

MIKE 21 Flow Model FM

Particle Tracking Module

Step-by-step training guide



DHI A/S headquarters

Agern Allé 5
DK-2970 Hørsholm
Denmark

+45 4516 9200 Telephone
mike@dhigroup.com
www.mikepoweredbydhi.com

Company Registration No.: DK36466871

CONTENTS

MIKE 21 FLOW MODEL FM Particle Tracking Module Step-by-step training guide

1	Introduction.....	1
1.1	Background.....	1
1.2	Objective.....	3
2	Creating the MIKE Zero Project.....	5
2.1	The Role of the MIKE Zero Project in this Example.....	5
2.2	Creating the Project File.....	5
3	Creating the Computational Mesh.....	9
4	Creating the Input Parameters to the MIKE 21 Flow Model FM.....	11
4.1	Generating Hydrodynamic Boundary Data.....	13
4.1.1	Importing measured water levels to time series file.....	14
4.1.2	Filtering time series.....	17
4.2	Generate Hydrodynamic Forcings.....	21
4.2.1	Wind forcing.....	21
4.3	Generate Float Track Source.....	22
5	Setup of the MIKE 21 Flow Model FM.....	25
5.1	Specifications for the MIKE 21 Flow Model FM.....	25
5.2	Basic Parameters.....	26
5.3	Hydrodynamic Module.....	29
5.4	HD Model Calibration.....	38
5.4.1	Measured values.....	38
5.4.2	Compare model result and measured values.....	41
5.4.3	Generate decoupled model data.....	44
5.5	The Particle Tracking Module.....	45
5.6	PT Model Calibration 1.....	50
5.6.1	Measured values.....	50
5.6.2	Model results compared with measured values.....	53
5.7	PT Model Calibration 2.....	54
5.7.1	Changed model specifications.....	54
5.7.2	Model results compared with measured values.....	55
5.8	PT Model Calibration 3.....	57
5.8.1	Changed model specifications.....	57
5.8.2	Model results compared with measured values.....	58
6	References.....	59

1 Introduction

This Step-by-step training guide relates to the Great Belt Fixed link, which is located between the Danish islands Funen and Zealand (see Figure 1.1).



Figure 1.1 Map showing the Great Belt area, which is located between Funen and Zealand, Denmark

1.1 Background

In 1989 the construction of an 18 km long link across the Great Belt in the Danish waters was initiated. The Great Belt Fixed link consists of a road suspension bridge and railway tunnel between Zealand and the islet Sprogø, as well as a box girder bridge between Sprogø and Funen.



Figure 1.2 The suspension bridge, known as the Eastern Bridge, is a part of the Great Belt Link connection and located between Zealand and Sprogø

As the establishment of causeways and bridge piers in connection with the Great Belt Link construction work inevitably would have increased the flow resistance in the Great Belt and thereby have affected the supply of saline, oxygen rich water from the North Sea into the Baltic Sea, it was decided to carry out compensation dredging of the sea bottom thereby equalising the flow resistance. Furthermore, it was foreseen that the compensation dredging activities, together with other dredging operations connected with the construction work, were likely to cause significant environmental impact caused by the spill of dredged material.

The Great Belt Sediment Plume Model, SPM, was developed to serve as a tool for the planning and monitoring of the dredging operations taking place in connection with the implementation of the fixed link.

The SPM model was applied in the planning and monitoring of the various dredging operations in connection with the Link, and before and during the construction extensive measurement campaigns were conducted.

The water density in Kattegat is larger than the density of the Baltic Sea. This means that if the water level is the same at the north and the south boundary, the water will actually flow to the south, but the MIKE 21 Flow model will simulate no flow. The difference between the water densities at the south and north boundary depends on the hydrodynamics of the previous period. Thus, this must be considered when calibrating the model.

During time periods where the difference in water level between Kattegat and the Baltic Sea is small or in shallow areas in shelter of the main current in the Belt, the current is dominated by interactions between wind and friction forces. The current may vary significantly in time and space according to local wind phenomena, such as sudden changes in speed and direction. This is a difficult current field to describe within the MIKE 21 model, and tracks of floats placed in areas or periods like these are difficult to reproduce.



Figure 1.3 Spilled sediment during construction of pier base in the case of south-going currents. The image view is directed towards south-west

1.2 Objective

The objective of this Step-by-step training guide is to set up a Particle Tracking MIKE 21 Flow Model with flexible mesh (MIKE 21 Flow Model FM, Particle Tracking Module) for the Great Belt area from scratch and to calibrate the model to a satisfactory level.

MIKE 21 is a one-layer hydrodynamic model and thus the model does not include the vertical density gradients. The water in the Great Belt is stratified most of the time. In some periods the two layer effects are so important that the flow field cannot be described satisfactorily within a one-layer model.

With the restrictions of the model in mind, the focus in this example is on particle tracking and it is assumed that you are familiar with setting up a hydrodynamic MIKE 21 Flow Model FM. If not please refer to the Step-by-step 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 Training Guide](#)

Attempts have been made to make this exercise as realistic as possible although some shortcuts 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, time consuming but indispensable process. In this example the 'raw' data have been provided as ASCII text files. Moreover, the hydrodynamic model could be calibrated to a higher extent than in this example.

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 already familiar with importing data into MIKE Zero format files, you do not have 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 MIKE Zero Project

2.1 The Role of the MIKE Zero Project in this Example

MIKE Zero enhances the ability to control and organise information related to almost all kinds of engineering studies including model simulations.

A MIKE Zero project is a collection of files, positioned in one or several folders recursively under a folder where the overall project file is positioned. For example, a DHI project will typically hold separate folders for measured data, model input, model results and documentation.

By selecting a template that fits the aim of the study, the logical organisation of the project folders helps streamline the work by adding informative, intuitive and helpful features. However, in this example only the directly relevant files for the study are given as part of the installation and the work flow is described in the present manual.

The MIKE Zero project will in this case mainly be useful for getting an overview of the available data and quick access to the files. It is thus decided to create the MIKE Zero project file, not by using any of the pre-defined templates, but by using the option 'Project from Folder'. This option creates a project where all existing files within the folder and subfolders are included in the project by default, maintaining the folder structure.

The creation and use of MIKE Zero projects are explained in larger detail in the MIKE Zero Step-by-step guide, which can be accessed via the MIKE Zero Documentation Index in the start menu:

[MIKE Zero, Project Oriented Water Modelling, Step-by-Step Training Guide](#)

2.2 Creating the Project File

To create a new project file showing the files supplied with the installation, open the MIKE Zero shell and click File→New→Project from Folder..., see Figure 2.1.

In the New Project from Folder dialogue, select the folder that contains the installed example and name the project. You can specify the Project Name anyway you like. In this case it is named 'Great Belt Project – Particle Tracking', see Figure 2.2.

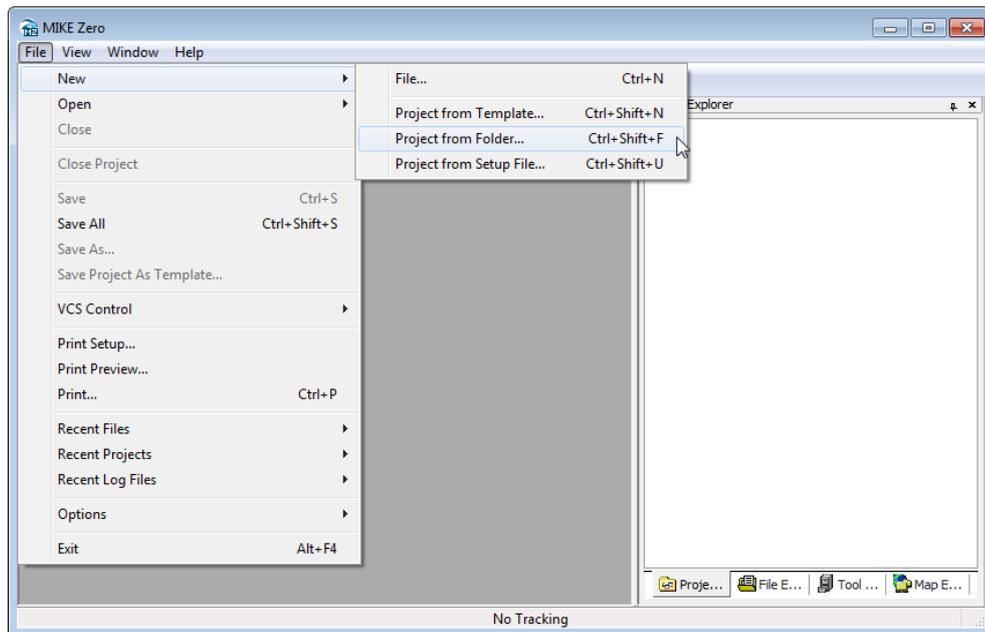


Figure 2.1 Selecting to create a new project from folder

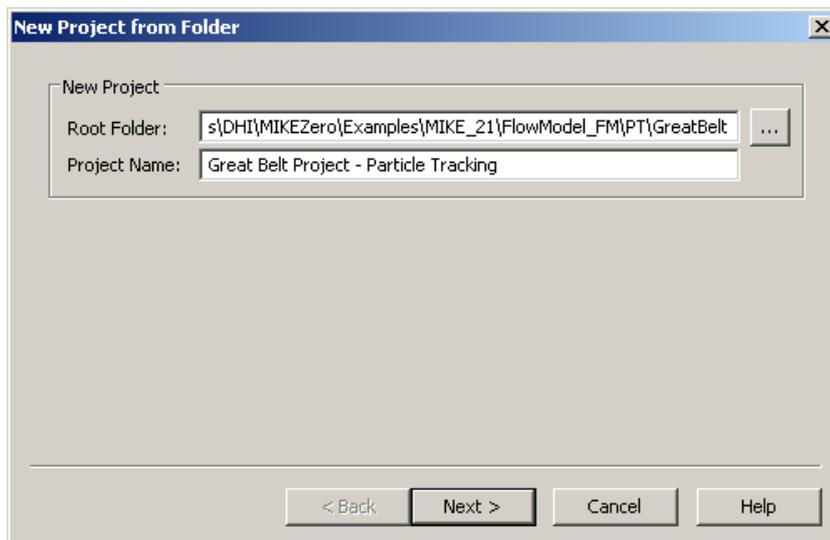


Figure 2.2 Specifying the root folder and project name for a new MIKE Zero project

The next dialogue lists the subfolders and files within the root folder and subfolders to be included in the MIKE Zero project, see Figure 2.3. All files and folders are selected by default, but you have the option to unselect some before creating the project.

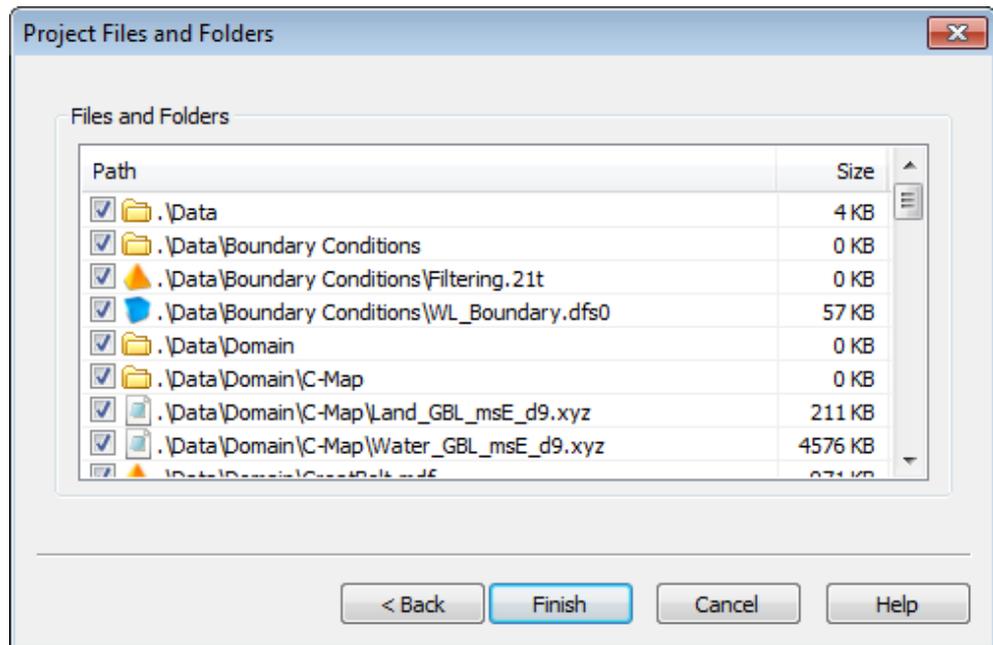


Figure 2.3 List of files and folders in root folder for MIKE Zero project

Press Finish and the project will now automatically be created with the selected files and folders. The project will be opened and shown in the Project Explorer in the MIKE Zero shell. By using the Project Explorer and File Explorer you can get an overview of all the files and folders contained in the project, see Figure 2.4 and Figure 2.5.

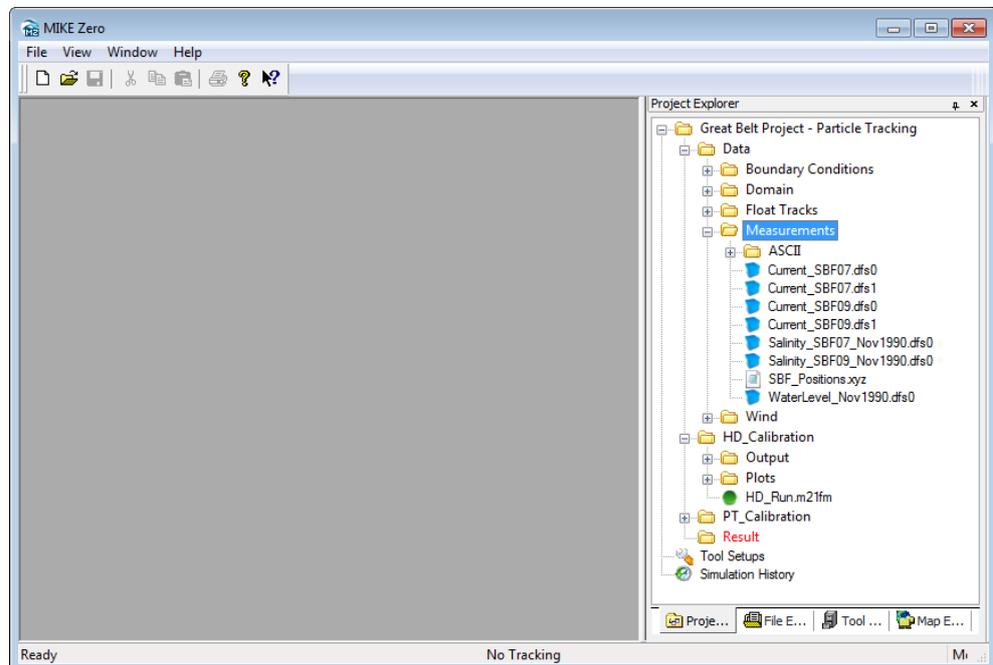


Figure 2.4 Project Explorer view showing the folders and some of the files supplied for the example by the installation.

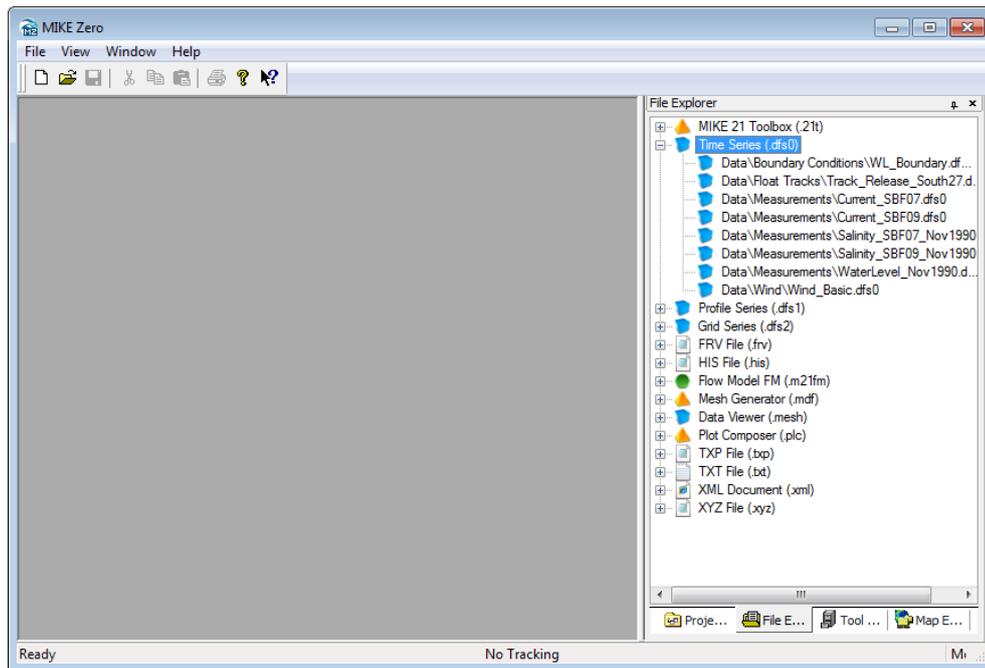


Figure 2.5 File Explorer view showing the file types and the time series (dfs0) files supplied for the example by the installation

It is also possible to include the pdf-file containing the present Step-by-step guide in the project. This will enable a quick access to the description of the example.

Right-click the project in Project Explorer, select 'Add existing file...' and select the document MIKE_FM_PT_Step_By_Step.pdf located in the manuals directory. Select the Copy Mode to create short-cut to the pdf-file. The pdf-file can now be easily accessed through the Project Explorer, see Figure 2.6.

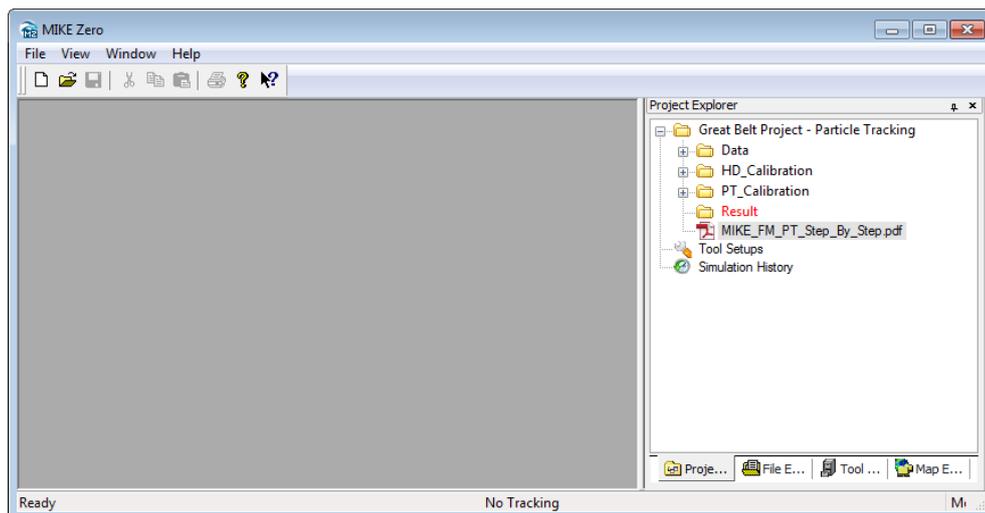


Figure 2.6 Shortcut to pdf-file included in the MIKE Zero project

3 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 for positions of land-boundaries and an xyz file that couples geographical positions with water depths. A bathymetry in dfs2 format 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 Step-by-step 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 Training Guide](#)

When creating the computational mesh you should work with the mdf-file, which contains information about the settings for the mesh. The focus in this example is on Particle Tracking, so the necessary bathymetry information (here given as xyz files and an old MIKE 21 bathymetry), the mdf-file and the mesh-file are already supplied with the example, so that the training time can be spent on specific Particle Tracking subjects.

The mesh, which is used in the example, is shown in Figure 3.1 and Figure 3.2 shows a graphical presentation of the mesh as it can be displayed with MIKE Animator.

The applied coordinate system is the SBF system, which is a local Transverse Mercator projection especially created for the construction of the Great Belt link.

