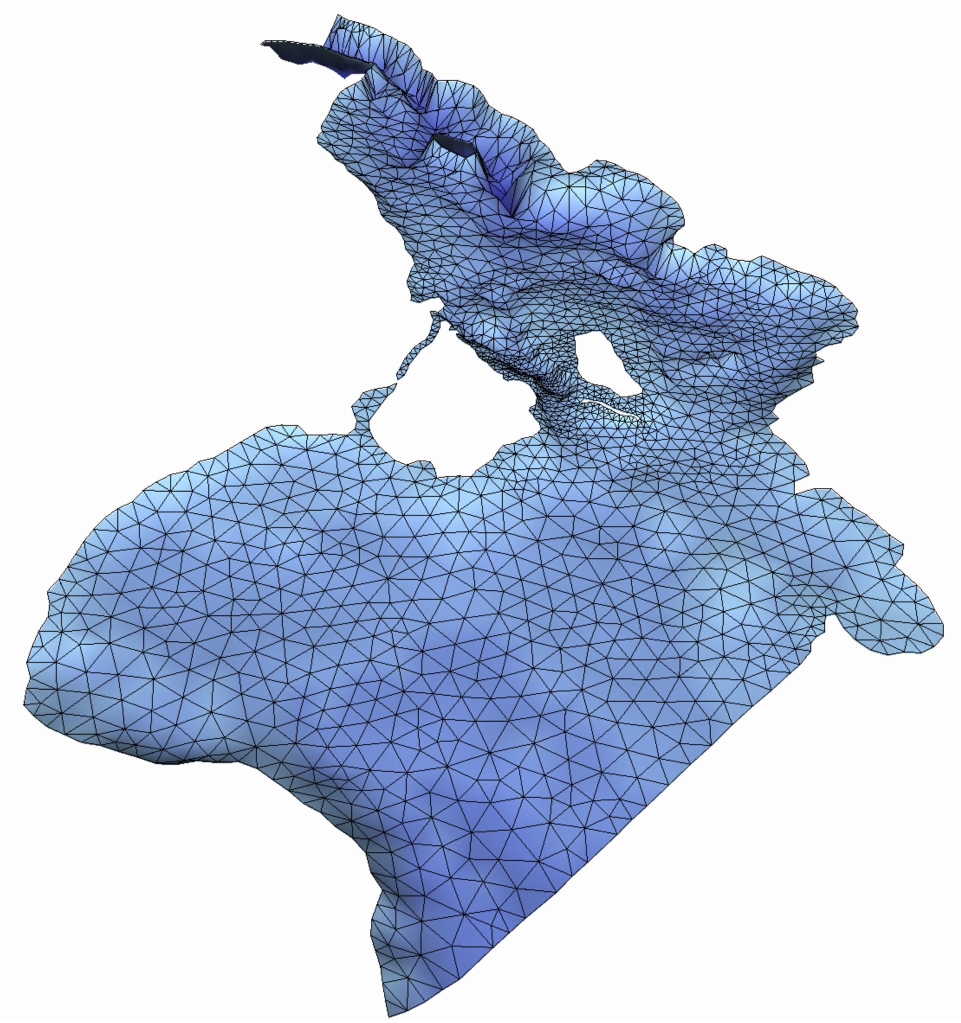


MIKE 21 & MIKE 3 Flow Model FM

Hydrodynamic 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 & MIKE 3 Flow Model FM
Hydrodynamic Module
Step-by-step training guide

1 Introduction

This Step-by-step training guide relates to the fixed link across the Sound (Øresund) between Denmark and Sweden.

Figure 1.1 Øresund, Denmark

1.1 Background

In 1994 the construction of a fixed link between Copenhagen (Denmark) and Malmö (Sweden) as a combined tunnel, bridge and reclamation project commenced. Severe environmental constraints were enforced to ensure that the environment of the Baltic Sea remains unaffected by the link. These constraints implied that the blocking of the uncompensated design of the link should be down to 0.5 %, and similarly maximum spillage and dredging volumes had been enforced. To meet the environmental constraints and to monitor the construction work a major monitoring programme was set-up. The monitoring programme included more than 40 hydrographic stations collecting water level, salinity, temperature and current data. In addition intensive field campaigns were conducted to supplement the fixed stations with ship-based ADCP measurements and CTD profiles. The baseline monitoring programme was launched in 1992 and continued into this century.

By virtue of the natural hydrographic variability in Øresund the blocking of the link can only be assessed by means of a numerical model.

Amongst the comprehensive data sets from the monitoring programme, which form a unique basis for modelling, a three-month period was selected as 'design' period such that it reflected the natural variability of Øresund. The design period was used in the detailed planning and optimisation of the link, and to define the compensation dredging volumes, which were required to reach a so-called Zero Solution.

1.2 Objective

The objective of this Step-by-step training guide is to set-up both a MIKE 21 Flow Model with Flexible Mesh (MIKE 21 Flow Model FM), and a MIKE 3 Flow Model FM for Øresund 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, time consuming but indispensable process. For this example guide 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 and located in the default installation folders:

.\Examples\MIKE_21\FlowModel_FM\HD\Oresund

.\Examples\MIKE_3\FlowModel_FM\HD\Oresund

You should install the examples from the MIKE Zero start page in your own folder.

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 needed to run the example are included and the simulation can start immediately if you want.

2 Creating the Computational Mesh

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.

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

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

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

Please also see the User Guide for the MIKE Zero Mesh Generator, 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

2.1 General Considerations before Creating a Computational Mesh

The bathymetry and mesh file should

1. describe the water depths in the model area
2. allow model results with a desired accuracy
3. give model simulation times acceptable to the user

To obtain this, you should aim at a mesh

1. with triangles without small angles (the perfect mesh has equilateral triangles)
2. with smooth boundaries
3. with high resolutions in areas of special interest
4. based on valid xyz data

Large angles and high resolutions in a mesh are contradicting with the need for short simulation times, so the modeller must compromise his choice of triangulation between these two factors.

The resolution of the mesh, combined with the water depths and chosen time-step governs the Courant numbers in a model set-up. The maximum Courant number shall be less than 0.5. So the simulation times dependency on the triangulation of the mesh, relates not only to the number of nodes in the mesh, but also the resulting Courant numbers. As a result of this, the effect on simulation time of a fine resolution at deep water can be relatively high compared to a high resolution at shallow water.

2.2 Creating the Øresund Computational Mesh

A chart of the area of interest in this example is shown in Figure 2.1. It covers the Sound between Denmark and Sweden. Based on a chart (e.g. from MIKE C-map), xyz data for shorelines and water depths can be generated. In this example xyz data for shorelines and water depths have already been generated. See Figure 2.14.

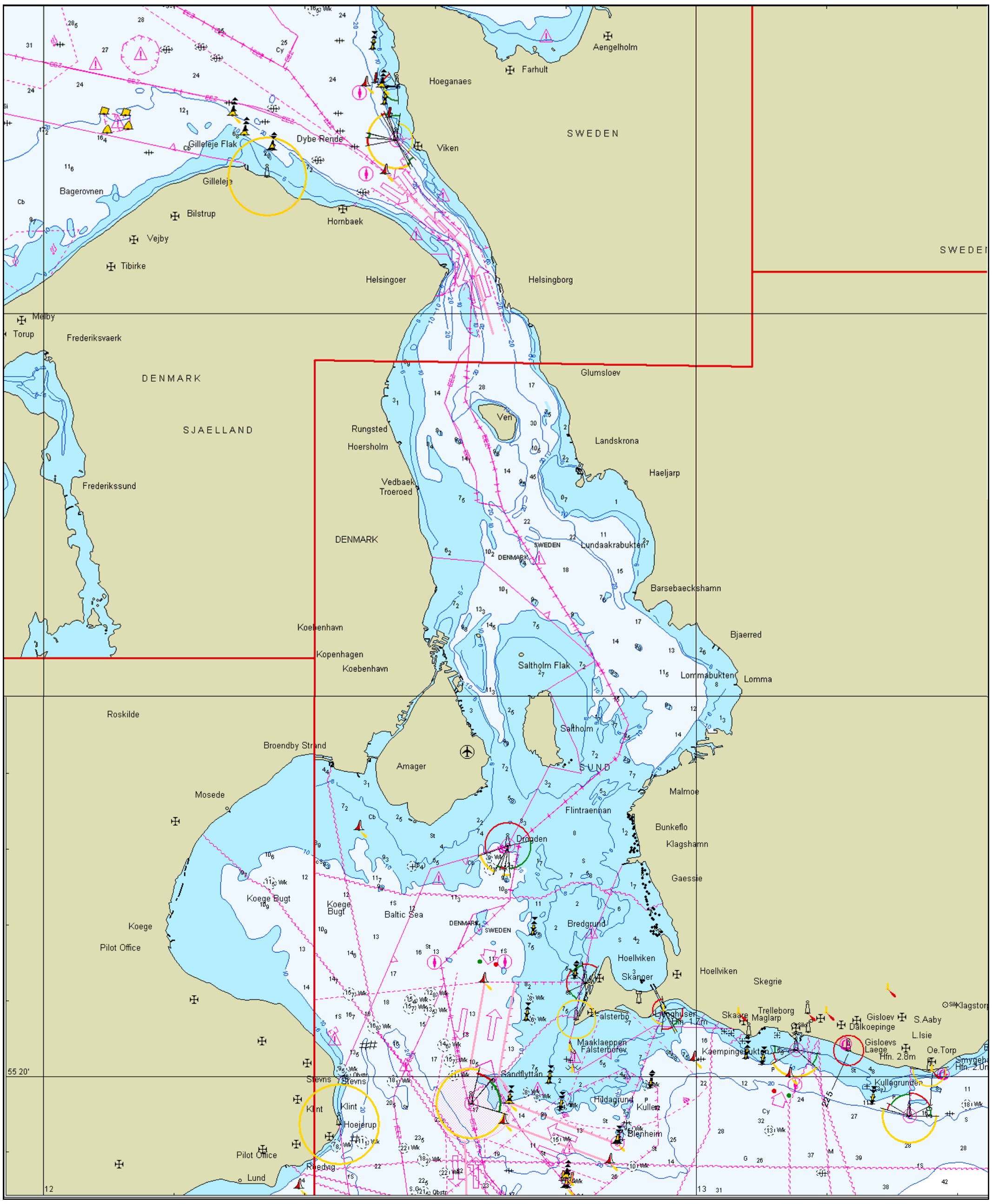


Figure 2.1 Chart covering the area of interest: Øresund, the Sound between Denmark and Sweden

2.2.1 Creating a mdf- file from the raw xyz data

The mesh file containing information about water depths and mesh is created with the Mesh Generator tool in MIKE Zero. First you should start the Mesh Generator (File→New→ File→Mesh Generator). See Figure 2.2.

After starting the Mesh Generator you should specify the projection system as UTM and the zone as 33 for the working area. See Figure 2.3.

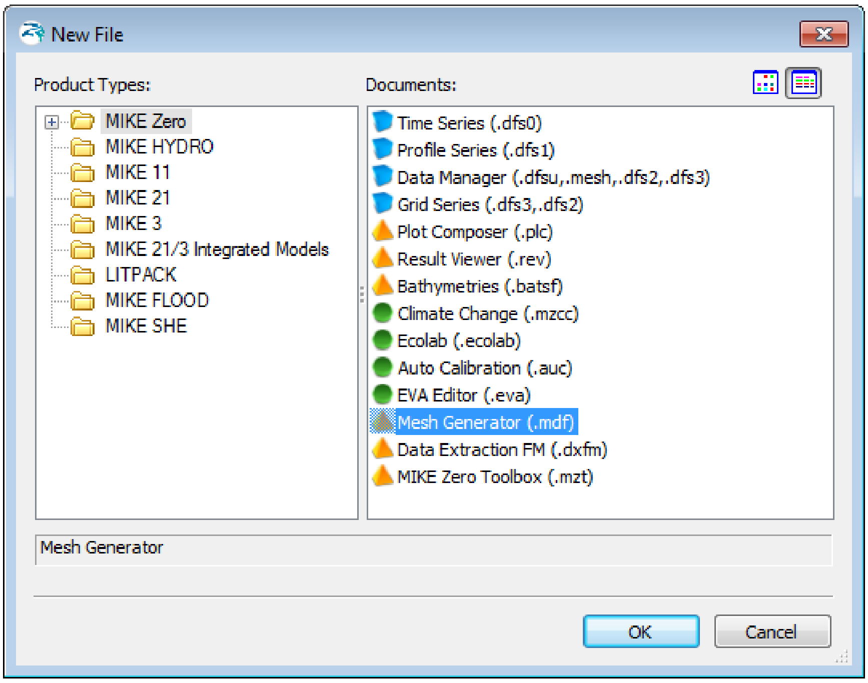


Figure 2.2 Starting the Mesh Generator tool in MIKE Zero

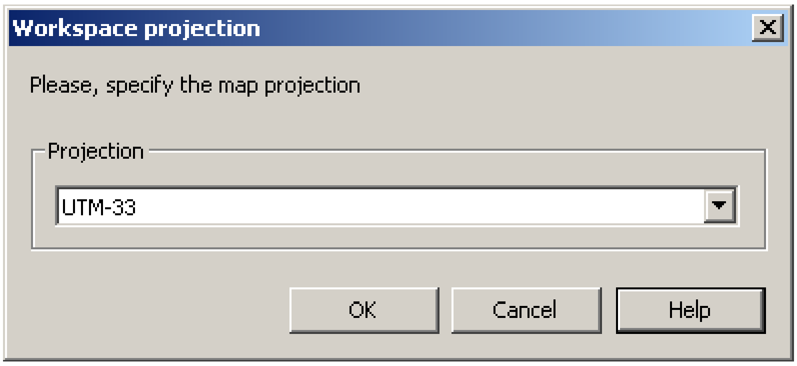


Figure 2.3 Defining the projection of the work space

The resulting working area is shown in Figure 2.4.

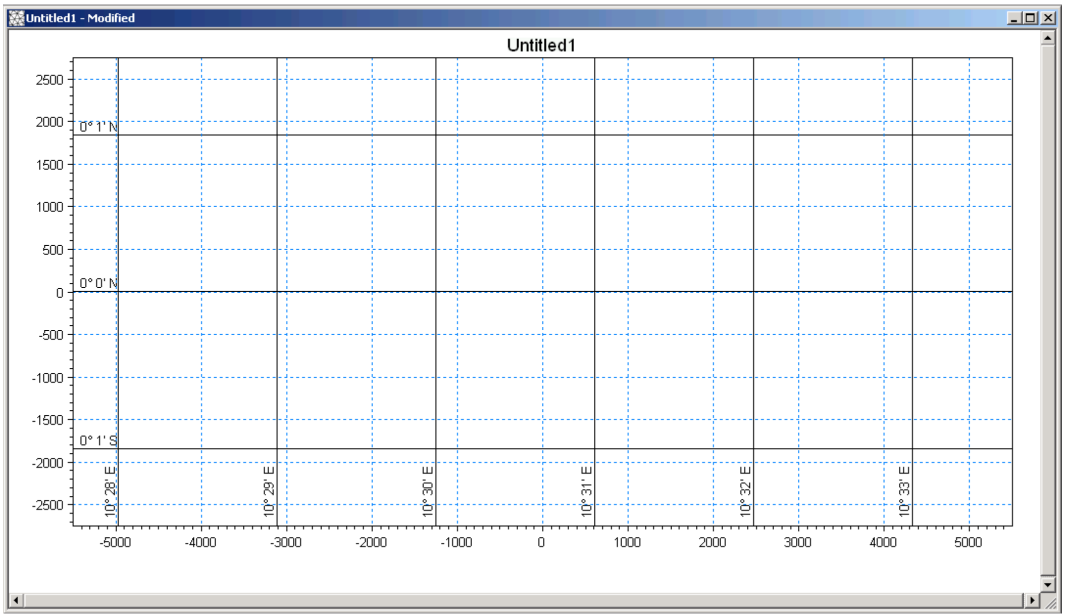


Figure 2.4 The work space as it appears in the Mesh Generator after choosing projection and before adding xyz data

Import digitised shoreline data from an ASCII file (Data→Import Boundary →Open XYZ file: land.xyz). See Figure 2.5.

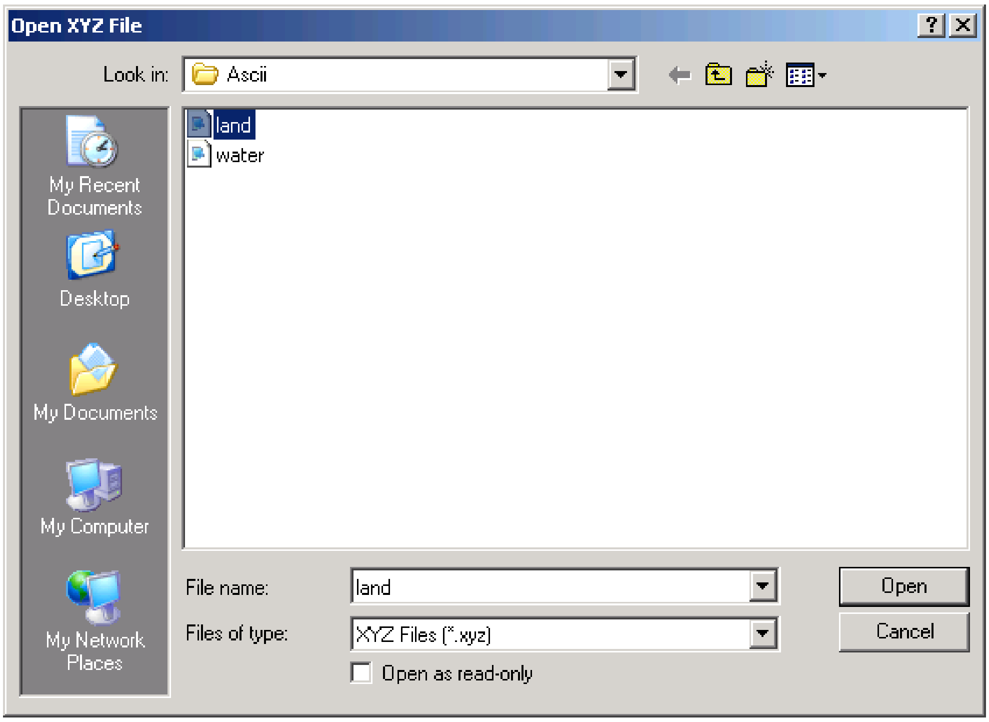


Figure 2.5 Import digitised shoreline data

Remember to convert from geographical co-ordinates: Choose Longitude/Latitude after importing shoreline date. See Figure 2.6.

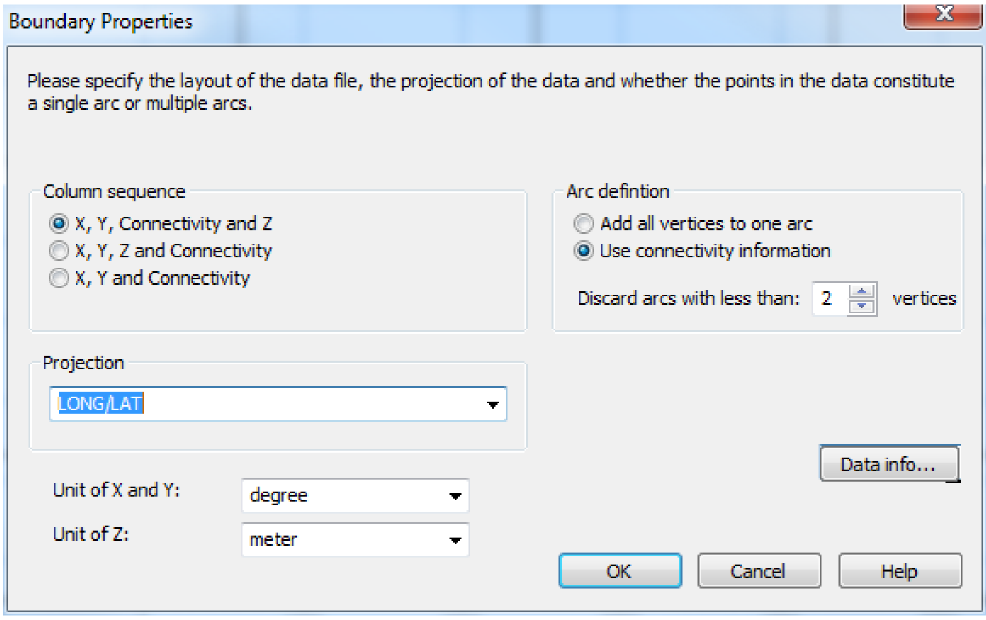


Figure 2.6 Import digitised shoreline data

The resulting workspace with the imported shoreline data will appear in a so-called Mesh Definition File (mdf-file) as shown in Figure 2.7.

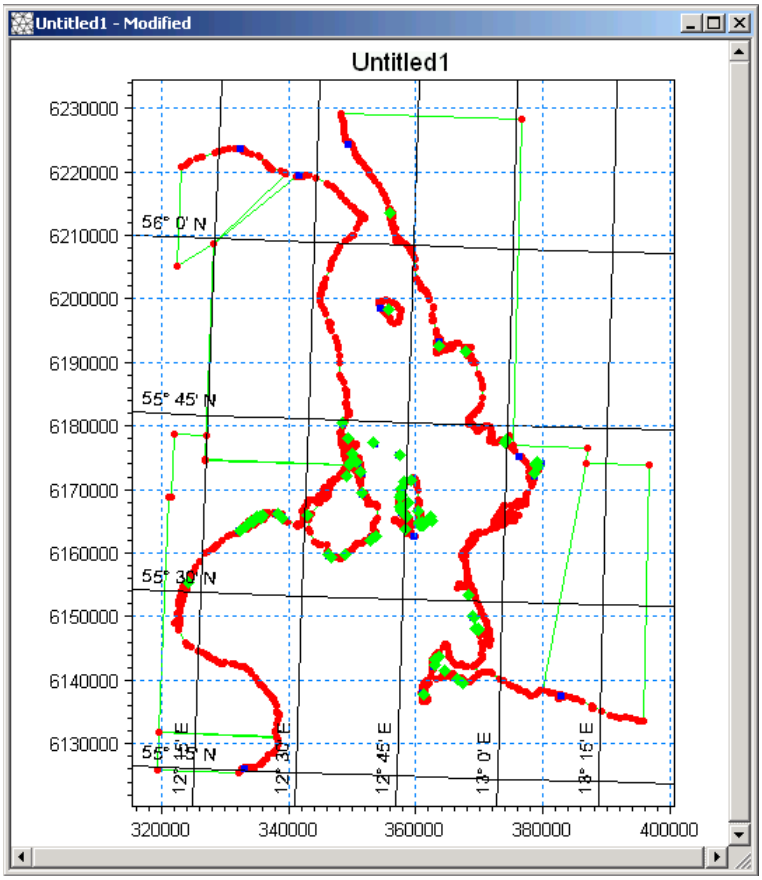


Figure 2.7 The mdf-file as it appears in the Mesh Generator after importing shoreline xyz data

The next step is to transform the raw data set, into a data set from which the selected domain can be triangulated.

2.2.2 Adjusting the boundary data into a domain that can be triangulated

This task should result in a file with boundary- and water boundaries (the green arcs) - forming a closed domain that can be triangulated.

Start by deleting the shoreline vertices and nodes (the red and blue points) that are not part of the shoreline of the area that you want to include in the bathymetry. This includes the nodes on land that you see in Figure 2.7.

Define a northern and southern boundary between Denmark and Sweden by adding arcs between 'Danish' nodes and 'Swedish' nodes. The boundaries should be placed near the co-ordinates for the boundary measurements given in Table 2.1.

Mark the North Boundary arc and choose properties. Set the arc attribute to 2 for the northern boundary. Mark 3 for the southern boundary arc. See Figure 2.8 and Figure 2.9.. These attributes are used for the model system to distinguish between the different boundary types in the mesh: the North Boundary (attribute 2) and the South Boundary (attribute 3). The land/water boundary (attribute 1) is automatically set by the Mesh Generator.

Table 2.1 Measured water level data

Station	Data File	Position	
		Easting (m)	Northing (m)
WL13 Viken	waterlevel_viken.txt	349744	6224518
WL14 Hornbæk	waterlevel_hornbaek.txt	341811	6219382
WL19 Skanør	waterlevel_skanor.txt	362748	6143316
WL20 Rødvig	waterlevel_rodvig.txt	333191	6126049

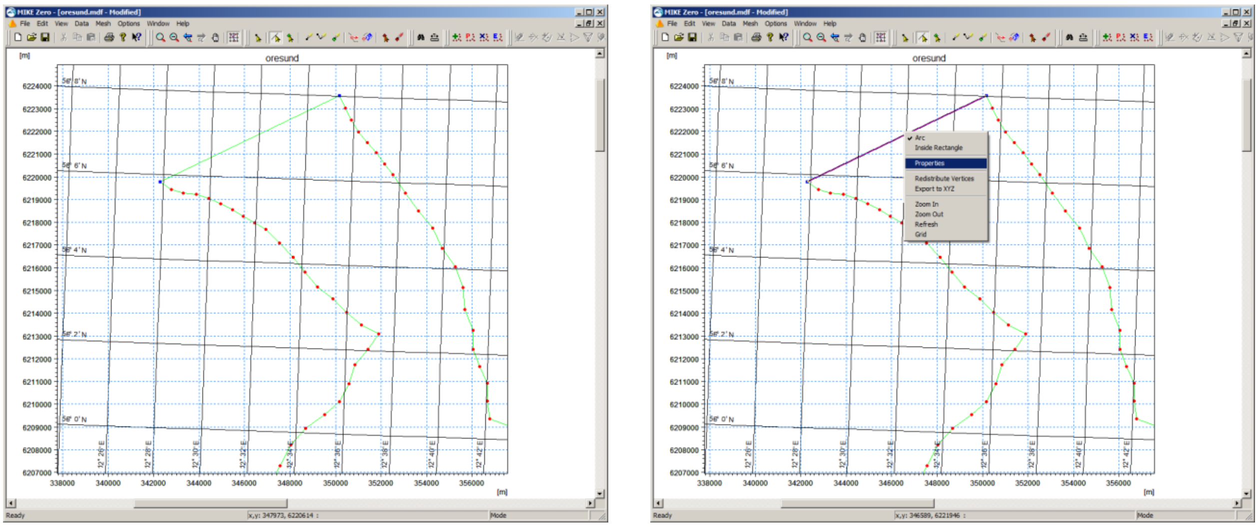


Figure 2.8 Selecting the arc for the northern boundary (purple arc) for editing properties (right click with mouse)

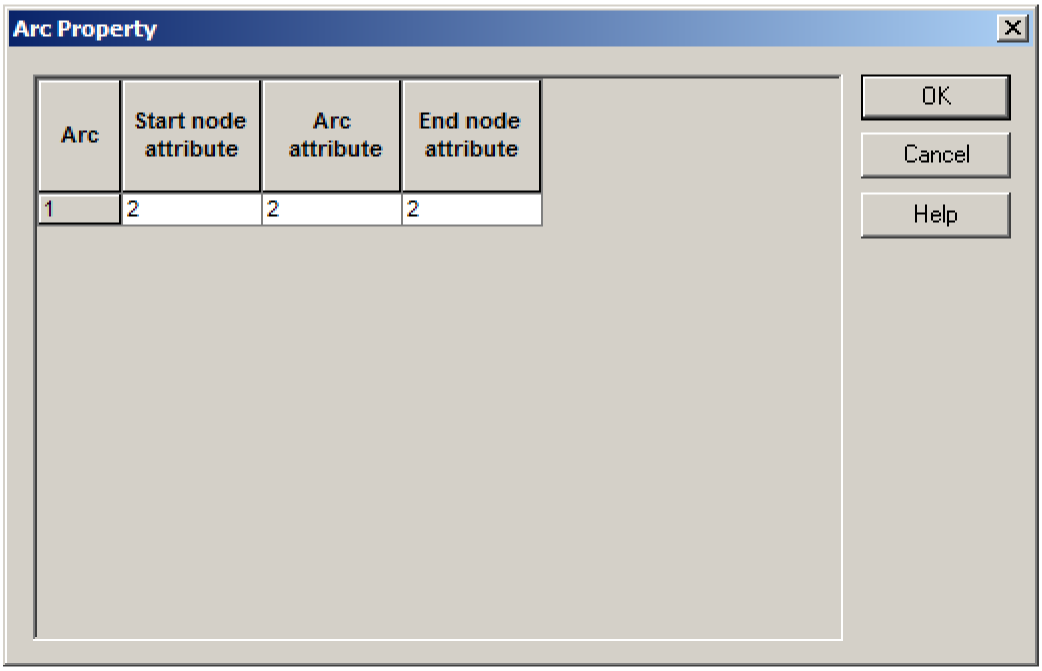


Figure 2.9 Editing the northern boundary arc properties

Now you have a closed area that can be triangulated, but first you should go through and smooth all the shorelines. It should be noted that the triangulation of the domain starts from the boundary polygon, thus the number of elements generated by the triangulation process are very dependent of the number of nodes and vertices on the shoreline.

You can use the tool for redistributing the vertices along an arc to form a more uniform shoreline. In areas of special interest you can redistribute the vertices along the land boundary with a shorter distance between them. You might consider not including the bathymetry in inner harbours and lagoons, if they do not have significant effect on the results in the areas, where you do have interest.

After editing all the shorelines you might end up with an mdf-file similar to the one found in this example called: oresund.mdf. See Figure 2.10. Use this file for the further work with this example.

2.2.3 Triangulation of the domain

The next step is to triangulate the domain. First you mark the closed areas (islands) that should not be triangulated as polygons (green marks).

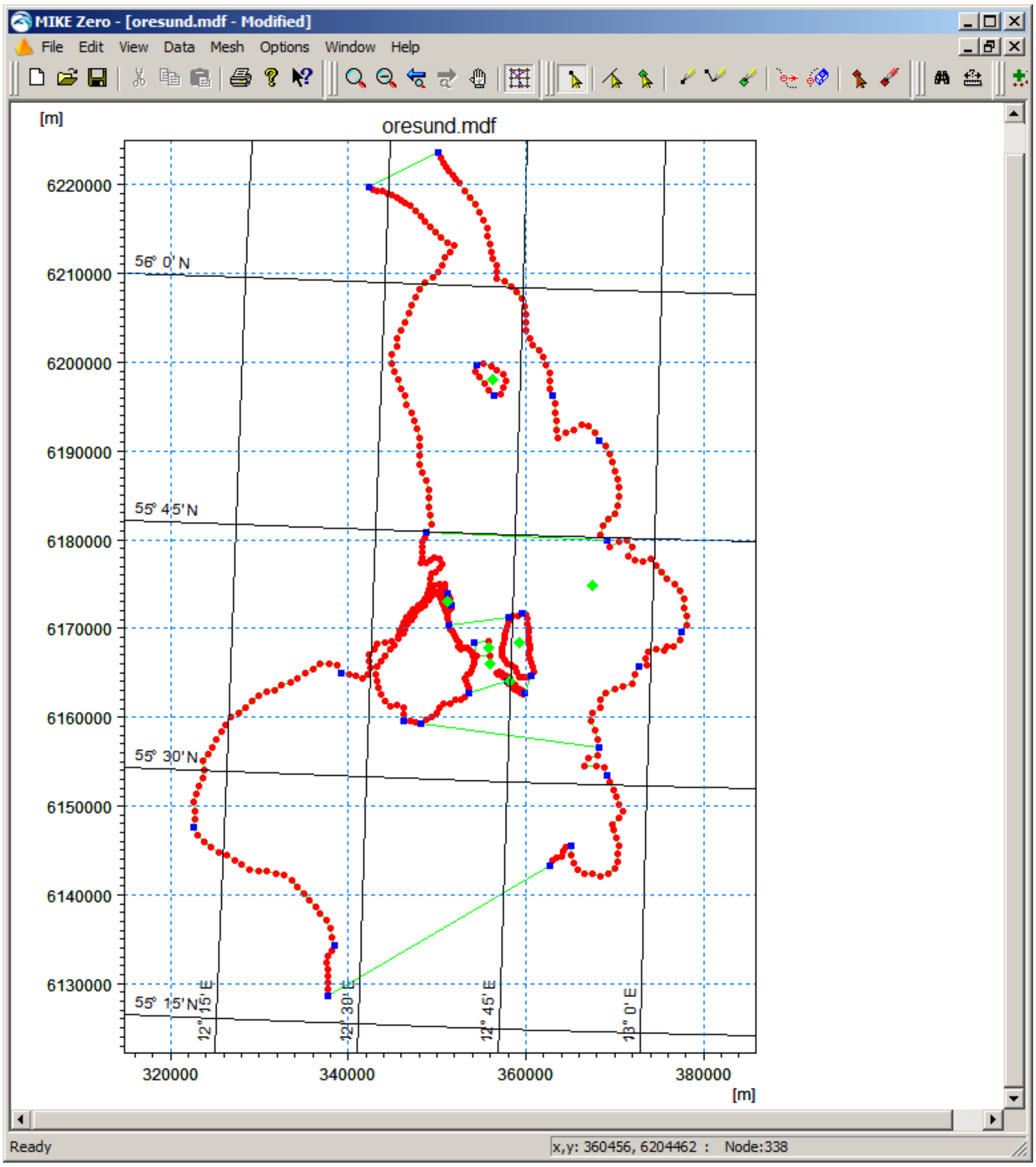


Figure 2.10 The mdf-file as it appears in the Mesh Generator after modifying Boundary xyz data to form a closed area surrounded by boundaries ready for triangulation

Try to make the first triangulation (Mesh→Generate Mesh). Use the triangulation option settings as in Figure 2.11. You can see that in this example, the area of interest has a higher resolution than the rest of the mesh. You might also consider refining the mesh in specific areas. You can do that by adding a polygon in the area of interest and giving the polygon local properties (add green mark in polygon and right click the mark for defining its properties).

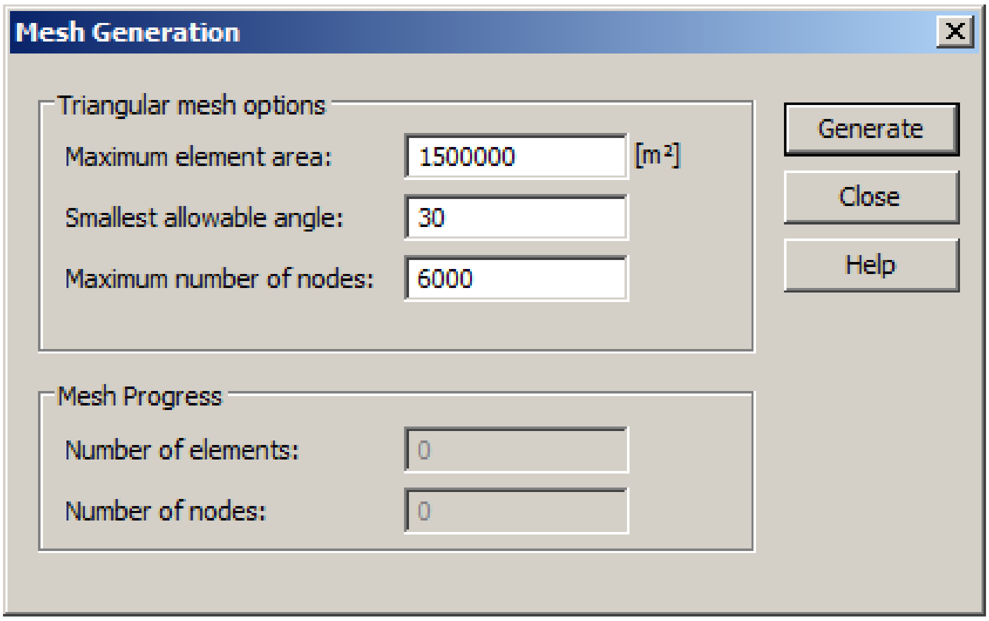


Figure 2.11 The triangulation options as they are set in this example

After the triangulation you can use a tool for smoothing the mesh (Mesh→Smooth Mesh). In this example the mesh has been smoothed 100 times. The resulting mesh will look as in Figure 2.12.

First import the xyz-file containing the water depths (Data→Manage Scatter Data→Add→water.xyz. Specify projection as Long/Lat for the 1993 data and UTM 33 for the 1997 data. The ASCII files are found in the folders:

.\Data\1993\Ascii\water.xyz	(for MIKE 21 modelling)
.\Data\1997\Ascii\water.xyz	(for MIKE 3 modelling)

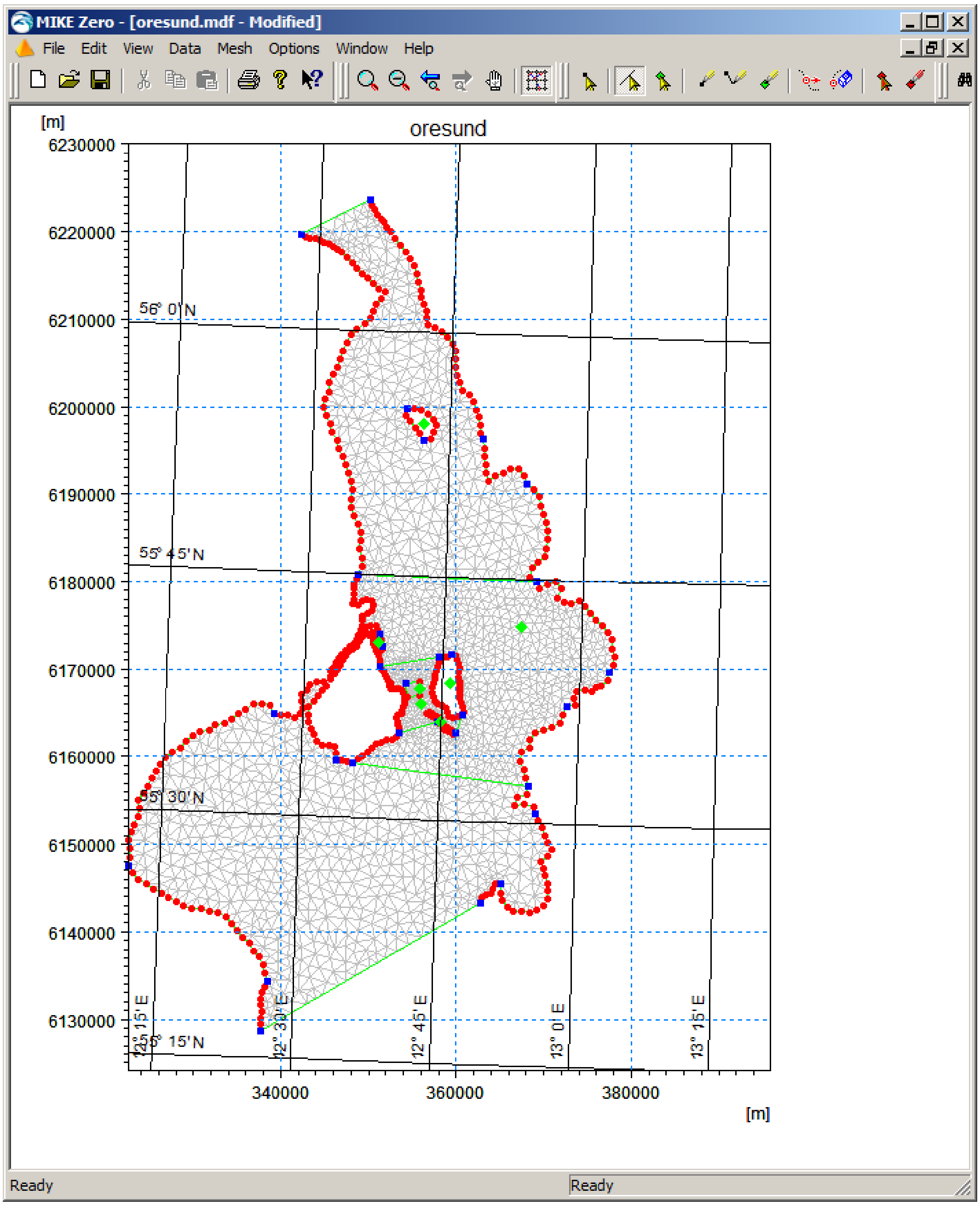


Figure 2.12 The mesh as it appears in the Mesh Generator after triangulation and smoothing 100 times. The triangulation options allow a maximum area of 1500000 m^2, a minimum angle of 30 degrees, and maximum of 6000 nodes. Local triangulation options are given in polygons around the areas of interest.

Afterwards you have to interpolate the water depths in the appropriate xyz data file into the mesh: water.xyz (Mesh→Interpolate). See Figure 2.13.

Please note that the water depths are different in the MIKE 21 FM example and the MIKE 3 FM example, because they cover two different years, so the morphology has changed. The resulting mdf-file for the MIKE 21 bathymetry for 1993 will look as in Figure 2.15.

