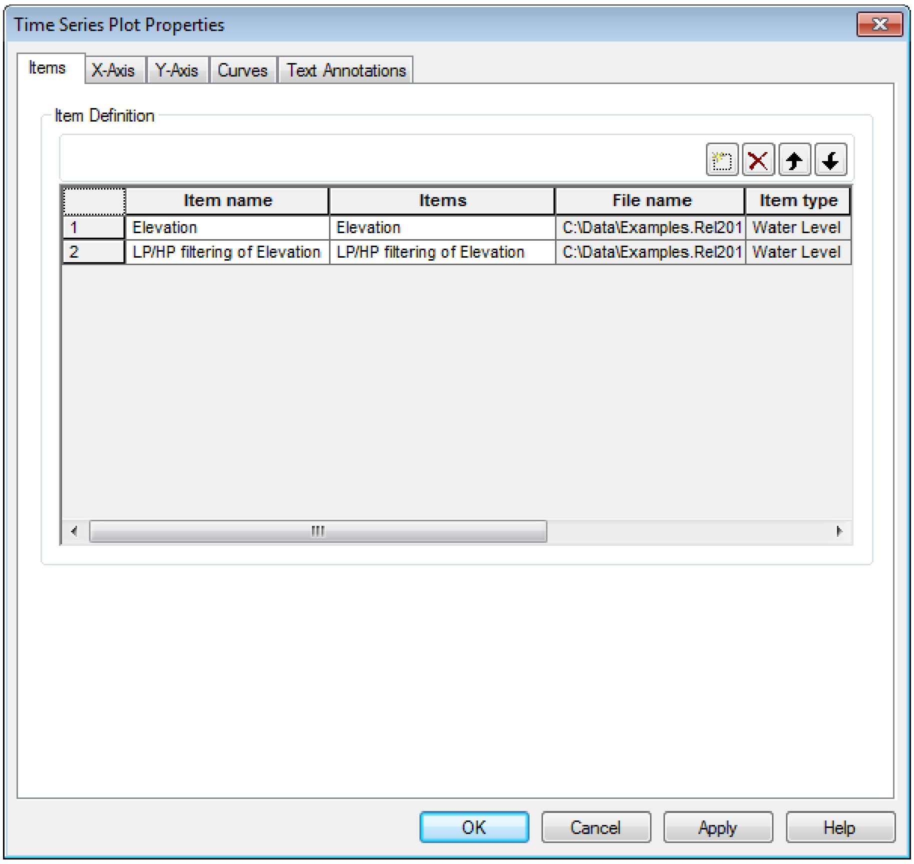

Figure 3.13 Plot Composer Time Series Plot Properties dialogue for selecting time series files and adjusting scales, curves, etc.

The resulting plot of the water level variation at the boundary during the simulation is shown in Figure 3.14.

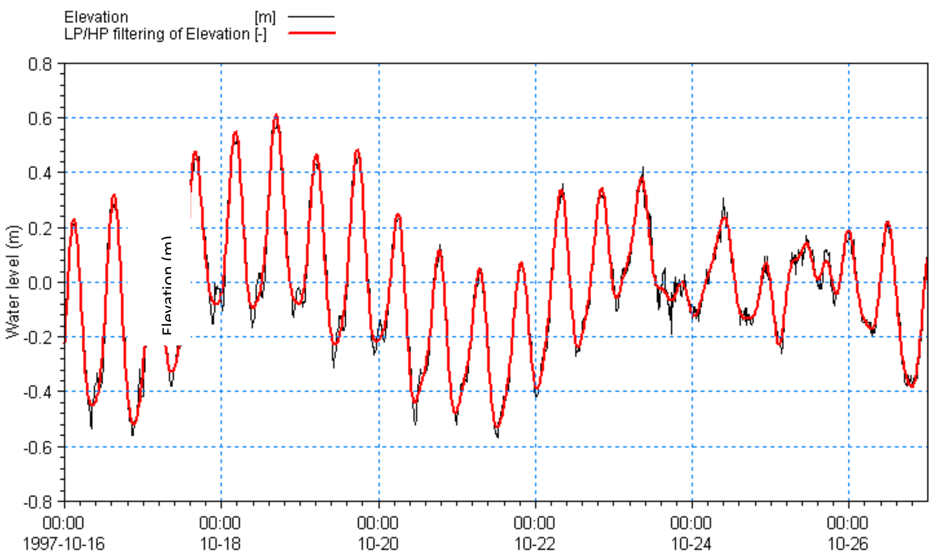

Figure 3.14 Water level variation in time, before and after low pass filtering

3.2 Wave Boundary Conditions

Wave conditions are available from a regional model[2]. Because of the open area and relatively small domain of the present model the wave conditions are set as constants along the offshore (west) boundary.

The north and south boundaries are selected to be lateral, as these are far away from the area in focus.

Figure 3.15 shows the wave conditions during the simulation period.

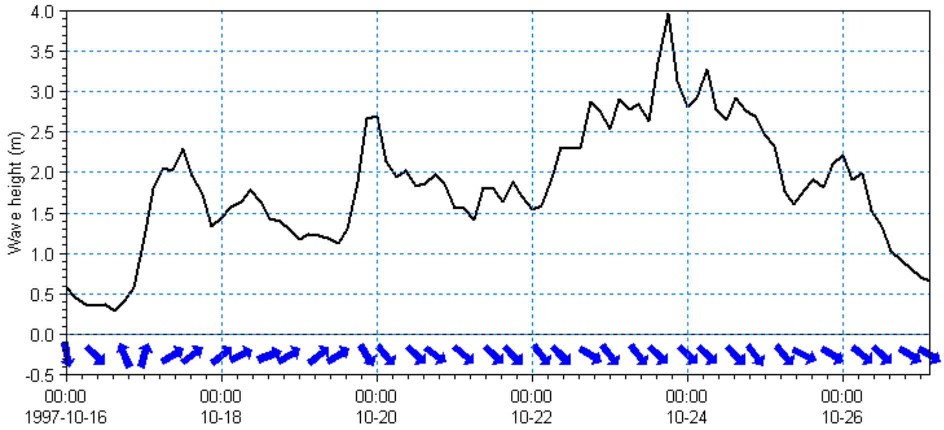

Figure 3.15 Time series of significant wave height and mean wave direction

In the beginning of the period there are fairly small waves from SW. After almost 4 days the wave direction shifts to NW and the wave height increases.

3.3 Initial Conditions (Hydrodynamic Module)

The initial surface level is estimated to be 0.0 m over the whole domain. Using the soft start facility the water level will adapt to the measured water level in Torsminde harbour within an hour.

3.4 Wind Forcing (Hydrodynamic Module)

Wind recordings from Torsminde Harbour will form the wind forcing as time series constant in space.

The wind recordings from Torsminde harbour have been loaded into the Time Series Editor and saved in 3-hour intervals in the file Wind_Torsminde_3h.dfs0.

Time series of the wind speed and direction are shown in Figure 3.16.

---

2 See e.g. www.waterforecast.com

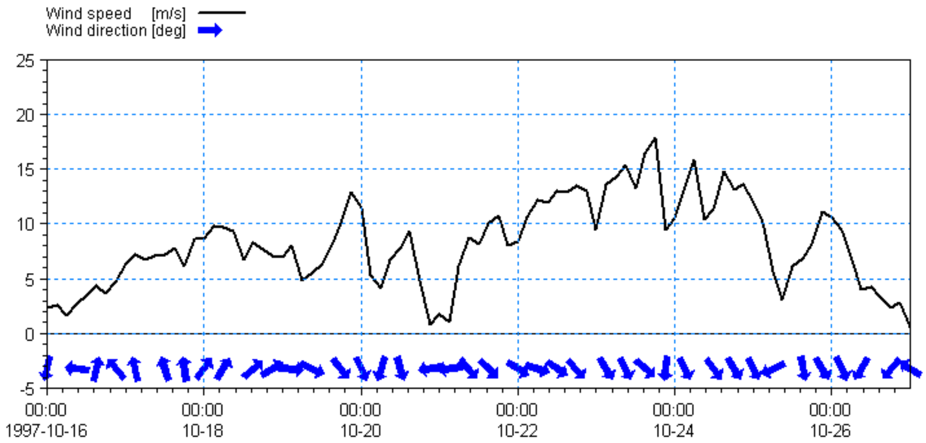

Figure 3.16 Wind speed and direction from Torsminde Harbour during the simulation period

A more descriptive presentation of the wind can be given as a wind speed (or wind rose) diagram. Figure 3.17 shows the wind rose for Torsminde Harbour.

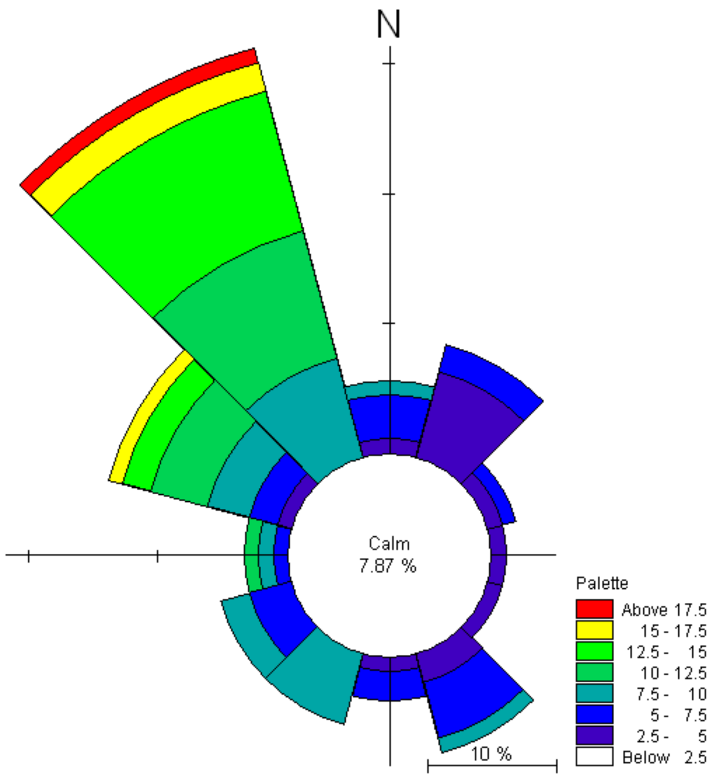

Figure 3.17 Wind rose from Torsminde Harbour for the simulation period as it can be illustrated in the Plot Composer. The dominating wind direction is northwest

3.5 Sediment Data

3.5.1 Sediment properties

The sediment properties are assumed to be constant over the area. The median grain size is 0.3 mm and the sediments are well sorted with a spreading of 1.45.

3.5.2 Generating sediment transport tables

Prior to execution of the Sand Transport Module, quasi-3D sediment transport look-up tables have to be generated. The extension of the tables must be such that the possible parameters from the simulation are covered.

The necessary extension of the transport tables is not known a priori the simulation. However, an educated guess can be made on basis of the boundary conditions, information about the area and previous experience.

To generate a transport table, please open the *MIKE 21 Toolbox* in MIKE Zero, select the *Generation of Q3D Sediment Tables* and press *New*, see Figure 3.18 and Figure 3.19.

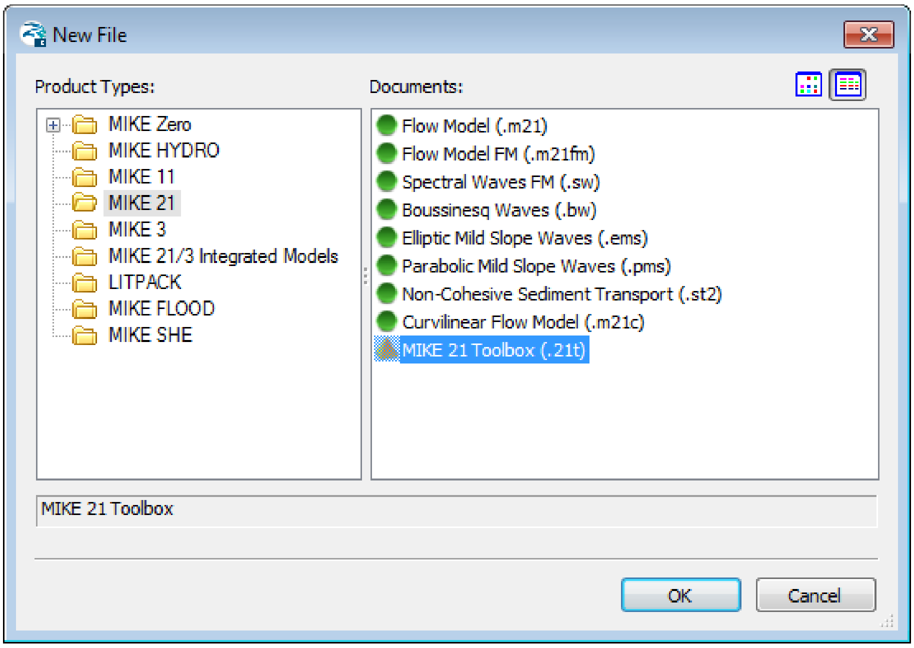

Figure 3.18 Starting the MIKE 21 Toolbox in MIKE Zero

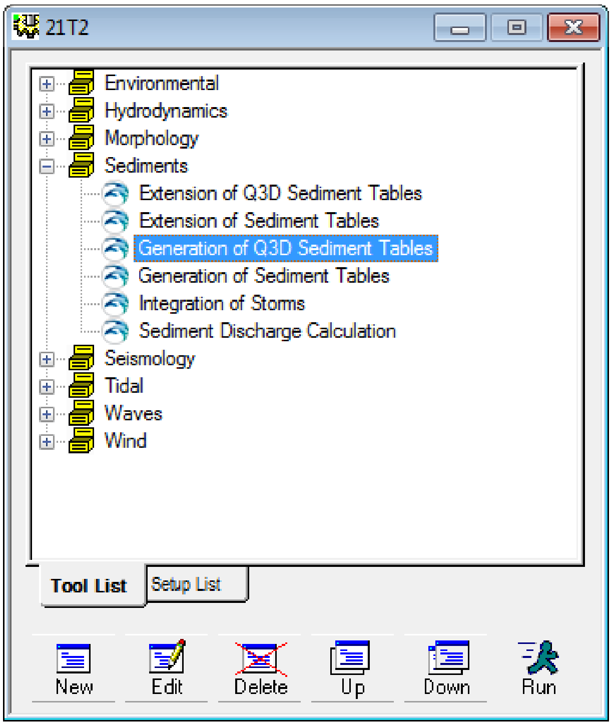

Figure 3.19 Selecting tool for generation of transport tables

Follow the tutorial, rename the setup and continue to the dialogue containing General Parameters and Additional Parameters. Press next for each to accept the default values. In the 'Wave Parameters' dialogue change the wave theory to Stokes 1st order, see Figure 3.20.

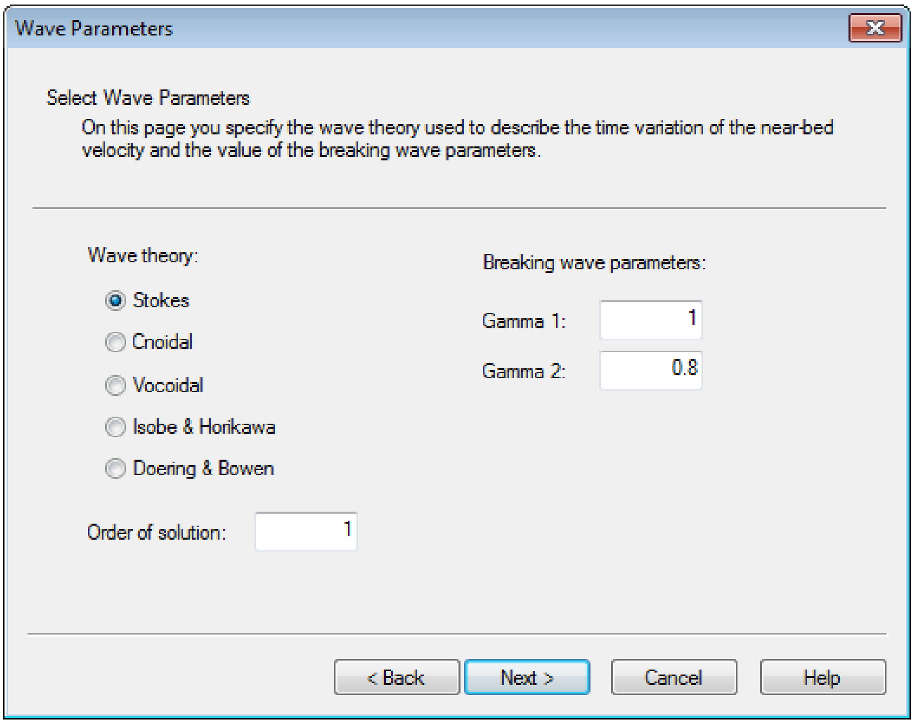

Figure 3.20 Selecting wave theory for transport tables

In the 'Sediment Transport Tables Axes' dialogue, the extension of the tables is defined, see Figure 3.21.

