

## MIKE 21 Flow Model FM

Mud Transport Module

Step-by-step training guide



**DHI headquarters**

Agern Allé 5  
DK-2970 Hørsholm  
Denmark

+45 4516 9200 Telephone

+45 4516 9333 Support

+45 4516 9292 Telefax

[mike@dhigroup.com](mailto:mike@dhigroup.com)

[www.mikepoweredbydhi.com](http://www.mikepoweredbydhi.com)

# CONTENTS

## MIKE 21 Flow Model FM Mud Transport Module Step-by-step training guide

<b>1</b>	<b>Introduction.....</b>	<b>1</b>
1.1	Background.....	2
1.2	Objective.....	3
<b>2</b>	<b>Creating the Computational Mesh .....</b>	<b>5</b>
<b>3</b>	<b>Creating the Input Parameters to the MIKE 21 Flow model FM .....</b>	<b>7</b>
3.1	Generate Water Level Boundary Conditions .....	7
3.1.1	Importing measured water levels to time series file.....	9
3.1.2	Creating boundary conditions .....	14
3.2	Initial Conditions.....	17
3.3	Wind Forcing.....	18
3.4	Generate Manning Number Data.....	19
3.5	Generate Source Data .....	22
3.5.1	Hydrodynamic source discharges.....	22
3.5.2	Suspended sediment source concentrations.....	23
3.6	Generate Mud Transport Model Constant Parameters.....	24
3.6.1	Critical shear stress for consolidated mud in layer 2 .....	24
3.7	Generate Wave Data .....	25
3.7.1	Wave database .....	26
3.8	Generate Mud Transport Model Initial Conditions .....	27
<b>4</b>	<b>Specifications for the MIKE 21 Flow Model FM .....</b>	<b>31</b>
4.1	Basic Parameters.....	32
4.2	Hydrodynamic Module .....	35
4.3	Mud Transport Module.....	46
4.4	Model Calibration 1 .....	57
4.4.1	Measured values.....	57
4.4.2	Compare model result and measured values .....	59
4.5	Model Calibration 2 .....	62
4.5.1	Generate decoupled model data.....	62
4.5.2	Changed model specifications .....	63
4.5.3	Model results compared with measured values.....	64
4.6	Model Calibration 3 .....	64
4.6.1	Decoupled model set-up .....	64
4.6.2	Changed model specifications .....	64
4.6.3	Model results compared with measured values.....	65



## 1 Introduction

This Step-by-step training guide relates to Ho Bay in the Grådyb tidal area located in western Jutland, Denmark. See Figure 1.2.



Figure 1.1 The Grådyb tidal area, Denmark. View towards south with the island Langli in the foreground. Photo: Morten Pejrup



Figure 1.2 Map of the Grådyb tidal area, which is located in the western part of Jutland, Denmark

## 1.1 Background

Ho Bay is situated in the northernmost part of the European Wadden Sea forming the northern part of the Grådyb tidal area (Figure 1.2). This is a unique area with the Varde River being the only major waterway to the Wadden Sea, which has not been regulated or altered through straightening.

The tidal estuary has a very dynamic morphology due to the significant and complicated sediment transport in the area. The surrounding widespread meadows constitute a very special type of nature. The meadows are flooded from time to time, and for the sections near the coast, the floods contain both saltwater and brackish water. This creates conditions for a very special flora and fauna. Because of these very particular natural conditions, the Varde River valley and the meadows around Ho Bay have attracted a great deal of attention, especially with regard to nature conservation.

Today, the area is intensively cultivated, mainly for the production of dried fodder pellets from grass. This intensive type of production conflicts with the wish to preserve the natural ecosystems in the area.

From a scientific point of view the area is also interesting, and therefore the area has been investigated with numerical models in several research projects.



Figure 1.3 View of Varde River from Ho Bay, Denmark. Photo: Thorbjørn Joest Andersen

## 1.2 Objective

The objective of this Step-by-step training guide is to set up a mud transport MIKE 21 Flow Model with Flexible Mesh (MIKE 21 Flow Model FM Mud Transport Module) for the Grådyb tidal area from scratch and to calibrate the model to a satisfactory level.

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 readily accepted by 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.

The 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 MIKE Zero format files, 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 a xyz file(s) for positions of land-boundaries and a xyz file(s) that couples geographical positions with water depths. A dfs2 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 HD Step-by-step training guide, which can be accessed via the MIKE Zero Documentation Index in the start menu:

[MIKE 21 & MIKE 3 Flow Model FM, Hydrodynamic Module, Step-by-Step](#)

When creating the computational mesh you should work with the mdf file, which contains information about settings for the mesh. The focus in Step-by-step training guide is on Mud Transport, so both the necessary xyz files and the mdf file and a mesh file are already supplied with the example, so that the training time can be spent on specific Mud Transport subjects.

The mesh that is used in the example is shown in Figure 2.1, and Figure 2.2 shows a graphical presentation of the mesh of the Grådyb tidal area as it can be displayed with MIKE Animator.

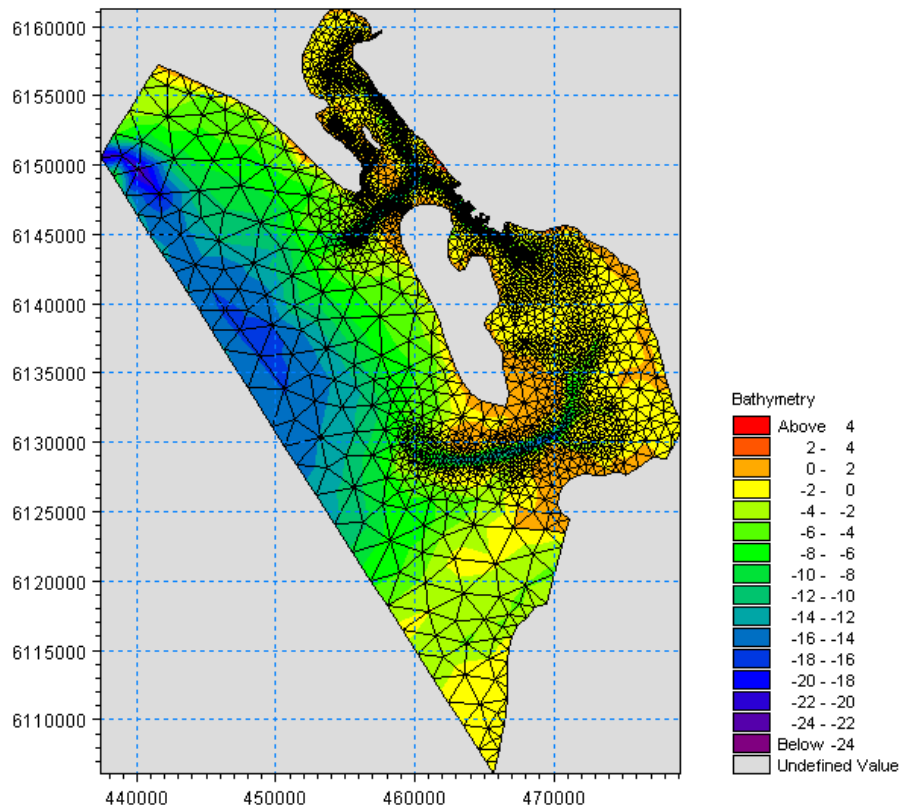


Figure 2.1 The Grådyb tidal area computational mesh as it can be presented with the Data Viewer tool

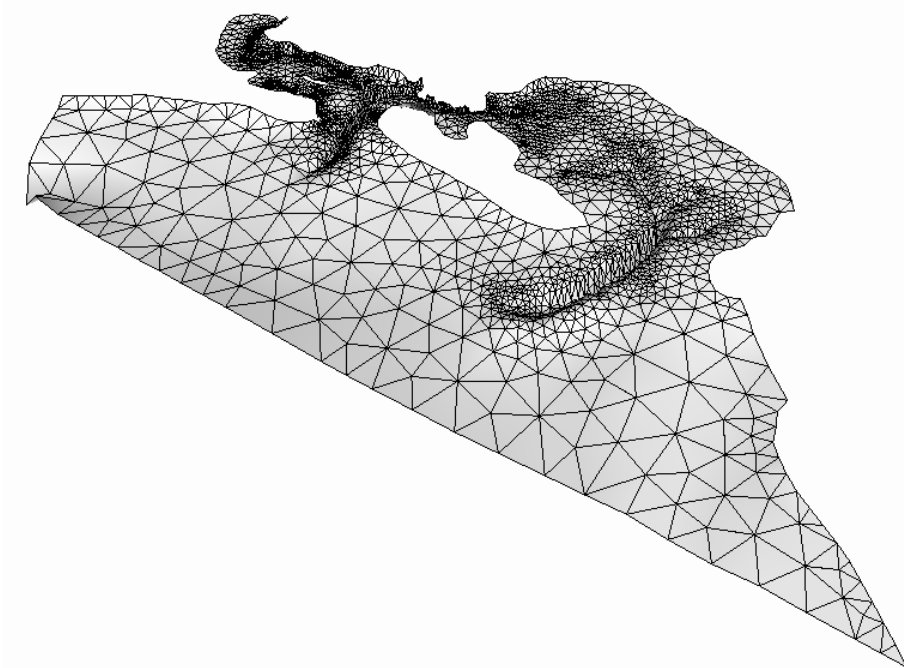


Figure 2.2 The Grådyb tidal area computational mesh as it can be presented with the MIKE Animator tool

## 3 Creating the Input Parameters to the MIKE 21 Flow model FM

Before the MIKE 21 Flow Model FM Mud Transport module can be set up, the input data must be generated from the measurements. The following measurements exist for year 2004:

1. Water levels at station Esbjerg Harbour, see Table 3.1
2. Current measurements at station RCM9, see Table 3.1
3. Turbidity measurements at station RCM9, see Table 3.1
4. Wind measurements from Esbjerg, Denmark
5. Wave model results of Blåvands Huk

The measured raw data is included in the folder:

.\Ho\_Bay\Data\Ascii

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 Generate Water Level Boundary Conditions

Measured water level recordings from monitoring station Esbjerg Harbour are available for the Grådyb tidal area model, see Figure 3.1.

The Grådyb tidal area model is forced with water level boundaries. Unfortunately, there are no measurements at the boundary, so the correct variation along the boundary is not known, and therefore the boundary is specified as a time series (dfs0 type file) constant along the boundary. An empirical method has been used to generate the water levels at the boundary based on the water levels measured in Esbjerg Harbour. The difference between the instant water levels and the average value of water levels in Esbjerg Harbour in the simulation period are multiplied by 110% and applied at the boundaries. The manipulated water levels at the boundary are also phase-shifted a half-hour forward in time relative to the water levels in Esbjerg Harbour to include the phase difference between the positions.

The physical explanation of the above empirical approach is that it describes the phase difference and also the general loss of energy from the boundary to Esbjerg Harbour. This approach has successfully been used in previous studies in the area.

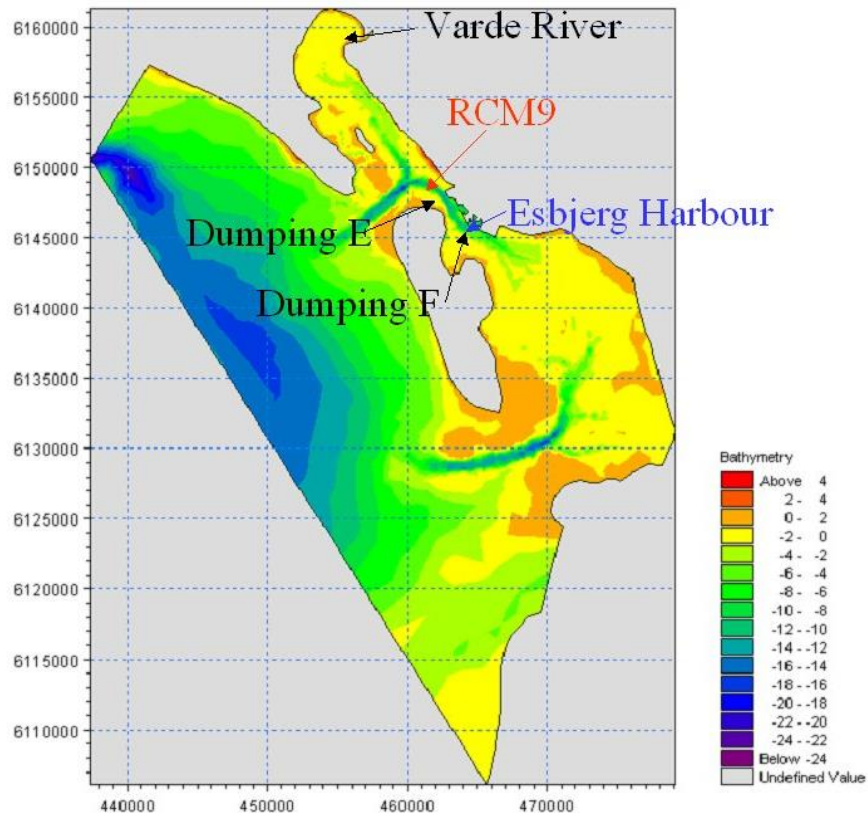


Figure 3.1 Map showing the positions of the water level station at Esbjerg Harbour, the current and turbidity monitoring station RCM9, and the source positions

Table 3.1 Measurements in the Grådyb tidal area. Projection UTM-32 (datum WGS-84)

Station	Easting [m]	Northing [m]	ASCII file
Esbjerg Harbour	464229	6145802	waterlevel_esbjerg.txt
RCM9	461745	6148663	current_rcm9.txt turbidity_rcm9.txt

### 3.1.1 Importing measured water levels to time series file

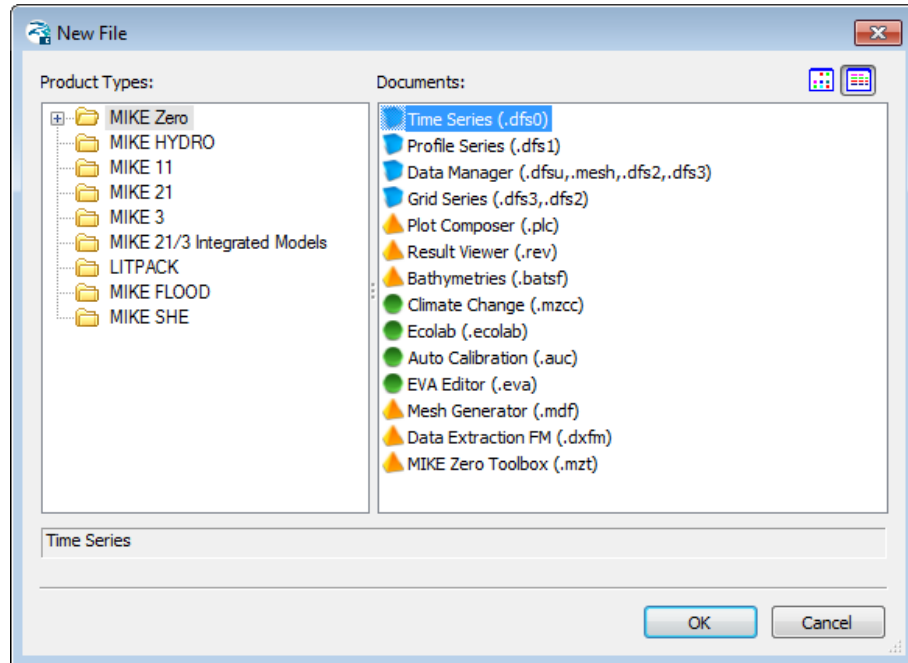


Figure 3.2 Starting the Time Series Editor in MIKE Zero

Open the Time Series Editor in MIKE Zero (File → New → MIKEZero → Time Series), see Figure 3.2. Select the ASCII template. Open the text file waterlevel\_esbjerg.txt (Figure 3.3). Change the time description to 'Non-equidistant Calendar Axis' and press OK, see Figure 3.4. Save the data in waterlevel\_esbjerg.dfs0 and the Time Series Editor will show the imported water levels from station Esbjerg Harbour (Figure 3.5).

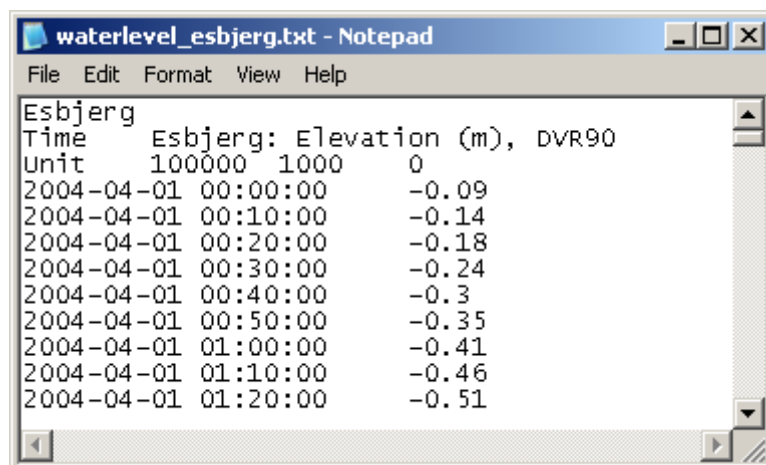


Figure 3.3 ASCII file with water level recordings from station Esbjerg Harbour

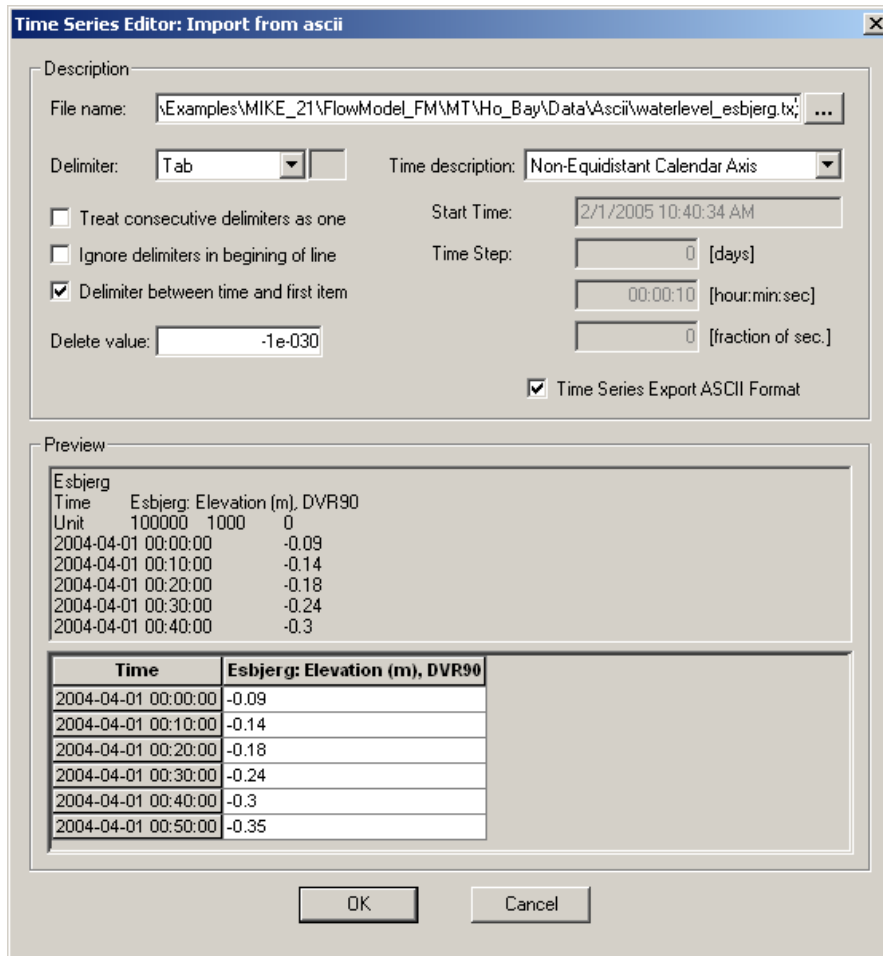


Figure 3.4 Time Series Editor Import Template

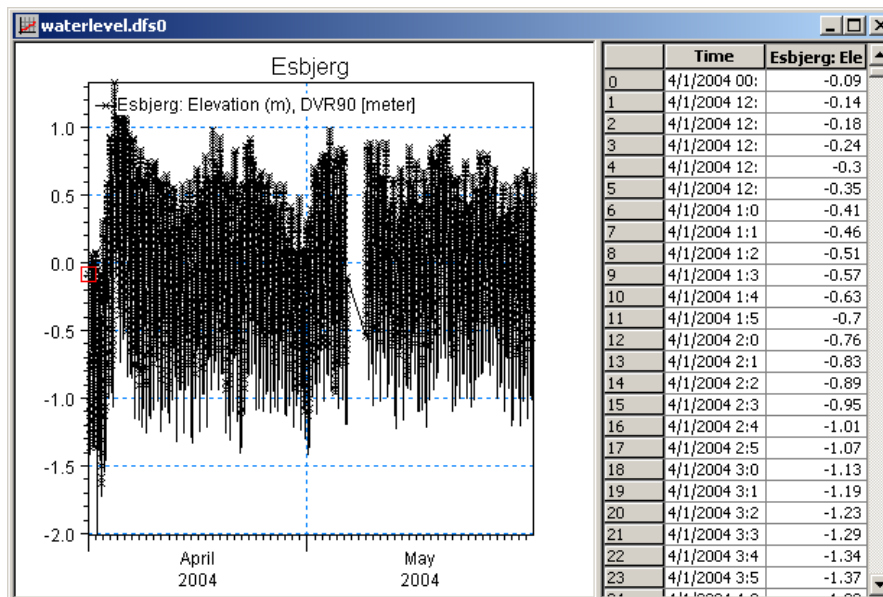


Figure 3.5 Time Series Editor with imported Water Levels from station Esbjerg Harbour

To make a plot of the water level time series open the Plot Composer in MIKE Zero, see Figure 3.6. Select 'plot' → 'insert a new plot object' and select 'Time Series Plot' (see Figure 3.7).

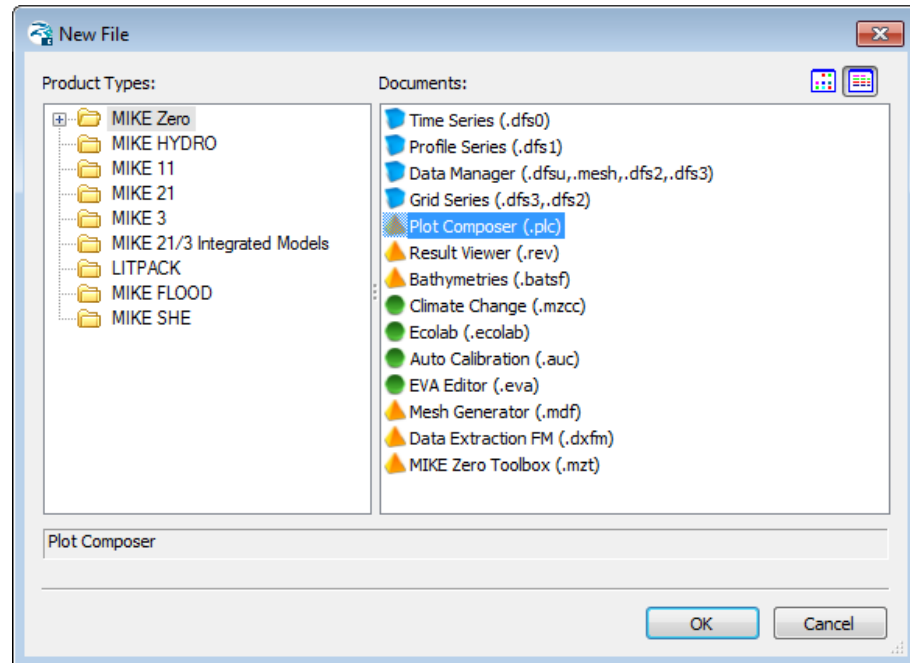


Figure 3.6 Starting the Plot Composer in MIKE Zero

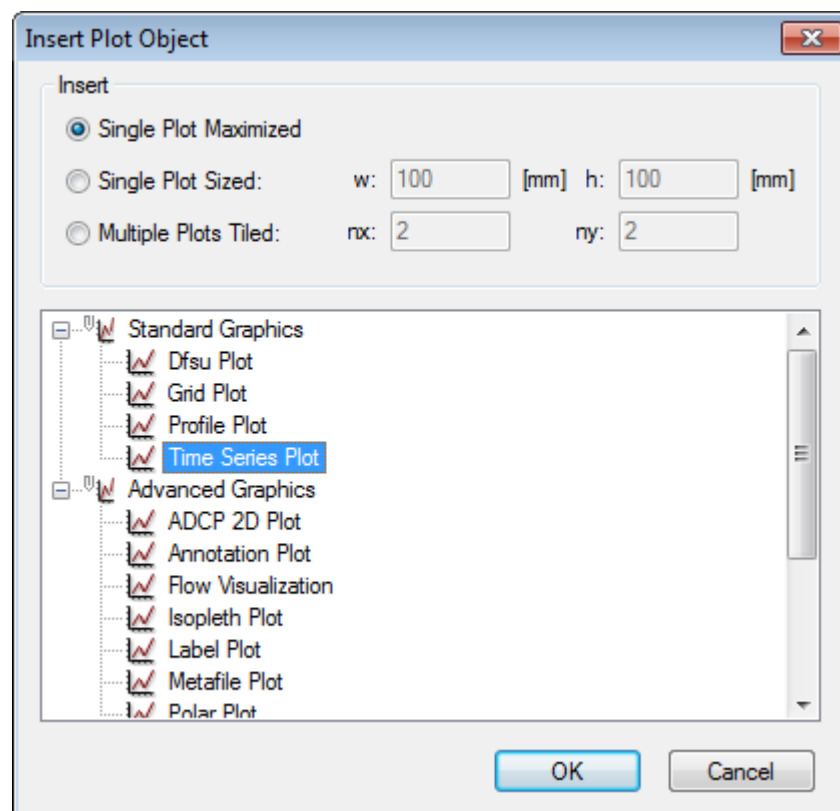



Figure 3.7 Insert a new Plot Object as Time Series in Plot Composer

Right click on the plot area and select properties. Add the actual time series file to the Plot Composer by clicking  and selecting the file, see Figure 3.9. It is possible to add more than one time series to the same plot. In the property dialogue it is possible to change some of the properties for the plot, such a colours, etc. (see Figure 3.8).