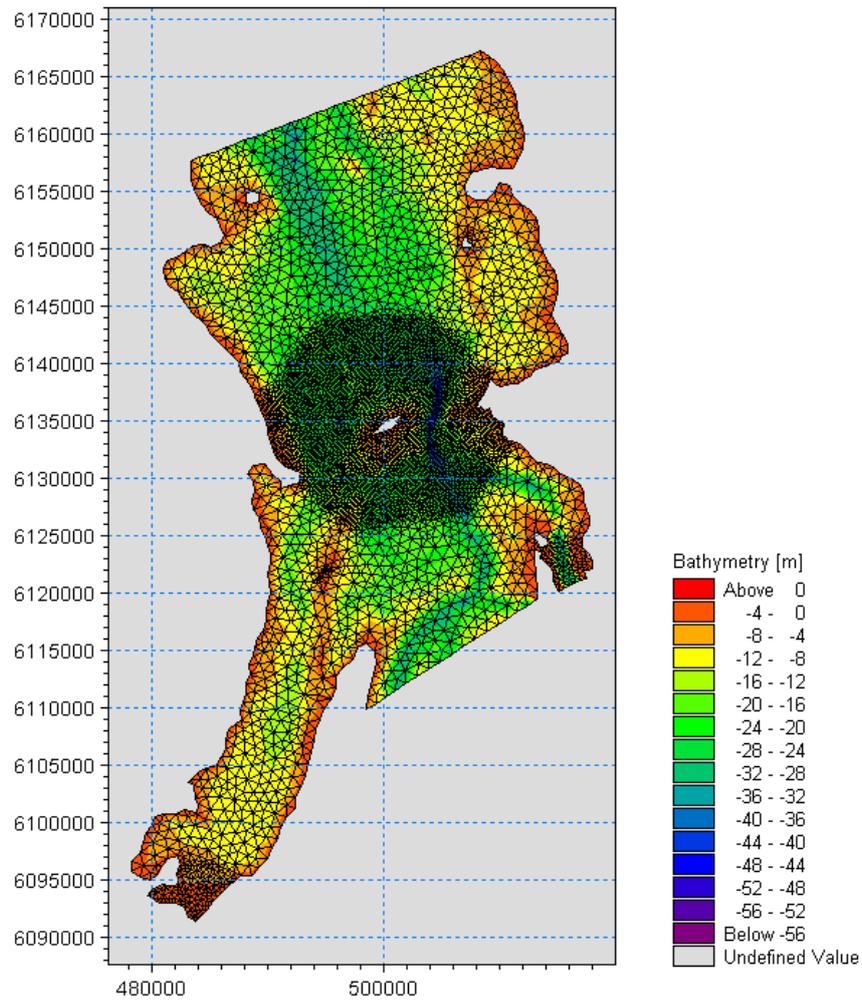


Figure 3.1 The Great Belt computational mesh as it can be presented with the Data Viewer tool

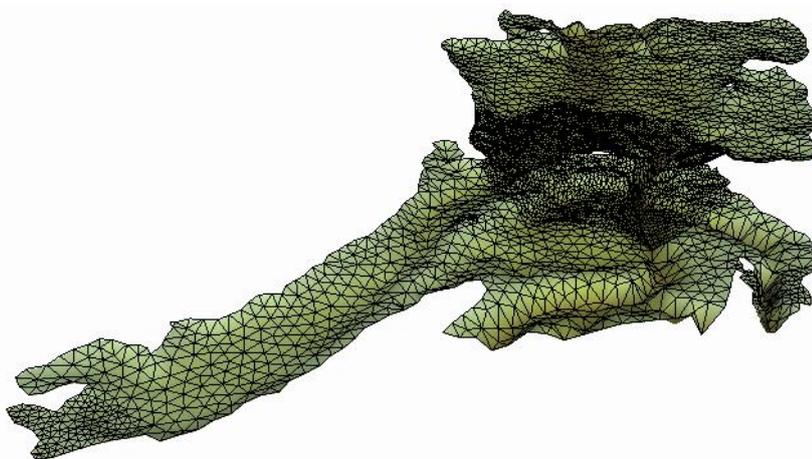


Figure 3.2 The Great Belt computational mesh as it can be presented with the MIKE Animator tool

4 Creating the Input Parameters to the MIKE 21 Flow Model FM

Before the MIKE 21 Flow Model FM can be set up, the input data to the model must be generated from the data collected.

The following data have been applied for this example:

1. Water levels at four stations (SBF02, SBF11, SBF13 and SBF14) in the Great Belt area
2. Current measurements at station SBF07 and SBF09
3. Wind measurements from Sprogø (SBF08)

The measured raw data from November 1990 are included in the folder:

.\MIKE_21\FlowModel_FM\PT\GreatBelt\Data\Measurements\ASCII

A map showing the positions of the monitoring stations is shown in Figure 4.1. The coordinates are given in Table 4.1.

Table 4.1 Measurements stations in the Great Belt Area. Projection SBF (ED50)

Station	Easting [m]	Northing [m]	Measurement
SBF-02	490435	6159807	Water level
SBF-07	495476	6131135	Current
SBF-08	500106	6134282	Wind
SBF-09	503449	6132231	Current
SBF-11	502045	6112259	Water level
SBF-13	517943	6120092	Water level
SBF-14	483443	6090708	Water level

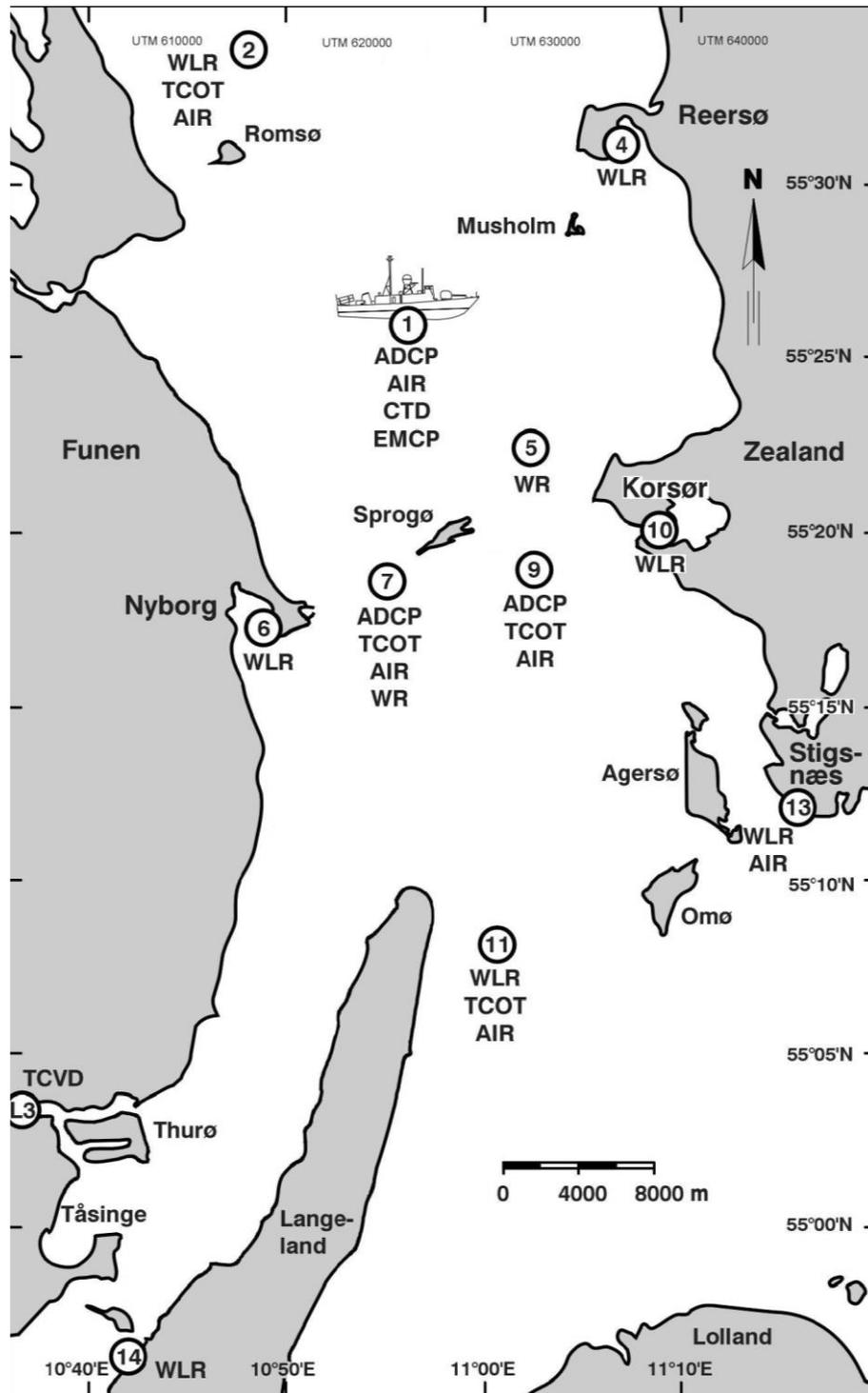


Figure 4.1 A map of the Great Belt showing the positions of monitoring stations:
WLR: Water level recordings
ADCP: Current measurements (Acoustic Doppler Current Profiler)

Furthermore, various float tracks have been conducted during 27 November 1990. These measurements will be used for the calibration of the Particle Tracking model.

We have chosen to start the simulation period at 21/11 1990 18:00 and end it at 28/11 1990 00:00. The period prior to the float track measurements can then be used for calibration of the hydrodynamic model.

4.1 Generating Hydrodynamic Boundary Data

The water levels at the various monitoring stations in the Great Belt area have been measured. To simulate the flow in the area, measured water levels south and north of the belt area are used.

The computational domain contains 5 open boundaries, which are located in the vicinity of a water level measuring station, see Figure 4.2.

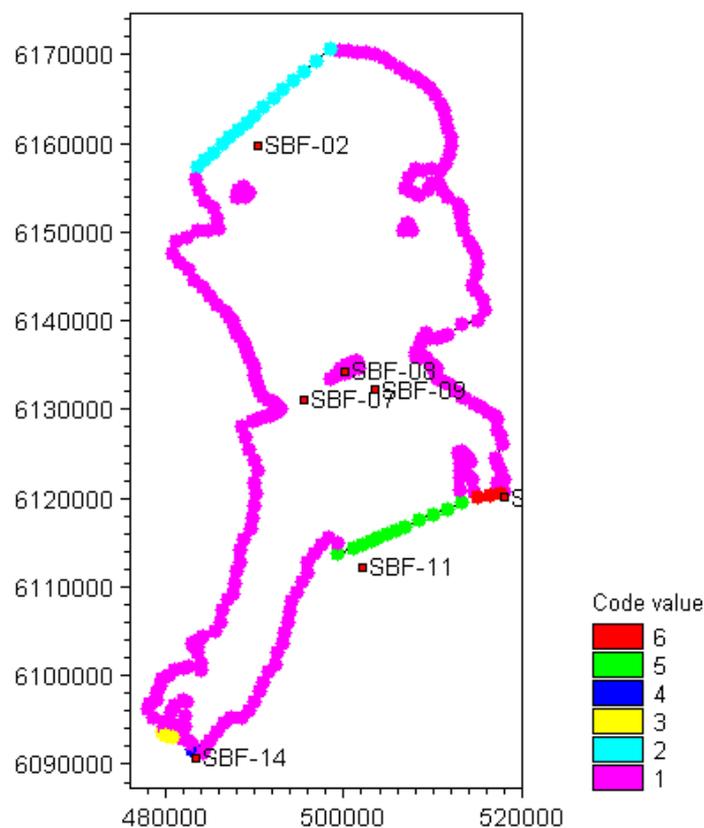


Figure 4.2 Position of boundaries and related measuring stations

The extension of the model area is limited and the water levels along the individual boundaries are not expected to have significant variation, which will influence the results. Therefore, it is assumed that it is sufficient to use a constant water level along each boundary.

4.1.1 Importing measured water levels to time series file

The water levels were measured by soundings. The results were saved in text files with four columns (see Figure 4.3):

- Column 1: Date and time in UTC (Number of seconds from 1 January 1970 at 00:00:00, the time step is 0.5 hours)
- Column 2: Water level (m) DNN
- Column 3: Soundings (m/s)
- Column 4: Reference target number

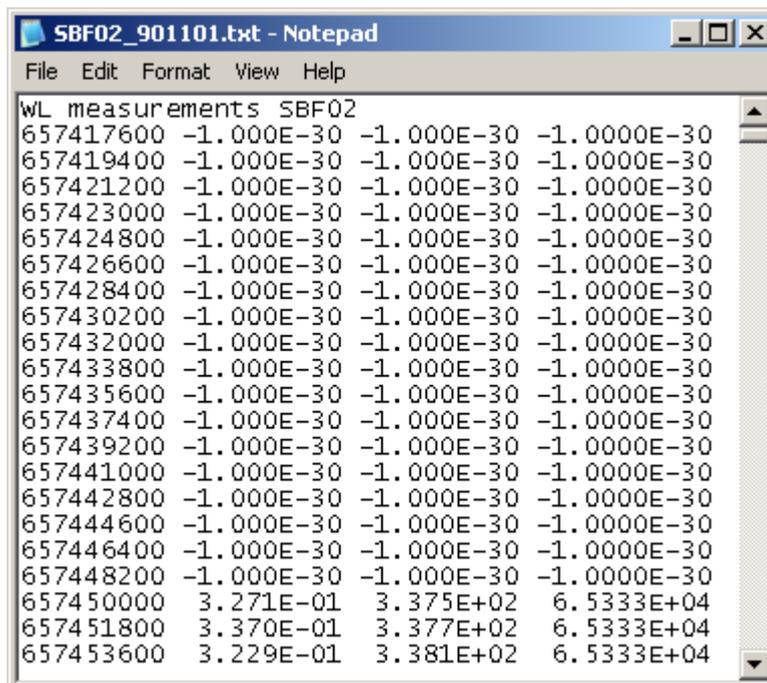


Figure 4.3 Example of measurements: water level recordings from Station SBF02 in November 1990.

To simplify the pre-processing of the data, all columns are imported initially in time series format, but it will only be column 2 with the water levels that will be used.

Open the Time Series Editor in MIKE Zero (File→New→File→Time Series), see Figure 4.4. Select the ASCII template and open the text file 'SBF02_901101.txt'.

Please note that time series must have equidistant time steps in MIKE 21 & MIKE 3 Flow Model FM. This means that if the raw data have time gaps without measurements, the gaps in the raw data must be filled before importing them.

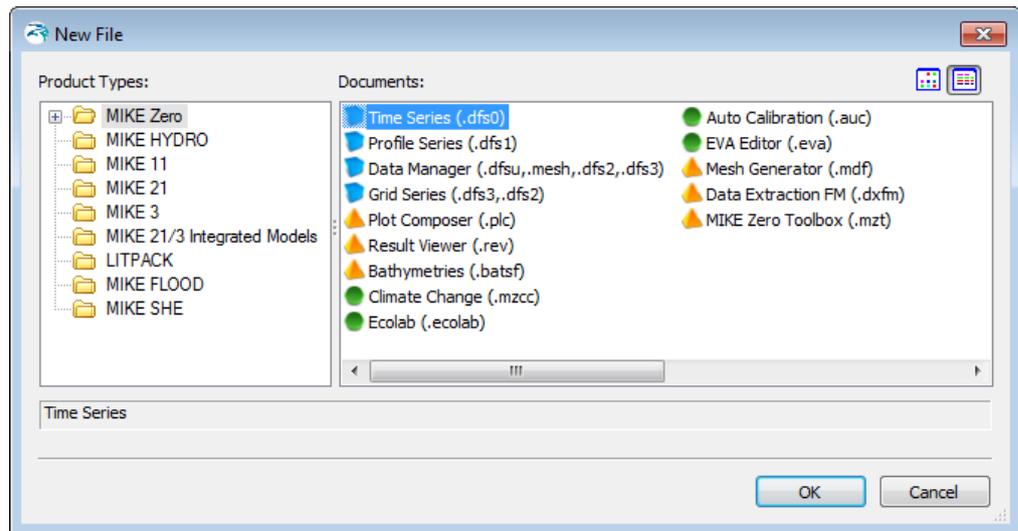


Figure 4.4 Starting the Time Series Editor in MIKE Zero

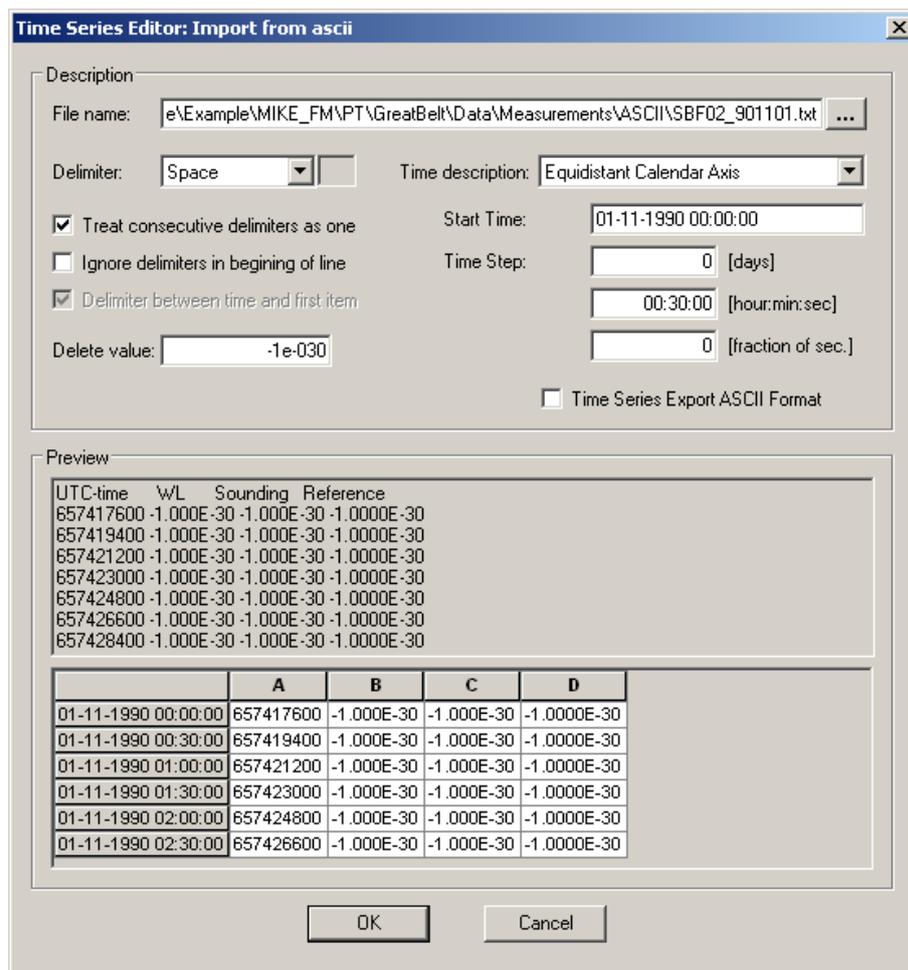


Figure 4.5 Time Series Editor Import Template

On the time series property page change the time description to 'Equidistant Calendar Axis', set the start time to 1 November 1990 at 00:00:00 and the time step to 30 minutes. Set the remaining parameters as shown in Figure 4.5 and press OK.

Then right-click on the generated data, select properties and change the title and type of item 2 to 'Water Level', see Figure 4.6. Save the data in the time series file WL_SBF02.dfs0.

The resulting time series can be seen in Figure 4.7.

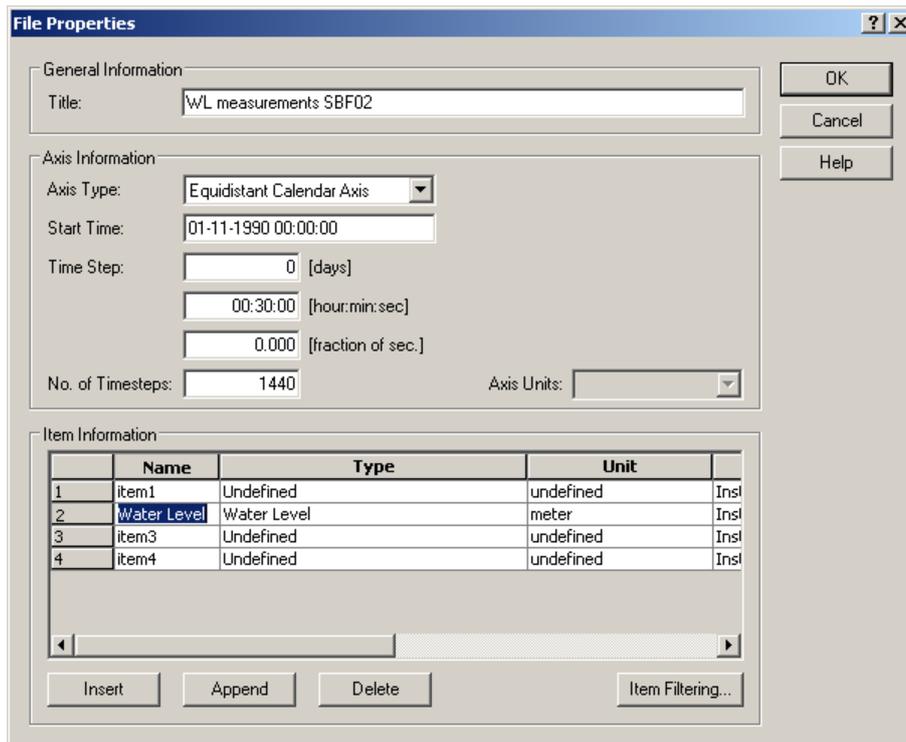


Figure 4.6 Time Series Properties

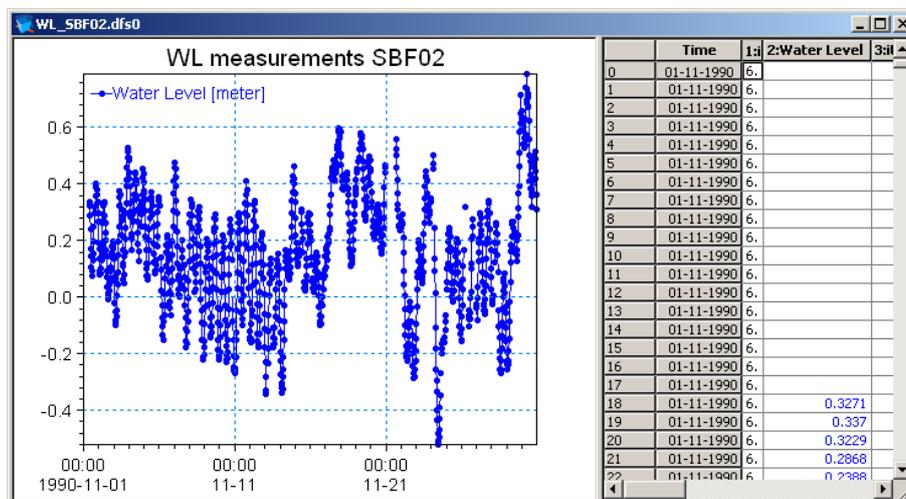


Figure 4.7 Time Series Editor with imported Water Levels from Station SBF02

Repeat the steps above to import the remaining water level recordings into time series format. For the coming processing, the measured water levels have been collected in a file named 'Waterlevel_Nov1990.dfs0'.

4.1.2 Filtering time series

To obtain a smooth variation of water levels, the measured water level recordings may be low pass filtered. This will ensure a better performance of the model.

The time series of measured water levels may contain delete values where no measurements were available. Before filtering the data these delete values must be replaced by real values. In this case the delete values are omitted from the time series by interpolating the data using the Time Series Editor (Tools→Interpolation) and saved in the file 'Waterlevel_Nov1990_1.dfs0'.

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 4.8.