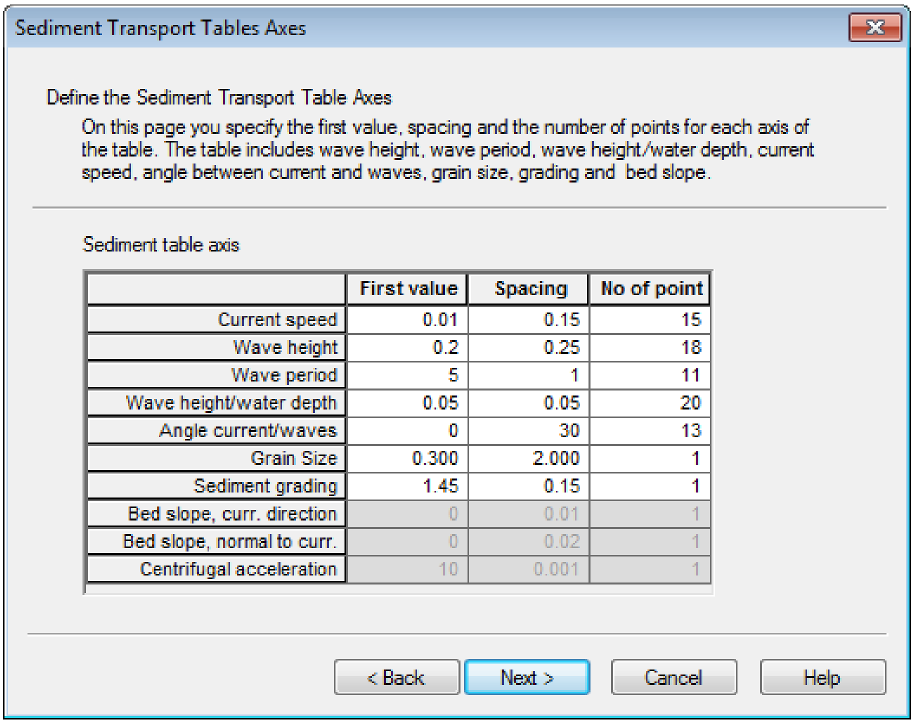

Figure 3.21 Defining the extension of the transport tables

Insert the values given in Figure 3.21, press next and continue to give the sediment transport tables a descriptive title. Go on with saving the setup and press 'Execute' to the tool in order to generate the sediment transport tables. Two ASCII files will be generated with the extensions .lon and .crs, respectively.

As generating sediment transport tables is computationally demanding, sediment transport tables for this example have been provided with the setup.

4 Setup of MIKE 21/3 Coupled Model FM

4.1 Specifications

We are now ready to set up the model using the Torsminde mesh with the pre-storm water depths, boundary conditions and forcings as described in Chapter 3.

In Table 4.1, the parameters applied for the simulation are listed in short.

Table 4.1 Specifications for the calibration simulation

Parameter	Value
Specification file	Sim1.mfm
Mesh and bathymetry	FineBathy.mesh 4211 Nodes, 7876 elements or CourseBathy.mesh 1997 nodes, 3674 elements
Simulation period	1997-10-16 00:00 – 1997-10-27 00:00 (11 days)
Time step interval	1800 s
No. of time steps	528
Module selection	Hydrodynamics (HD) Sand Transport (ST) Spectral Waves (SW)
HD: Solution technique	Low order, fast algorithm Minimum time step: 0.1 Maximum time step: 1800 Critical CFL number: 1
HD: Flood and dry	Standard flood and dry
HD: Initial conditions	Constant 0.0 m
HD: Wind forcing	Wind_SimulationPeriod.dfs0 soft start: 3600 s wind friction varying with wind speed
HD: Wave radiation	Wave radiation from SW simulation
HD: Water level boundaries North and south West boundary	Waterlevels_Torsminde.dfs0 Include radiation stress correction, soft start 3600 s Exclude radiation stress correction, soft start 3600 s
HD: Eddy viscosity	Smagorinsky formulation, Constant 0.28 m^2/s
HD: Bed resistance	Manning number, FineManning.dfsu or CourseManning.dfsu
HD: Coriolis forcing	None
ST: Model type	Wave and current
ST: Sediment transport table	QsTable.lon / QsTable.crs
ST: Time step factor	2
ST: Sediment properties	Constant, d50=0.3 mm, spreading 1.45
ST: Forcings	Wave field from SW simulation

Parameter	Value
ST: Morphology	Include feedback on hydrodynamic, wave and sand transport calculation
ST: Boundaries	Zero sediment flux gradient for outflow, zero bed change for inflow
SW: Basic equations	Spectral: Directionally decoupled parametric formulation Time: Quasi-stationary formulation
SW: Discretization	Type: directional sector 14 directions with a minimum value of 135 degrees and a maximum value of 45 degrees.
SW: Solution technique	Newton-Raphson iteration Tolerance: $1.0 \cdot 10^{-5}$
SW: Water level conditions	Water level variation from HD simulation
SW: Current conditions	Current variation from HD simulation
SW: Wind forcing	Wind_SimulationPeriod.dfs0
SW: Wave breaking	Constant gamma value: 0.8
SW: Bottom friction	Constant Nikuradse roughness, k_N: 0.01 m Include effect on mean wave frequency
SW: Initial conditions	Type: Zero spectra
SW: North & south boundary	Lateral boundary
SW: West boundary	Varying in time, constant along line File: NSW_timeseries_okt97.dfs0
Result files	HD_results.dfsu, SW_results.dfsu, ST_results.dfsu
Simulation time	About 27 minutes with a 2.2 GHz PC, 8 GB RAM, 4 cores, using the coarse bathymetry About 2.0 hours with a 2.2 GHz PC, 8 GB RAM, 4 cores, using the fine bathymetry

In the following screen dumps, the individual input pages are shown and a short description is provided.

Additional information about the modules can be found in the User Guides listed below, which can be accessed via the MIKE Zero Documentation Index in the start menu:

MIKE 21 Flow Model FM, Hydrodynamic Module, User Guide

MIKE 21 Flow Model FM, Sand Transport Module, User Guide

MIKE 21 SW, Spectral Waves FM Module, User Guide

4.2 Specifications for Basic Parameters

To create a new setup file, open the MIKE Zero shell and select File→New→File, MIKE 21/3 Integrated Models→Coupled Model FM. Specify the bathymetry and the mesh file FineBathy.mesh in the 'Domain' dialogue, see Figure 4.1. The projection zone has already been specified in the mesh generation process as UTM-32. In the mesh file, each boundary has been given a code. In this example the North Boundary has 'Code 2', the West Boundary has 'Code 3' and the South Boundary has 'Code 4'. Rename the boundary 'Code 2' to 'North', 'Code 3' to 'West' and 'Code 4' to 'South' in the 'Boundary Names' dialogue.

Please note that you can see a graphical view of the computational mesh and the boundaries by selecting 'Boundary Conditions' in the Hydrodynamic Module. Here you can also view the code values (second item), see Figure 4.2.

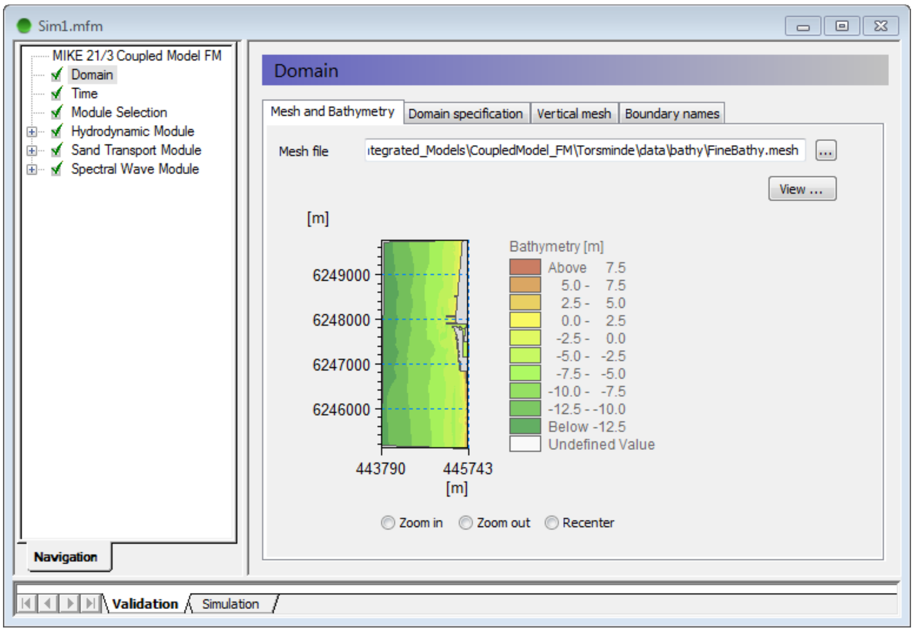

Figure 4.1 MIKE 21/3 Coupled Model FM: Specify Domain

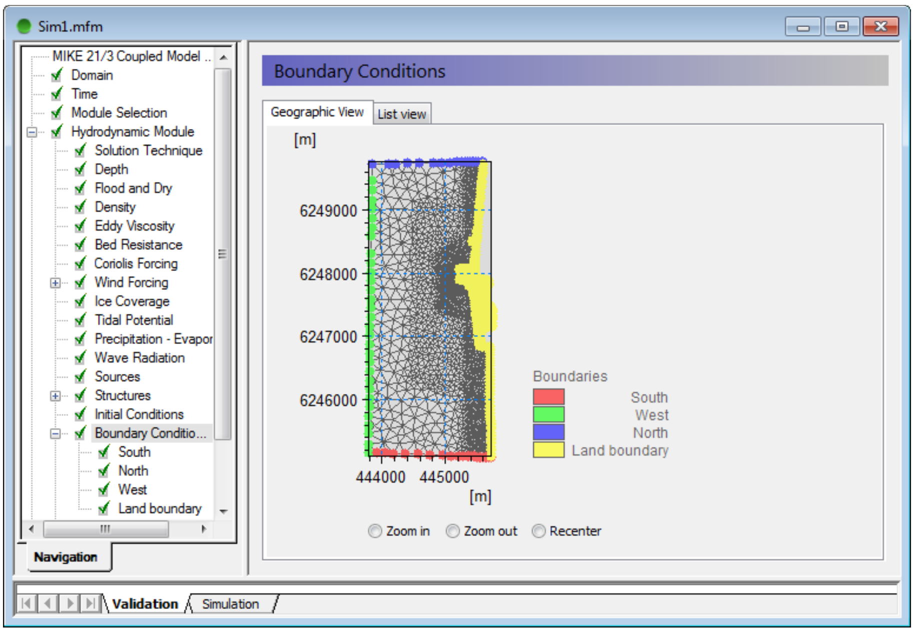

Figure 4.2 Computational mesh and code values of the boundaries

In the MIKE 21/3 Coupled Model FM the modules run simultaneously, allowing feedback of the results between the involved modules at a frequency determined by the overall time step.

Set the start date to 1997-10-16 00:00:00, specify the time interval to 1800 s and the number of time steps to 528 in order to simulate a total period of 11 days, see Figure 4.3.

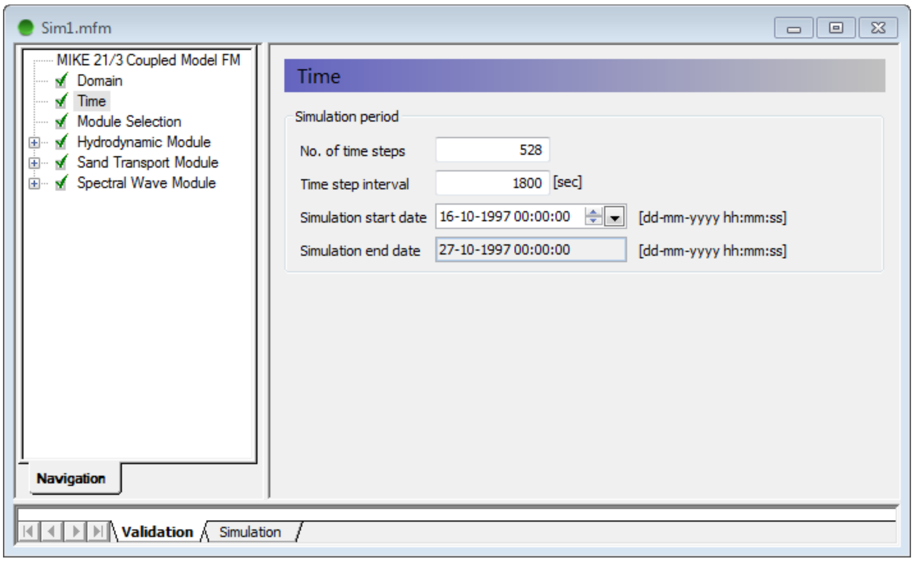

Figure 4.3 MIKE 21/3 Coupled Model FM: Simulation period

For this example include the 'Hydrodynamic', 'Sand Transport' and 'Spectral Waves' modules in the 'Module Selection' dialogue, see Figure 4.4.

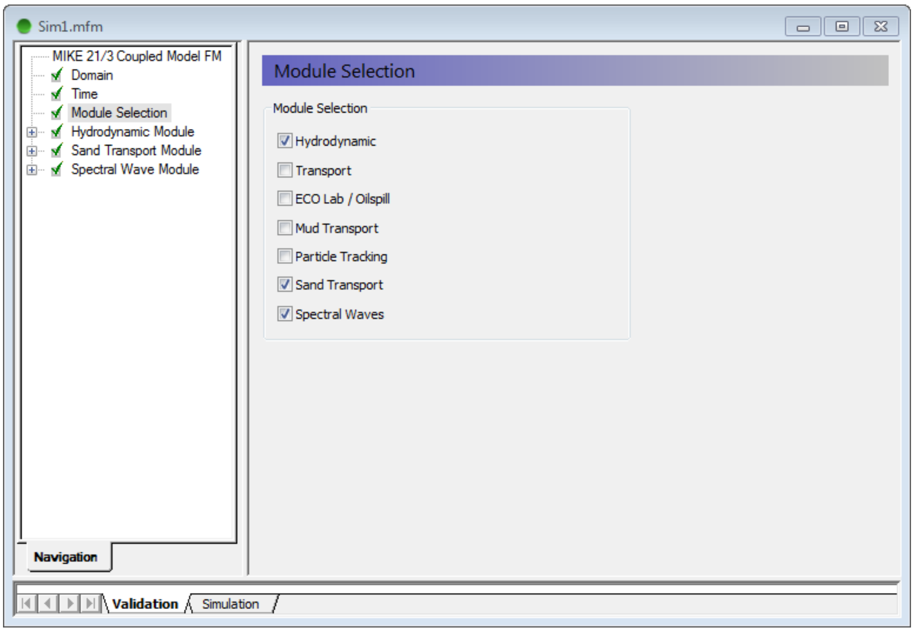

Figure 4.4 MIKE 21/3 Coupled Model FM: Module Selection

4.3 Specifications for the Hydrodynamic Module

The shallow water equations are solved using 'Low order, fast algorithm' for both time integration and space discretization, see Figure 4.5.

The minimum time step is set to 0.1 s to ensure that the CFL number always is less than the critical CFL number of 1. The maximum time step is set to the overall time step of 1800 s.