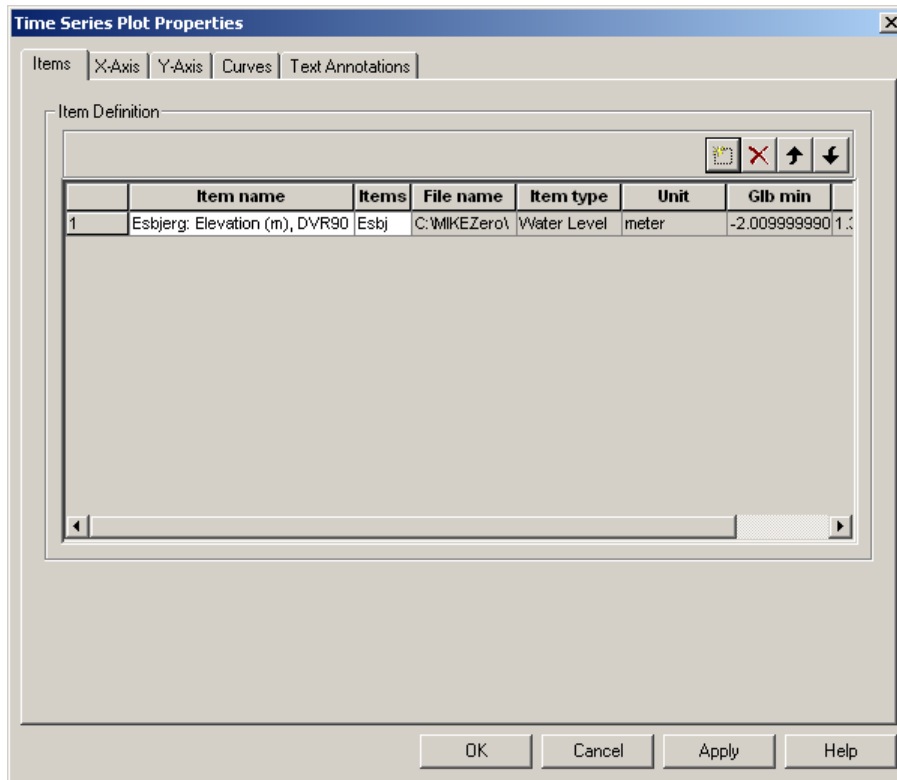


Figure 3.8 Plot Composer Time Series Plot Properties dialogue for selecting time series files and adjusting scales, curves, etc.



Figure 3.10 shows the measured water levels in Esbjerg Harbour.

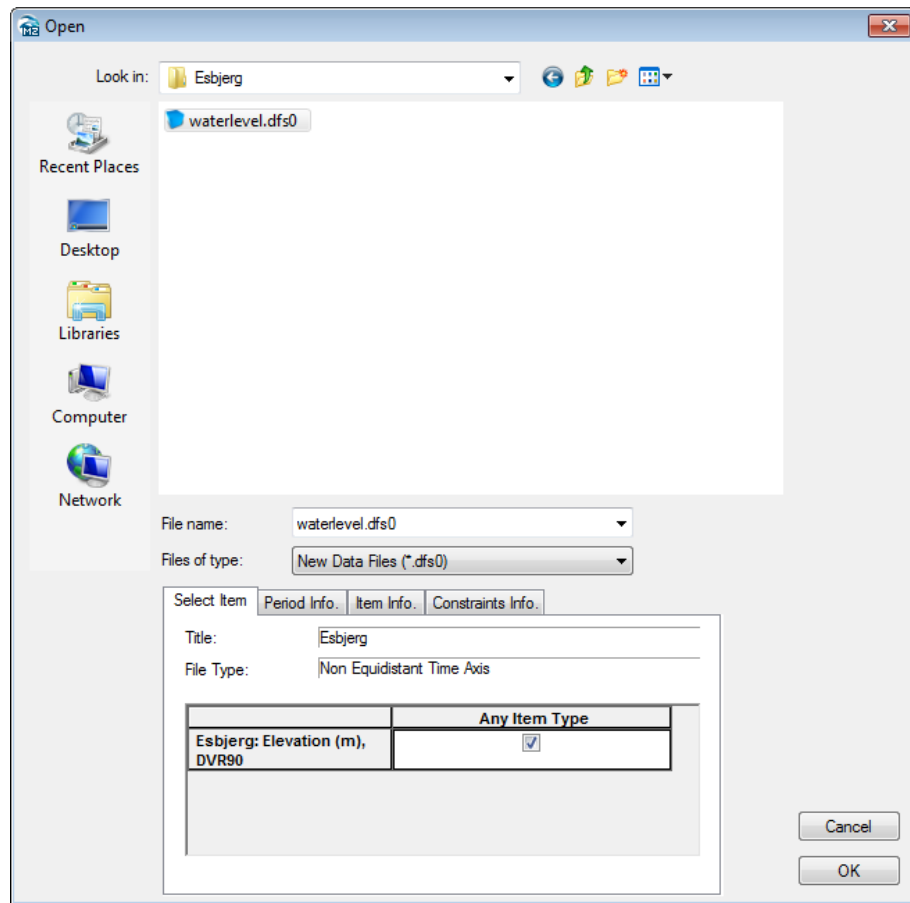


Figure 3.9 Selection of time series files in the Plot Composer

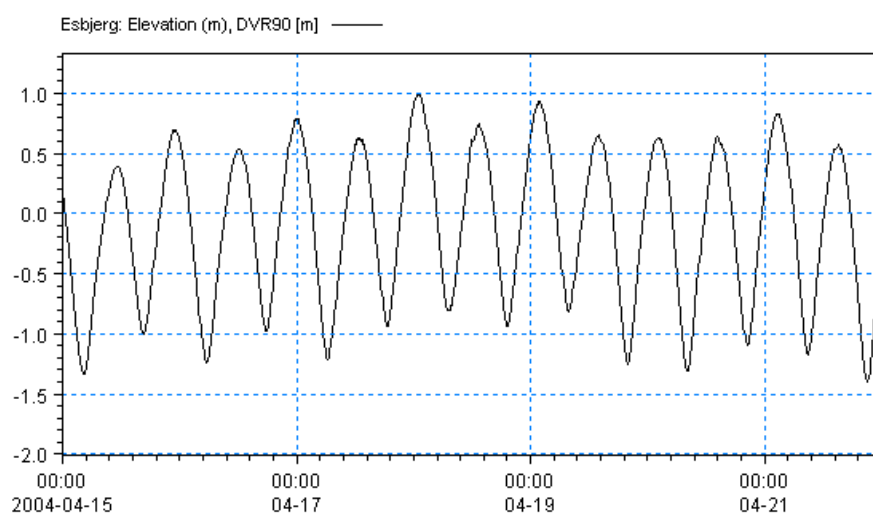


Figure 3.10 Time series of water levels in the Esbjerg Harbour as it can be viewed with the Plot Composer tool

### 3.1.2 Creating boundary conditions

The next step is to create the boundary water level time series from the generated time series in Esbjerg Harbour. Open the MIKE Zero Toolbox time series interpolation tool: New → MIKE Zero Toolbox → Time series → Interpolate Time Series, see Figure 3.11 and Figure 3.12.

**Please note** that time series must have equidistant time steps in the present version of MIKE 21 & MIKE 3 Flow Model FM. That means if the raw data has a Non-equidistant Calendar Axis, it must be converted to Equidistant Calendar Axis first before applying it as model input. Therefore we start by creating a time series similar to the one we created in Section 3.1.1, but with Equidistant Calendar Axis.

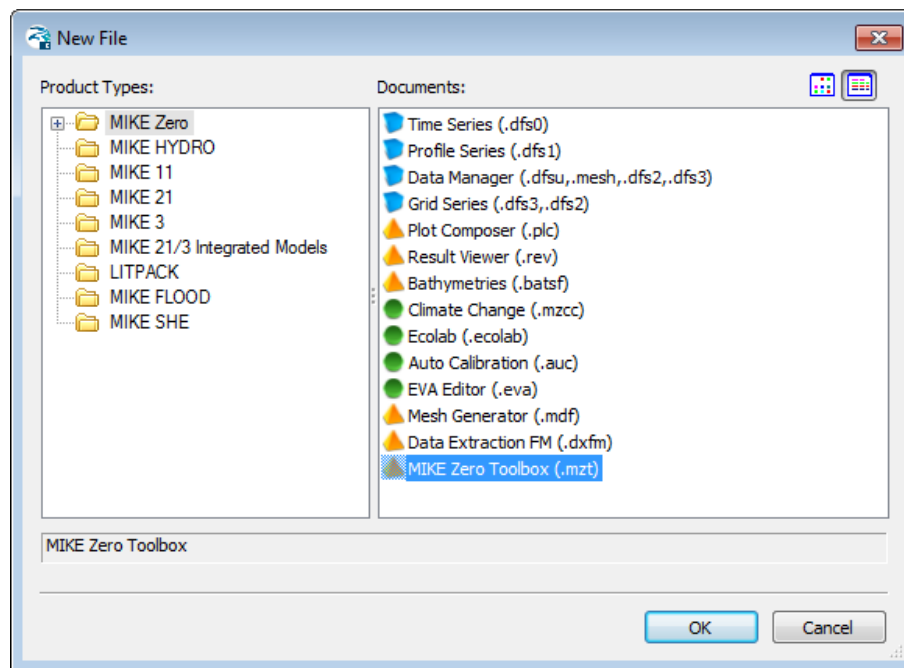


Figure 3.11 Starting the MIKE Zero Toolbox in MIKE Zero

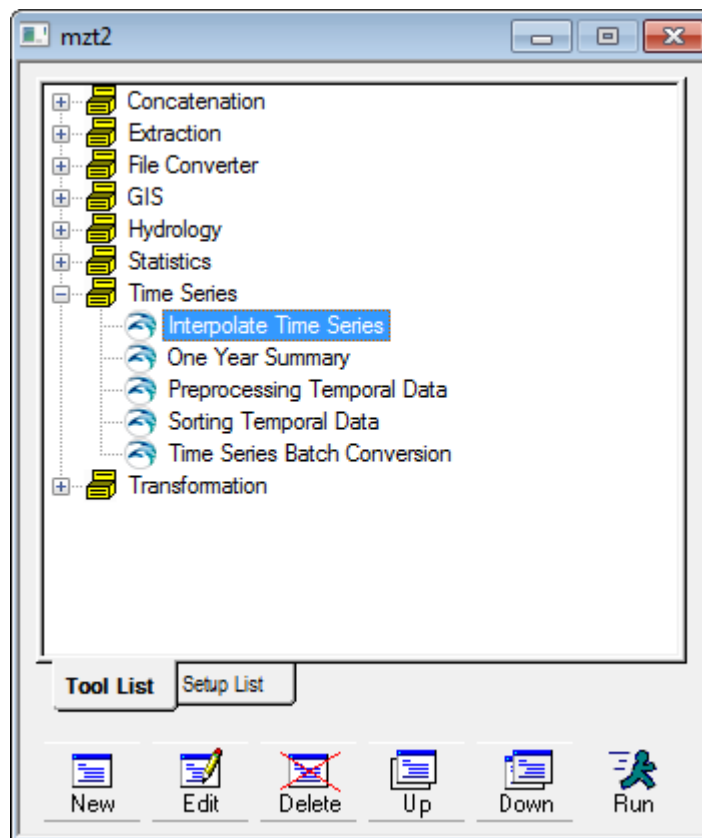


Figure 3.12 Starting the Tool for Interpolating Time Series

Follow the tutorial and specify the input file name: waterlevel\_esbjerg.dfs0 on the first page. On the next page choose 'Linear' interpolation method, and select a time interval of 1800 seconds.

Finally, specify output file name for the new interpolated time series file. Save it as:

'.\Ho\_Bay\Data\Boundary\_Conditions\waterlevel\_west.dfs0'

On the final page you click  , and the new time series with Equidistant Calendar Axis will be generated.

Open waterlevel\_west.dfs0. Find the average value for the simulation period:

15-4-2004 to 22-4-2004

Go to: Tools → Statistics. Specify the period in the Sub-series dialogue. Go to the Statistics dialogue and read the mean value: -0.118635 m, see Figure 3.13.

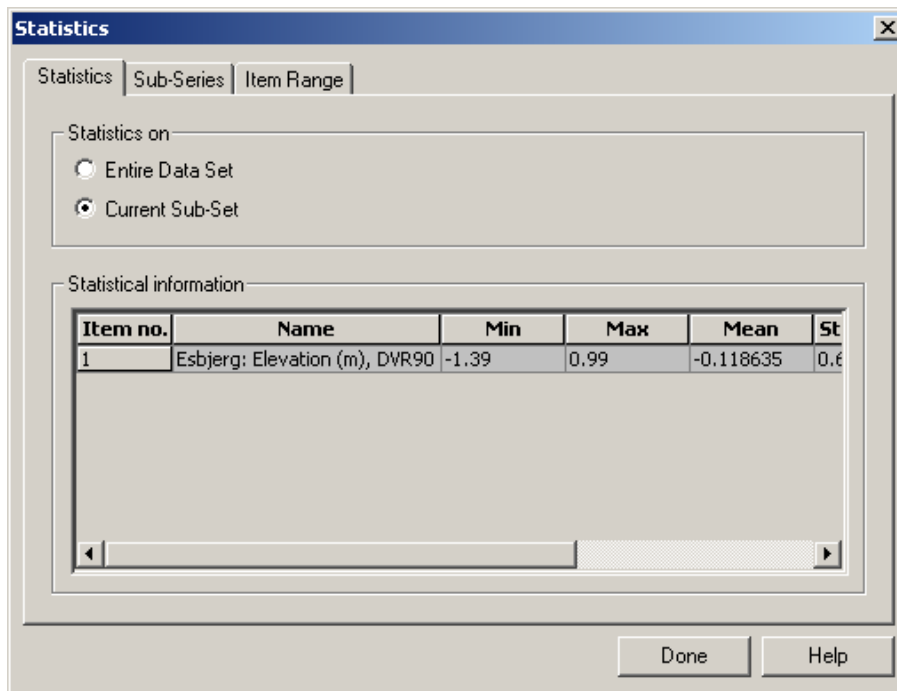


Figure 3.13 Statistics Tool for Time Series: Reading mean value of water levels in Esbjerg Harbour in the period 15-4-2004 to 22-4-2004: -0.12 m (-0.118635 m)

Go to: Tools → Calculator

Add 0.118635 m to the time series, see Figure 3.14.

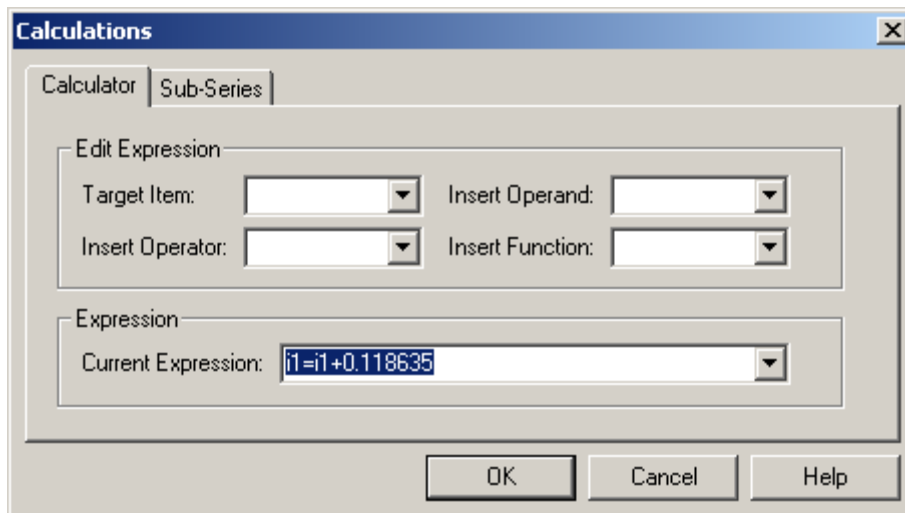


Figure 3.14 Calculator Tool for Time Series: Adding 0.118635 to the Time Series

Next you should use the calculator to multiply the time series by 1.1:

$i1=i1*1.1$

Afterwards you should subtract 0.118635 m again with the calculator.

Finally, the time series should be phase-shifted by ½ hour. You do that by marking row 1 (4/1/2004 12:30) to row 2928 (6/1/2004 00:00) with the mouse. Then type Ctrl+C and click row 0 (4/1/2004 00:00), type Ctrl+V.

The time series has now been adjusted and can be used as boundary condition for the western boundary. The difference between the measurements in Esbjerg Harbour and the manipulated time series at the boundary can be seen in Figure 3.15.

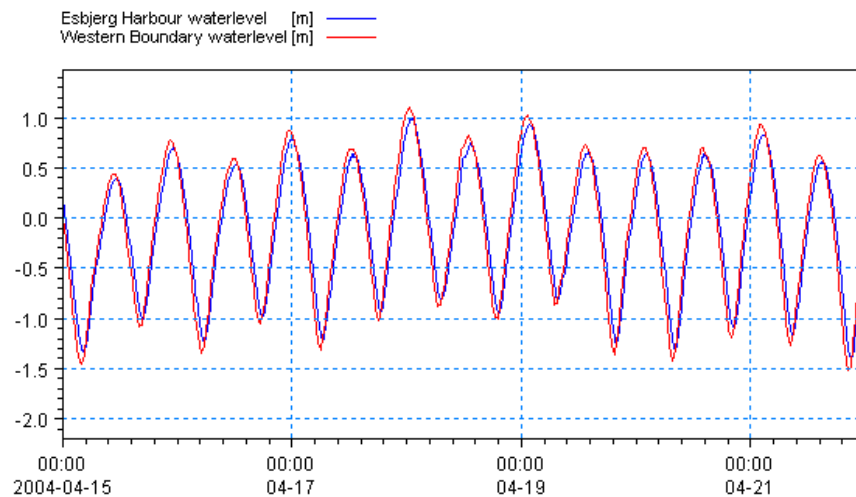


Figure 3.15 Comparison of water levels at Esbjerg Harbour and at the Western Boundary

### 3.2 Initial Conditions

The initial surface level is estimated the same as the measured in Esbjerg Harbour at the beginning of the simulation. Check the measured time series file for Esbjerg Harbour. We will use 0.17 m as constant value for the whole domain as initial condition for the water levels.

### 3.3 Wind Forcing

Wind recordings from Esbjerg Harbour will form the wind forcing as time series constant in space. Load the Time Series Editor and import the ASCII file 'wind\_esbjerg.txt' (Figure 3.16) with the equidistant calendar axis. Save the file in 'wind\_esbjerg.dfs0'. Time series of the wind speed and direction are shown in Figure 3.17.

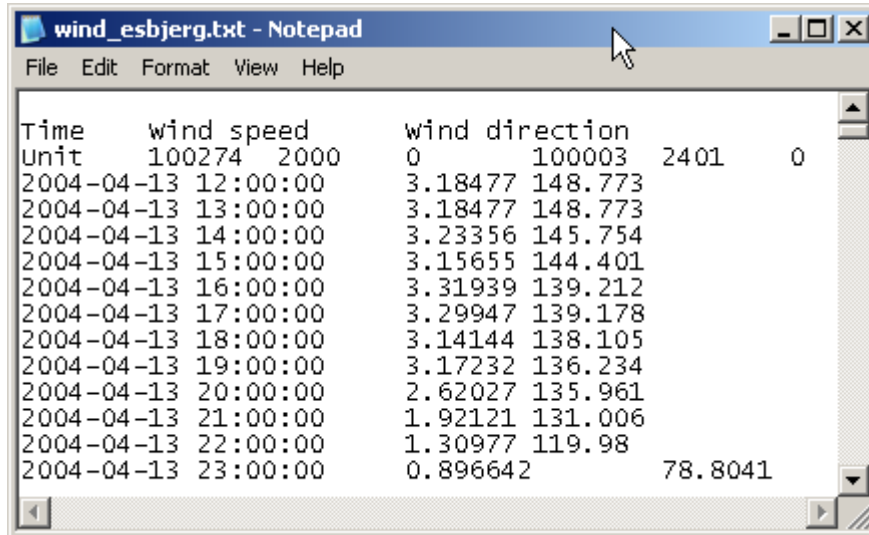


Figure 3.16 ASCII file with wind speed and direction from Esbjerg Harbour

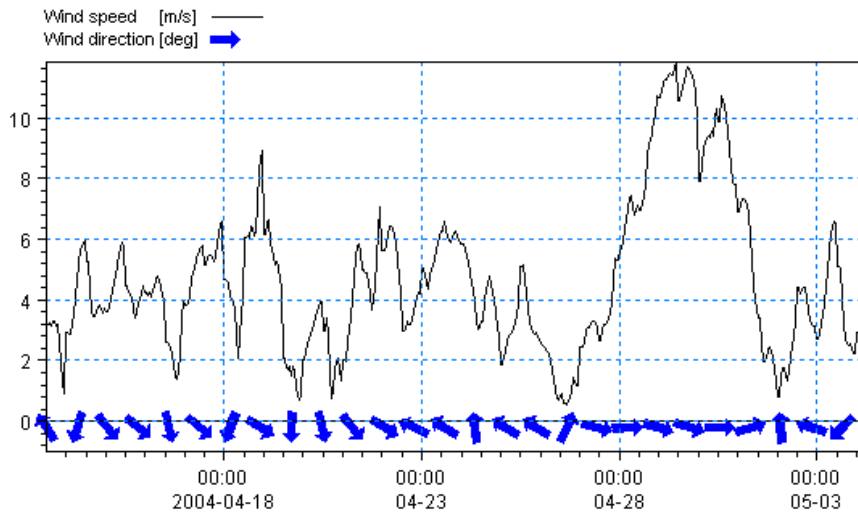


Figure 3.17 Wind speed and direction from Esbjerg Harbour, as it can be illustrated in the Plot Composer (Time Series Direction plot control)

A more descriptive presentation of the wind can be given as a wind speed (or wind rose) diagram. Start the 'Plot Composer' insert a new plot object, and select 'Wind/Current Rose Plot' and then select properties and select the newly created file 'wind\_esbjerg.dfs0' and change the properties of the plot as you prefer with respect to appearance (colours, etc.). The result is shown in Figure 3.18.