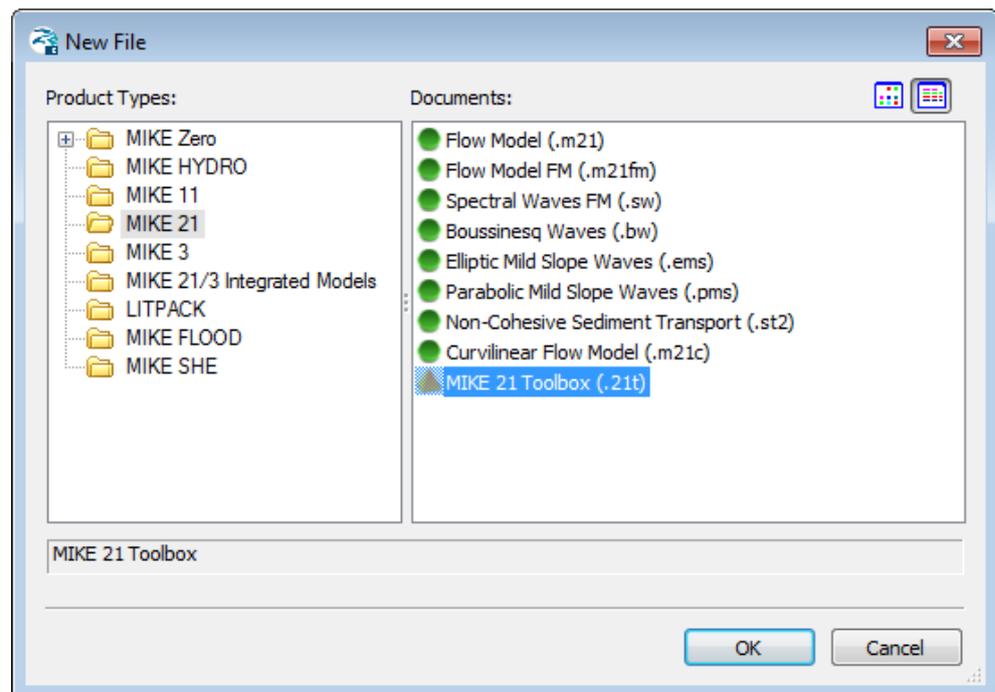


Figure 4.8 Selecting the MIKE 21 Toolbox in MIKE Zero

Select the tool *FFT Filtering of Time series* in the *Waves* group, see Figure 4.9.

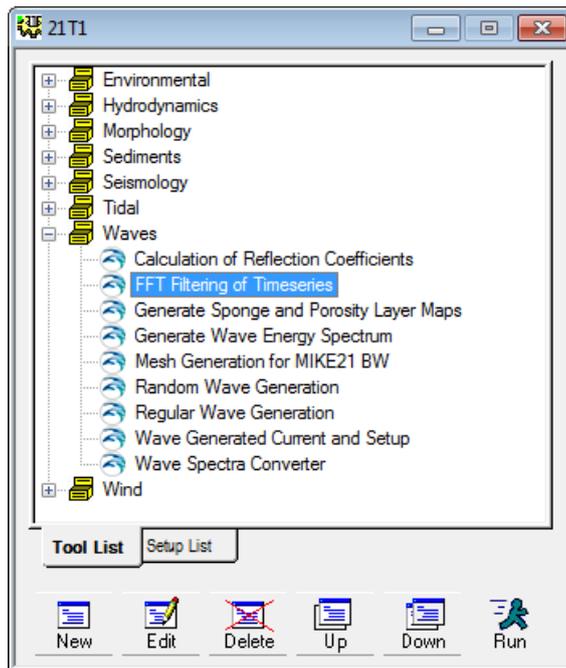


Figure 4.9 Selecting the FFT Filtering of Time Series tool in MIKE 21 Toolbox

Press the *New* button and follow the dialogues. Select the interpolated time series file as the input file. In the next dialogue box, define the frequency range as shown in Figure 4.10. The maximum frequency of interest is set to 9.259e-005 Hz, corresponding to 3 hours.

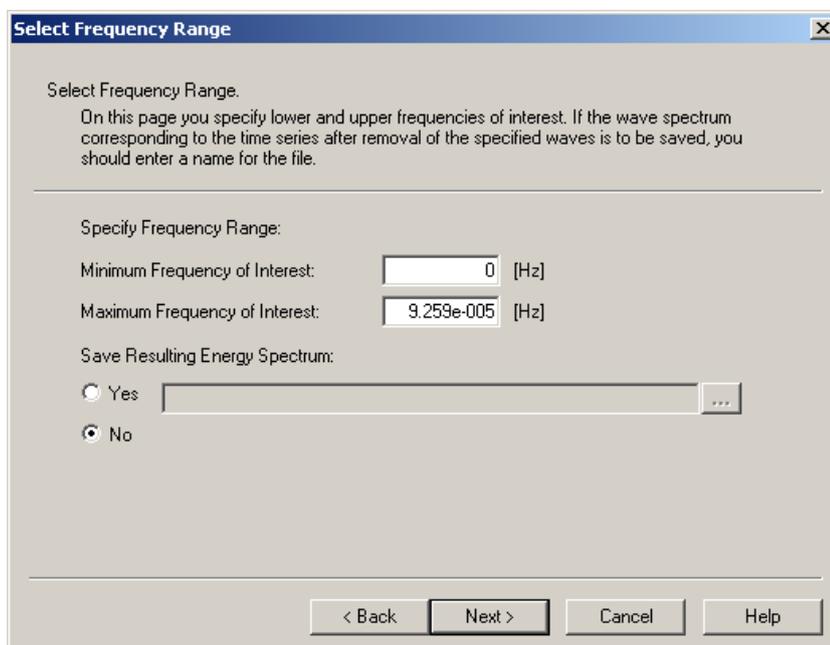


Figure 4.10 Specifying frequency range in FFT Filtering of Time Series tool

Save the data as 'WL_Boundary.dfs0'.

Now you are ready to use the values in this file as boundary conditions in the simulation.

To make a plot of the water level time series open the Plot Composer in MIKE Zero (File→New→File→Plot Composer), see Figure 4.11. Select 'plot' → 'insert a new plot object' and select 'Time Series Plot' (see Figure 4.12).

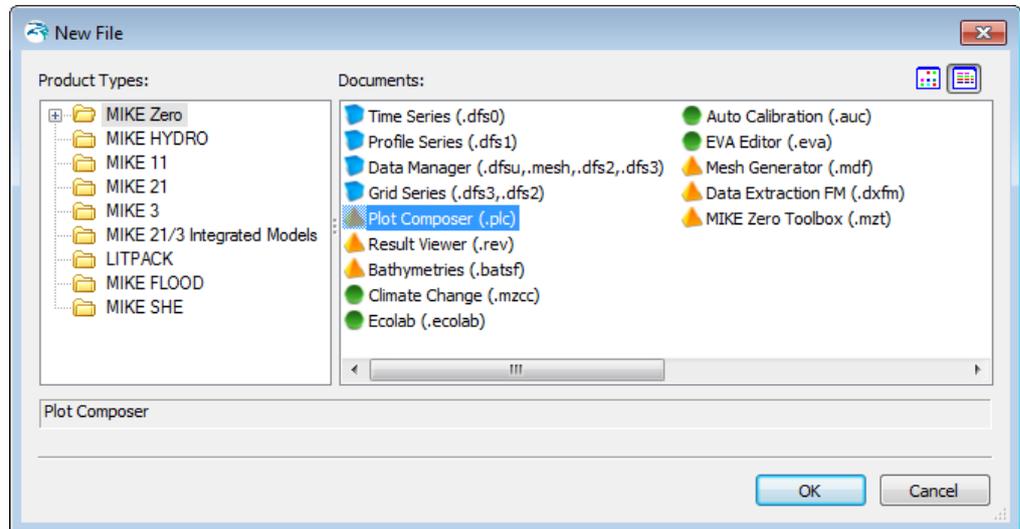


Figure 4.11 Starting the Plot Composer in MIKE Zero

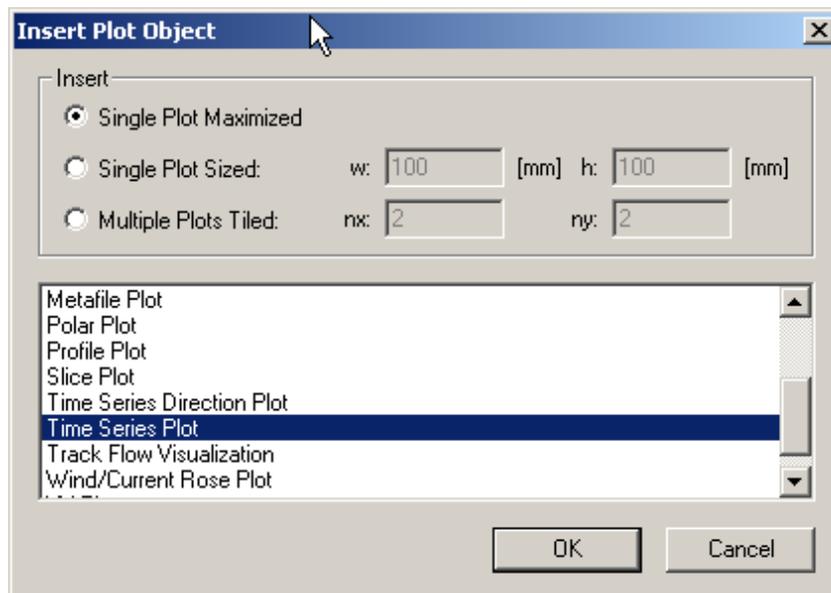


Figure 4.12 Insert a new Plot Object as Time Series in the Plot Composer

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 4.13. 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 colours, etc. (see Figure 4.14).

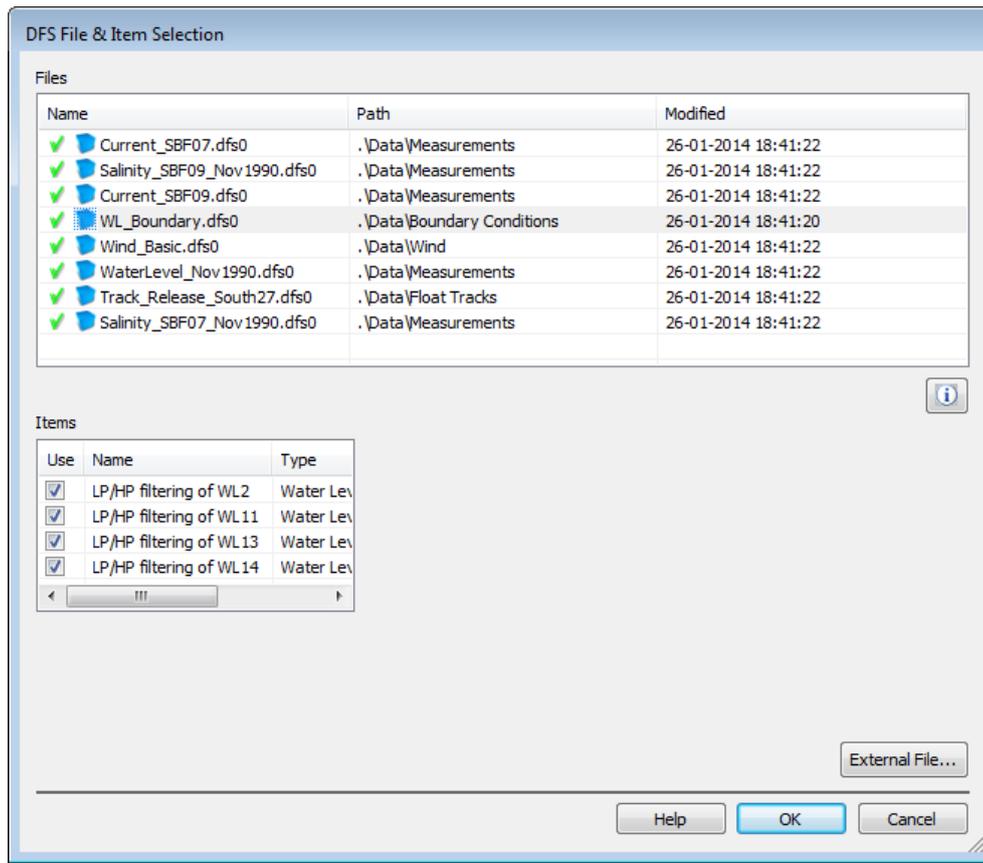


Figure 4.13 Selection of time series files in the Plot Composer

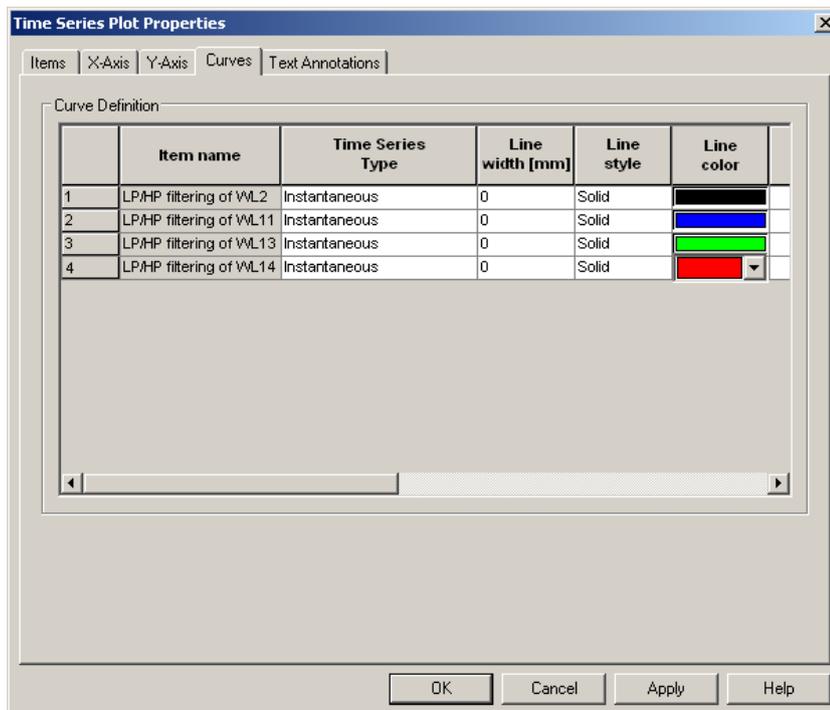


Figure 4.14 Plot Composer Time Series Plot Properties dialogue for selecting time series files and adjusting scales, curves, etc.

Figure 4.15 shows the measured water levels during the simulation period for the four measuring stations.

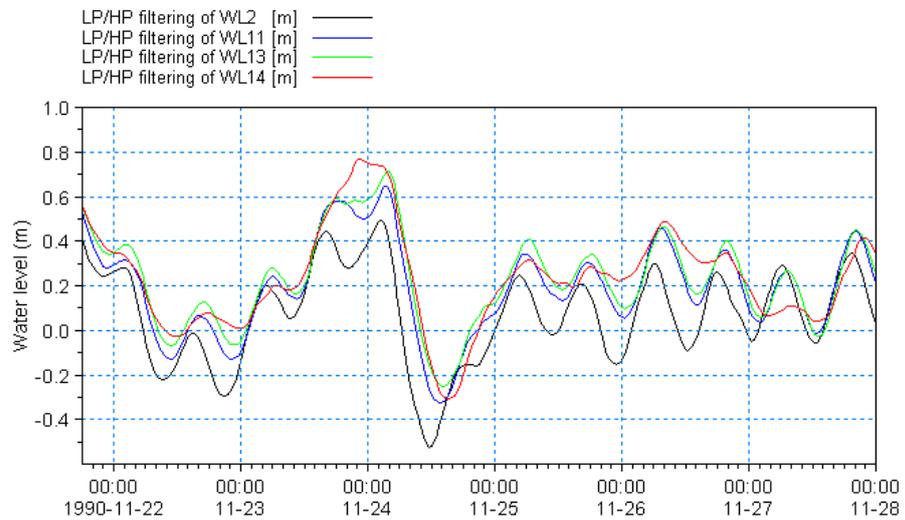


Figure 4.15 The four boundary water level time series plotted with Plot Composer

4.2 Generate Hydrodynamic Forcings

The measurements of wind should also be entered in MIKE Zero format.

4.2.1 Wind forcing

Wind recordings from Sprogø (SBF08) will form the wind forcing as time series constant in space. The recordings were saved in text files with three columns (see Figure 4.16):

- Column 1: Date and time in UTC (Number of seconds from 1 January 1970 at 00:00:00, the time step is 3.0 hours)
- Column 2: Wind direction (deg N) at 10 metres above surface
- Column 3: Wind speed (m/s) at 10 metres above surface

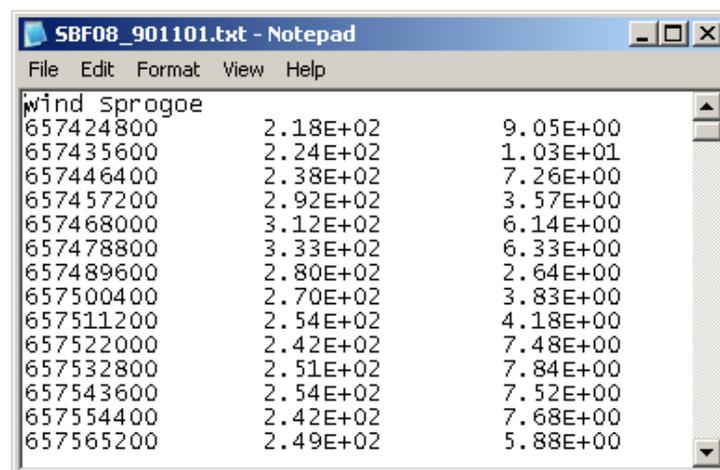


Figure 4.16 ASCII file with wind speed and direction from Sprogø

Load the time series editor and import the ASCII file 'SBF08_901101.txt' with equidistant calendar axis as described in Section 4.1.1. Save the file in 'wind.dfs0'. Time series of the wind speed and direction can be illustrated with the Time Series Direction Plot in Plot Composer, see Figure 4.17.

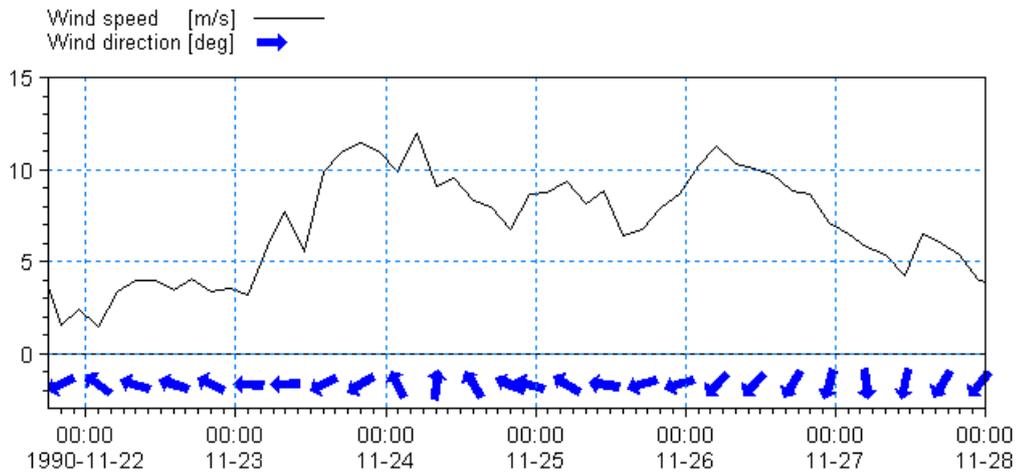


Figure 4.17 Wind speed and direction at Sprogø during simulation period as it can be illustrated in the Plot Composer with the Time Series Direction Plot template

4.3 Generate Float Track Source

Float tracking measurements were conducted in the area south of Sprogø in an area with water depths varying between 15 and 20 metres, from 8 am to 12.30 pm on 27 November 1990.

The floats were placed at a depth of 2.5 meters. A listing of the geographic positions is given in Table 4.2 and a plot is shown in Figure 4.18.

Table 4.2 Measured float track positions in time

Float number	Time GMT	Easting (m)	Northing (m)	Duration (s)
1	07:59:25	502951	6130519	14242
	09:25:13	502623	6129991	
	10:52:14	502185	6129660	
	11:56:47	501586	6129519	
2	08:09:06	501785	6130495	14764
	09:31:04	501416	6129843	
	10:59:17	500562	6129327	
	12:15:10	499627	6129285	
3	08:13:53	500984	6130507	14876
	09:35:32	500412	6129812	
	11:04:27	499359	6129394	
	12:21:49	498453	6129400	
4	08:19:49	499987	6130499	14891
	09:40:29	499300	6129789	
	11:08:52	498327	6129369	
	12:28:00	497370	6129442	
5	08:25:54	498967	6130493	14795
	09:44:31	498402	6129623	
	12:32:29	496799	6129173	
7	08:31:12	497941	6130495	14851
	09:48:07	497645	6129730	
	11:14:39	496983	6129409	
	12:38:43	496175	6129702	
8	08:36:05	496974	6130501	4598
	09:52:43	496848	6130213	
10	09:43:04	495687	6130508	8986
	12:44:34	495401	6130299	

Each of the floats were released at a certain position and at a certain time. To model this in the particle tracking model, a time series file was generated with 8 items - each item describing the source of one of the floats. The position of the source will be defined by a fixed location.