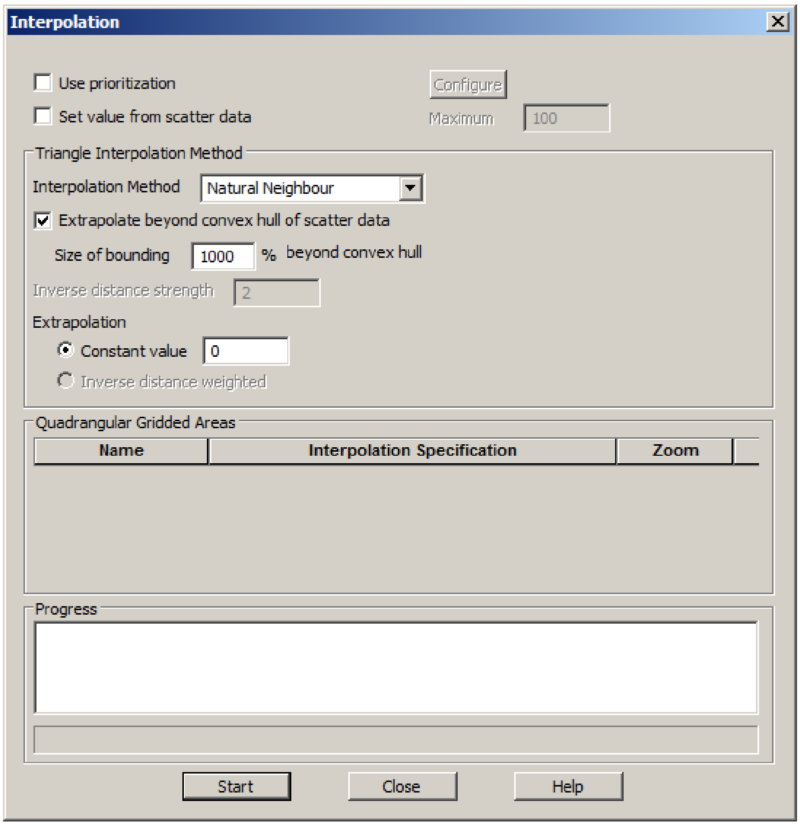
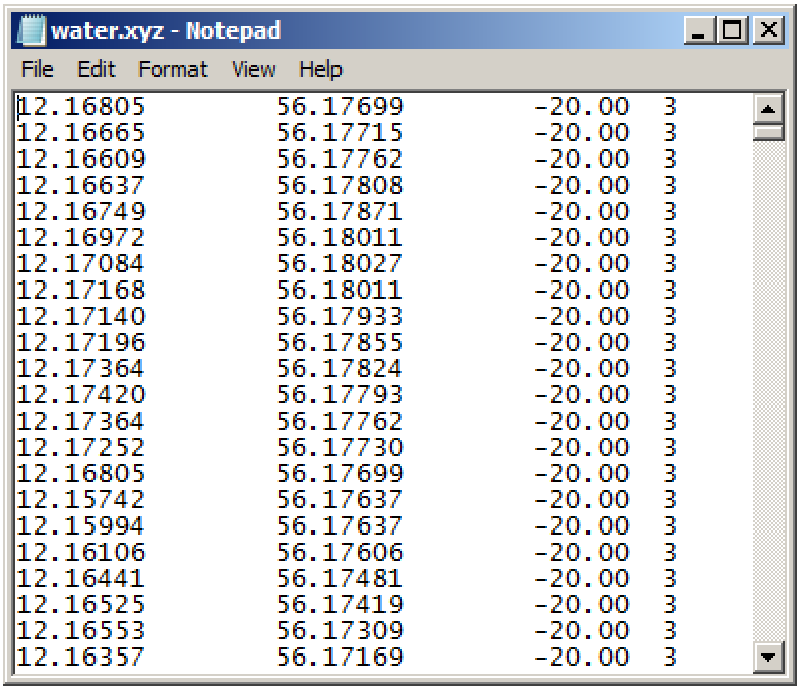


Figure 2.13 Interpolating water depths into the domain from imported XYZ file


water.xyz - Notepad

File Edit Format View Help

12.16805	56.17699	-20.00	3
12.16665	56.17715	-20.00	3
12.16609	56.17762	-20.00	3
12.16637	56.17808	-20.00	3
12.16749	56.17871	-20.00	3
12.16972	56.18011	-20.00	3
12.17084	56.18027	-20.00	3
12.17168	56.18011	-20.00	3
12.17140	56.17933	-20.00	3
12.17196	56.17855	-20.00	3
12.17364	56.17824	-20.00	3
12.17420	56.17793	-20.00	3
12.17364	56.17762	-20.00	3
12.17252	56.17730	-20.00	3
12.16805	56.17699	-20.00	3
12.15742	56.17637	-20.00	3
12.15994	56.17637	-20.00	3
12.16106	56.17606	-20.00	3
12.16441	56.17481	-20.00	3
12.16525	56.17419	-20.00	3
12.16553	56.17309	-20.00	3
12.16357	56.17169	-20.00	3

Figure 2.14 ASCII file describing the depth at specified geographical positions (Longitude, Latitude and Depth). Please note that if MIKE C-map is used you are not allowed to view the data in a text editor because the data is encrypted

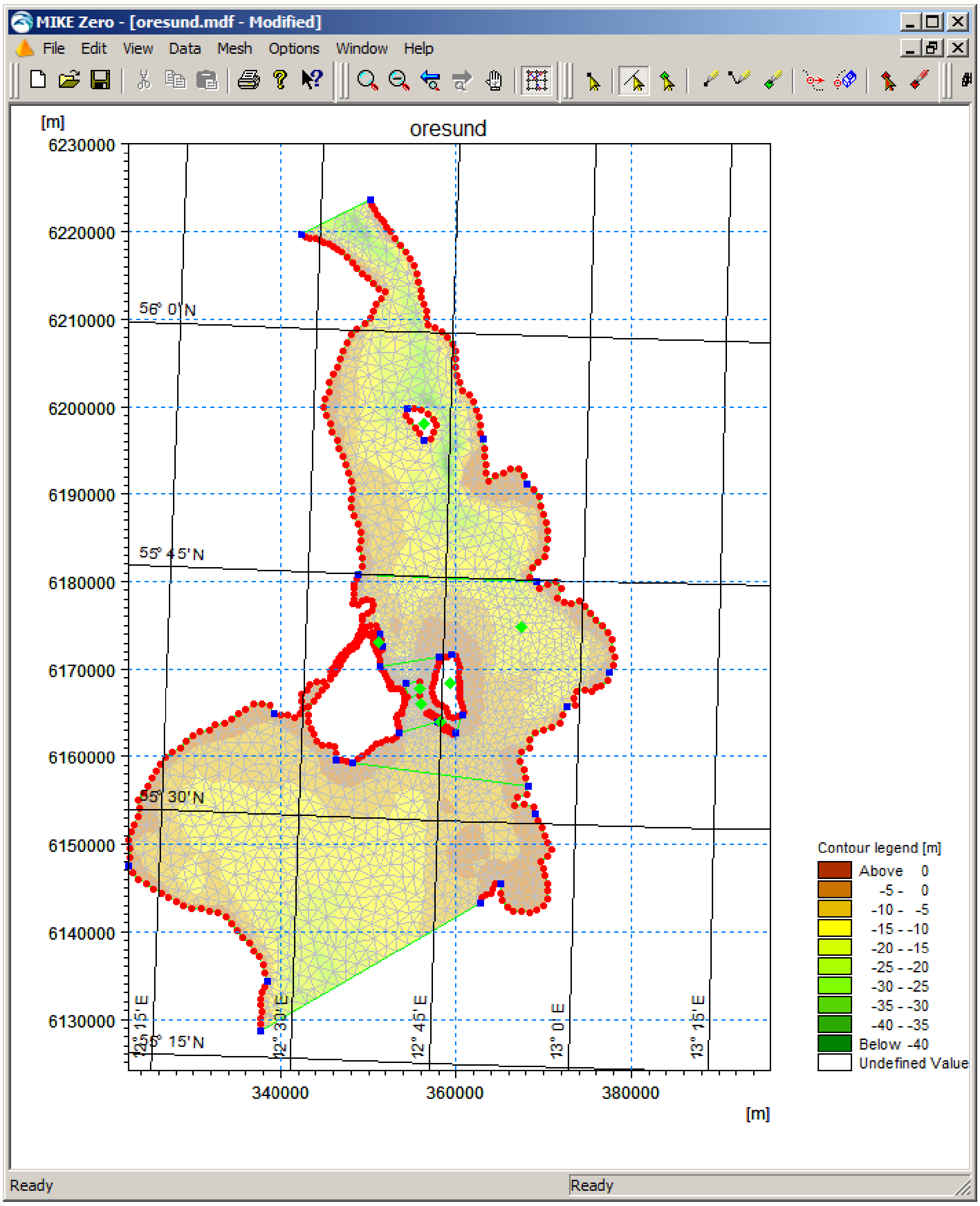


Figure 2.15 The mesh as it appears in the Mesh Generator after interpolating water depth xyz data into the mesh

Now you are ready to export the data set into a mesh file that can be used in MIKE 21 & MIKE 3 Flow Model FM (Mesh→Export Mesh). Save the file as oresund.mesh.

You can view (and edit) the resulting mesh file in the Data Viewer (Figure 2.16) or view it in MIKE Animator Plus (Figure 2.17).

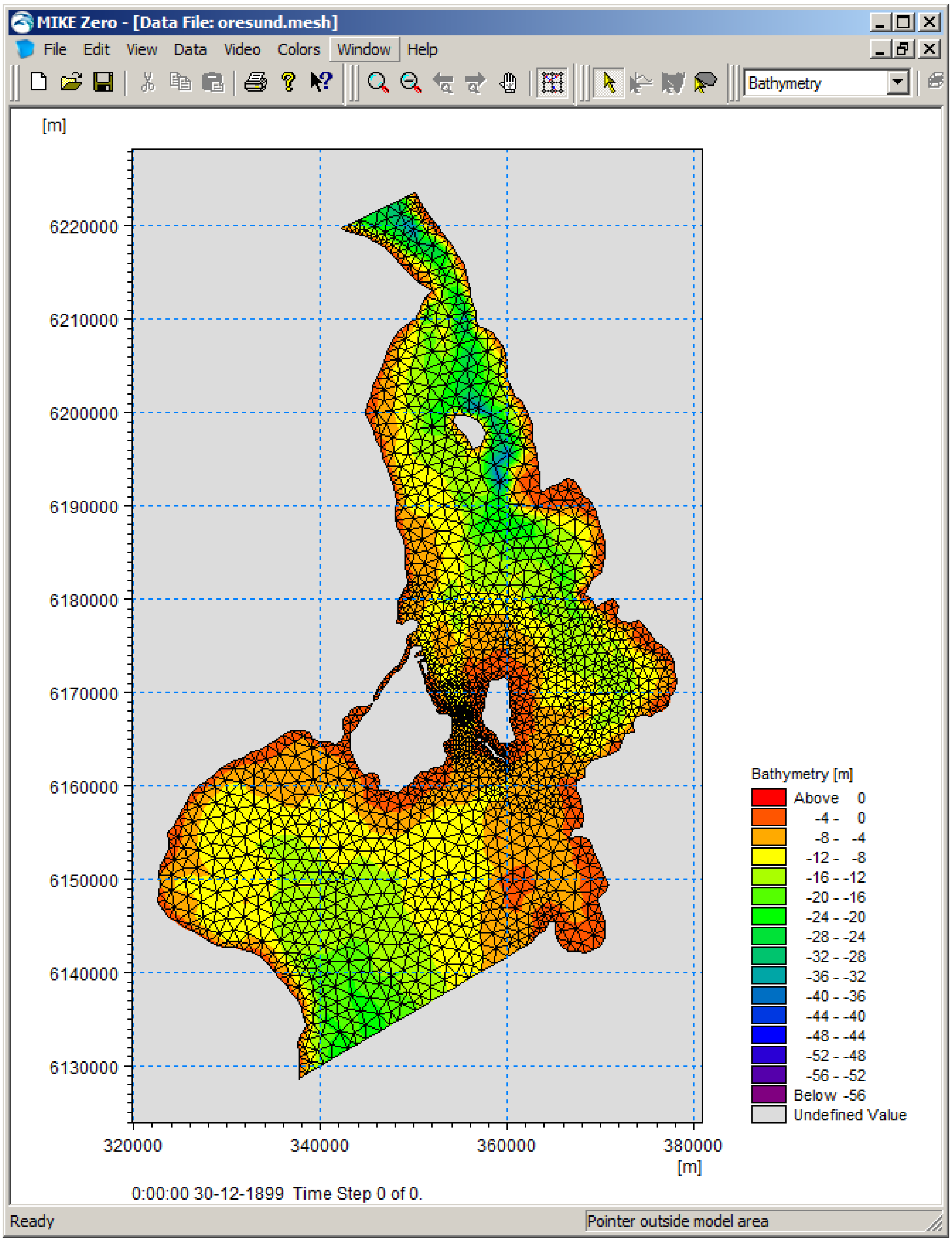


Figure 2.16 The Øresund mesh file as it appears in the Data Viewer

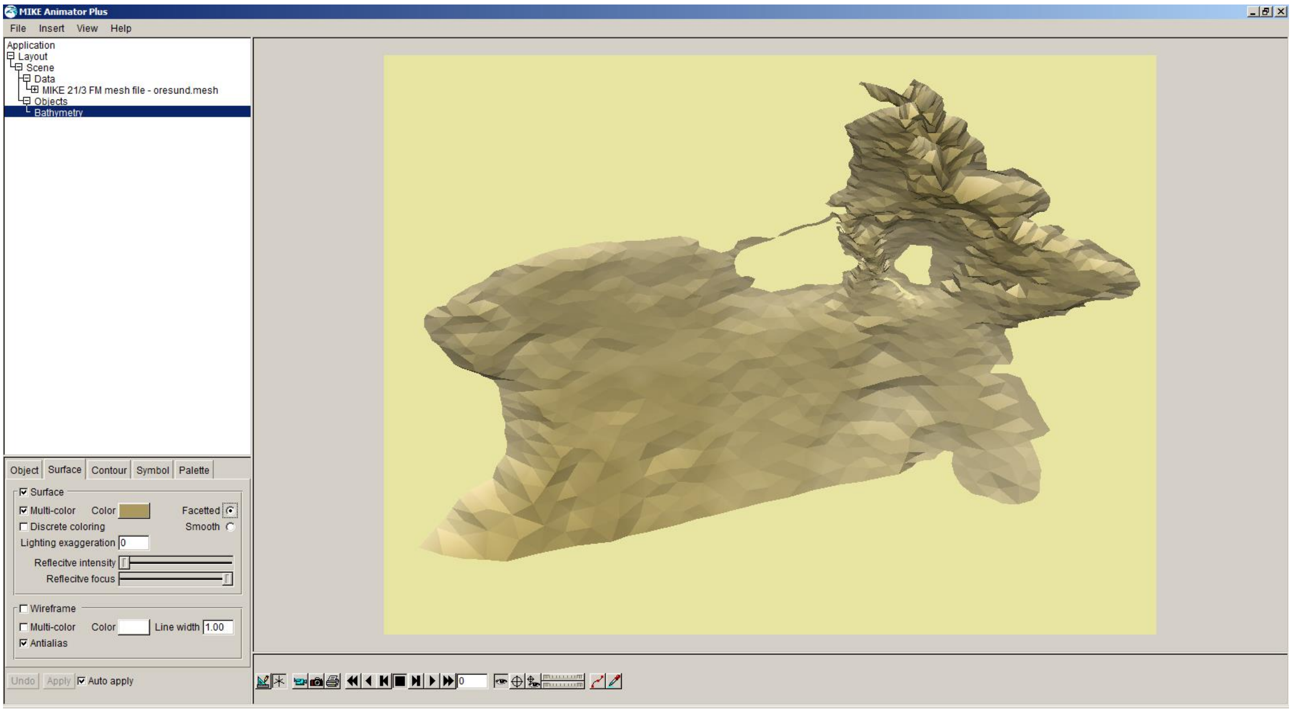


Figure 2.17 The Øresund mesh as it can be presented with the MIKE Animator Plus tool

3 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 must be generated from the measurements. Measurements for 1993 exist for

- Water levels at the boundaries
- Wind at Kastrup Airport (Copenhagen, Denmark)

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 four stations located near the open model boundaries are available for the Øresund model, see Figure 3.1.

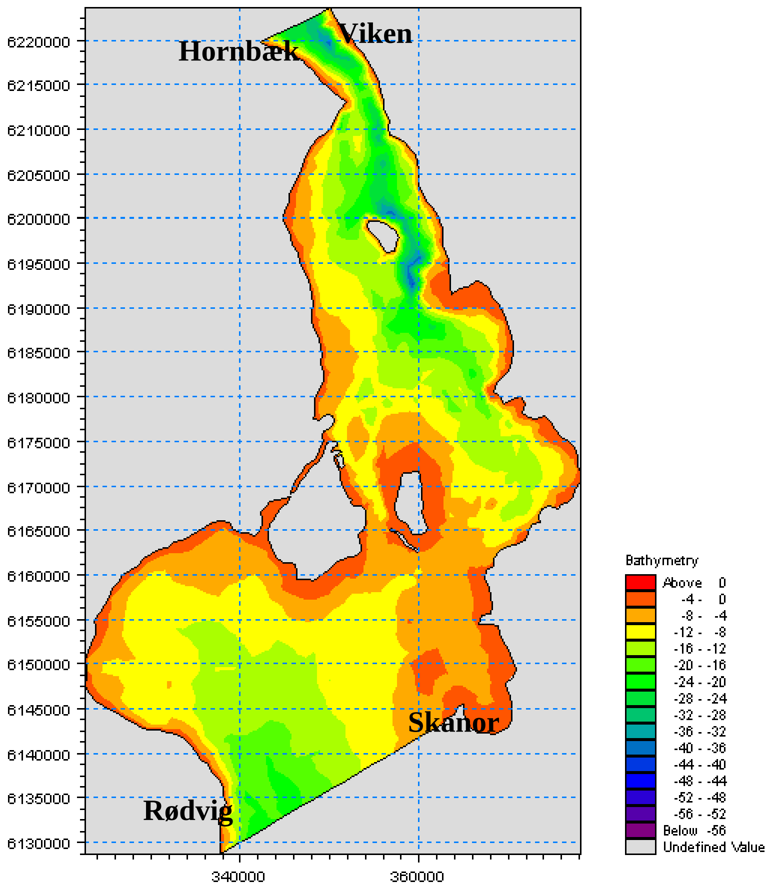


Figure 3.1 Map showing the water level stations at the open boundaries: Hornbæk, Viken, Skanör, and Rødvig

The Øresund model is forced with water level boundaries. The water level measurements indicate that the variations along the boundaries are significant, so the water level boundaries should be specified as line series (dfs1 type data file) containing an interpolation between the two measurements at each boundary.

In the following, two line series (dfs1 type data file) with water level variations will be created on basis of measured recordings from four stations on two boundaries. The locations of the four stations are listed in Table 2.1

3.1.1 Importing measured water levels to time series file

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

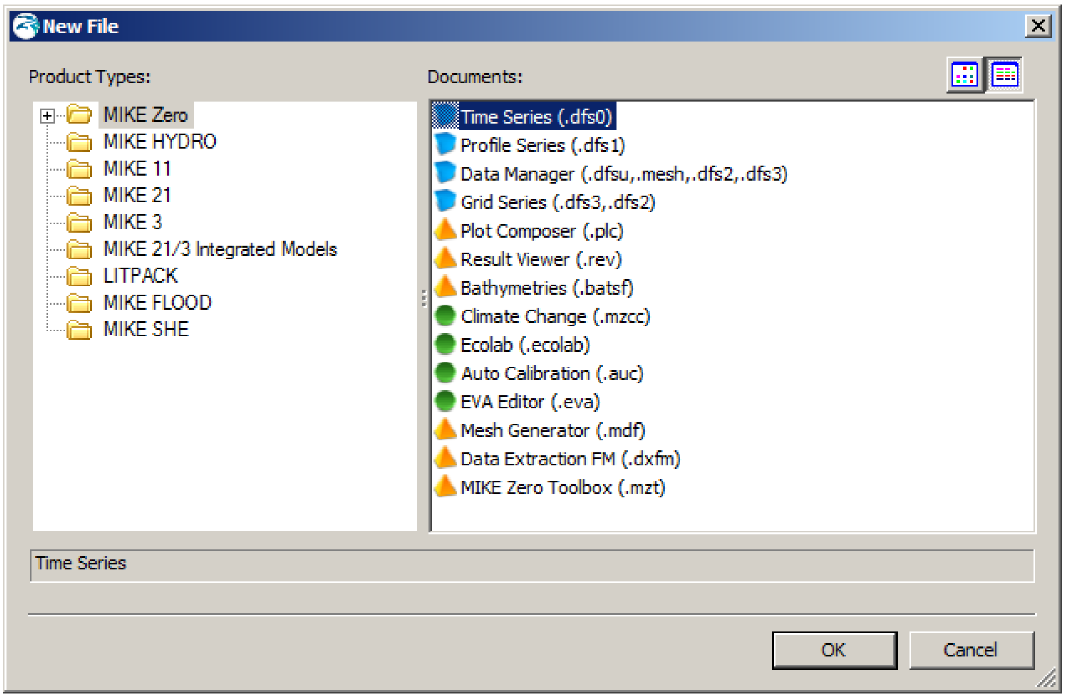


Figure 3.2 Starting the Time Series Editor in MIKE Zero

Select the ASCII template. Open the text file waterlevel_hornbaek.txt shown in Figure 3.3. Change the time description to 'Equidistant Calendar Axis' and press OK, see Figure 3.4. Then right click on the generated data in the Time Series Editor and select properties, see Figure 3.5. Change the item type to 'Water Level', see Figure 3.6, and finally, save the data in waterlevel_hornbaek.dfs0.

Repeat these steps for the remaining three stations.

Please note that time series must have equidistant time steps in the present version of MIKE 21 & MIKE 3 Flow Model FM. That means that if the raw data have time gaps without measurements, the gaps in the raw data must be filled (e.g. by interpolation) before importing it.

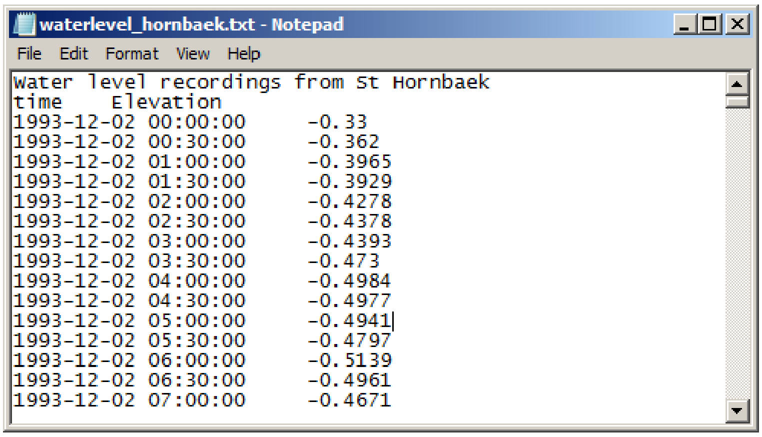


Figure 3.3 ASCII file with water level recordings from Station Hornbæk

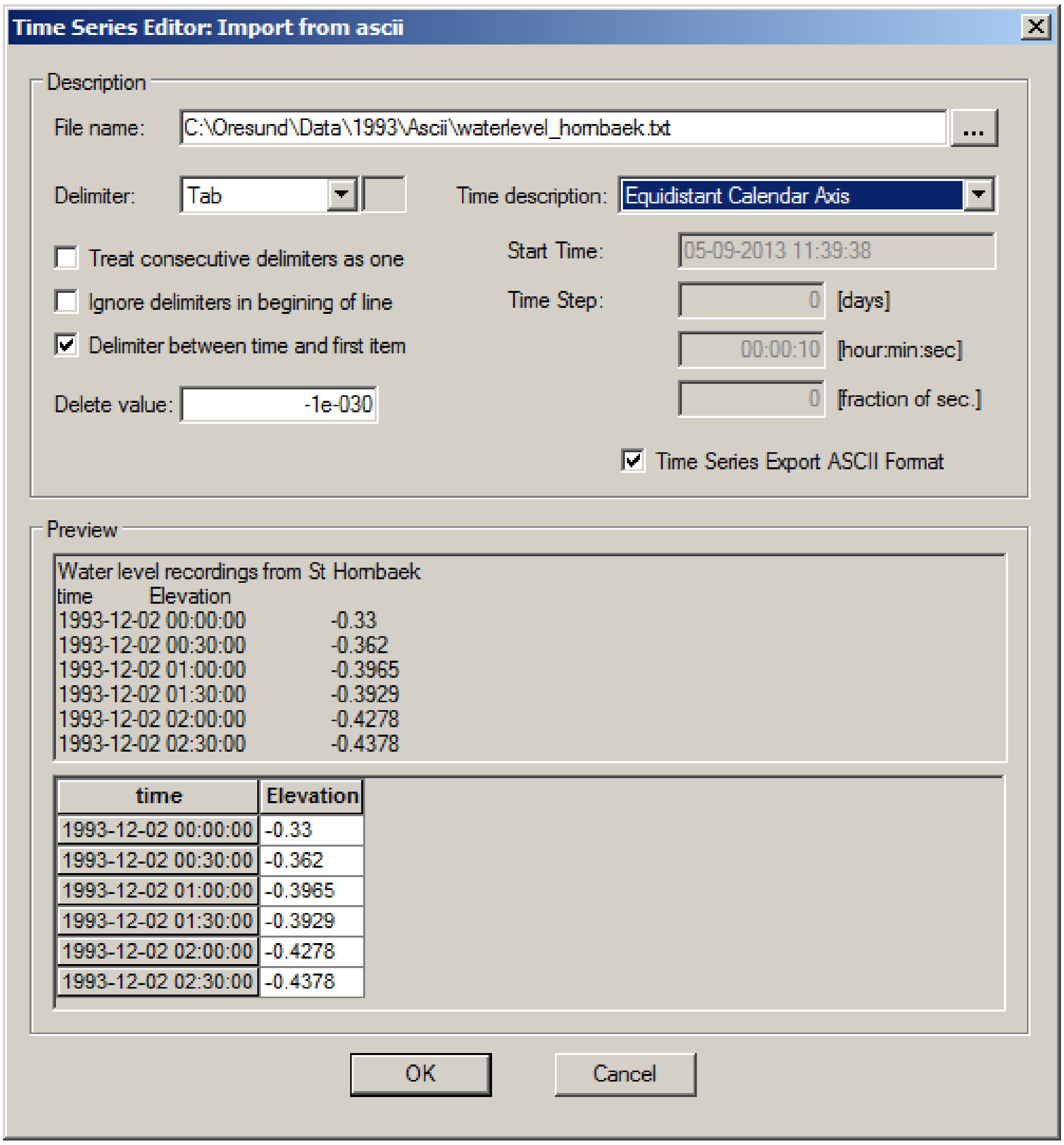


Figure 3.4 Time Series Editor: Import from ascii

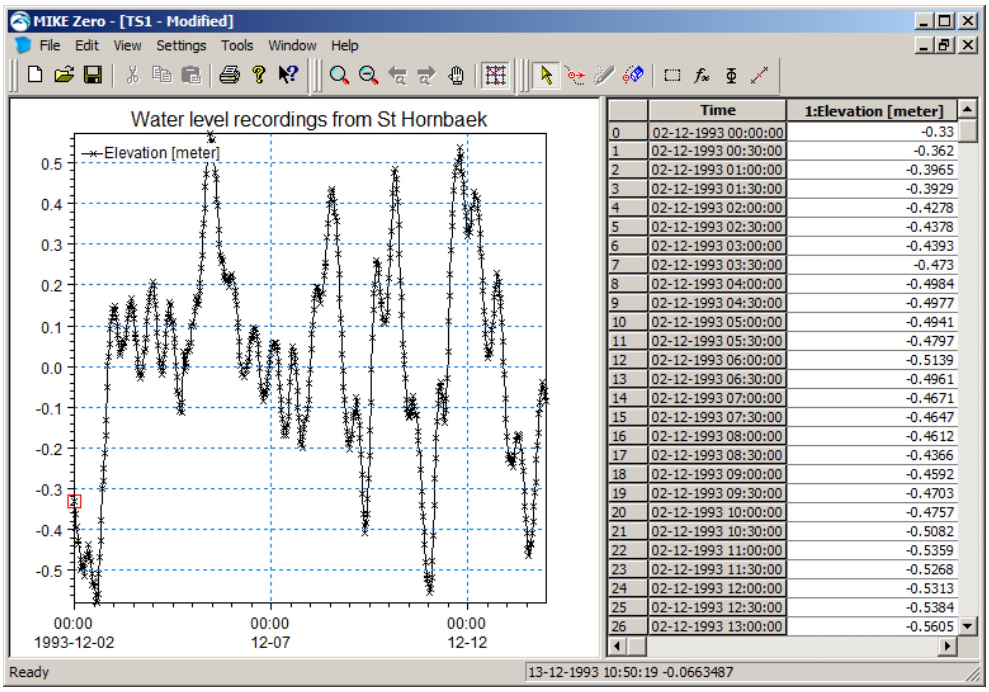


Figure 3.5 Time Series Editor with imported Water Levels from Station 1: Hornbæk

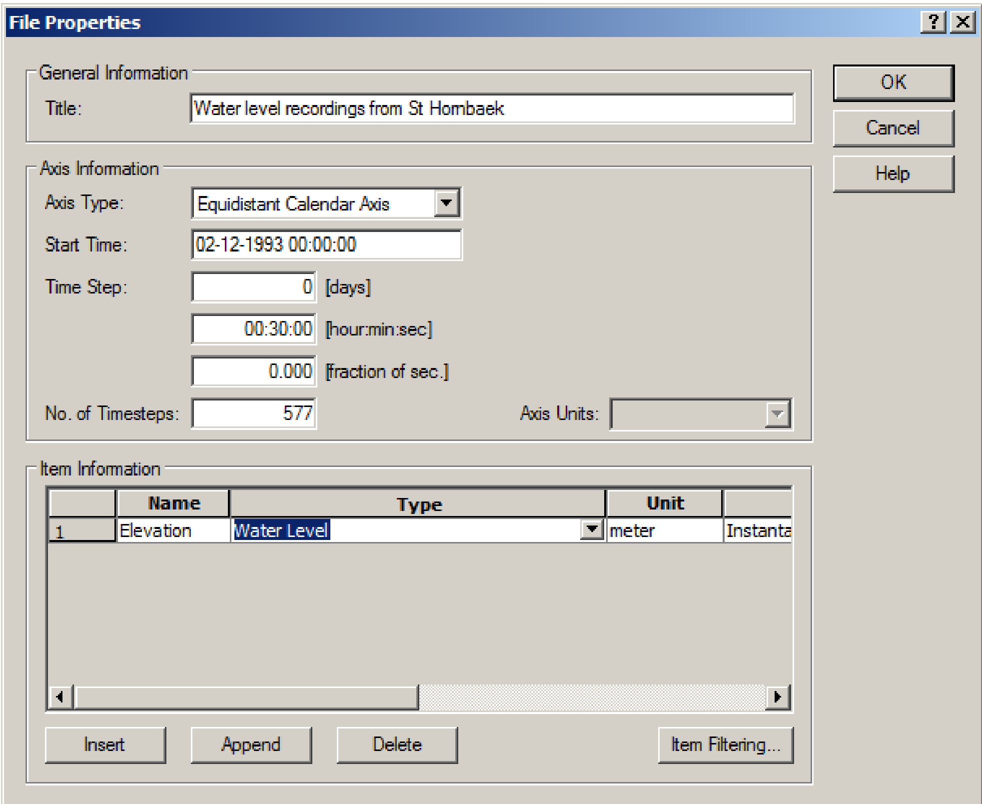


Figure 3.6 Time Series Properties

To make a plot of the water level time series open the Plot Composer in MIKE Zero, see Figure 3.7. Select 'plot' → 'insert a new plot object' and select 'Time Series Plot' (see Figure 3.8).

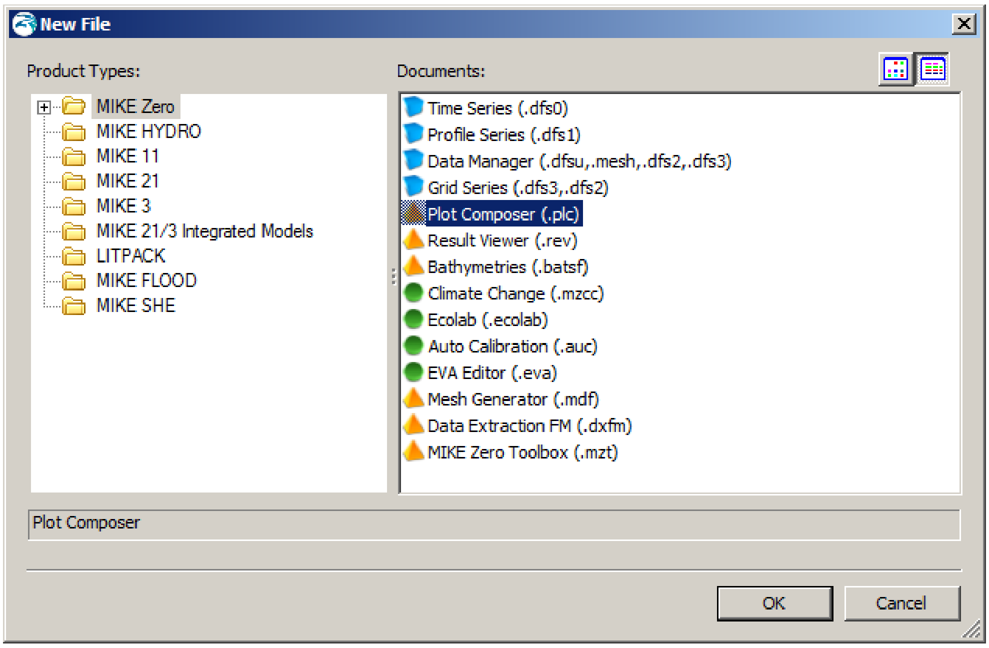


Figure 3.7 Starting the Plot Composer in MIKE Zero

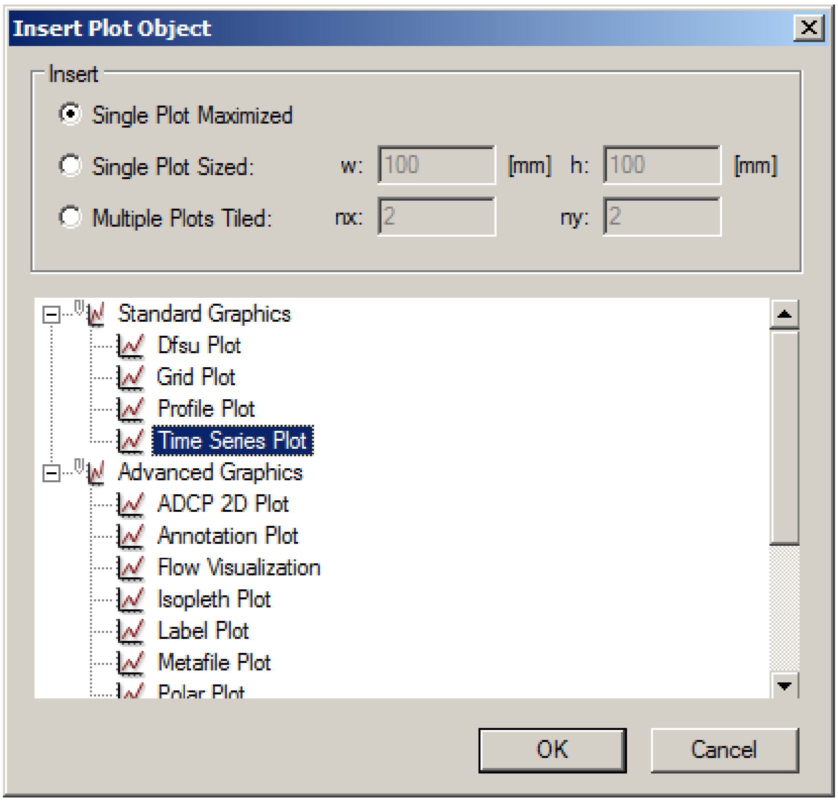


Figure 3.8 Insert a new Plot Object as Time Series in Plot Composer

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 Time Series Plot Properties dialogue it is possible to change some of the properties for the plot, such as colours, etc. (see Figure 3.10).

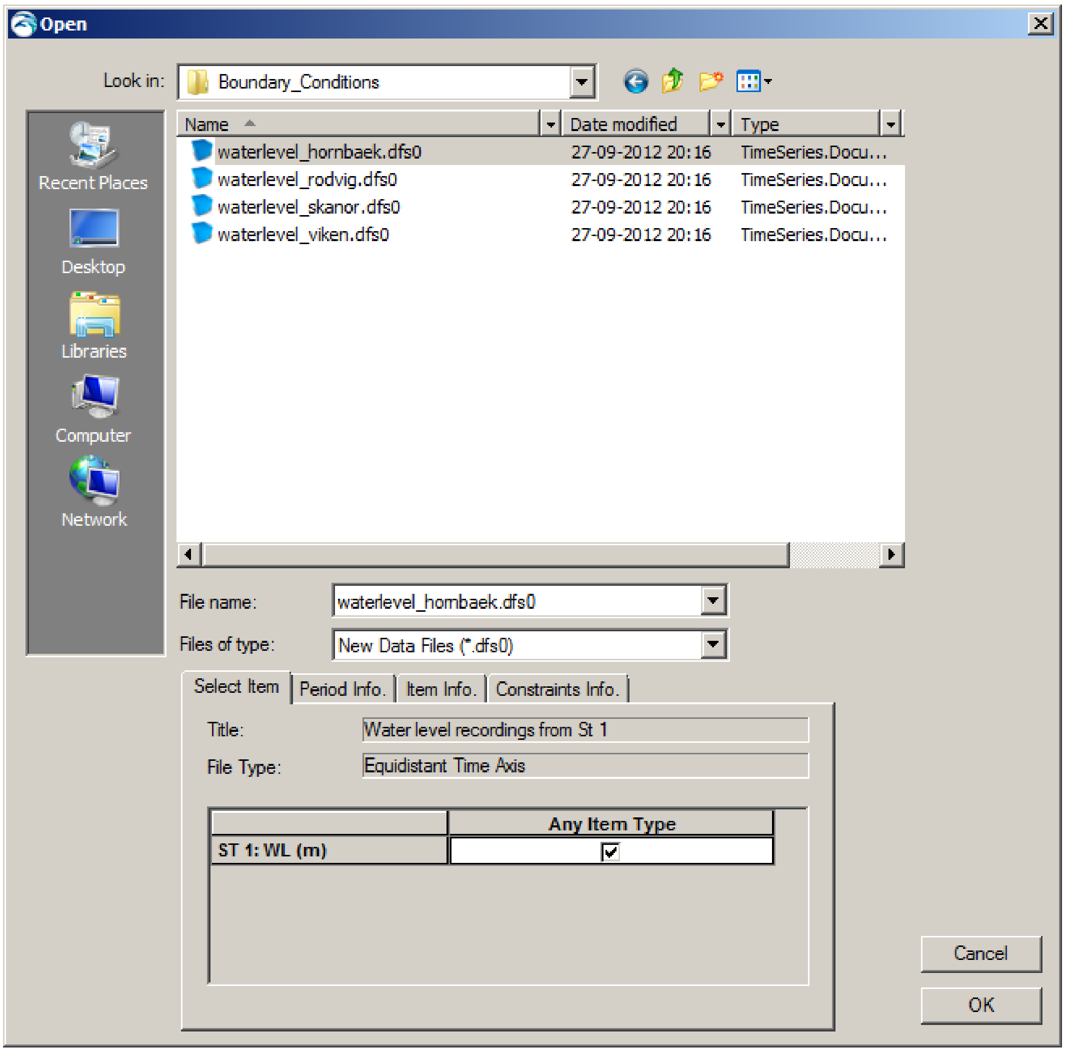


Figure 3.9 Selection of time series files in the Plot Composer

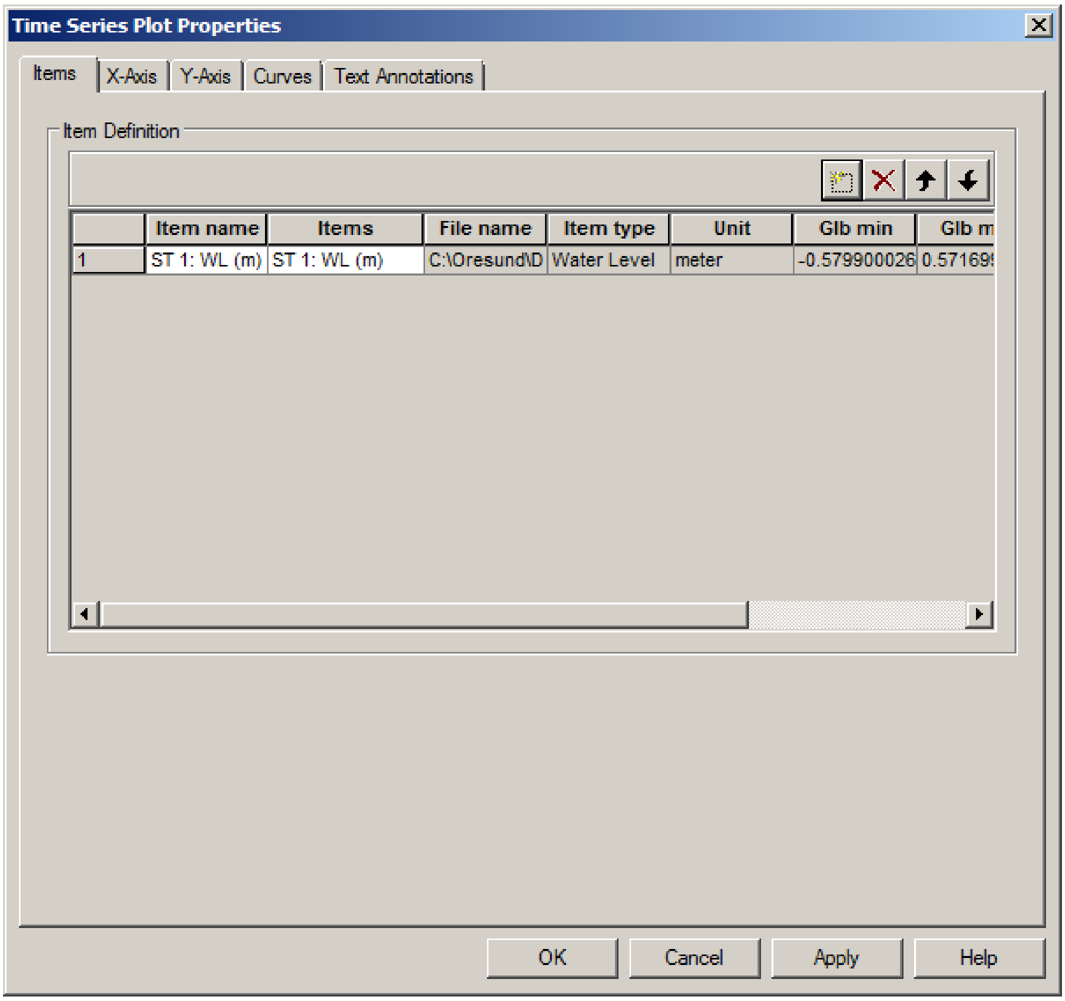


Figure 3.10 Plot Composer Time Series Plot Properties dialogue for selecting time series files and adjusting scales, curves, etc.