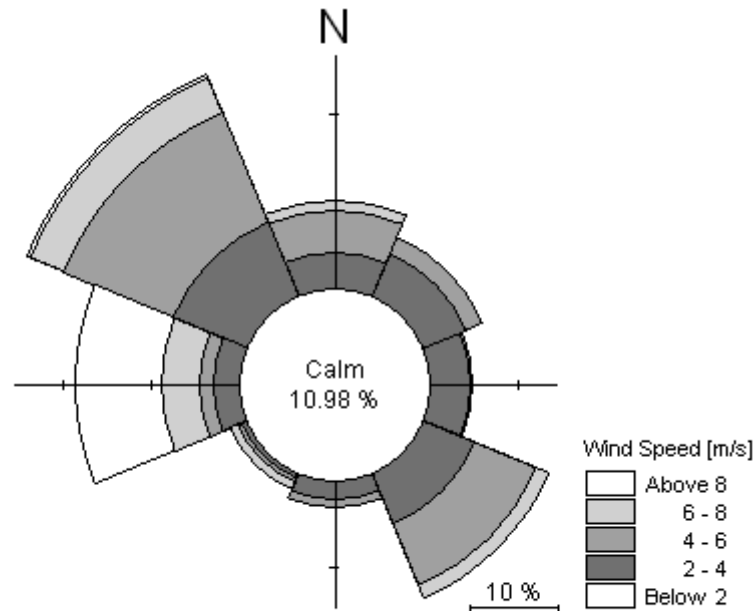


Figure 3.18 Wind rose from Esbjerg Harbour as it can be illustrated in the Plot Composer. The dominating wind direction is northwest

### 3.4 Generate Manning Number Data

From a previous MIKE 21 modelling study, it has been suggested that the Manning Number should have a depth dependent distribution in the Grådyb tidal area. So the Manning Number has the value  $35 [m^{1/3} \cdot s^{-1}]$  in the deeper parts of the Bay and  $45 [m^{1/3} \cdot s^{-1}]$  in the rest.

In this example, all the data are given as an ASCII file (manning\_number.txt) containing data, which shall be imported to a dfs2 file with the Grid Series Editor (see Figure 3.19, Figure 3.20, Figure 3.21 and Figure 3.22):

New → Grid Series → From Ascii File → Select file name

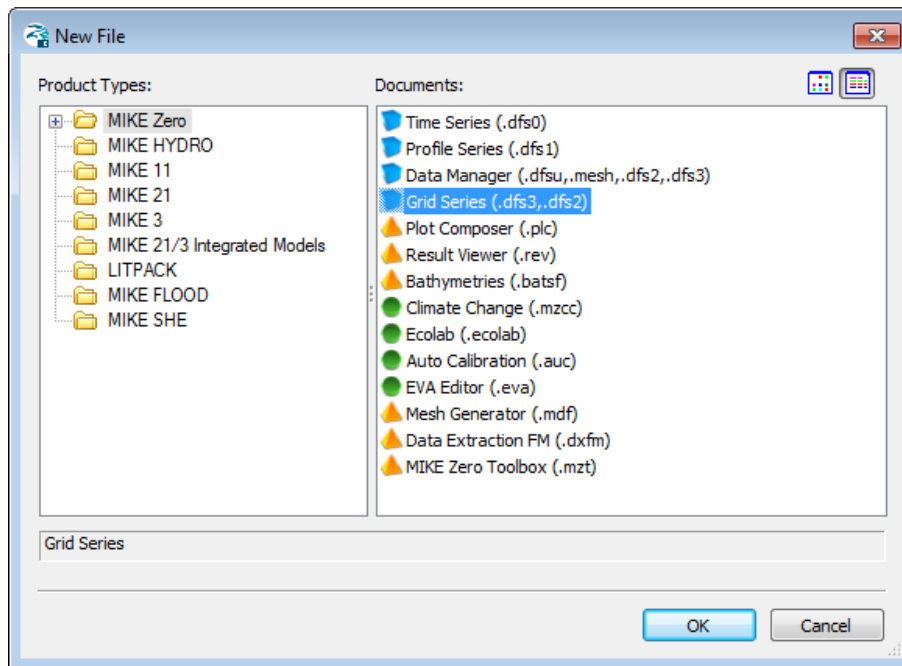


Figure 3.19 Starting to create a new Grid series

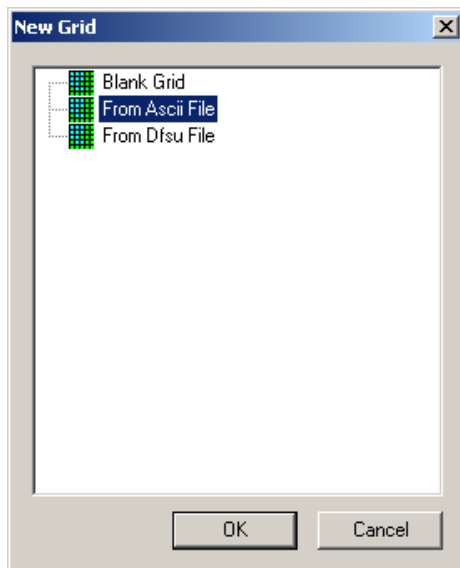


Figure 3.20 Creating dfs2 file from an ASCII file



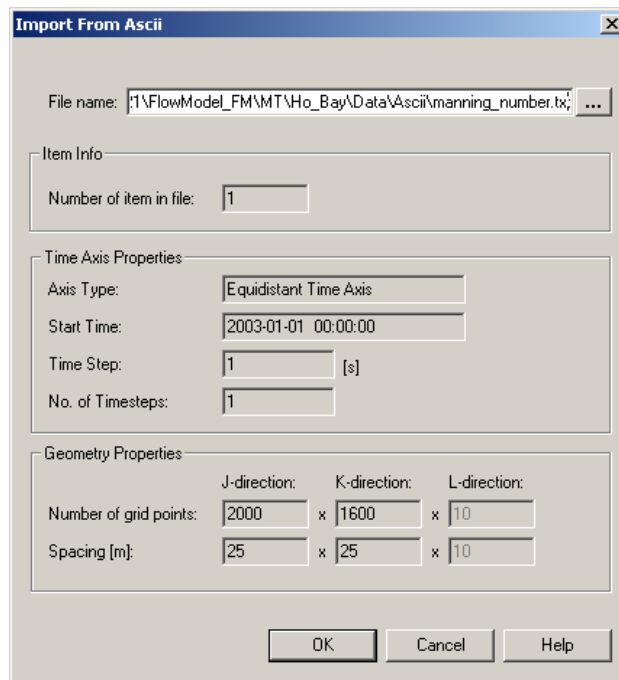


Figure 3.21 Specifying the ASCII file to import for creation of dfs2 file

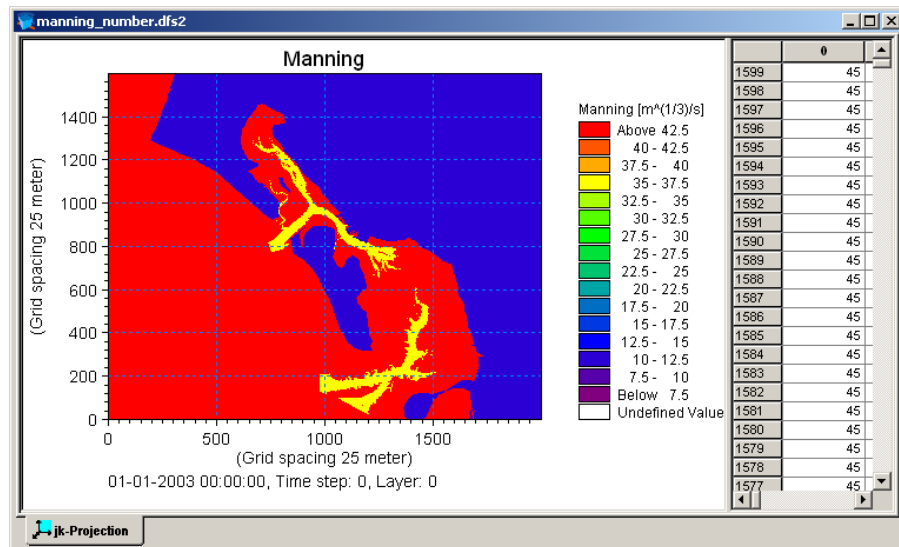


Figure 3.22 Manning number distribution in the Grådyb tidal area, given in the file manning\_number.dfs2

## 3.5 Generate Source Data

### 3.5.1 Hydrodynamic source discharges

There are 3 sources in the Grådyb tidal area model domain: water discharge from Varde River and also sediment discharge at two artificial dumpings. The best thing is to get the source data as frequent measurements of discharge, or as model results of the stream in question, but sometimes no data exists and you have to make estimates of the discharges in a stream or other source. In this case there are measurements of the discharge flows in Varde River. The raw data is supplied in the ASCII file:

varde\_river\_discharge.txt

which can be found in the folder:

.\Ho\_Bay\Data\Ascii

Generate the source file with the Time Series Editor by importing the ASCII files and saving it as:

.\Ho\_Bay\Data\Sources\varde\_river\_discharge.dfs0

The annual cycle of flow in Varde River is illustrated in Figure 3.23.

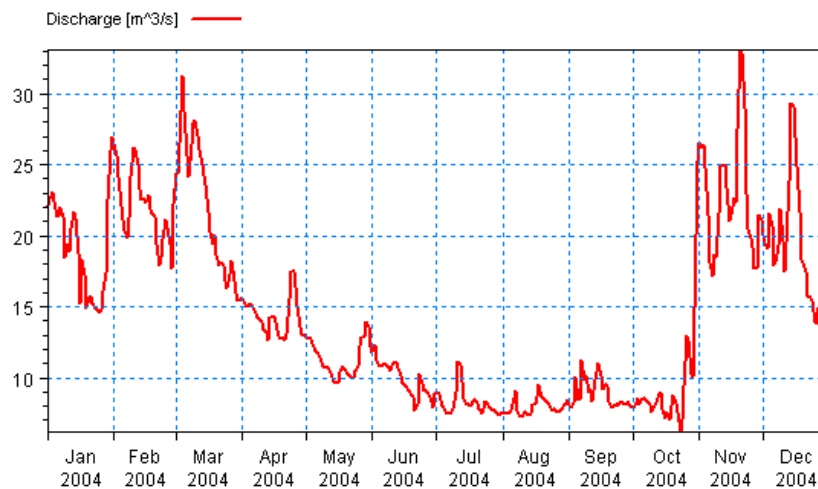


Figure 3.23 Annual cycle of discharge in Varde River in year 2004

The discharge from the sediment dumping discharge is fictive, because the loading of mass is actually dry without a real hydrodynamic discharge. We have chosen to use a discharge of 0.001 m<sup>3</sup>/s.

### 3.5.2 Suspended sediment source concentrations

The concentration in Varde River of sediment in suspension must also be specified. No measurements are available, so we will use a constant concentration  $0.01 \text{ kg/m}^3$ .

Based on the data for M and discharge Q (see Section 3.5.1), the sediment concentration C can be estimated as:  $C=M/Q$ .

The mass flux M of 'Fraction 1 mud concentrations' in the dumping sources is measured. For dumping location E and F the data is available in the ASCII file:

.\Ho\_Bay\Data\Ascii\dumping\_mass.txt.

Import the ASCII file with the Time Series Editor. Save the file as:

.\Ho\_Bay\Data\Sources\sediment\_dumping.dfs0.

Go to: Tools → Calculator

Divide the mass flux with the discharge:  $0.001 \text{ m}^3/\text{s}$ . Now the discharge  $[\text{m}^3/\text{s}]$  and the mass flux have been used to estimate a corresponding concentration  $[\text{kg}/\text{m}^3]$  in the file sediment\_dumping.dfs0.

The time series of the mass flux from the dumping E and F can be seen in Figure 3.24.

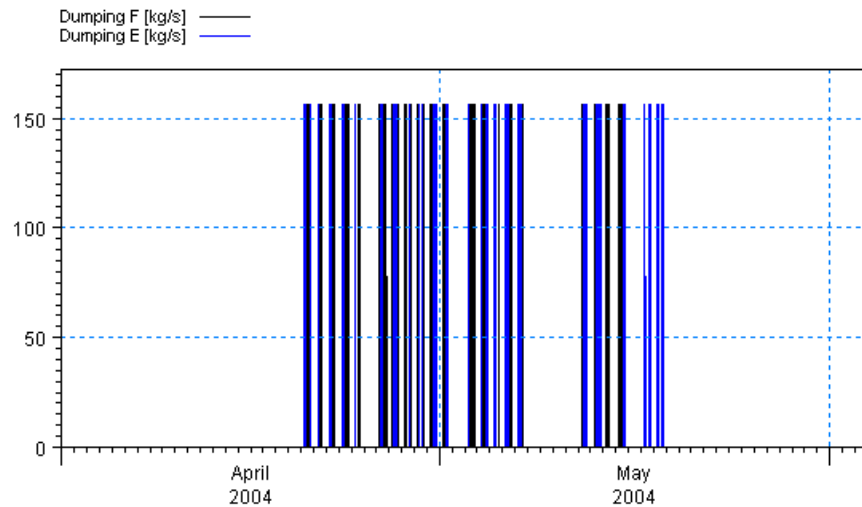


Figure 3.24 The time series of mass flux of dumped sediment [kg/s] at location E and F

## 3.6 Generate Mud Transport Model Constant Parameters

The Mud Transport model is flexible with respect to how many mud fractions and sediment layers the model shall consist of.

The choice of number of mud fractions is based on considerations regarding

1. computation time
2. user wishes for the specific fractions
3. available data

In this example, the data is only supporting 1 fraction, and there is not a user wish to simulate specific fractions, so only 1 fraction will be applied.

The choice of number of sediment layers is based on considerations regarding:

1. computation time
2. length of simulation and how dynamic the morphology in the area is
3. available data

In this example, the model will consist of 1 fraction of mud and 2 sediment layers. The top layer of the sediment (Layer 1) is considered to be soft mud, and Layer 2 is considered to be more consolidated mud. When only simulating 1 week it is assumed that 2 layers are enough.

### 3.6.1 Critical shear stress for consolidated mud in layer 2

The critical shear stress is an important factor that influences the resuspension of sediment. The value for critical shear stress depends on how consolidated and dense the sediment is. It is a parameter for calibration. From another study data exists that contain the spatial distribution of critical shear stress in layer 2 in the Grådyb tidal area.

Generate a dfs2 file from the ASCII file:

critical\_shear\_stress\_layer2.txt

with the Grid Series Editor, and save the file as:

critical\_shear\_stress\_layer2.dfs2

This generated dfs2 file can be used directly in this study, see Figure 3.25.

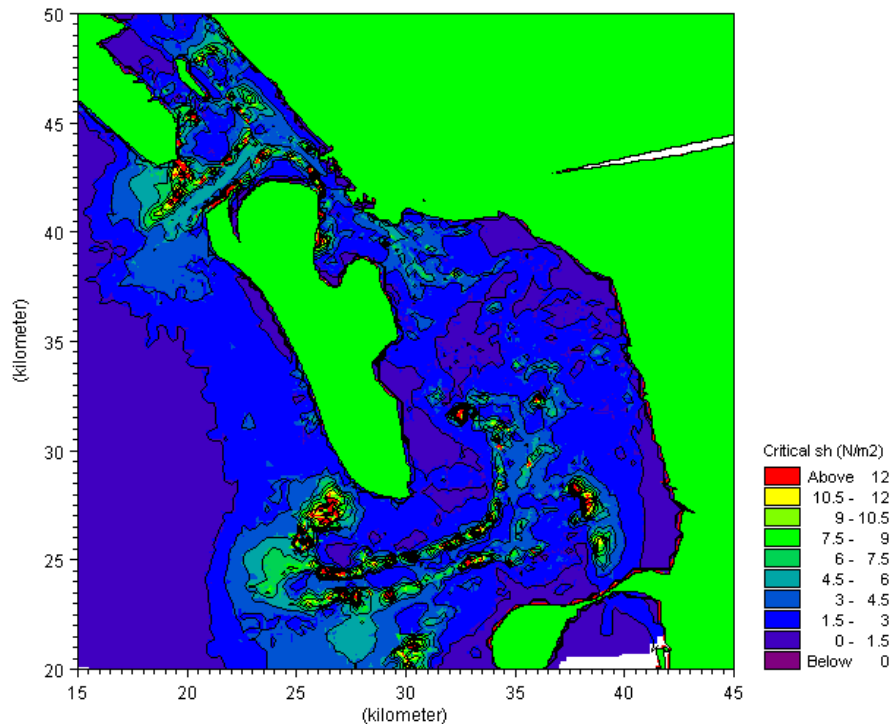


Figure 3.25 Critical Shear stress for Layer 2 sediments [ $\text{N/m}^2$ ]

### 3.7 Generate Wave Data

Waves can have an impact on the sediment transport in an area. There are two methods for inclusion of waves in the simulation. Both require that you have a result file from a wave model of the area. It is required that the wave result file contains data of wave heights, wave periods, and wave directions. You can directly specify a result file of a MIKE 21 SW (Spectral Wave) model of the area, or you can analyse the wave result file regarding wind and water level and generate a database with pictures of typical wave patterns for different wind situations and different water levels. The first method means that during simulation the wave result file is read. The second method requires that the area is dominated by wind-waves, and that the waves respond quickly to a change in water level, wind speed and direction. The latter method will save hard disk space, because it will interpolate the typical pictures of wave variables with respect to actual wind conditions and water levels during simulation.

In this example we will use the database method.

Run the wave model with artificial wind covering 4 directions and a wind speed of 5 m/s and 15 m/s. Run with static boundary conditions, so that a situation with water levels around  $-1$  m is obtained. For each situation you should save an output file. Repeat the exercise with a water level of 2 m.

### 3.7.1 Wave database

This exercise has already been done and the 16 wave output files are ready to use in the example, see the list of files in Table 3.2.

Table 3.2 List of wave output files for different wind direction, wind speeds, and water levels

Wind direction	Wind speed [m/s]	Water level [m]	File name
N	5	-1	n5--1.dfsu
N	5	2	n5-2.dfsu
N	15	-1	n15--1.dfsu
N	15	2	n15-2.dfsu
E	5	-1	e5--1.dfsu
E	5	2	e5-2.dfsu
E	15	-1	e15--1.dfsu
E	15	2	e15-2.dfsu
S	5	-1	s5--1.dfsu
S	5	2	s5-2.dfsu
S	15	-1	s15--1.dfsu
S	15	2	s15-2.dfsu
W	5	-1	w5--1.dfsu
W	5	2	w5-2.dfsu
W	15	-1	w15--1.dfsu
W	15	2	w15-2.dfsu

Each picture (dfsu file) illustrates typical wave patterns that can be expected at specific wind and water level conditions, see Figure 3.26.

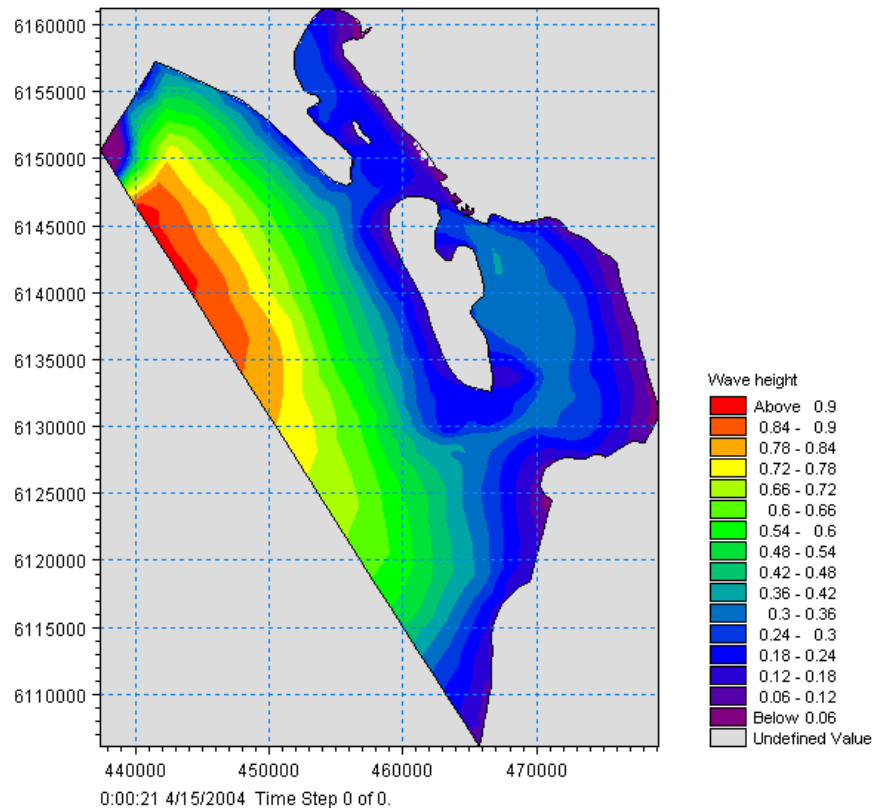


Figure 3.26 Example from database of wave height in situation with easterly wind of 5 m/s and a water level of 1 m

### 3.8 Generate Mud Transport Model Initial Conditions

The initial conditions are very important in a short simulation as used in this example. At the same time they are difficult to measure and estimate correctly, and therefore, a difficult part of a Mud Transport Model to define. The following iterative approach is often used to estimate the initial conditions:

1. The initial conditions are first based on sedimentological maps of the area.
2. Afterwards some test model simulations indicate where deposition can take place.
3. From these results the maps are adjusted.

This exercise has already taken place in another study of the area. A first guess of a set of initial condition files are therefore already supplied for the first calibration simulation with this example as dfs2 files:

1. Fraction 1 mud concentration: mt\_version1.dfs2 (item 1)
2. Layer 1 thickness: mt\_version1.dfs2 (item 2)
3. Layer 2 thickness: mt\_version1.dfs2 (item 3)

Initial conditions must be specified for 'Fraction 1 mud concentrations', and 'Thickness of Layer 1 and Layer 2'.

The supplied dfs2 files are shown in Figure 3.27, Figure 3.28, and Figure 3.29. The files can be generated from the corresponding ASCII files included in the example using the Grid Series Editor.