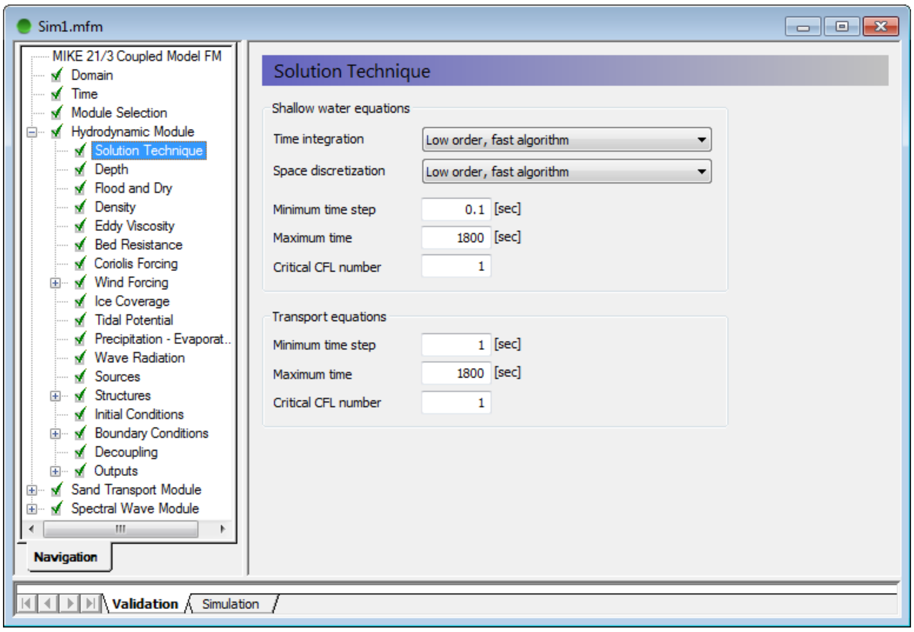

Figure 4.5 MIKE 21/3 Coupled Model FM, Hydrodynamic Module, Solution Technique

In this example the standard flooding and drying should be included (see Figure 4.6), because the tidal variation of the water level will cause part of the beach to dry out and flood during the simulation. If you choose not to include flooding and drying, the model will blow up in situations with dry areas.

In our case select a drying depth of 0.005 m and a flooding depth of 0.05 m. The wetting depth should be 0.1 m. These are default values.

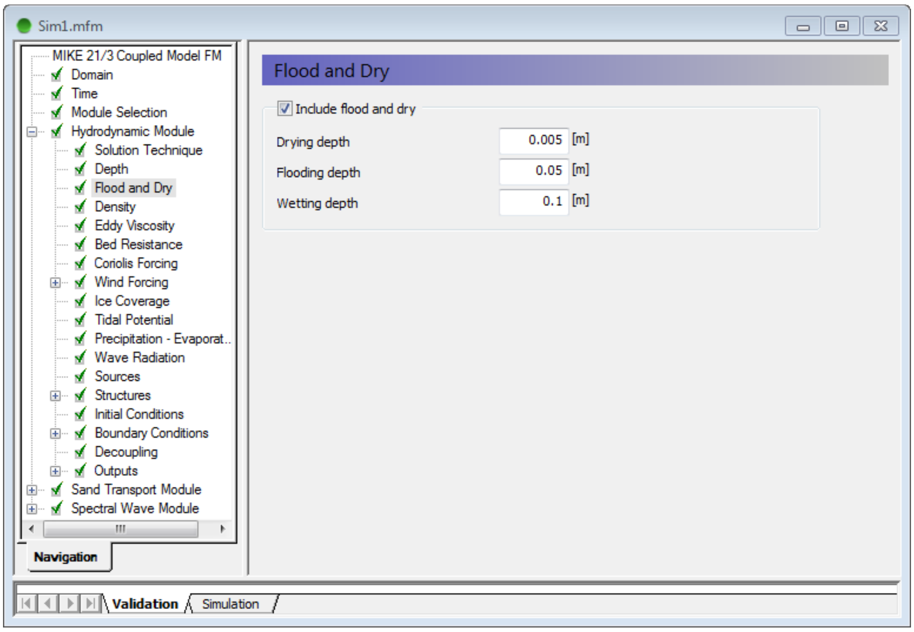

Figure 4.6 MIKE 21/3 Coupled Model FM, Hydrodynamic Module: Flood and Dry

As the density variation is not taken into account in this example, the density should be specified as 'Barotropic', see Figure 4.7.

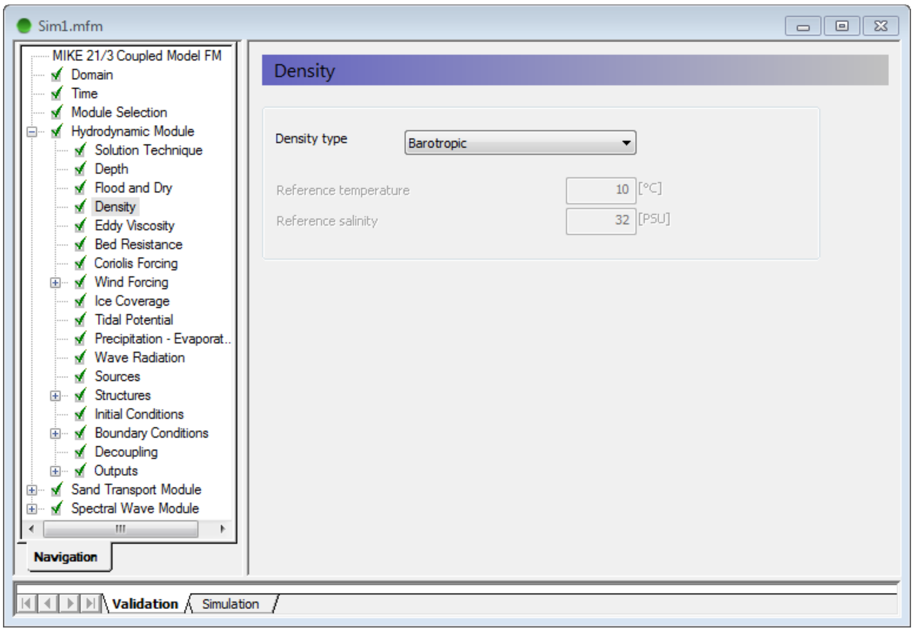

Figure 4.7 MIKE 21/3 Coupled Model FM, Hydrodynamic Module: Density specification

The default setting for the Horizontal Eddy Viscosity is a Smagorinsky formulation with a coefficient of 0.28, see Figure 4.8.

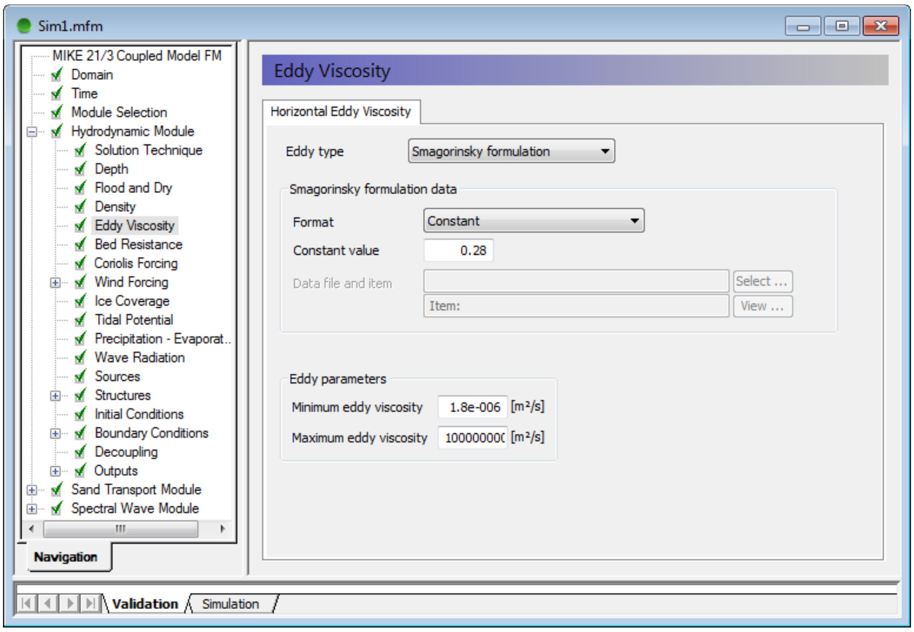

Figure 4.8 MIKE 21/3 Coupled Model FM, Hydrodynamic Module: Eddy Viscosity (Horizontal)

The Bed Resistance will be given as a Manning map, see Figure 4.9. The Manning number is $40^{1/3}$/s in the overall area, but smaller at the south and north boundaries to avoid instabilities during simulation.

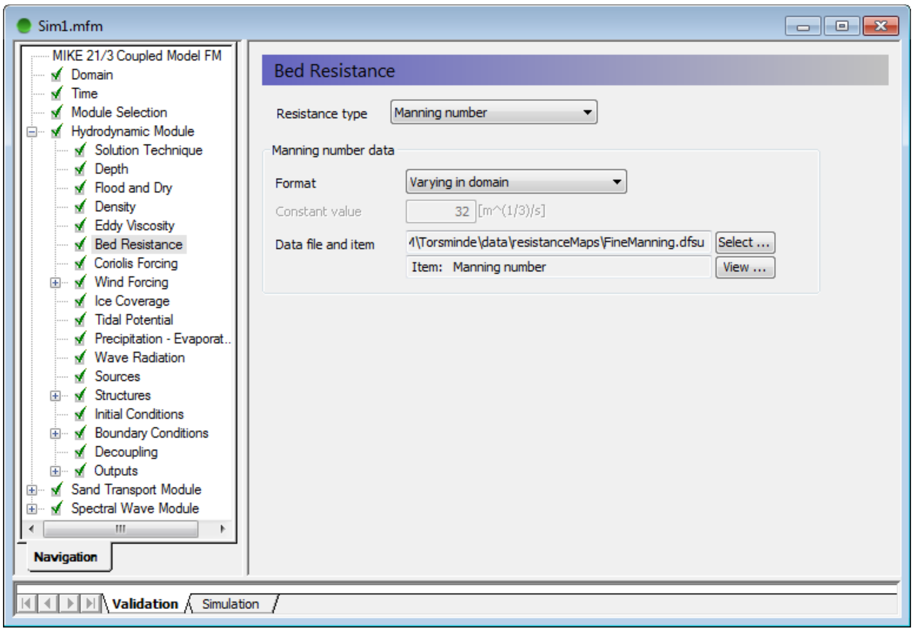

Figure 4.9 MIKE 21/3 Coupled Model FM, Hydrodynamic Module: The Bed Resistance is specified as Varying in domain: FineManning.dfsu

The format must be set as 'Varying in domain' in order to specify the bed resistance by a 2D data file (dfsu or dfs2).

Click Select ... to select the data file in the open file window that appears.

The Wind Forcing will be included as a time series file, see Figure 4.10.

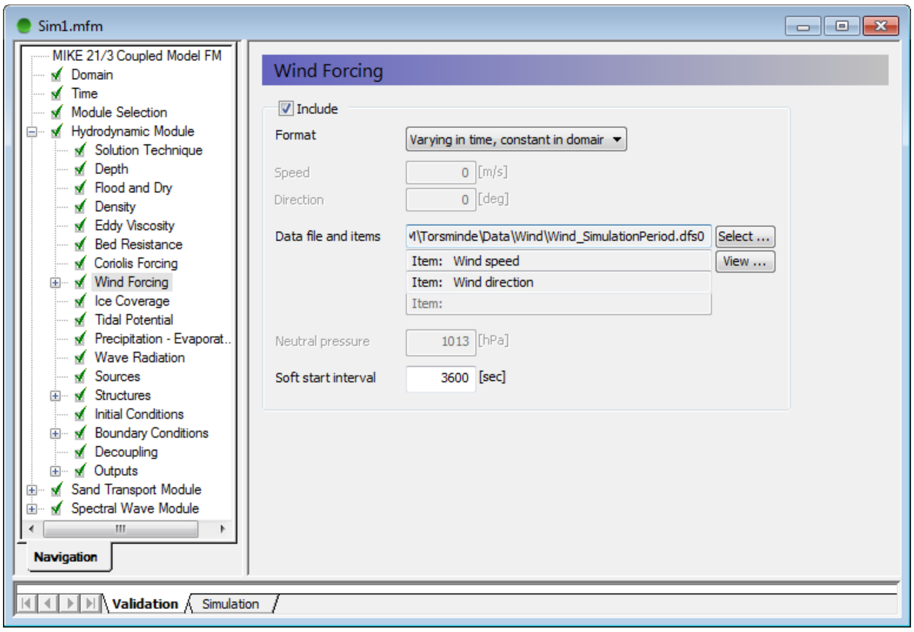

Figure 4.10 MIKE 21/3 Coupled Model FM, Hydrodynamic Module: Wind Forcing

The format must be set as 'Varying in time, constant in domain' in order to specify the wind conditions by a time series file. A soft start of 3600 s is applied.

Click Select ... to select the data file in the Open File window that appears.

The Wind Friction type is set to 'Varying with Wind Speed' with the default values, see Figure 4.11.

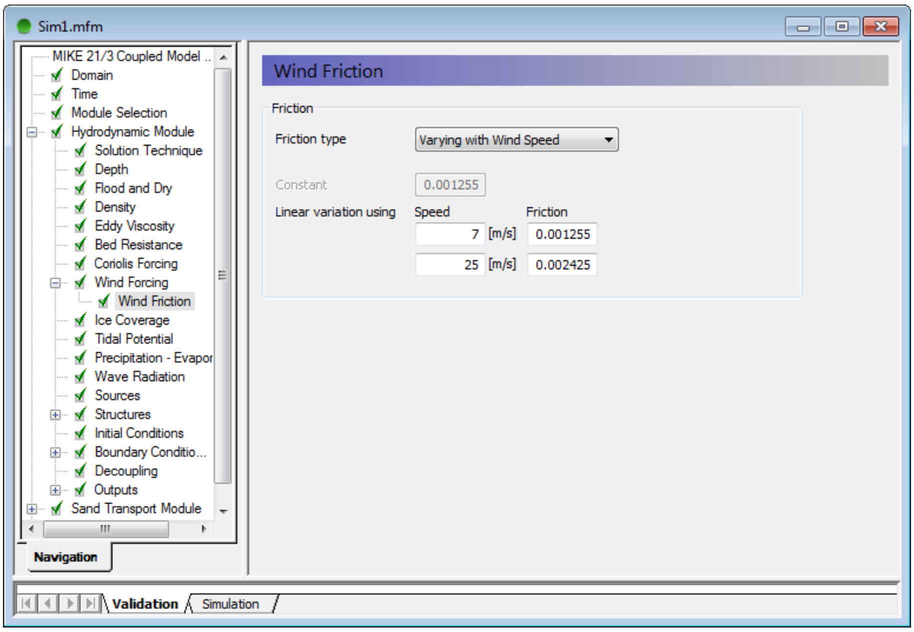

Figure 4.11 MIKE 21/3 Coupled Model FM, Hydrodynamic Module: Wind Friction

Please note that this example **does not** include the parameters of the following subjects:

- Coriolis Force
- Ice Coverage
- Precipitation-Evaporation
- Sources
- Tidal Potential
- Structures
- Decoupling

and the associated dialogue pages are not considered.

The wave radiation should be specified in the 'Wave Radiation' dialogue. In this case the flow, wave and sediment modules are executed in coupled mode, so you should select the type 'Wave radiation from SW simulation', see Figure 4.12.

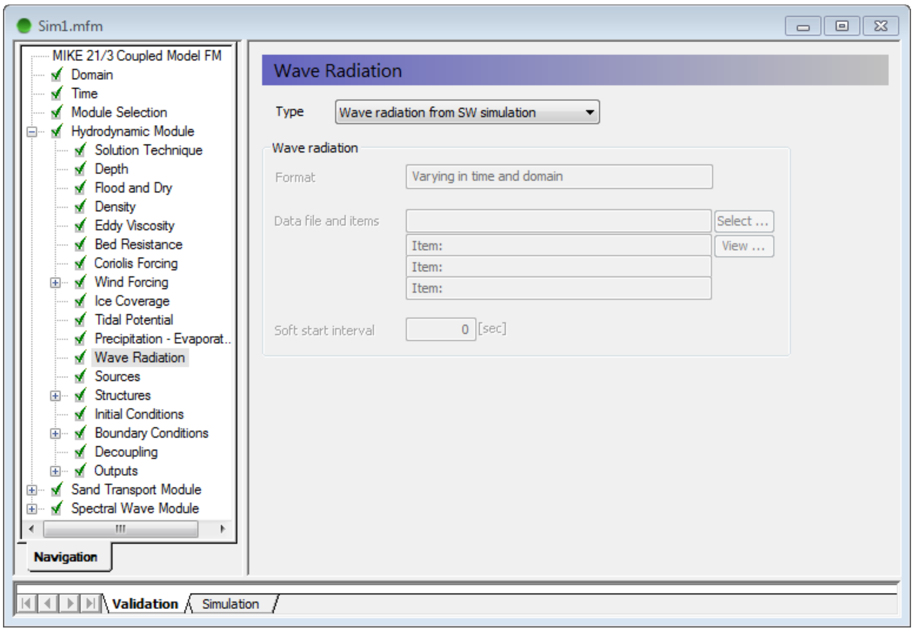

Figure 4.12 MIKE 21/3 Coupled Model FM, Hydrodynamic Module: Wave Radiation

Now you should check the boundary conditions at the simulation start time to estimate the initial surface level. In this case we will use a constant level of 0.0 m, which is the measured value in Torsminde harbour 1 hour after the start of the simulation, see Figure 4.13.