Figure 3.11 and Figure 3.12 show the measured water levels at the two boundaries.

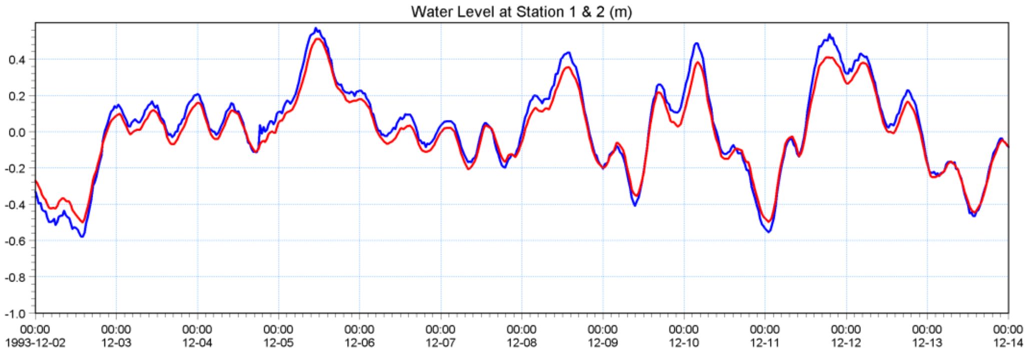


Figure 3.11 Combined time series at the North Boundary, Stations 1 and 2: Hornbæk and Viken

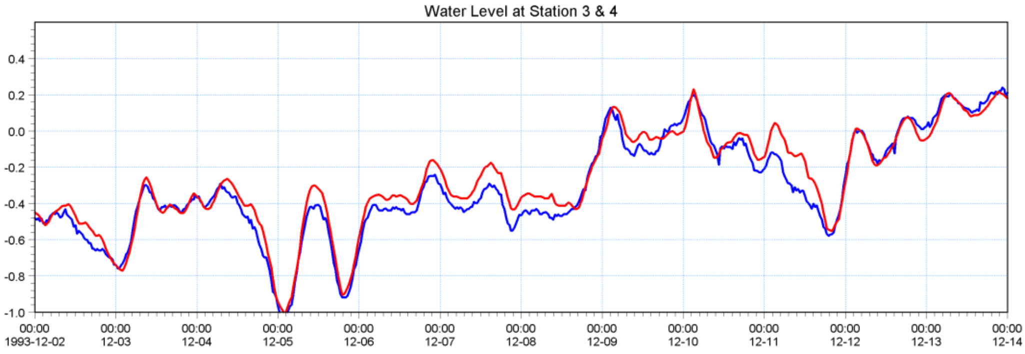


Figure 3.12 Combined time series at the South Boundary, Stations 3 and 4: Skanör and Rødvig

3.1.2 Creating boundary conditions

The next step is to create line series from the generated time series. Load the Profile Series in MIKE Zero and select 'Blank ...', see Figure 3.13.

Fill in the required information as shown in Figure 3.14:

North Boundary

- Start date 1993-12-02 00:00:00
- Time step: 1800s
- No. of time steps: 577
- No. of grid points: 2
- Grid Step: 9200m (width of boundary, actually not necessary, because MIKE 21 & MIKE 3 Flow Model FM interpolates the line series to the boundary nodes without respect to this distance, see the MIKE 21 and MIKE 3 Flow Model FM User Guides:
  - MIKE 21 Flow Model FM, Hydrodynamic Module, User Guide
  - MIKE 3 Flow Model FM, Hydrodynamic Module, User Guide

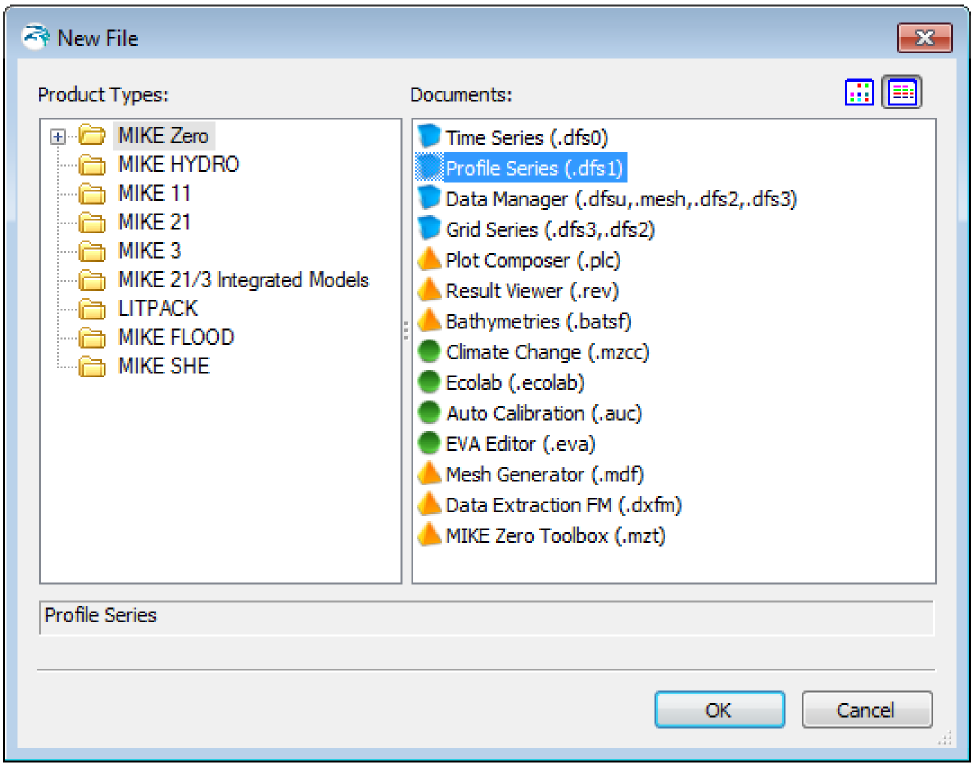


Figure 3.13 Starting the Profile Series Editor in MIKE Zero

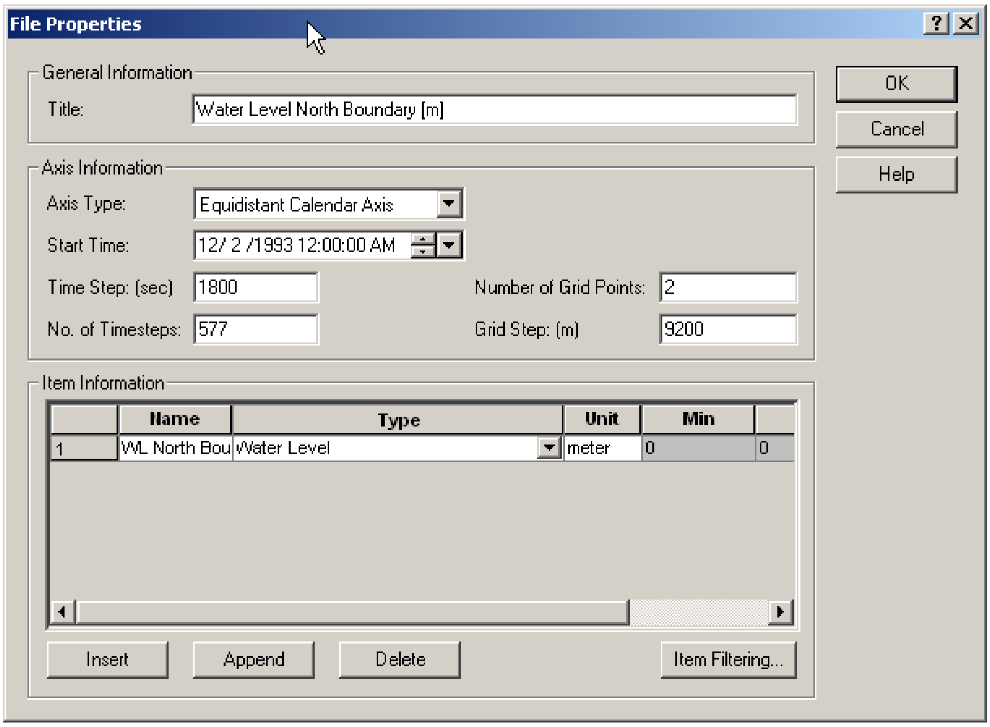
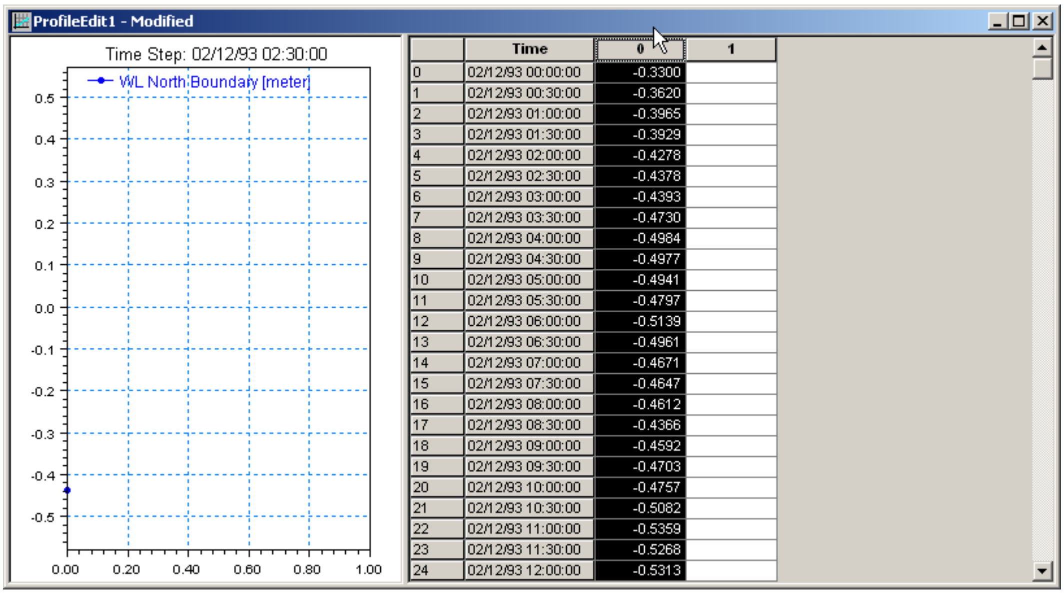


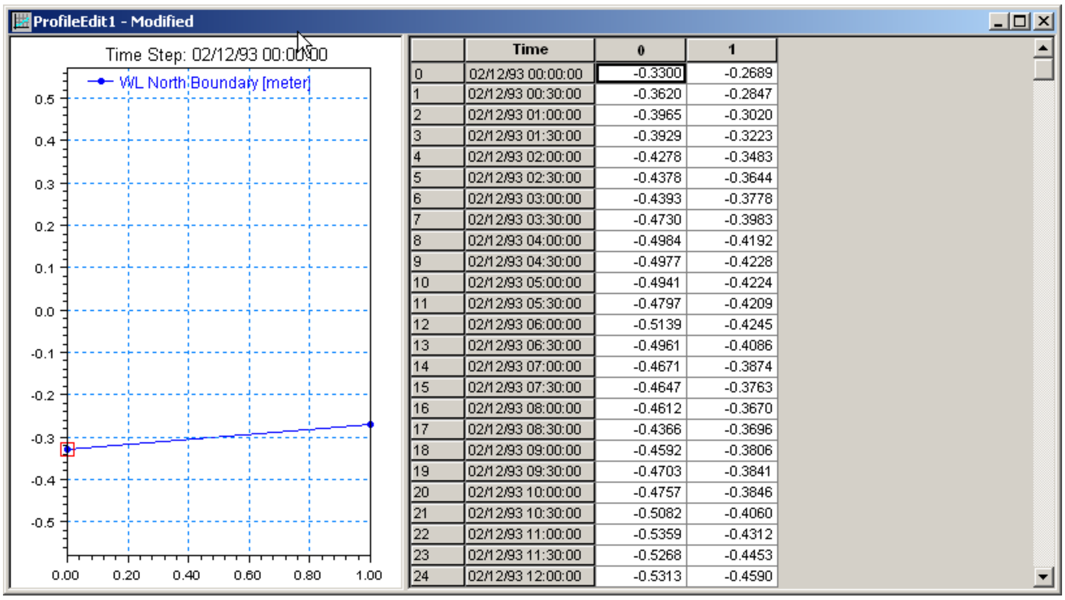
Figure 3.14 Profile Series Properties

Load Station Hornbæk (waterlevel_hornbaek.dfs0) and copy and paste the water levels to the profile Series Editor at point 0. Next load Station Viken (waterlevel_viken.dfs0) and copy and paste the levels into point 1 (see Figure 3.15). Save the profile series as waterlevel_north.dfs1 (see Figure 3.16).



	Time	0	1
0	02/12/93 00:00:00	-0.3300	
1	02/12/93 00:30:00	-0.3620	
2	02/12/93 01:00:00	-0.3965	
3	02/12/93 01:30:00	-0.3929	
4	02/12/93 02:00:00	-0.4278	
5	02/12/93 02:30:00	-0.4378	
6	02/12/93 03:00:00	-0.4393	
7	02/12/93 03:30:00	-0.4730	
8	02/12/93 04:00:00	-0.4984	
9	02/12/93 04:30:00	-0.4977	
10	02/12/93 05:00:00	-0.4941	
11	02/12/93 05:30:00	-0.4797	
12	02/12/93 06:00:00	-0.5139	
13	02/12/93 06:30:00	-0.4961	
14	02/12/93 07:00:00	-0.4671	
15	02/12/93 07:30:00	-0.4647	
16	02/12/93 08:00:00	-0.4612	
17	02/12/93 08:30:00	-0.4366	
18	02/12/93 09:00:00	-0.4592	
19	02/12/93 09:30:00	-0.4703	
20	02/12/93 10:00:00	-0.4757	
21	02/12/93 10:30:00	-0.5082	
22	02/12/93 11:00:00	-0.5359	
23	02/12/93 11:30:00	-0.5268	
24	02/12/93 12:00:00	-0.5313	

Figure 3.15 Copying water levels from Station 1 (Hornbæk) into the Profile Series Editor (Ctrl V)



	Time	0	1
0	02/12/93 00:00:00	-0.3300	-0.2689
1	02/12/93 00:30:00	-0.3620	-0.2847
2	02/12/93 01:00:00	-0.3965	-0.3020
3	02/12/93 01:30:00	-0.3929	-0.3223
4	02/12/93 02:00:00	-0.4278	-0.3483
5	02/12/93 02:30:00	-0.4378	-0.3644
6	02/12/93 03:00:00	-0.4393	-0.3778
7	02/12/93 03:30:00	-0.4730	-0.3983
8	02/12/93 04:00:00	-0.4984	-0.4192
9	02/12/93 04:30:00	-0.4977	-0.4228
10	02/12/93 05:00:00	-0.4941	-0.4224
11	02/12/93 05:30:00	-0.4797	-0.4209
12	02/12/93 06:00:00	-0.5139	-0.4245
13	02/12/93 06:30:00	-0.4961	-0.4086
14	02/12/93 07:00:00	-0.4671	-0.3874
15	02/12/93 07:30:00	-0.4647	-0.3763
16	02/12/93 08:00:00	-0.4612	-0.3670
17	02/12/93 08:30:00	-0.4366	-0.3696
18	02/12/93 09:00:00	-0.4592	-0.3806
19	02/12/93 09:30:00	-0.4703	-0.3841
20	02/12/93 10:00:00	-0.4757	-0.3846
21	02/12/93 10:30:00	-0.5082	-0.4060
22	02/12/93 11:00:00	-0.5359	-0.4312
23	02/12/93 11:30:00	-0.5268	-0.4453
24	02/12/93 12:00:00	-0.5313	-0.4590

Figure 3.16 Water level line series at the North Boundary

Repeat the same step with the southern boundary with the similar information except the grid step and using the recorded water levels at Station Rødvig (waterlevel_rodvig.dfs0) and Station Skanör (waterlevel_skanor.dfs0) and save the resulting file as waterlevel_south.dfs1.

South Boundary

- Start date 1993-12-02 00:00:00
- Time step: 1800s
- No. of time steps: 577
- No. of grid points: 2
- Grid Step: 33500m (actually not necessary, because MIKE 21 & MIKE 3 Flow Model FM interpolates the line series to the boundary nodes without respect to this distance)

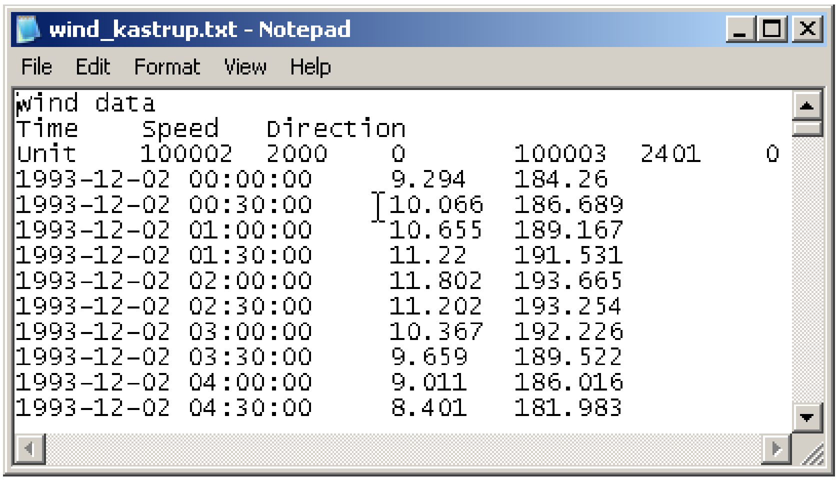
3.2 Initial Conditions

The initial surface level is calculated as a mean level between the northern and the southern boundary at the beginning of the simulation. Load the two boundary files and approximate a mean level at the start of the simulation. We will use –0.37 m.

3.3 Wind Forcing

Wind recordings from Kastrup Airport will form the wind forcing as time series constant in space. Load the time series editor and import the ASCII file 'wind_kastrup.txt' with equidistant calendar axis. Save the file in 'wind_kastrup.dfs0'. Time series of the wind speed and direction is shown in Figure 3.17.

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_kastrup.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.




```
wind data
Time      Speed    Direction
Unit      100002   2000     0       100003   2401     0
1993-12-02 00:00:00     9.294   184.26
1993-12-02 00:30:00     10.066  186.689
1993-12-02 01:00:00     10.655  189.167
1993-12-02 01:30:00     11.22   191.531
1993-12-02 02:00:00     11.802  193.665
1993-12-02 02:30:00     11.202  193.254
1993-12-02 03:00:00     10.367  192.226
1993-12-02 03:30:00     9.659   189.522
1993-12-02 04:00:00     9.011   186.016
1993-12-02 04:30:00     8.401   181.983
```

Figure 3.17 ASCII file with wind speed and direction from Kastrup Airport

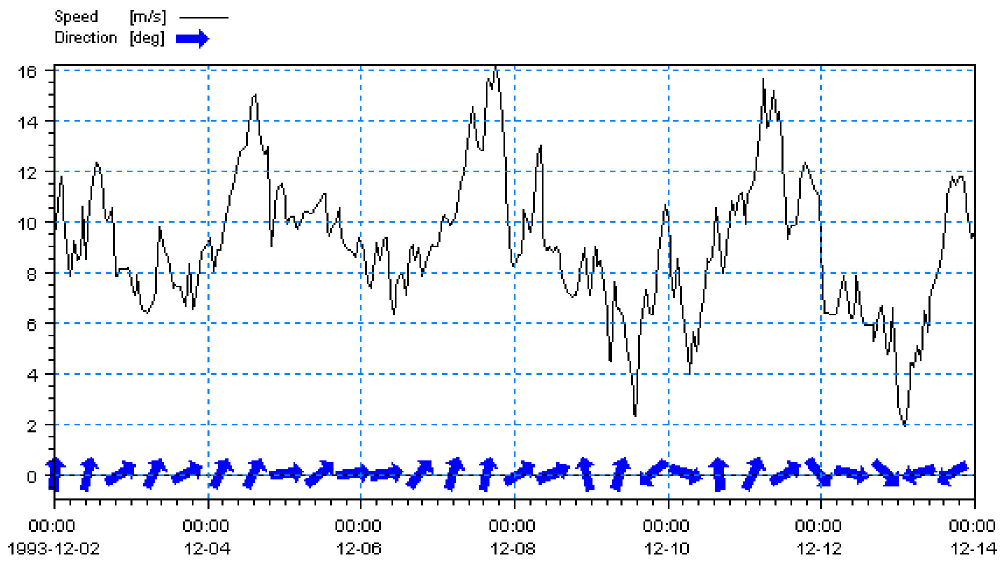


Figure 3.17 Wind speed and direction from Kastrup Airport as it can be illustrated in the Plot Composer (Time Series Direction plot control)

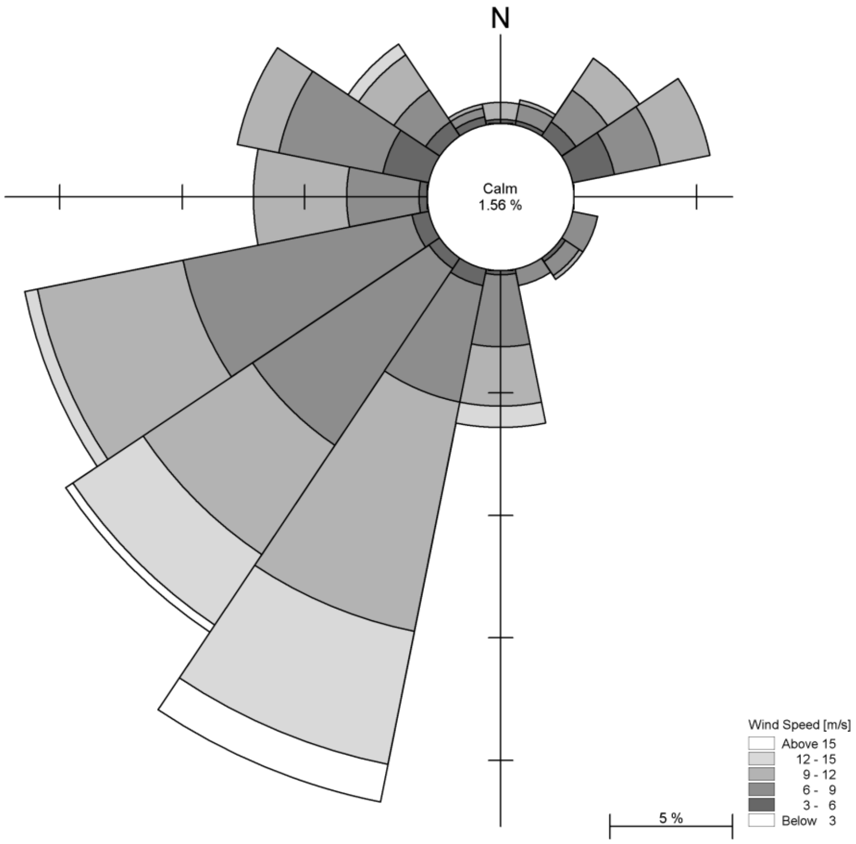


Figure 3.18 Wind rose from Kastrup Airport as it can be illustrated in the Plot Composer and South Boundary

4 Creating the Input Parameters to the MIKE 3 Flow Model FM

Before the MIKE 3 Flow Model FM can be set up, the input data must be generated from the measurements. Measurements for 1997[1] exist for

1. Water levels at the boundaries
2. Salinity at the boundaries
3. Temperature at the boundaries
4. Wind at Ven

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

4.1 Generate Boundary Conditions

4.1.1 Water levels

Measured water level recordings from four stations located near the open model boundaries are available for the Øresund model, see Figure 4.1.

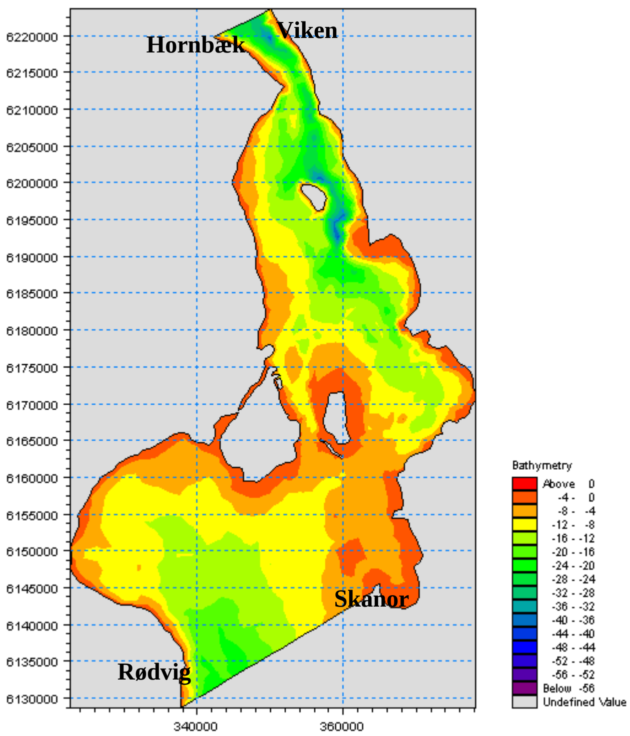


Figure 4.1 Map showing the water level stations at the boundaries: Hornbæk, Viken, Skanör, and Rødvig

[1] Measurements for 1993 were used for the MIKE 21 Flow Model example in Chapter 3

The Øresund model is forced with water level boundaries. The water level measurements indicate that the variations along the boundaries are significant, so the water level boundaries should be specified as line series (dfs1 type data file) containing an interpolation between the two measurements at each boundary.

In the following, two line series (dfs1 type data file) with water level variations will be created on basis of measured recordings from the four stations on the two boundaries. The locations of the four stations are listed in Table 2.1.

4.1.2 Importing measured water levels to time series file

Open the Time Series Editor in MIKE Zero (File→New→ File→Time Series), see Figure 4.2. Select the ASCII template. Open the text file waterlevel_rodvig.txt shown in Figure 4.3. Change the time description to 'Equidistant Calendar Axis' and press OK, see Figure 4.4. Then right click on the generated data in the Time Series Editor and select properties, see Figure 4.5. Change the item type to 'Water Level', see Figure 4.6, and finally, save the data in waterlevel_rodvig.dfs0.

Repeat these steps for the remaining three stations.

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

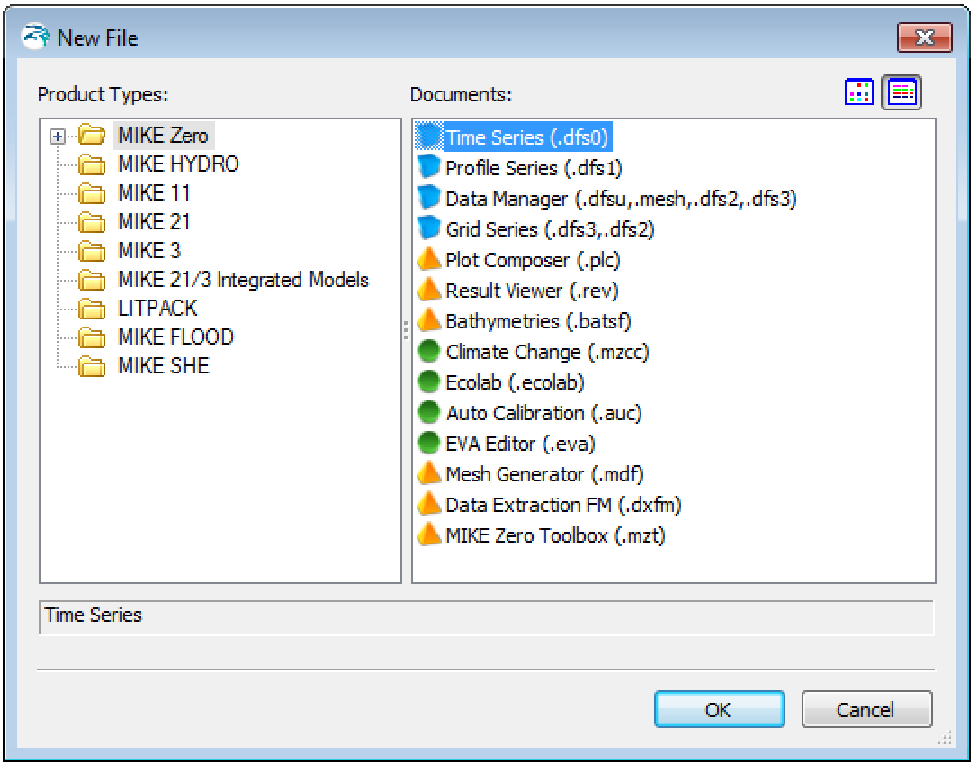


Figure 4.2 Starting the Time Series Editor in MIKE Zero

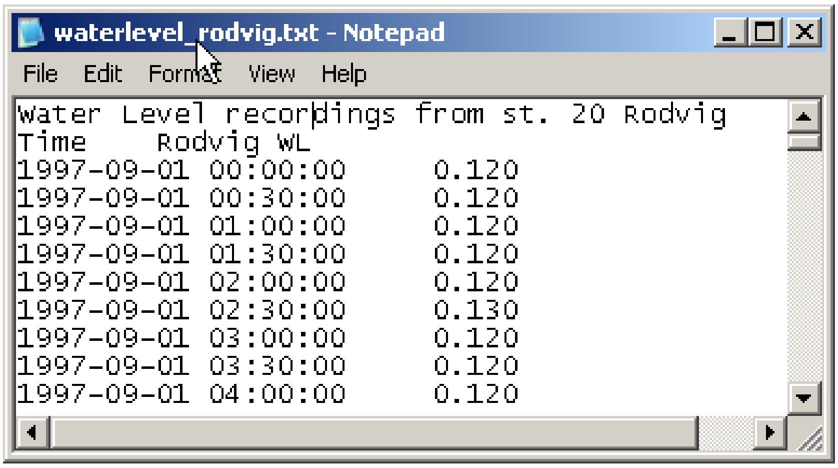


Figure 4.3 ASCII file with water level recordings from Station Rødvig

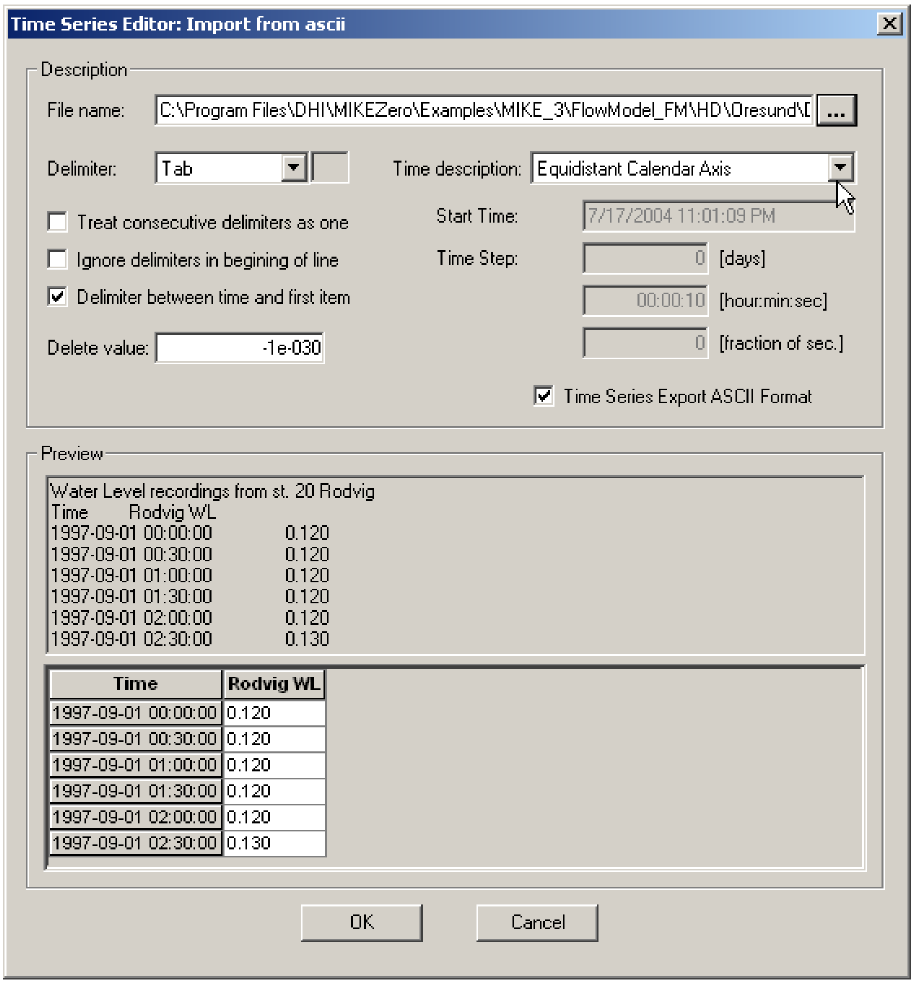


Figure 4.4 Time Series Editor: Import from ascii

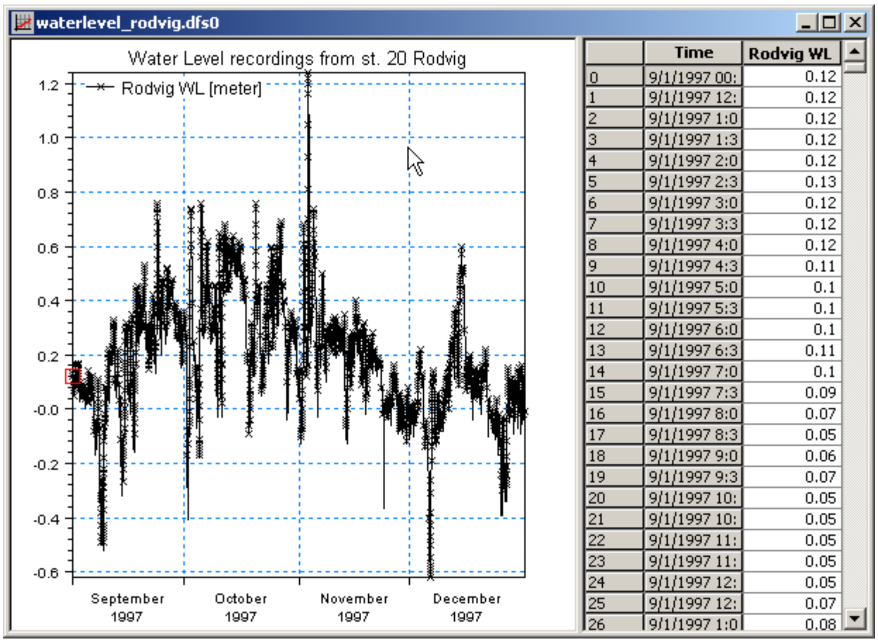


Figure 4.5 Time Series Editor with imported water levels from Station: Rødvig

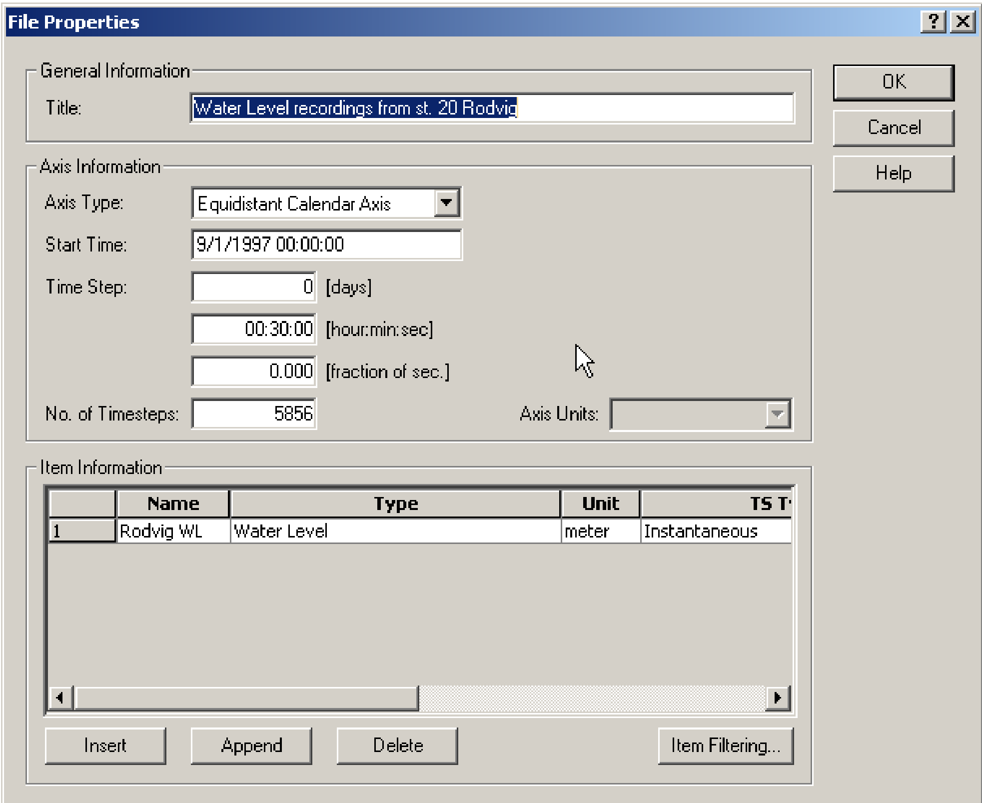


Figure 4.6 Time Series Properties

To make a plot of the water level time series open the Plot Composer in MIKE Zero, see Figure 4.7. Select 'plot' → 'insert a new plot object' and select 'Time Series Plot' (see Figure 4.8).