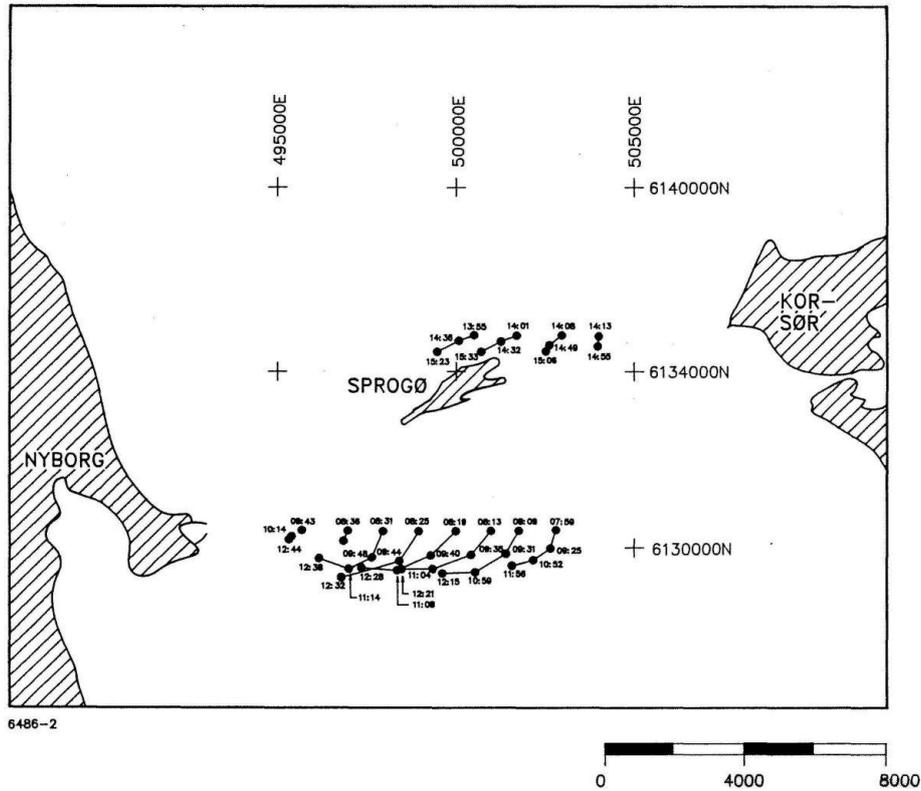


Figure 4.18 Position of the floats in the Great Belt on 27 November 1990. A listing of the geographic positions is given in Table 4.2

Only one particle is to be applied per float track, thus to simplify the setup, the time step of the time series file is set to the overall time step. A dummy value of 100 kg/s is used for the magnitude at the given time step, and in the remaining time the magnitude is 0. The generated time series file will look similar to the one shown in Figure 4.19.

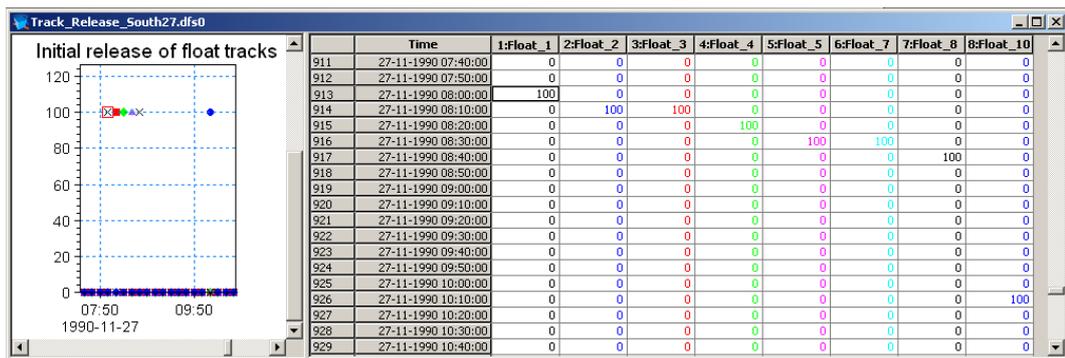


Figure 4.19 Time series file describing the source of floats

5 Setup of the MIKE 21 Flow Model FM

5.1 Specifications for the MIKE 21 Flow Model FM

We are now ready to create the specification file for the Particle Tracking modelling using the Great Belt mesh generated in Chapter 2 and the input parameters generated in Chapter 4.

The Particle Tracking technique uses a discrete approach in the calculations with information from the overall time step. This implies that the calculation of particle tracks can be applied in a decoupled mode without losing any accuracy in the results.

Running a simulation in decoupled mode can significantly reduce the computation time. If the basic flow information from a hydrodynamic simulation is saved in a number of special decoupling output files, it is possible to re-run the simulation in a decoupled mode, where instead of calculating the flow information it is read from the output files. Hence, it is decided to calibrate the hydrodynamic model alone, before setting up a decoupled model for the particle tracking simulations. The setup of the first calibration simulation for the hydrodynamic module consists of the parameters shown in Table 5.1.

Table 5.1 Specifications for the calibration simulation of HD

Parameter	Value
Specification File	HD_Run.m21fm
Mesh and Bathymetry	GreatBelt.mesh 4442 Nodes
Simulation Period	1990-11-21 18:00 – 1990-11-28 00:00 (~6 days)
Time Step Interval	600 s
No. of Time Steps	900
HD: Solution Technique	Low order, fast algorithm Minimum time step: 0.01 s Maximum time step: 600 s Critical CFL number: 0.8
HD: Flood and Dry	Standard flood and dry
HD: Initial Conditions	Surface elevation: 0.4 m
HD: Wind	Varying in time, constant in domain: wind_basic.dfs0 Soft start: 21600 s
HD: Wind Friction	Constant: 0.0025
HD: Boundary Conditions	Specified level Type 0 data: WL_Boundary.dfs0 Include Coriolis and wind correction Soft start: 21600 s from 0.4 m
HD: Eddy Viscosity	Smagorinsky formulation, Constant 0.28
HD: Resistance	Manning number. Constant value 28 m ^{1/3} /s
HD: Decoupling	Include, time step frequency: 1
HD: Result Files	HD_out.dfsu, HD_out.dfs0, (+ Decoupled files)
CPU Simulation Time	About 3 minutes using 4 threads with a 70 Ghz PC, 8 GB RAM

The setup of the first calibration simulation for the particle tracking module consists of the parameters shown in Table 5.2.

Table 5.2 Specifications for the first calibration simulation of PT

Parameter	Value
Specification File	PT_Run1.m21fm
Mesh and Bathymetry	GreatBelt.mesh 4442 Nodes
Simulation Period	1990-11-21 18:00 – 1990-11-28 00:00 (~6 days)
Module Selection	Particle Tracking
Hydrodynamics	Decoupled results from HD : Run_HD_DecouplingFlux.dfsu Run_HD_DecouplingArea.dfsu
PT: Number of Classes	8 (one per float track)
PT: Number of Sources	8 Track_Release_South27.dfs0
PT: Decay	Not included
PT: Settling	Not included
PT: Dispersion	Not included
PT: Erosion	Not included
PT: Drift profile	Use raw data from hydrodynamics
PT: Result Files	PT_1.dfsu, PT_1.XML
CPU Simulation Time	A few seconds

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

Additional information about the Particle Tracking Module can be found in the User Guide, which can be accessed via the MIKE Zero Documentation Index in the start menu:

[MIKE 21 Flow Model FM, Particle Tracking Module, User Guide](#)

5.2 Basic Parameters

Specify the bathymetry and mesh file GreatBelt.mesh in the 'Domain' dialogue, see Figure 5.1. The projection zone has already been specified in the mesh as SBF.

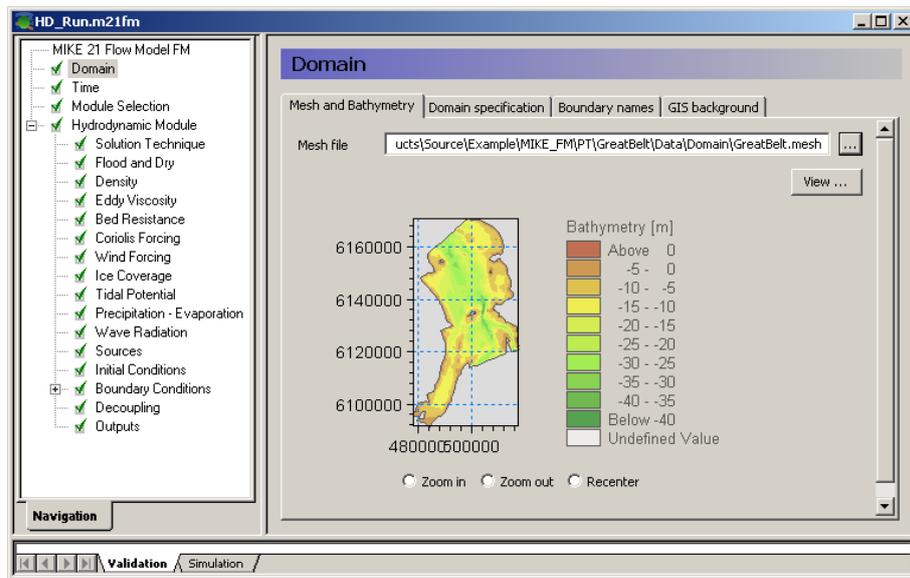


Figure 5.1 MIKE 21 Flow Model FM: Specify Domain

In the mesh file, each boundary has been given a code, see Figure 5.2. In this example the boundaries has Code 2 to Code 6, starting at the north boundary and moving anti-clockwise around the mesh. Rename the boundary 'Code 2' to 'North', the boundary codes 3 and 4 to 'West1' and 'West2', the boundary code 5 to 'South' and the boundary code 6 to 'East' in the 'Domain' dialogue box under 'Boundary names', see Figure 5.2.

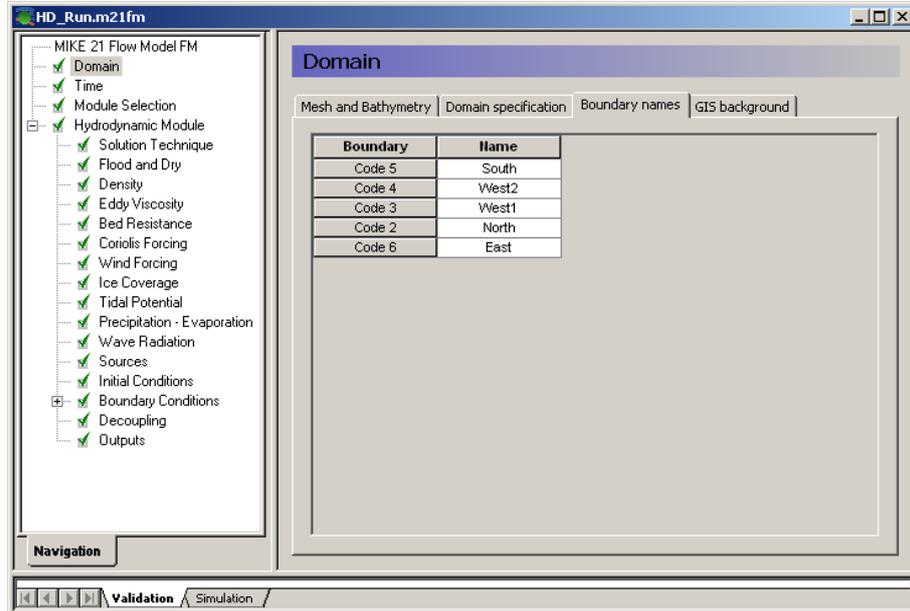


Figure 5.2 M21 Flow Model FM: Renaming boundaries

Specify the overall time step to 600 s in the 'Time' dialogue. The time step range must be specified to 900 time steps in order to simulate a total period of 6 days and 6 hours. Set the simulation start date to 21-11-1990 18:00:00, see Figure 5.3.

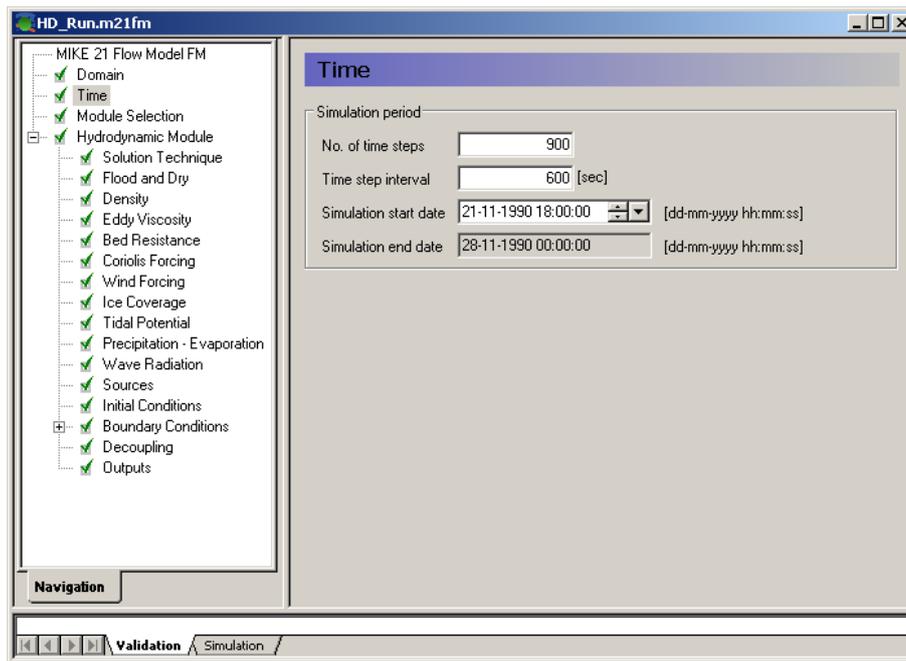


Figure 5.3 MIKE 21 Flow Model FM: Simulation period

In this example only the Hydrodynamic Module will be used, see Figure 5.4.

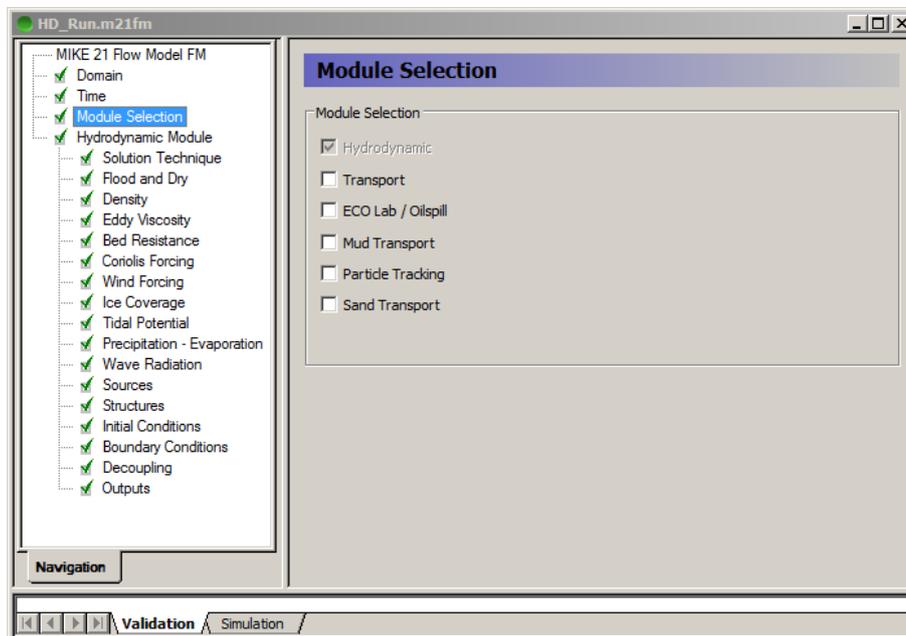


Figure 5.4 MIKE 21 Flow Model FM: Module Selection

5.3 Hydrodynamic Module

The shallow water equations are solved using 'Low order, fast algorithm' for both time integration and space integration, see Figure 5.5.

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 600 s.

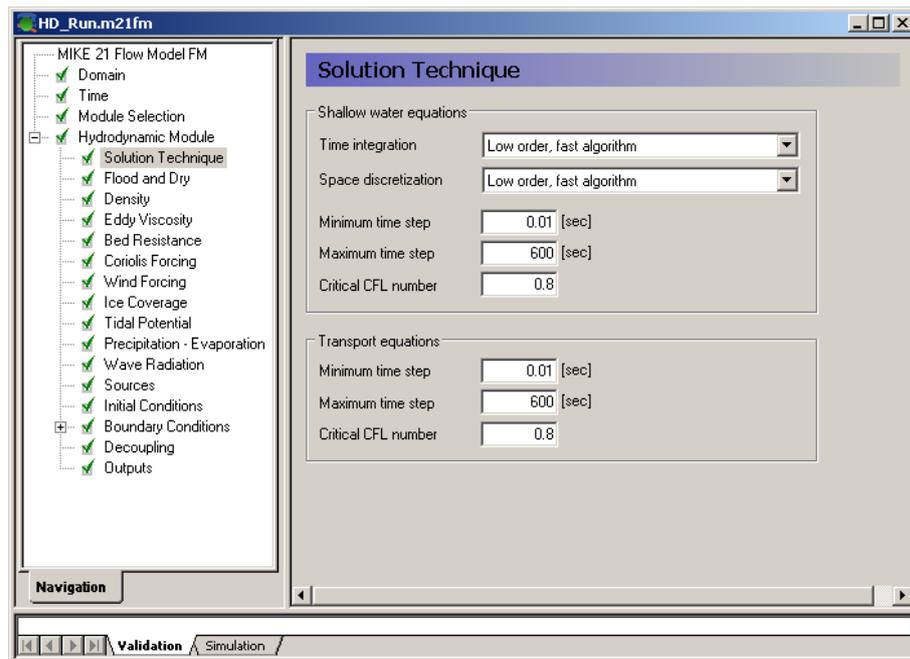


Figure 5.5 MIKE 21 Flow Model FM, Hydrodynamic Module: Solution Technique

In this case the Flood and Dry function is included, see Figure 5.6. You just have to set the Drying depth to 0.005 m and the Flooding depth to 0.05 m. The Wetting depth should be 0.1 m. These are the default values.

Unfortunately, the densities at the boundaries for the domain were not measured during the float track period. As no boundary data exist it is not possible to simulate the density variation in the domain to any realistic values. Thus, it is decided to omit the influence of any possible density variations. As the density variation is not to be taken into account in this example the density should be specified as 'Barotropic' in the 'Density' dialogue, see Figure 5.7.

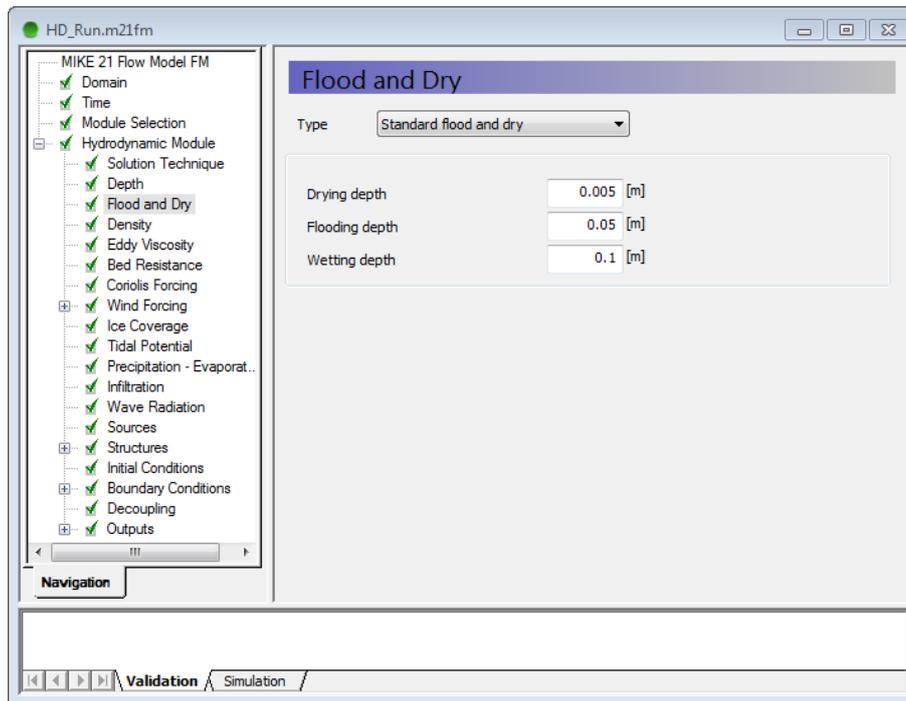


Figure 5.6 MIKE 21 Flow Model FM, Hydrodynamic Module: Flood and Dry

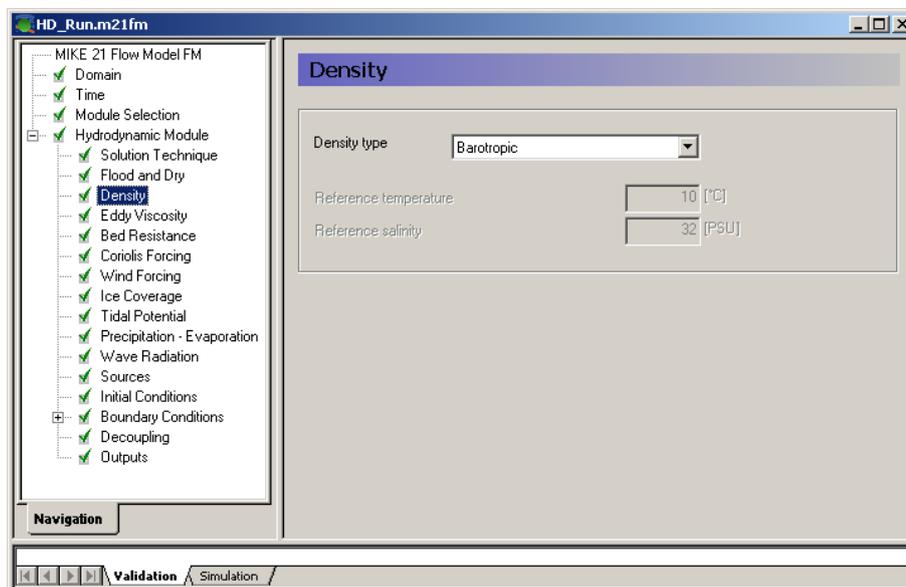


Figure 5.7 MIKE 21 Flow Model FM, Hydrodynamic Module: Density specification

The default setting for the Horizontal Eddy Viscosity is a Smagorinsky formulation with a coefficient of 0.28, see Figure 5.8.