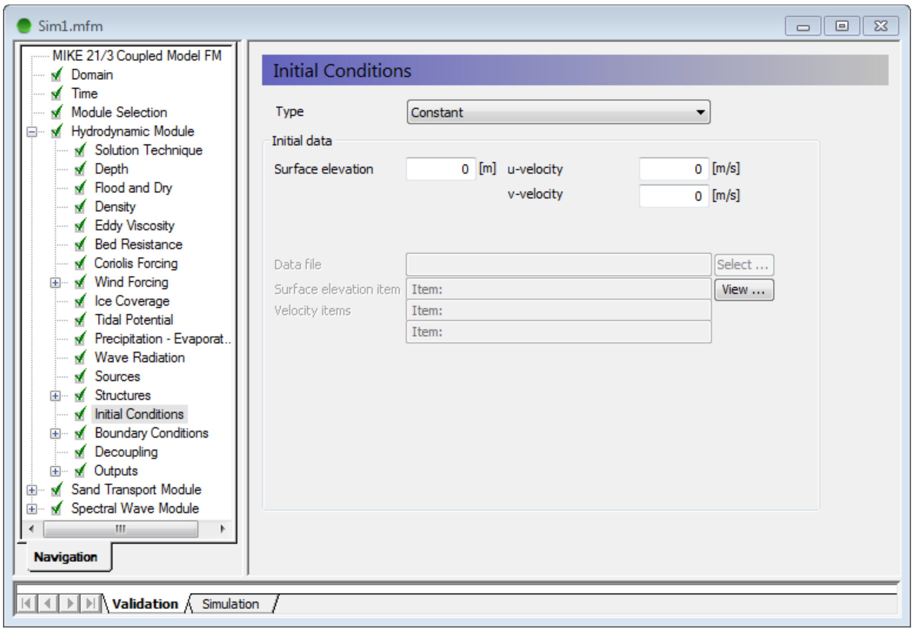

Figure 4.13 MIKE 21/3 Coupled Model FM, Hydrodynamic Module: Initial Conditions

In the 'Boundary Conditions' dialogue, the boundary conditions should be specified for the boundary names, which were specified in the 'Domain' dialogue, see Figure 4.1. There is a North Boundary, a West Boundary and a South Boundary, and the time series that were generated in Chapter 3 should be used on all three boundaries.

In this example the boundary type is 'Specified Level' (Water Level), because only Water Level measurements are available. 'Specified Level' means that the Water Levels are forced at the boundaries and the discharge across the boundary is unknown and estimated during simulation.

The boundary format must be set as 'Varying in time, constant along boundary' in order to specify the boundary as a time series file (dfs0).

Click Select ... to select the data file in the Open File window that appears.

At the North and South boundary you must select the 'Include radiation stress correction'. This automatically changes the water level at the boundary to include the water level setup caused by the breaking waves. There is no reason to this correction for the West Boundary as the wave setup here is negligible.

It is a good practice to use a soft start interval. In our case 3600 s is suggested with a reference value corresponding to the initial value of 0.0 m. The soft start interval is a period in the beginning of a simulation where the effect of the boundary water levels does not take full effect.

In the beginning of the soft start period the effect of the specified Boundary Condition is zero. The effect increases gradually until the boundaries have full effect in the model at the end of the soft start interval period. See Figure 4.14.

Please note that an easy way to see the boundary data file is to simply click View ... in the 'Boundary' dialogue.

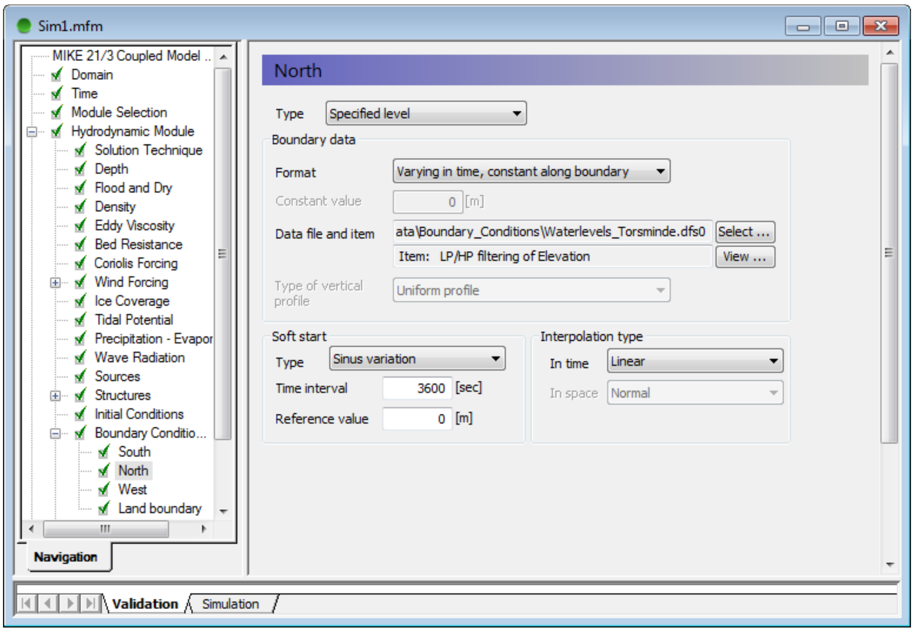

Figure 4.14 MIKE 21/3 Coupled Model FM, Hydrodynamic Module. The Boundary Conditions for the North Boundary are specified as constant along the boundary: Waterlevels_Torsminde.dfs0

Now you may specify one output as an area series (2D) and also specify the resulting output file name, see Figure 4.15. Specify the file name HD_results.dfsu for the first simulation.

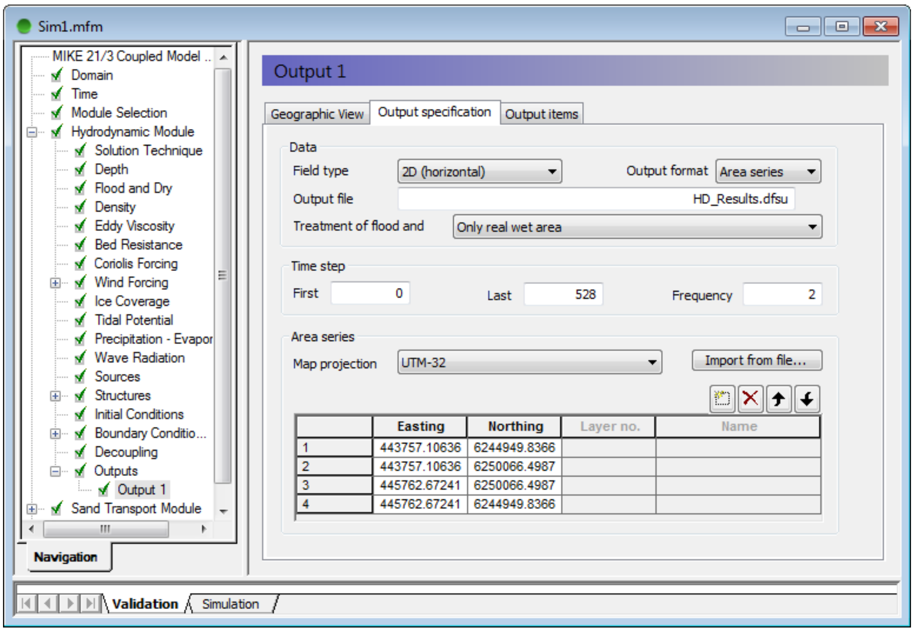

Figure 4.15 MIKE 21/3 Coupled Model FM, Hydrodynamic Module: Output type and location. Results can be specified as point, line or area series

Reduce the output size for the area series to a reasonable amount by selecting an output frequency of 3600 s, which is a reasonable output frequency for a tidal simulation. As our time step is1800 s, the specified output frequency is 3600/1800 = 2. As default, the full area is selected.

Pick the parameters to include in the output file as shown in Figure 4.16.

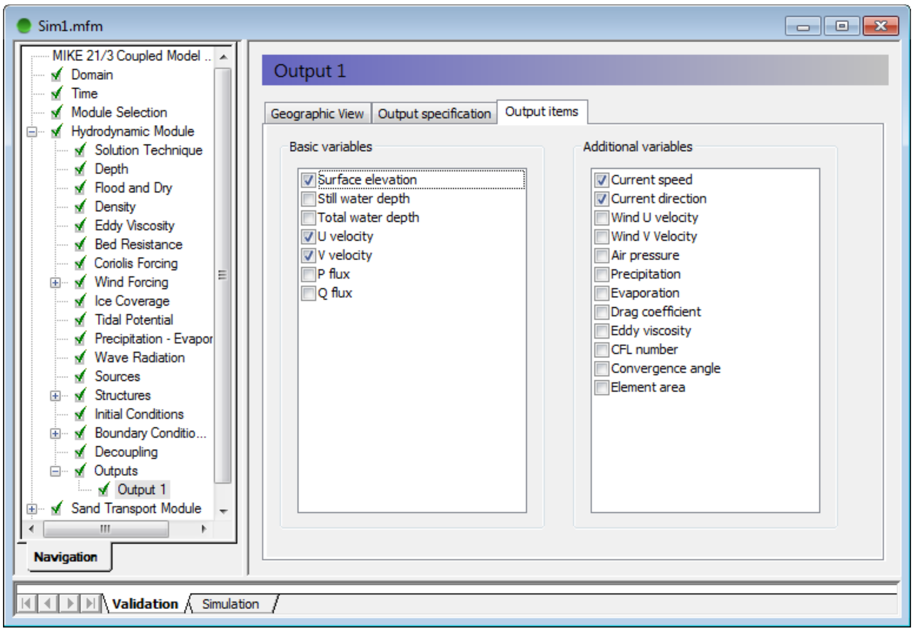

Figure 4.16 MIKE 21/3 Coupled Model FM, Hydrodynamic Module: Output items

4.4 Specifications for the Spectral Wave Module

In this simulation waves derived from a regional numerical model act as boundary conditions for the local wave simulation.

The time formulation is selected as quasi-stationary formulation. This has been chosen because of the limited size of the area. Also the simulation time for the quasi-stationary simulation is much smaller than for the instationary simulation.

The spectral formulation of the wave model is selected as directionally decoupled parametric formulation, see Figure 4.17.

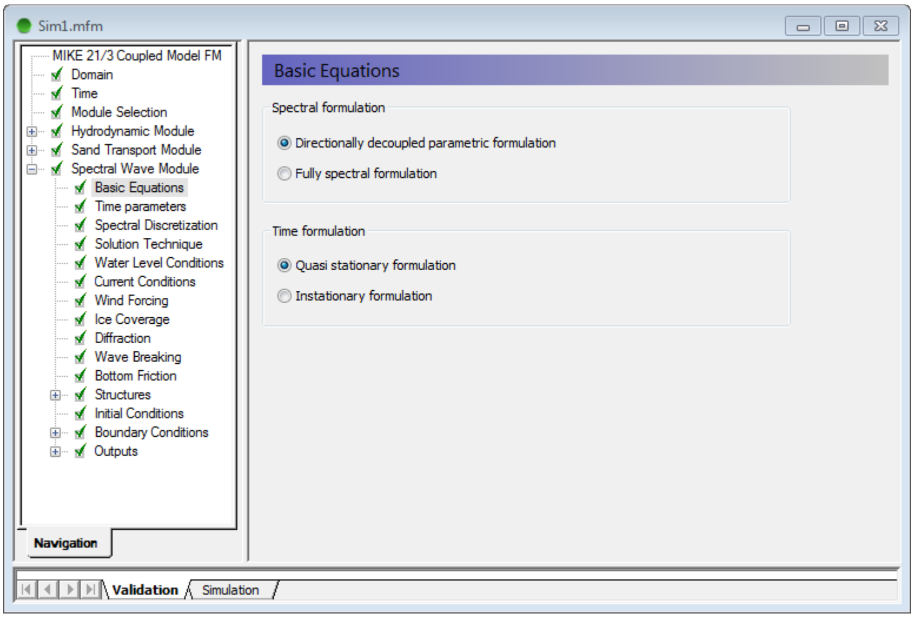

Figure 4.17 MIKE 21/3 Coupled Model FM, Spectral Wave Module: Basic Equations

The start time step for the wave calculations is specified in the 'Time parameters' dialogue. The time step for the wave simulation is defined by the overall time step.

The wave simulation should start at the same time as the hydrodynamic simulation, e.g. start time step is 0 as shown in Figure 4.18.

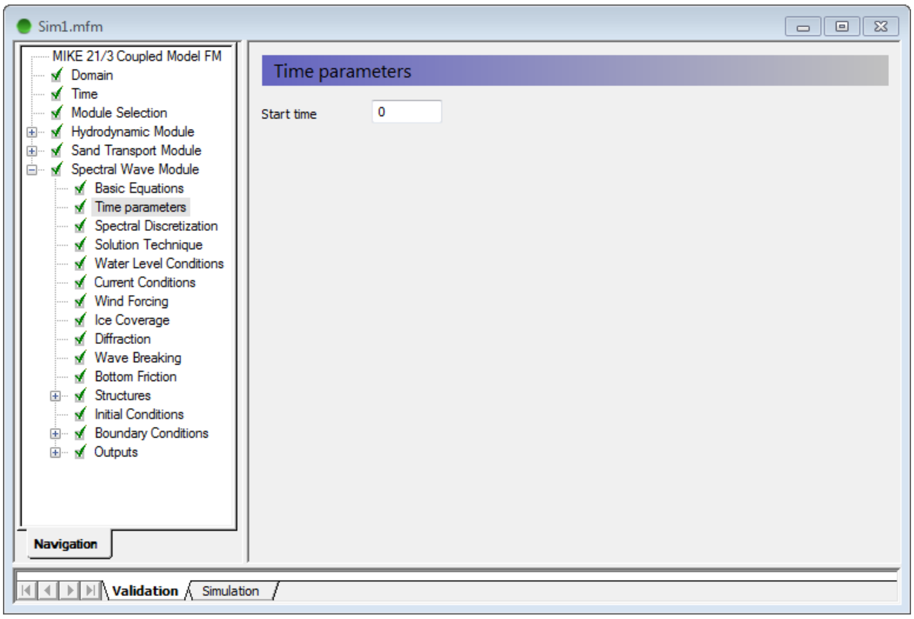

Figure 4.18 MIKE 21/3 Coupled Model FM, Spectral Wave Module: Time parameters

In the 'Spectral Discretization' dialogue you specify the spectrum from which the wave simulations are based. Specify the values as given in Figure 4.19.

Please note that increasing the number of directions will give more accurate results, but also longer simulation time. If the expected wave directions fall within a pre-defined range of directions, the choice of a directional sector type of discretization is recommended to reduce the computational time.

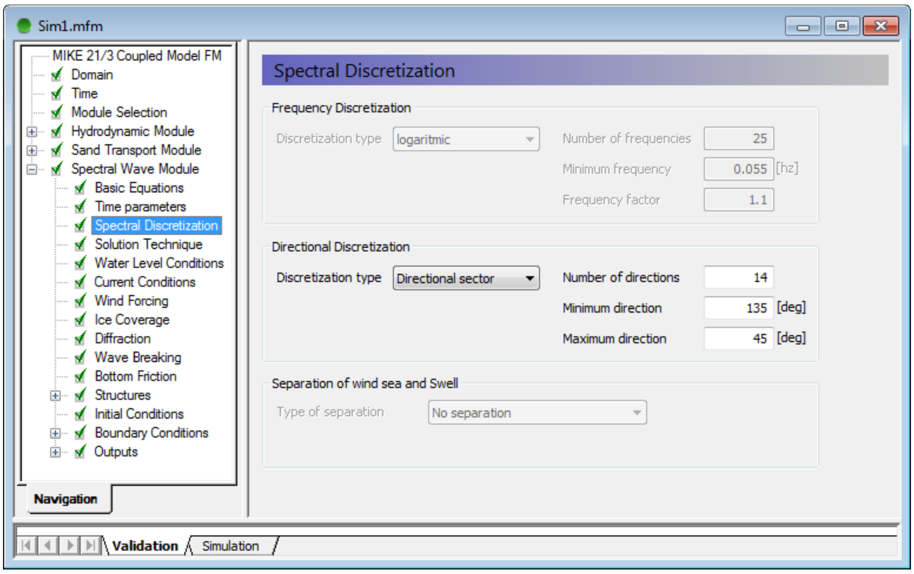

Figure 4.19 MIKE 21/3 Coupled Model FM, Spectral Wave Module: Spectral Discretization

In the 'Solution Technique' dialogue you specify the method as Newton-Raphson iteration. Specify the values as given in Figure 4.20.