### Fraction 1

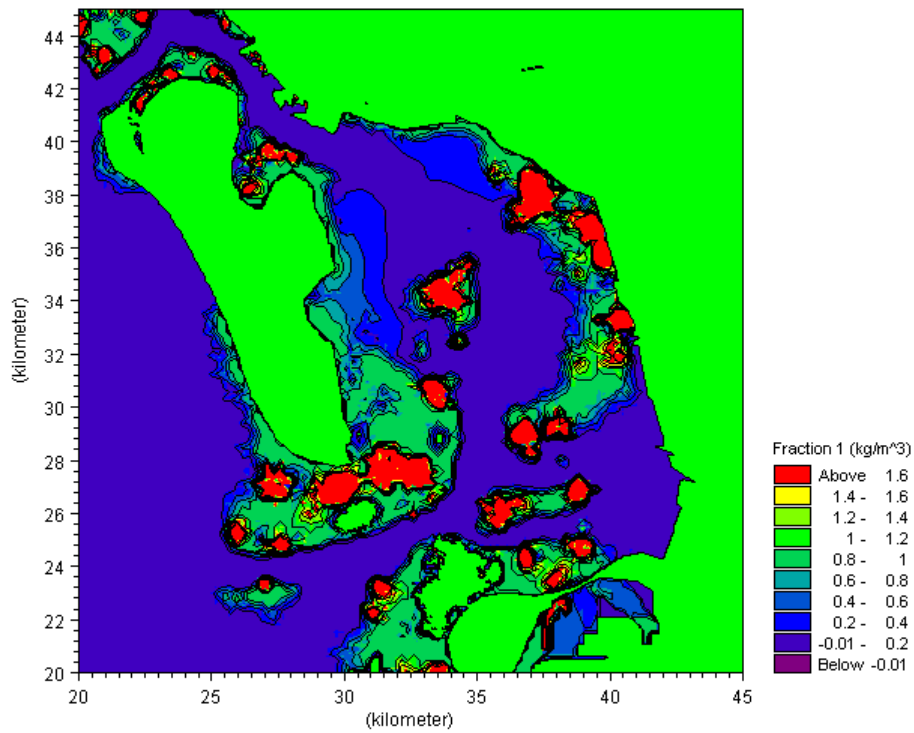


Figure 3.27 Initial condition for Fraction 1 mud concentrations [ $\text{kg/m}^3$ ]

### Layer 1

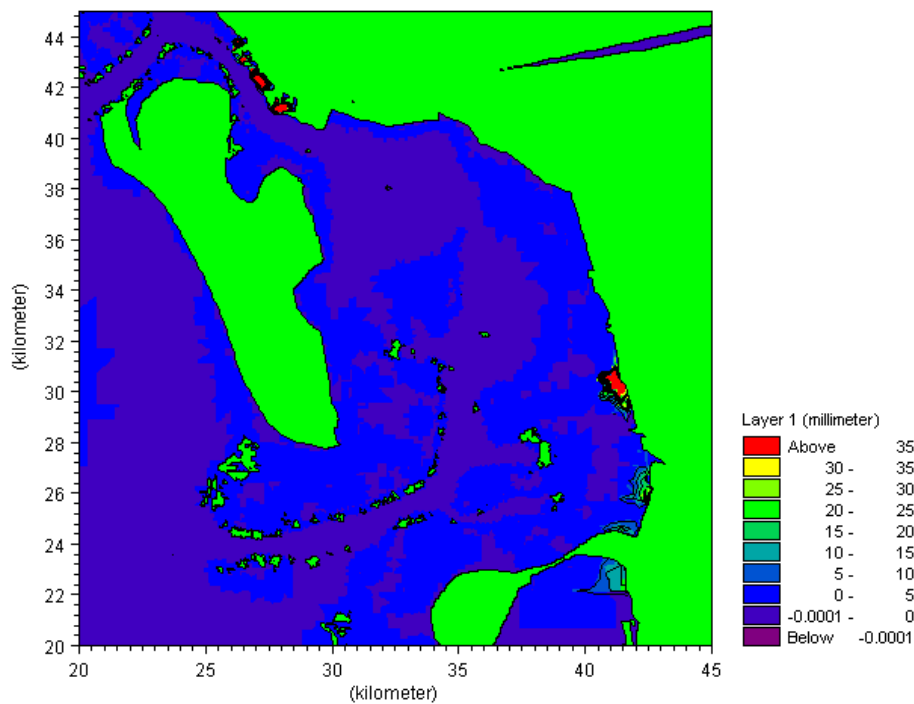


Figure 3.28 Thickness of Layer 1, initial condition given as dfs2 file [mm]



## Layer 2

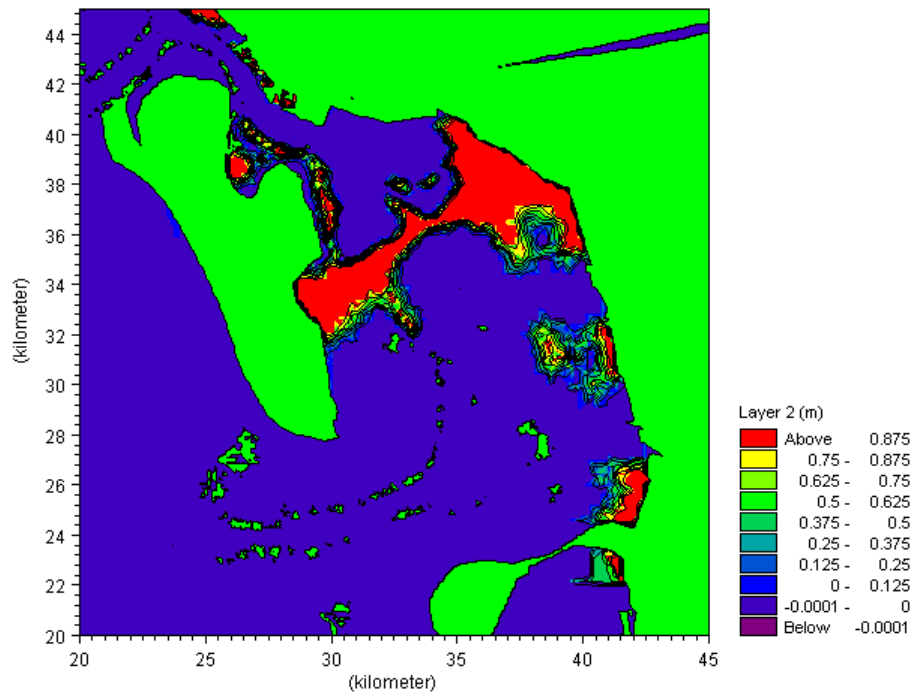


Figure 3.29 Thickness of Layer 2, initial condition given as dfs2 file [m] Set-Up of MIKE 21 Flow Model FM



## 4 Specifications for the MIKE 21 Flow Model FM

We are now ready to create the specification file for the Mud Transport modelling using the Grådyb tidal area mesh with year 2004 input data as generated in Chapter 3. Initially we will use the default parameters and not take into account the effect of the density variation at the boundaries. The set-up in the first calibration simulation consists of the parameters shown in Table 4.1.

Table 4.1 Specifications for the calibration simulation

Parameter	Value
Specification File	ho.m21fm
Mesh and Bathymetry	ho.mesh 3266 Nodes
Simulation Period	2004-04-15 00:00 – 2004-04-22 00:00 (7 days)
Time Step Interval	60 s
No. of Time Steps	10080
HD: Solution Technique	Low order, fast algorithm Minimum time step: 0.01 s Maximum time step: 60 s Critical CFL number: 0.8
HD: Enable Flood and Dry	Yes
HD: Initial Surface Level	0.2 m
HD: Wind	Varying in time, constant in domain: wind_esbjerg.dfs0
HD: Wind Friction	Constant: 0.001255
HD: North and West Water level Boundary	Type 0 data: waterlevel_west.dfs0
HD: Eddy Viscosity	Smagorinsky formulation, Constant 0.28 m <sup>1/3</sup> /s
HD: Resistance	Manning number. Varying in domain: manning_number.dfs2
MT: Number of Fractions	1
MT: Number of Layers	2
MT: Settling Settling velocity coefficient	5 m/s
MT: Erosion: Critical Shear Stress layer 2	Varying in domain: critical_shear_stress_layer2.dfs2
MT: Bed Roughness	0.001 m
MT: Initial Cond. Fraction 1	Varying in domain: mt_version1.dfs2, item 1
MT: Initial Cond. Layer 1	Varying in domain: mt_version1.dfs2, item 2
MT: Initial Cond. Layer 2	Varying in domain: mt_version1.dfs2, item 3
MT: Include waves Shear stress formulation	Yes, database specification Max Soulsby
Result Files	hd.dfsu esbjerg_hd.dfs0 rcm9_hd.dfs0 mt.dfsu rcm9_mt.dfs0
Simulation Time	About 7 minutes with a 2.2 GHz PC, 1'8 GB RAM, 4 cores

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

Additional information about the Mud Transport Module can be found in the MIKE 21 Flow Model FM Mud Transport Module User Guide in the Manuals folder.

## 4.1 Basic Parameters

Specify the bathymetry and mesh file `ho.mesh` in the Domain dialogue, see Figure 4.1. The projection zone has already been specified in the mesh as UTM-32. In the mesh file, each boundary has been given a code, see Figure 4.3. In this Grådyb example the West Boundary has 'Code 2' and the North Boundary has 'Code 3'. Rename the boundary 'Code 2' to 'West' and 'Code 3' to 'North' in the dialogue Boundaries window, see Figure 4.2.

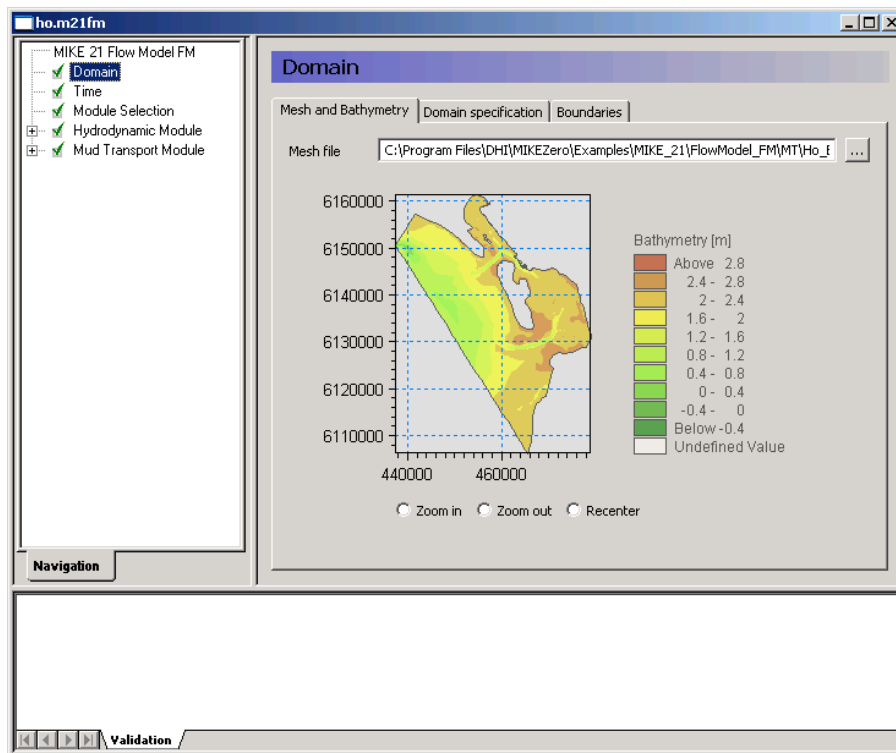


Figure 4.1 MIKE 21 Flow Model FM: Specify Domain

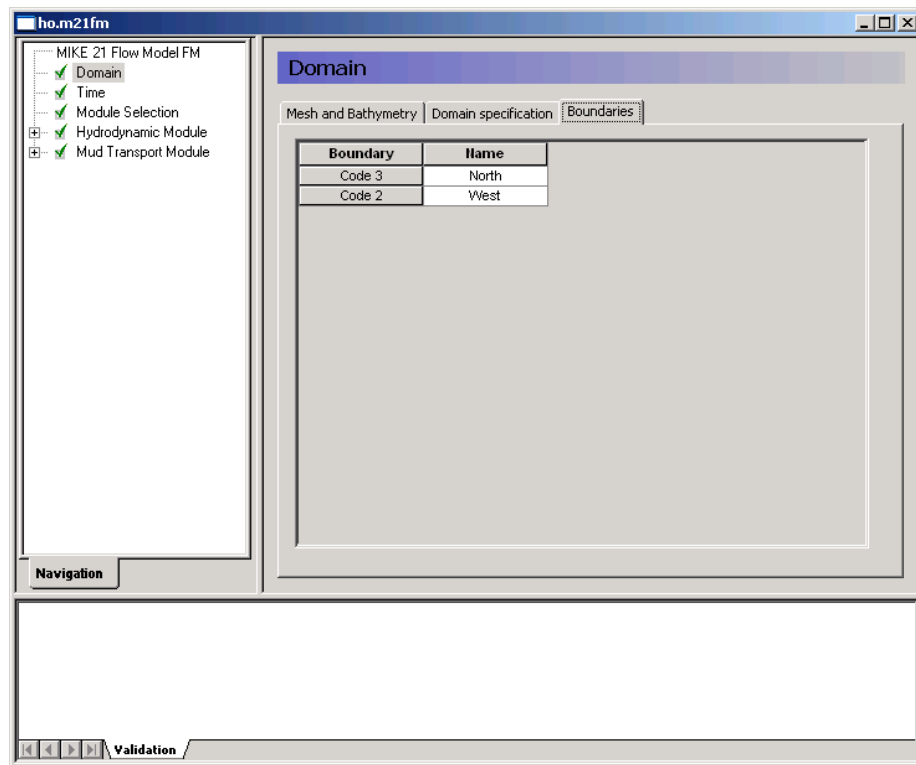


Figure 4.2 M21 Flow Model FM: Renaming boundaries

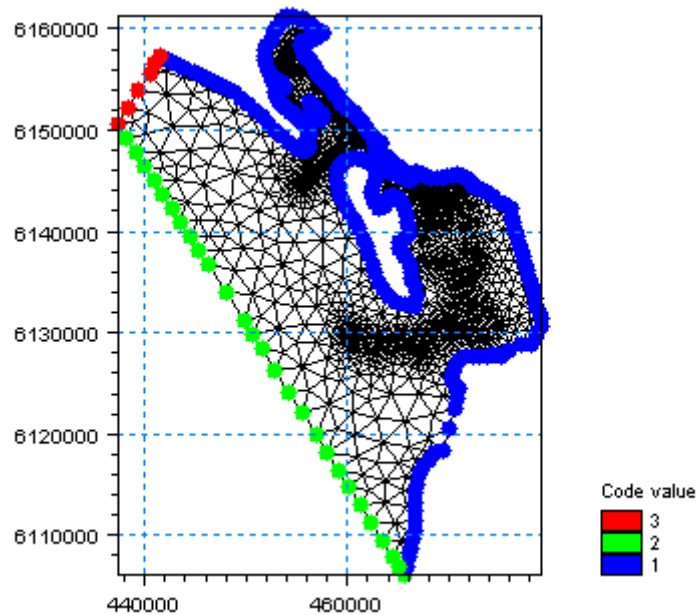


Figure 4.3 In the Data Viewer it is also possible to view the code values of the boundaries

Specify the overall time step in the 'Time' dialogue to 60 s. The time step range must be specified to 10080 time steps in order to simulate a total period of 7 days. See Figure 4.4.

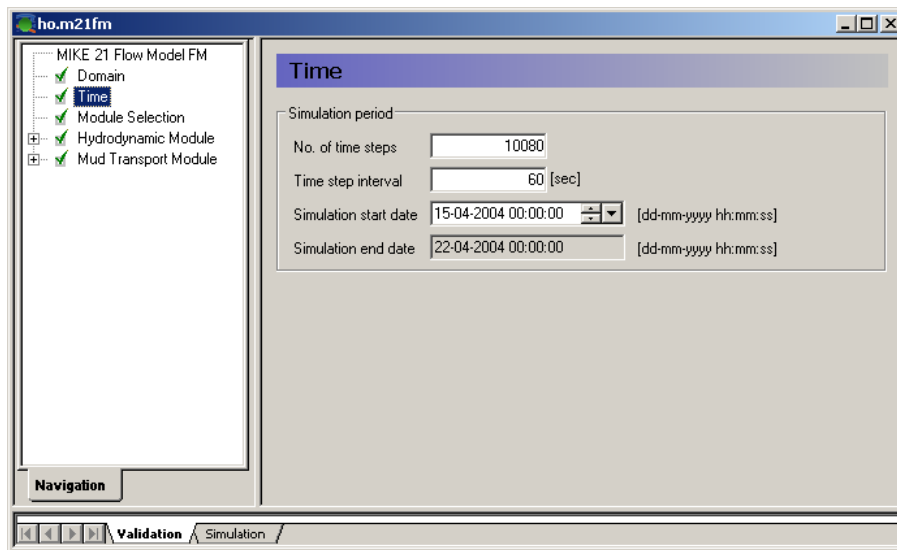


Figure 4.4 MIKE 21 Flow Model FM: Simulation period

Include the 'Mud Transport Module' in the 'Module Selection' dialogue, see Figure 4.5.

In this example, the Hydrodynamic Module and the Mud Transport Module will be used.

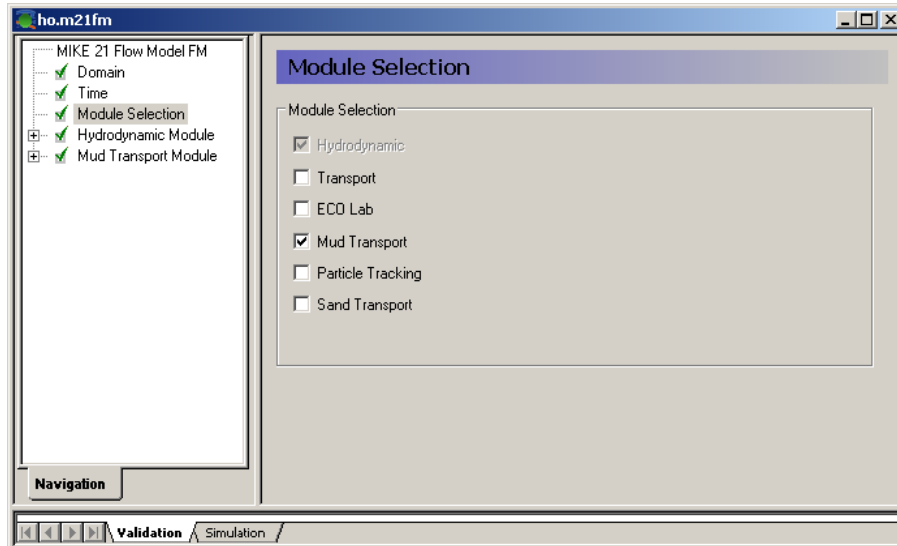


Figure 4.5 MIKE 21 Flow Model FM: Module Selection

## 4.2 Hydrodynamic Module

The shallow water equations are solved using 'Low order, fast algorithm' for both time integration and space integration, see Figure 4.6.

The minimum time step is set to 0.01 to ensure that the CFL number always is lower than the critical CFL number, here defined as 0.8. The maximum time step is set to the overall time step of 60 s.

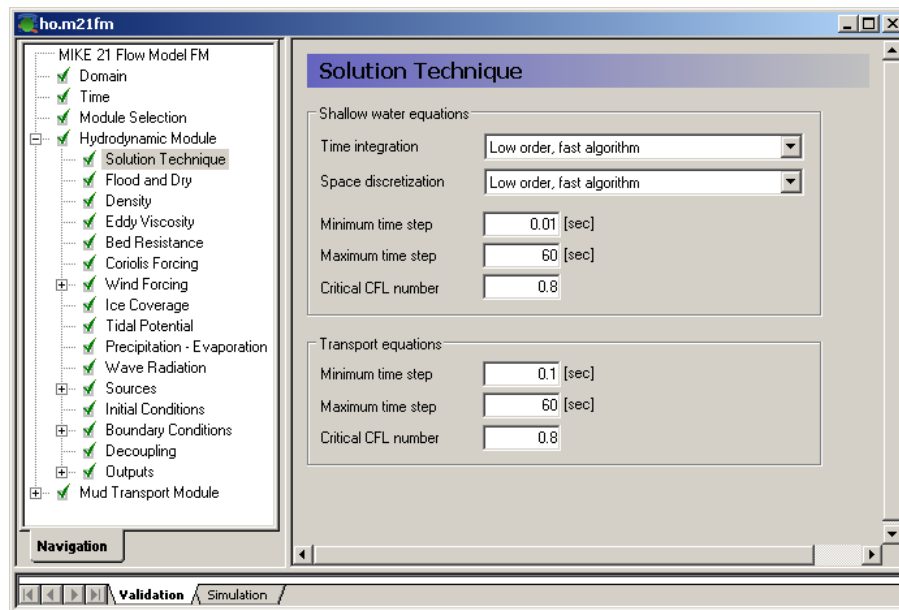


Figure 4.6 MIKE 21 Flow Model FM: Solution Technique

In this example the flooding and drying should be included, because many areas in the Grådyb tidal area will dry out during the simulation. If you choose not to include flooding and drying, the model will blow up in situations with dry areas.

In the 'Flood and Dry' dialogue it is possible to include flood and dry, see Figure 4.7. 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.

As the density variation is not taken into account in this example the density should be specified as 'Barotropic' in the 'Density' dialogue, see Figure 4.8.

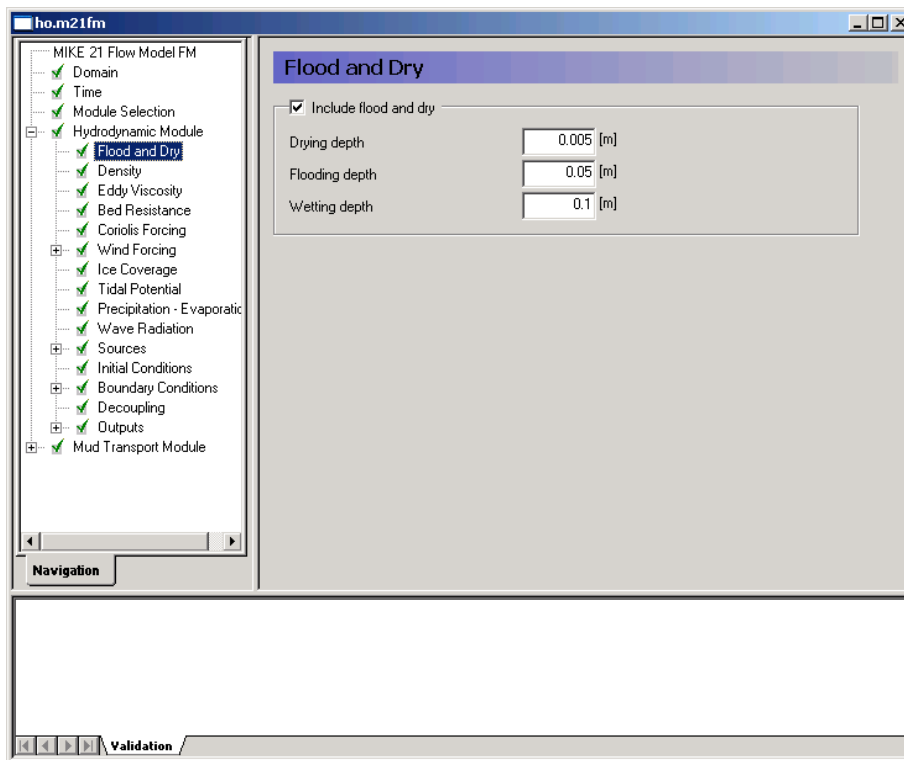


Figure 4.7 MIKE 21 Flow Model FM: Flood and Dry

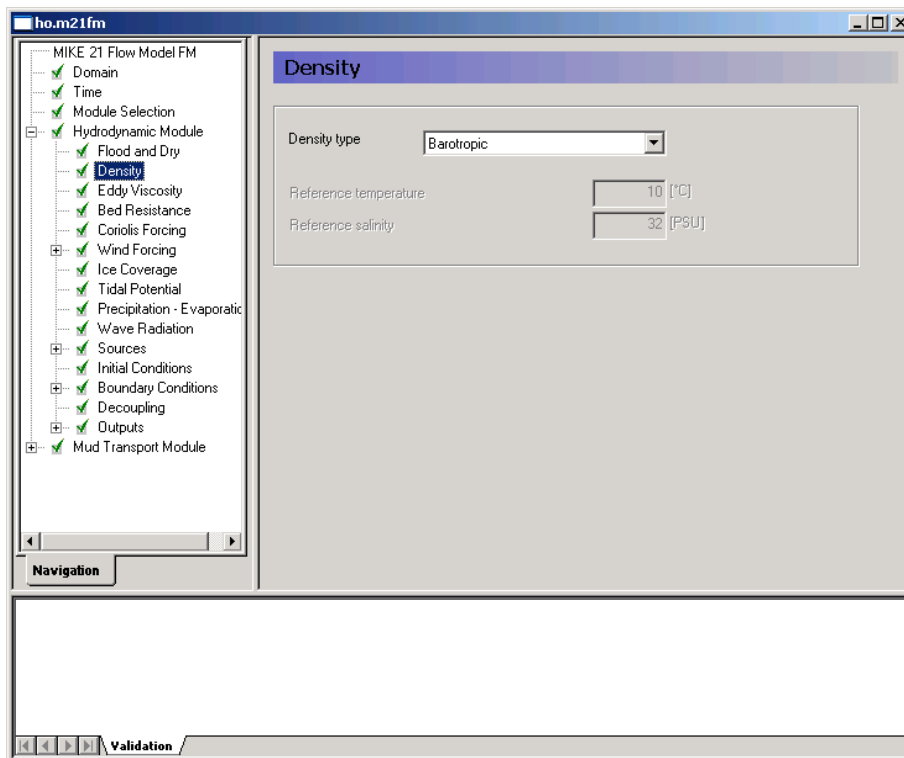


Figure 4.8 MIKE 21 Flow Model FM: Density specification



The default setting for the Horizontal Eddy Viscosity is a Smagorinsky formulation with a coefficient of 0.28, see Figure 4.9.