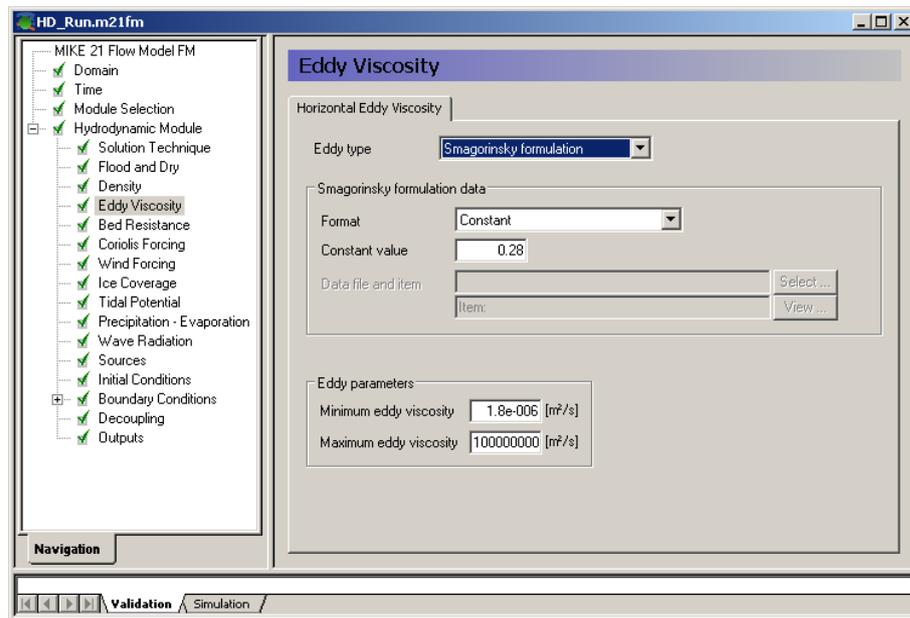


Figure 5.8 MIKE 21 Flow Model FM, Hydrodynamic Module: Eddy Viscosity (Horizontal)

The Bed Resistance will be given as a constant Manning number of $28 \text{ m}^{1/3}/\text{s}$, see Figure 5.9.

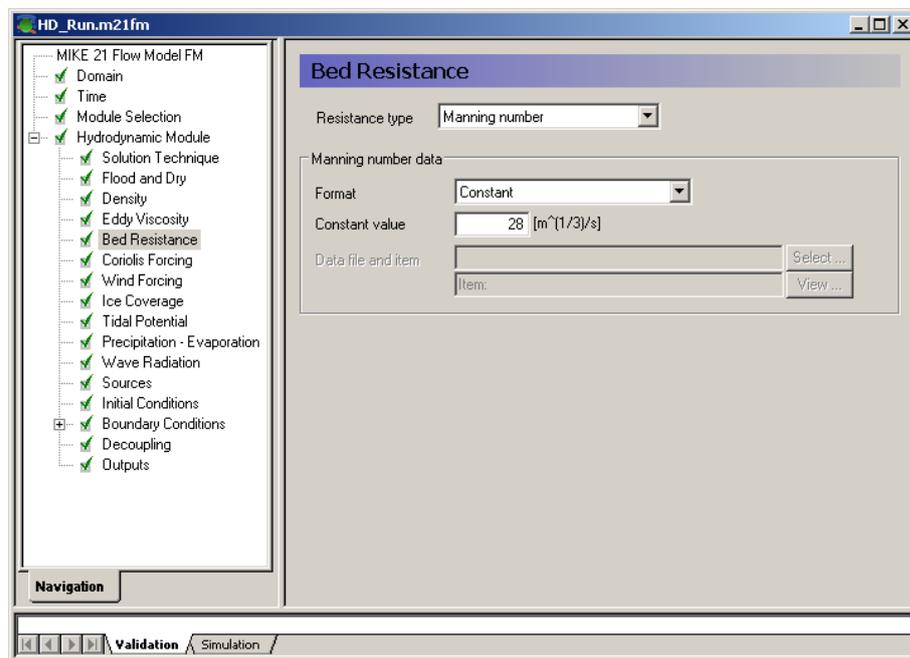


Figure 5.9 MIKE 21 Flow Model FM, Hydrodynamic Module: Bed Resistance

Coriolis forcing is always included in real applications. Only for laboratory type of simulations Coriolis is most often ignored, see Figure 5.10.

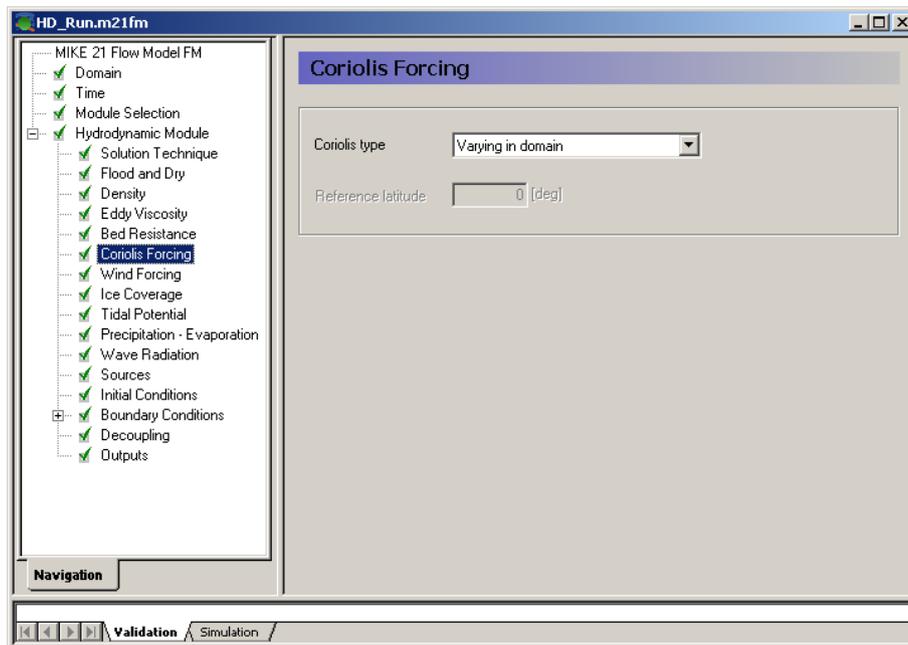


Figure 5.10 MIKE 21 Flow Model FM, Hydrodynamic Module: Coriolis Forcing

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_basic.dfs0.

Please note that an easy way to see the wind data file is to simply click  in the 'Wind Forcing' dialogue, see Figure 5.11.

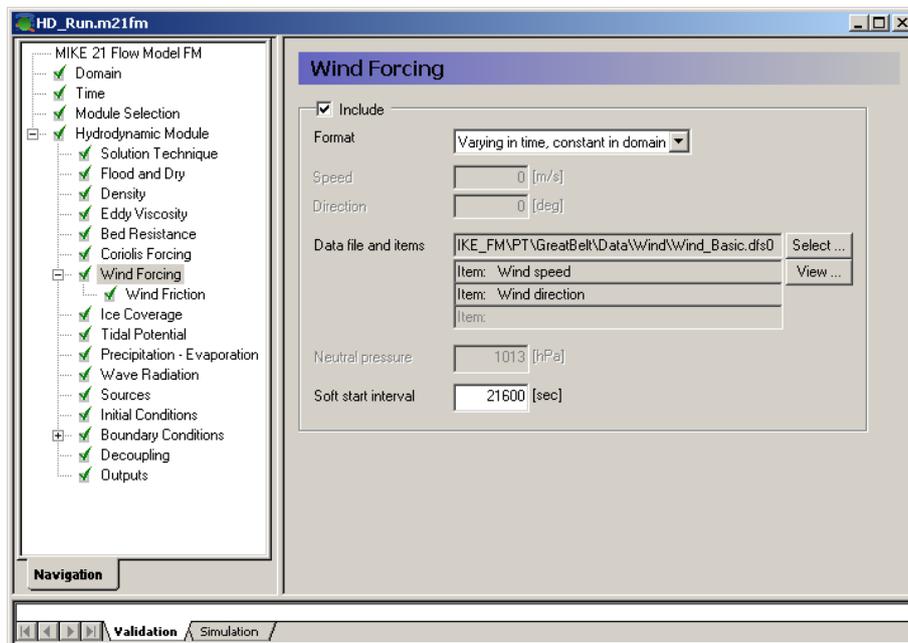


Figure 5.11 MIKE 21 Flow Model FM, Hydrodynamic Module: Wind Forcing

Furthermore, when you run a simulation it is good practice to use a soft start interval. In this case 21600 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 value 0.0025 for the Wind Friction, see Figure 5.12. The wind friction value, which has been set to a higher value than the default value as field measurements of the drag coefficient collected over enclosed areas, such as the Great Belt area, indicates that the drag coefficient is larger here than in the open sea.

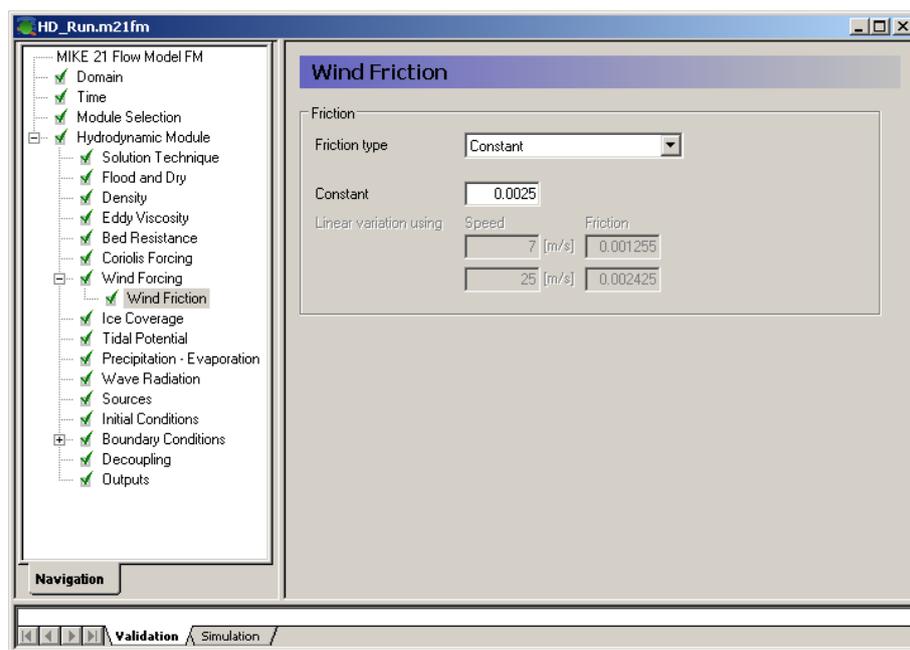


Figure 5.12 MIKE 21 Flow Model FM, Hydrodynamic Module: Wind Friction

Please note that in this example the following parameters are **not** included:

- Ice Coverage
- Tidal Potential
- Precipitation-Evaporation
- Infiltration
- Wave Radiation
- Sources
- Structures

and their respective dialogue pages are not considered further.

Now you should check the boundary conditions at the simulation start time to estimate the initial surface level specified in the 'Initial Conditions' dialogue see Figure 5.13. In this case we will use a surface elevation level of 0.4 m, which is close to the measured value for all the boundaries at the start of the simulation.

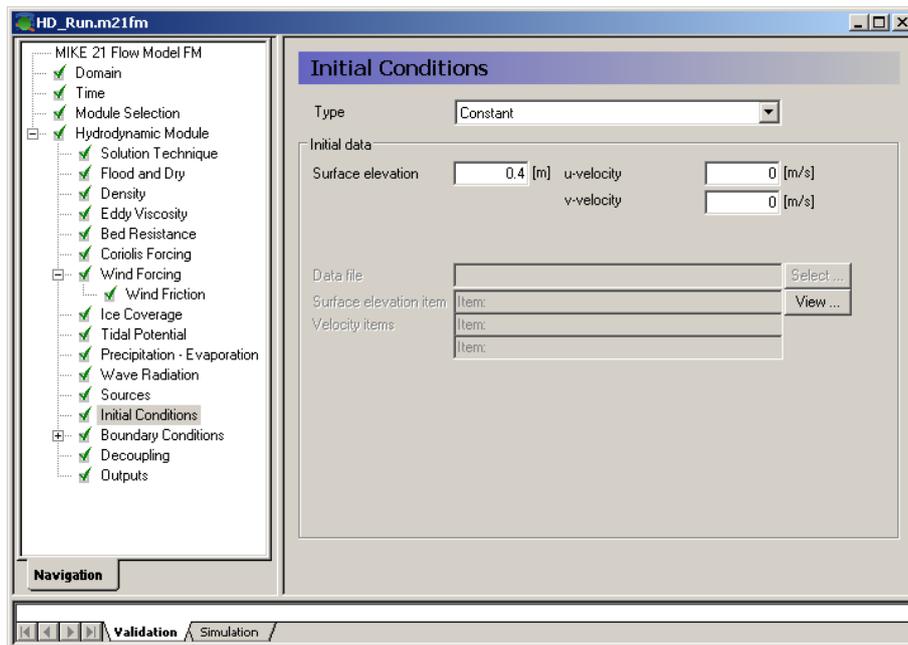


Figure 5.13 MIKE 21 Flow Model FM, Hydrodynamic Module: Initial Conditions

In the 'Boundary Conditions' dialogue, the boundary conditions should be specified for the boundary names, which were denoted in the 'Domain' dialogue (Figure 5.2). There are 5 boundaries; North, West1, West2, South and East.

In this example the boundary type is 'Specified Level' (Water Level), because only Water Level measurements are available, see Chapter 3. '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 (e.g. WL_Boundary.dfs0) in the Open File window that appears, see Figure 5.14.

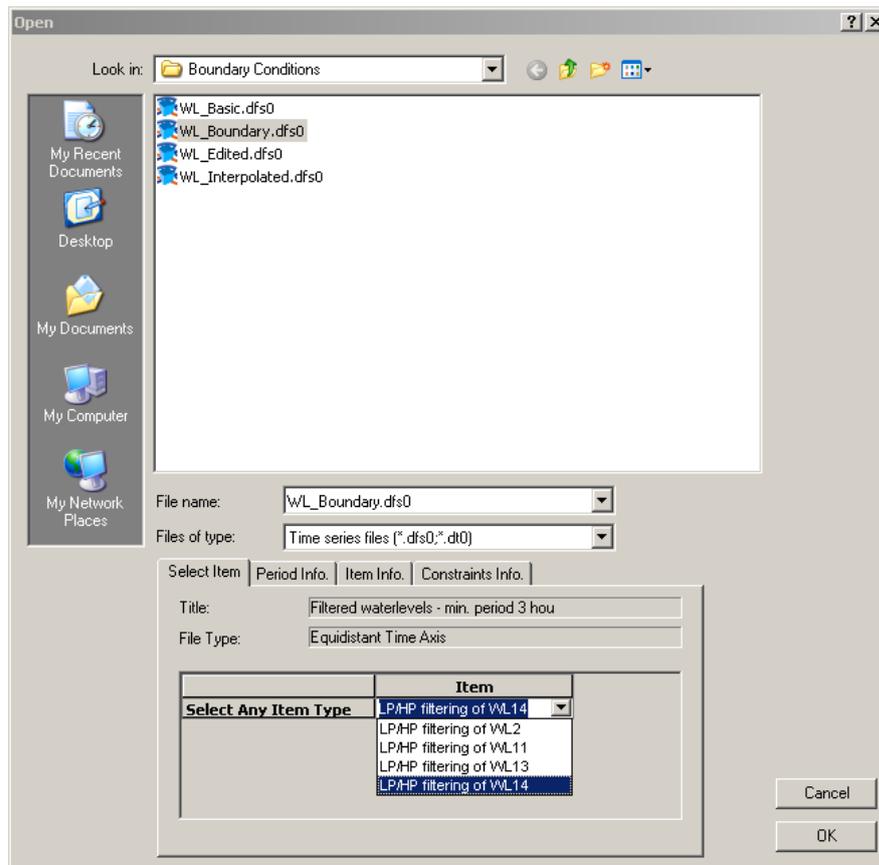


Figure 5.14 MIKE 21 Flow Model FM, Hydrodynamic Module: Boundary Selecting file and item for boundary

Select item 1 in the time series file for the North boundary as this item contains the filtered water levels from measuring station SBF02 (WL2). For the boundaries West1 and West2 select item 4 (WL14), for the South boundary select item 2 (WL11) and for the East boundary select item 4 (WL13).

For the soft start you should use linear variation with the same time interval of 21600 s as in the 'Wind Forcing' dialogue (Figure 5.11) and a reference value of 0.4 m corresponding to the initial value. See Figure 5.15. Also all the boundaries should include Coriolis and wind correction.

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

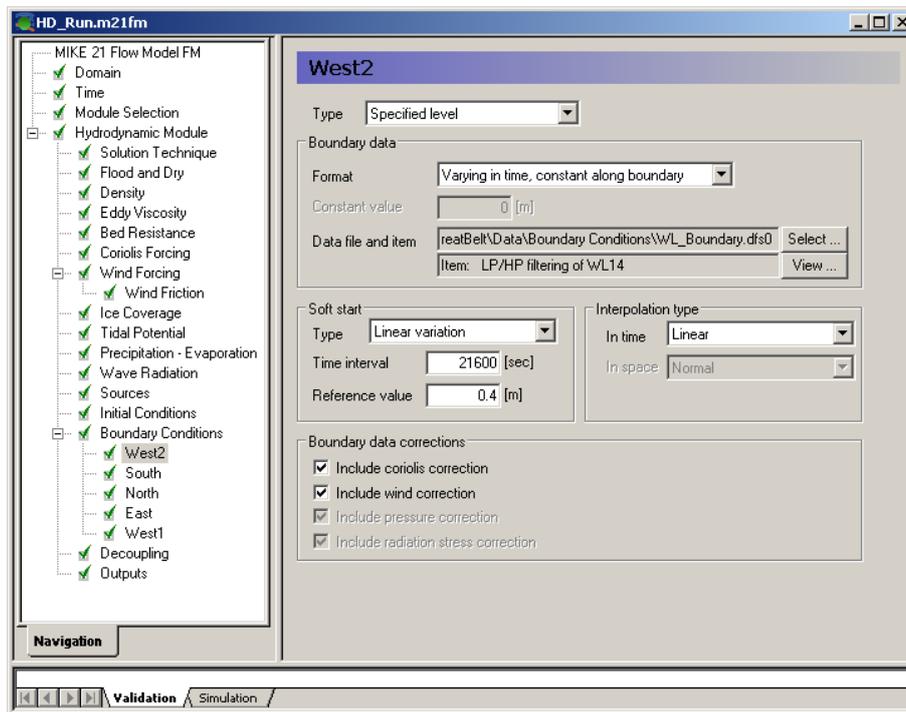


Figure 5.15 MIKE 21 Flow Model FM, Hydrodynamic Module: Boundary Conditions for the West Boundary are specified as constant along the boundary: WL_Boundary.dfs0, item WL14

The first simulation of MIKE 21 Flow Model FM is for calibrating the hydrodynamic model alone. Thus, to save computation time in the simulation, the generation of decoupled files for future particle tracking simulations is not considered here. Once the flow model has been calibrated, files for subsequent decoupled model simulations can be generated as described in Section 5.4.3.

Specify one output as an area series (2D) and specify the resulting output file name as HD_out.dfsu, see Figure 5.16. 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 600 s then the specified output frequency is $3600/600 = 6$. As default, the full area is selected.

Also specify an output file as point series at the positions of the monitoring stations SBF07 and SBF09, see Figure 5.17. Save every time step and use discrete values as interpolation type. 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.

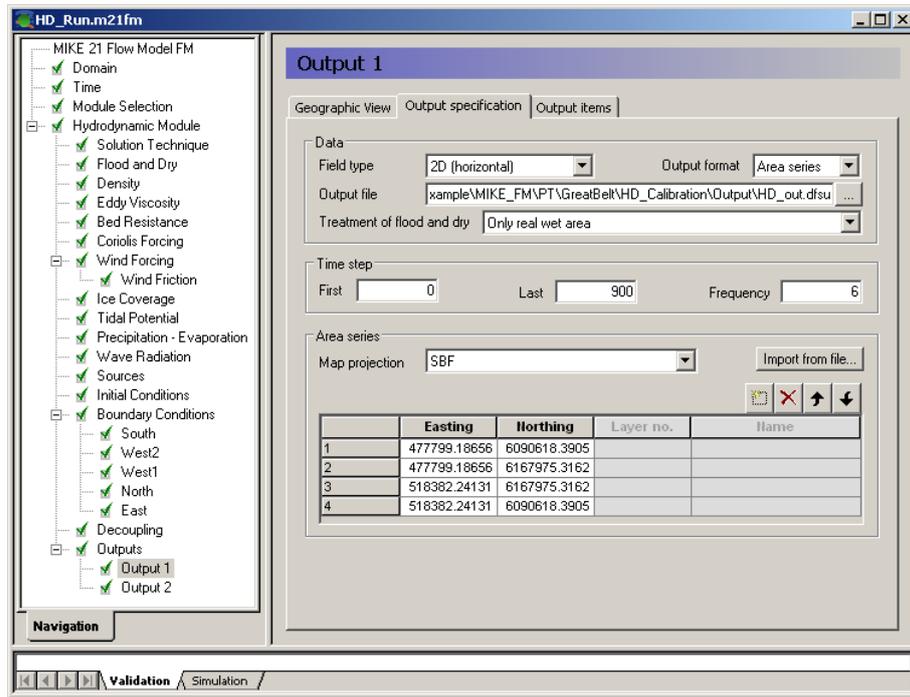


Figure 5.16 MIKE 21 Flow Model FM, Hydrodynamic Model: Output of area series

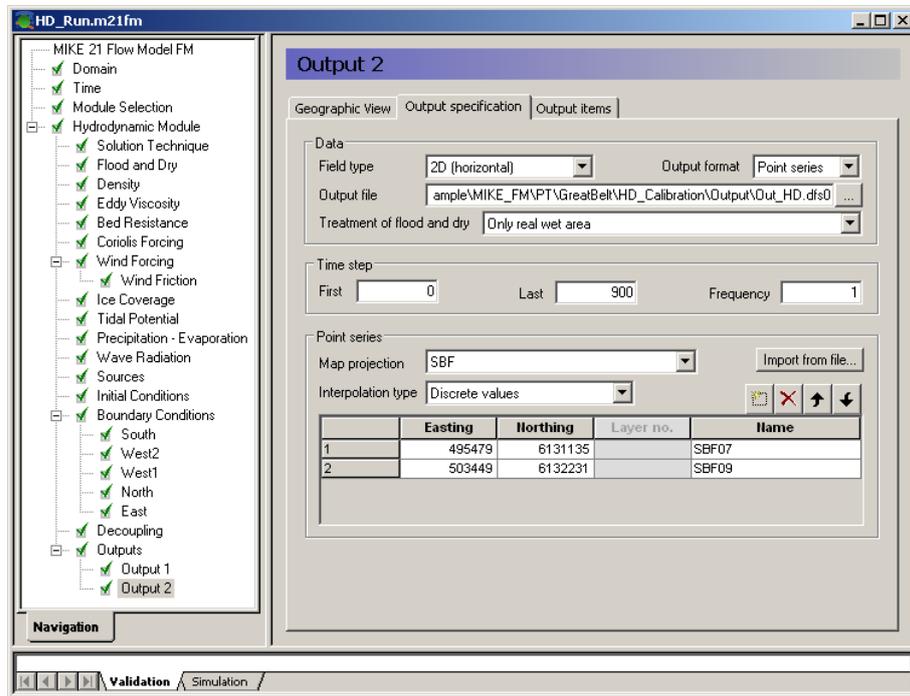


Figure 5.17 MIKE 21 Flow Model FM, Hydrodynamic Model: Output of point series

Pick the parameters to include in the output file as shown in Figure 5.18.