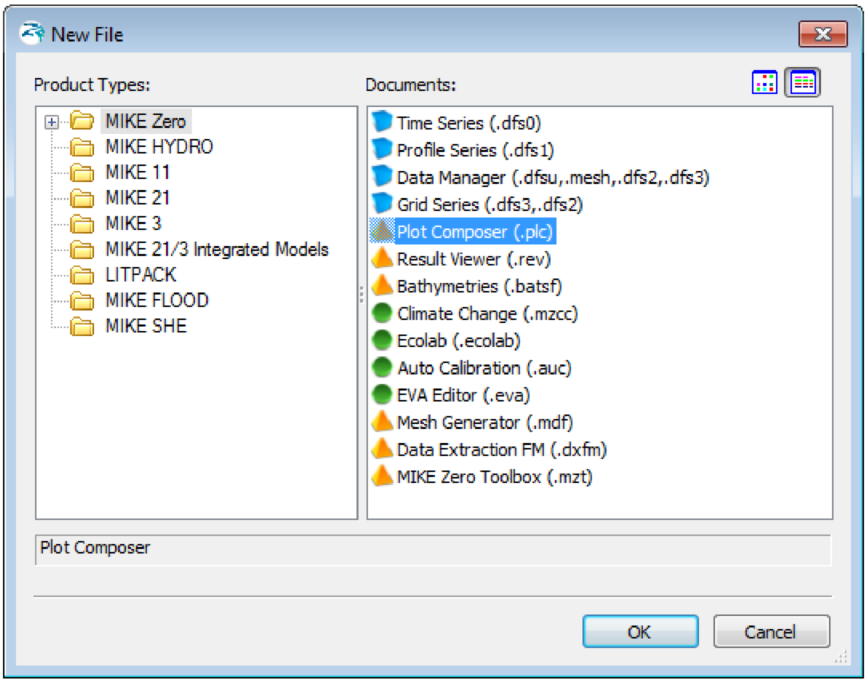


Figure 4.7 Starting the Plot Composer in MIKE Zero

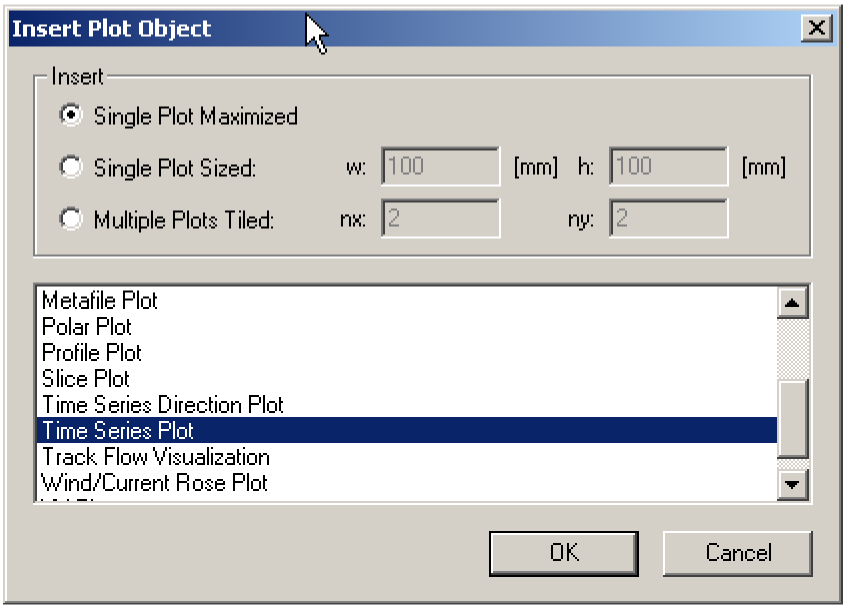


Figure 4.8 Plot Composer inserted a new Plot Object as Time Series

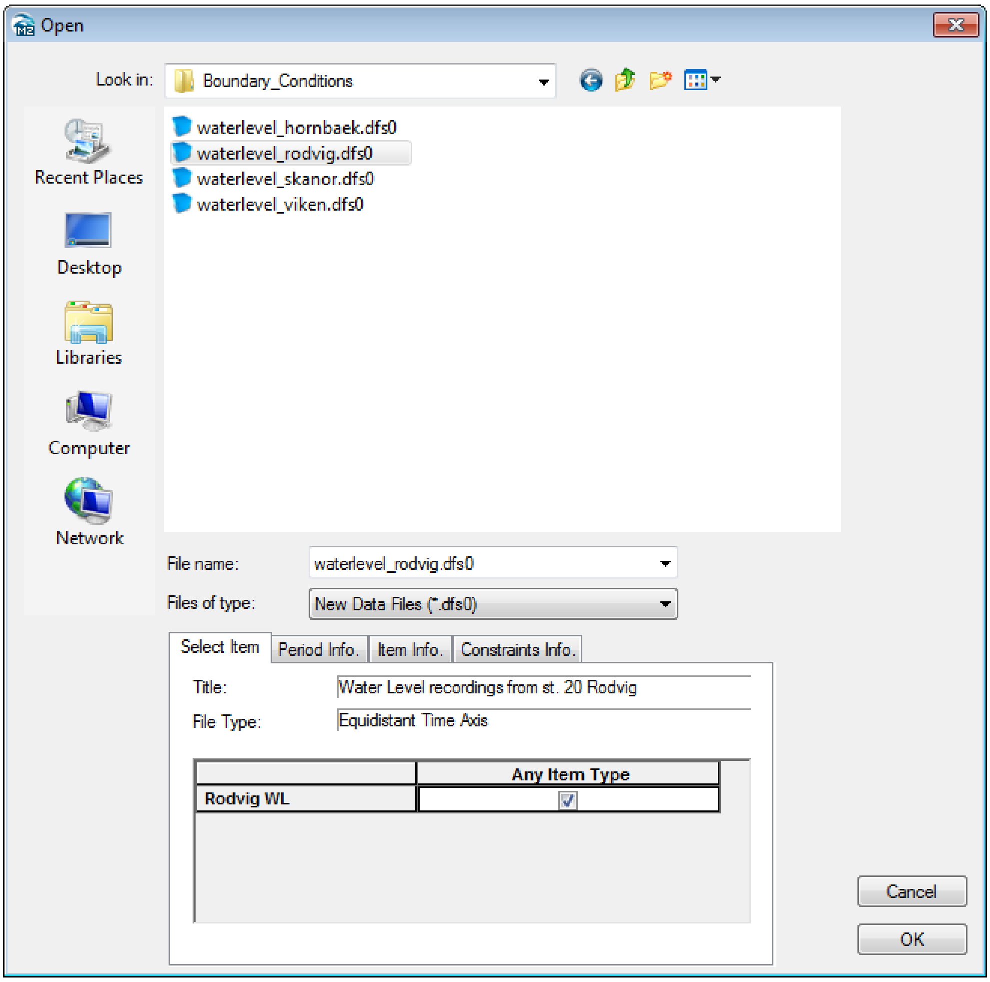
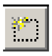


Figure 4.9 Plot Composer properties select time series to plot

Add the actual time series file to the Plot Composer by clicking  and selecting the file, see Figure 4.9. It is possible to add more than one time series to the same plot. You might also change some of the properties for the plot, such as colours, etc. (see Figure 4.10).

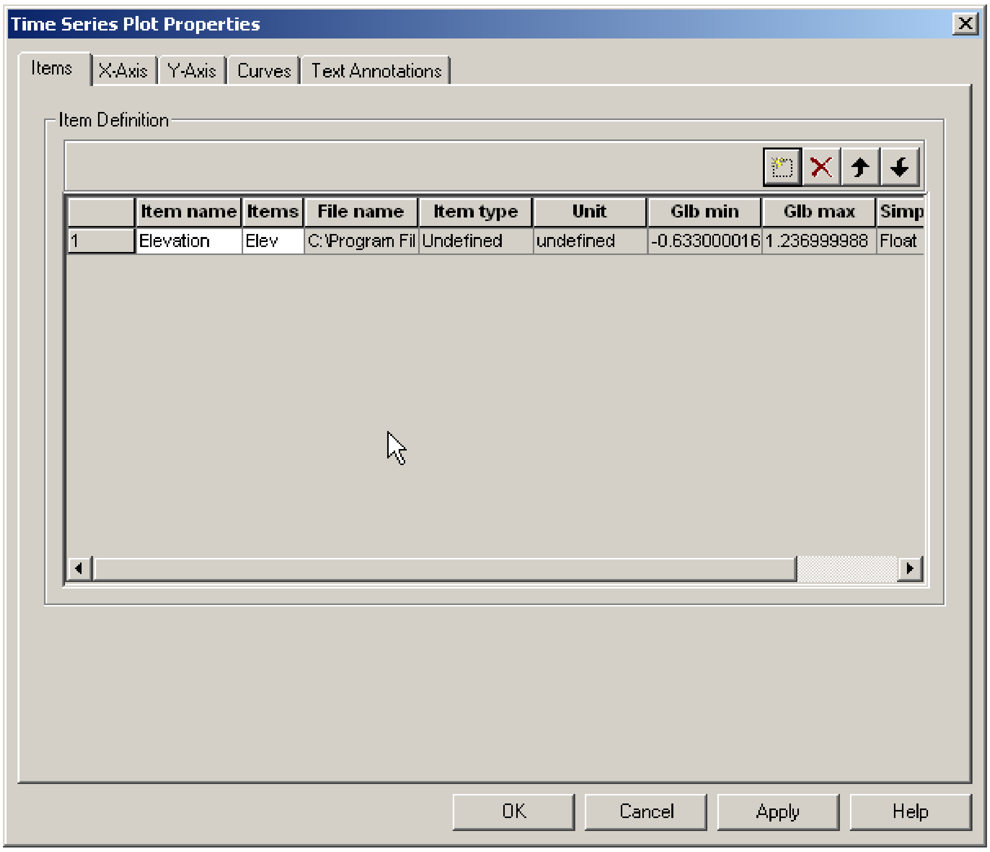


Figure 4.10 Plot Composer Time Series Plot Properties dialogue for selecting time series files and adjusting scales, curves, etc.

Figure 4.11 and Figure 4.12 show the measured water levels at the two boundaries.

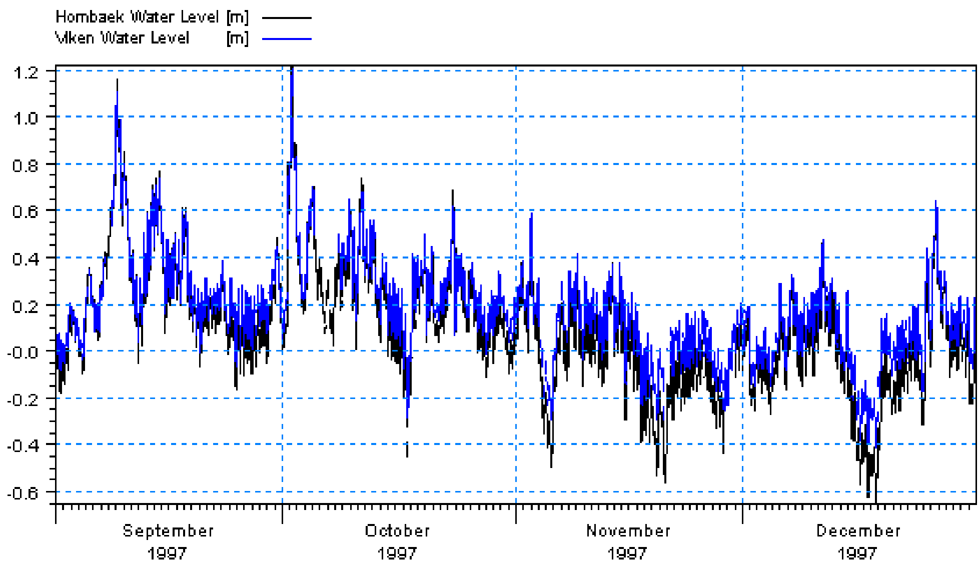


Figure 4.11 Combined Time Series at the North Boundary, Stations 1 and 2: Hornbæk and Viken

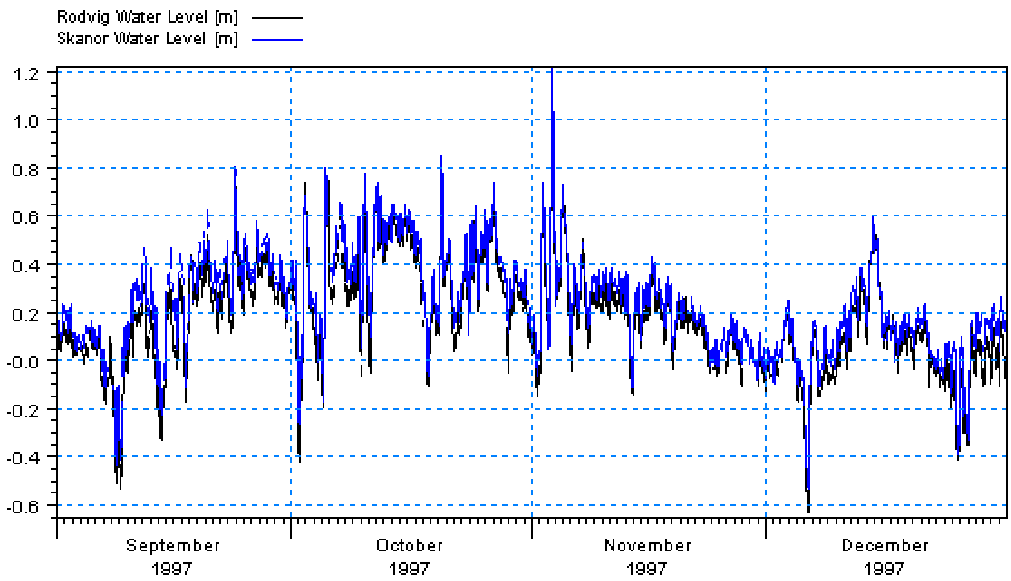


Figure 4.12 Combined Time Series at the South Boundary, Stations 3 and 4: Skanör and Rødvig

4.1.3 Creating boundary conditions

The next step is to create line series from the generated time series.

Load the Profile Series in MIKE Zero (File→New→ File→Profile Series) and select 'Blank ...'. Use default values for the geographical information in the first dialogue.

In the next dialogue, fill in the required information listed below and as shown in Figure 4.13:

North Boundary

- Start date 1997-09-01 00:00:00
- Time step: 1800s
- No. of time steps: 5856
- No. of grid points: 10
- Grid Step: 1000m (actually not necessary, because MIKE 21 & MIKE 3 Flow Model FM interpolates the line series to the boundary nodes without respect to this distance)

Load Station Viken (waterlevel_viken.dfs0) in the Time Series Editor and copy and paste the water levels to the profile Series Editor at column 0 and 1. Next load Station Hornbæk (waterlevel_hornbaek.dfs0) and copy and paste the levels into column 8 and 9 (see Figure 4.14). This way the water levels are kept constant close to the coast, where the current velocities are expected to be low due to bottom friction. Interpolate the columns 2-7 using the values in columns 1 and 8 (Tools→Interpolation).

Save the profile series as waterlevel_north.dfs1 (see Figure 4.15).

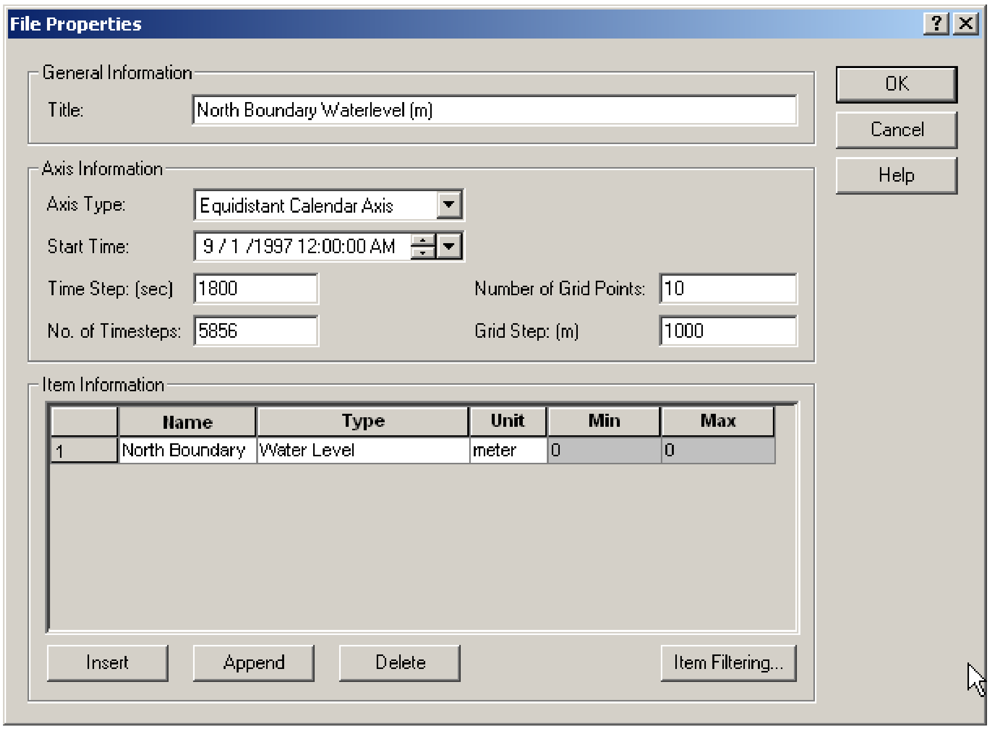


Figure 4.13 Profile Series Properties

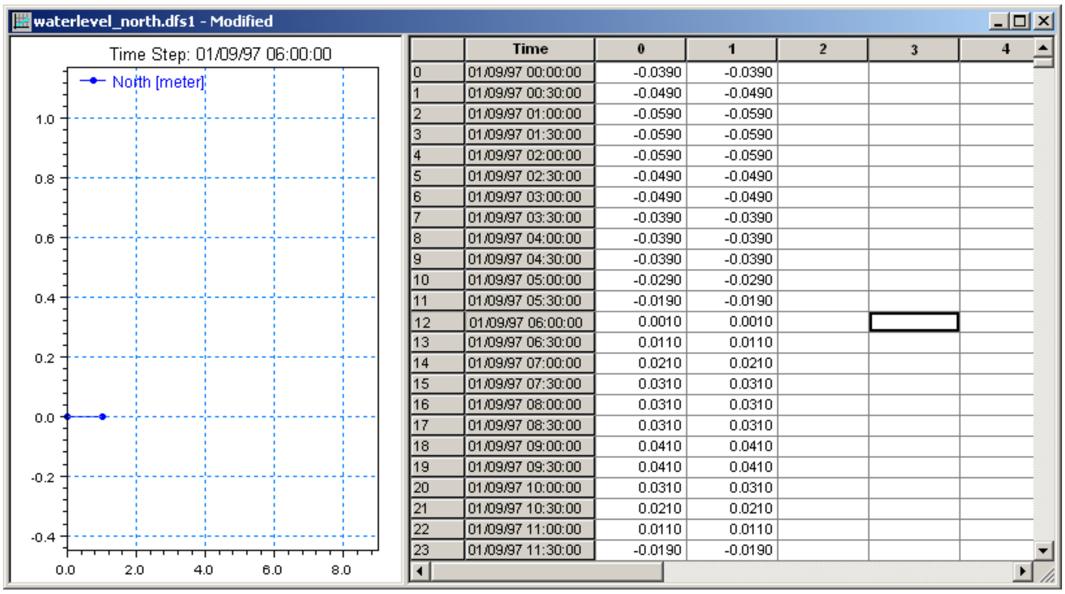
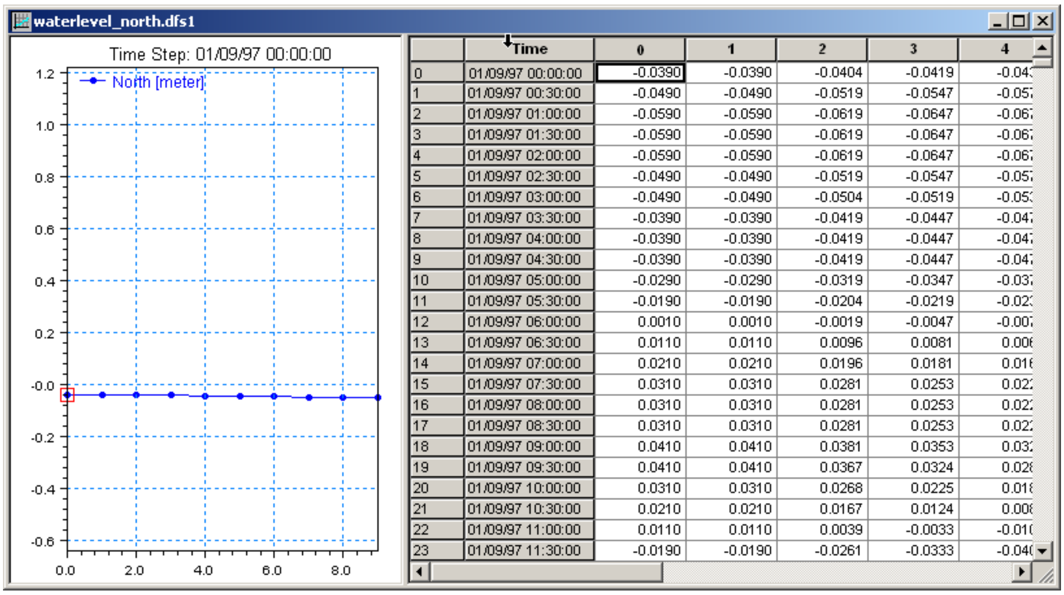


Figure 4.14 Copying Viken water levels into Profile Series Editor



	Time	0	1	2	3	4
0	01/09/97 00:00:00	-0.0390	-0.0390	-0.0404	-0.0419	-0.04
1	01/09/97 00:30:00	-0.0490	-0.0490	-0.0519	-0.0547	-0.05
2	01/09/97 01:00:00	-0.0590	-0.0590	-0.0619	-0.0647	-0.06
3	01/09/97 01:30:00	-0.0590	-0.0590	-0.0619	-0.0647	-0.06
4	01/09/97 02:00:00	-0.0590	-0.0590	-0.0619	-0.0647	-0.06
5	01/09/97 02:30:00	-0.0490	-0.0490	-0.0519	-0.0547	-0.05
6	01/09/97 03:00:00	-0.0490	-0.0490	-0.0504	-0.0519	-0.05
7	01/09/97 03:30:00	-0.0390	-0.0390	-0.0419	-0.0447	-0.04
8	01/09/97 04:00:00	-0.0390	-0.0390	-0.0419	-0.0447	-0.04
9	01/09/97 04:30:00	-0.0390	-0.0390	-0.0419	-0.0447	-0.04
10	01/09/97 05:00:00	-0.0290	-0.0290	-0.0319	-0.0347	-0.03
11	01/09/97 05:30:00	-0.0190	-0.0190	-0.0204	-0.0219	-0.02
12	01/09/97 06:00:00	0.0010	0.0010	-0.0019	-0.0047	-0.00
13	01/09/97 06:30:00	0.0110	0.0110	0.0096	0.0081	0.00
14	01/09/97 07:00:00	0.0210	0.0210	0.0196	0.0181	0.01
15	01/09/97 07:30:00	0.0310	0.0310	0.0281	0.0253	0.02
16	01/09/97 08:00:00	0.0310	0.0310	0.0281	0.0253	0.02
17	01/09/97 08:30:00	0.0310	0.0310	0.0281	0.0253	0.02
18	01/09/97 09:00:00	0.0410	0.0410	0.0381	0.0353	0.03
19	01/09/97 09:30:00	0.0410	0.0410	0.0367	0.0324	0.02
20	01/09/97 10:00:00	0.0310	0.0310	0.0268	0.0225	0.01
21	01/09/97 10:30:00	0.0210	0.0210	0.0167	0.0124	0.00
22	01/09/97 11:00:00	0.0110	0.0110	0.0039	-0.0033	-0.01
23	01/09/97 11:30:00	-0.0190	-0.0190	-0.0261	-0.0333	-0.04

Figure 4.15 Water level line series at the North Boundary

Repeat the same step with the southern boundary with a similar approach.

Load Station Rødvig (waterlevel_rodvig.dfs0) in the Time Series Editor and copy and paste the water levels to the profile Series Editor at column 0 and 1. Next load Station Skanör (waterlevel_skanor.dfs0) and copy and paste the levels into columns 16 to 19 (Because of a larger area with shallow water near the Swedish coast at the southern boundary). This way the water levels are kept constant close to the coast, where the current velocities are expected to be low due to bottom friction. Add values in columns 2-15 by interpolating the data (Tools→Interpolation). Save the resulting file as waterlevel_south.dfs1.

South Boundary

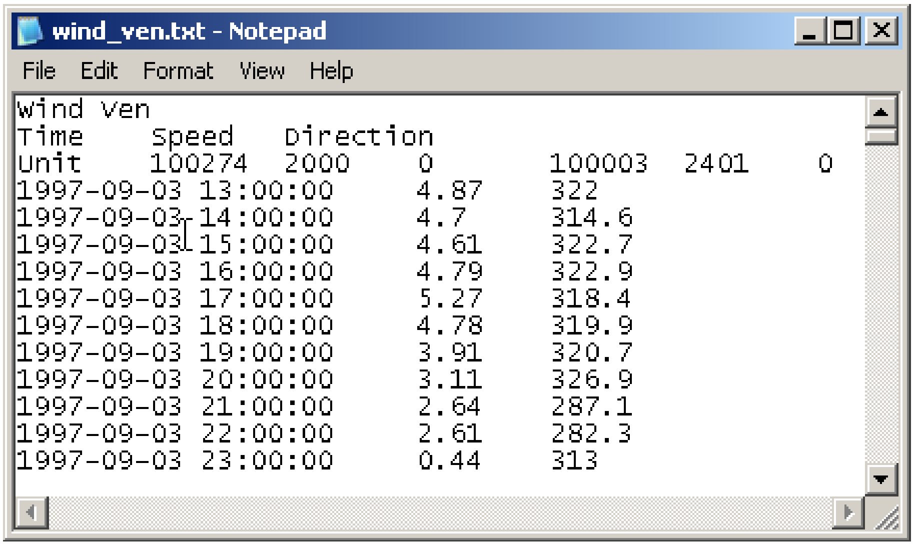
- Start date 1997-09-01 00:00:00
- Time step: 1800s
- No. of time steps: 5856
- No. of grid points: 20
- Grid Step: 1600 m (actually not necessary, because MIKE 21 & MIKE 3 Flow Model FM interpolates the line series to the boundary nodes without respect to this distance)

4.2 Initial Conditions

The initial surface level is calculated as a mean level between the northern and the southern boundary at the beginning of the simulation. Load the two boundary files and approximate a mean level at the start of the simulation. We will use 0 m.

4.3 Wind Forcing

Wind recordings from Ven Island will form the wind forcing as time series constant in space. Load the time series editor and import the ASCII file 'wind_ven.txt' as equidistant calendar axis (See Figure 4.16). Save the file in 'wind_ven.dfs0'. Time series of the wind speed and direction is shown in Figure 4.17.



```
wind_ven.txt - Notepad
File  Edit  Format  View  Help
Wind Ven
Time       Speed     Direction
Unit       100274    2000     0        100003   2401     0
1997-09-03 13:00:00        4.87     322
1997-09-03 14:00:00        4.7      314.6
1997-09-03 15:00:00        4.61     322.7
1997-09-03 16:00:00        4.79     322.9
1997-09-03 17:00:00        5.27     318.4
1997-09-03 18:00:00        4.78     319.9
1997-09-03 19:00:00        3.91     320.7
1997-09-03 20:00:00        3.11     326.9
1997-09-03 21:00:00        2.64     287.1
1997-09-03 22:00:00        2.61     282.3
1997-09-03 23:00:00        0.44     313
```

Figure 4.16 ASCII file with Wind speed and direction from Ven Island in Øresund

A more descriptive presentation of the wind can be given as a wind - speed diagram. Start the 'Plot composer' insert a new plot object select 'Wind/Current Rose Plot' and then select properties and select the newly created file 'wind_ven.dfs0' and change properties to your need. The result is shown in Figure 4.18.

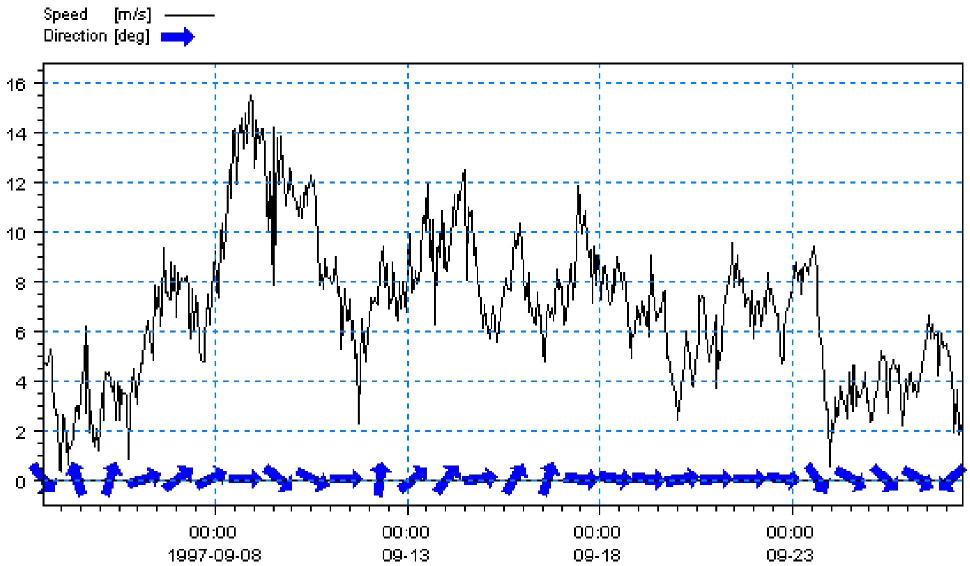


Figure 4.17 Wind Speed and Direction from Ven Island as it can be viewed in the Plot Composer

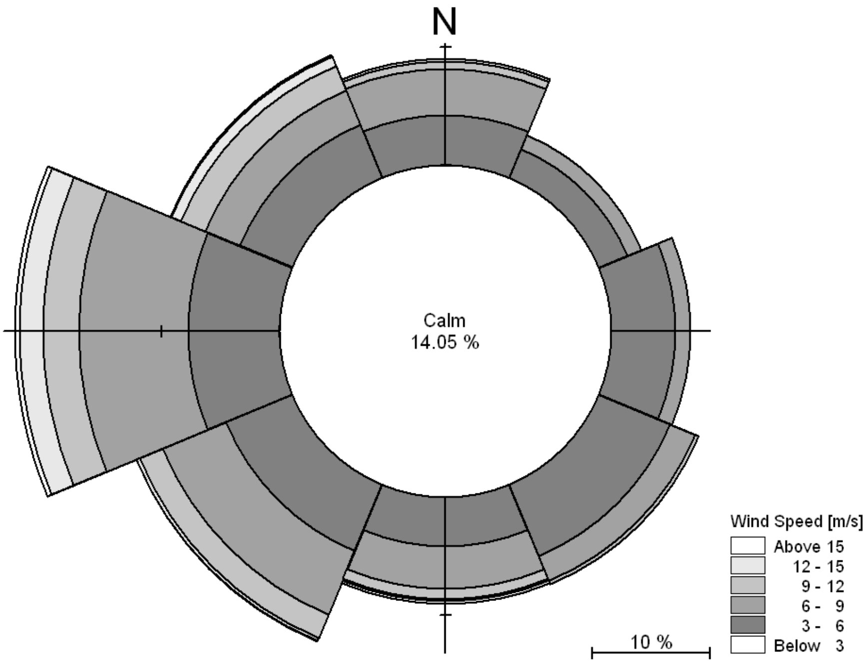


Figure 4.18 Wind Rose from Ven Island as it can be viewed in the Plot Composer

4.4 Density Variation at the Boundary

As the area of interest is dominated with outflow of fresh water from the Baltic Sea and high saline water intruding from the North Sea, measurement of salinity and temperature has taken place at the boundaries. You can use two methods to generate a grid series boundary files from the measurements. You can for instance use a spread sheet to create an ASCII file from the raw data to import in MIKE Zero. In that case you must follow the same syntacs as the ASCII file in Figure 4.19. The first 13 lines in the ASCII file constitute a header containing information about for instance:

1. Title
2. Dimension
3. UTM zone
4. Start date and time
5. No. of time steps
6. Time step in seconds
7. No. of x points and z points
8. x and z-spacing in meters
9. Name, type and unit of item in file
10. Delete value

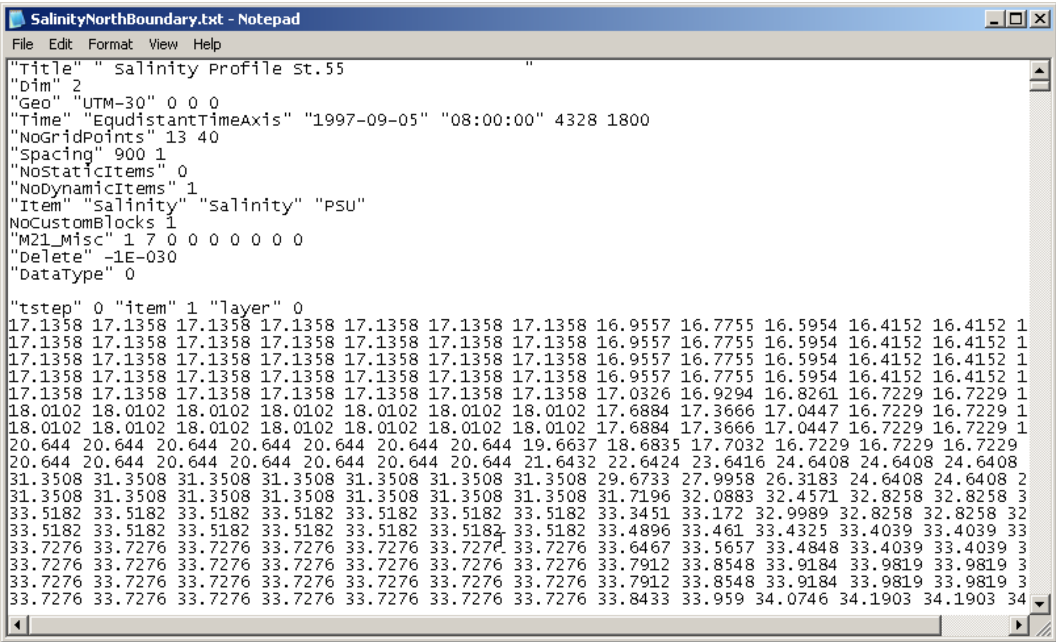
After the mandatory header, the profile measurements are grouped in an x-z matrix for each item and time step.

Another option is to enter the data manually directly in a new Grid Series file in MIKE Zero, see Figure 4.20. The choice of method depends on the amount of data, their format, and the users own experience with data processing, but both methods can be quite time consuming[2]. In this example the ASCII files to import in MIKE Zero, containing the vertical profile values of the measurements for every 900 metres along the boundary, are already made and given in the ASCII files named:

- salinity_north_boundary.txt
- salinity_south_boundary.txt
- temperature_north_boundary.txt
- temperature_south_boundary.txt

An example is shown in Figure 4.19. You should import these ASCII files with the Grid Series editor and save the files with the same name but with extension dfs2. A plot of a grid series boundary file for salinity is shown in Figure 4.20.

Note that if the boundary condition has a fluctuating and complicated spatial variation it is difficult to measure and therefore generate correct boundary conditions. In that case it is often a good strategy to extend the model area so that the boundaries are placed at positions where the boundaries are less complicated and therefore easier to measure.



```
SalinityNorthBoundary.txt - Notepad
File Edit Format View Help
"Title" " Salinity Profile St.55                    "
"Dim" 2
"Geo" "UTM-30" 0 0 0
"Time" "EqudistantTimeAxis" "1997-09-05" "08:00:00" 4328 1800
"NoGridPoints" 13 40
"Spacing" 900 1
"NoStaticItems" 0
"NoDynamicItems" 1
"Item" "Salinity" "Salinity" "PSU"
NoCustomBlocks 1
"M21_Misc" 1 7 0 0 0 0 0 0 0
"Delete" -1E-030
"DataType" 0

"tstep" 0 "item" 1 "layer" 0
17.1358 17.1358 17.1358 17.1358 17.1358 17.1358 17.1358 16.9557 16.7755 16.5954 16.4152 16.4152 1
17.1358 17.1358 17.1358 17.1358 17.1358 17.1358 17.1358 16.9557 16.7755 16.5954 16.4152 16.4152 1
17.1358 17.1358 17.1358 17.1358 17.1358 17.1358 17.1358 16.9557 16.7755 16.5954 16.4152 16.4152 1
17.1358 17.1358 17.1358 17.1358 17.1358 17.1358 17.1358 16.9557 16.7755 16.5954 16.4152 16.4152 1
17.1358 17.1358 17.1358 17.1358 17.1358 17.1358 17.1358 17.0326 16.9294 16.8261 16.7229 16.7229 1
18.0102 18.0102 18.0102 18.0102 18.0102 18.0102 18.0102 17.6884 17.3666 17.0447 16.7229 16.7229 1
18.0102 18.0102 18.0102 18.0102 18.0102 18.0102 18.0102 17.6884 17.3666 17.0447 16.7229 16.7229 1
20.644 20.644 20.644 20.644 20.644 20.644 20.644 19.6637 18.6835 17.7032 16.7229 16.7229 16.7229
20.644 20.644 20.644 20.644 20.644 20.644 20.644 21.6432 22.6424 23.6416 24.6408 24.6408 24.6408
31.3508 31.3508 31.3508 31.3508 31.3508 31.3508 31.3508 29.6733 27.9958 26.3183 24.6408 24.6408 2
31.3508 31.3508 31.3508 31.3508 31.3508 31.3508 31.3508 31.7196 32.0883 32.4571 32.8258 32.8258 3
33.5182 33.5182 33.5182 33.5182 33.5182 33.5182 33.5182 33.3451 33.172 32.9989 32.8258 32.8258 32
33.5182 33.5182 33.5182 33.5182 33.5182 33.5182 33.5182 33.4896 33.461 33.4325 33.4039 33.4039 33
33.7276 33.7276 33.7276 33.7276 33.7276 33.7276 33.7276 33.6467 33.5657 33.4848 33.4039 33.4039 3
33.7276 33.7276 33.7276 33.7276 33.7276 33.7276 33.7276 33.7912 33.8548 33.9184 33.9819 33.9819 3
33.7276 33.7276 33.7276 33.7276 33.7276 33.7276 33.7276 33.7912 33.8548 33.9184 33.9819 33.9819 3
33.7276 33.7276 33.7276 33.7276 33.7276 33.7276 33.7276 33.8433 33.959 34.0746 34.1903 34.1903 34
```

Figure 4.19 Measured transect of temperature at the North Boundary

[2] DHI's MATLAB DFS function (see http://www.mikepoweredbydhi.com/download/mike-by-dhi-tools may be another option

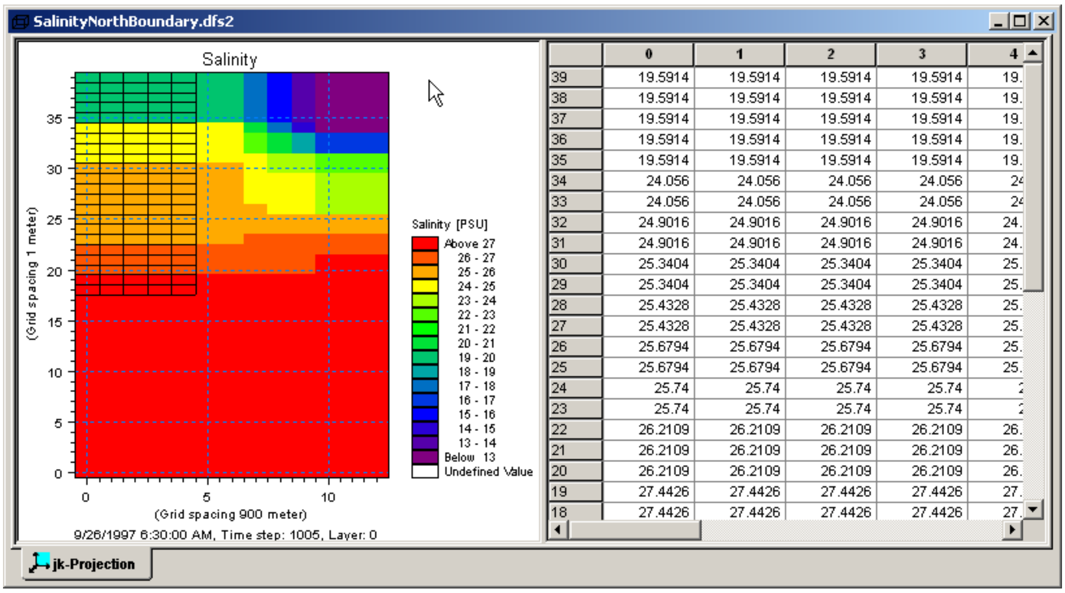


Figure 4.20 Grid Series Boundary file of salinity at the North Boundary

5 Set-Up of MIKE 21 Flow Model FM

5.1 Flow Model

We are now ready to set up the MIKE 21 Flow Model FM[3] model using the Øresund mesh with 1993 water depths, and boundary conditions and forcing 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 5.1.

Table 5.1 Specifications for the calibration simulation

Parameter	Value
Specification File	oresund.m21fm
Mesh and Bathymetry	oresund.mesh (1993) 2057 Nodes in file
Simulation Period	1993-12-02 00:00 – 1993-12-13 00:00 (11 days)
Time Step Interval	120 s
No. of Time Steps	7920
Solution Technique	Low order, fast algorithm Minimum time step: 0.01 s Maximum time step: 120 s Critical CFL number: 0.8
Enable Flood and Dry	Drying depth 0.01 m Flooding depth 0.05 m Wetting depth 0.1 m
Initial Surface Level	-0.37 m
Wind	Varying in time, constant in domain: wind_kastrup.dfs0
Wind Friction	Varying with wind speed: 0.001255 at 7 m/s 0.002425 at 25 m/s
North Boundary	Type 1 data: waterlevel_north.dfs1
South Boundary	Type 1 data: waterlevel_south.dfs1
Eddy Viscosity	Smagorinsky formulation, Constant 0.28
Resistance	Manning number. Constant value 32 $m^{1/3}/s$
Result Files	flow.dfsu ndr_roese.dfs0
CPU Simulation Time	About 25 minutes with a 2.4 GHz PC, 512 MB DDR RAM

3 The MIKE 21 Flow Model FM, Hydrodynamic Module, User Guide can be accessed via the MIKE Zero Documentation Index in the start menu

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

The dialogue for setting up the MIKE 21 Flow Model FM is initiated from MIKE Zero, see Figure 5.1 (File→New→ File→MIKE 21→Flow Model FM).

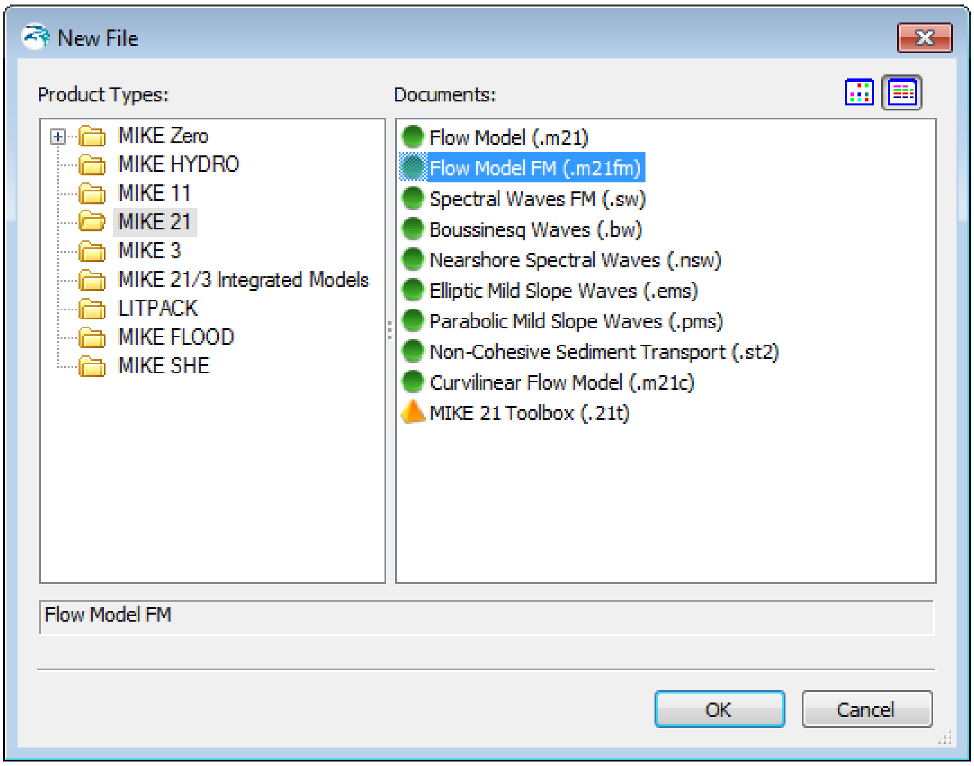


Figure 5.1 Starting MIKE 21 Flow Model FM in MIKE Zero

Specify the bathymetry and mesh file oresund.mesh in the Domain dialogue, see Figure 5.2. A graphical view of the computational mesh will appear. The projection zone has already been specified in the mesh as UTM-33. In the domain file each boundary has been given a code. In this Øresund example the North Boundary has the code 2 and the South Boundary has the code 3. Rename the boundary 'Code 2' to 'North' and 'Code 3' to 'South' in the 'Boundaries' window in the Domain dialogue, see Figure 5.3.