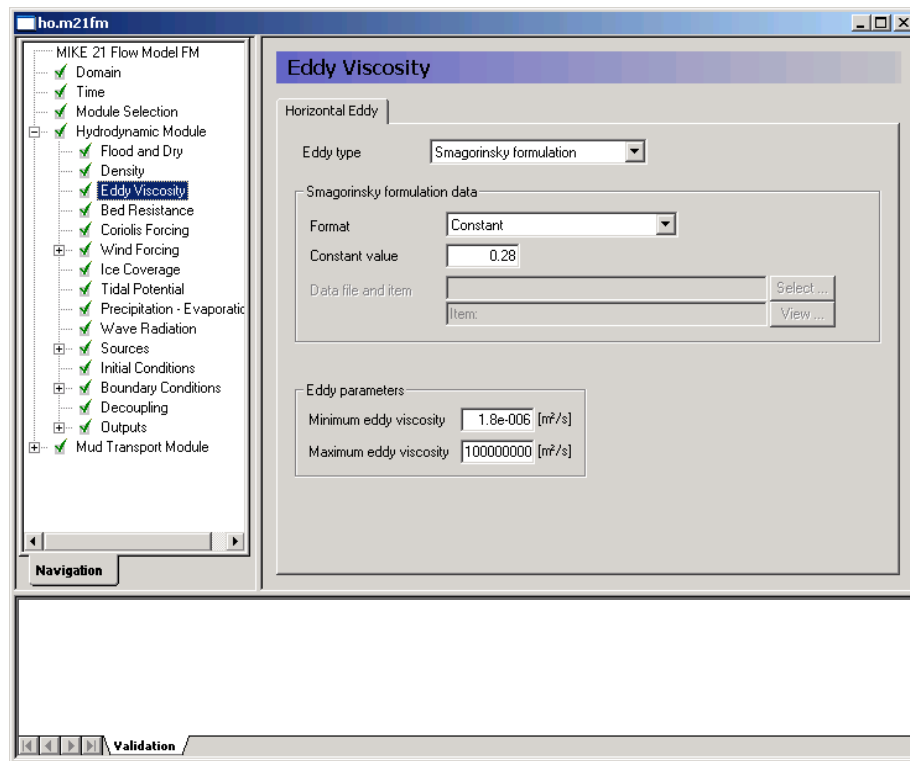


Figure 4.9 MIKE 21 Flow Model FM: Eddy Viscosity (Horizontal)

The Bed Resistance will be given as a map with Manning numbers varying in the domain, see Figure 4.10. Use the format 'Varying in domain' and specify the dfs2 file 'manning\_number.dfs2' that was generated earlier.

Coriolis is most often included in real applications. Only for laboratory type of simulations Coriolis is most often ignored, see Figure 4.11.

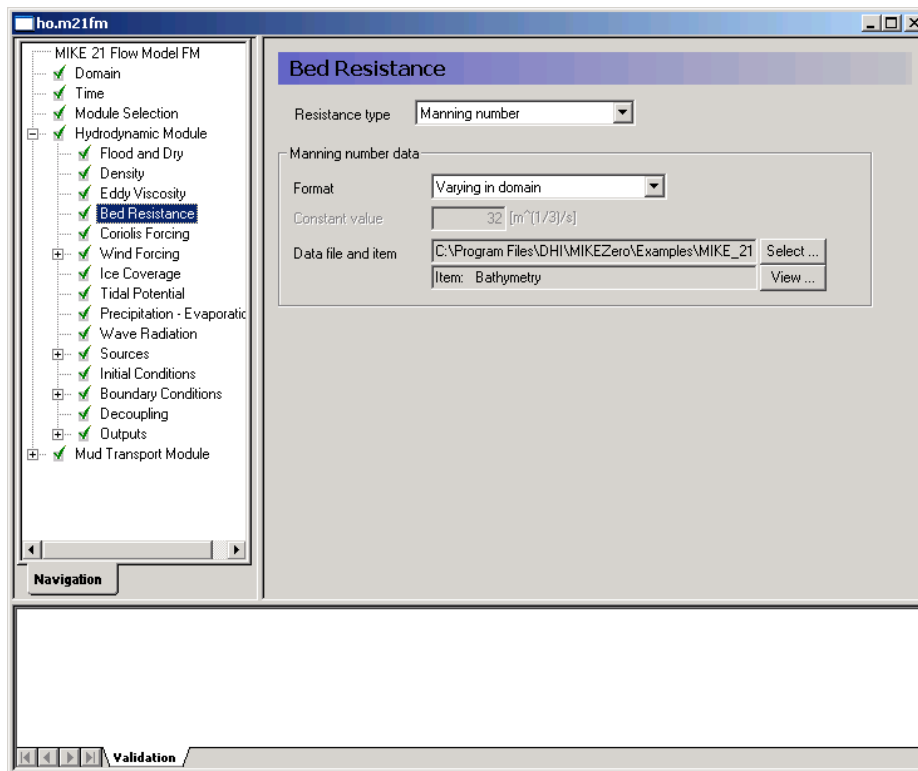


Figure 4.10 MIKE 21 Flow Model FM: Bed Resistance

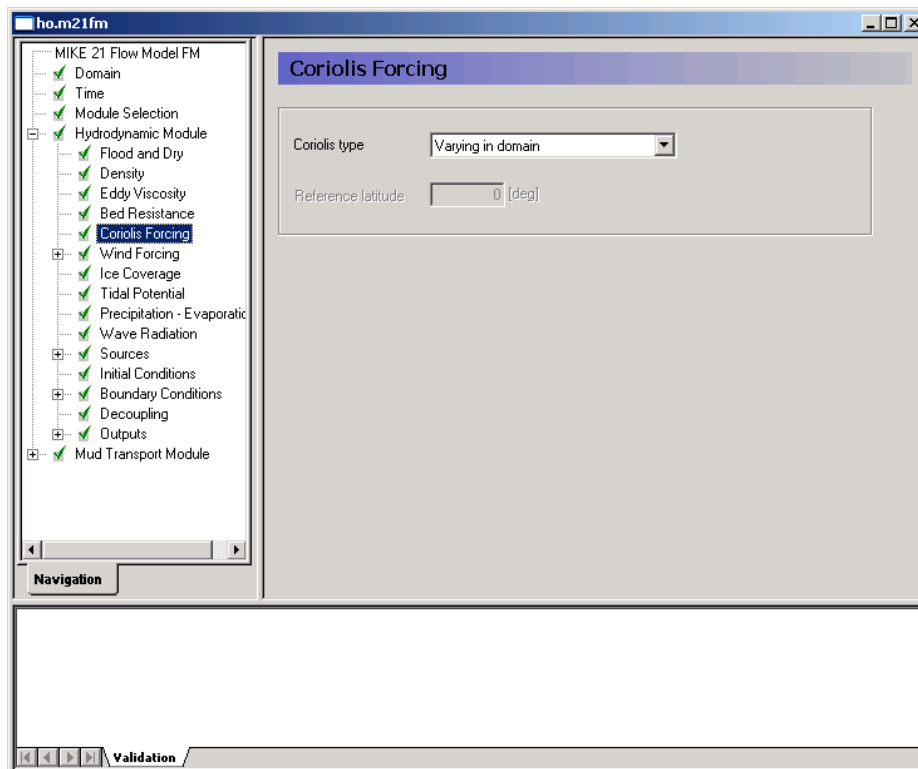



Figure 4.11 MIKE 21 Flow Model FM: Coriolis Forcing

To use the generated wind time series specify it as 'Variable in time, constant in domain' in the Wind Forcing dialogue, and locate the time series wind\_esbjerg.dfs0.

**Please note** that an easy way to see the wind data file is to simply click  in the Wind Forcing dialogue, see Figure 4.12.

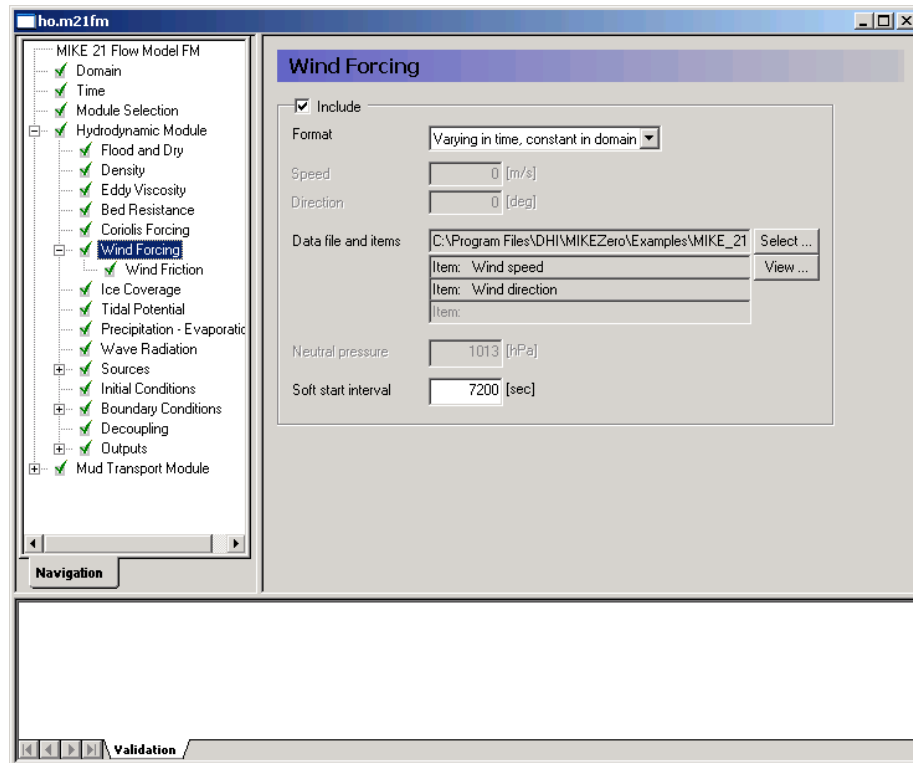


Figure 4.12 MIKE 21 Flow Model FM: Wind Forcing

It is a good practice to use a soft start interval. In our case 7200 s is suggested. The soft start interval is a period at the beginning of a simulation where the effect of the wind does not take full effect. At the beginning of the soft start period the effect of the specified Wind Forcing is zero and then it increases gradually until it has full effect on the model at the end of the soft start interval period. Specify the Wind Friction type as 'Constant' and use the default values for the Wind Friction: 0.001255. See Figure 4.13.

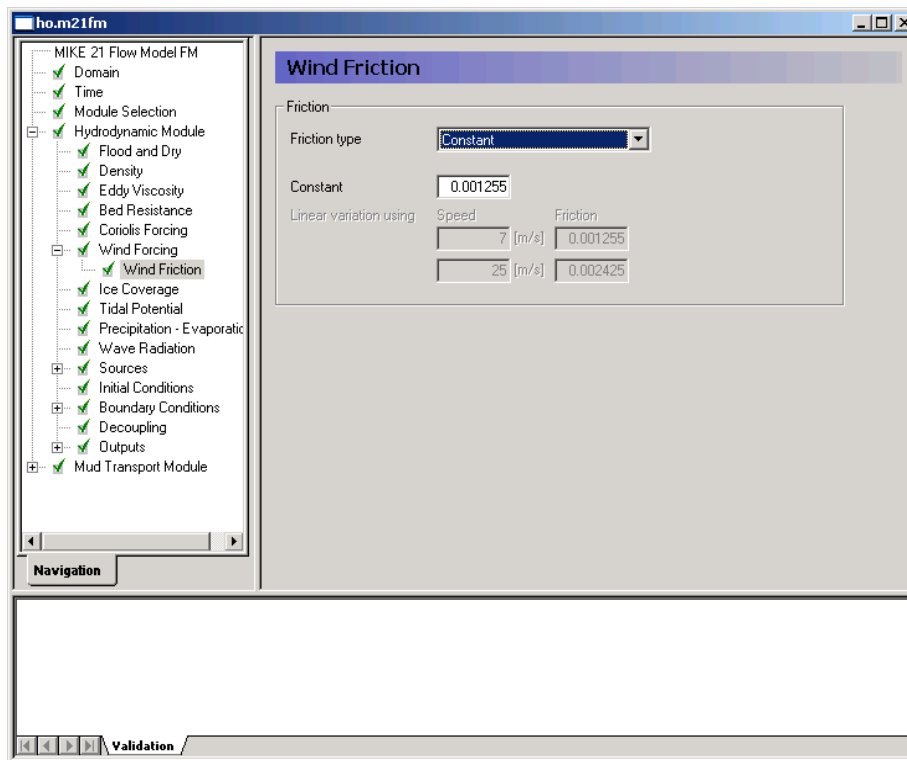


Figure 4.13 MIKE 21 Flow Model FM: Wind Friction

In this example

- Ice Coverage is not included
- Precipitation-Evaporation is not included
- Wave Radiation is not included
- Tidal Potential is not included

and their dialogue pages are not explained.

The discharge and velocity for each source and sink should be specified in the 'Sources' dialogue. There are 3 sources in this example: Varde River, Dumping E and Dumping F. Varde River is specified with a time series discharge, and the two dumping sources with constant discharges of 0.001 m<sup>3</sup>/s. See Figure 4.14.

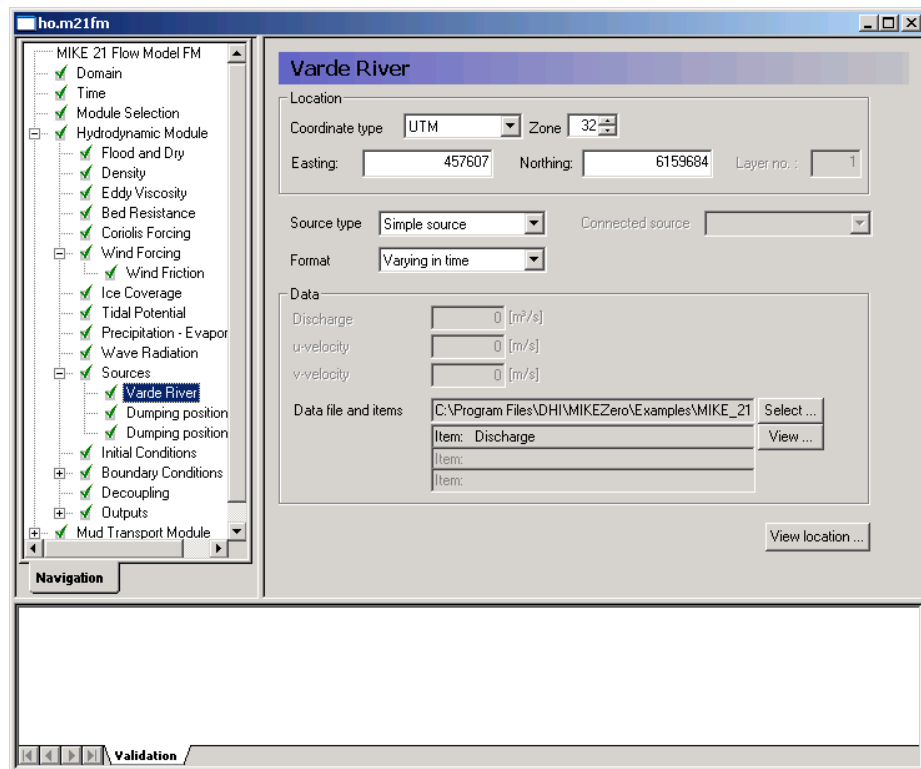


Figure 4.14 MIKE 21 Flow Model FM: Sources

The location of the sources must also be specified. Use the coordinates in Table 4.2.

Table 4.2 Position of sources (UTM-32)

Source	Easting	Northing	Discharge
Varde River	457607	6159684	varde_river_discharge.dfs0
Dumping position E	461771	6148416	0.001 m³/s
Dumping Position F	464646	6145524	0.001 m³/s

When you have specified the source coordinates, you should see a graphical map indicating the positions of the sources, cf. Figure 4.15.

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 0.2 m, which is the measured value in Esbjerg harbour at the start of the simulation, see Figure 4.16.

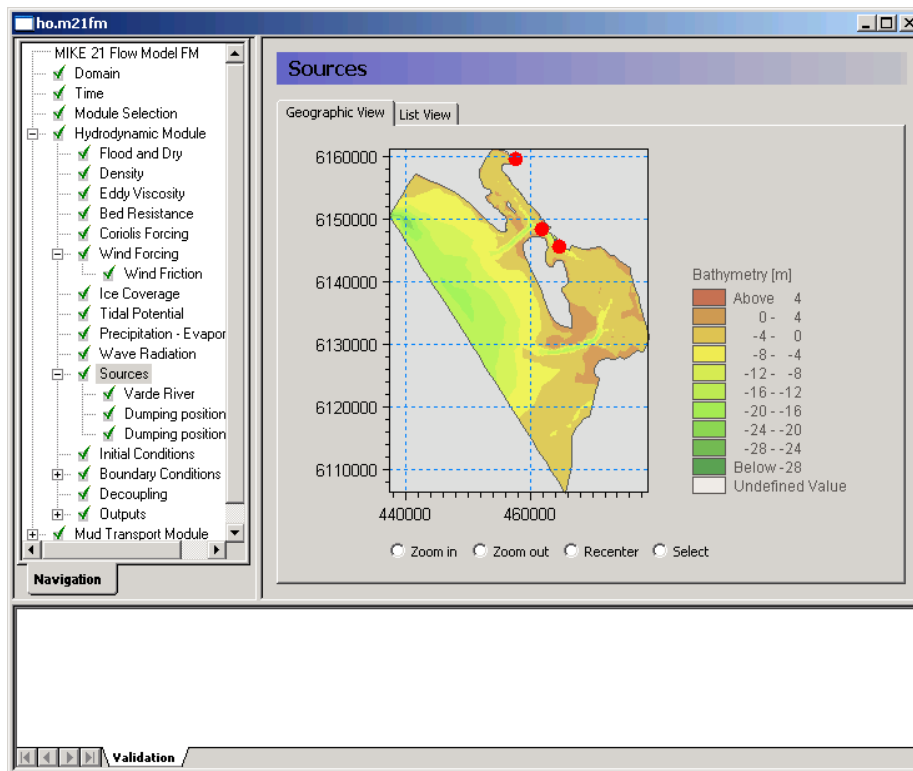


Figure 4.15 MIKE 21 Flow Model FM: Position in domain of specified sources

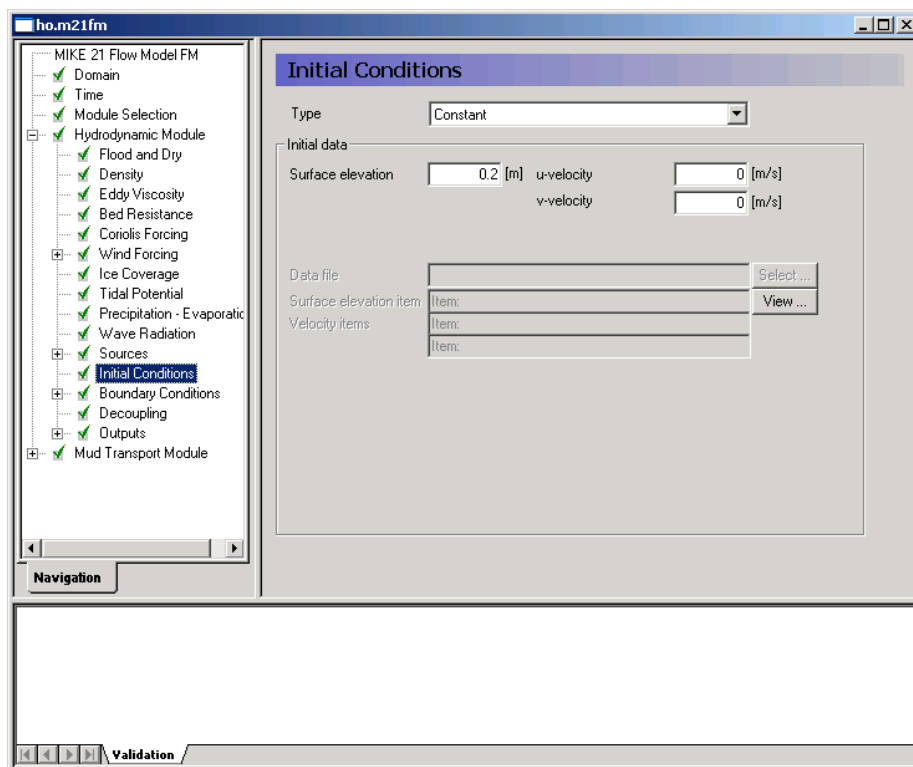


Figure 4.16 MIKE 21 Flow Model FM: Initial Conditions

In the Boundary Conditions dialogue, the boundary conditions should be specified for the boundary names, which were specified in the Domain dialogue. There is a North Boundary and a West Boundary, and the time series that were generated in Chapter 3 should be used on both 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 ...** and select the appropriate data file in the Open File window that appears, see Figure 4.17.

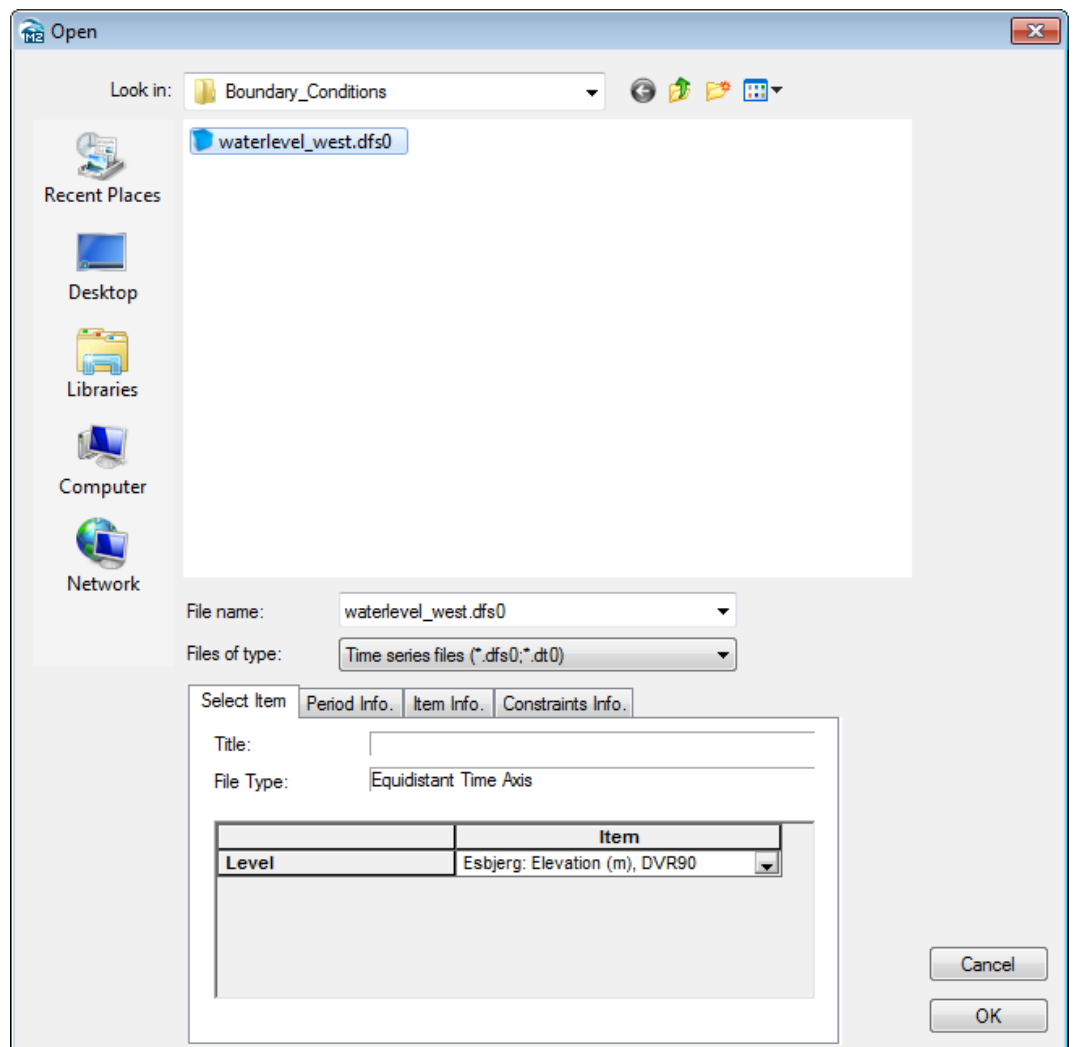



Figure 4.17 MIKE 21 Flow Model FM: Boundary Select File

For the North Boundary select the waterlevel\_west.dfs0, and also for the West Boundary select waterlevel\_west.dfs0. The files are the same because the water levels are assumed to be the same along the two boundaries.

Use a soft start interval of 7200 s and a reference value corresponding to the initial value of 0.2 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. At 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 4.18.