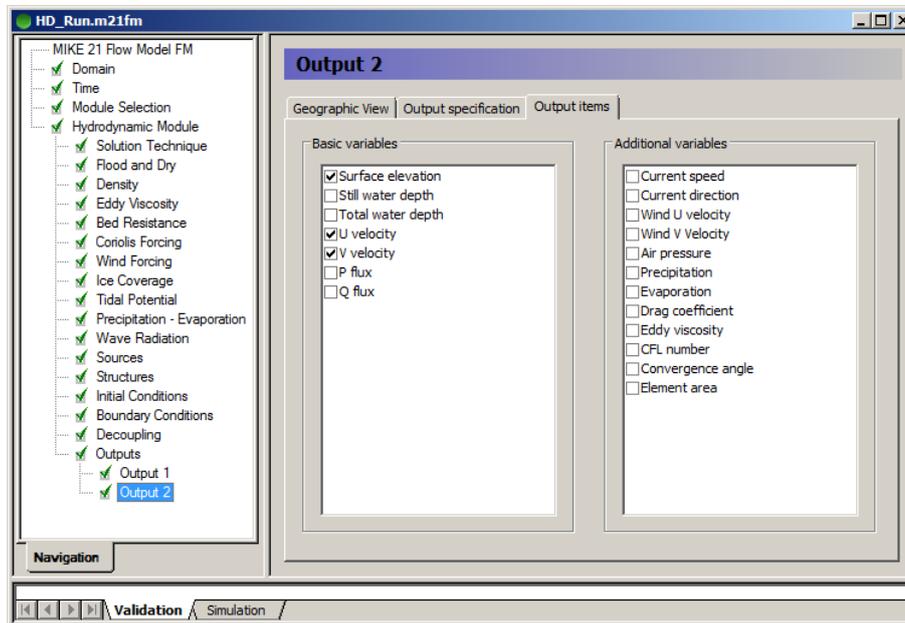


Figure 5.18 MIKE 21 Flow Model FM: Output Parameters

Now you are ready to run the MIKE 21 Flow Model FM, simulating the hydrodynamic conditions for the Great Belt Area.

The specification file for this example has already been made, and it can be found in the Examples folder:

.\MIKE_21\FlowModel_FM\PT\GreatBelt\HD_Calibration\HD_Run.m21fm

5.4 HD Model Calibration

5.4.1 Measured values

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

The hydrodynamic model is calibrated by comparing current measurements from the two monitoring stations SBF07 and SBF09, positioned south-west and south-east of Sprogø, respectively (see Figure 4.1). Vertical current profiles were obtained by Acoustic Doppler Current Profilers (ADCPs) mounted on the sea bed.

The ADCP recordings were saved in text files with columns. The first column showed date and time in UTC (Number of seconds from 1 January 1970 at 00:00:00 with a 0.5-hour time step). The next columns showed measurements from a number of levels in the water column. Each measurement (bin) was represented by four columns (east, north and vertical current component + error value).

The levels in the water column are described by

SBF07: Water depth = 21.4 m DNN
 $bin(i) = -19.5 + (i-1) \cdot 1.0, \quad i=1,17$

SBF09: Water depth = 18.7 m DNN
 $bin(i) = -16.8 + (i-1) \cdot 1.0, \quad i=1,15$

The measured current data have been imported into dfs1 files. Figure 5.19 and Figure 5.20 show the current profile (east and north component) measured on 27 November at stations SBF07 and SBF09, respectively. Please note that the upper part of the profile, which may be largely influenced by wind, is not shown in the plots.

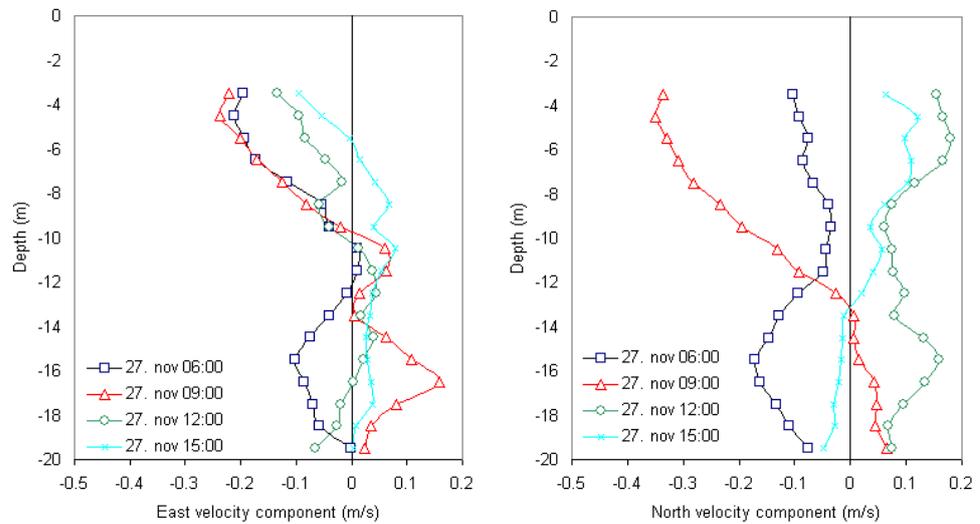


Figure 5.19 Current profiles (east and north component) measured at SBF07

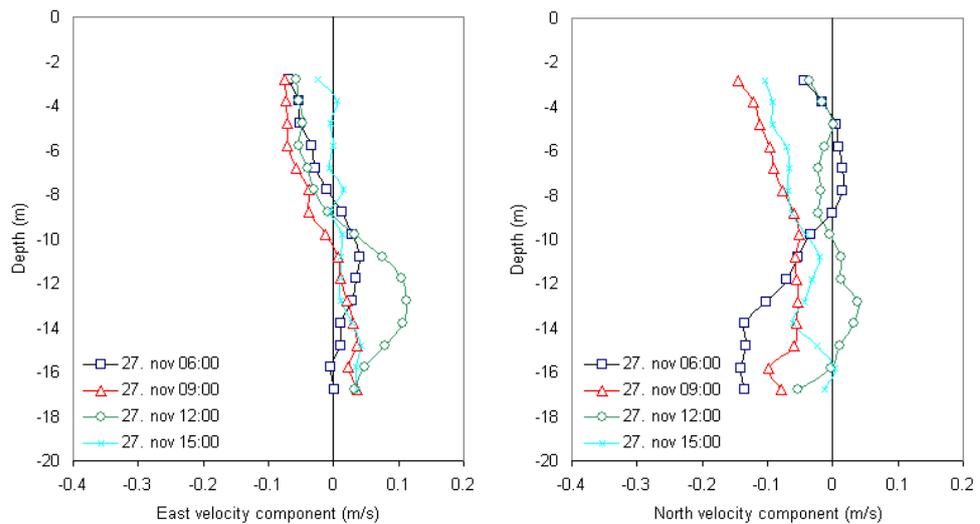


Figure 5.20 Current profiles (east and north component) measured at SBF09

The depth averaged values have been calculated and saved into dfs0 files. Figure 5.21 and Figure 5.22 show the East and North components of the depth averaged current in station SBF07 and SBF09, respectively. During the simulation period, the current was predominantly towards north.

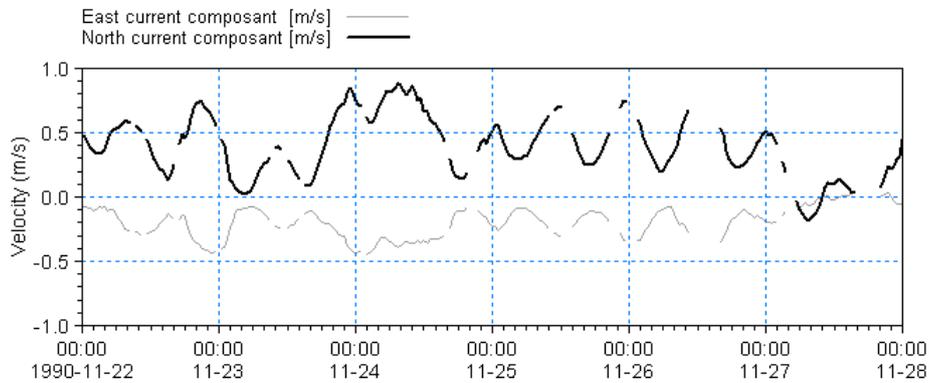


Figure 5.21 Measured depth-averaged current components for SBF07

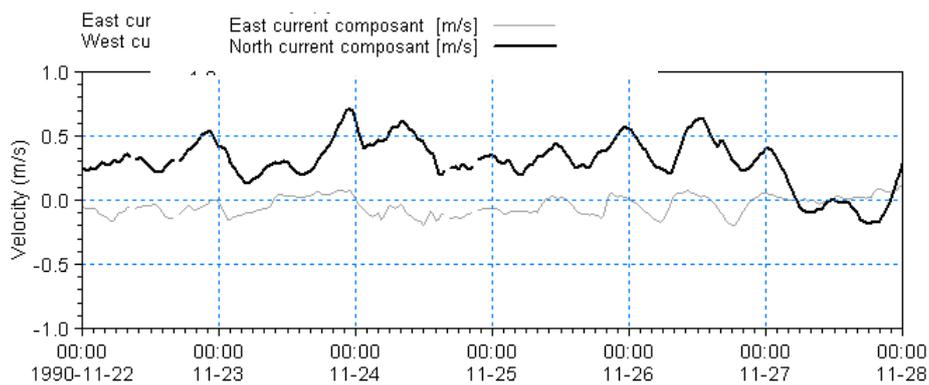


Figure 5.22 Measured depth-averaged current components for SBF09

The data files with the measured current data are included in the folder:

.MIKE_21\FlowModel_FM\PT\GreatBelt\Data\Measurements

During the float track period on 27 November, the current was weak. From 8-10 am the current was weak with a mean north component of 0.04 m/s to the south. From 10 to 12 AM the current decreased further and turned from south to a weak north-going current at SBF 07, which resulted in the very curved float tracks shown in Figure 4.18.

5.4.2 Compare model result and measured values

An example of the simulated flow field for the area around Sprogø is shown in Figure 5.23.

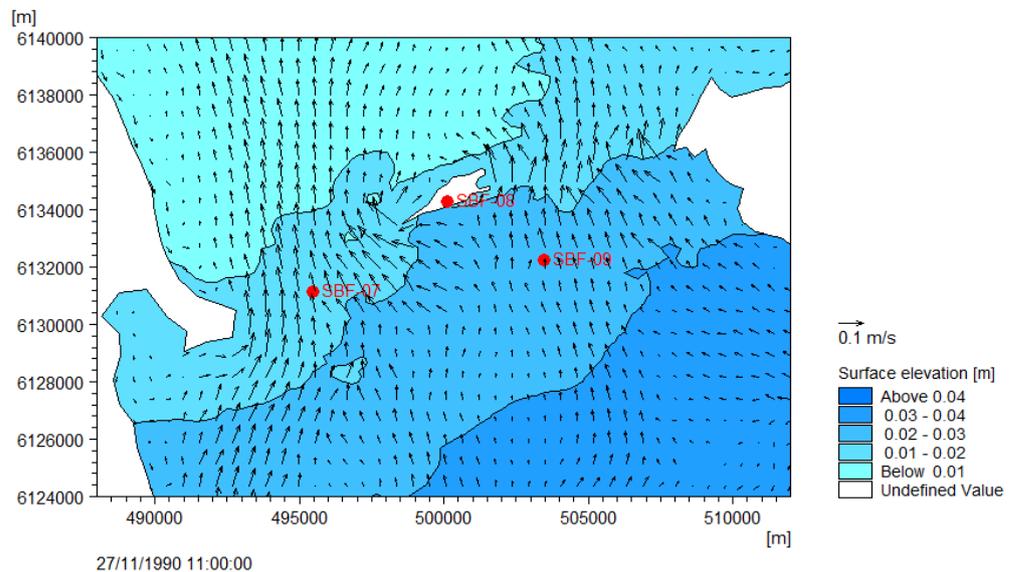


Figure 5.23 Simulated flow field in the area around Sprogø during a period with north-going currents

Use the Plot Composer to plot the simulated current velocities at the positions for SBF07 and SBF09.

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

.MIKE_21\FlowModel_FM\PT\GreatBelt\HD_Calibration\Plots

Open the files CurrentSpeed_CompareSBF07.plc and CurrentSpeed_CompareSBF09.plc to see a comparison of measurements and model output of currents at the two measuring stations, respectively.

If your simulation is successful, plots as shown in Figure 5.24 and Figure 5.25 will appear. The float track period is indicated by a grey area.

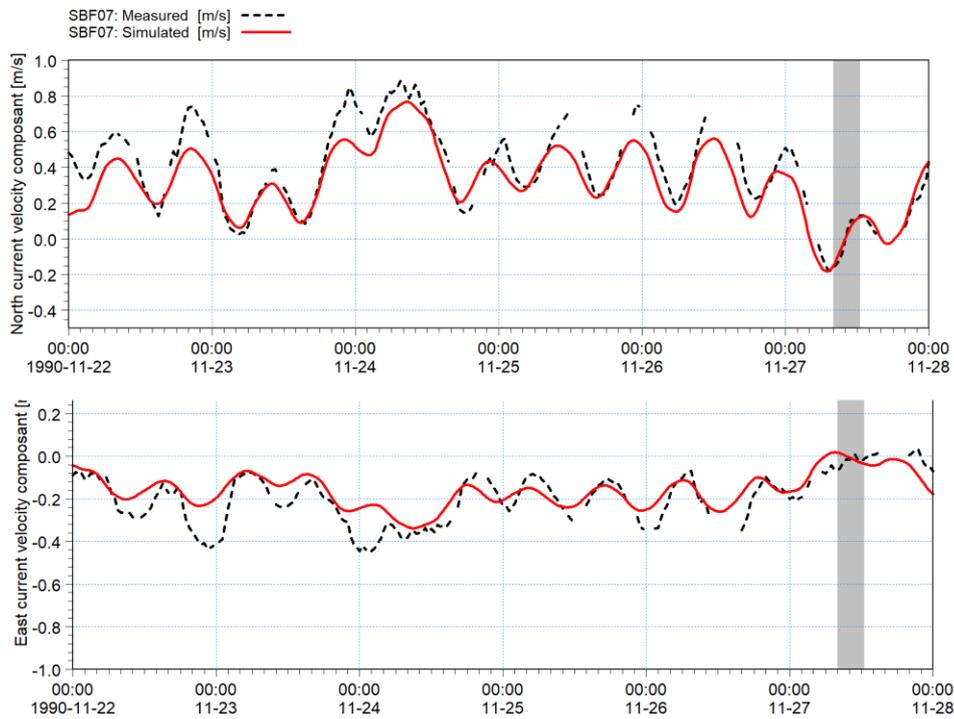


Figure 5.24 Comparison of measurements and model output of current at SBF07.
Dotted black line: depth-averaged measured values, Solid red line: simulation
Upper: North component, Lower: East component

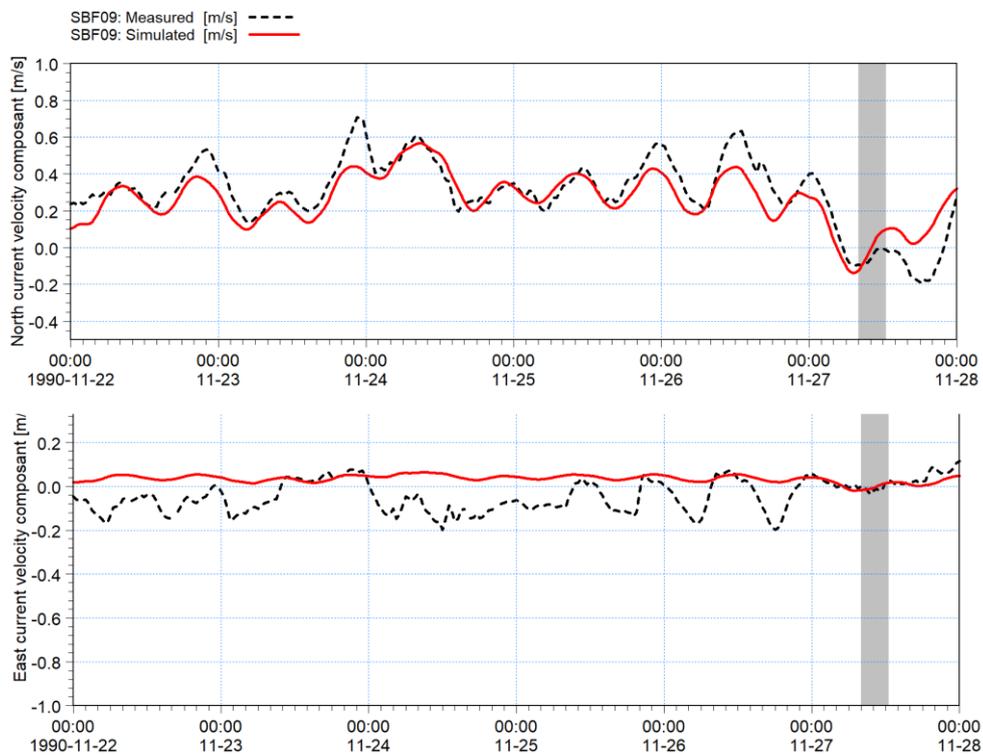


Figure 5.25 Comparison of measurements and model output at SBF09.
Dotted black line: depth-averaged measured values, Solid red line: simulation
Upper: North component, Lower: East component

The comparisons between measured and calculated current velocities indicate that the simulated north component of the current is close to the measured whereas the east component shows too little variation compared to measurements. The results from the float track period on 27 November are close to the measured values. However it should be noticed that this is a period with low current speeds in general.

The density in Kattegat is larger than the density of the Baltic Sea. This means that if the water level at the north and south boundary is the same, the water will actually flow to the south, but the model will simulate no flow. The difference between the densities on the south and north boundary depends on the hydrodynamics of the previous period. Thus, the model simulation in general underestimates south-going currents and overestimates north-going currents.

Unfortunately, actual measurements of the densities on the boundaries did not take place during the float track period and have therefore not been included in the simulation. However, salinity measurements were made at stations SBF07 and SBF09 at various depths ranging from 7.0 m to 19.2 m. The measured salinity values for the upper and lower depth levels are shown in Figure 5.26.

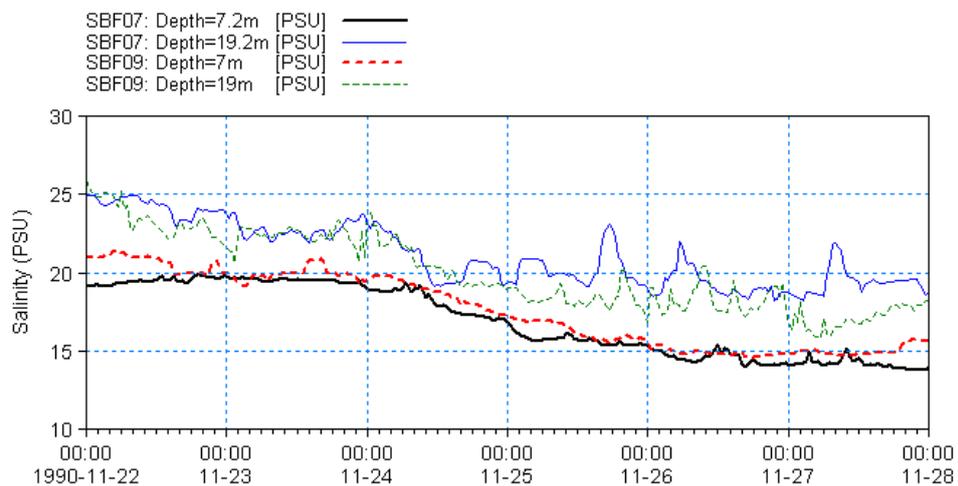


Figure 5.26 Measured salinity at stations SBF07 and SBF09.
 Full thick black line: salinity at upper level (7.2 m), SBF07
 Full thin blue line: salinity at lower level (19.2 m), SBF07
 Dotted thick red line: salinity at upper level (7.0 m), SBF09
 Dotted thin green line: salinity at upper level (19.0 m), SBF09

Figure 5.26 shows that some stratification exists in the area. In general the salinity values in the upper layer are less than the salinity layers in the lower layer, but with no major differences between stations SBF07 and SBF09. However, during the float track period on 27 November the salinity at the lower level at station SBF09 was significantly lower than at the lower level at station SBF07. This indicates that additional currents due to density gradients between the two stations were influencing the track of the floats.

Another influence that the presence of stratification induces is the Coriolis force. In the lower layer the current is changing from north to south as a result of the tidal variations. When the current is changing direction to south-going, the water in the upper layer will flow towards west as a result of the Coriolis force, forcing the water in the lower layer to flow towards east. When the north-south current decelerates, the flow towards west in the upper layer will decrease, forcing the lower layer to flow to the west. This phenomenon is shown in Figure 5.19.