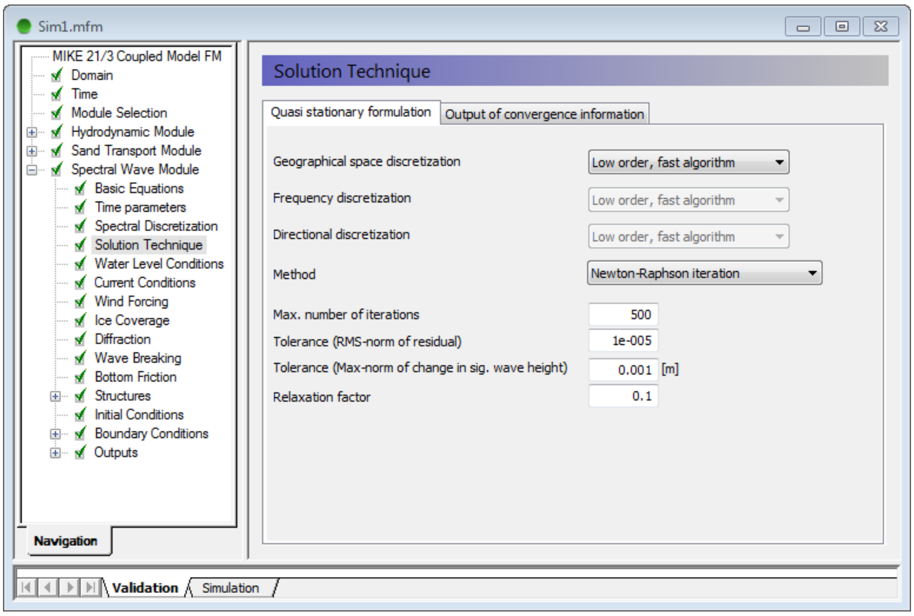

Figure 4.20 MIKE 21/3 Coupled Model FM, Spectral Wave Module: Solution Technique

In the 'Water Level Conditions' dialogue you specify the type as 'Water level variation from HD simulation', see Figure 4.21.

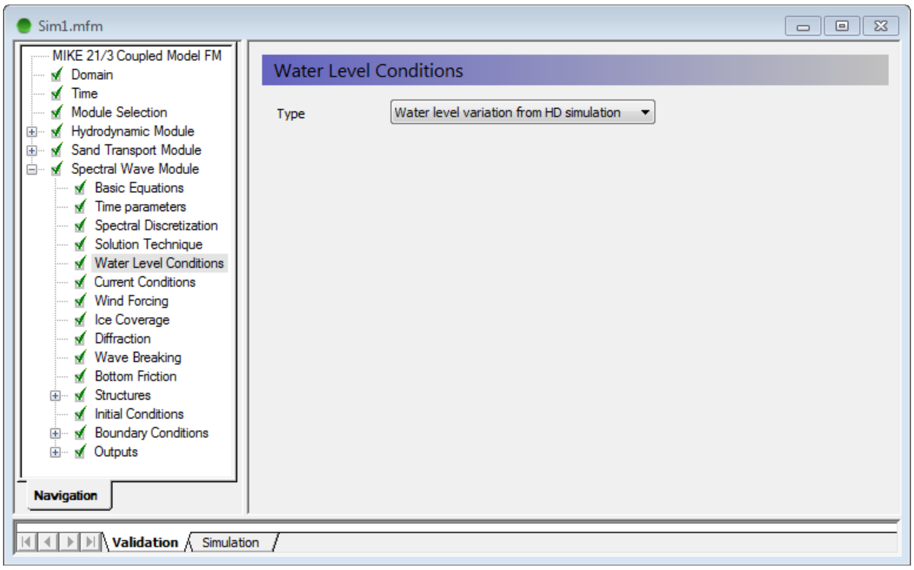

Figure 4.21 MIKE 21/3 Coupled Model FM, Spectral Wave Module: Water Level

Likewise, in the 'Current Conditions' dialogue you specify the type to be 'Current variation from HD simulation', see Figure 4.22.

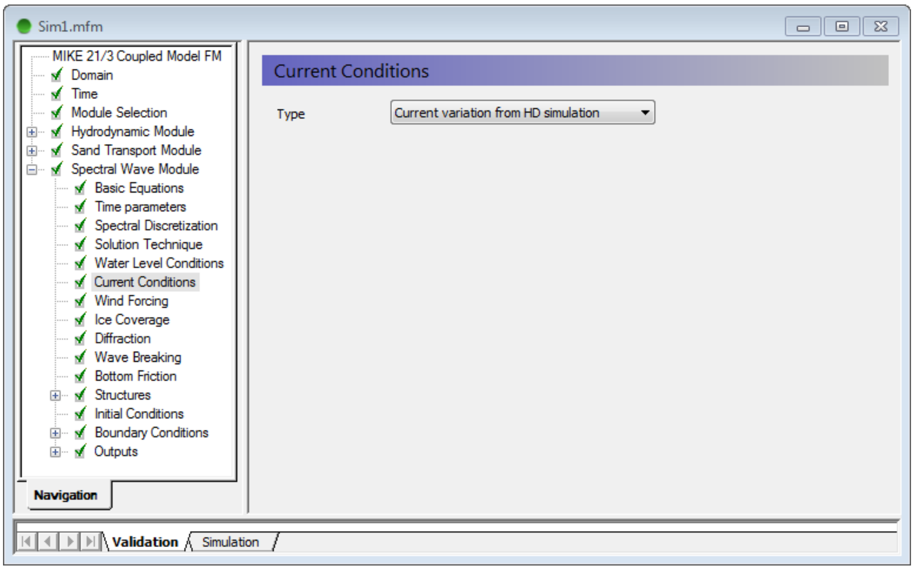

Figure 4.22 MIKE 21/3 Coupled Model FM, Spectral Wave Module: Current Conditions

The Wind Forcing is included as a time series file similar to the input for the Hydrodynamic module, see Figure 4.23.

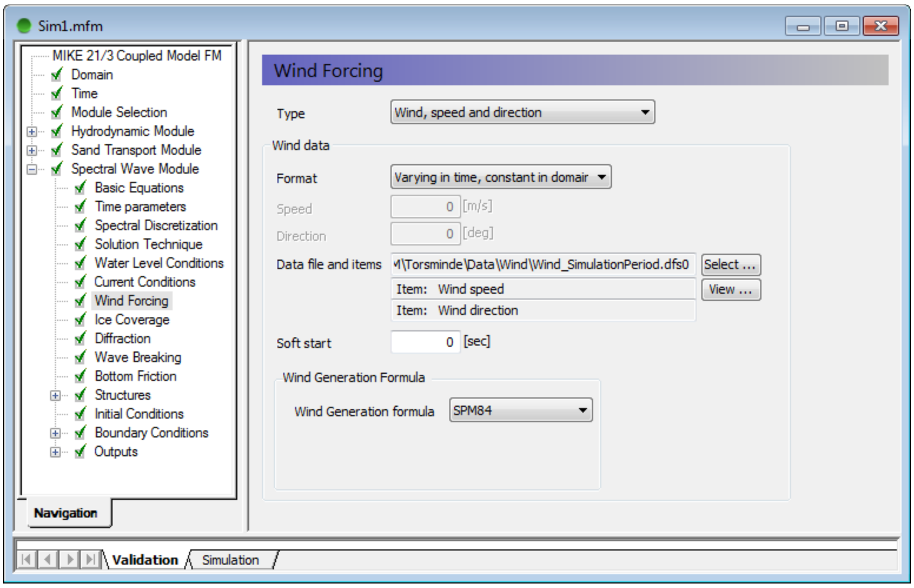

Figure 4.23 MIKE 21/3 Flow Model FM, Spectral Wave Module: Wind Forcing

In this example 'Ice Coverage' and ‘Diffraction’ are not included and the dialogue pages are is not explained.

Use the default values in the 'Wave Breaking' dialogue, see Figure 4.24.

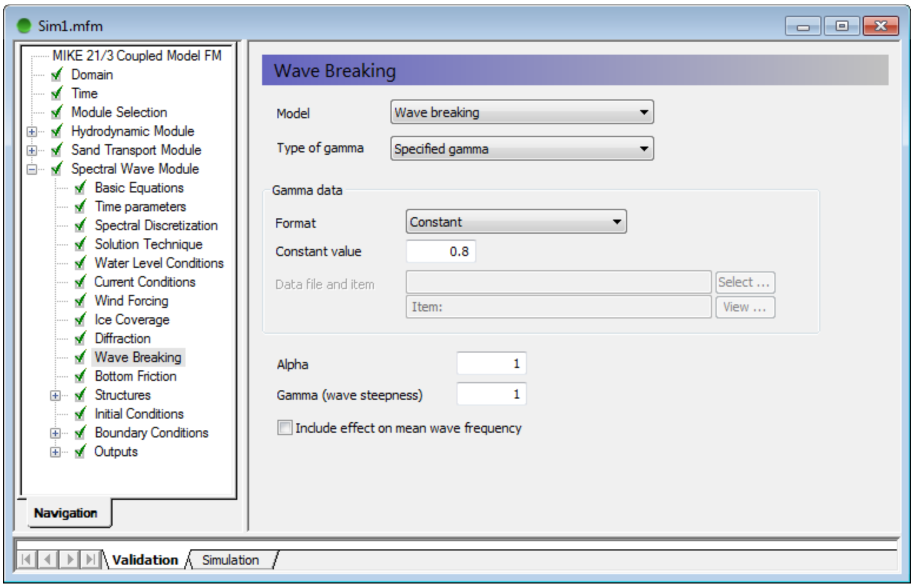

Figure 4.24 MIKE 21/3 Coupled Model FM, Spectral Wave Module: Wave Breaking

In the 'Bottom Friction' dialogue, the model for the friction is selected to Nikuradse roughness, k_N. The bottom friction is set to a constant value of 0.01 m. As default the effect on mean wave frequency is included, see Figure 4.25.

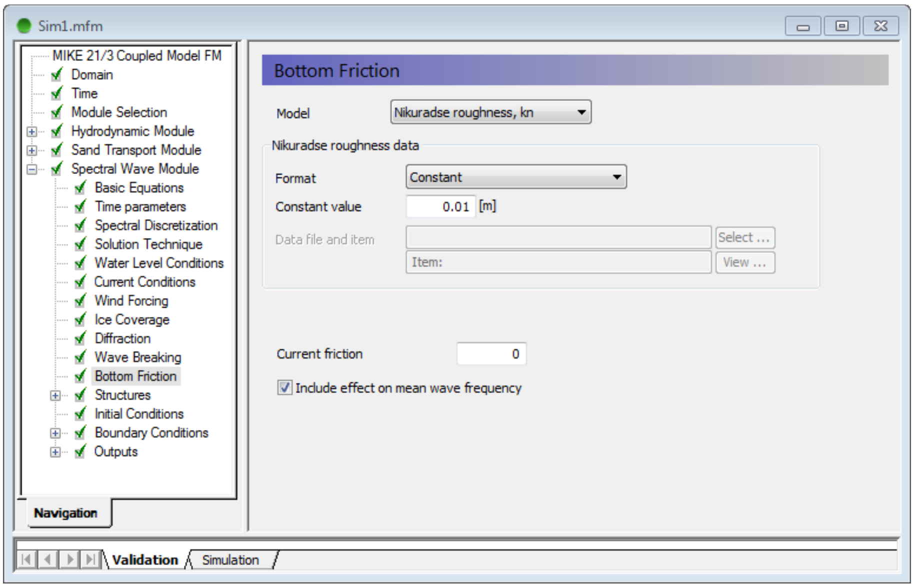

Figure 4.25 MIKE 21/3 Coupled Model FM, Spectral Wave Module: Bottom Friction

In this case the directionally decoupled parametric formulation is started as a cold-start simulation. Select Zero spectra in the 'Initial Conditions' dialogue, see Figure 4.26.

In this example 'Structures' is not included and the dialogue page is not described.

In the 'Boundary Conditions' dialogue, the boundary conditions should be specified for the boundary names, which were specified in the 'Domain' dialogue, see Figure 4.1. There is a North Boundary, a West Boundary and a South Boundary.

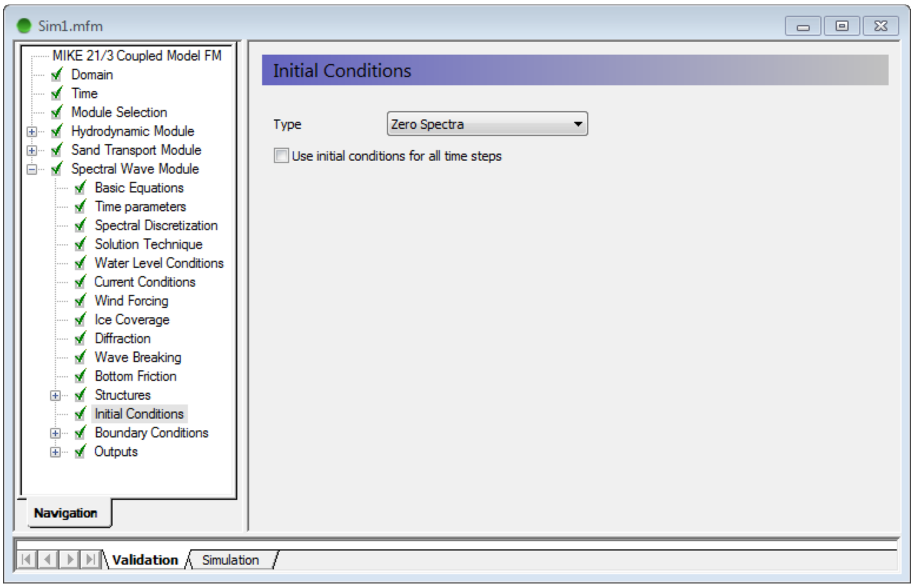

Figure 4.26 MIKE 21/3 Coupled Model FM, Spectral Wave Module: Initial Conditions

In this example, the waves on the West boundary are defined by a time series file whereas the North and South boundary are defined as Lateral, see Figure 4.27. As the lateral boundary conditions is an approximation, inaccurate results near the boundary may be expected, but as the area of interest is well inside the model domain it is assumed that this will have no effect on the conclusive results.

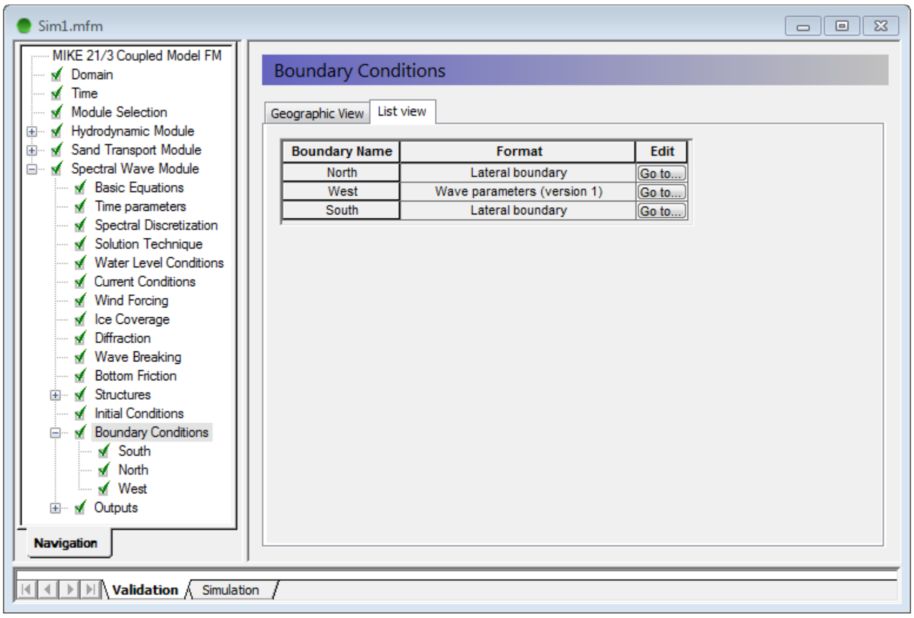

Figure 4.27 MIKE 21/3 Coupled Model FM, Spectral Wave Module: Boundary Condition List

The time series of waves for the West boundary is obtained from a previous simulation as described in Chapter 3. The type is Wave Parameters (Version 1) and the boundary format must be set as 'Varying in time, constant along boundary' in order to specify the boundary as a time series file (dfs0).