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

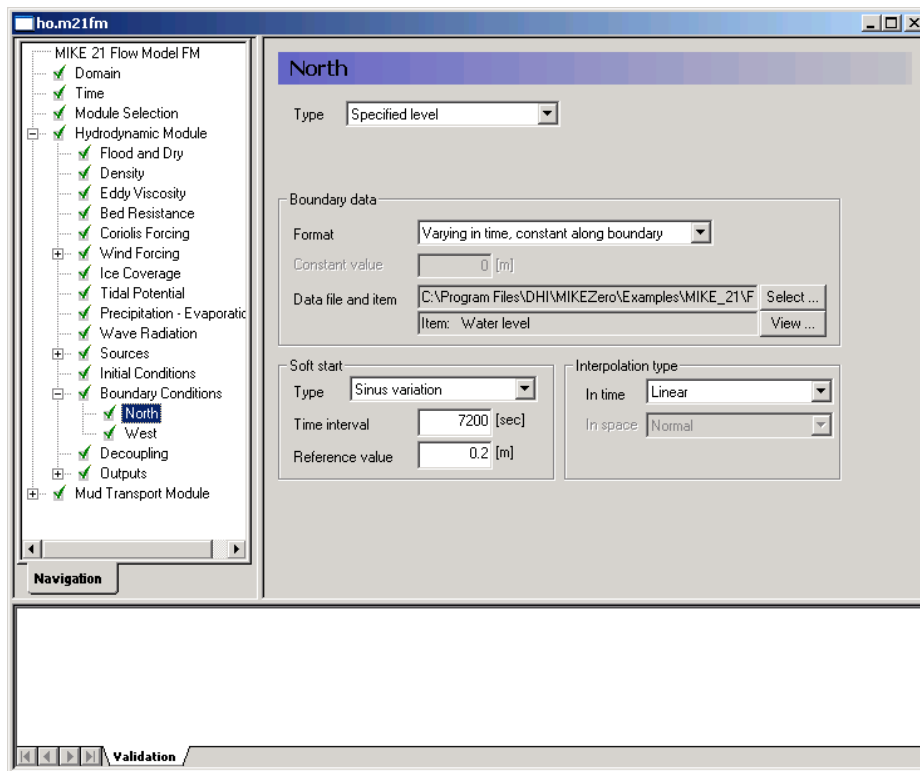


Figure 4.18 MIKE 21 Flow Model FM. The Boundary Conditions for the West Boundary are specified as constant along the boundary: waterlevel\_west.dfs0

Then you may specify one output as an area series (2D) and also specify the resulting output file name, see Figure 4.19. Specify the file name hd.dfsu for the first simulation. Make sure the required disk space is available on the hard disk. 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 is 60 s then the specified output frequency is  $3600/60 = 60$ . As default, the full area is selected.



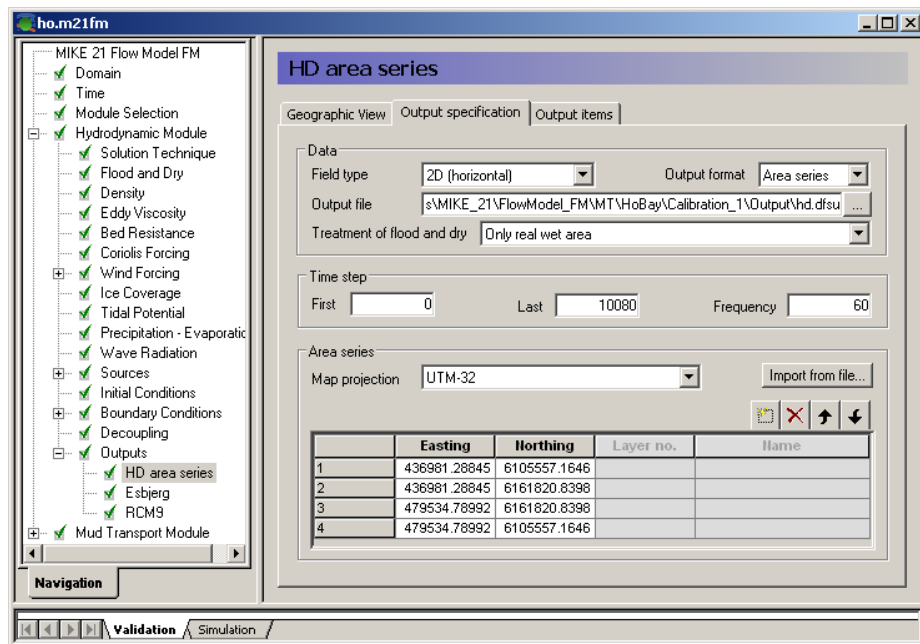


Figure 4.19 MIKE 21 Flow Model FM: Results can be specified as point, line or area series

Pick the parameters to include in the output file as in Figure 4.20.

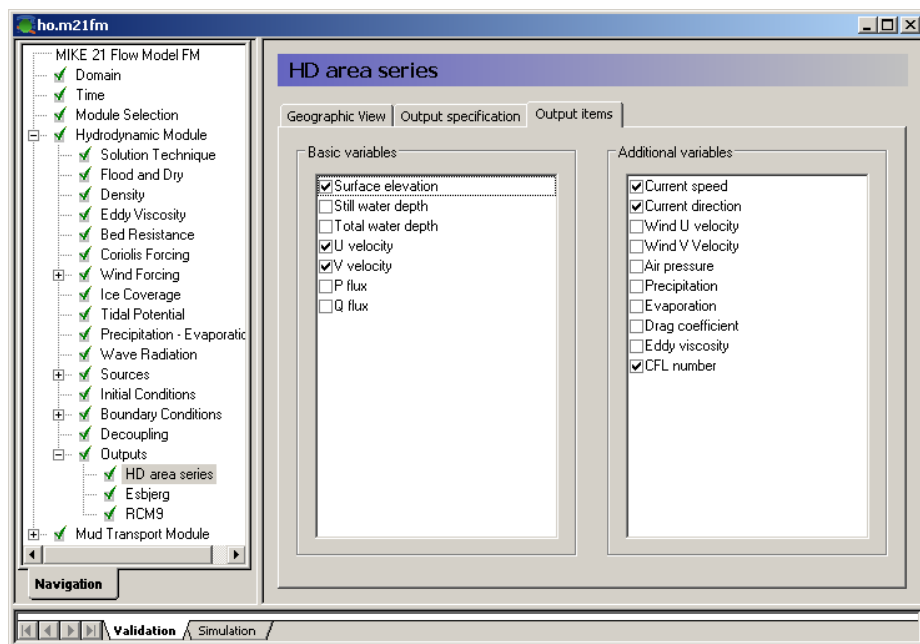


Figure 4.20 MIKE 21 Flow Model FM: Output Parameters

Also specify an output file as point series at the monitoring stations in RCM9 and Esbjerg Harbour, see the positions in Table 4.3. You might consider saving other time series from neighbouring points, so that you can see how much the results vary in the area near the monitoring station.

## 4.3 Mud Transport Module

The next step is to select the parameters for the specific Mud Transport Model.

The following features are not included in this example:

1. Sand fractions
2. Transition between layers
3. Dredging
4. Morphology

In the Parameter Selection dialogue, you must specify how many particle size fractions and mud bed layers that the model shall contain, see Figure 4.21.

It is possible to include up to 8 fractions and up to 12 sediment layers. So there are actually 96 possible combinations. For this example, please select 1 fraction and 2 sediment layers.

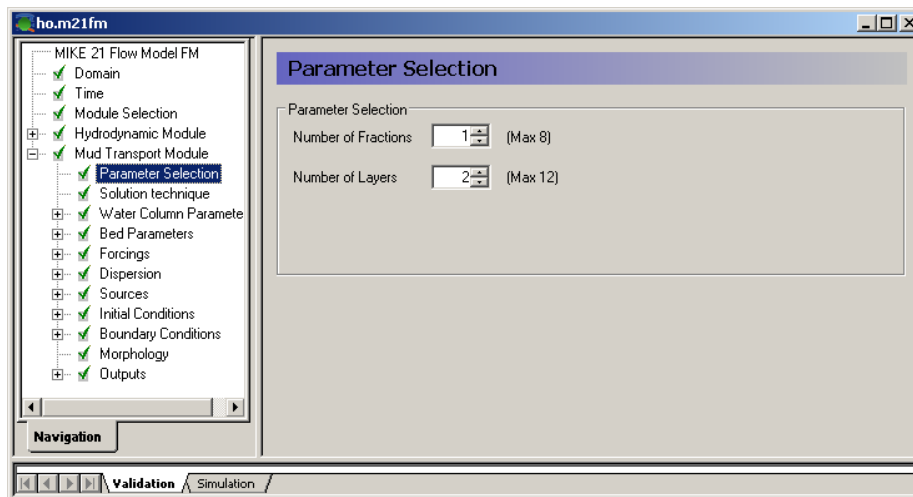


Figure 4.21 MIKE 21 Flow Model FM: Parameter Selection in the Mud Transport dialogue

The solution technique defined in the next dialogue is set to the same as for the hydrodynamic calculations, namely 'Low order, fast algorithm'.

The water column parameters must be specified in a way, so that properties relating to processes in the water column can be covered, i.e. settling and deposition.

In the Settling dialogue the general settling properties and the specific settling properties of each particle size fraction (in this case only one) must be specified. Flocculation should be included with a flocculation criteria of  $0.01 \text{ kg/m}^3$ , which means that flocculation as a function of suspended sediment concentration will start when  $\text{SSC} > 0.01 \text{ kg/m}^3$ . See Figure 4.22. The sediment density is specified as  $2650 \text{ kg/m}^3$ .

The settling velocity coefficient must also be specified. As only one fraction is modelled it must cover a range of grain sizes.

As flocculation is included, the settling velocity of the suspended sediment is not given directly, but as a coefficient from which the actual settling velocity will be derived. We will use the default value of 5 m/s for Fraction 1 mud concentrations specified with a constant format, see Figure 4.23.

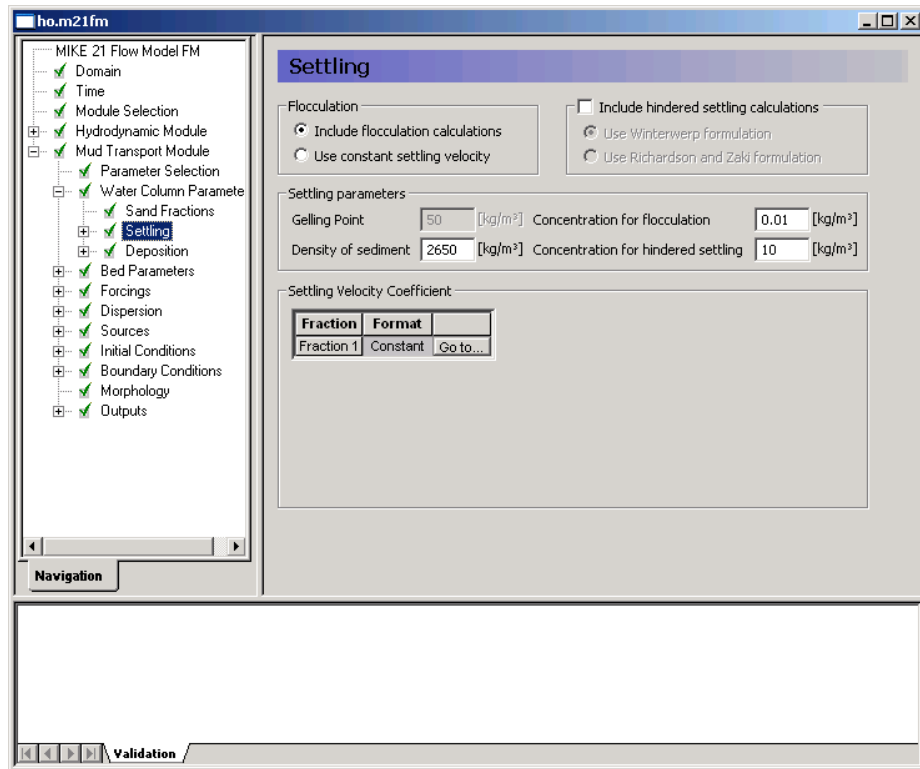


Figure 4.22 MIKE 21 Flow Model FM: Specification of settling properties in the Mud Transport dialogue

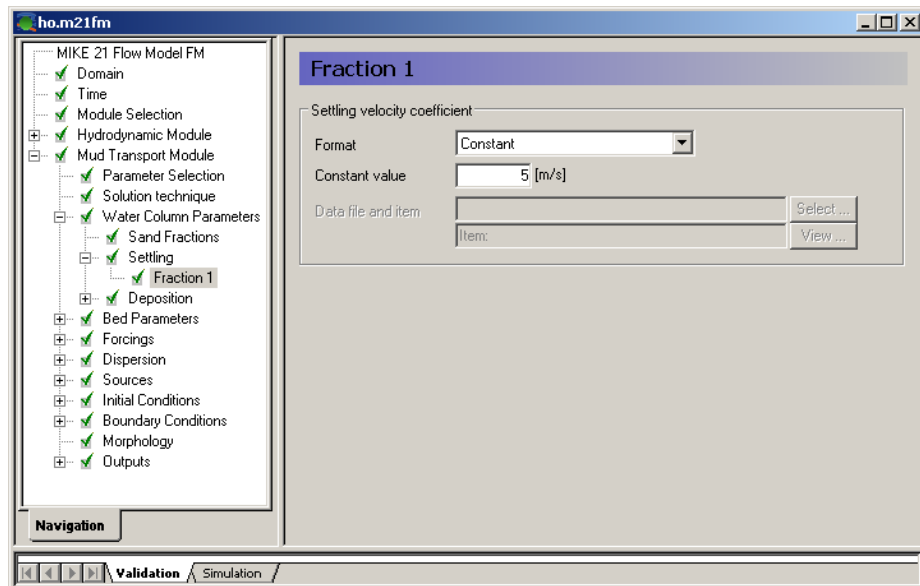


Figure 4.23 MIKE 21 Flow Model FM: Settling velocity coefficient in the Mud Transport dialogue

**Please note** that the magnitude of a Settling Velocity Coefficient is not directly comparable with Settling Velocity, because factors are multiplied by the coefficient to estimate the resulting settling velocity.

In order to describe the distribution of suspended matter in the water column, a profile equation must be picked:

1. Teeter Profile
2. Rouse Profile

We will use the default Teeter profile. The Critical Shear Stress for Deposition uses the default value 0.07 N/m<sup>2</sup>, see Figure 4.24 and Figure 4.25.

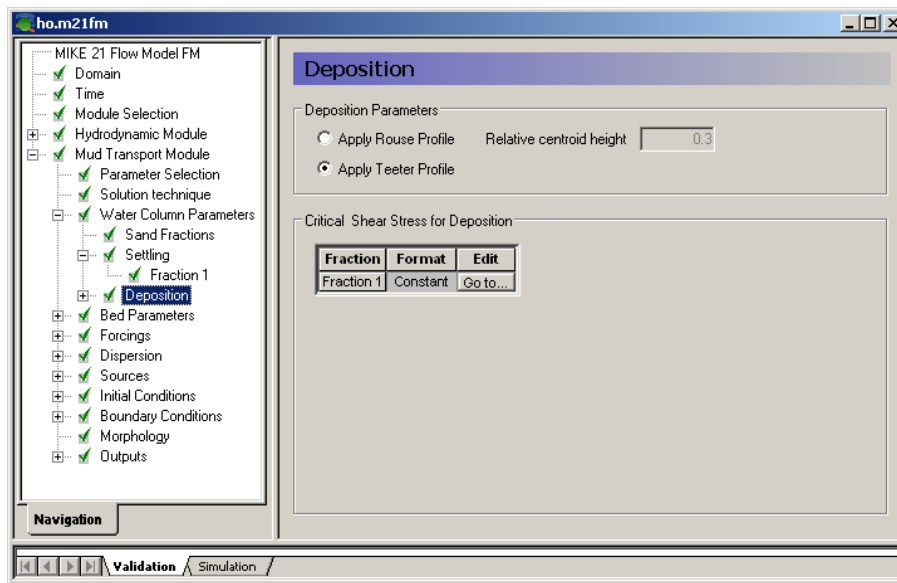


Figure 4.24 MIKE 21 Flow Model FM: Deposition specification in the Mud Transport dialogue

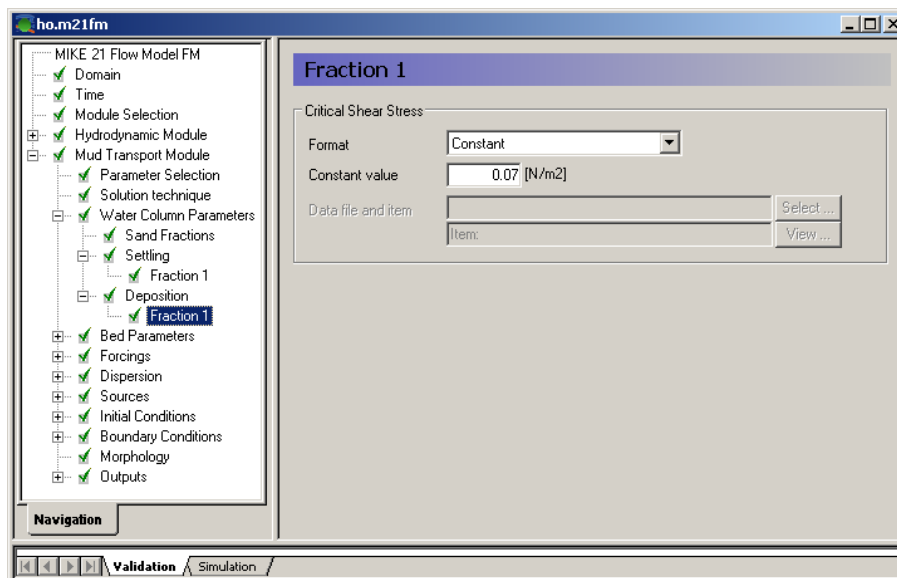


Figure 4.25 MIKE 21 Flow Model FM: Specifying Critical Shear Stress for deposition for of Fraction 1 suspended matter in the Mud Transport dialogue

The Bed Parameters must be specified, so that properties relating to processes in the sediment layers may be described including erosion and density of the layers.

For erosion the maximum concentration allowed due to erosion is specified as 50 kg/m<sup>3</sup>. See Figure 4.26. If the limit is reached the erosion process stops.

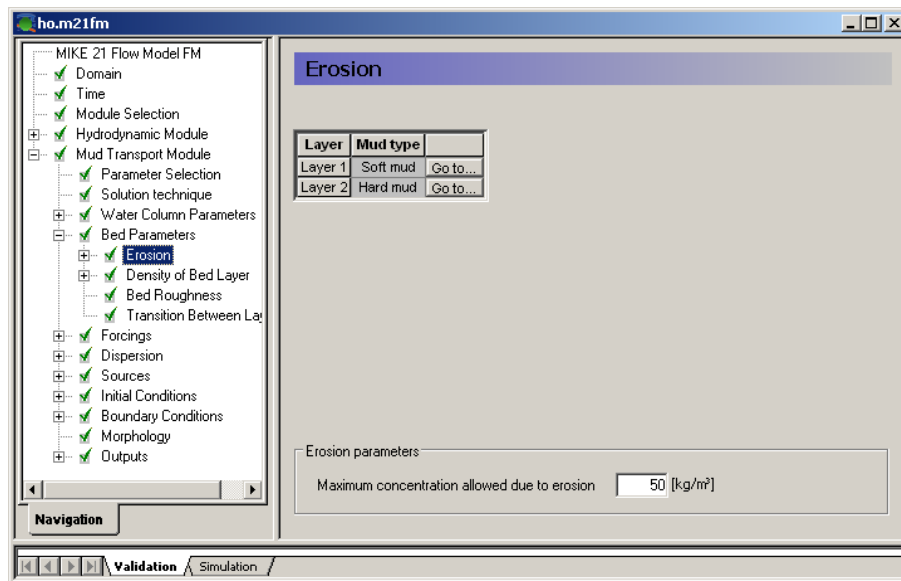


Figure 4.26 MIKE 21 Flow Model FM: Erosion specification in the Mud Transport dialogue

Select the erosion description as ‘Soft mud’ for layer 1 and ‘Hard mud’ for layer 2.

Set ‘Power of erosion’ as 8.3 for layer 1 and 1 for layer 2.

For each layer the erosion coefficient must be specified. Use a constant value of 0.00005 kg/m<sup>2</sup>/s in layer 1 and layer 2. See Figure 4.27.

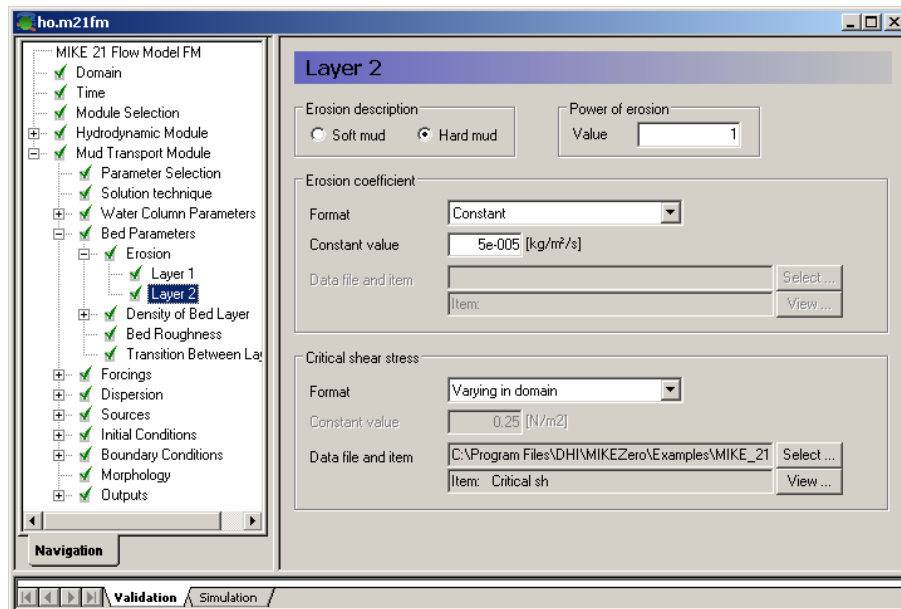


Figure 4.27 MIKE 21 Flow Model FM: Erosion parameters for Layer 2 in the Mud Transport dialogue

For layer 1 the critical shear stress is specified as constant: 0.1 N/m<sup>2</sup>.

For layer 2 the critical shear stress should be specified as varying in domain with the file generated in Chapter 3: `critical_shear_stress_layer2.dfs2`

The density of the sediment layers must also be specified. The more consolidated, the denser the layers are. For Layer 1 specify a density of  $180 \text{ kg/m}^3$ , and for the denser layer 2 a density of  $300 \text{ kg/m}^3$ . See Figure 4.28.

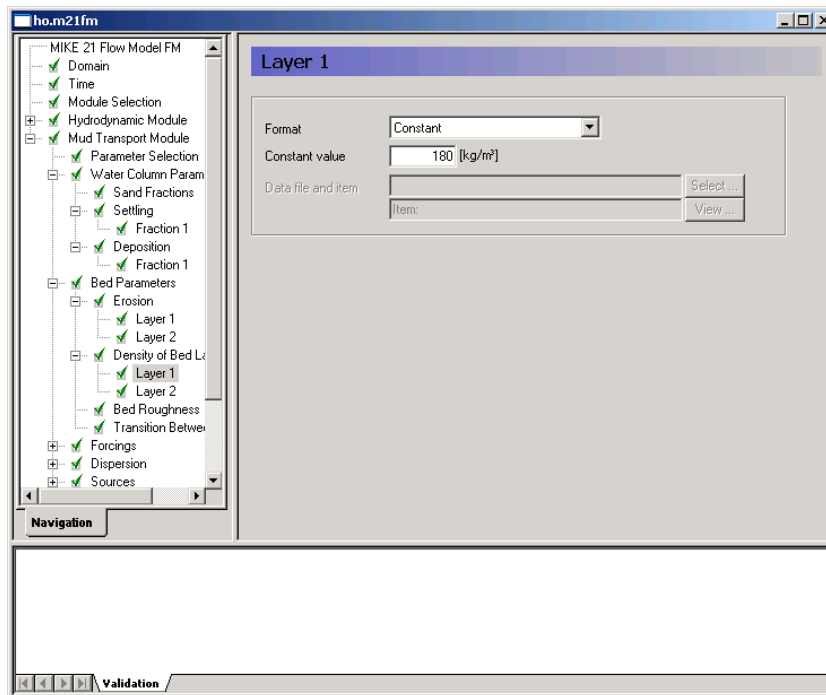


Figure 4.28 MIKE 21 Flow Model FM: Specifying density for Layer 1 in the Mud Transport dialogue

Set the bed roughness is specified as constant with a value of  $0.001 \text{ m}$ , see Figure 4.29.