The above mentioned factors indicate that the current pattern in the present domain cannot be described in sufficient detail by a two-dimensional flow model. However, despite the many reservations that can be said about the validity of the results and level of calibration of the hydrodynamic model, it is assumed that the accuracy of the hydrodynamic simulation is adequate for the hydrodynamic conditions to be used in the Particle Tracking simulations. Therefore, for the hydrodynamic results we will not calibrate further.

5.4.3 Generate decoupled model data

The first simulation showed that the hydrodynamic results need no further calibration. Yet, it can save time in calibration runs of the particle tracking module, 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 (if you have not done so already).

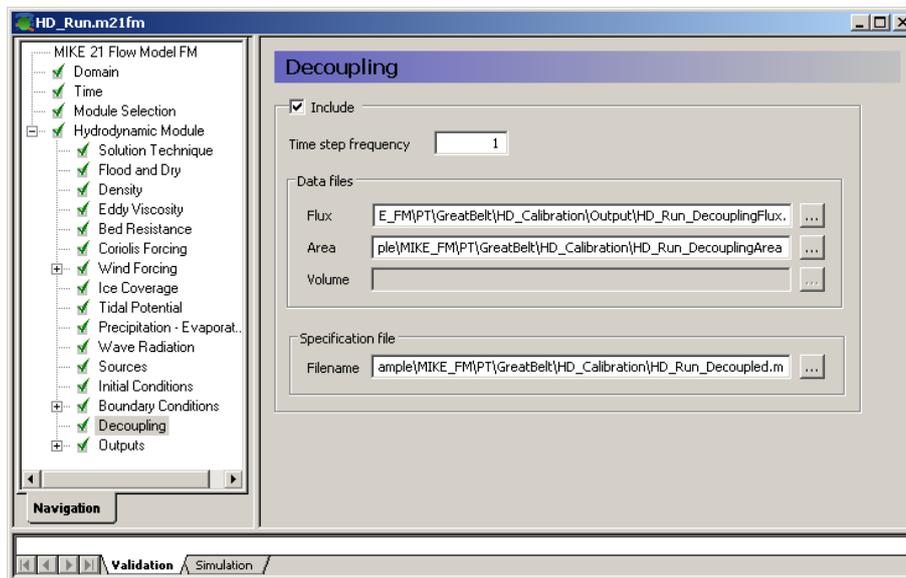


Figure 5.27 MIKE 21 Flow Model FM, Hydrodynamic Module: Decoupling specifications

The decoupled data for use in future simulations are defined in the 'Decoupling' dialogue in the Hydrodynamic Module, see Figure 5.27.

The decoupled data are generated and saved every 10 minutes, i.e. with a time step frequency of 1. The generated data files and specification file are defined by default names.

5.5 The Particle Tracking Module

To use the decoupled file setup, generated by the HD simulation, open the generated file 'HD_Run_Decoupled.m21fm', and save it under a new name.

The basic domain and time parameters in the input file are already determined by the decoupled input data files. Include the 'Particle Tracking Module' in the 'Module Selection' dialogue, see Figure 5.28.

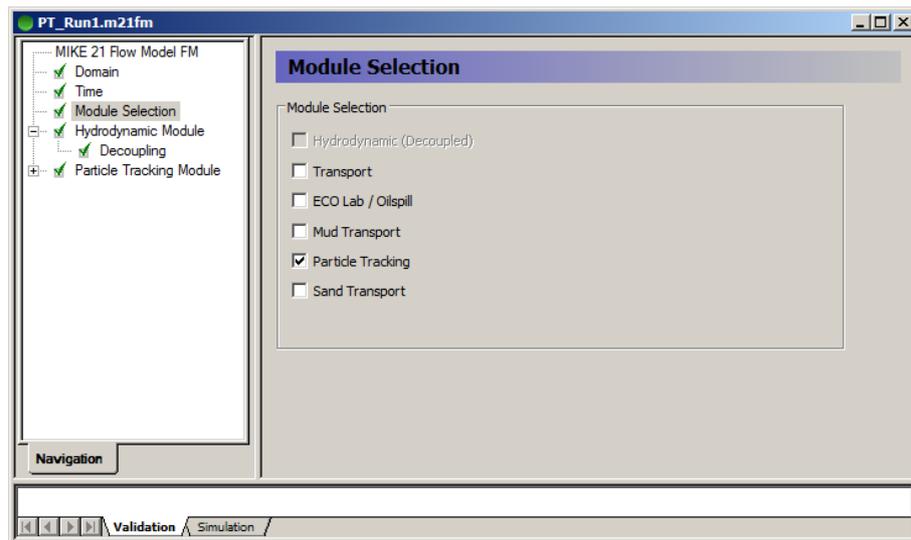


Figure 5.28 MIKE 21 Flow Model FM: Module Selection

The hydrodynamics are fixed and specified by the generated decoupling files, see Figure 5.29.

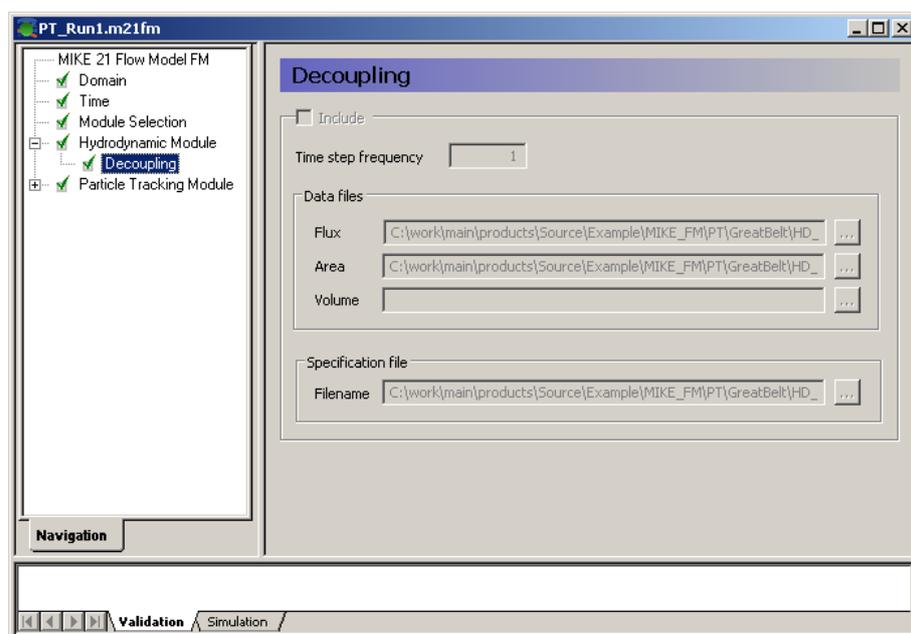


Figure 5.29 MIKE 21 Flow Model FM, Hydrodynamic Module: Decoupled Files

The next step is to select the parameters specific for the Particle Tracking Model.

In the 'Classes' dialogue you must describe the different particle classes that are to be included in the calculation. A class is a group of particles with the same properties regarding settling, decay, dispersion and erosion. An example of a class could be silt.

Specify a particle class for each of the float tracks, and describe each class separately, see Figure 5.30 and Figure 5.31.

Set the minimum particle mass to a small value above 0. This will exclude particles with insignificant masses, i.e. in this example all particles with magnitude 0, shown in Figure 4.19. The computational simulation time is proportional to the number of particles in the simulation, so therefore it is often a good idea to exclude the particles that have insignificant masses.

Each class can be given a maximum particle age, corresponding to the duration of the measured float track as given in Table 4.2. This enables a direct comparison between measured and simulated float tracks.

Also, when the particle exceeds the maximum age the particle is removed from the simulation. If the simulation contains many particles this can save a lot of simulation time. In this context, please note that particles released in the beginning of the simulation are not be considered.

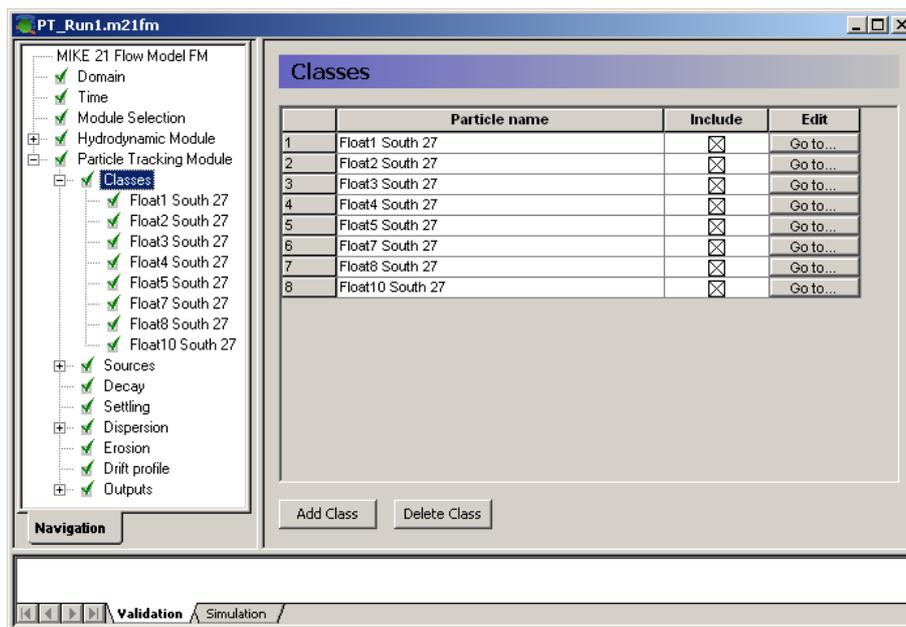


Figure 5.30 MIKE 21 Flow Model FM, Particle Tracking Module: Classes in setup

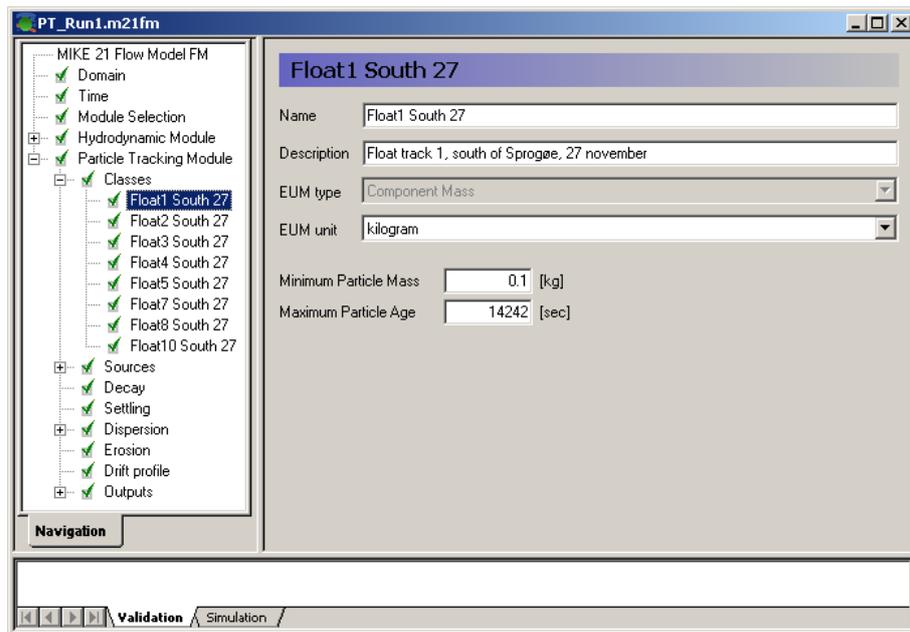


Figure 5.31 MIKE 21 Flow Model FM, Particle Tracking Module: Class specification

In the ‘Sources’ dialogue you specify how the particles are entered into the simulation. In this example each float track is defined by a separate source, see Figure 5.32.

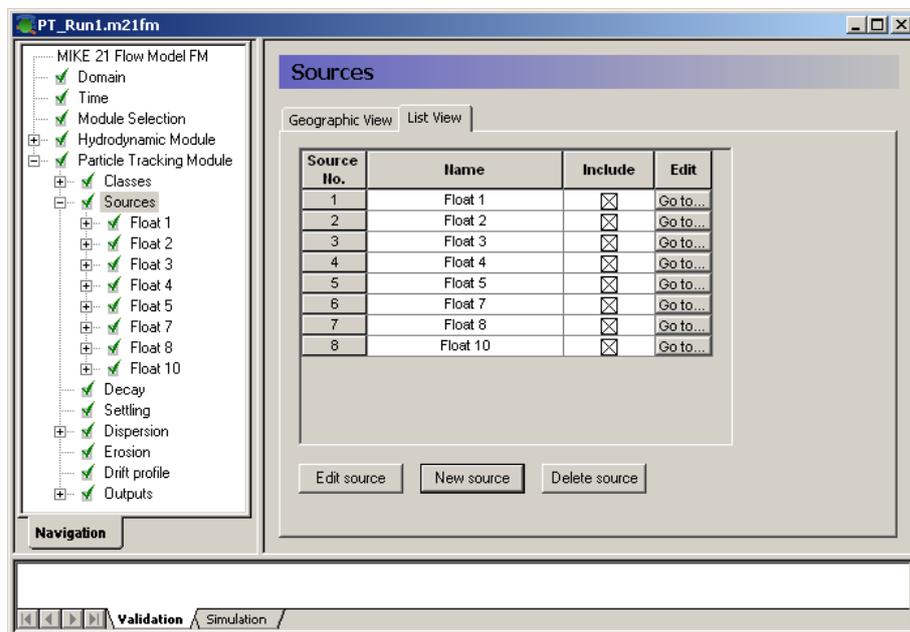


Figure 5.32 MIKE 21 Flow Model FM, Particle Tracking Module: Sources in setup

For each source you must specify the location and which particles classes that are released by the source. In this example the location of the source is fixed, corresponding to the initial position of the float track and only one particle class is included in each source, see Figure 5.34.

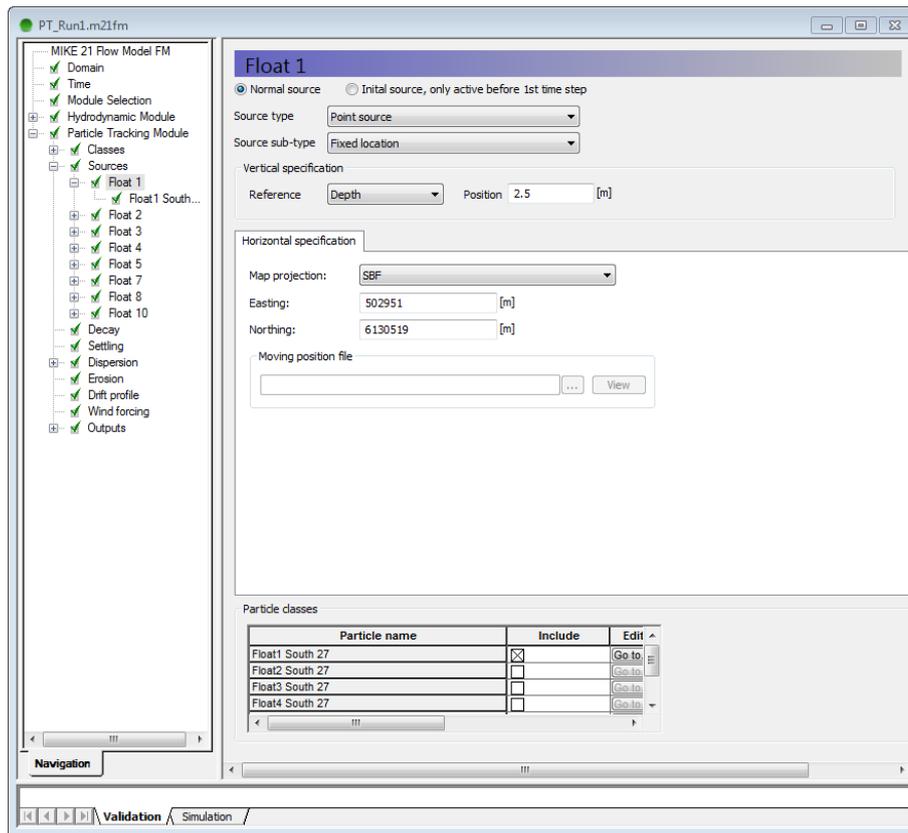


Figure 5.33 MIKE 21 Flow Model FM, Particle Tracking Module: Source specification

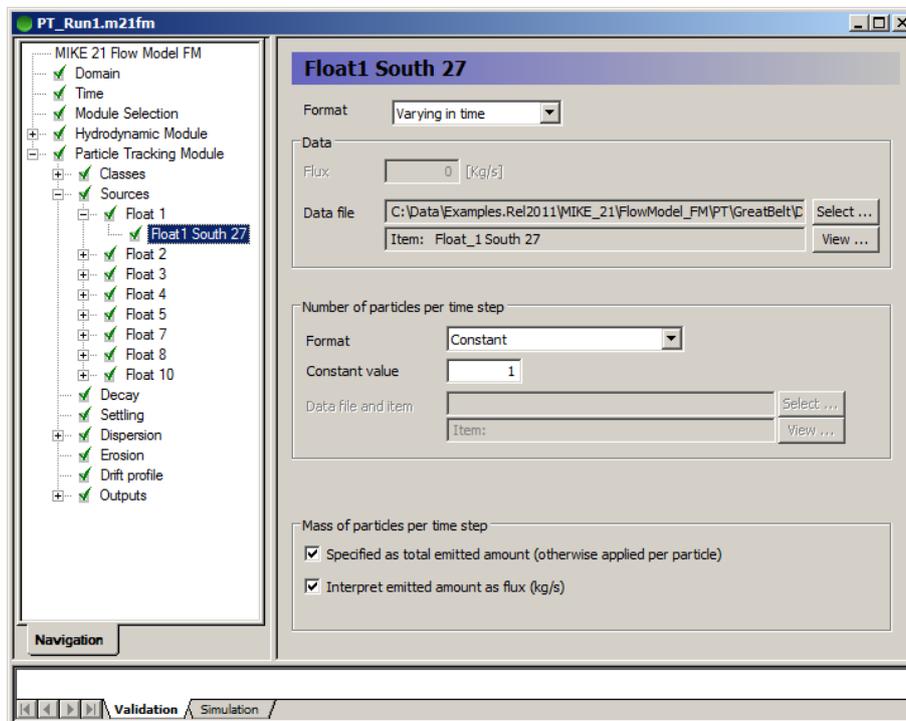


Figure 5.34 MIKE 21 Flow Model FM, Particle Tracking Module: Source specification for individual class

For each particle class in the source you must specify how the particles are released. In this example the release of the particles is defined by the time-varying source, as described in Section 4.3 and the dialogue will look as in Figure 5.34.

The particles simulate floats. The floats can be described as conservative, with no decay, erosion or settling. Therefore these effects are not included in the simulation.

In order to obtain a result, which can be reproduced, the present setup does not include dispersion. By including dispersion the simulated float position would by random be a bit dislocated.

The drift profile used in the simulation is based on the results from the hydrodynamic simulation. Optionally it is possible to consider other factors that may influence the drift profile and hence the influence on the particles, such as wind and bed shear stresses.

For the first simulation of the particle tracking, the default type is applied, see Figure 5.35.

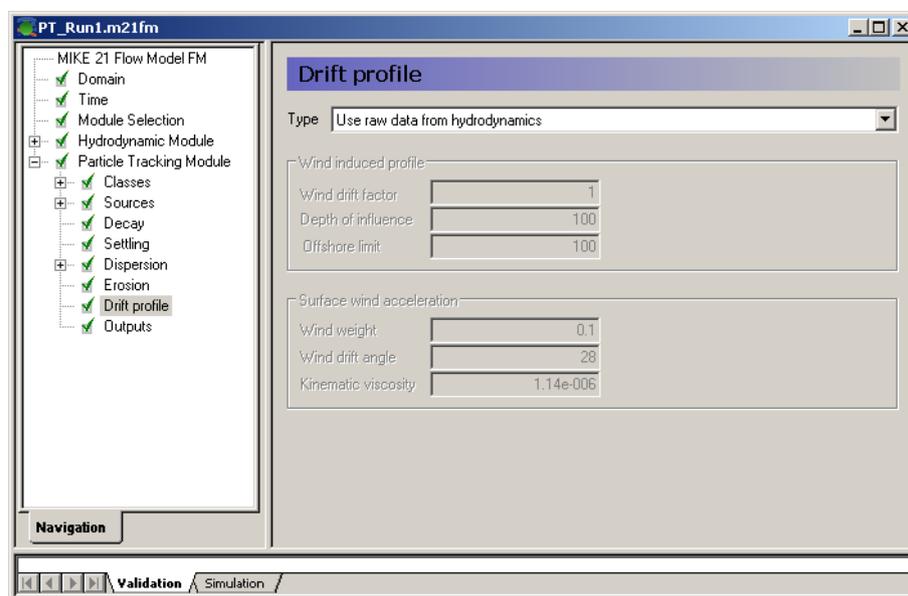


Figure 5.35 MIKE 21 Flow Model FM, Particle Tracking Module: Drift Profile specification

Finally the output must be specified. You should save the data in a particle track file. Save all particle locations by entering 1 for the frequency, see Figure 5.36.