Click Select ... to select the data file in the Open File window that appears.

In this case the waves at the start of the simulation are small so it is assumed that a soft start interval is not necessary, see Figure 4.28.

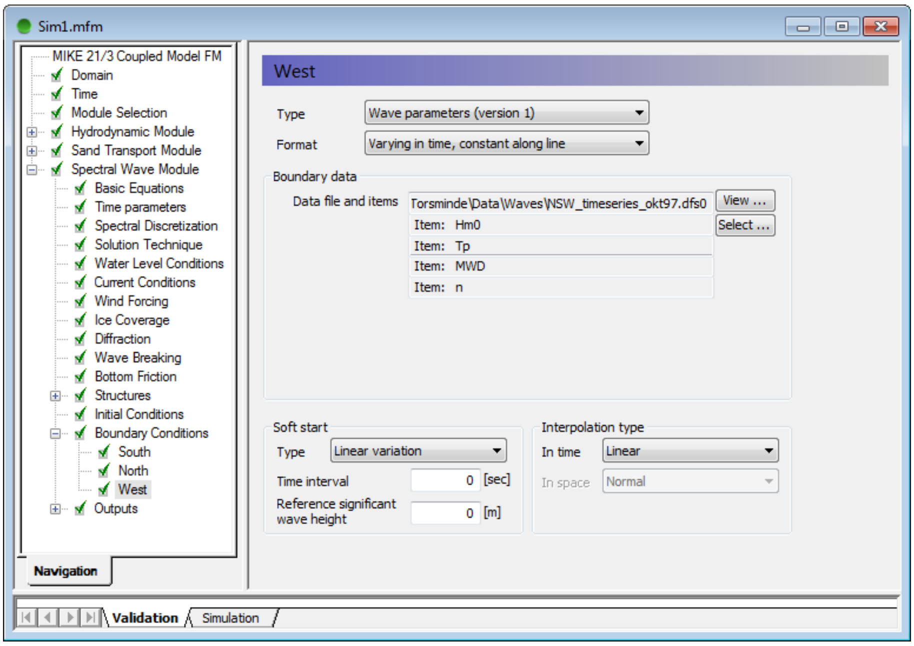

Figure 4.28 MIKE 21/3 Coupled Model FM, Spectral Wave Module: Boundary Conditions

The output of the wave simulation is defined as an area series (2D), see Figure 4.29. Reduce the output size for the area series by selecting an output frequency of 3600 s, similar to the HD output. As default, the full area is selected.

Pick the parameters to be included in the output file as in Figure 4.30.

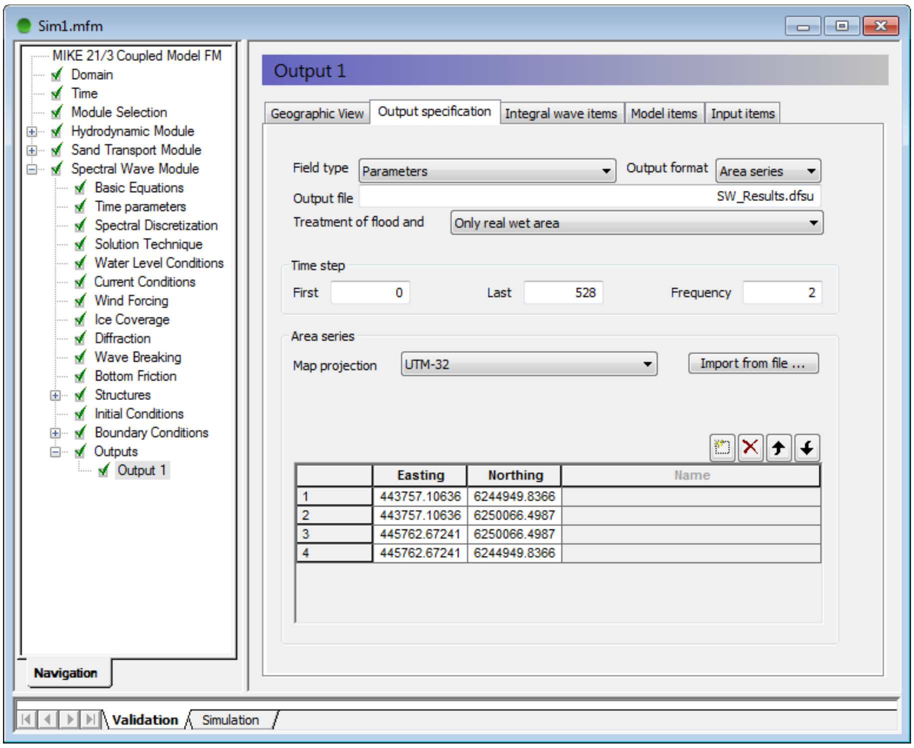

Figure 4.29 MIKE 21/3 Coupled Model FM, Spectral Wave Module: Output type and location

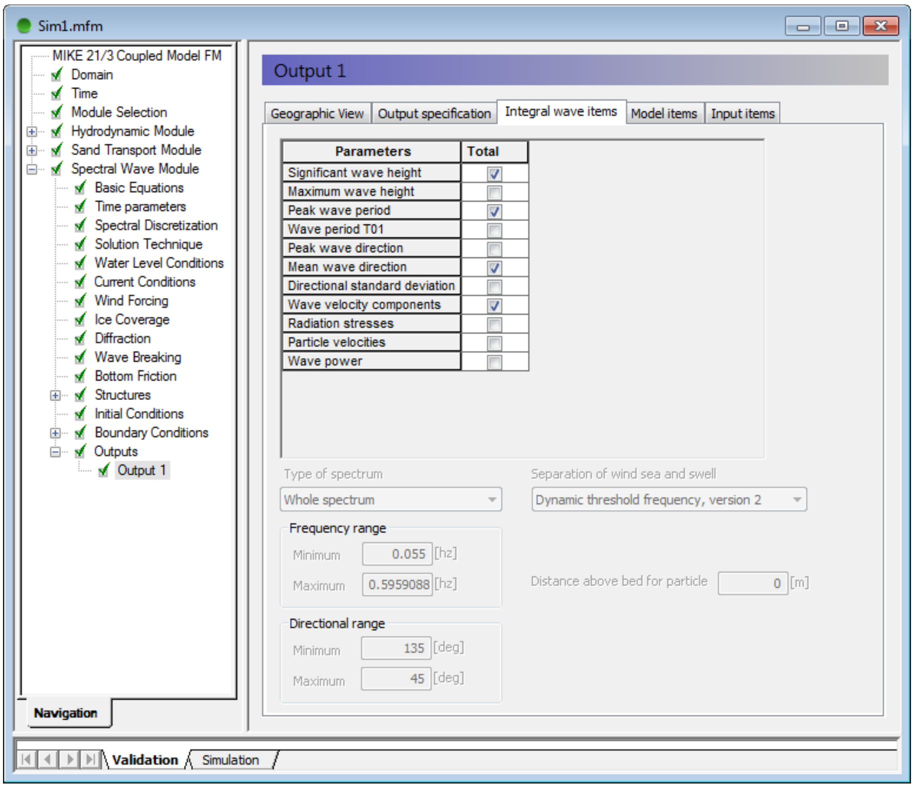

Figure 4.30 MIKE 21/3 Coupled Model FM, Spectral Wave Module: Output items

4.5 Specifications for the Sand Transport Module

In the 'Model Definition' dialogue you select the 'Model type' to *Wave and current* as the sediment transport calculations in this simulation are based on the simultaneous calculation of waves and currents, see Figure 4.31.

To limit computational time, the sediment transport rates for this model type are found by interpolation in a table containing sediment transport rates for an array of representative events. The method of generating such a sediment transport table is described in Chapter 3. Click  to select the sediment transport table 'QsTable.lon' in the Open File window that appears.

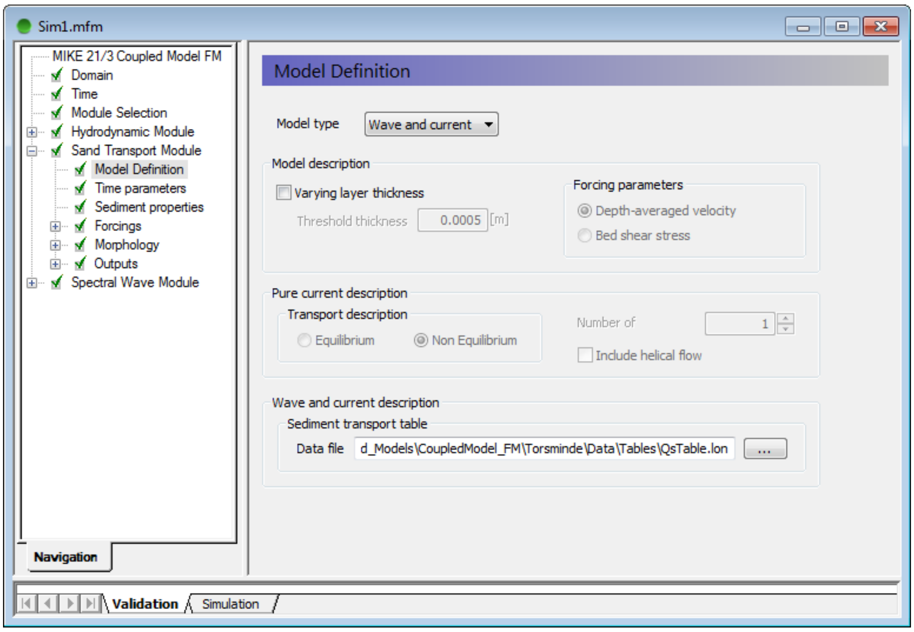

Figure 4.31 MIKE 21/3 Coupled Model FM, Sand Transport Module: Model Definition

The time step for the sediment calculations is specified in the 'Time parameters' dialogue. The time step for the ST simulation is defined in relation to the overall time step.

The sediment transport simulation should start at the same time as the hydrodynamic simulation, i.e. start time step is 0. The time step for the ST simulation should be defined such that the update of the sediment transport rates reflects the bathymetric changes. In this case an update frequency of ½ hour is chosen, hence the time step factor must be set to 1 (1800 s), see Figure 4.32.

Please note that decreasing the time step factor may increase the simulation time.

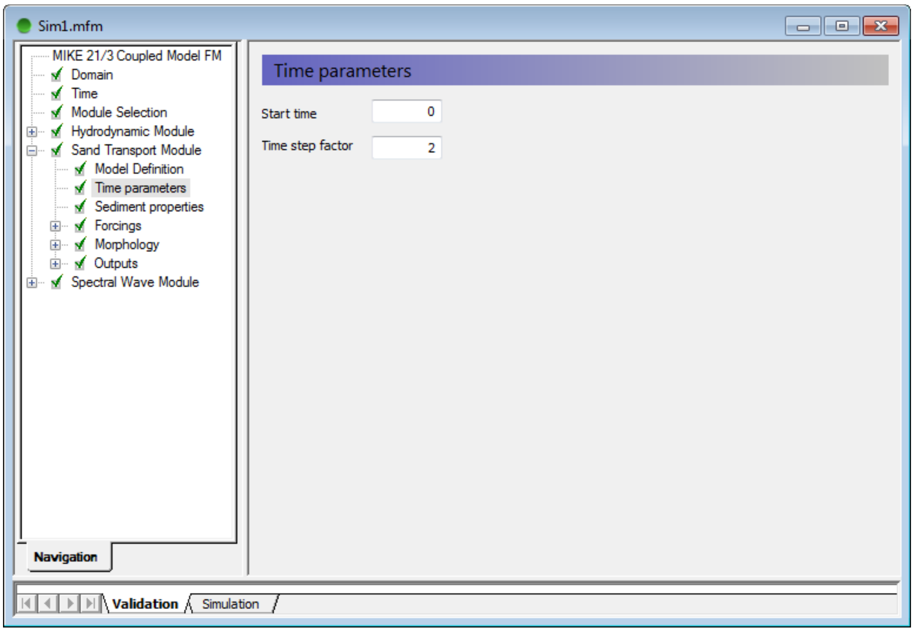

Figure 4.32 MIKE 21/3 Coupled Model FM, Sand Transport Module: Time parameters

The sediment properties must be defined in the 'Sediment Properties' dialogue. Per default the porosity is set to 0.4.

Important: The specified grain diameter or grading coefficient must be contained in the sediment transport table. If the sediment data is outside the limits of the table, the simulation will stop!

In this case the sediment data in the area is specified as Constant with a mean Grain diameter of 0.3 mm and a Grading coefficient of 1.45, see Figure 4.33. (It is the type of sediment, which is described in the 'Sediment transport table' in Figure 4.31).

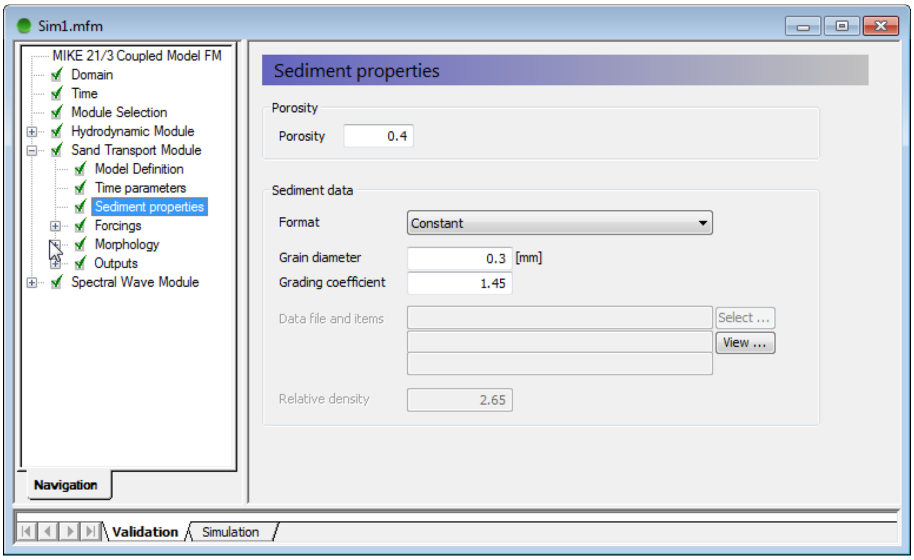

Figure 4.33 MIKE 21/3 Coupled Model FM, Sand Transport Module: Sediment properties

In this simulation it is implicit that the current fields are found from the ongoing HD simulation. In the 'Waves' dialogue you specify the type as 'Wave field from SW simulation', such that you use the results from the ongoing SW simulation, see Figure 4.34.

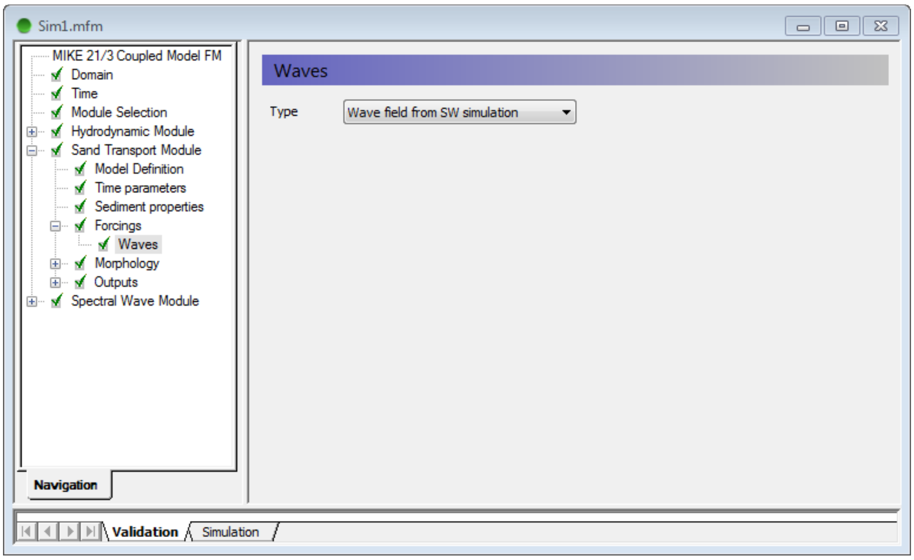

Figure 4.34 MIKE 21/3 Coupled Model FM, Sand Transport Module: Forcings, Waves

The focus of this simulation is to model the morphological evolution during a storm period, i.e. bathymetry changes. In the 'Model Definition' dialogue you select to include the feedback on hydrodynamic, wave and sand transport calculation. This enables a dynamic update of the bathymetry that effects the future calculations, see Figure 4.35.