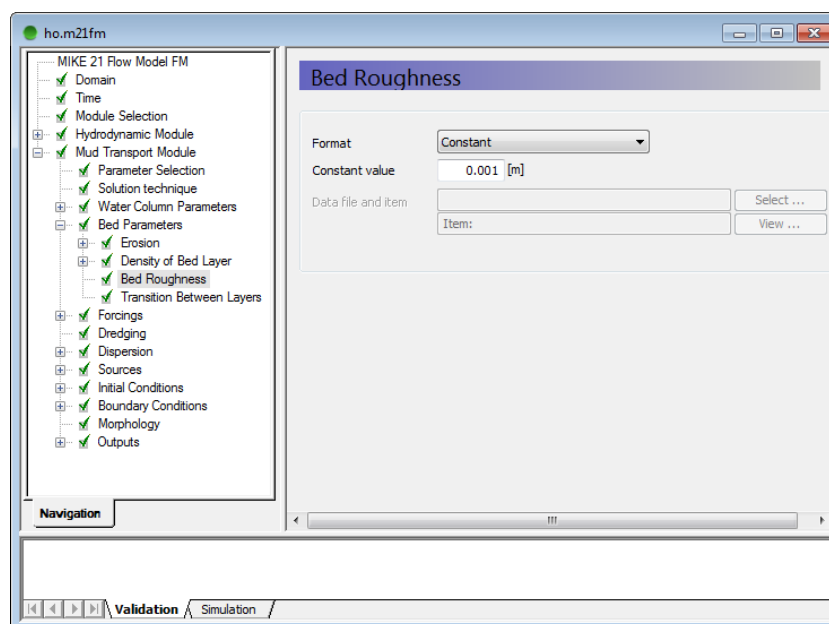


Figure 4.29 MIKE 21 Flow Model FM: Specifying Bed Roughness

Forcings include the impact from waves. These shall be included with a database with 16 wave patterns representing different wind and water level conditions. MIKE 21 Flow Model FM Mud Transport Module interpolates the patterns during simulation with respect to simulated water levels and the wind conditions based on the wind file given in Chapter 3.3. There are 2 wind speed situations, 4 wind direction situations, and 2 water level situation. Thus, they can be combined in 16 different ways. The files to specify are shown in Table 3.2.

The minimum water depth is set to 0.1 m and the shear stress formulation is selected as Max (Soulsby), see Figure 4.30.

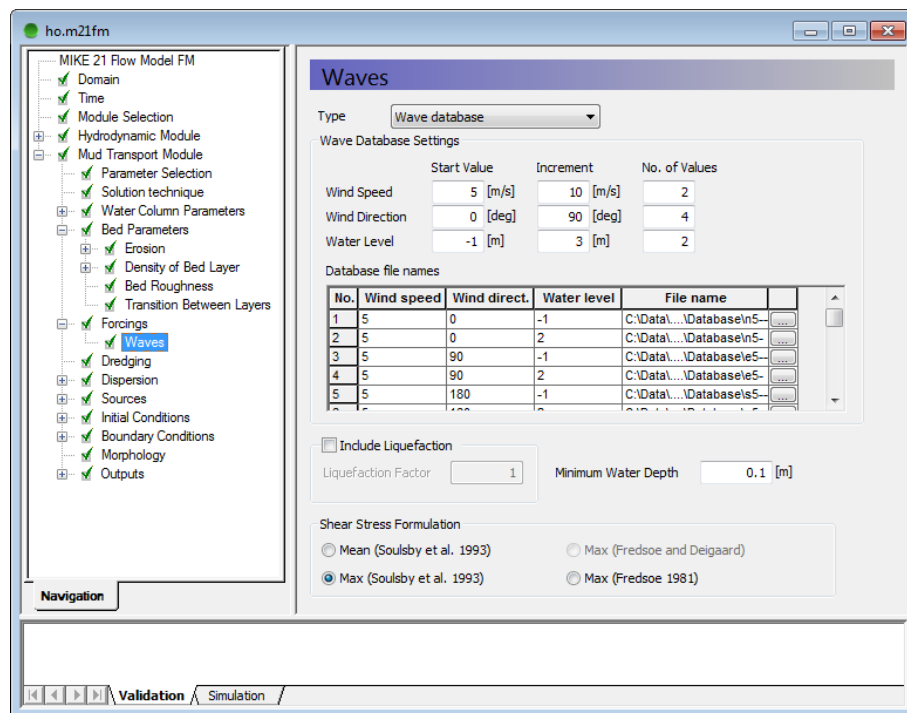


Figure 4.30 MIKE 21 Flow Model FM: Specifying the Wave database parameters in the Mud Transport dialogue

The dispersion properties are specified with a scaled eddy viscosity formulation using a scaling factor of 1. See Figure 4.31.

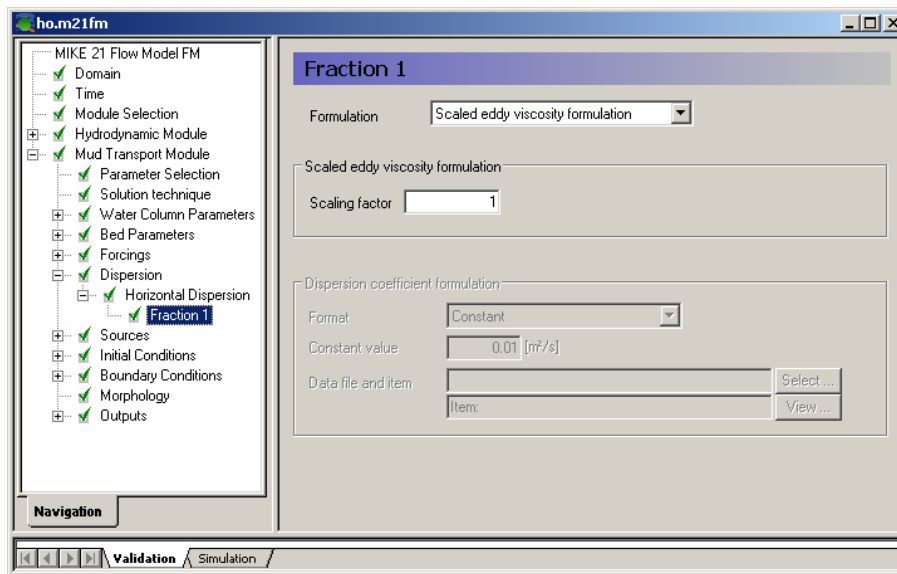


Figure 4.31 MIKE 21 Flow Model FM: Dispersion coefficient specifications in the Mud Transport dialogue

The source concentrations of Fraction 1 suspended matter must be specified, see Figure 4.32. For Varde River the concentration is not known, but use an estimate of a constant concentration 0.01 kg/m<sup>3</sup>.

For the dumping sources specify the time series related to the time series generated in Chapter 3.5:

Dumping position E:	sediment_dumping.dfs0	item 1
Dumping position F:	sediment_dumping.dfs0	item 2

**Please note** that a shortcut to see the location of the source is to click 'View location ...'



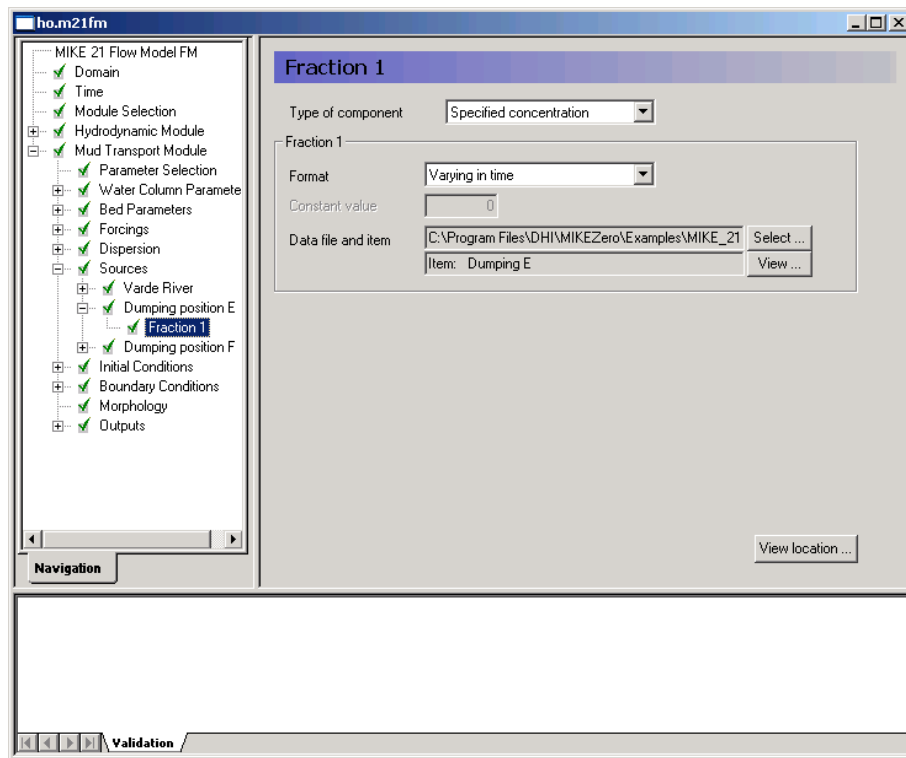


Figure 4.32 MIKE 21 Flow Model FM: Specifying Mud concentrations of Fraction 1 in sources in the Mud Transport dialogue

Initial conditions for Fraction 1 concentration and the thickness of the sediment layers must be specified. Use the dfs2 file that was generated in Chapter 3:

Fraction 1 concentration:	mt_version1.dfs2	item1
Layer 1 thickness:	mt_version1.dfs2	item2
Layer 2 thickness:	mt_version1.dfs2	item3

See Figure 4.33 and Figure 4.34.

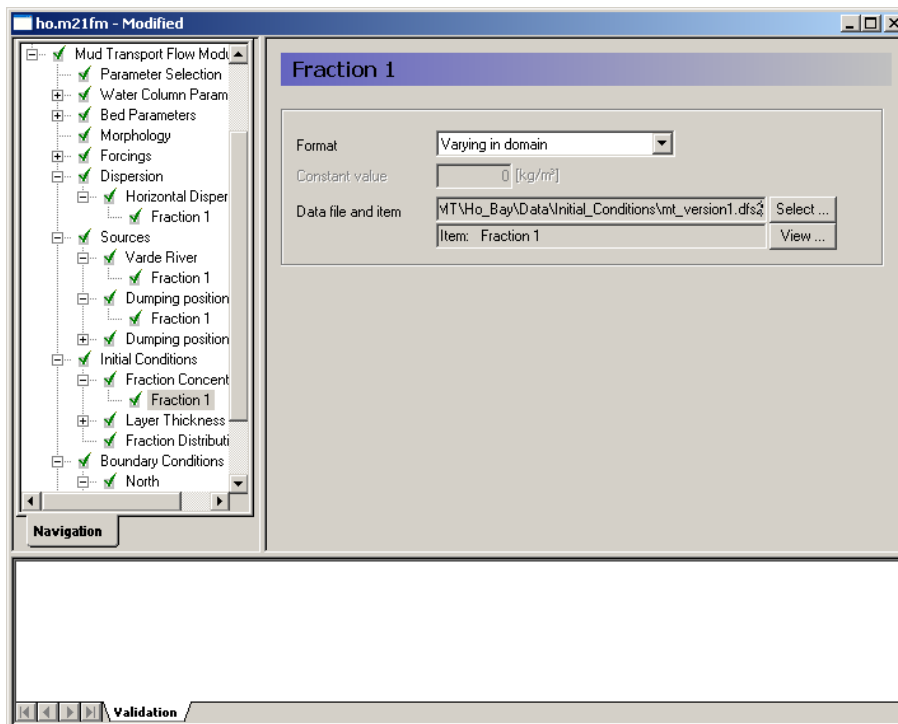


Figure 4.33 MIKE 21 Flow Model FM: Specifying initial conditions for the Fraction 1 mud concentrations in the Mud Transport dialogue

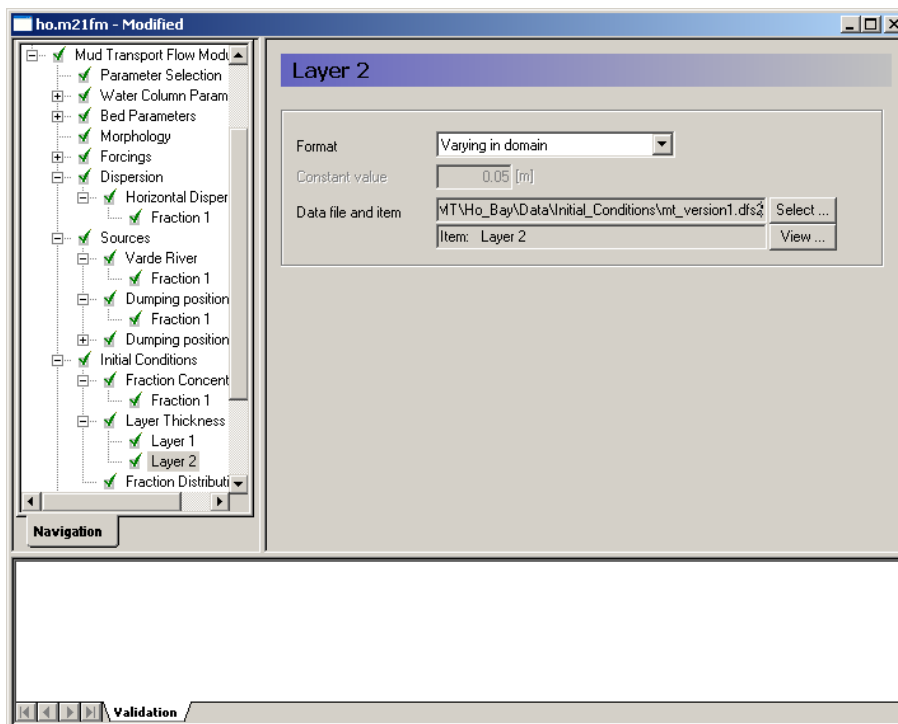


Figure 4.34 MIKE 21 Flow Model FM: Specifying initial condition of Layer 2 thickness in the Mud Transport dialogue

The Boundary concentration of suspended matter must be specified for Fraction 1 concentrations. Use an estimate of  $0.004 \text{ kg/m}^3$ . Also specify a soft start of 7200 s. See Figure 4.35.

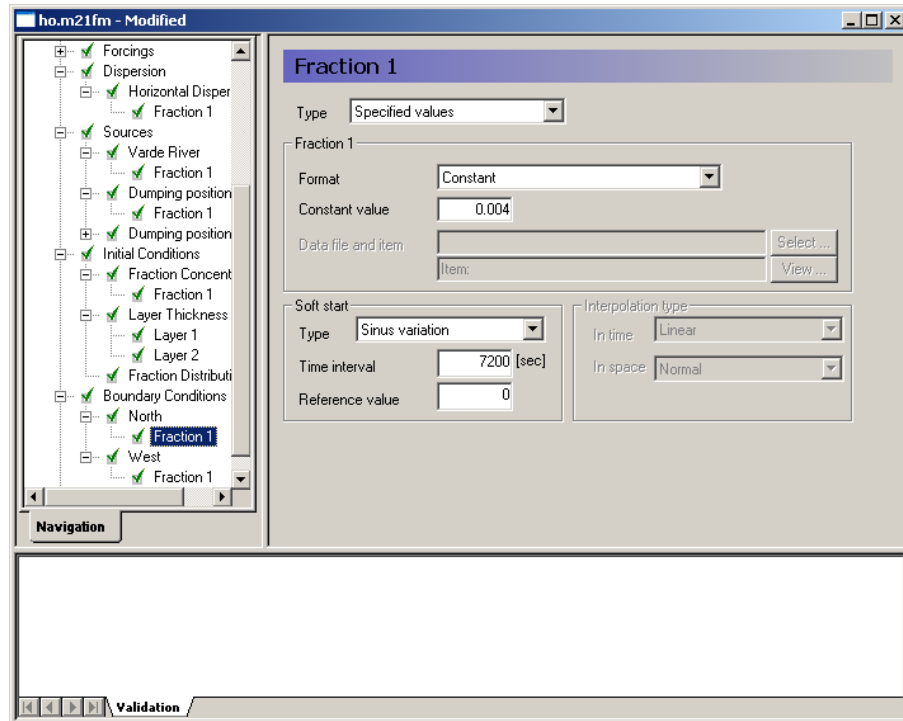


Figure 4.35 MIKE 21 Flow Model FM: Specifying boundary conditions for the Fraction 1 mud concentrations in the Mud Transport dialogue

Finally, the output files must be specified. See Figure 4.36, and Figure 4.37.

You should save 2 output files:

- a 2D area series representing the whole domain
- a time series at the monitoring station RCM9

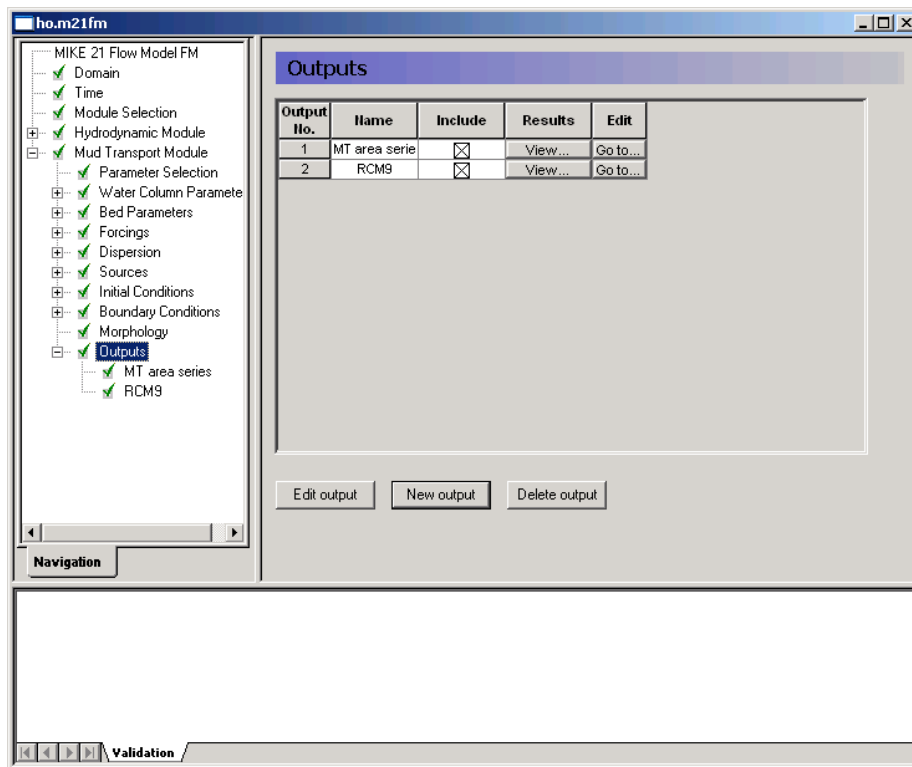


Figure 4.36 MIKE 21 Flow Model FM: Specifying output file in the Mud Transport dialogue

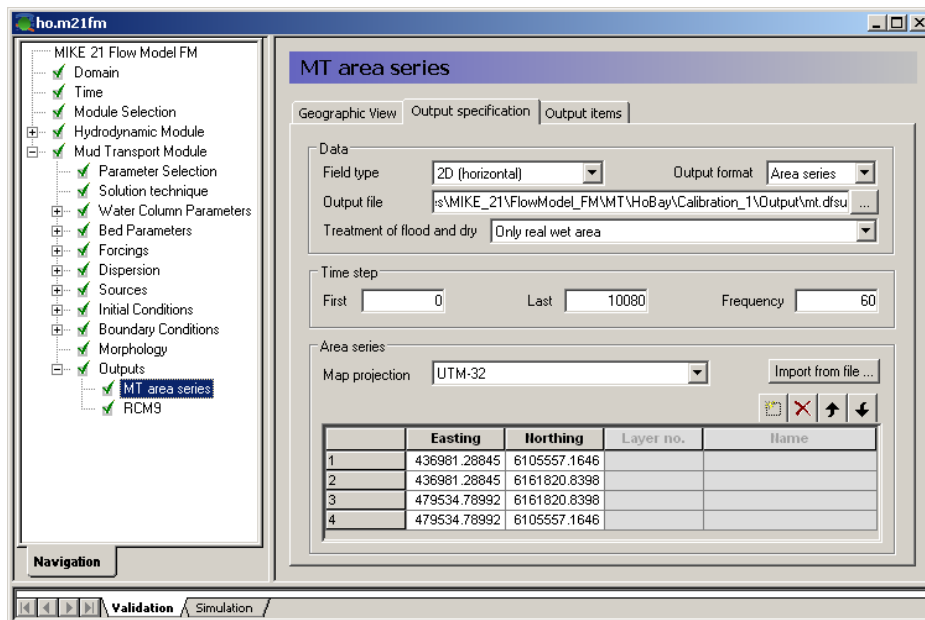


Figure 4.37 MIKE 21 Flow Model FM: Specifying an Area Series output file in the Mud Transport dialogue

Now we are ready to run the MIKE 21 Flow Model FM. (Run → Start simulation...).

The specification file for this example has already been made:

.\Ho\_Bay\Calibration\_1\Ho.m21fm

**Please note** that if you experience an abnormal simulation, you should look in the log file to see what causes the problem. The log file is displayed in the lower simulation window of the main GUI, but is also accessible from a list in the file drop-down menu 'File → Recent log files'.

After the simulation is finished, use the Plot Composer (or Data Viewer) to inspect and present the results.

The simulation data can be manipulated and extracted directly from dfsu result files by use of the Data Manager or the Data Extraction FM tool:

- New → MIKE Zero → Data Manager
- New → MIKE Zero → Data Extraction FM

The Post Processing Tools (data extraction, statistics, etc.) developed for the dfs2 and dfs3 formats can also be used for dfsu files. It requires a conversion of the dfsu file to a dfs2 or dfs3 file first. There is a tool available for that conversion:

- New → MIKE Zero → Grid Series → From Dfsu File

## 4.4 Model Calibration 1

### 4.4.1 Measured values

In order to calibrate the model some measurements are required inside the model domain. Measurements of water level, current velocities and turbidity are available, see Table 4.3.

Table 4.3 Measurements in the Grådyb tidal area. Projection UTM-32 (datum WGS-84)

Station	Data Files	Position	
		Easting (m)	Northing (m)
Esbjerg	waterlevel_esbjerg.txt	464229	6145802
RCM9	turbidity_rcm9.txt current_rcm9.txt	461745	6148663

Measurements of water levels are given at Esbjerg Harbour (waterlevel\_esbjerg.txt). Import this ASCII file using the Time Series Editor (how to import ASCII files to time series, see Section 3.1.1).

The water level at Esbjerg is shown in Figure 4.38.

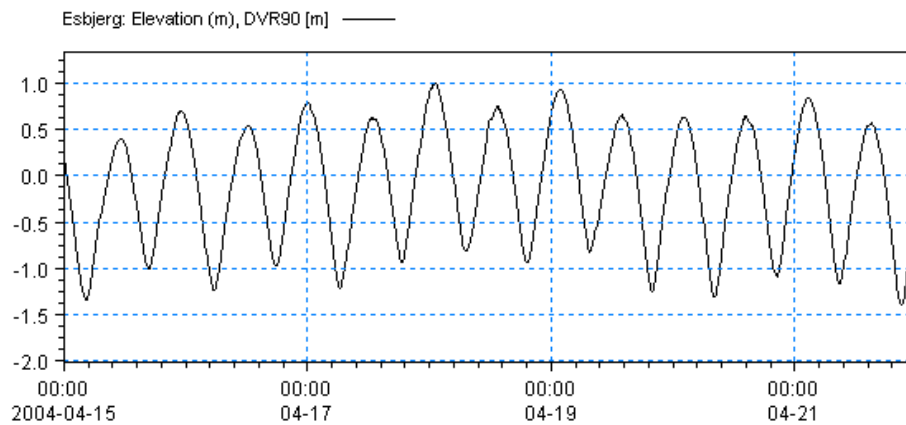


Figure 4.38 Esbjerg Harbour: Measured water levels

Measurements of current speed are given for station RCM9 (currents\_rcm9.txt). Import this file with the Time Series Editor.