Specify an overall time step of 120 s in the Time dialogue. The time step range must be specified to 7920 time steps in order to simulate a total period of 11 days. See Figure 5.4.

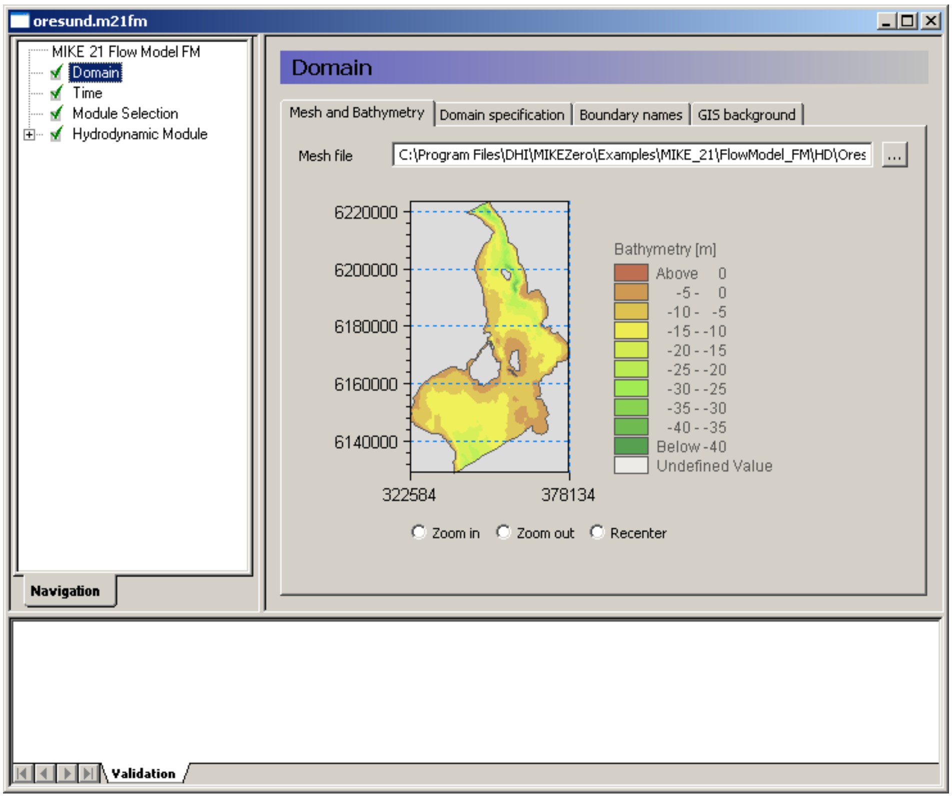


Figure 5.2 MIKE 21 Flow Model FM: Specify Domain

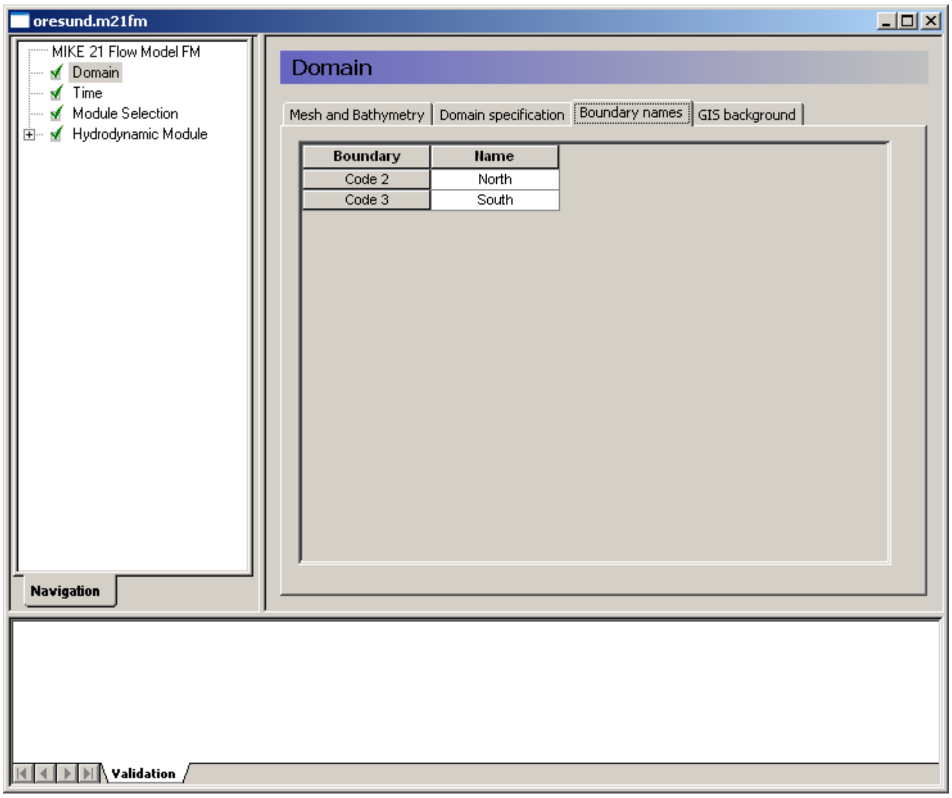


Figure 5.3 MIKE 21 Flow Model FM: Rename boundary codes to descriptive names in the Domain dialogue

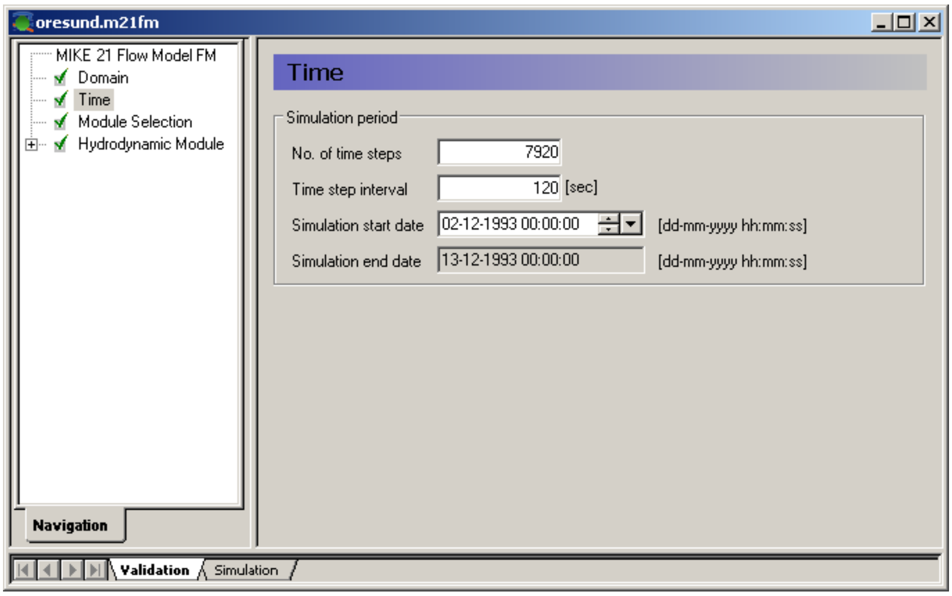


Figure 5.4 MIKE 21 Flow Model FM: Simulation period

In the Module Selection dialogue it is possible to include the 'Transport Module', the environmental 'ECO Lab Module', the 'Mud Transport Module', the 'Particle Tracking Module' and the 'Sand Transport Module'. Furthermore, it is possible to select 'Inland Flooding', see Figure 5.5.

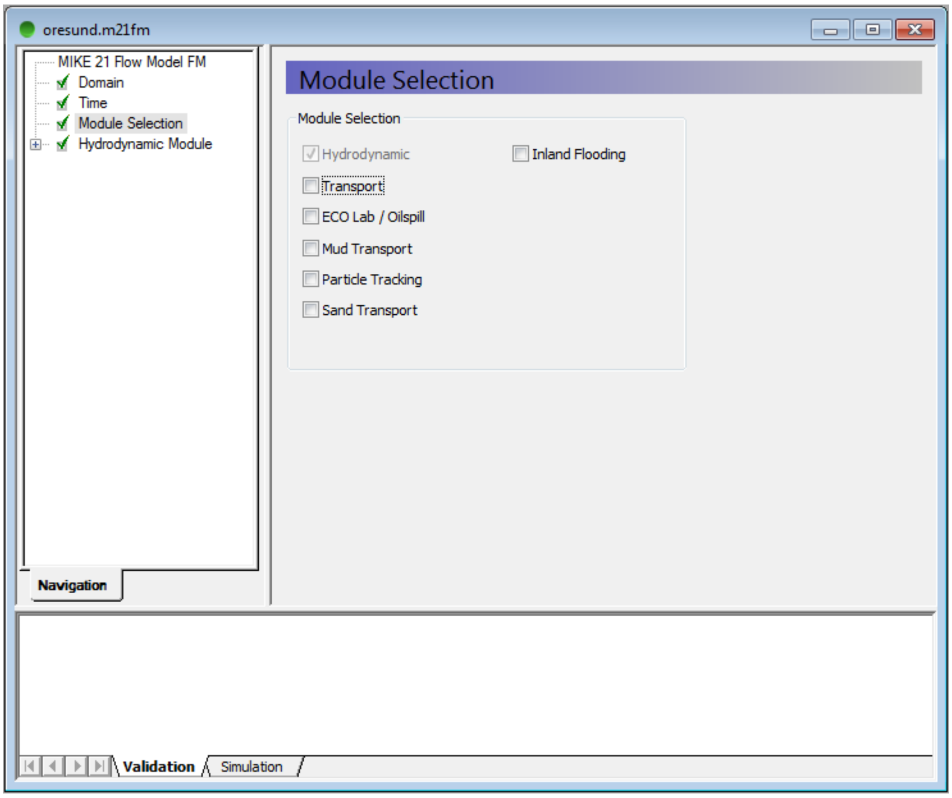


Figure 5.5 MIKE 21 Flow Model FM: Module Selection

In this example, only the Hydrodynamic Module will be used. Please see the Step-by-Step guides for the other Modules if you want to extend your hydrodynamic model with any of these modules.

In the Solution Technique dialogue set the minimum and maximum time step to 0.01 and 120 s, respectively. The critical CFL number is set to 0.8 to ensure stability of the numerical scheme throughout the simulation.

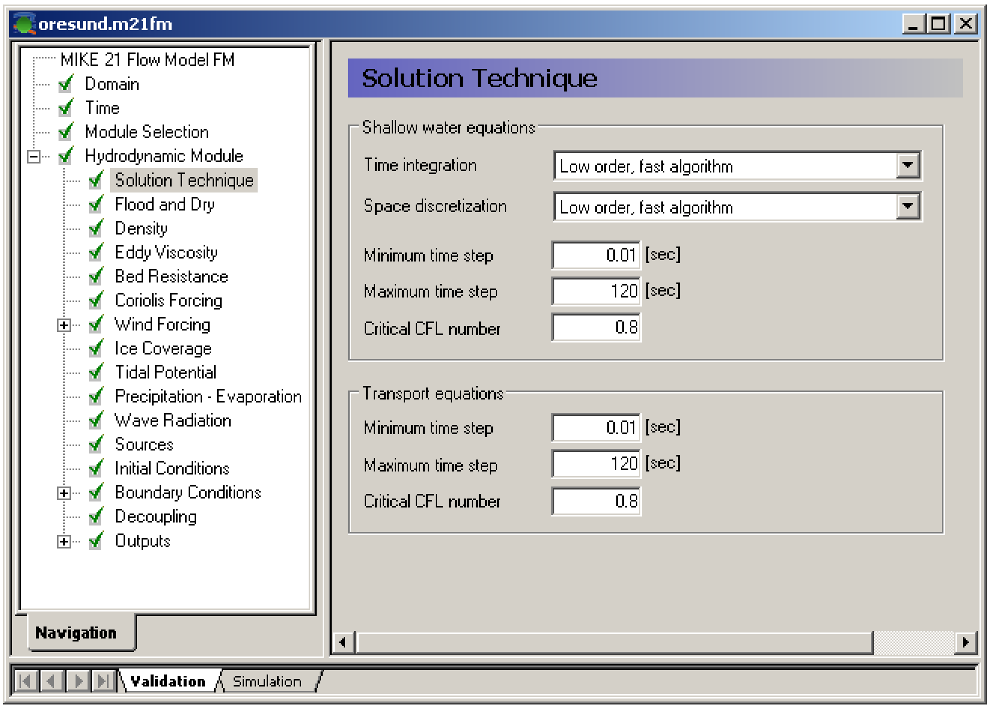


Figure 5.6 MIKE 21 Flow Model FM: Solution Technique

In the Depth dialogue it is possible to include a spatially varying depth correction. In this example, the bathymetry is kept as constant.

In the Flood and Dry t is possible to include flood and dry, see Figure 5.7. In our case select a Drying depth of 0.01 m and a Flooding depth of 0.05 m. The Wetting depth should be 0.1 m.

In this example the flooding and drying in the model should be included, because some areas along the shores of Saltholm 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. Including flooding and drying can however influence stability of the model, so if the areas that dry out are not important for the model study, you might consider not including flooding and drying. In that case you should manipulate the mesh file and make greater depths in the shallow areas to prevent those areas from drying out, and then run the model without flooding and drying.

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 5.8.

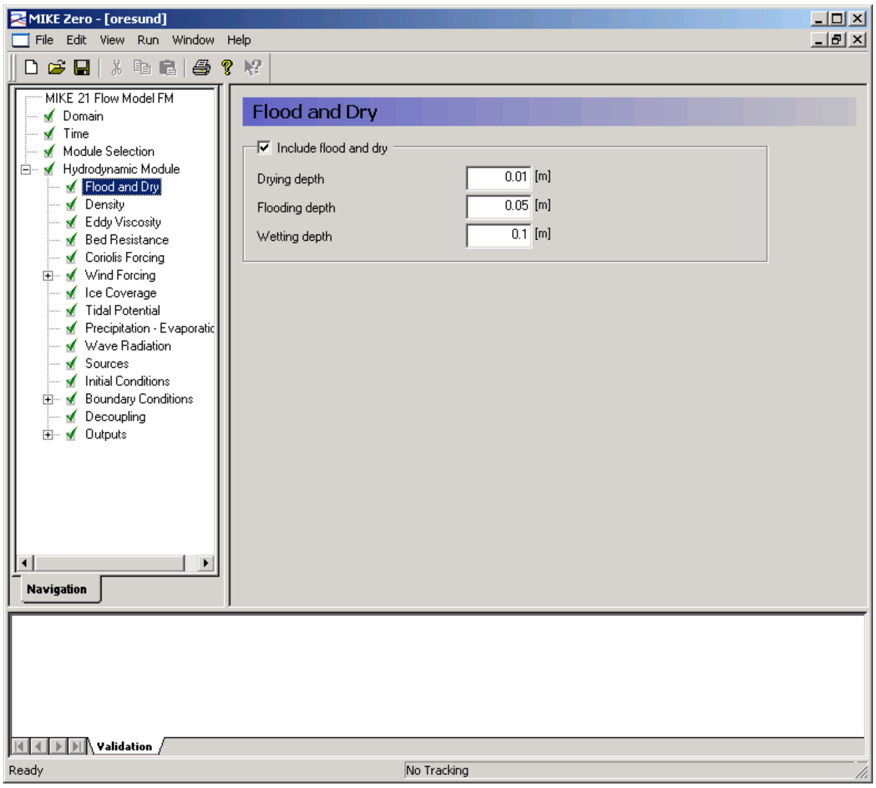


Figure 5.7 MIKE 21 Flow Model FM: Flood and Dry

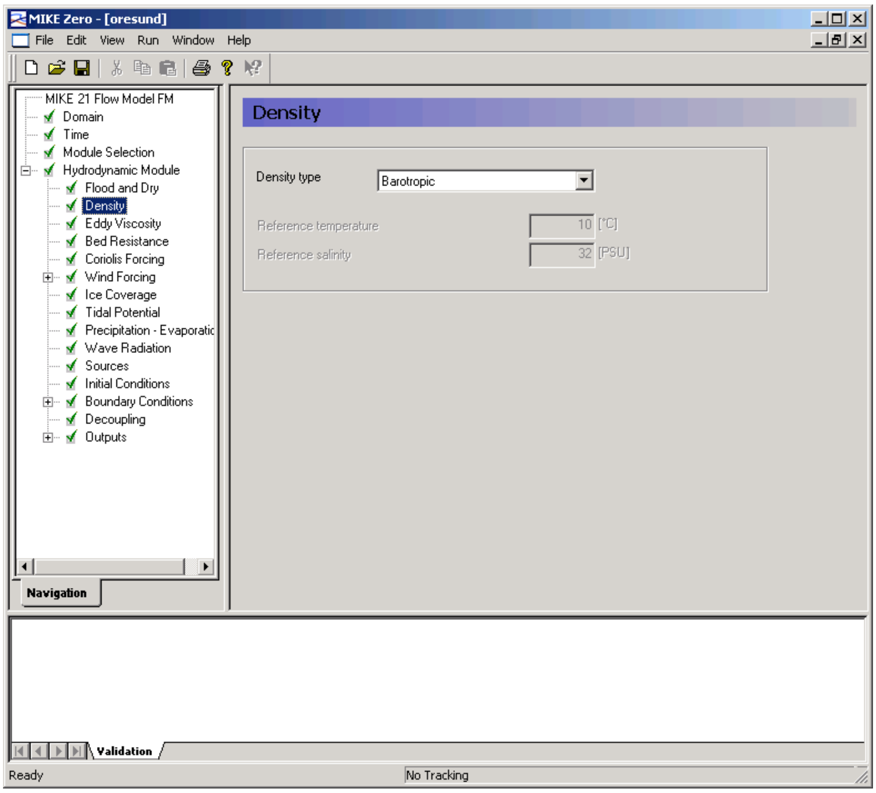


Figure 5.8 MIKE 21 Flow Model FM: Density specification

The default setting for the Horizontal Eddy Viscosity is a Smagorinsky formulation with a coefficient of 0.28, see Figure 5.9.

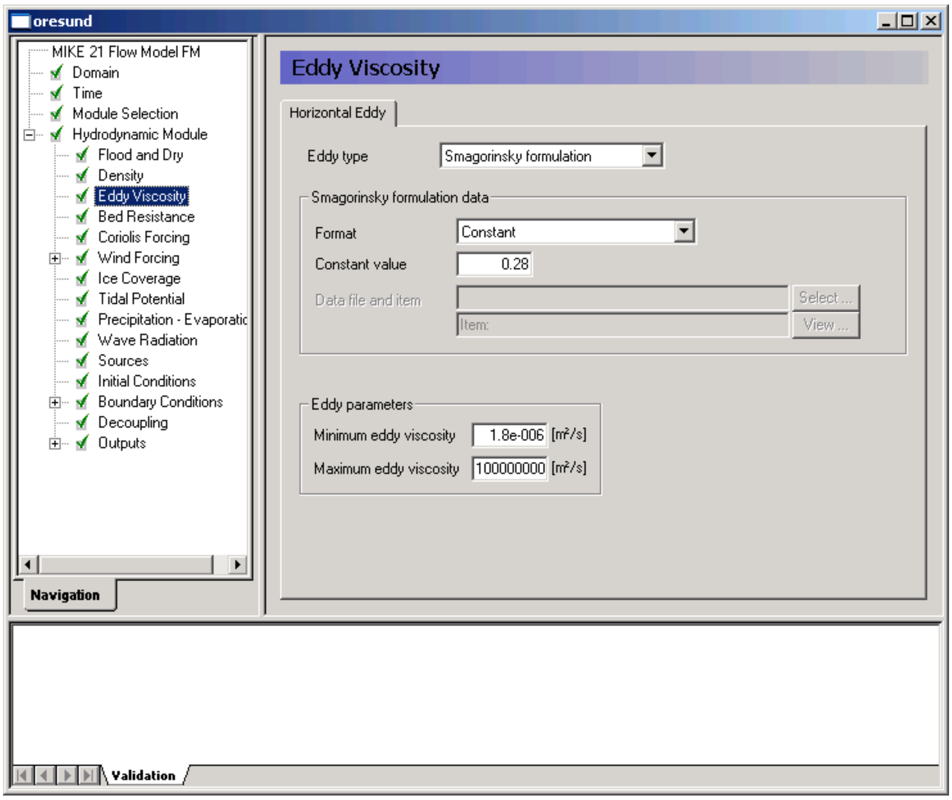


Figure 5.9 MIKE 21 Flow Model FM: Eddy Viscosity

The default Bed Resistance with a value given as a Manning number of 32 $m^{1/3}/s$ will be used for the first calibration simulation. In later calibration simulations this value can be changed, see Figure 5.10.

Even though there are often strong currents in Øresund, the effect of Coriolis forces is not so significant, because the strait is rather narrow.

However, Coriolis is always included in real applications. Only for laboratory type of simulations Coriolis is sometimes not included, see Figure 5.11.

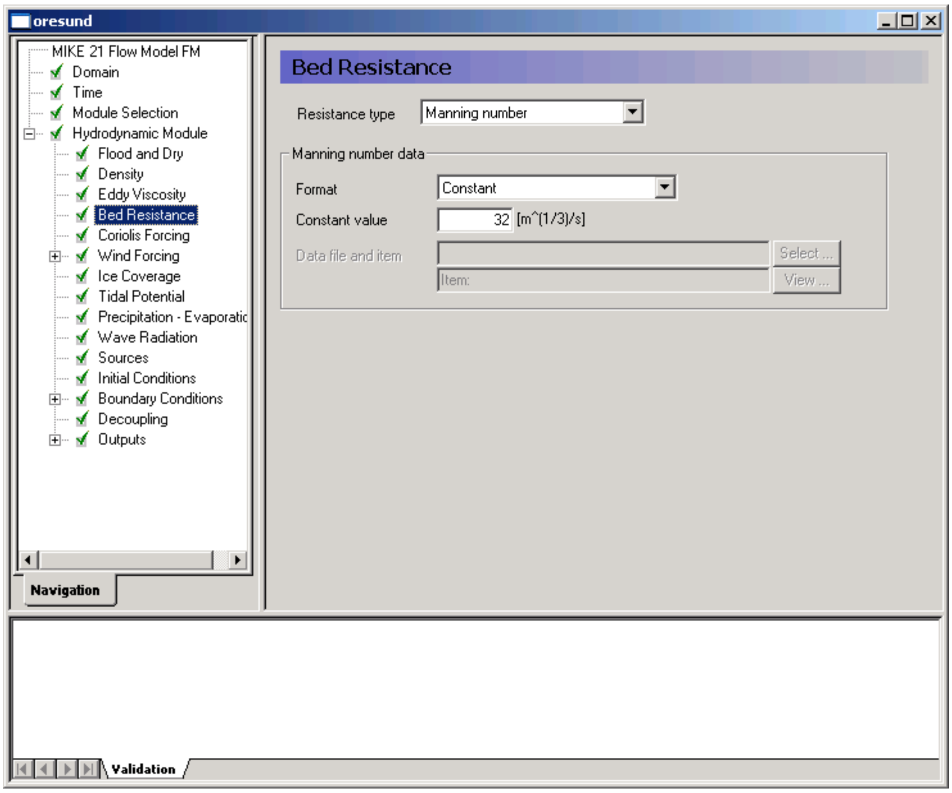


Figure 5.10 MIKE 21 Flow Model FM: Bed Resistance

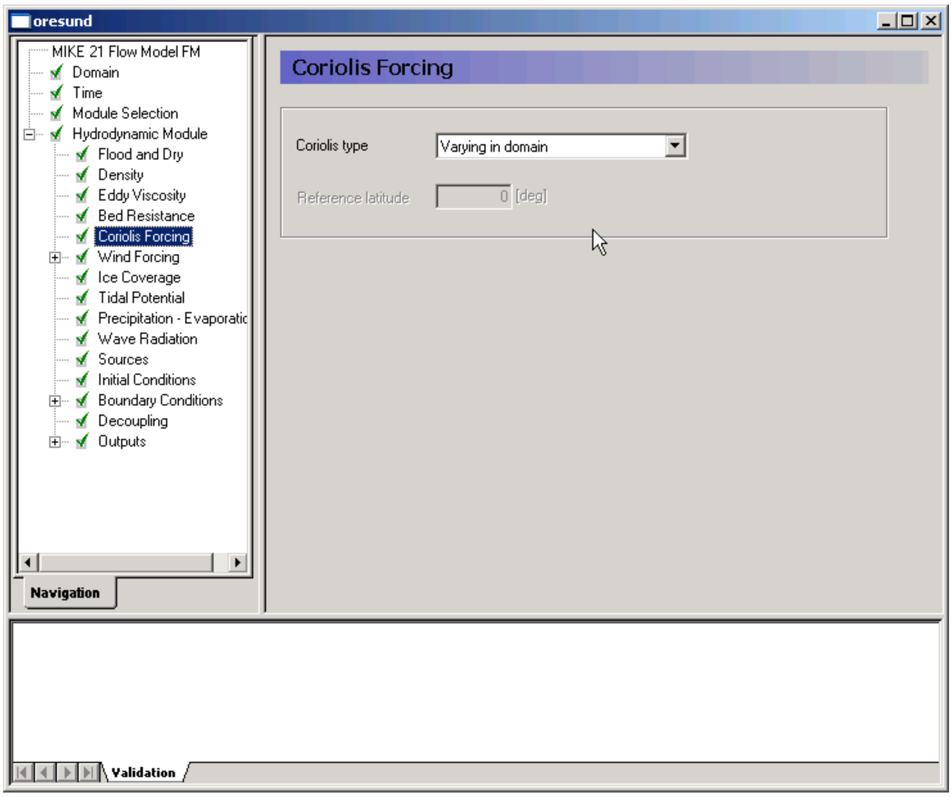


Figure 5.11 MIKE 21 Flow Model FM: Coriolis Forcing

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_kastrup.dfs0. It is a good practice to use a soft start interval. In our case 7200 s should be specified. The soft start interval is a period in the beginning of a simulation where the effect of the wind does not take full effect. In the beginning of the soft start interval 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 as 'Varying with Wind Speed' and use the default values for the Wind friction. See. Figure 5.13.

Note that an easy way to see the wind data file is to simply click View ... in the Wind Forcing dialogue, see Figure 5.12.

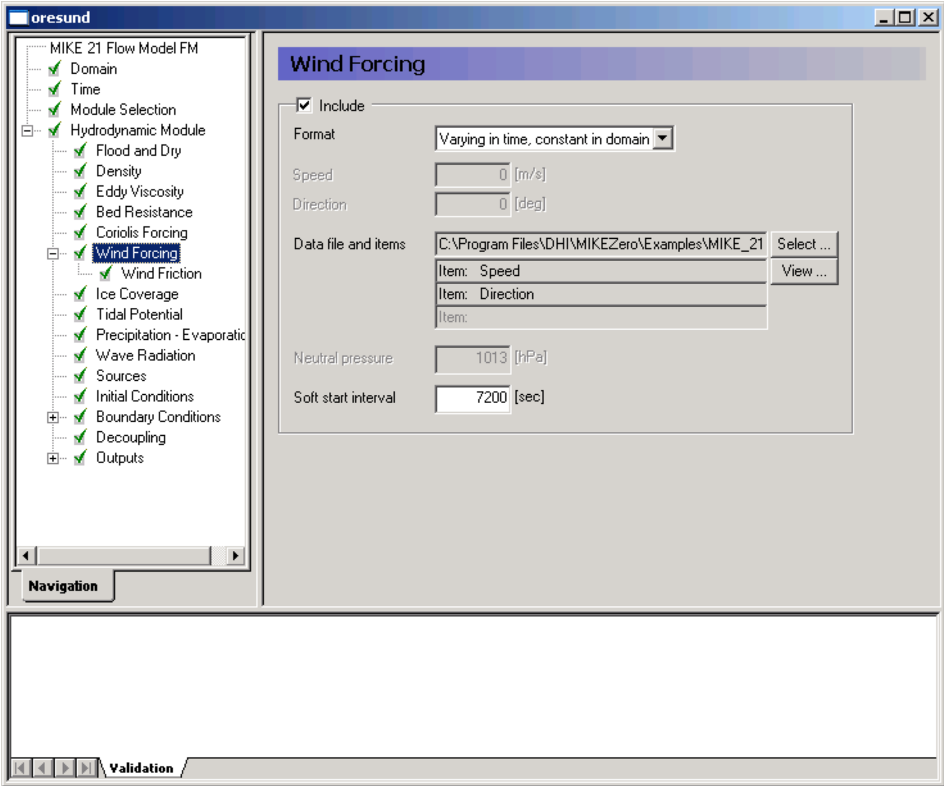


Figure 5.12 MIKE 21 Flow Model FM: Wind Forcing

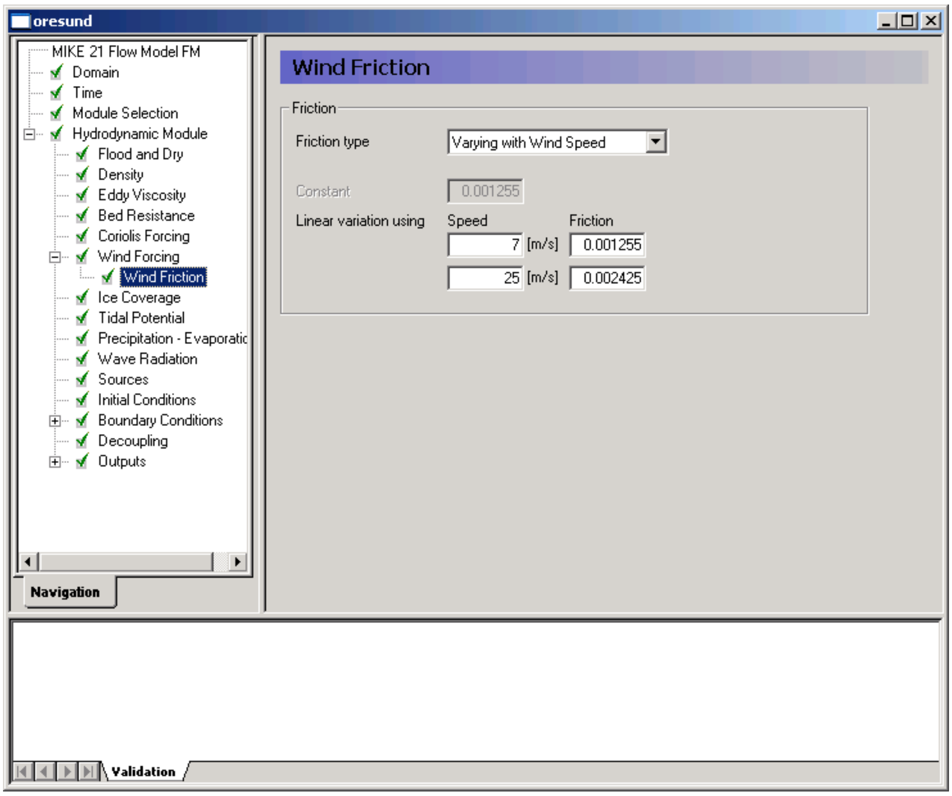


Figure 5.13 MIKE 21 Flow Model FM: Wind Friction

In this example

- Ice Coverage is not included
- Tidal Potential is not included
- Precipitation-Evaporation is not included
- Infiltration is not included
- Wave Radiation is not included
- Structures are not included
- Decoupling is not included

The discharge magnitude and velocity for each source and sink should be specified in the Sources dialogue. But, because the sources in Øresund are too small to have significant influence on the hydrodynamics in Øresund they are not included in this example. So since we do not have any sources, leave it blank. See Figure 5.14.

After inspection of the boundary conditions at the simulation start time decide the initial surface level. In this case we will use a constant level of -0.37m, which is the average between our North and South Boundary at the start of the simulation, see Figure 5.15.

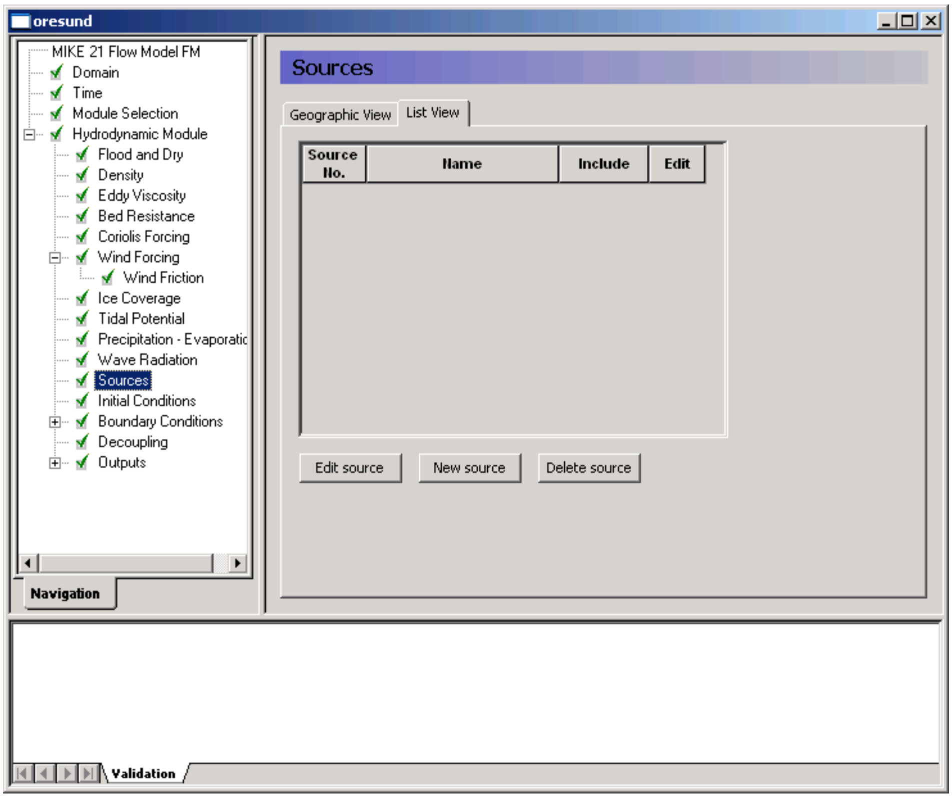


Figure 5.14 MIKE 21 Flow Model FM: Sources

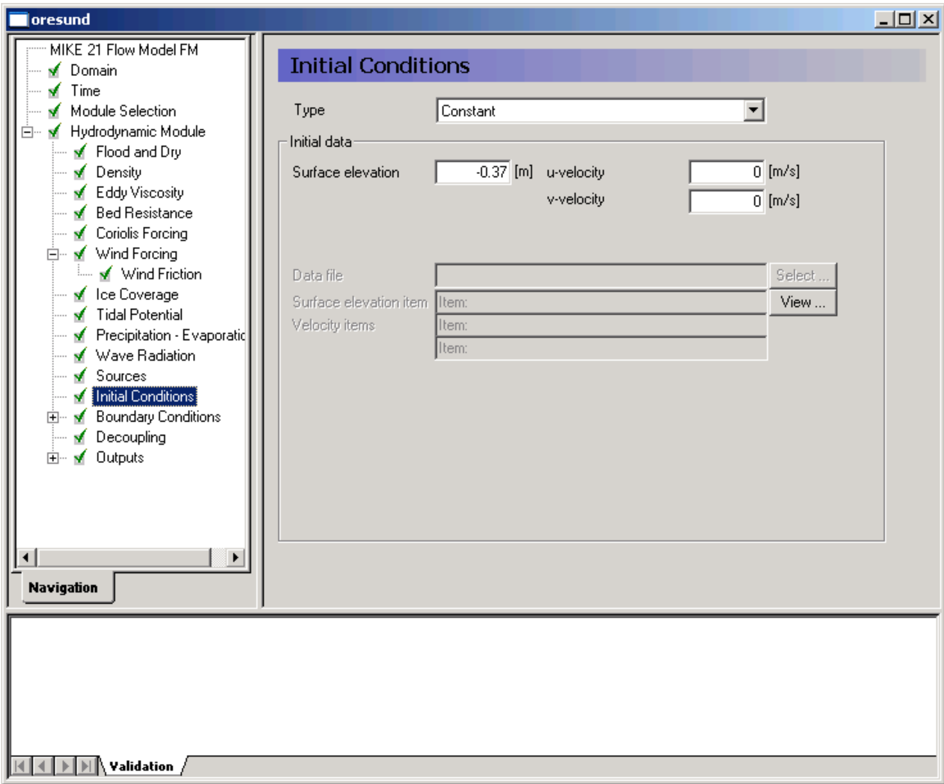


Figure 5.15 MIKE 21 Flow Model FM: Initial Conditions

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 South Boundary and the line series that were generated in Chapter 3 should be used. See Figure 5.16.

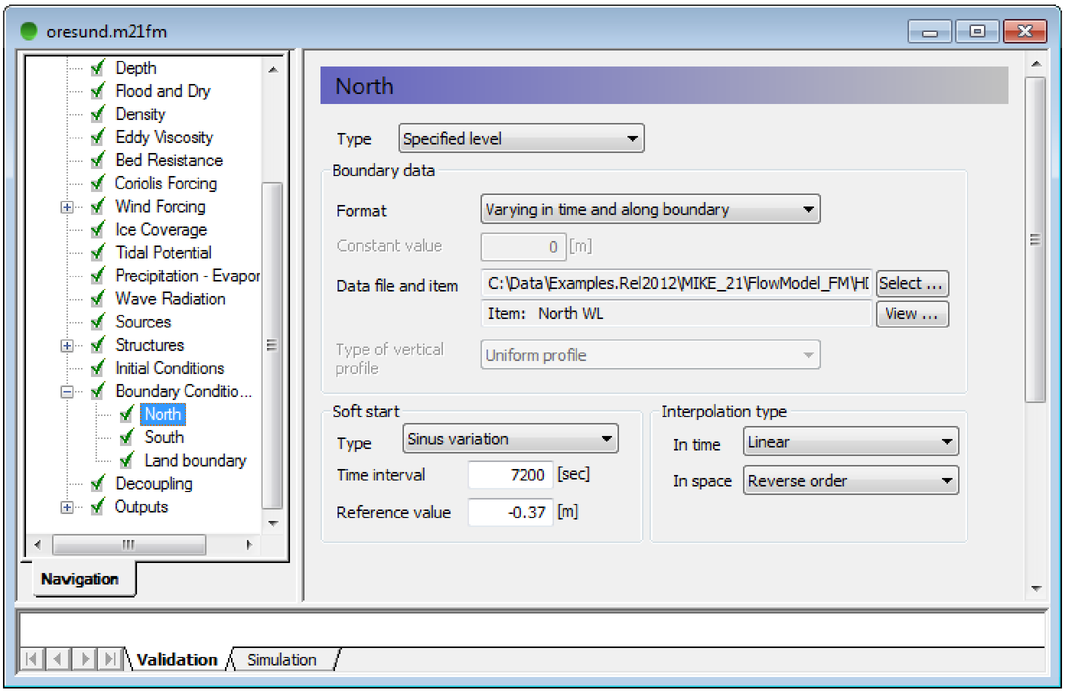


Figure 5.16 MIKE 21 Flow Model FM. The Boundary Conditions for the North Boundary are specified as waterlevel_north.dfs1

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

The boundary format must be set as 'Variable in time and along boundary' in order to specify the boundary as a line series file (dfs1).

Click Select ... and select the appropriate data file in the Open File window that appears, see Figure 5.18.

For the North Boundary select the waterlevel_north.dfs1, and for the South Boundary select waterlevel_south.dfs1.

Please note: when specifying a line series at the boundary it is important to know how MIKE 21 FM defines the first and last node of the boundary. The rule is: follow the shoreline with the discretised domain on the left hand side, see Figure 5.17. When a boundary is reached, this is the first node of the boundary.

In Section 3.1.2 the data for the north boundary was generated using data from Hornbæk in the first grid point and data from Viken in the last grid point of the boundary line. This is opposite the way MIKE 21 FM defines the direction of the boundary, and thus, the interpolation type for the north boundary should be set to reverse.