Set the maximum bed level change to 2 metres/day to avoid instabilities in the simulation because of abrupt changes at the boundaries.

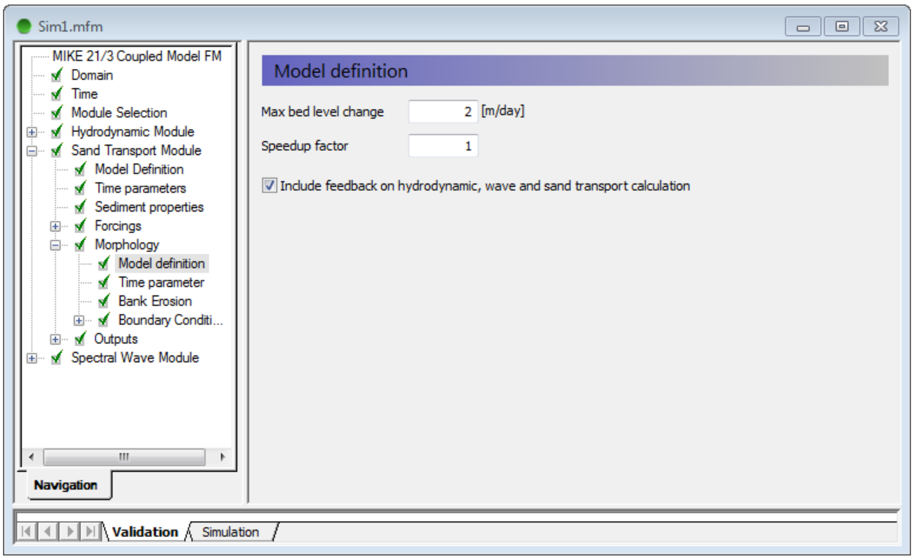

Figure 4.35 MIKE 21/3 Coupled Model FM, Sand Transport Module, Morphology: Model Definition

The time step for the morphological update is the same as the time step for the sediment calculations, i.e. the speed-up factor is 1.

You can start the morphological update of the bathymetry at a later time than the start of the sediment transport calculations, but in the present case you should set the start time step to 0, see Figure 4.36.

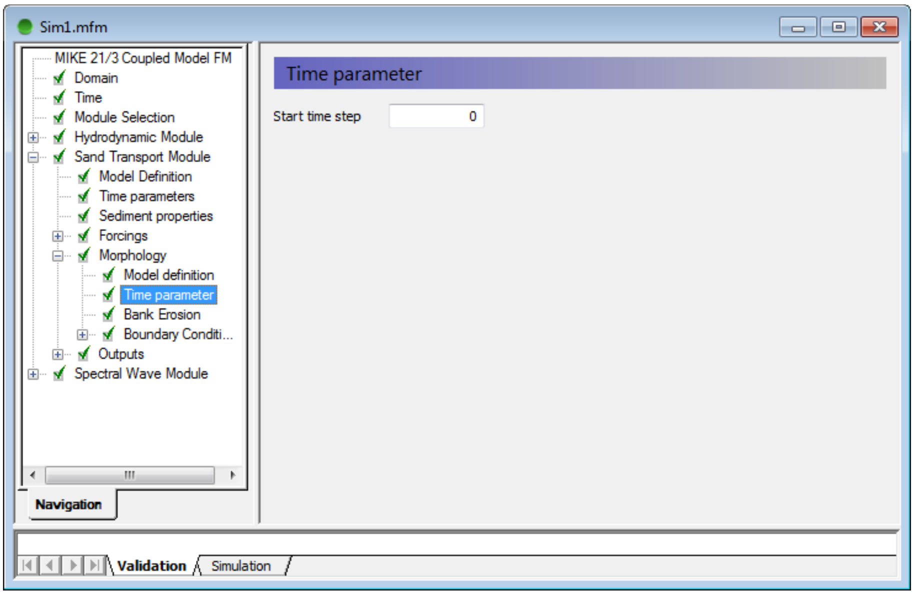

Figure 4.36 MIKE 21/3 Coupled Model FM, Sand Transport Module, Morphology: Time parameter

Bank erosion is not included in the model as the focus area is in front of the harbour entrance where no slope failure mechanisms will influence the result.

In the 'Boundary Conditions' dialogue, the boundary conditions should be specified for the boundary names, which were specified in the 'Domain' dialogue, see Figure 4.1. There is a North Boundary, a West Boundary and a South Boundary.

Select the boundary type for all three boundaries as shown in Figure 4.37.

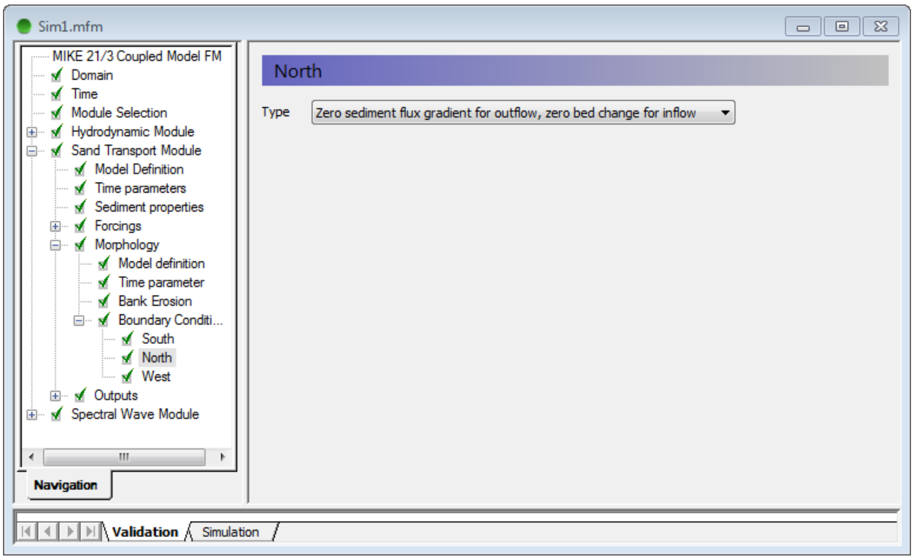

Figure 4.37 MIKE 21/3 Coupled Model FM, Sand Transport Module, Morphology: Boundary conditions

The output of the sediment transport simulation is defined as an area series (2D), see Figure 4.38. Select an output frequency of 3600 s, twice the time step of the HD simulation. As default, the full area is selected.

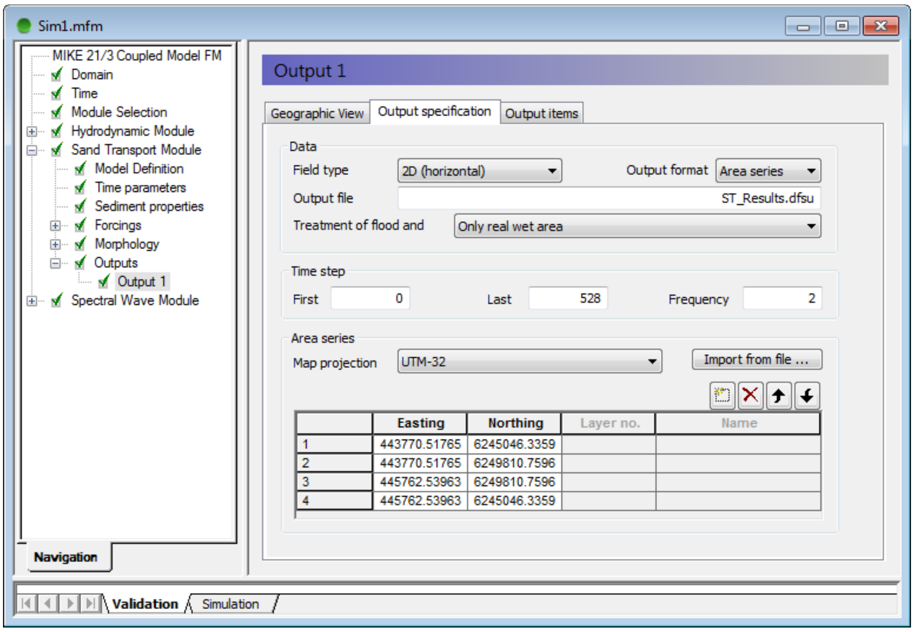

Figure 4.38 MIKE 21/3 Coupled Model FM, Sand Transport Module: Output type and location

Pick the parameters to include in the output file as in Figure 4.39.

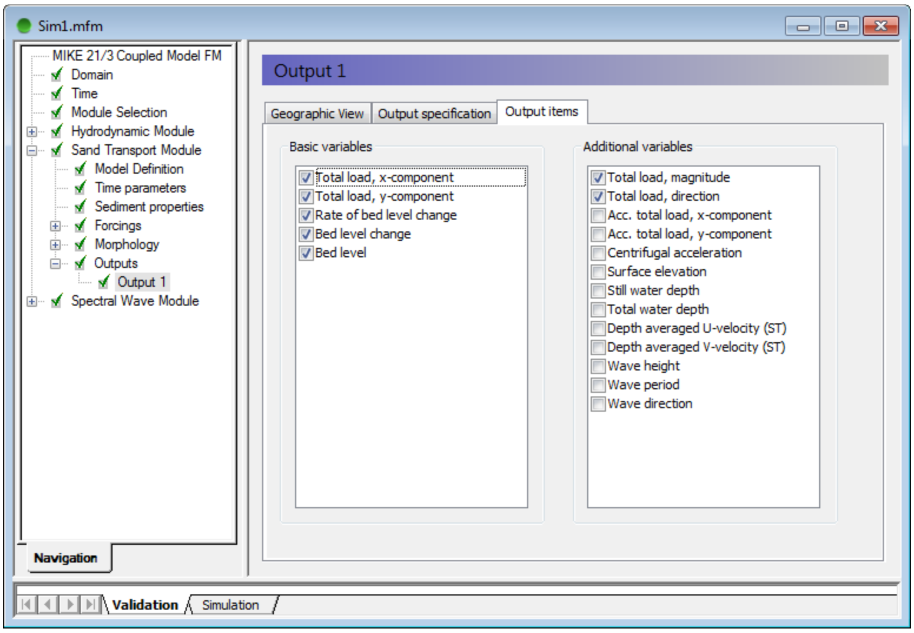

Figure 4.39 MIKE 21/3 Coupled Model FM, Sand Transport Module: Output items

5 Model Calibration

In order to calibrate the model, some measurements are required inside the model domain.

5.1 Measured Bathymetries

In this example, the bed level in front of the harbour entrance has been surveyed on 16/10 and 27/10 1997, respectively. These two dates are the beginning and end of the simulation period. The findings are shown in Figure 5.1 as maps of the survey data.

The maps show that two areas in front of the harbour entrance are significantly more shallow than the surroundings. One area is an isolated outer bar, positioned 300 m from the coastline. Another area is an inner bar located close to the northern jetty, extending in front of the harbour entrance.

Before the storm, on 16 October, 1997, the outer bar extended 100 in the north-south direction and 50 m in the west-east direction. It rose up to -3.0 m in an area where the bed levels were below -4 m. The inner bar extended 200 m along the coast and 50 m across. The southern 100 m of the bar was located just in front of the harbour entrance. The bar rose to above -2 m in an area where the bed generally was between -4 m and -5 m.

After the storm, on 27 October, 1997, the outer bar remained in its general appearance. However, it seems like the most shallow area did migrate a bit northwards. The inner bar now stretches even further across the entrance towards the south, though the top level of the bar has decreased.

These bathymetry changes can be related to the hydrodynamic conditions. In the beginning of the period, the fairly small waves from southwest would generate a north-bound littoral drift. On 20 October 1997 the wave direction shifted to northwest and the wave height increased thus generating a larger south-bound littoral drift. In total this would cause the sediment in the area to move towards south, as seen for the inner bar.

In reality the sluice inside the harbour was closed during the storm and reopened after the storm, but before the post-storm measurement. The influence of the outflow on the bed level change can be seen as the two seawards extensions on the inner bar.

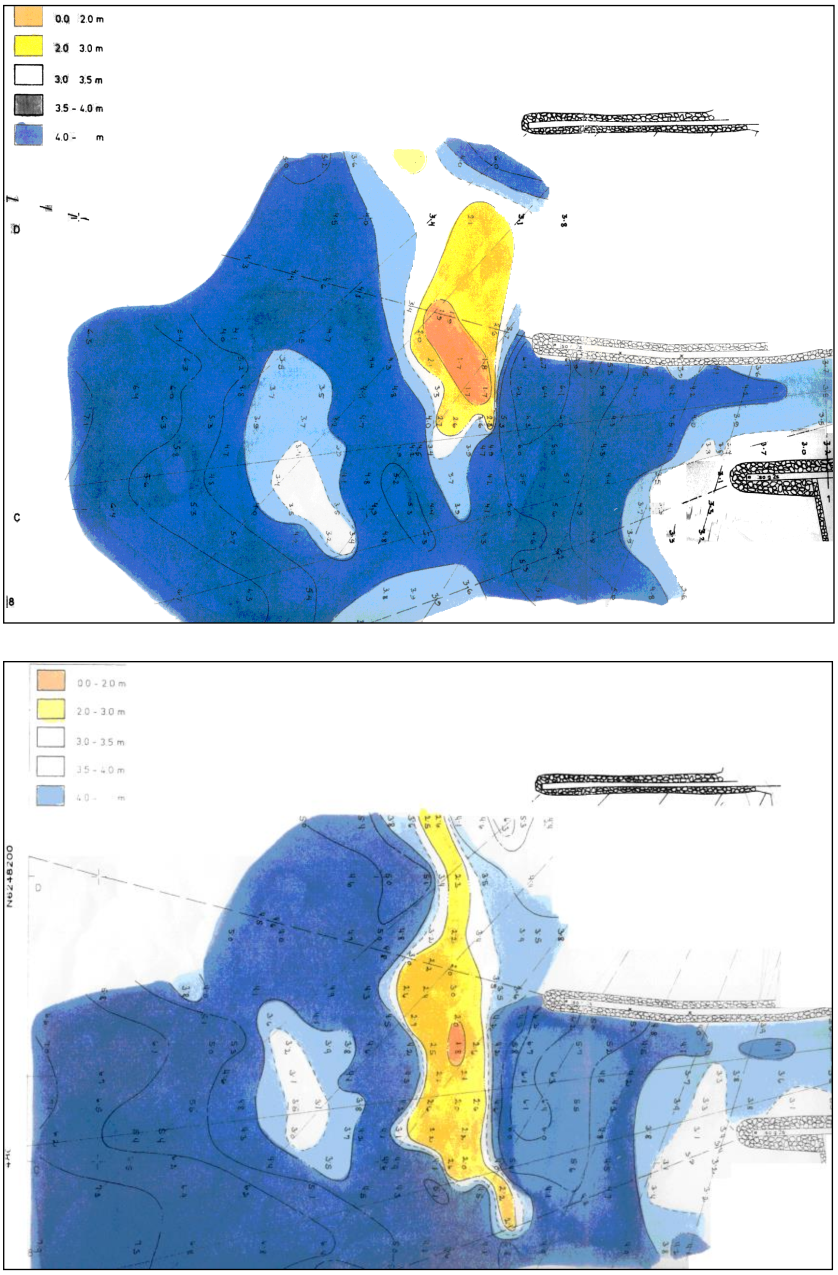

Figure 5.1 Measured bed level in front of harbour entrance.
Upper: measured 16/10 1997
Lower: measured 27/10 1997

5.2 Comparison of Measured and Calculated Bathymetries

Use the Data Viewer to plot the initial bathymetry and the calculated bed level after the 11 days of simulation. A comparison can easily be made by applying a colour legend that resembles the scanned map.