A plot of measured current speed and direction is shown in Figure 4.39.

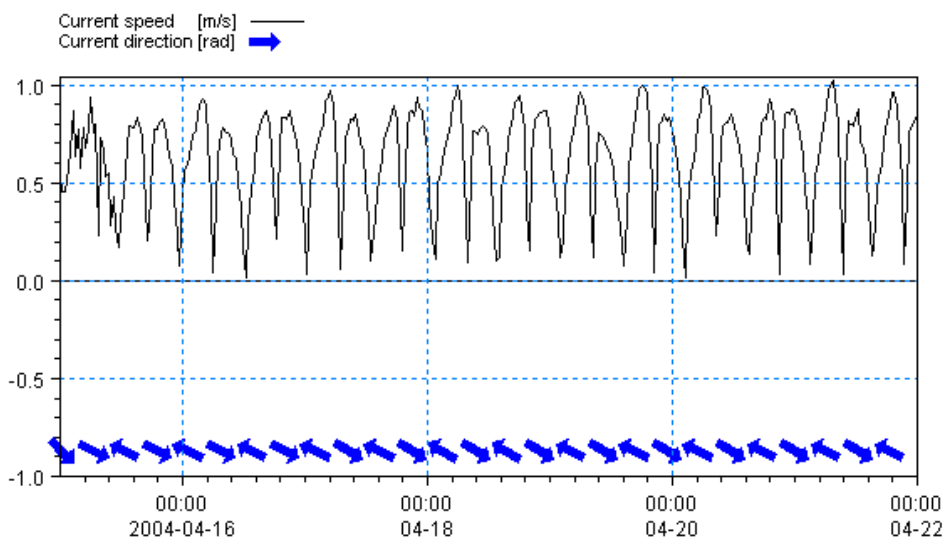


Figure 4.39 Station RCM9: Measured current speed and direction

The turbidity has been measured 4 metres above the seabed (approximately in the middle of the water column), so the depths of the measurements vary with the tides from 2 m to 5.2 m. The measurements are compared directly with the results from the depth-integrated model.

This means that a vertical profile is assumed with an average value 4 metres above the seabed. This is of course an approximation to the real conditions.

Measurements of turbidity are given for station RCM9 (turbidity\_rcm9.txt).

Import this file with the Time Series Editor. A plot of measured turbidity is shown in Figure 4.40.

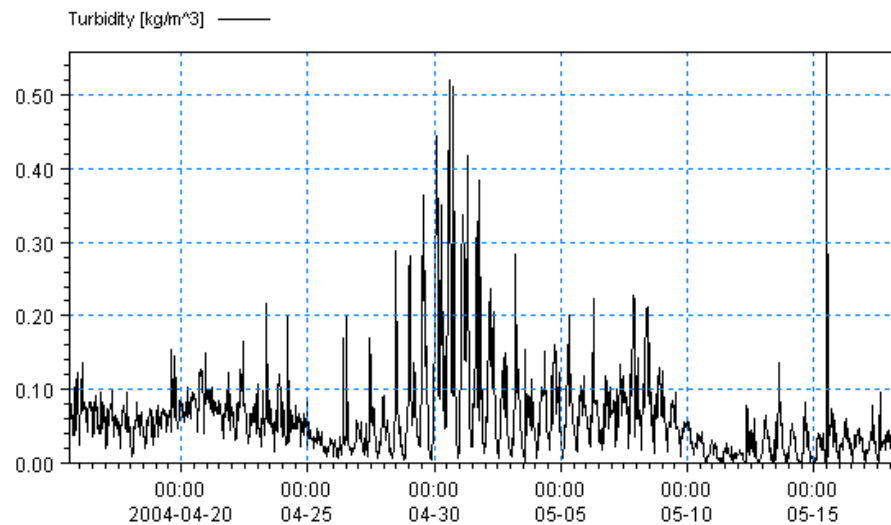


Figure 4.40 Station RCM9: Measured turbidity [kg/m<sup>3</sup>]

#### 4.4.2 Compare model result and measured values

Use the Plot Composer to plot the simulated and measured water levels, currents and turbidity.

For this example, Plot Composer files have already been made. The files are found in the following folder:

.\Ho\_Bay\Calibration\_1\Plots

In order to see a comparison of measurements and model output of water levels at Esbjerg Harbour and current velocity and turbidity at station RCM9 you need to open the following files:

1. waterlevel\_esbjerg.plc
2. current\_speed\_rcm9.plc
3. turbidity\_rcm9.plc

If your simulation is successful, plots as shown in Figure 4.41, Figure 4.42, and Figure 4.43 will appear.

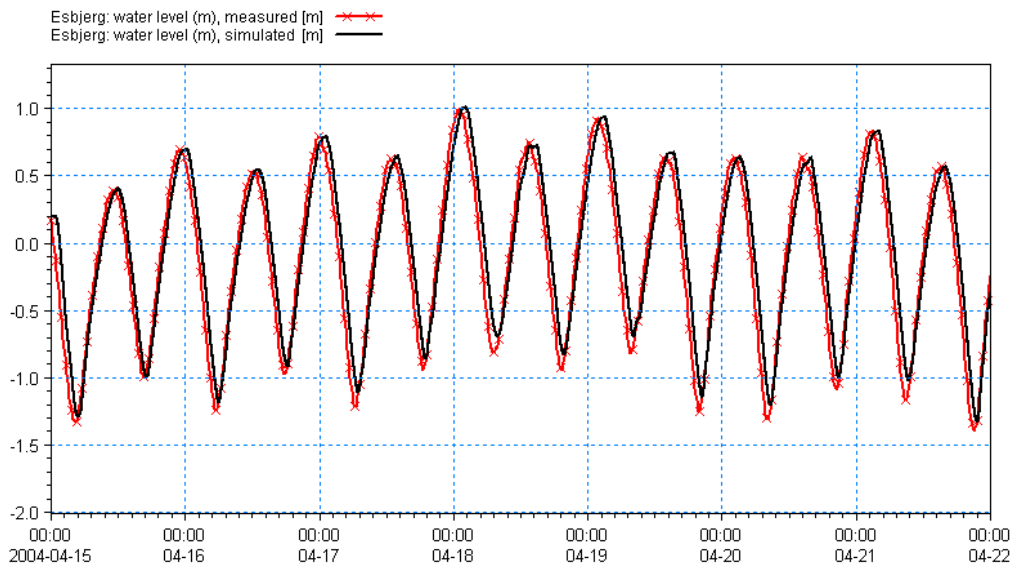


Figure 4.41 Comparison of measurements and model output of water levels at Esbjerg Harbour, Calibration 1

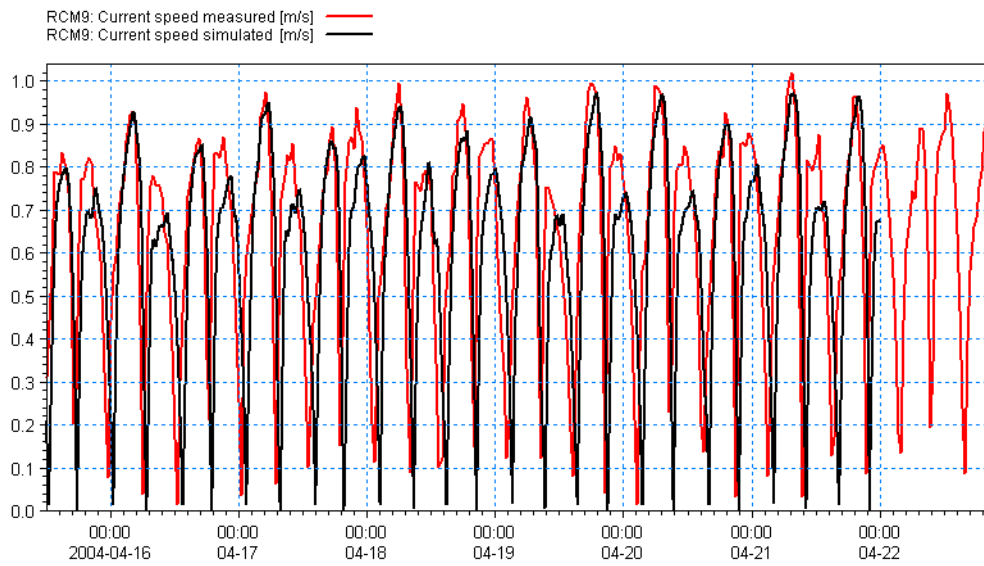


Figure 4.42 Comparison of measurements and model output of currents at monitoring station RCM9, Calibration 1



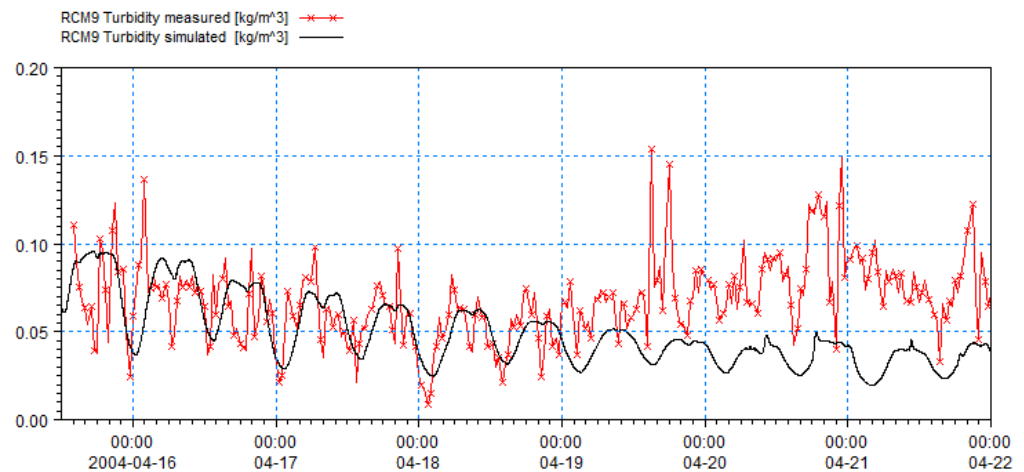


Figure 4.43 Comparison of measurements and model output of turbidity at monitoring station RCM9, Calibration 1

The comparisons between measured and calculated water levels and current speed indicate that the calibration is good for the hydrodynamic conditions.

Therefore, for the hydrodynamic results we will not calibrate further.

The comparison of measured and simulated values of turbidity (Figure 4.43) shows that the simulated values are reasonable in the beginning of the calculation but decrease too much as the simulation continues. Also, it seems that the additional mud, which was dumped during the last two days of the simulation - and probably contributed to the increased turbidity in the measurements at the end of the period – is not giving any effect in the modelled data. This result can be improved by calibration.

The dominance of the tidal cycle in turbidity during the simulation indicates that the basic physical processes of the sediment (settling and resuspension) are not resolved correctly by the parameters.

Calibration simulations concerning various parameters are given in the following sections.

## 4.5 Model Calibration 2

### 4.5.1 Generate decoupled model data

The first simulation showed that the hydrodynamic results need no further calibration. However, it can save time in future calibration runs if the basic flow information is read from a file instead of calculating the flow information during the simulation. This is possible when running a simulation in decoupled mode.

First, you need to generate the decoupling data.

The decoupled data for use in future simulations are defined in the Decoupling dialogue in the Hydrodynamic Module, see Figure 4.44.

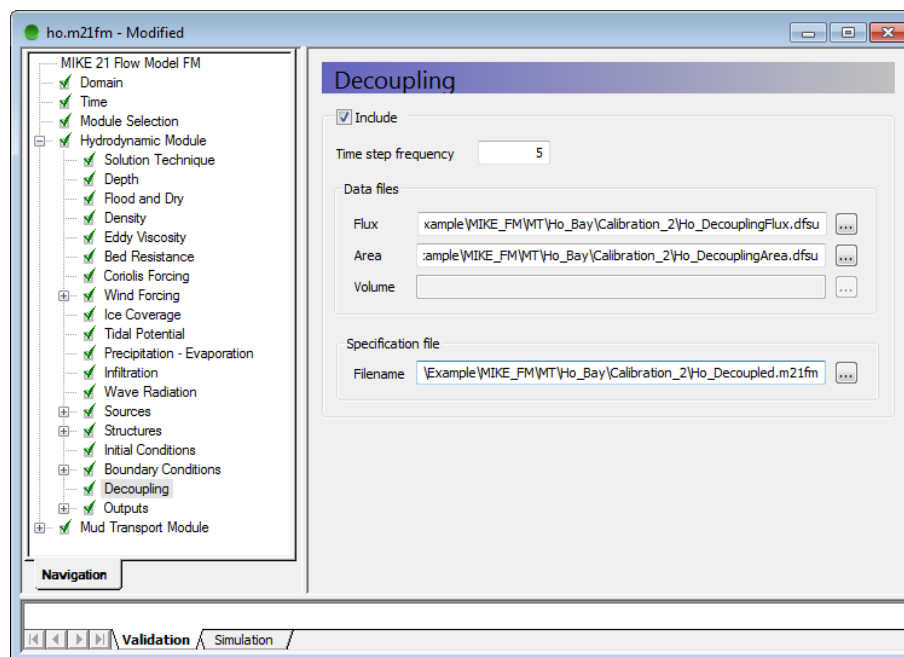


Figure 4.44 MIKE 21 Flow Model FM: Decoupling specifications

The decoupled data are generated and saved every 5 minutes, i.e. with a time step frequency of 5. The data files and specification file are defined by default names.

#### 4.5.2 Changed model specifications

The results from Calibration 1 showed the turbidity values to be far too dominated by the tidal process. This could be due to many factors, e.g. the mud settling too slowly or the calculation of the erosion and deposition rates being underestimated.

Furthermore, as the results in the first calibration showed a general trend towards a decrease in turbidity, a number of parameters have been changed in Calibration 2. All parameters have been modified in order to increase the moveability of mud to and from the bed.

The set-up for Calibration 2 is similar to Calibration 1 with the exception of the parameters listed in Table 4.4.

Table 4.4 Changed parameters in Calibration 2

Parameter	Value
Time Step Interval	60 s
No. of Time Steps	10080
HD: Decoupling	Time step interval: 5 Ho_DecouplingFlux.dfsu Ho_DecouplingArea.dfsu Ho_Decoupled.m21fm
MT: Settling	Settling velocity coefficient: 10 m/s
MT: Erosion (Soft mud)	Critical shear stress: 0.08 N/m <sup>2</sup>
MT: Deposition	Critical shear stress: 0.04 N/m <sup>2</sup>
MT: Include waves	Shear stress formulation: Max. Fredsøe
Result Files	hd.dfsu    esbjerg_hd.dfs0    rcm9_hd.dfs0 mt.dfsu    rcm9_mt.dfs0
CPU Simulation Time	About 100 minutes with a 1.8 GHz PC, 1 GB RAM

### 4.5.3 Model results compared with measured values

The resulting turbidity at station RCM9 is shown in Figure 4.45.

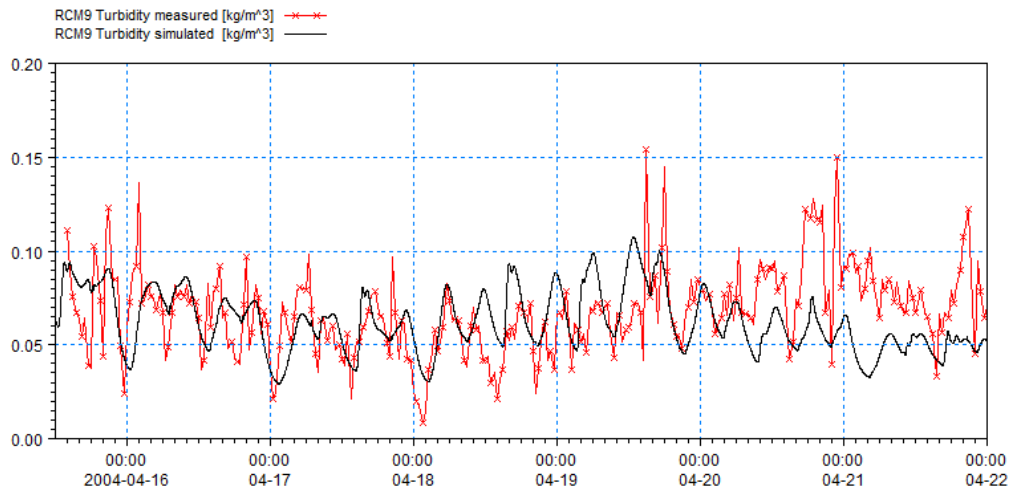


Figure 4.45 Comparison of measurements and model output data of turbidity at monitoring station RCM9, Calibration 2

The simulated results are more comparable with the measured turbidity values, but there is still room for improvement. Generally, when calibrating you should only change one single parameter at a time and track the changes in a log.

## 4.6 Model Calibration 3

### 4.6.1 Decoupled model set-up

The decoupled model data were saved during the simulation of Calibration 2. Now you should copy the two files 'Ho\_DecouplingFlux.dfsu' and 'Ho\_DecouplingArea.dfsu' to the folder 'Calibration\_3' (the set-up file has already been installed).

When using a decoupled set-up you cannot change the basic parameters in the Time, Domain or Hydrodynamic dialogues, but you are allowed to change the parameters in the Mud Transport Module.

### 4.6.2 Changed model specifications

The good results in the end of Calibration 2 indicate that the basic physical processes of the sediments (settling and resuspension) now have reasonable parameters. In this case the problem is that too little sediment is transported to the monitoring station from neighbouring points. Therefore, Calibration 3 will concern the initial conditions only and not the parameters for settling and resuspension. In this phase it is often a good practice to run a sensitivity analysis for the parameters that you want to change in your calibration.

If you run 3 sensitivity simulations as follows, you will find that the initial conditions of Fraction 1 is very important in the simulation results, Layer 1 thickness is not so important, and Layer 2 has some effect (a longer lasting effect, however):

1. Fraction 1 set to zero
2. Layer 1 thickness set to zero
3. Layer 2 thickness set to zero

For Calibration 3 we want to keep the good results in the start of the simulation, but increase the simulated concentrations in the end. Therefore, the initial conditions of Layer 2 has been changed by adding 1 mm to the layer thickness in the entire area in order to increase the amount of material transported to the area of interest.

Try replacing the file for the initial conditions in Calibration 1 (mt\_version1.dfs2) with mt\_version2.dfs2 and run the set-up.

### 4.6.3 Model results compared with measured values

The resulting turbidity at station RCM9 is shown in Figure 4.46. Now the model output turbidity is increased, but it does not improve the results. Changing the initial conditions in a 2D varying map is difficult, because the exercise is to guess the distribution at a previous time step, which is a difficult task. In this situation you should look at the results in an area series, see Figure 4.47.

Try to calibrate further by changing the initial conditions more. For each calibration simulation compare the results with the measurements. You should only change a single parameter at a time and track the changes in a log. Also be careful not to give values outside a realistic range.

**Please note** the way the folders are organised in the example: You can simply copy the Calibration\_3 folder and rename it to Calibration\_4. This way the Plot Composer file will also work for the new calibration simulation, because the path to the files in the Plot Composer file is relative to the present folder. If you are making many simulations this trick can save a lot of time.

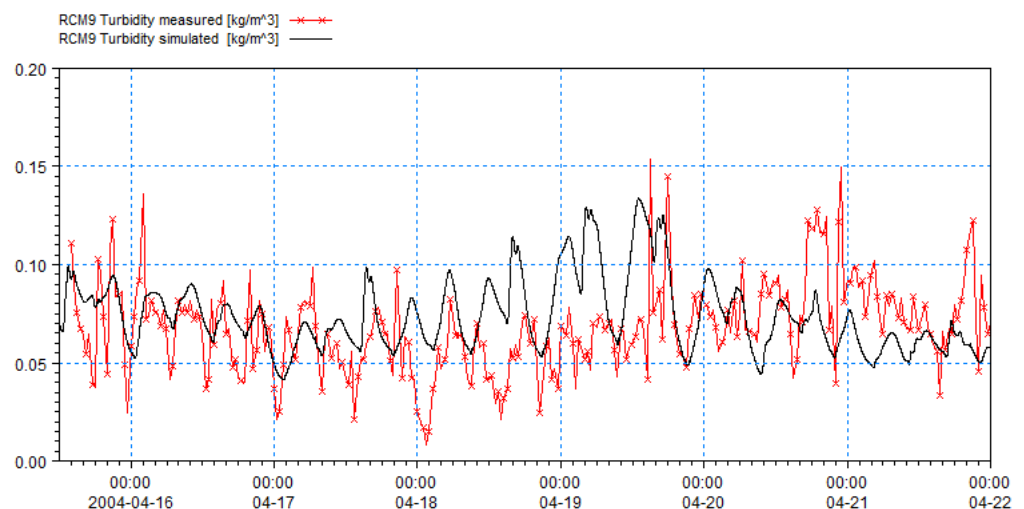


Figure 4.46 Comparison of measurements and model output of turbidity at monitoring station RCM9, Calibration 3

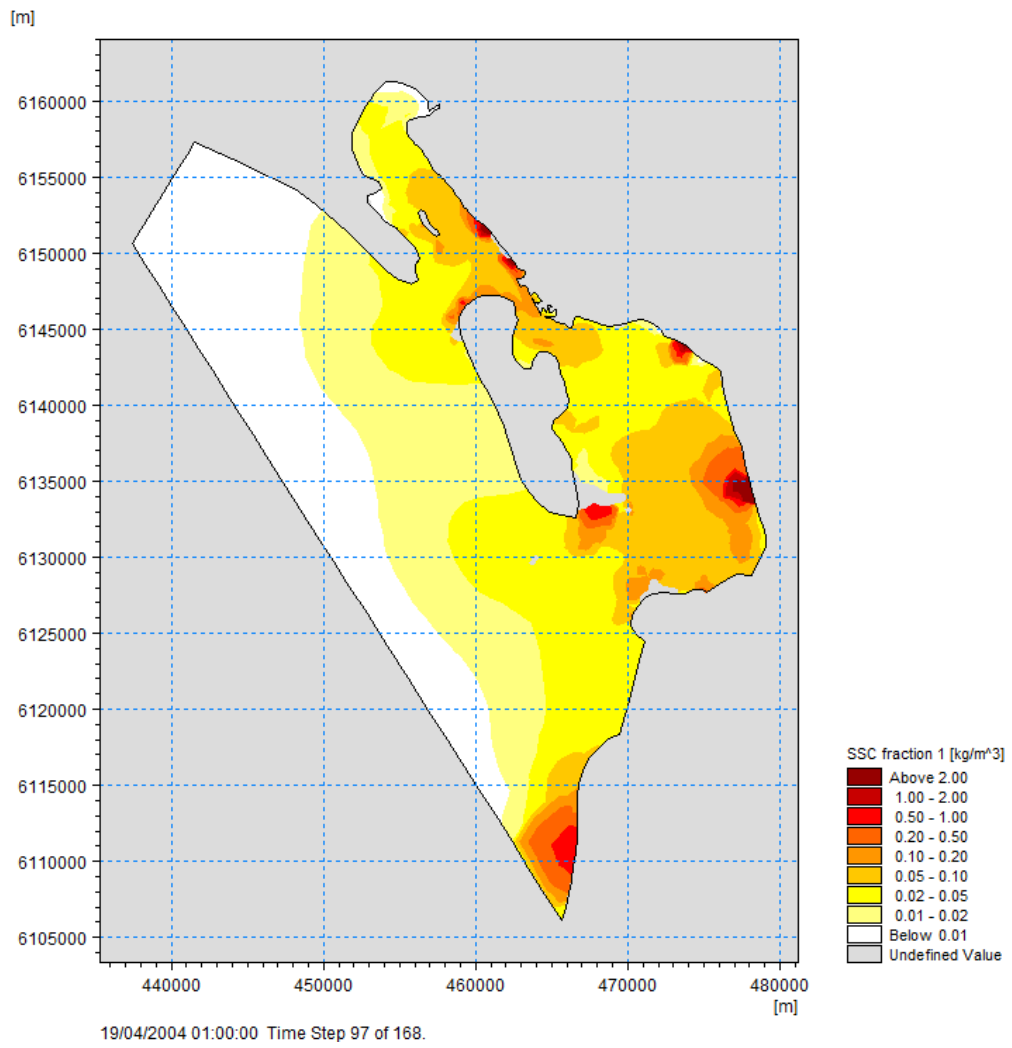


Figure 4.47 Area Series of output from the MIKE 21 Flow Model FM Mud Transport Module