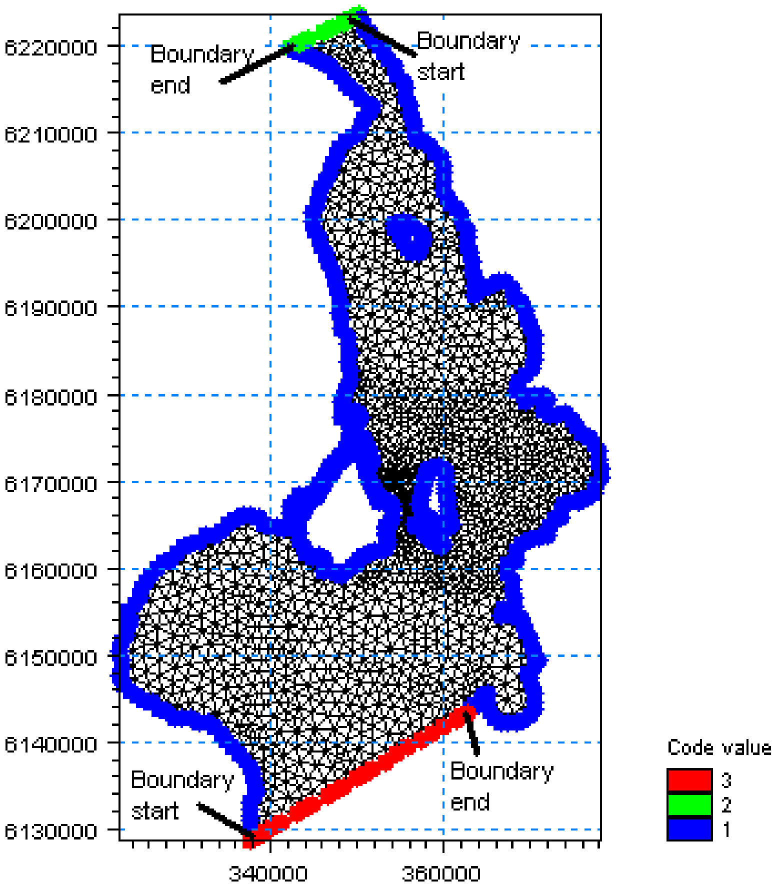


Figure 5.17 MIKE 21 Flow Model FM. The Boundaries are defined by codes in the mesh file. In this case Code 2 is North boundary, code 3 is South boundary and code 1 is land boundary

Use a soft start interval of 7200 s and a reference value corresponding to the initial value of –0.37 m. The soft start interval is a period in the beginning of a simulation where the effect of the boundary water levels does not take full effect. In the beginning of the soft start interval the effect of the specified Boundary Condition is zero and then the effect increases gradually until the boundaries has full effect on the model at the end of the soft start interval period.

The boundary data corrections due to Coriolis and wind are omitted because the spatial extension of the boundary is relatively small.

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

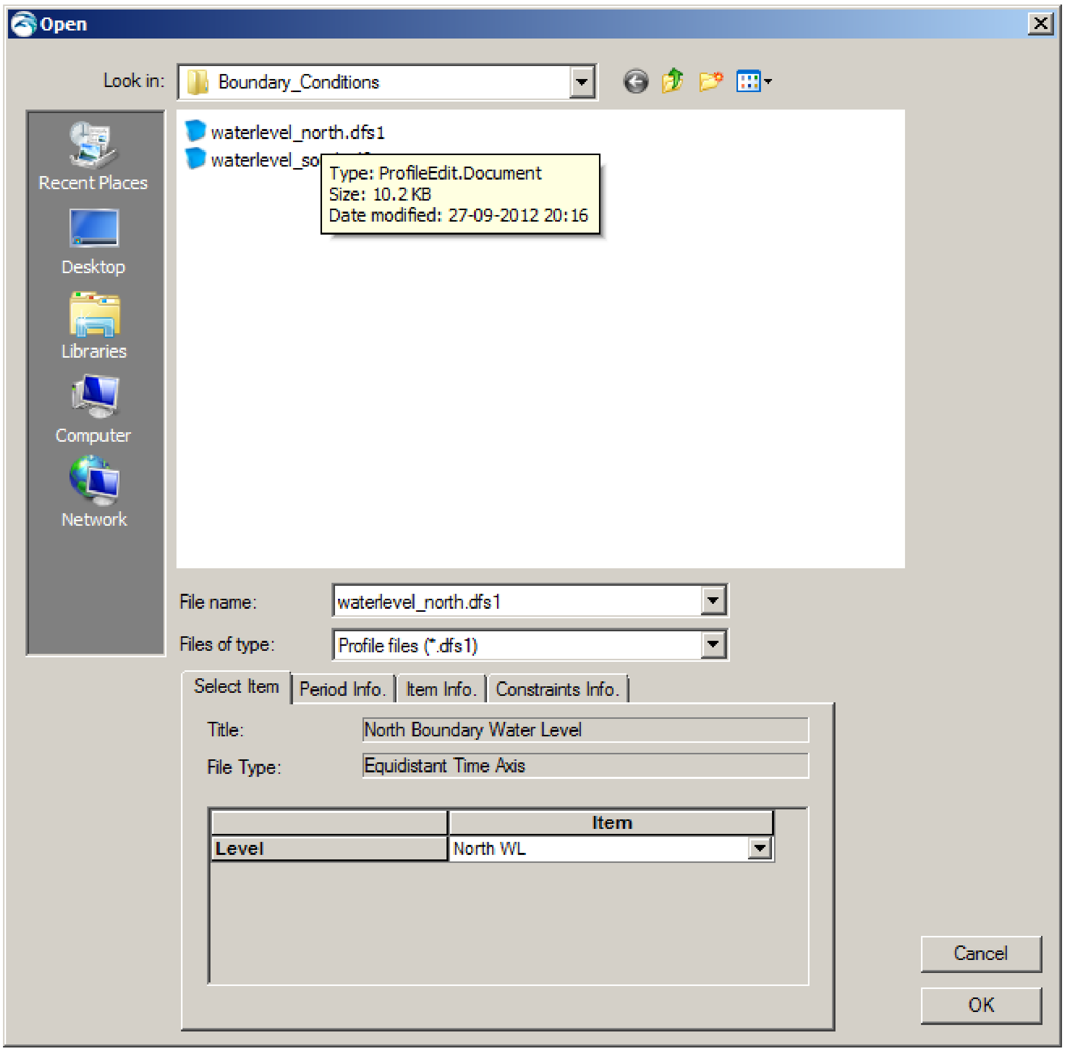


Figure 5.18   MIKE 21 Flow Model FM: Boundary Select File

Specify one output as area series and specify the resulting output file name, see Figure 5.19. Specify the file name flow.dfsu for our first simulation. Make sure the required disk space is available on the hard disk. Reduce the output size for the area series to a reasonably amount by selecting an output frequency of 3600 s which is a reasonably output frequency for a tidal simulation. As our time step is 120 s, the specified output frequency is 3600/120 = 30. As default, the full area is selected.

Pick the parameters to include in the output file as in Figure 5.20.

Also specify an output file as point series at the calibration station at Ndr. Roese, see the position in Table 5.2. 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.

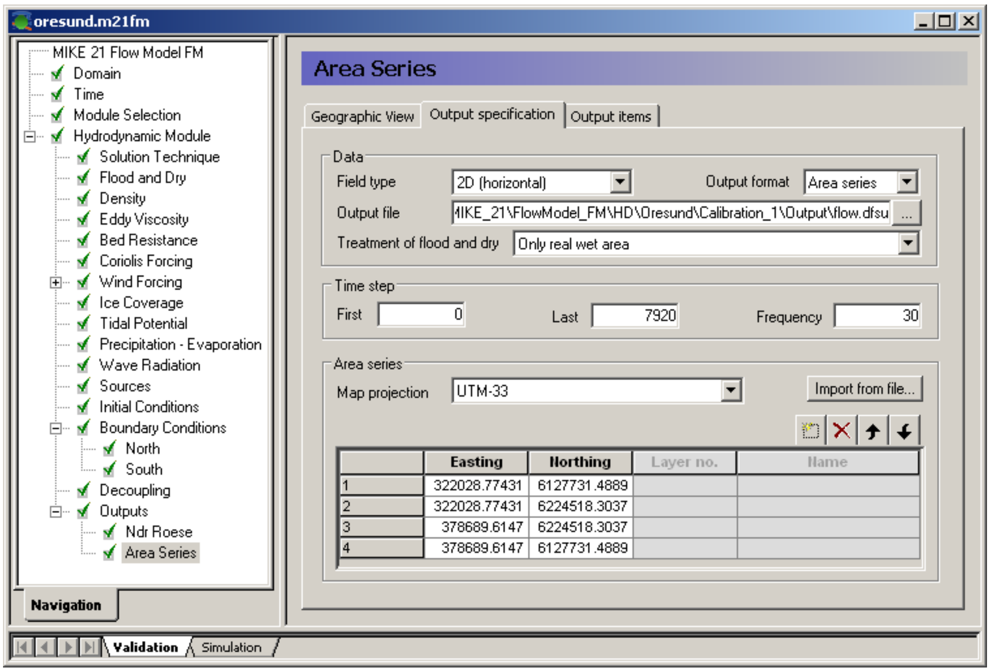


Figure 5.19 MIKE 21 Flow Model FM: Results can be specified as point, line or area series

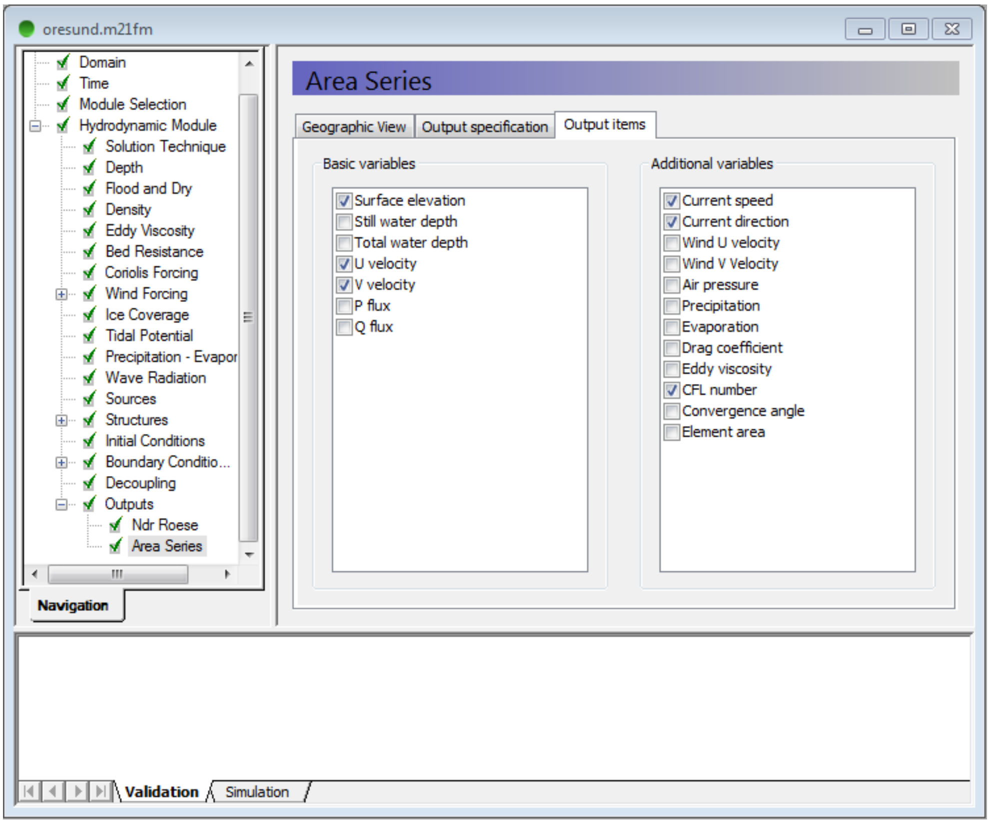


Figure 5.20 MIKE 21 Flow Model FM: The output Parameters specification

Table 5.2 Measurements at Ndr. Roese

Station	Data Files	Position	
		Easting (m)	Northing (m)
Ndr. Roese	waterlevel_ndr_roese.txt currents_ndr_roese.txt	354950	6167973

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:

.\Calibration_1\oresund.m21fm

Please note that if you experience an abnormal simulation, you should look in the log file (shown in the simulation tabs in the GUI) to see what causes the problem. Alternatively, you can use (File→Recent log file list) to select and open the log file in a separate window.

After the simulation use the Plot Composer (or Data Viewer) to inspect and present the results. In the following Figure 5.21 and Figure 5.22 two plots are shown; one with currents towards North and one with currents towards South.

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:

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

The Post Processing Tools (statistics, etc.) that are 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:

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

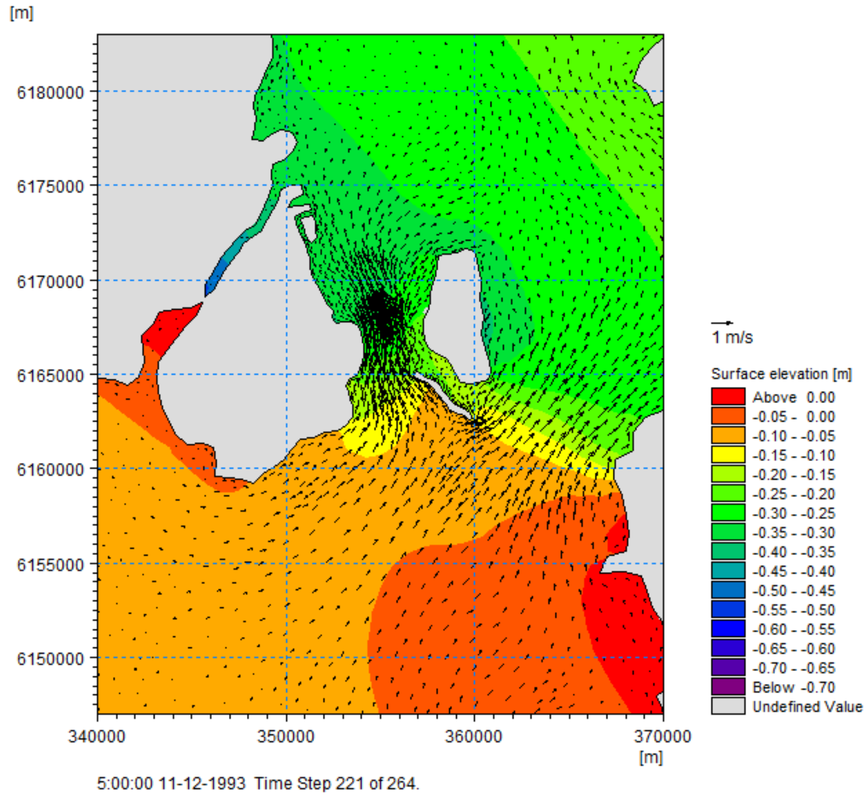


Figure 5.21 Current speed and water level during current towards North

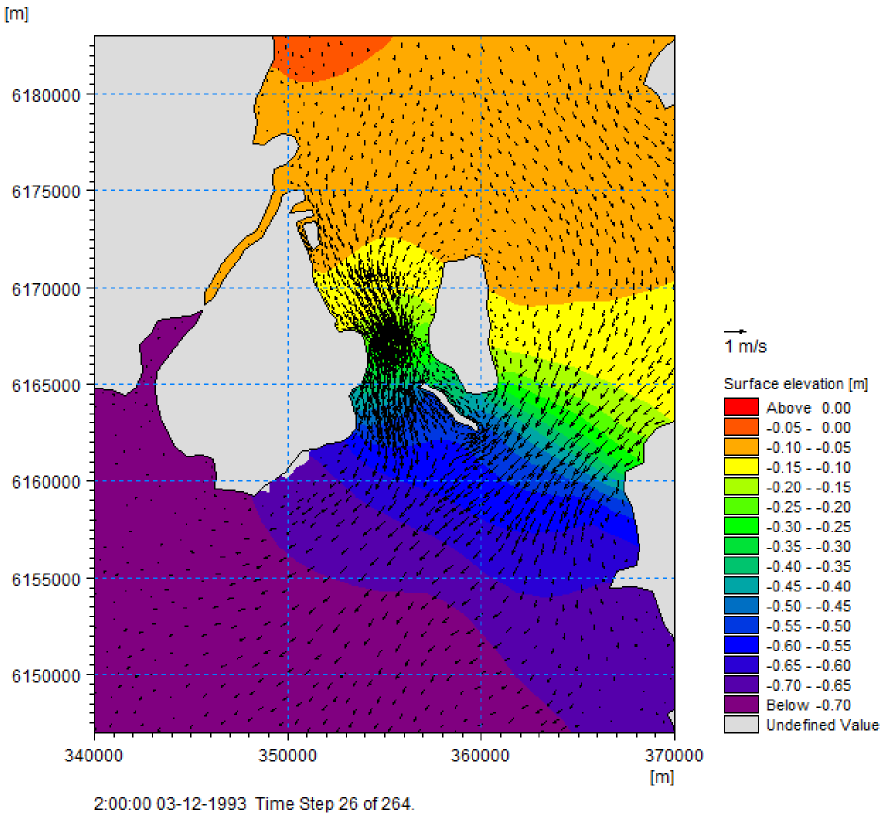


Figure 5.22 Current speed and water level during current towards South

5.2 Model Calibration

In order to calibrate the model we need some measurements inside the model domain. Measurements of water level and current velocities are available, see Table 5.2.

5.2.1 Measured water levels

Measurements of water level are given at Station Ndr. Roese (waterlevel_ndr_roese.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 levels at Ndr. Roese are shown in Figure 5.23.

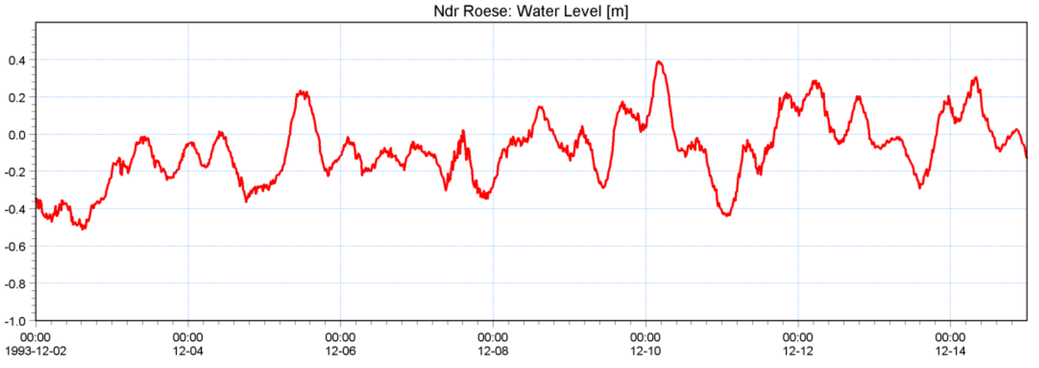


Figure 5.23 Ndr. Roese: Measured water level

5.2.2 Measured current velocity

Measurements of current velocities are given at station Ndr. Roese (currents_ndr_roese.txt). Import this file with the Time Series editor. Plots of current velocity and current speed and direction are shown in Figure 5.24, Figure 5.25 and Figure 5.26.

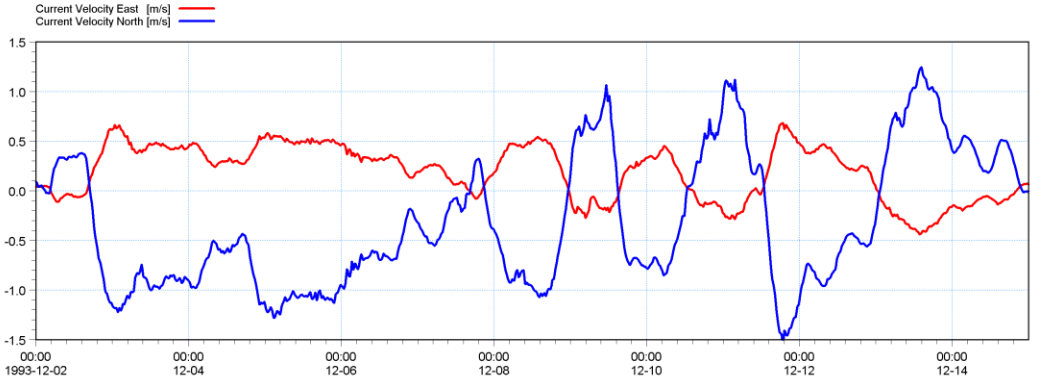


Figure 5.24 Ndr. Roese: Measured current velocity East and North component

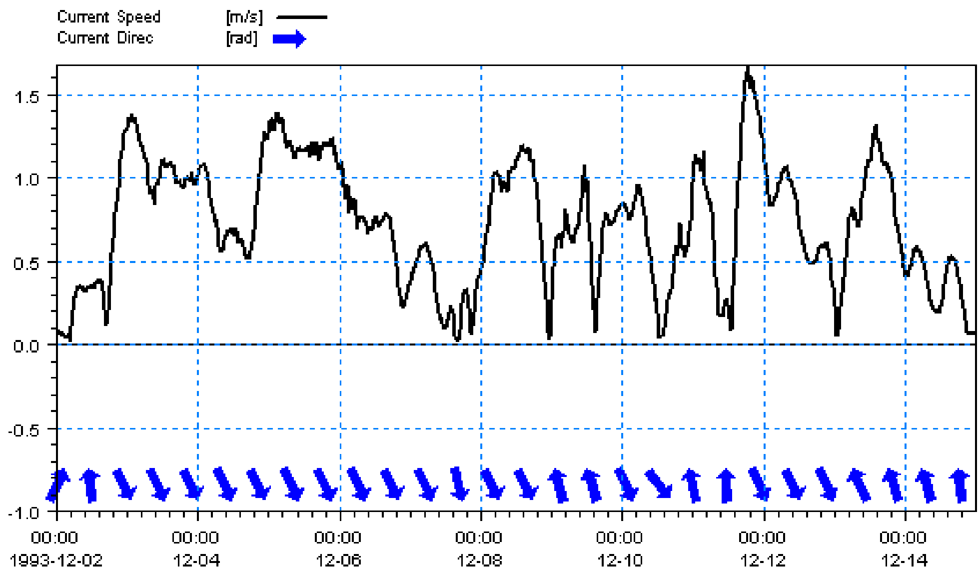


Figure 5.25 Ndr. Roese: Measured current speed and direction

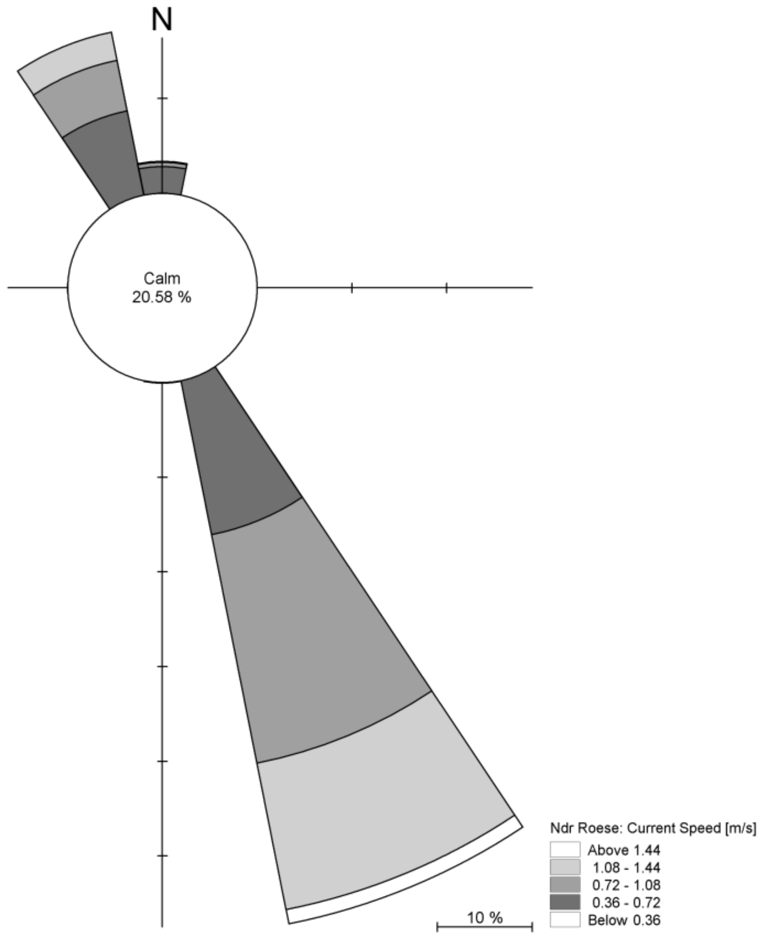


Figure 5.26 Ndr. Roese current rose, as it can be viewed with the Plot Composer

5.2.3 Compare model result and measured values

Use the Plot Composer to plot the simulated and measured water level and current. A Plot Composer file for this purpose is already made in this example. Open the file ndr_roese.plc in:

.\Calibration_1\Plots

to see a comparison of measurements and model output of water levels and currents at Ndr. Roese. Plots as shown in Figure 5.27 will appear (only if your simulation was successful).

The comparisons between measured and calculated water level and currents indicate that calibration might improve the results.

Try changing the Bed Resistance to a Manning number of 45 $m^{1/3}$/s and the Eddy Viscosity to a Smagorinsky value of 0.24 $m^{1/3}$/s and run the set-up again. Now the model output fit the measurements better, see Figure 5.28.

Try to calibrate further by changing the Manning number and the Eddy Coefficient with your own values. 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.

Note the way the folders are organised in the example: You can simply copy the Calibration_2 folder and rename it to Calibration_3. 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 are relative to the present folder. If you are making many simulations this trick can save a lot of time.

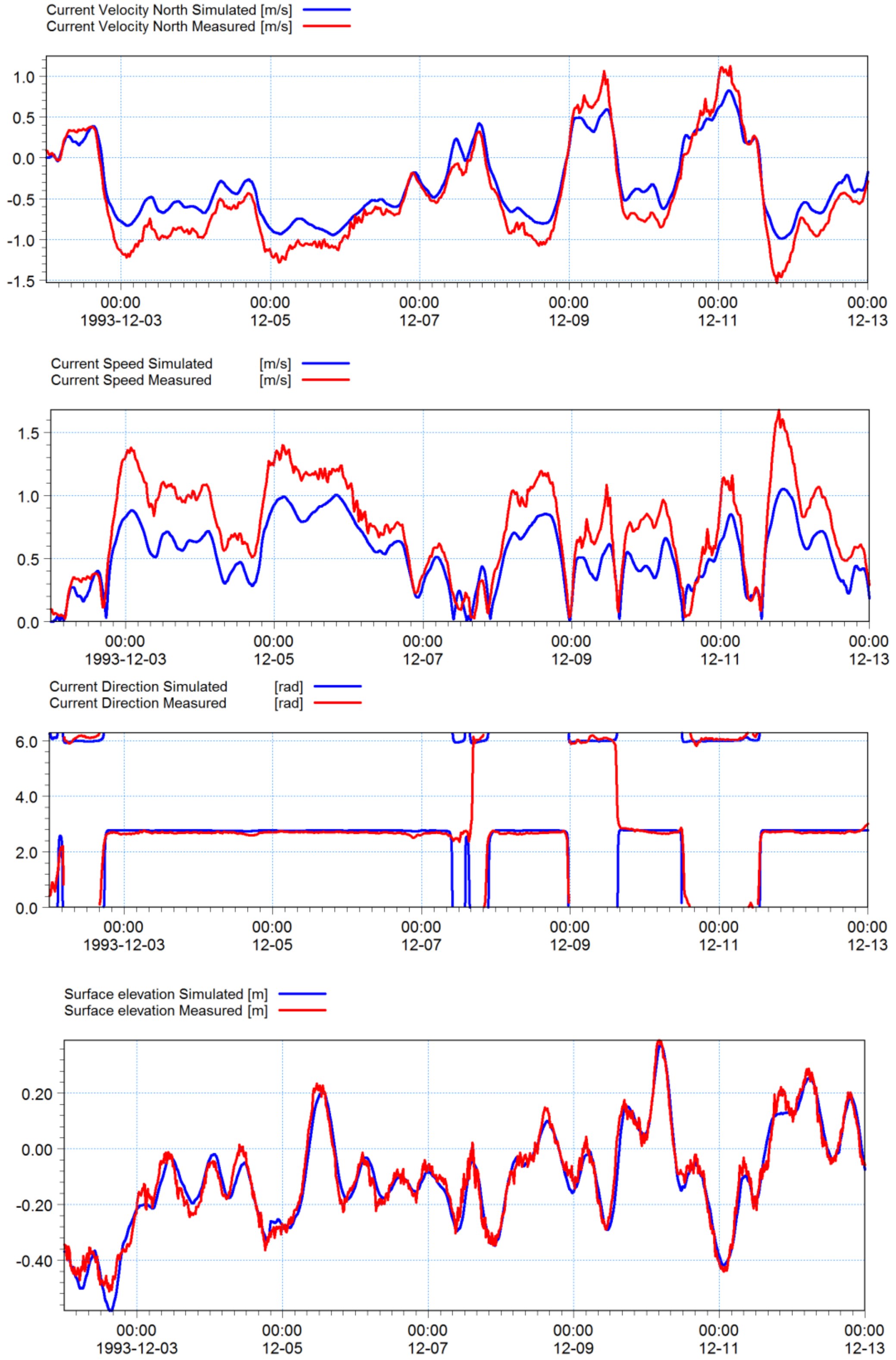


Figure 5.27 Comparison of measurements and model output of water levels and currents at Ndr. Rose with default parameters

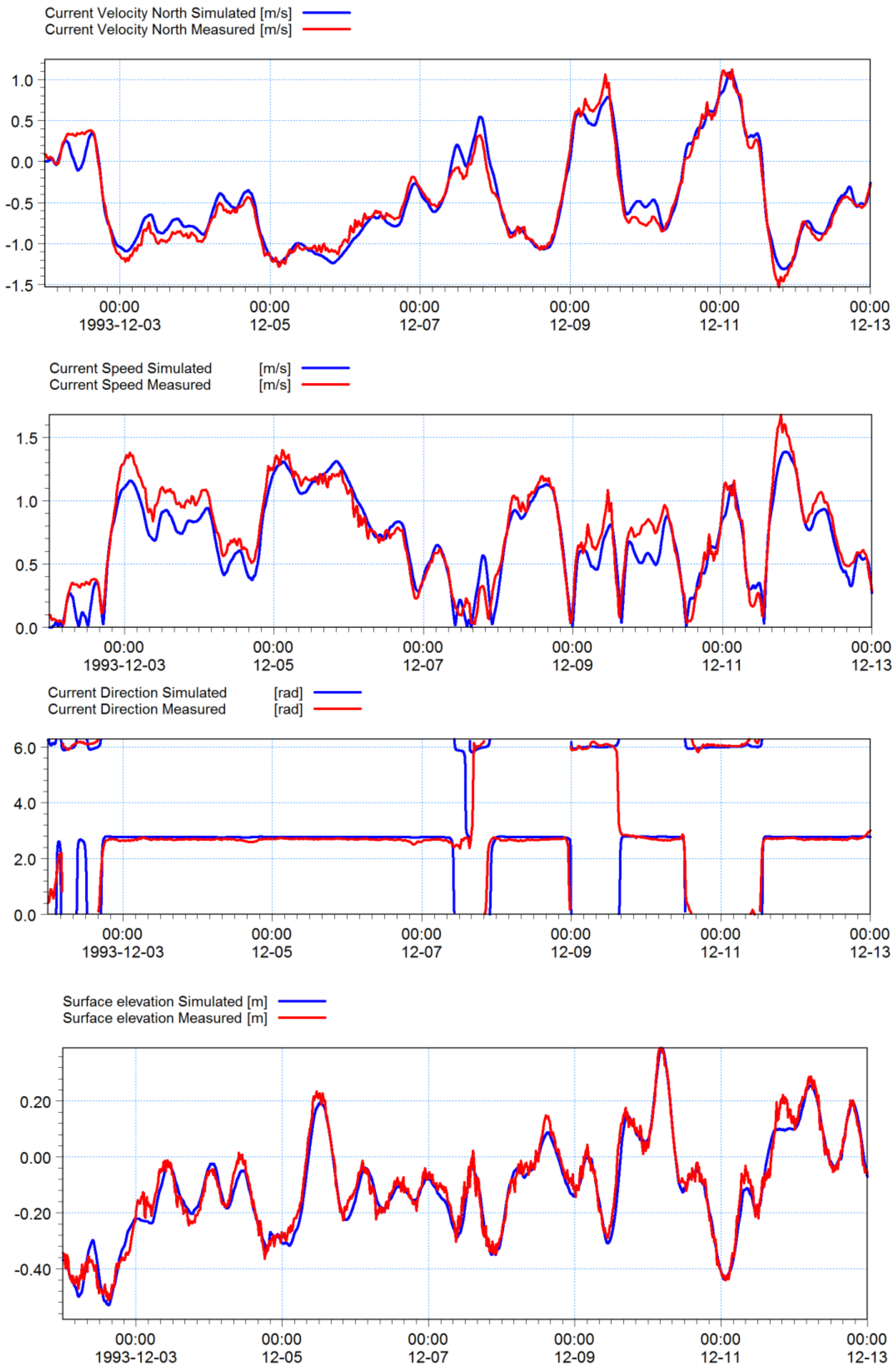


Figure 5.28 Comparison of measurements and model output of water levels and currents at Ndr. Rose after calibration

6 Set-Up of MIKE 3 Flow Model FM

6.1 Flow Model

We are now ready to set up the MIKE 3 Flow Model FM[4] model using the Øresund mesh with 1997 water depths, and the boundary conditions and forcings that were generated in Chapter 4.

The example has been divided into three sub periods:

1. A warm up period (3 days simulation used to warm up the model)
2. Period 1 (6 days simulation)
3. Period 2 (12 days simulation)

It is possible to open a specification file for each period, so that the example will not be too time consuming. The 'hot' initial conditions from the previous period to period 1 and period 2 are already supplied in the example. This way period 2, for instance, can be run without running period 1 first.

It is recommended that you start working with the warm up period first in this example. Afterwards start looking at period 1 and 2.

For the purpose of training do not start by opening the included specification files, but make your own specification files. The included specification files can be used for comparison.

The data material in this example allows the user to make a new simulation period 3 up until December 1997.

Hint for advanced users: Note that the total of 21 days can be simulated without opening the included specification files. Simply click on the bat file:

.\Calibration_1\run_warmup_period1_period2.bat

The trick with bat files can be useful when making many simulations. Starting the bat file will however overwrite the included hot initial conditions for period 1 and 2. So only start the bat job if it is intended to run the whole 21-day period. Otherwise open each specification file in the MIKE 3 Flow Model FM dialogues.

Table 6.1 gives a summary of the set-ups:

---

[4] The MIKE 3 Flow Model FM, Hydrodynamic Module, User Guide can be accessed via the MIKE Zero Documentation Index in the start menu

Table 6.1 Summary of set-ups

Parameter		Value	
Specification File		oresund.m3fm	
Mesh and Bathymetry		oresund.mesh (1997), 2090 Nodes in file, 10 vertical equidistant layers	
Simulation Period	(Warm up) (Period 1) (Period 2)	1997-09-06 00:00 – 1997-09-09 00:00 1997-09-09 00:00 – 1997-09-15 00:00 1997-09-15 00:00 – 1997-09-27 00:00	(3 days) (6 days) (12 days)
Time Step Interval	(Warm up) (Period 1) (Period 2)	7.2 s 7.2 s 108 s (Period 2)	
No. of Time Steps	(Warm up) (Period 1) (Period 2)	36000 72000 9600	
Solution Technique	(Warm up) (Period 1) (Period 2)	Critical CFL number: 1.0 Critical CFL number: 1.0 Critical CFL number: 0.8	
Enable Flood and Dry		Drying depth 0.01 m Flooding depth 0.05 m Wetting depth 0.1 m	
Initial Surface Level Initial Temperature Initial Salinity Initial Conditions Initial Conditions	(Warm up) (Warm up) (Warm up) (Period 1) (Period 2)	0.0 15.0 deg. C From file: salinity_sept_6_1997.dfs3 Hot started from warm up period simulation Hot started from period 1 simulation	
Wind		Varying in time, constant in domain: wind_ven.dfs0	
Wind Friction		Varying with Wind Speed: 0.0015 at 0 m/s, 0.0026 at 25 m/s	
North Boundary	Water Level Temperature Salinity	Type 1 data: waterlevel_north.dfs1 Type 2 data: temperature_north.dfs2 Type 2 data: salinity_north.dfs2	
South Boundary	Water Level Temperature Salinity	Type 1 data: waterlevel_south.dfs1 Type 2 data: temperature_south.dfs2 Type 2 data: salinity_south.dfs2	
Result Files		2D_ndr_roese_flow.dfs0 (Time series) 3D_ndr_roese_flow.dfs0 (Time series) 2D_flow.dfsu (2D result file) 3D_flow.dfsu (3D result file) Hot files	
CPU Simulation Time	(Warm up) (Period 1) (Period 2)	About 3½ hours with a 2.4 GHz PC, 512 MB DDR RAM About 7 hours with a 2.4 GHz PC, 512 MB DDR RAM About 15 hours with a 2.4 GHz PC, 512 MB DDR RAM	

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

The dialogue for setting up the MIKE 3 Flow Model FM is initiated from MIKE Zero, see Figure 6.1. (File→New→File→MIKE 3→Flow Model FM)

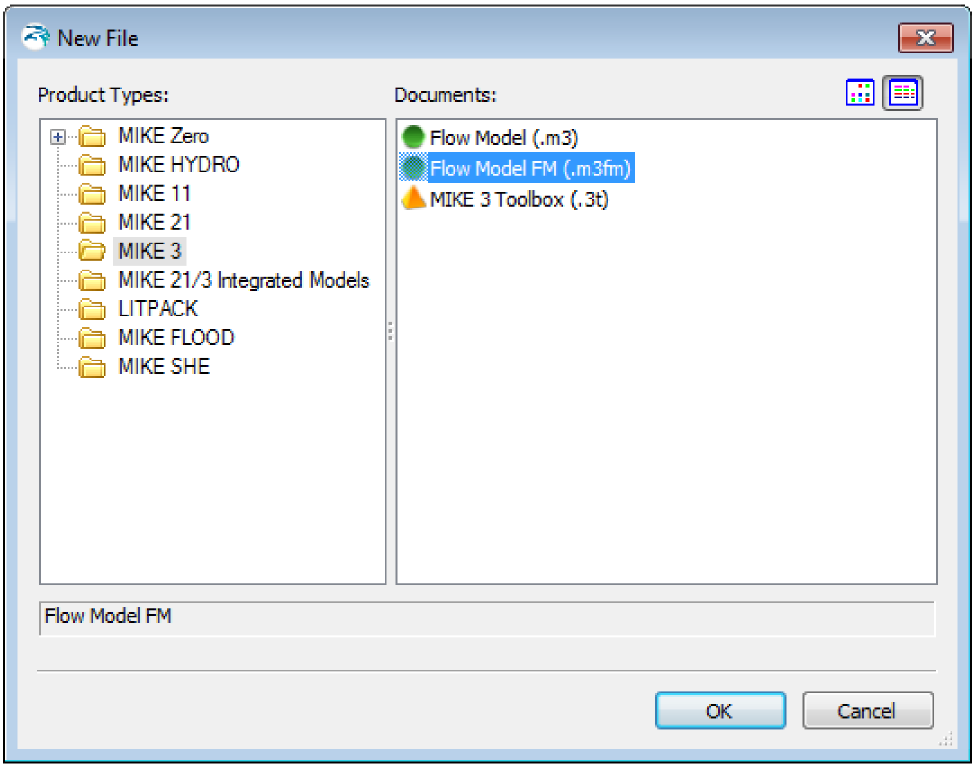


Figure 6.1 Starting MIKE 3 Flow Model FM in MIKE Zero

Specify the bathymetry and mesh file oresund.mesh in the Domain dialogue, see Figure 6.2. The projection zone has already been specified in the mesh as UTM-33. You should specify sigma mesh type and 10 equidistant vertical layers in the 'Vertical mesh' window. See Figure 6.3

The mesh file contains information about boundaries. In this case there are 2 boundaries, and it is possible to give the boundaries more describing names than numbers. In this Øresund example the North Boundary has the code 2 and the South Boundary has the code 3. Rename the boundary 'Code 2' to 'North' and 'Code 3' to 'South' in the dialogue Boundary Names, see Figure 6.4.

Specify the time step in the Time dialogue to 7.2 s. The time step range must be specified to 36000 time steps in order to simulate a total period of 3 days. See Figure 6.5.