Figure 5.2 shows the measured and simulated bathymetries before and after the storm event.

It is seen that the bar in front of the harbour entrance in the simulation shows the same tendency of southward migration as observed in nature. However, the form of the bar is different. This is probably because the outflow due to the opening of the sluices in the harbour at the end of the storm period is not considered in the simulation.

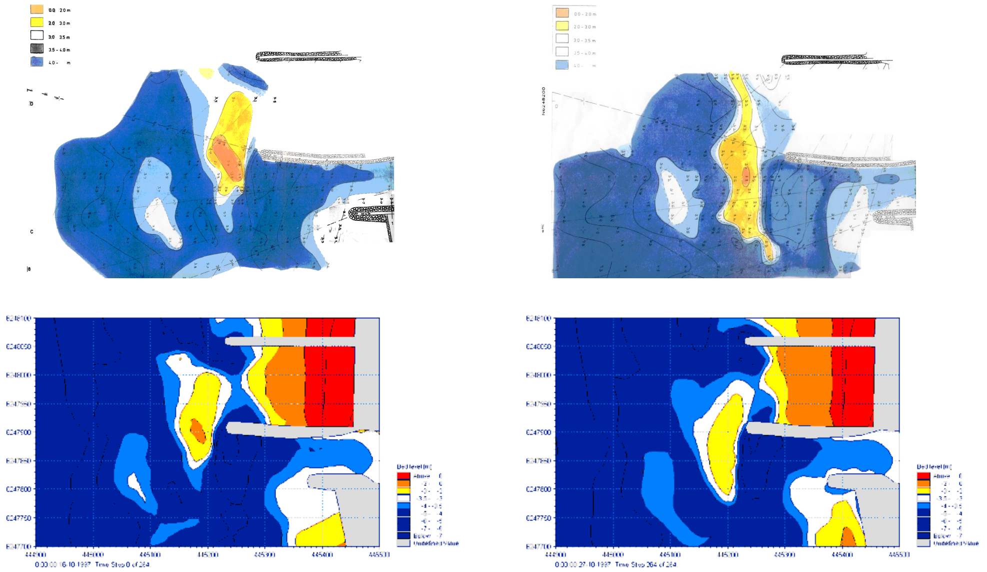

Figure 5.2 Comparison of measured and simulated bathymetries in front of harbour entrance.
Upper: measured
Lower: calculated

5.3 Line Series Extraction

The development of the inner bar can be reviewed in detail by extracting lines from the result file.

To extract lines from a dfsu file, use the Data Extraction FM tool in MIKE Zero, see Figure 5.3.

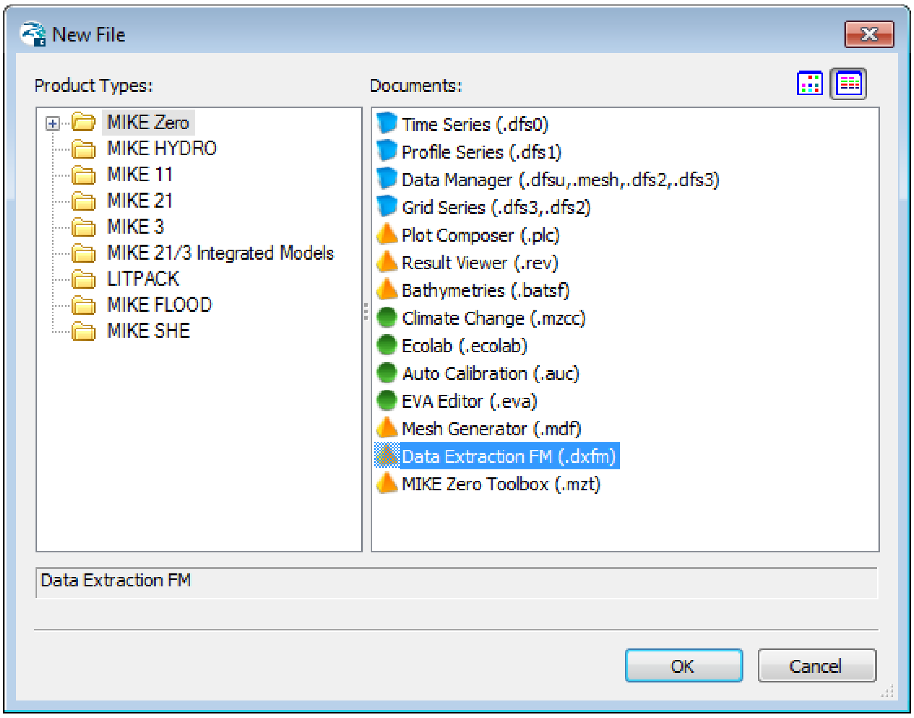

Figure 5.3 Starting the Data Extraction FM tool in MIKE Zero

In the 'Input' dialogue specify the name of the file containing the simulation results, see Figure 5.4.

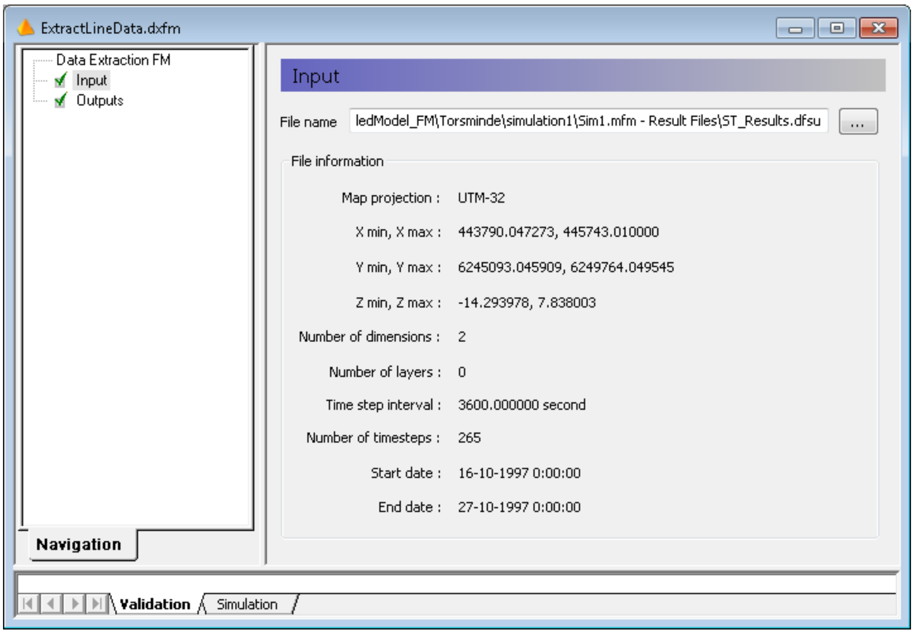

Figure 5.4 Selecting input file

In the 'Output' dialogue you define an output file for each line you want to extract, see Figure 5.5.

In this case three lines are defined as listed in Table 5.1.

Table 5.1 Extraction lines from the Output dialogue

Lines	Start	End	Description
Line 1	(445100, 6247880)	(445300, 6247880)	North cross-shore line
Line 2	(445100, 6247830)	(445300, 6247830)	South cross-shore line
Line 3	(445200, 6248050)	(445200, 6247750)	Longshore line

The lines are outlined in Figure 5.6.

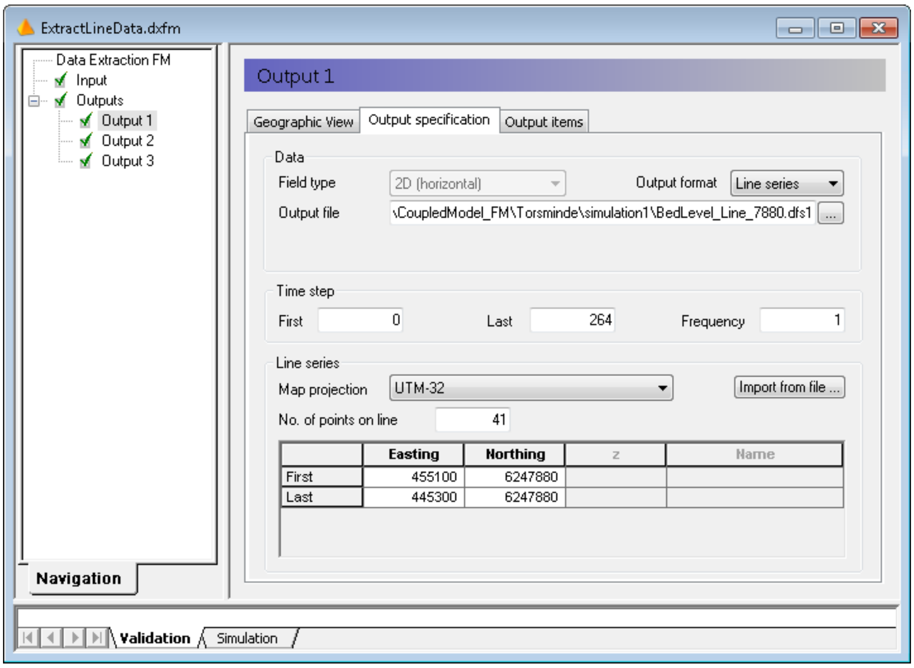

Figure 5.5 Defining output file and line position

Choose ‘Run’ → ‘Start Data Extraction’ to extract the line data.

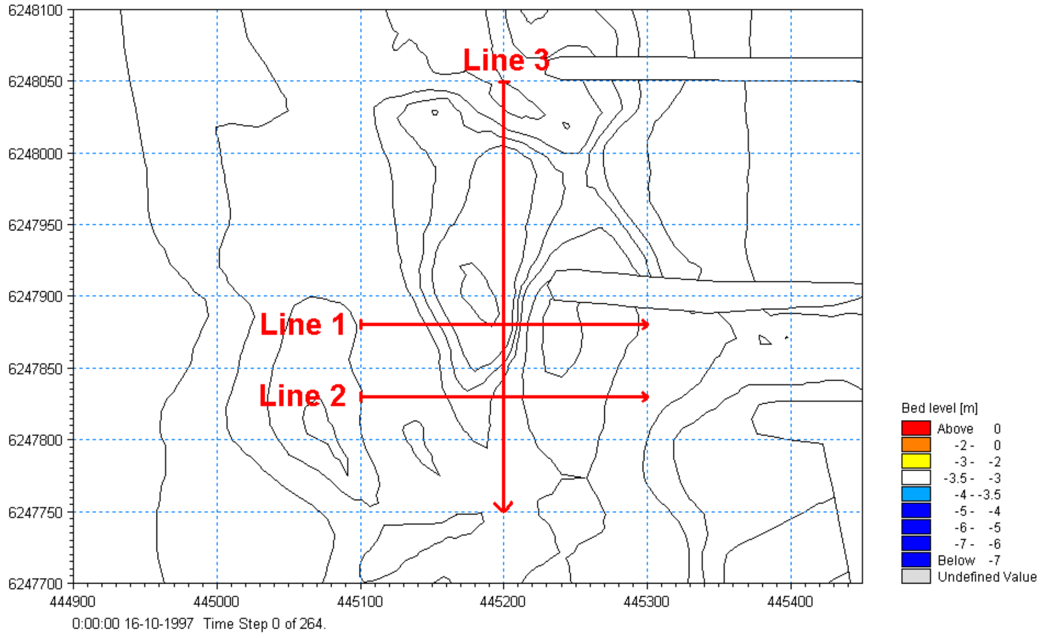

Figure 5.6 Position of extraction lines

The development of the bed level along Line 1, Line 2 and Line 3 are shown in Figure 5.7, Figure 5.8 and Figure 5.9, respectively. Line 3 is positioned to follow the top level of the bar.

It is seen from Figure 5.7 and Figure 5.8 that the bed level does not change significantly in the sheltered area behind the breakwater, i.e. east of position 140 m on the x-axis. Figure 5.7 shows that the inner bar just adjacent to the northern jetty has been subject to some erosion. The contrary is the case for Figure 5.8 where a distinct bar has developed.

Figure 5.9 shows that the existing inner bar has migrated towards south during the simulation, thus covering the harbour entrance entirely. The harbour entrance is positioned between position 150 m and 250 m on the x-axis.

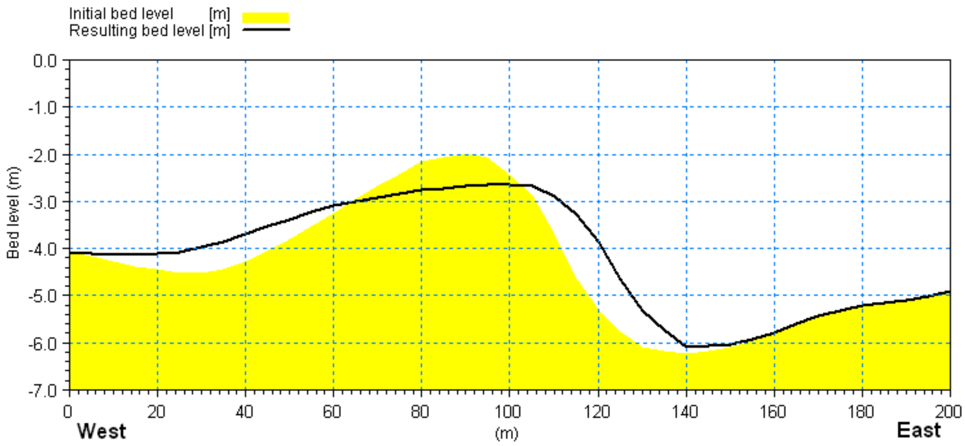

Figure 5.7 Bed level along Line 1: before and after simulation

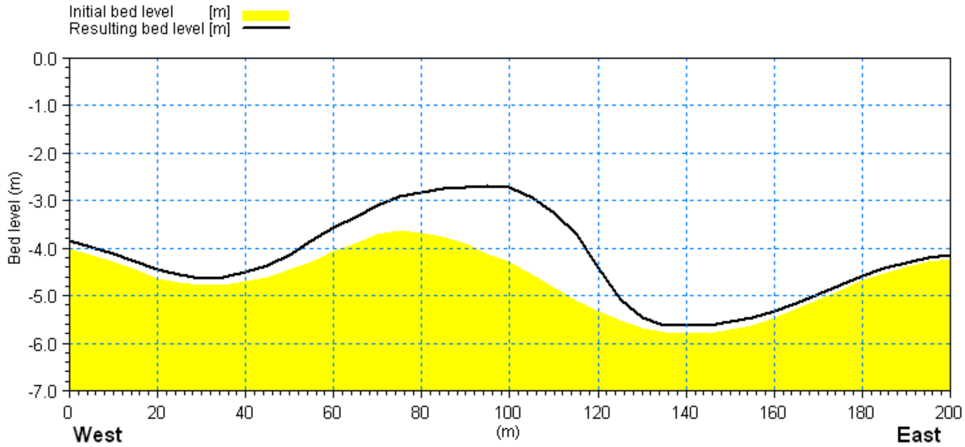

Figure 5.8 Bed level along Line 2: before and after simulation

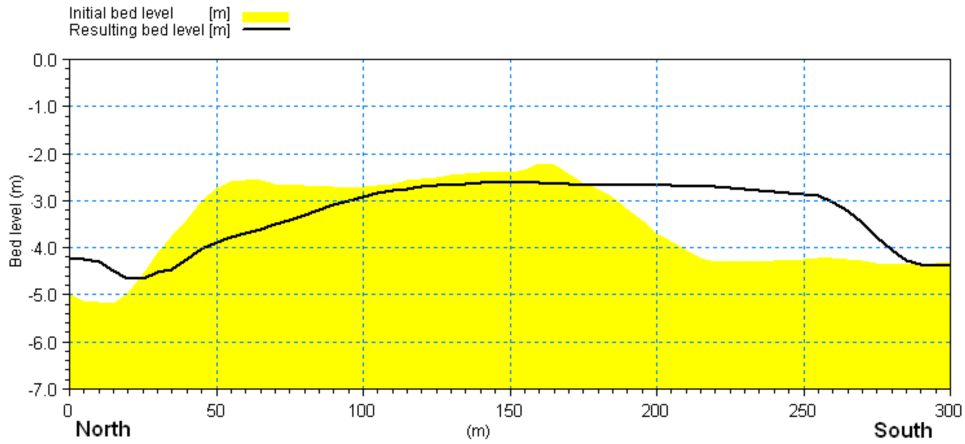

Figure 5.9 Bed level along Line 3: before and after simulation