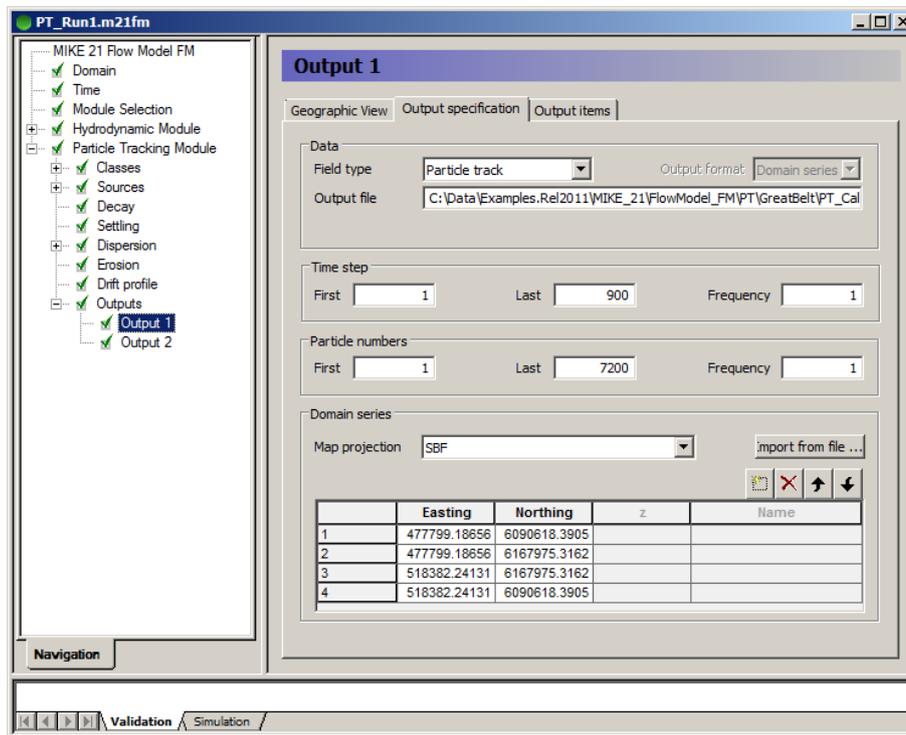


Figure 5.36 MIKE 21 Flow Model FM, Particle Tracking Module: Output specification

Furthermore, save the 2D information of the particle concentrations and hydrodynamics during the floats track period in another file.

Now we are ready to run the MIKE 21 Flow Model FM, specified by a decoupled setup modelling particle tracks.

The specification file for this example has already been made, and it can be found in the Examples folder:

.\MIKE_21\FlowModel_FM\PT\GreatBelt\PT_Calibration\PT_Run1.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'.

5.6 PT Model Calibration 1

5.6.1 Measured values

The float tracks south of Sprogø were performed in an area with water depths varying between 15 and 20 meters. From 8:00 to 10:00 hours the current was weak, running towards south with a mean velocity of 0.04 m/s. From 10:00 to 12:00 hours the current decreased further and turned from south to a weak north-going current, resulting in the very curved float tracks shown in Figure 4.18. Plots of the simulated flow pattern during the float track period are shown in Figure 5.37.

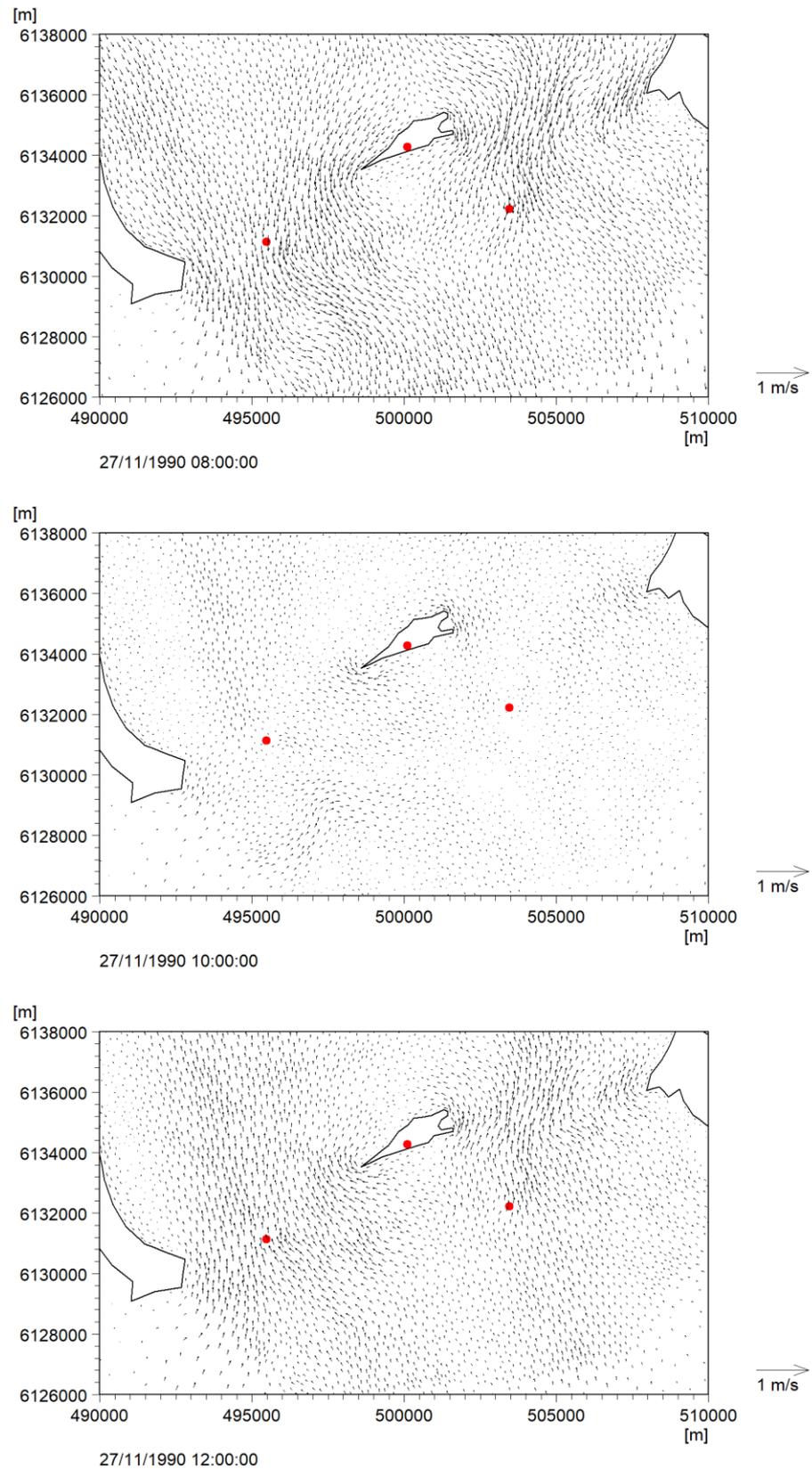


Figure 5.37 Simulated flow pattern at the beginning, middle and end of the float track period. Position of SBF07, SBF08 and SBF09 are indicated.

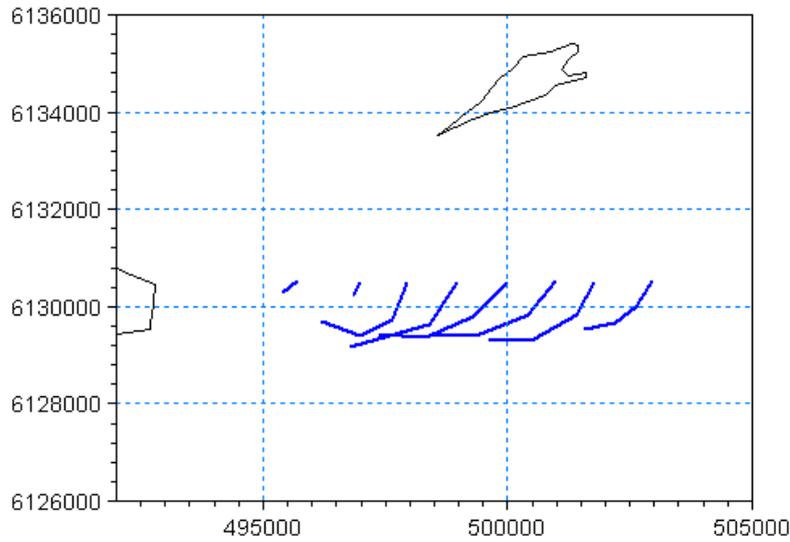


Figure 5.39 Measured float track visualised by a particle track overlay in the Data Viewer. Tracks are measured 27/11 in the period from 07:59 to 12:44

5.6.2 Model results compared with measured values

The measured and simulation float tracks are compared by importing the XML-files for measured and simulated data into the Data Viewer as layers, see Figure 5.40.

A layer file for displaying measured and simulated float tracks has already been generated. To activate it you just have to open the generated file 'PT_Run1.dfsu' in the PT output directory.

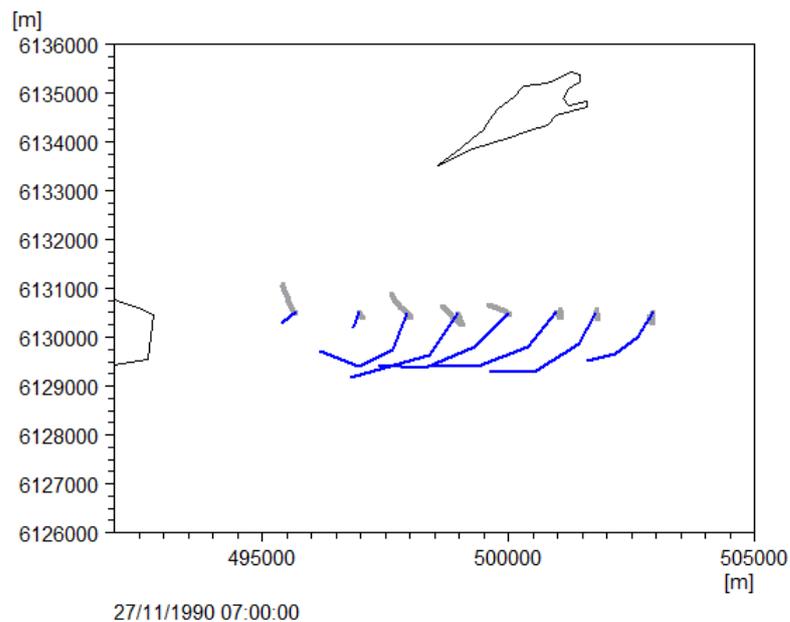


Figure 5.40 Particle track overlays. Blue lines: Measured, Grey lines: Simulated run 1

The first particle tracking simulation does not describe the float tracks well.

During time periods with small water level difference between Kattegat and the Baltic Sea, or in shallow areas placed in the shelter of the main current in the Belt, the current is dominated by interactions between wind and friction forces. The current can vary significantly in time and space according to local wind phenomena, such as sudden changes in speed and direction. This is a difficult current field to describe within a 2D model and tracks of floats placed in areas or periods like this are difficult to reproduce.

The tendency of the particles first going towards south-west and then changing direction towards west are indicated in the simulation by some of the floats, but the particles do not travel as far as expected by the measurements.

This indicates that there are other factors than the raw hydrodynamics from the two-dimensional HD simulation that need to be considered and/or that the HD simulation needs further calibration.

5.7 PT Model Calibration 2

5.7.1 Changed model specifications

The floats are positioned near the water surface. This implies that the wind has an influence on the drift profile. That this is the case is actually observed by the ADCP measurements of the current profile, see Figure 5.19 and Figure 5.20.

During the float track period the wind was towards south, with a wind speed of about 5 m/s, see Figure 5.41.

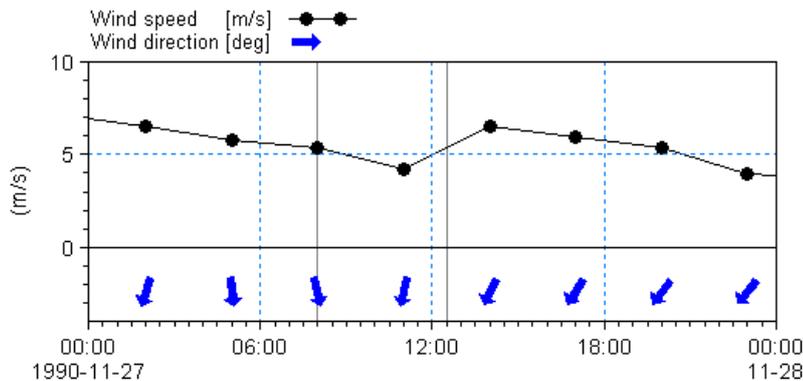


Figure 5.41 Wind conditions during float track period indicated by lines

The second simulation with particle tracking is therefore carried out using a drift profile considering the bed shear profile and wind induced profile, see Figure 5.42.

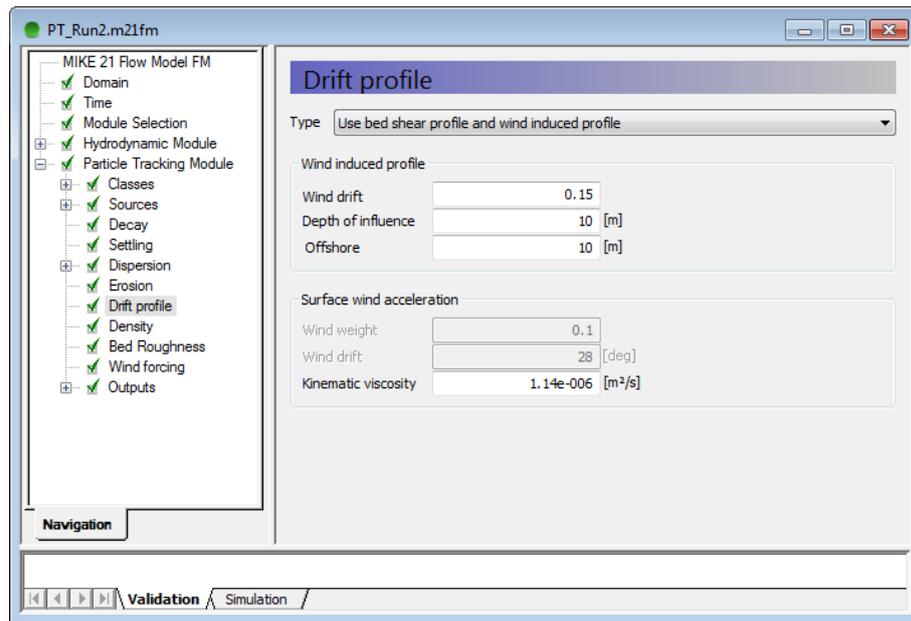


Figure 5.42 MIKE 21 Flow Model FM, Particle Tracking Module: Selecting new drift profile

The wind induced profile will have a large effect, as it will induce additional currents in the surface layer. The depth of influence and offshore limit are both set to 10 m as this limit is indicated by the measured current profiles.

The wind drift factor describes the magnitude of the additional influence the wind induced profile will have on the flow. This factor has a common value that varies from 3 to 4 percent of the wind speed 10 m above the water surface. Thus a wind drift factor of 0.15 is applied in the simulation.

5.7.2 Model results compared with measured values

A layer file for displaying measured and simulated float tracks for calibration run 2 has already been generated. To activate it you just have to open the generated file 'PT_Run2.dfsu' in the PT output directory.

The measured and simulation float tracks are compared in Figure 5.43.

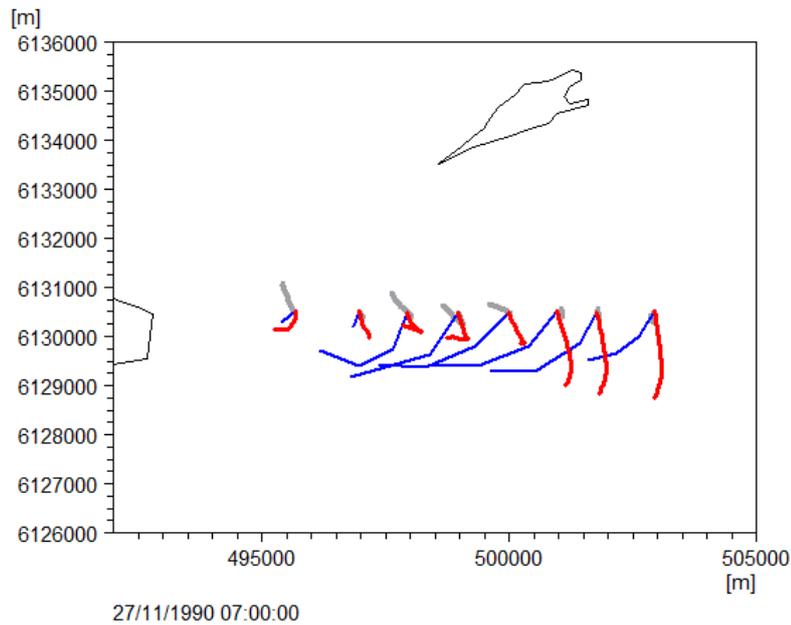


Figure 5.43 Particle track overlays. Blue lines: Measured, Grey lines: Simulated run 1, Red lines: Simulated run 2

The length of the simulated float tracks are now of the correct magnitude, but the float tracks all have a too weak turn towards the west.

One reason for this could be the fact that the floats were placed in an area in the lee of the island and a period which is difficult to describe. However, it is probably due to the fact that the hydrodynamic model is insufficiently calibrated and shows too poor correlation between measured and calculated current speeds in the eastern direction.

5.8 PT Model Calibration 3

5.8.1 Changed model specifications

The floats were positioned at 2.5 m water depth. The floats were constructed by two large 2 x 2 m screens held together as a cross. Due to this large size, the depth of the source could as well have been defined as 3 m.

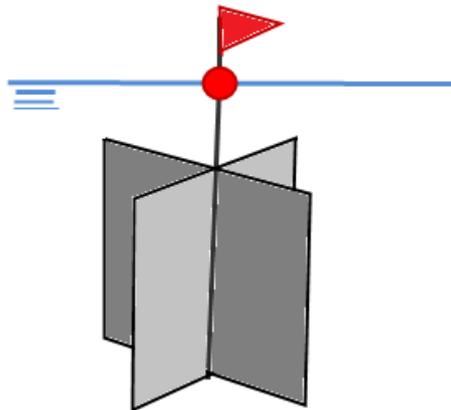


Figure 5.44 Sketch of floating device

To investigate the influence of the source depth, the third simulation with particle tracking is therefore carried out using a location of the source at 3 m below the water surface, see Figure 5.45.

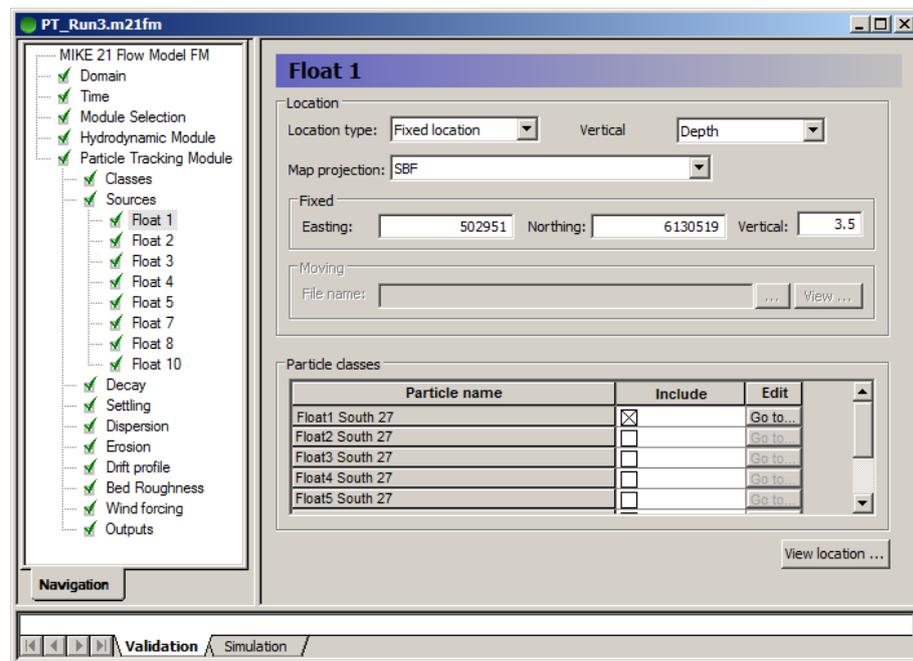


Figure 5.45 MIKE 21 Flow Model FM, Particle Tracking Module: Selecting new source depth

To compare the simulation results with the results from calibration run 2, the drift profile settings are set to the same as in calibration 2.

5.8.2 Model results compared with measured values

A layer file for displaying measured and simulated float tracks for calibration run 3 has already been generated. To activate it you just have to open the generated file 'PT_Run3.dfsu' in the PT output directory.

The measured and simulation float tracks are compared in Figure 5.46.

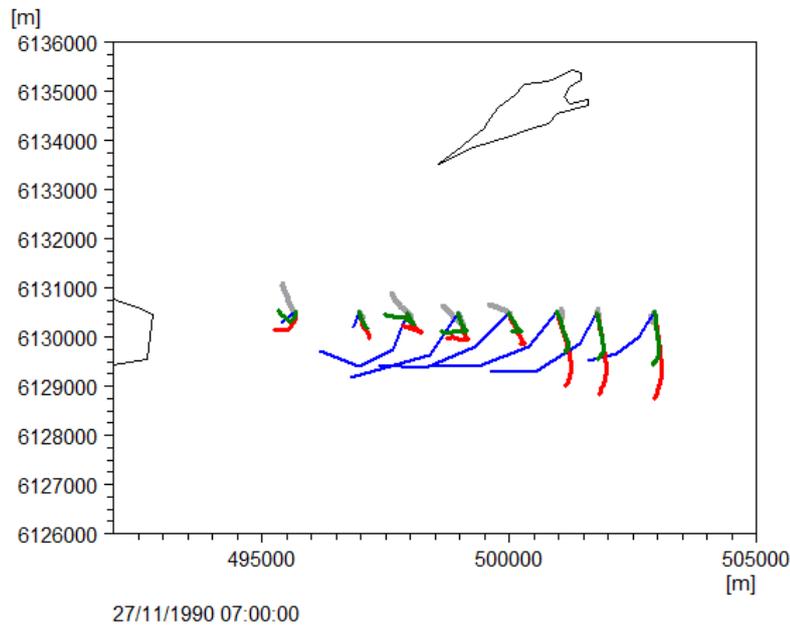


Figure 5.46 Particle track overlays. Blue lines: Measured, Red lines: Simulated run 2, Green lines: Simulated run 3

Figure 5.46 shows that by lowering the source location 0.5 m, the float tracks are turned more towards the west. This is due to the decreased influence of the wind drift profile.

However, it seems like the hydrodynamic conditions are not sufficiently accurate to form the basis for the particle tracking simulations.

More accurate information of the flow field can be obtained by using a hydrodynamic model in three dimensions, where phenomena such as variation in salinity and temperature in the domain are considered.

6 References

- /1/ COWI/VKI, Storebælt, Eelgrass Growth Experiment, 11 Sep-2 Oct.1990, Biological Monitoring Programme for the Period 1989-1992. August 1991.