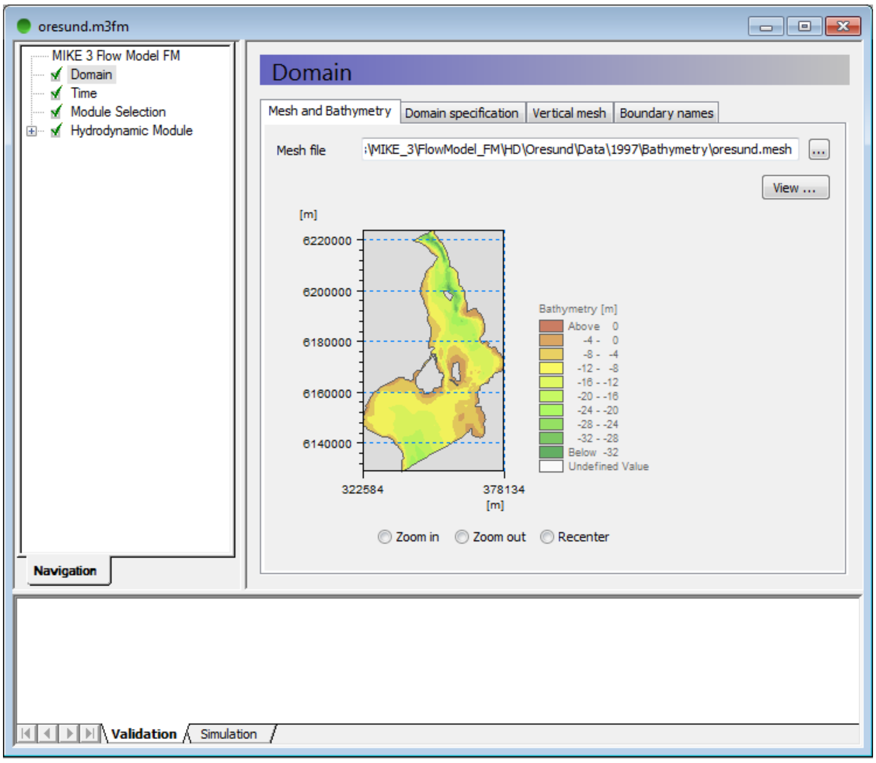


Figure 6.2 MIKE 3 Flow Model FM: Specify Domain

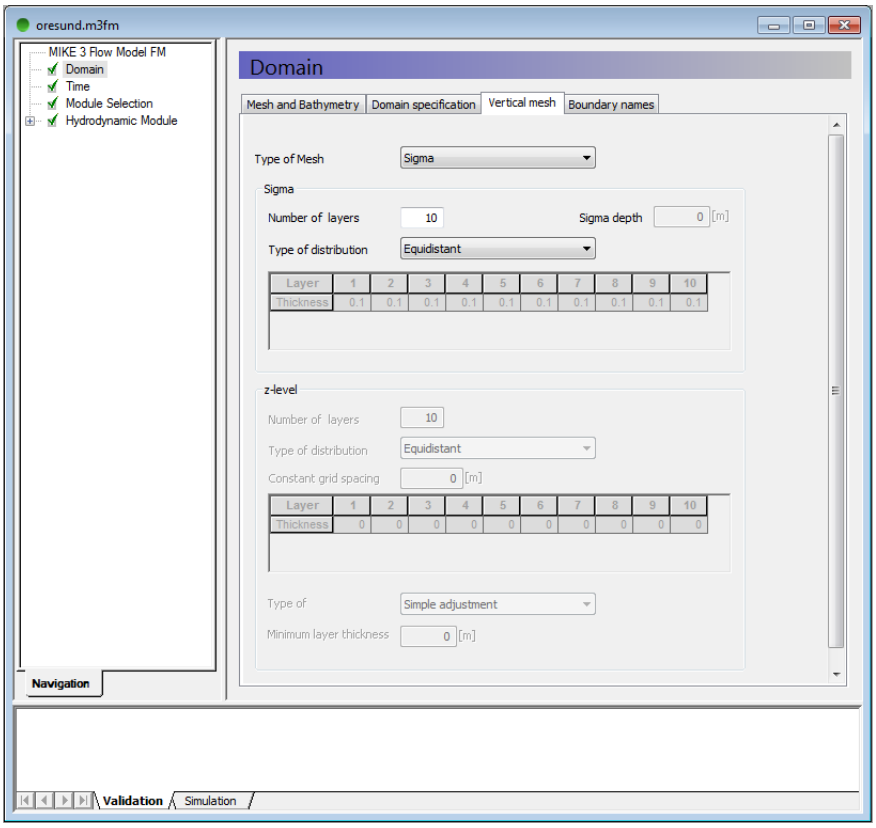


Figure 6.3 MIKE 3 Flow Model FM: Specify number of layers

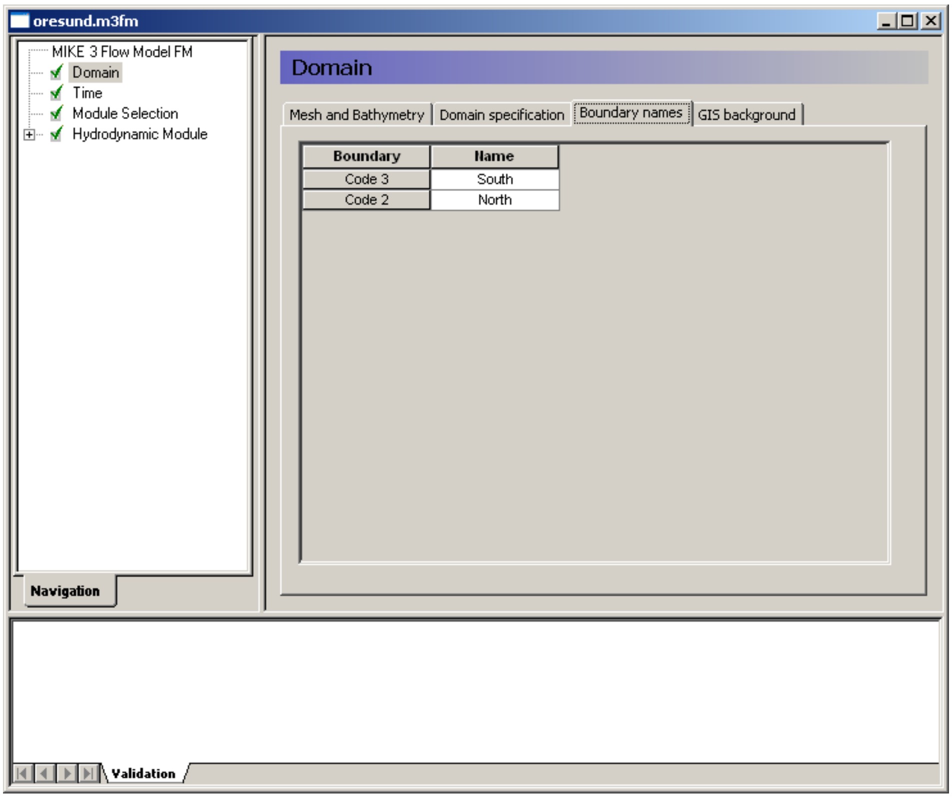


Figure 6.4 MIKE 3 Flow Model FM: Specify Boundary Names: For instance North and South Boundary

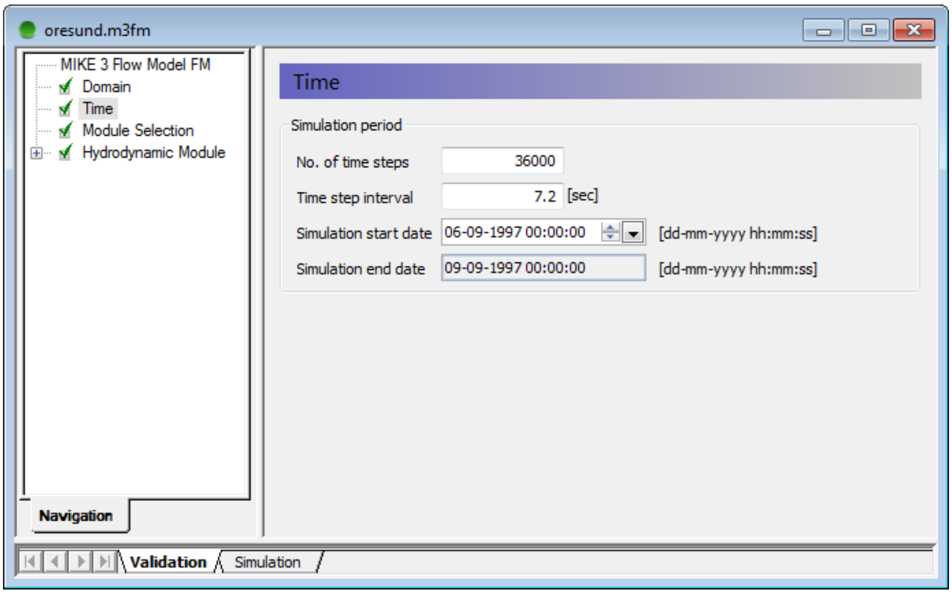


Figure 6.5 MIKE 3 Flow Model FM: Simulation period

In the Module Selection dialogue it is possible to include the 'Transport Module', the environmental 'ECO Lab Module', the 'Mud Transport Module', The 'Particle Tracking' Module or the 'Sand Transport' Module, see Figure 6.6.

In this example, only the Hydrodynamic Module will be used. Please also see the relevant step-by-step training guides if you want to extend your hydrodynamic model with an ecological model, or a mud model.

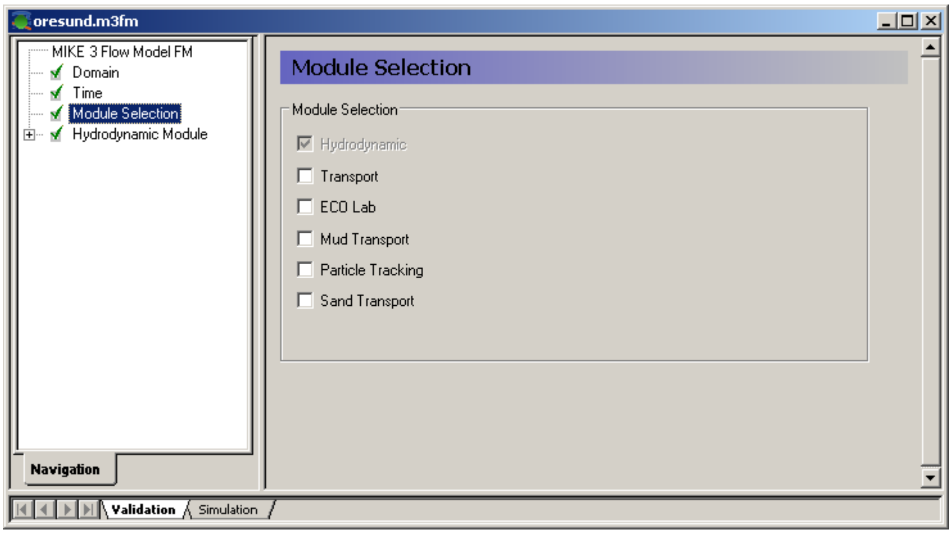


Figure 6.6 MIKE 3 Flow Model FM: Module Selection

In the Basic Equations dialogue the Shallow Water Equations should be selected, since the non-hydrostatic pressure, which is the assumption in the Navier-Stokes Equations, has no significant impact on the modelling results for this example.

In the Solution Technique dialogue the maximum time step is set to the overall time step. For the Warm up period and Period 1, the overall time step is small, itself ensuring a stable solution. Thus, the critical CFL number is set to 1.0. For Period 2, the overall time step is large. In this case the critical CFL number is set to 0.8 to ensure stability. Specify ('Lower order, fast algorithm') as the solution method and set the minimum time step (0.01s).

In the Depth dialogue it is possible to include a spatially varying depth correction. In this example the bathymetry is kept as constant.

In this example the Flooding and Drying in the model should be included, because some areas along the shores of Saltholm 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. Including Flooding and Drying can however influence stability of the model, so if the areas that dry out are not important for the model study, you might consider not including Flooding and Drying. In that case you should manipulate the mesh file and make greater depths in the shallow areas to prevent those areas from drying out, and then run the model without Flooding and Drying. In our case Flooding and Drying is included, so select a Drying depth of 0.01 m, a Flooding depth of 0.05 m, and a Wetting depth of 0.1 m, see Figure 6.7.

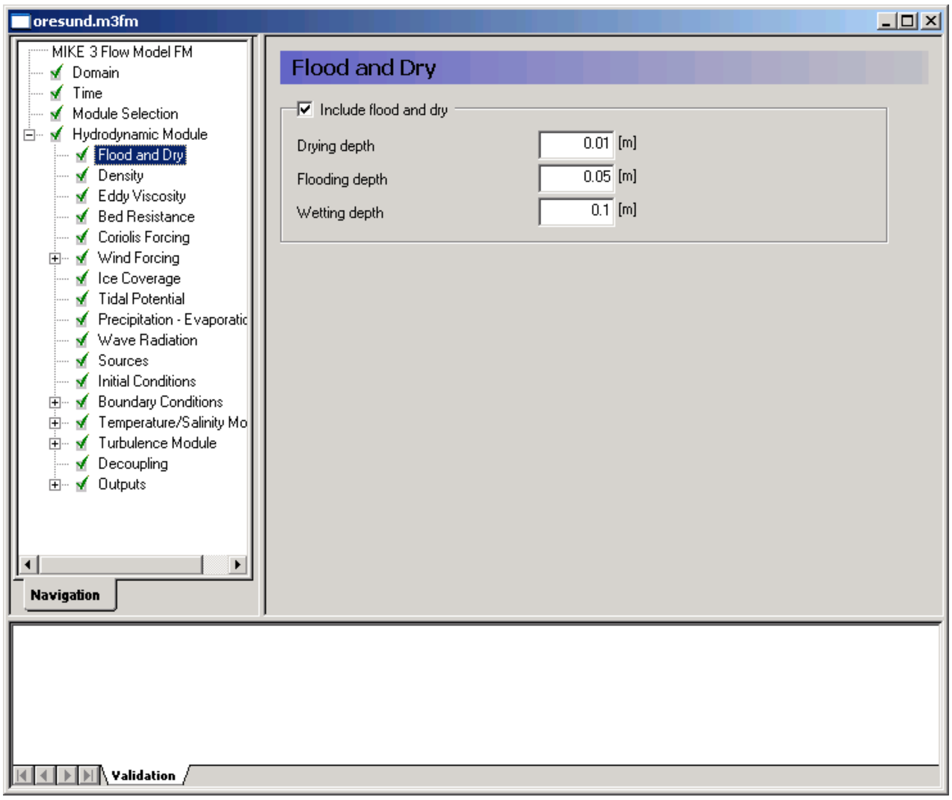


Figure 6.7 MIKE 3 Flow Model FM: Flood and Dry specification

Øresund is situated in a transition zone between the brackish Baltic Sea to the South and the more saline Kattegat to the North. Therefore, Øresund is often stratified with dense, saline water from Kattegat in the lower part of the water column and the brackish water from the Baltic Sea in the upper part of the water column. A sill across Øresund called Drogden allows the saline plumes from north to only pass the sill under certain long periods of uniform wind conditions. These wind conditions are rather rare and therefore large-scale intrusion of the saline Kattegat water across Drogden into the Baltic only happens every 7 years in average.

In this example we will try to describe the salinities and temperatures in Øresund. Therefore the density should be specified as 'Function of temperature and salinity', see Figure 6.8.

Specify the Horizontal Eddy Viscosity as Smagorinsky formulation with a constant value of 0.1, see Figure 6.9.

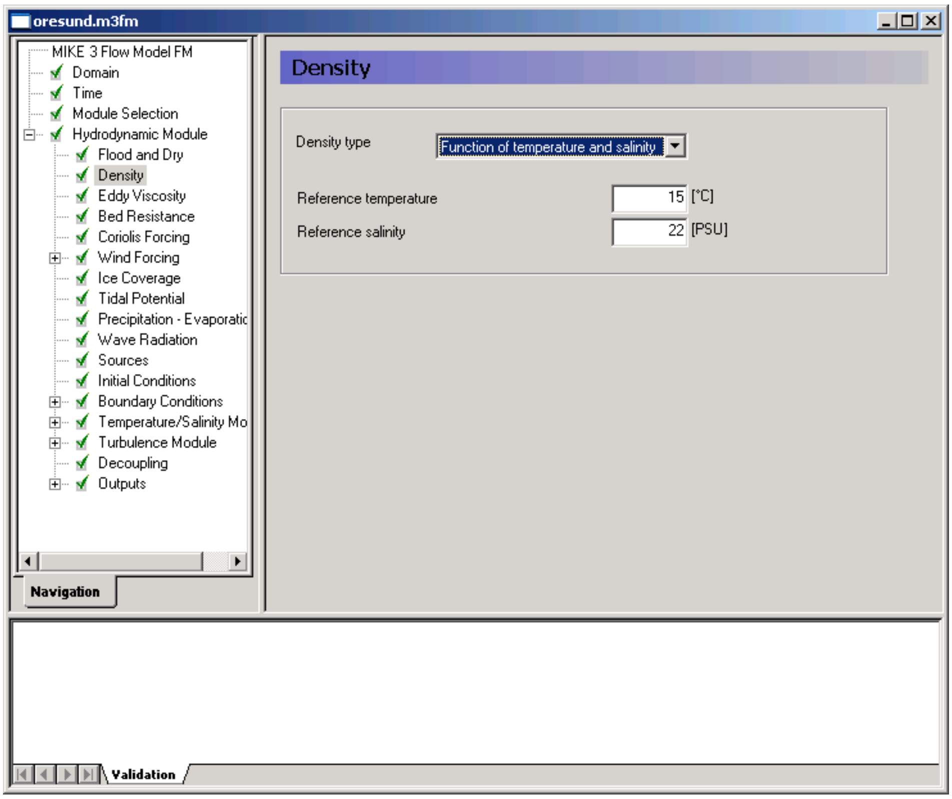


Figure 6.8 MIKE 3 Flow Model FM. Density specification. Density should in this example be a function of salinity and temperature

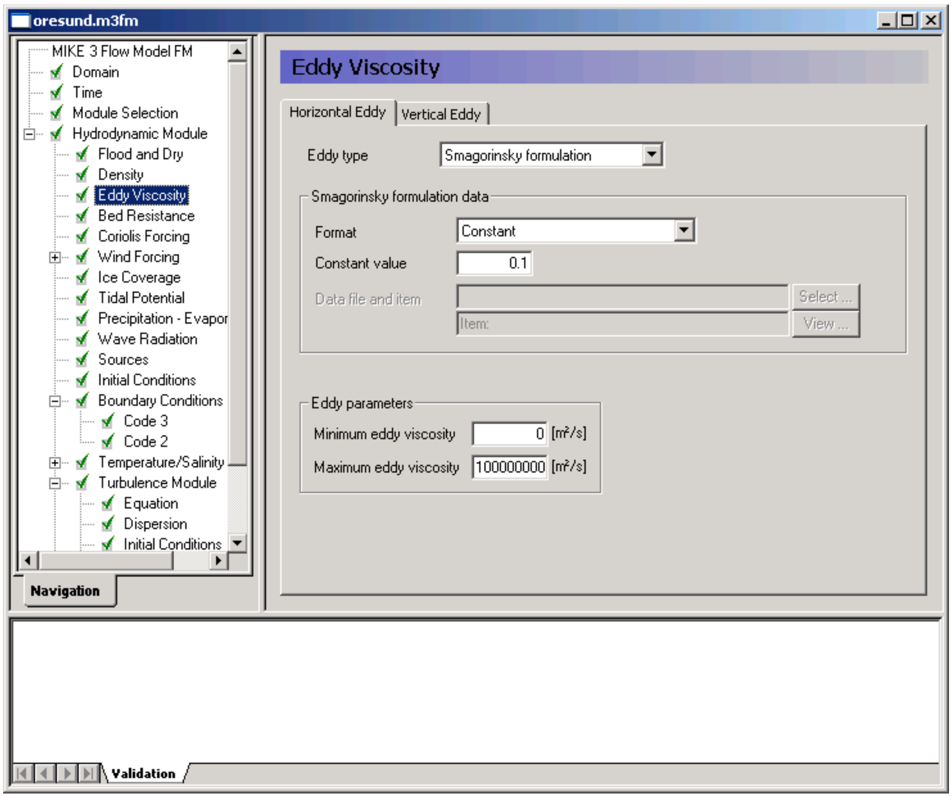


Figure 6.9 MIKE 3 Flow Model FM. Horizontal Eddy Viscosity specification

Choose a two-equation turbulence model for the Vertical Eddy Viscosity. This formulation implies that the Turbulent Kinetic Energy must be determined with the Turbulence Module, see Figure 6.10.

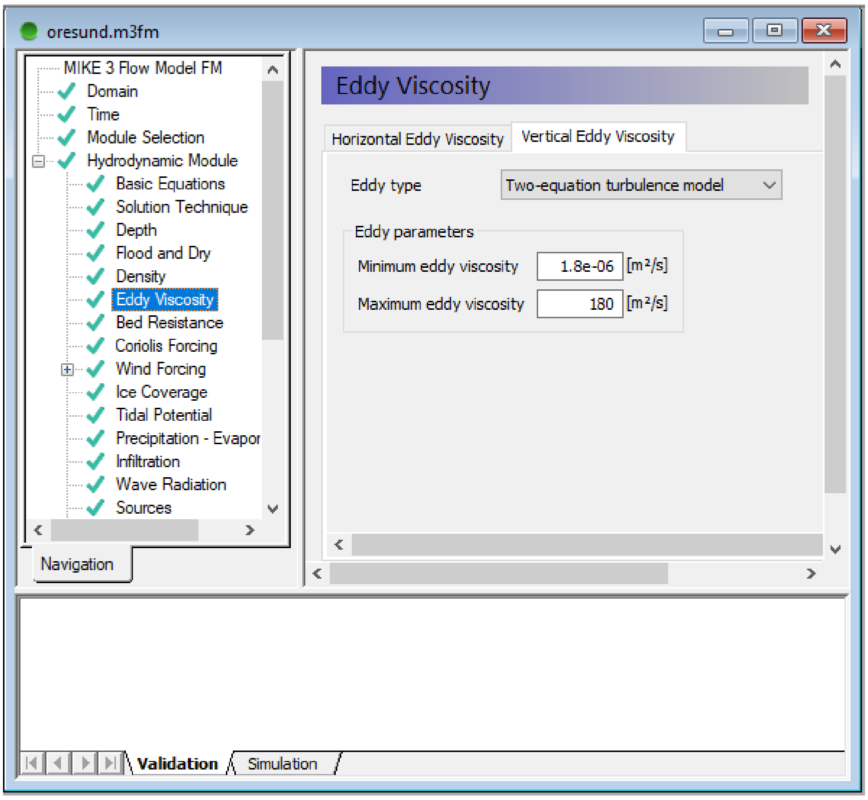


Figure 6.10 MIKE 3 Flow Model FM. Vertical Eddy Viscosity specification

Next the Bed Resistance should be specified. In general the reasons for experiencing blow-ups are numerous.

Two strategies can often be used to prevent instabilities:

1. Increasing the Eddy Viscosity will 'smoothen' the values and can sometimes solve instability problems
2. Increasing Bed Resistance can 'smoothen' Water Levels, and it can also dampen instabilities in some situations.

It is best to solve the problem locally around the area causing problems, so that the trick of solving an instability problem does not significantly influence the calibration of results in other areas.

A roughness height specified as a constant of 0.1 m is used, see Figure 6.11.

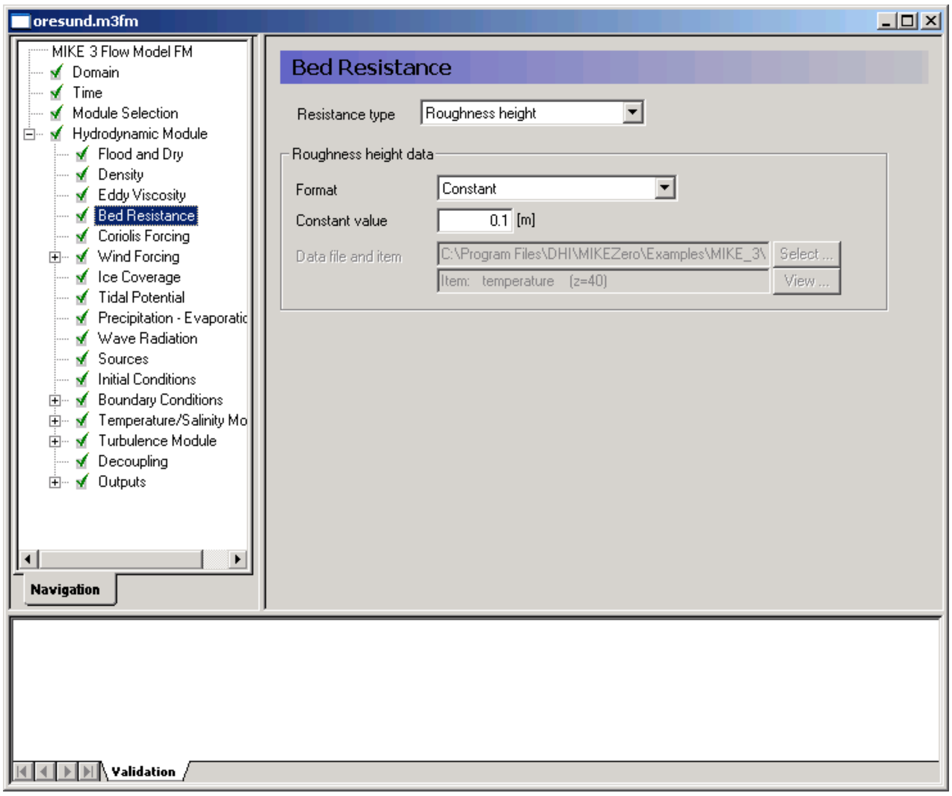


Figure 6.11 MIKE 3 Flow Model FM. Bed Resistance specification

Even though there are often strong currents in Øresund, the effect of Coriolis forces is not so significant, because the strait is rather narrow. However, Coriolis is always included in real case applications. Only for laboratory type of simulations Coriolis is sometimes not included. So there is no reason for excluding Coriolis Forcing, because the effect on simulation time is small, see Figure 6.12.

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

Note that an easy way to see the Wind data file is to simply click View ... in the Wind Forcing dialogue, see Figure 6.13.

It is often a good practice to use a soft start interval. In our case 7200 s should be specified. The soft start interval is a period in the beginning of a simulation where the effect of the wind does not take full effect. In the beginning of the soft start interval 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.

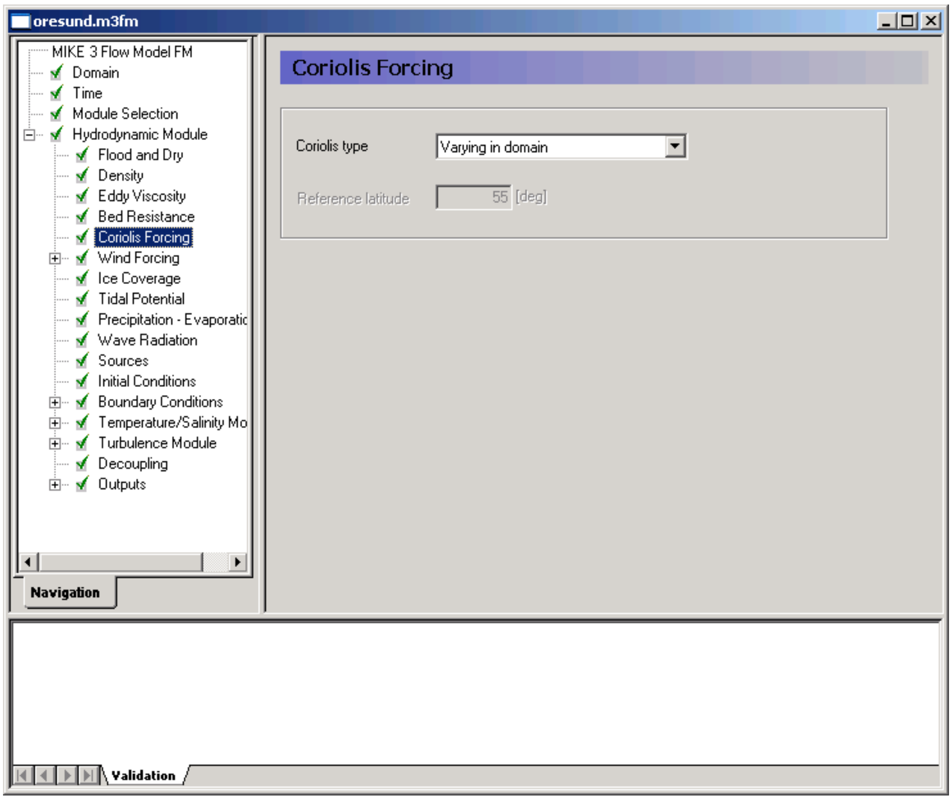


Figure 6.12 MIKE 3 Flow Model FM. Coriolis specification

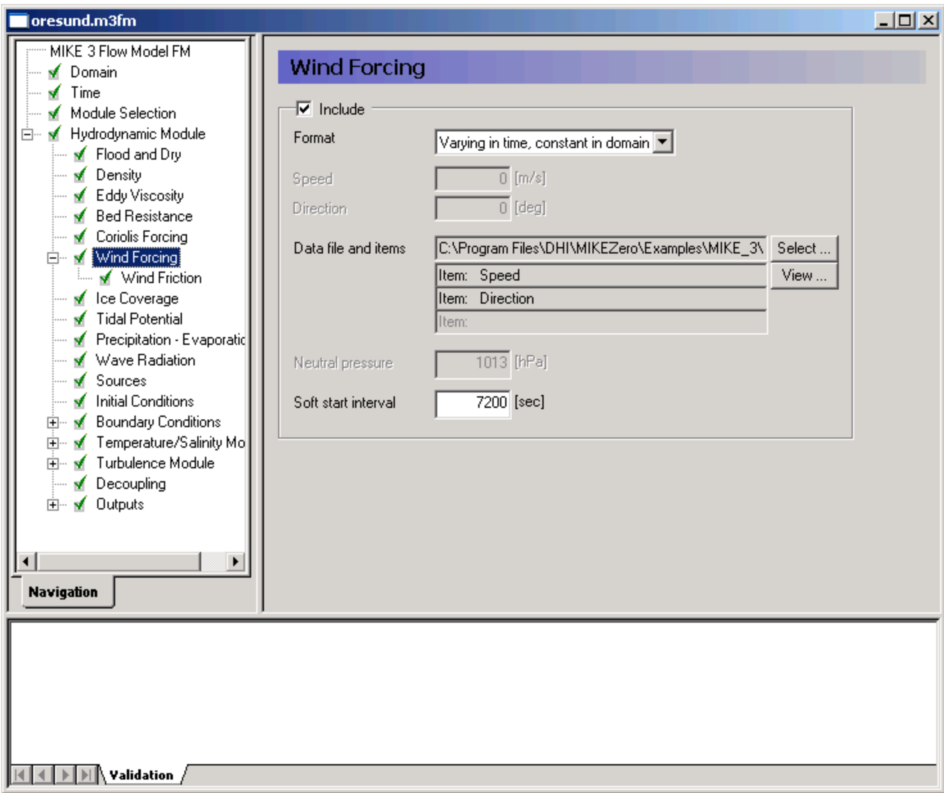


Figure 6.13 MIKE 3 Flow Model FM. Wind Forcing specification

Specify the Wind friction as 'Varying with Wind Speed' and use the values for the Wind friction as in Figure 6.14.

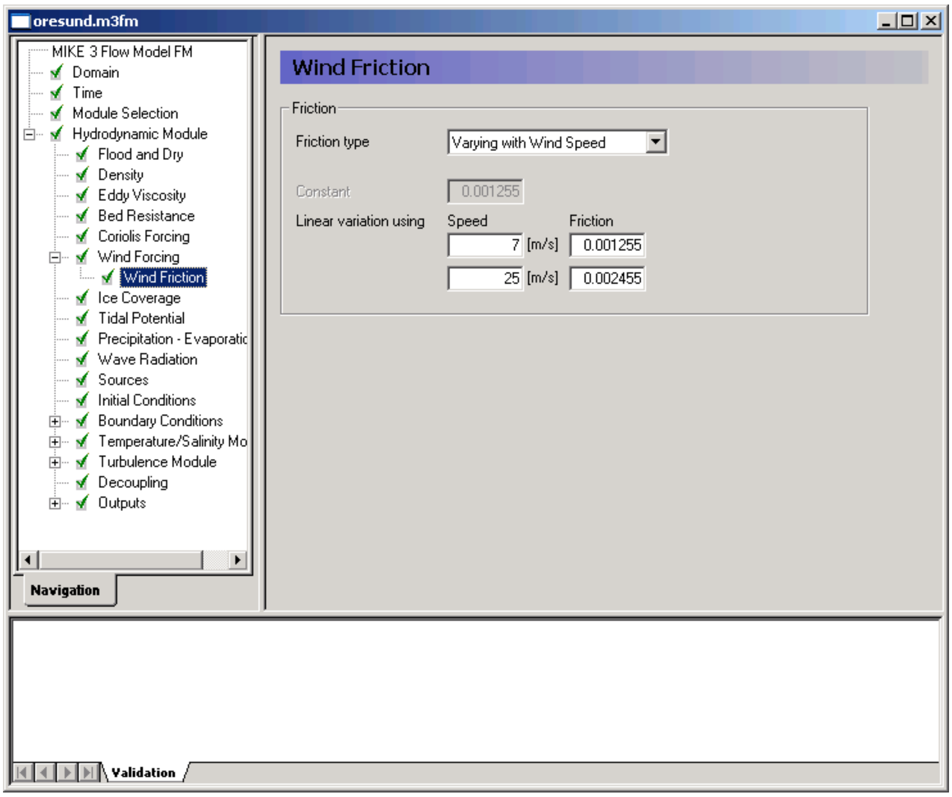


Figure 6.14 MIKE 3 Flow Model FM. Wind Friction specification

In this example

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

The discharge magnitude and velocity for each Source and Sink should be specified in the Sources dialogue. But, because the sources in Øresund are too small to have significant influence on the overall hydrodynamics in Øresund they are not included in this example. So since we do not have any sources, leave it blank. See Figure 6.15.

After inspection of the boundary conditions at the simulation start time you should decide the initial surface level. In this case we will use a constant Surface Elevation of 0.0 m for the warm up period, which is about the average value between our North and South Boundary at the start of the simulation, see Figure 6.16.

The Period 1 and Period 2 simulations use the 'hot', simulated results from the previous period as initial conditions.

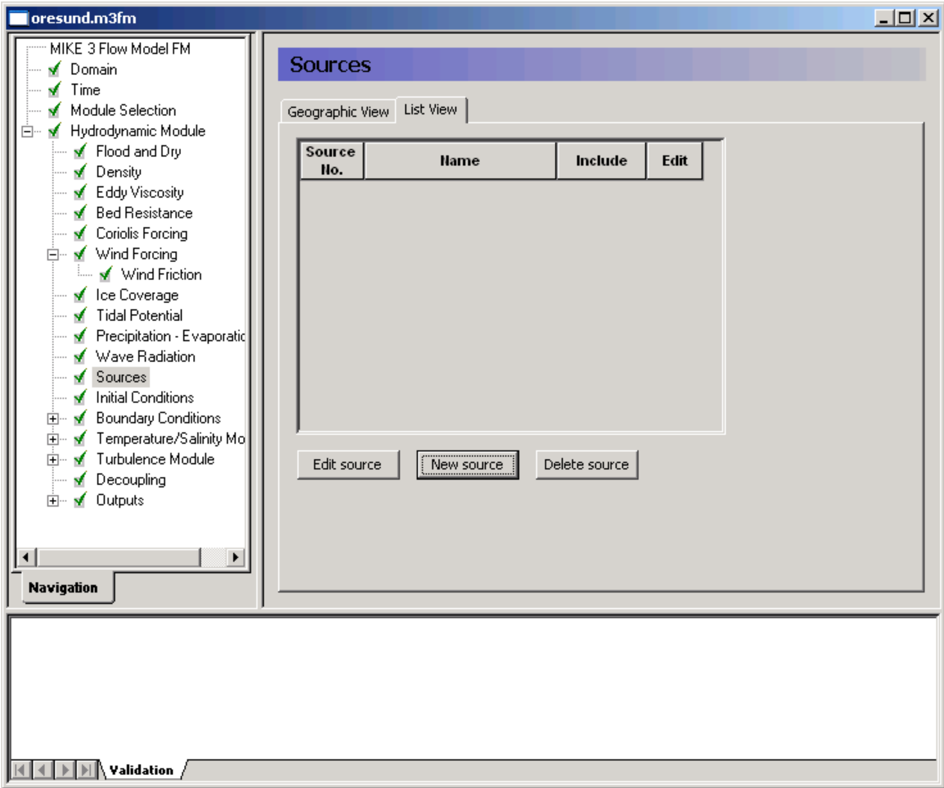


Figure 6.15 MIKE 3 Flow Model FM: Source and Sink

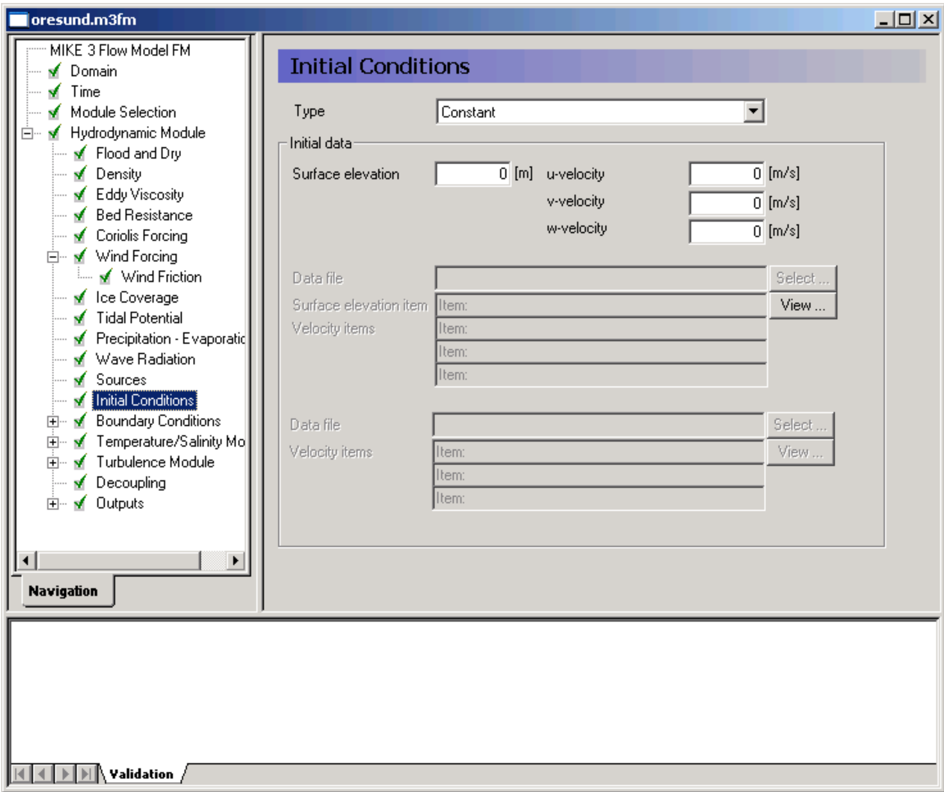


Figure 6.16 MIKE 3 Flow Model FM: Hydrodynamic Initial Conditions

In the Boundary Conditions dialogue the boundary conditions should be specified for the boundary names that was specified in the Domain dialogue, see Figure 6.4. There is a North Boundary and a South Boundary and the line series that was generated in chapter 3 shall be used.

In this case the boundary type is 'Specified Level' (Water Level), because only Water Level measurements are available at the boundaries. 'Specified Level' means that the Water Levels are forced at the boundaries, and the discharge across the boundary is unknown and estimated during simulation. In case you choose the boundary type as 'Specified discharge' the discharge is forced and the water levels at the boundary are unknown and estimated during simulation. A stable model in an area with two boundaries can often be obtained if one boundary is type 'Specified discharge' and another is type 'Specified Level'.
The Boundary Format must be set as 'Variable in time and along boundary' in order to specify the boundary as a line series file (dfs1).

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

For the North Boundary select the waterlevel_north.dfs1, and for the South Boundary select waterlevel_south.dfs1.

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

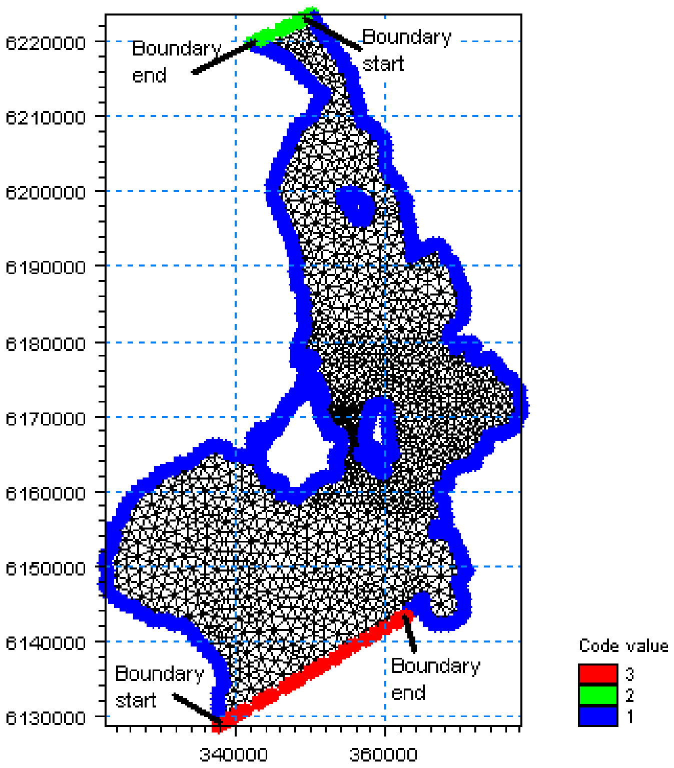


Figure 6.17 MIKE 3 Flow Model FM. The Boundaries are defined by codes in the mesh file. In this case code 3 is South boundary, code 2 is North boundary and code 1 is land boundary

Please note: When specifying a Line Series, or an Area Series at the boundary it is important to know how MIKE 21 FM defines the first and last nodes of the boundary. The rule is: follow the shoreline with the domain on the left hand side, see Figure 6.17. When a boundary is reached, this is the first node of the particular boundary.

Use a soft start interval of 7200 s and a reference value corresponding the initial value of 0.0 m. The soft start interval is a period in the beginning of a simulation where the effect of the boundary water levels does not take full effect. In the beginning of the soft start interval the effect of the specified Boundary Condition is zero and then the effect increases gradually until the boundaries has full effect on the model at the end of the soft start interval period. See. Figure 6.18.

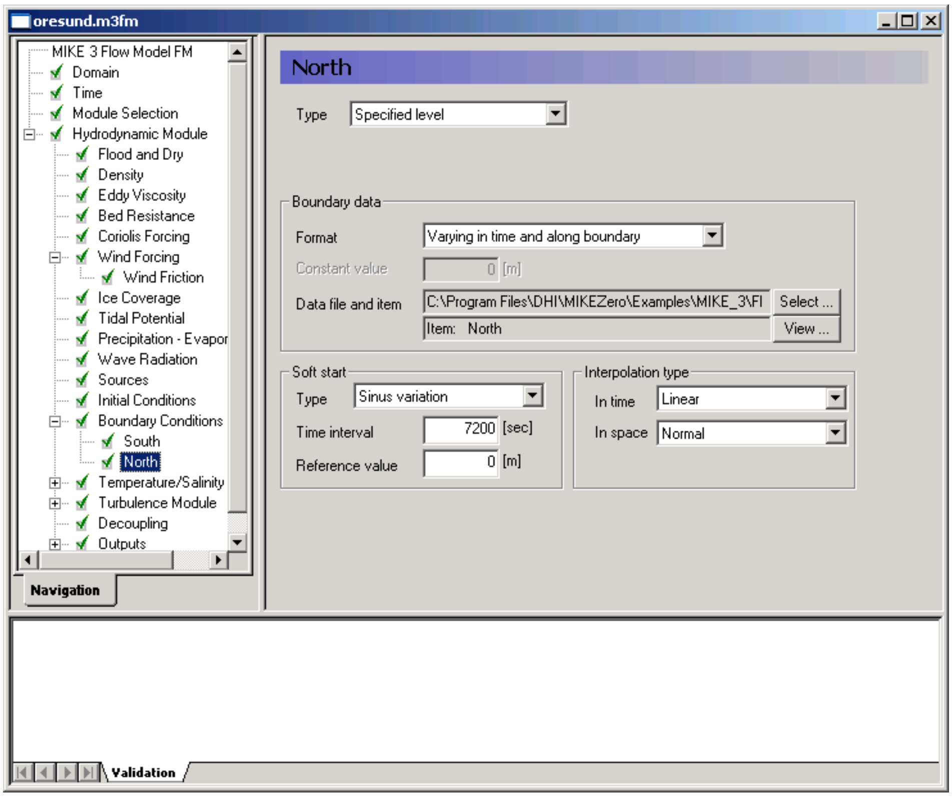


Figure 6.18 MIKE 3 Flow Model FM. The Boundary conditions for the North Boundary are specified as 'Variable in time and along boundary': waterlevel_north.dfs1

The density in this example is a function of salinity and temperature. So the model specifications are enhanced with specifications for modelling salinity and temperature. In case you have any problems with stability of salinity or temperature, you have the option of range checking salinity and temperature in the Equation dialogue. Values outside the specified range are cut off, see Figure 6.19. So you should only use range checking if you cannot manage instabilities of salinity or temperature with other methods, because you may violate the mass budget with this method.

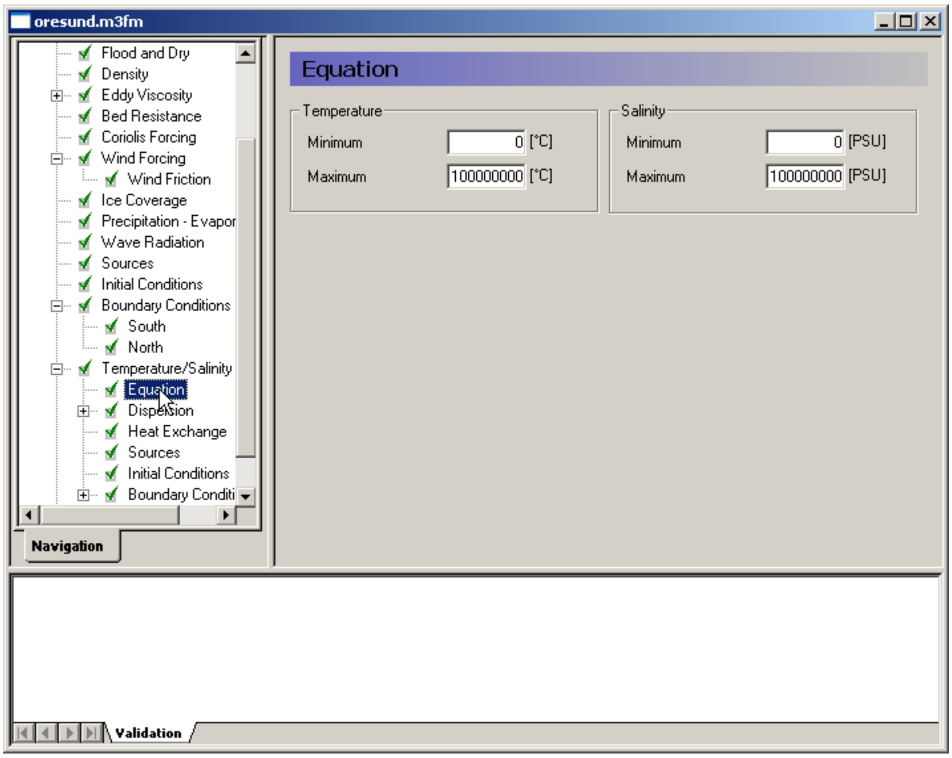


Figure 6.19 MIKE 3 Flow Model FM: Temperature/Salinity module range checking. Values outside the specified range of minimum and maximum values are cut off

Use the same solution technique as in the Hydrodynamic module.

The dispersion settings for the Temperature and Salinity module is for both the horizontal and vertical dispersion specified as Scaled Eddy Viscosity, see Figure 6.20. The horizontal sigma value should be set as 1. The vertical dispersion sigma value should be set as 0.1 to support a stratification of Øresund. Note that the dispersion settings are common for salinity and temperature.

The heat exchange with the surroundings is included to simulate the temperatures better in the model. The default values are used and the air temperature is specified as 'Varying in time and constant in domain'. Use the file: temperature.dfs0. See Figure 6.21.

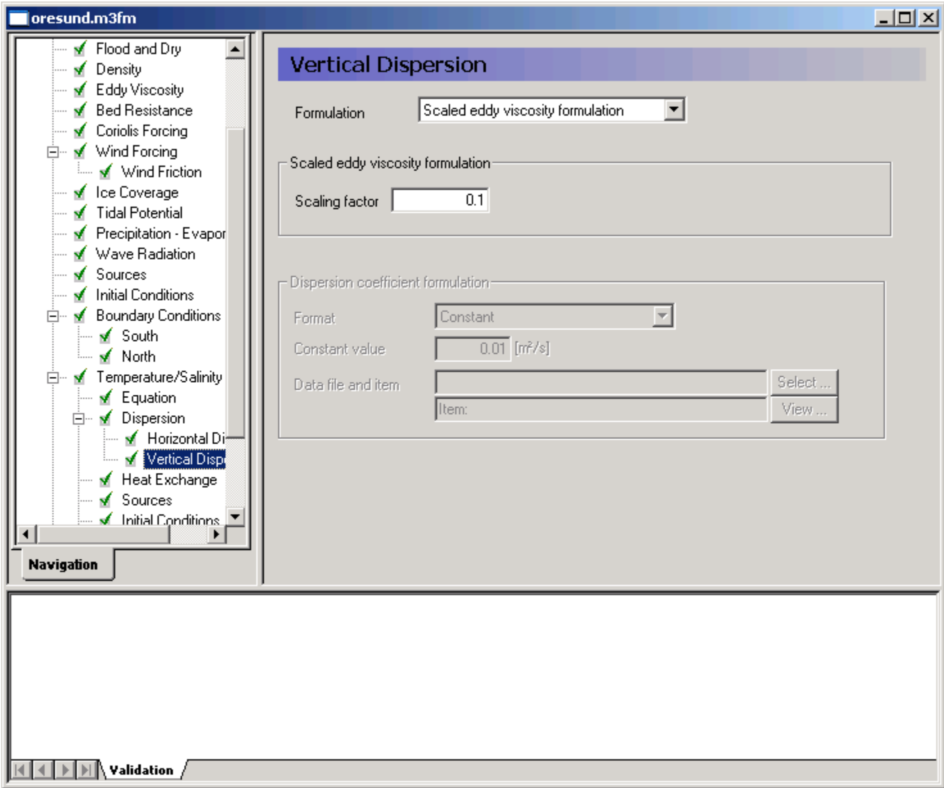


Figure 6.20 MIKE 3 Flow Model FM: Vertical Dispersion in the Temperature/Salinity module

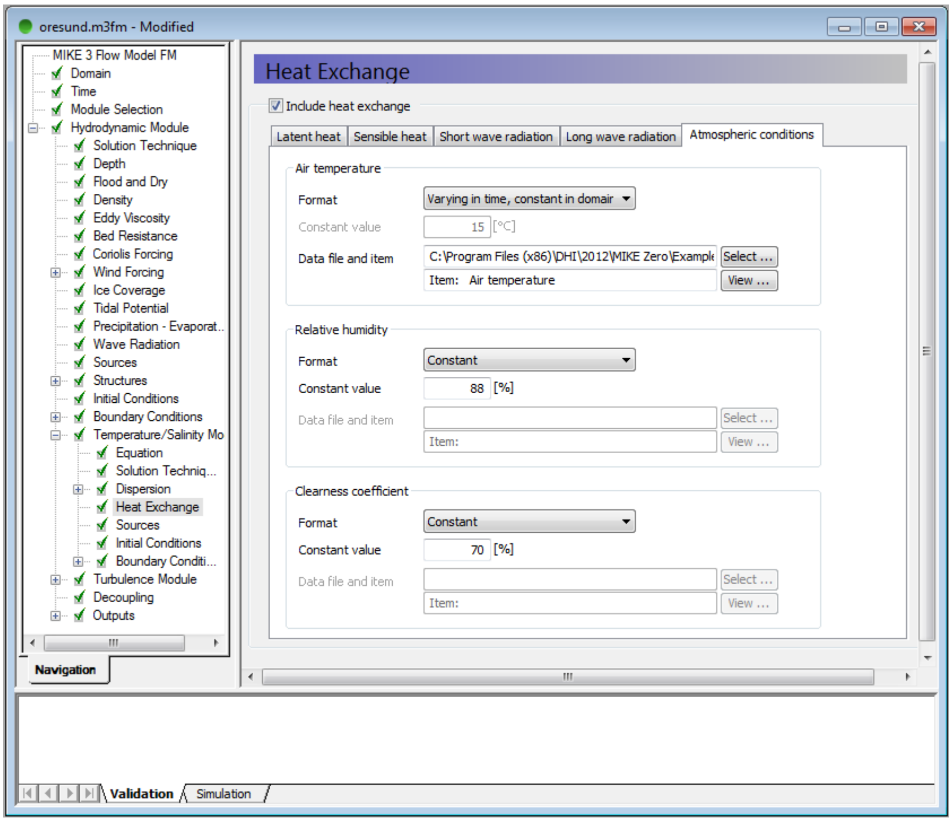


Figure 6.21 MIKE 3 Flow Model FM: Heat Exchange in the Temperature/Salinity module

When starting the simulation from scratch it takes approximately one to two weeks to warm up the salinity and temperature distributions in the Øresund model. Therefore, the results of salinity and temperature for the warm up period and the first part of Period 1 cannot be expected to be right. In simulation period 2 (after 9 days' simulation), the salinity and temperature distributions should be quite 'hot'. This example has already been warmed up for the warm up period and Period 1 and the initial conditions are supplied with this example, so you can start the example for Period 2 with initial conditions as if the model had been running for a period of 9 days.

In the warm up period, the initial conditions are specified as constant of 15°C for Temperature, and a type 3 data file: salinity_sept_6_1997.dfs3 is used for the salinity, see Figure 6.22.

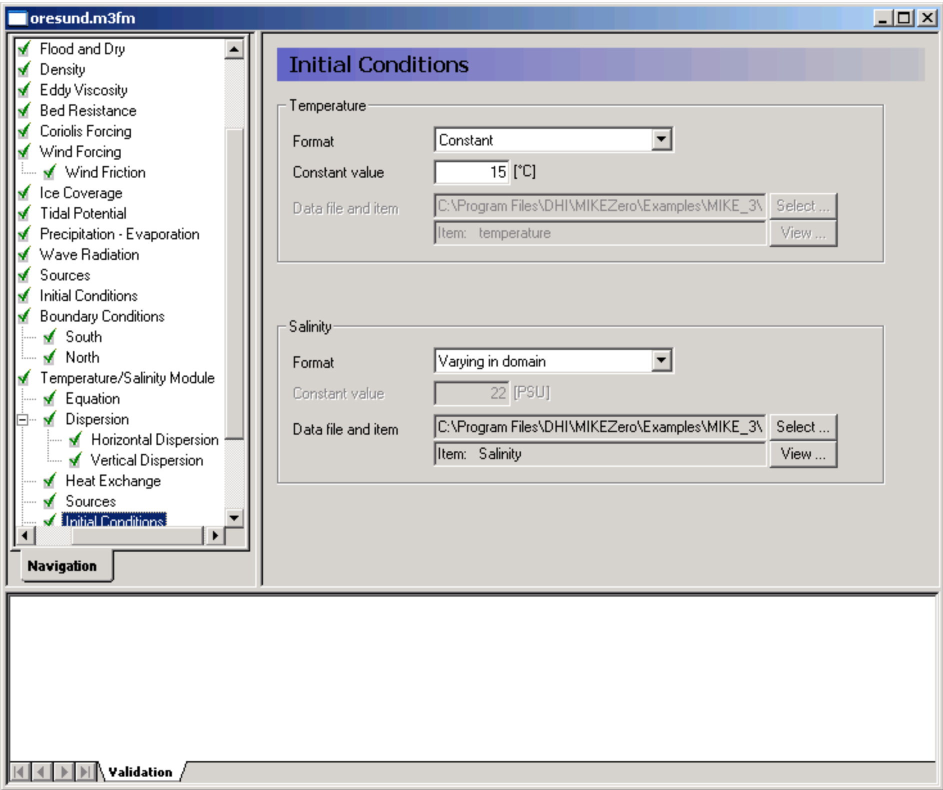


Figure 6.22 MIKE 3 Flow Model FM: Initial Conditions in the Temperature/Salinity module in the Warm up simulation

The file salinity_sept_6_1997.dfs3 is a very rough description of the horizontal distribution of salinity in Øresund in the beginning of September 1997. The dfs3 file was generated from a result file and measured values were interpolated into it. It is based on measurements and it does not contain information about the stratification and the vertical variation.

In the Boundary Conditions dialogue, the boundaries for temperature and salinity should be specified for the boundary names, which were specified in the Domain dialogue, see Figure 6.4. Enable the use of the grid series files that were generated in Chapter 4 by selecting the boundary data format 'Varying in time and along boundary'.

For the south boundary, select the files 'temperature_south.dfs2' and 'salinity_south' for the temperature and salinity at the boundary, respectively.

For the north boundary select the files 'temperature_north.dfs2' and 'salinity_north' for the temperature and salinity at the boundary, respectively.

Please note that for the north boundary, the contents of the dfs2 file are expressed in a west-east direction, i.e. opposite to the default direction of the north boundary. Hence the interpolation type in space must be defined as being in Reverse order.

Use a soft start with a sinus variation and a time interval of 7200 seconds. For the temperature, use a reference value of 15º Celsius, and for the salinity, use a reference value of 22 PSU.

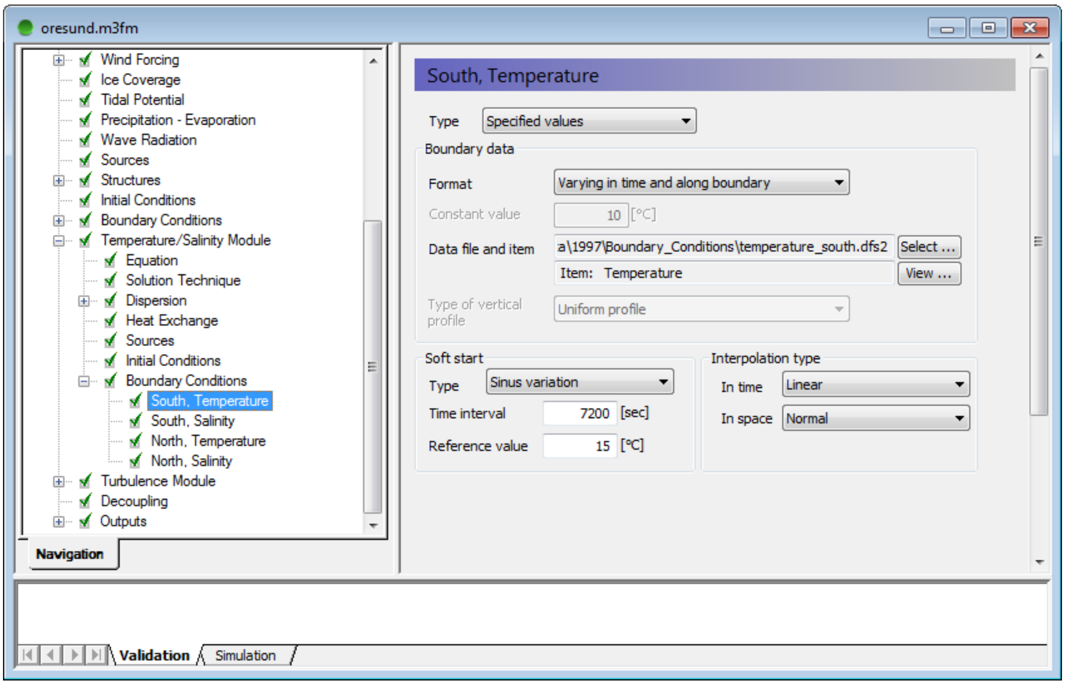


Figure 6.23 MIKE 3 Flow Model FM: Boundary conditions for temperature at the south boundary in the Temperature/Salinity module

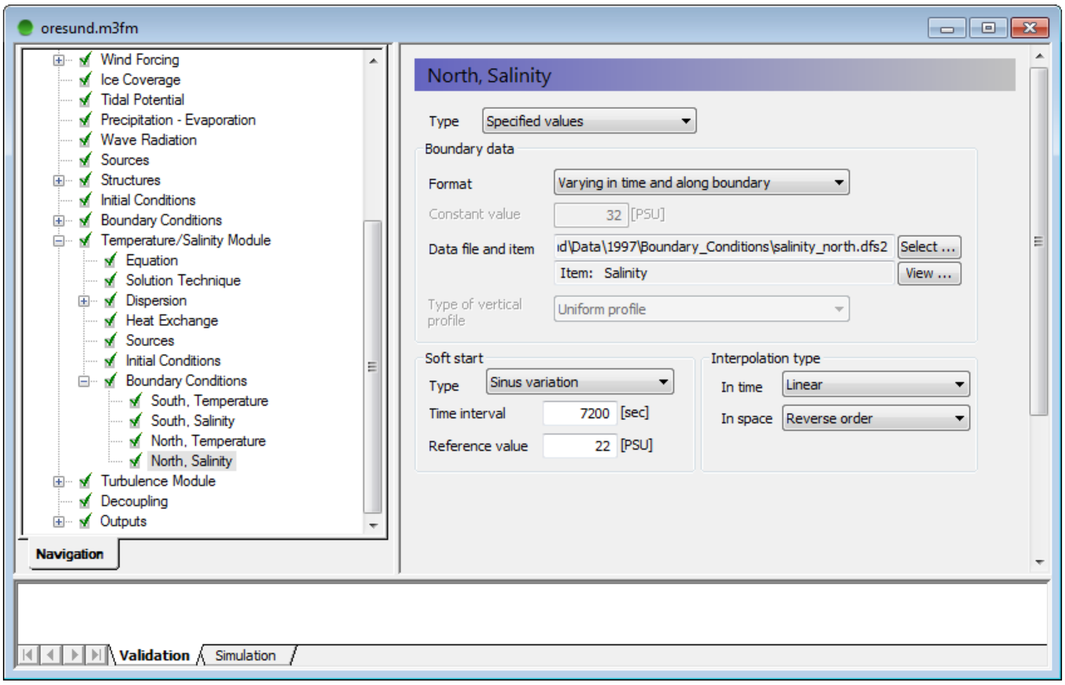


Figure 6.24 MIKE 3 Flow Model FM: Boundary conditions for salinity at the north boundary in the Temperature/Salinity module

On the Equation dialog for the Turbulence Module, the k-epsilon formulation is chosen for Model type. The parameters for the model are kept as default values, see Figure 6.25 and Figure 6.26.

Use the same Solution Technique as in the Hydrodynamic module.

The boundary conditions for turbulence are set as constants, using the default values.

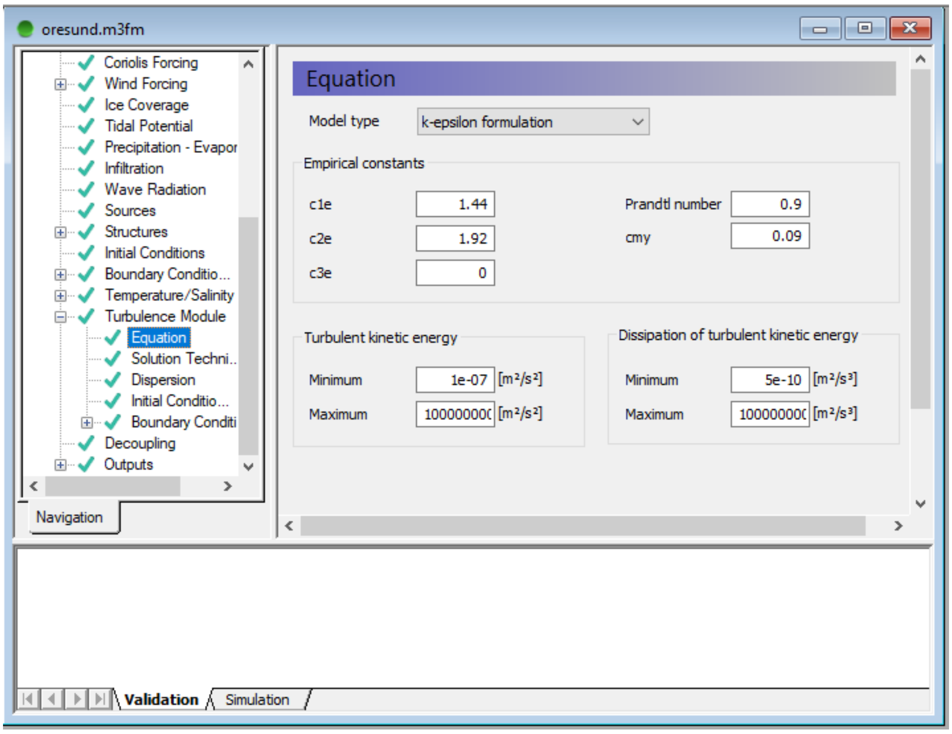


Figure 6.25 MIKE 3 Flow Model FM: Turbulence Equation specifications

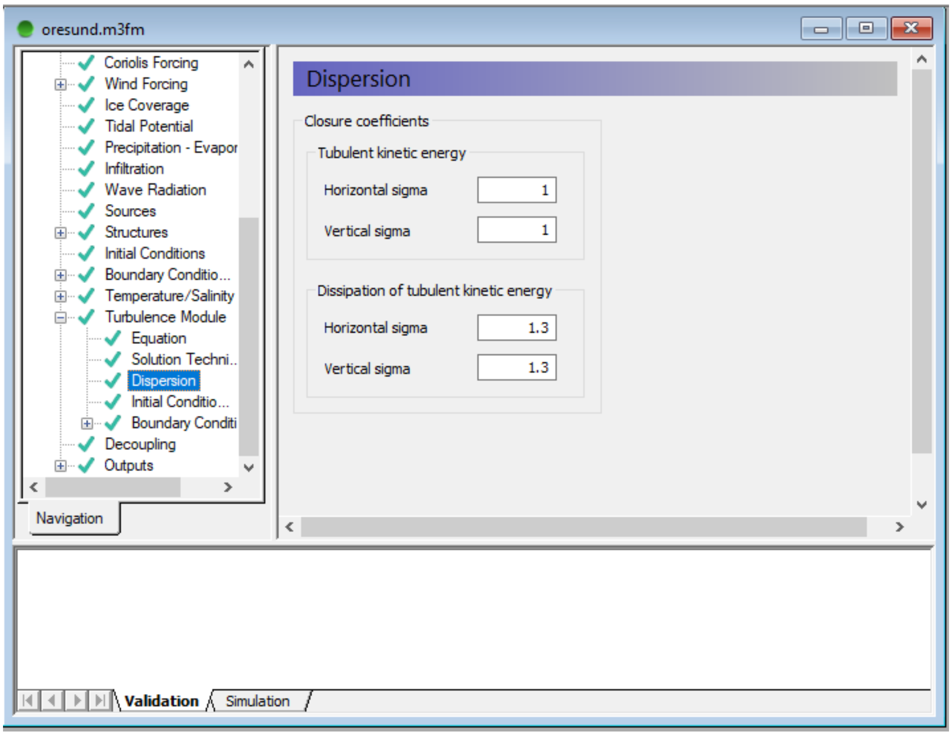


Figure 6.26 MIKE 3 Flow Model FM: Turbulence Dispersion specifications

Decoupling is not included in the model.

The Model Output is divided into two groups:

- 3D variables
- 2D variables

2D variables can be specified as Point Series, Line Series or Area Series. 3D variables can be specified as Point Series, Line Series or Volume Series. See example of 3D output in Figure 6.27.

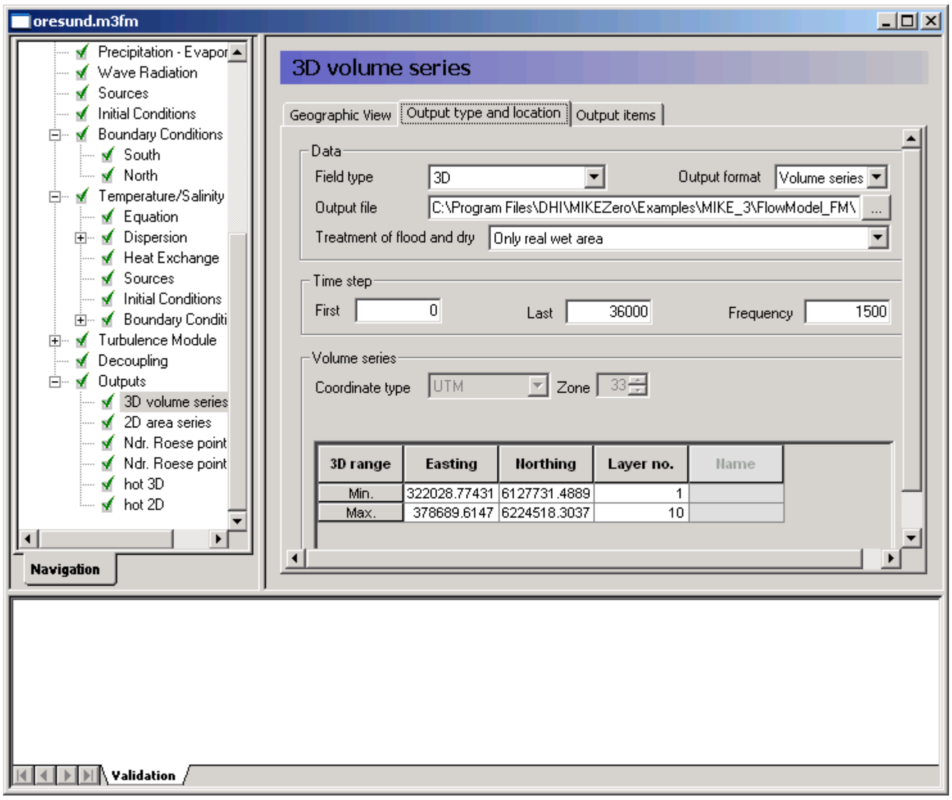


Figure 6.27 MIKE 3 Flow Model FM: Results can be specified as point, line or area series

Specify 6 output files:

1. Volume Series and specify the resulting output file name. Specify the file name: 3D_flow.dfsu.
2. Area Series and specify the resulting output file name. Specify the file name: 2D_flow.dfsu.
3. Time Series at Ndr. Roese: 3D_ndr_roese.dfs0
4. Time Series at Ndr. Roese: 2D_ndr_roese.dfs0
5. A Hot Volume Series (only the last time step) and save in the Initial_Conditions folder.
6. A Hot Area Series (only the last time step) and save in the Initial_Conditions folder.

Make sure the required disk space is available on the hard disk. Reduce the output size for the Area Series and the Volume Series to a reasonably amount by selecting an output frequency of 3 hours. As the time step is 7.2 s then the specified output frequency is 10800/7.2 = 1500. Default, the full area is selected for both Area Series and Volume Series.

The output files include a Point series at the calibration station at Ndr. Roese, see the position in Table 6.2. 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.

Table 6.2 Measurements at Ndr. Roese

Station	Data Files	Position	
		Easting (m)	Northing (m)
Ndr. Roese	salinity_ndr_roese_2.9.txt	354950	6167973
	salinity_ndr_roese_5.9.txt	354950	6167973
	salinity_ndr_roese_10.6.txt	354950	6167973
	temperature_ndr_roese_2.9.txt	354950	6167973
	temperature_ndr_roese_5.9.txt	354950	6167973
	temperature_ndr_roese_10.6.txt	354950	6167973
	u_velocity_ndr_roese_3.0.txt	354950	6167973
	u_velocity_ndr_roese_6.0.txt	354950	6167973
	u_velocity_ndr_roese_10.5.txt	354950	6167973
	v_velocity_ndr_roese_3.0.txt	354950	6167973
	v_velocity_ndr_roese_6.0.txt	354950	6167973
	v_velocity_ndr_roese_10.5.txt	354950	6167973
	waterlevel_ndr_roese.txt	354950	6167973
		354950	6167973

Pick the 3D parameters to include in the output files as shown in Figure 6.28 and the 2D parameters as shown in Figure 6.29.

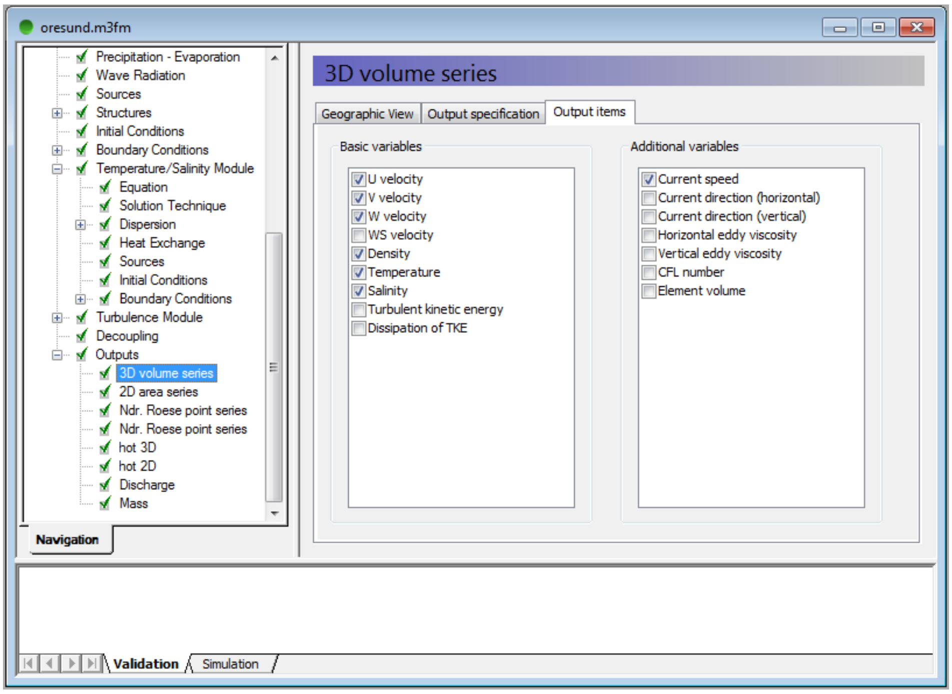


Figure 6.28 MIKE 3 Flow Model FM: The 3D Output Parameters specification

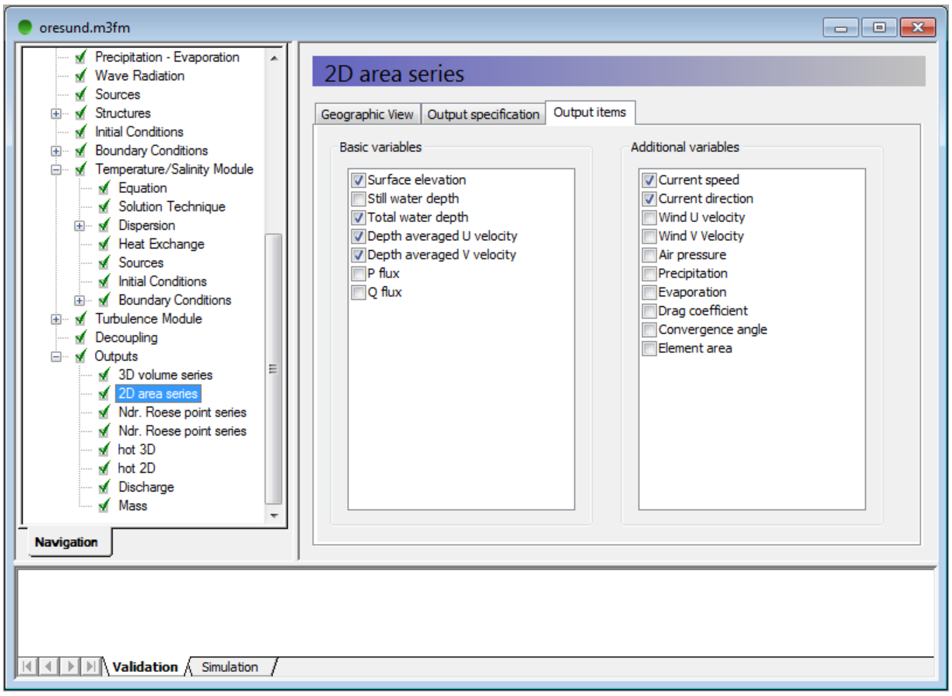


Figure 6.29 MIKE 3 Flow Model FM: The 2D Output Parameters specification

Now we are ready to run the MIKE 3 Flow Model FM.

The specification files for this example have already been made:

.\Calibration_1\Varmup\oresund.m3fm

.\Calibration_1\Period_1\oresund.m3fm

.\Calibration_1\Period_2\oresund.m3fm

After the simulation, you should use the Plot Composer (or Grid Editor or Data Viewer) to inspect and present the results. Two plots are shown below; one top view with salinities and vectors for current speed and direction (Figure 6.30) Another plot is a side view showing a profile along a line through the middle of Øresund from North to South (Figure 6.31).

The Post Processing Tools (data extraction, statistics, etc.) that are 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:

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

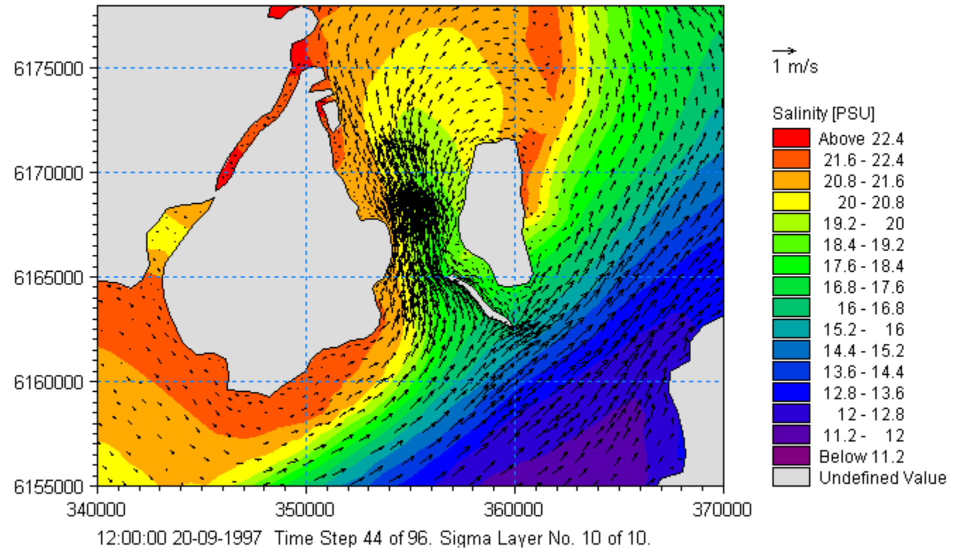


Figure 6.30 Top view with salinities and vectors for current speed and direction

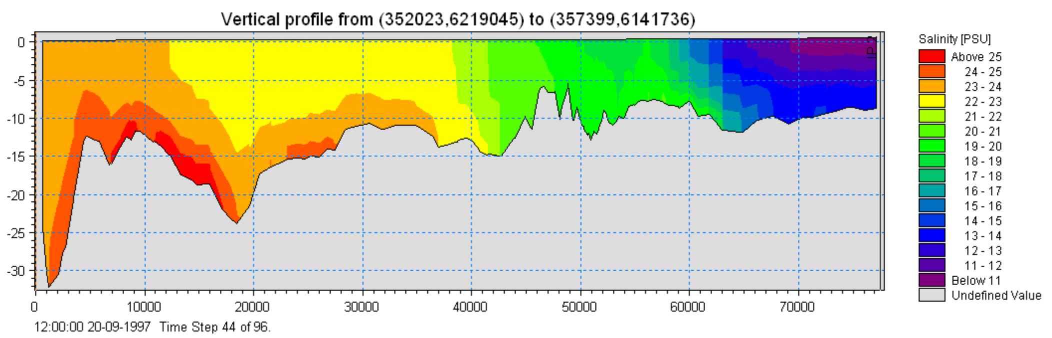


Figure 6.31 Side view showing the salinity profile along a line through the middle of Øresund from North (left) to South (right)

6.2 Model Calibration

In order to calibrate the model we need some measurements inside the model domain. Measurements of water level, current velocities, salinities, temperatures at Ndr Roese are available for different depths, see Table 6.2.

6.2.1 Measurements of water levels, salinities, temperature and currents

Measurements of water levels are given at Ndr. Roese. Import this ASCII file using the Time Series editor. See Figure 6.32.

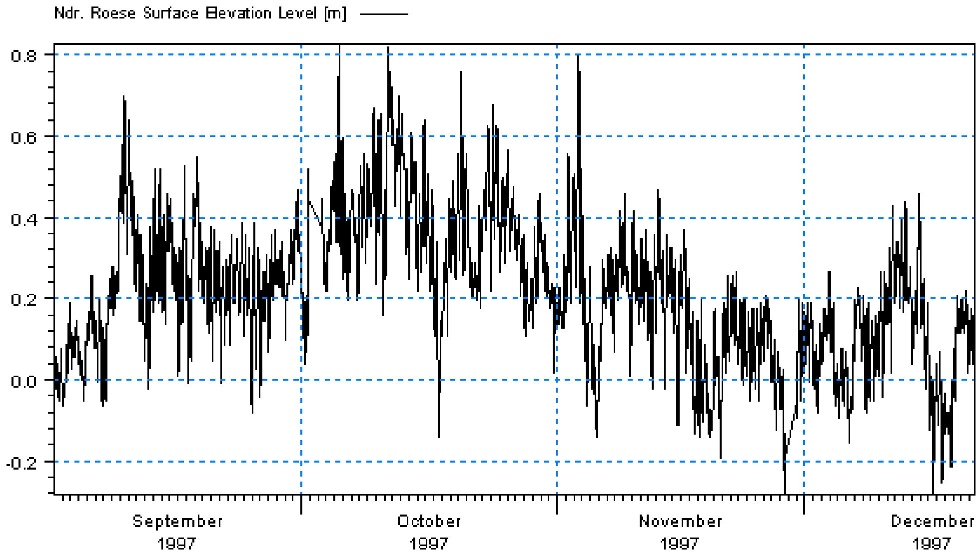


Figure 6.32 Ndr. Roese Measured Water Level

The salinities, temperatures and current velocities are given at 3 depths at station Ndr. Roese, see Table 6.2. Import these files with the Time Series editor.

6.2.2 Compare model result and measured values

Compare the simulated values at Ndr. Roese with measured values. Use the Plot Composer to plot the simulated and measured values of water level, currents, salinity, and temperature for both the Period 1 and Period 2.

The comparison is shown in Figure 6.33 and Figure 6.34for water levels at Ndr. Roese in Period 1 and Period 2. The water levels look reasonable.

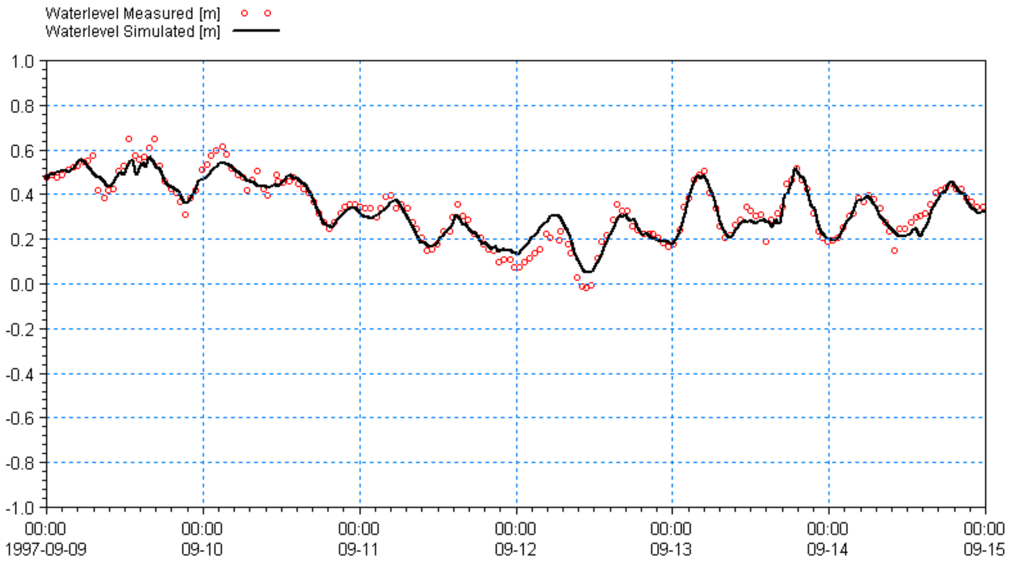


Figure 6.33 Water level comparison at Ndr. Roese. Period 1

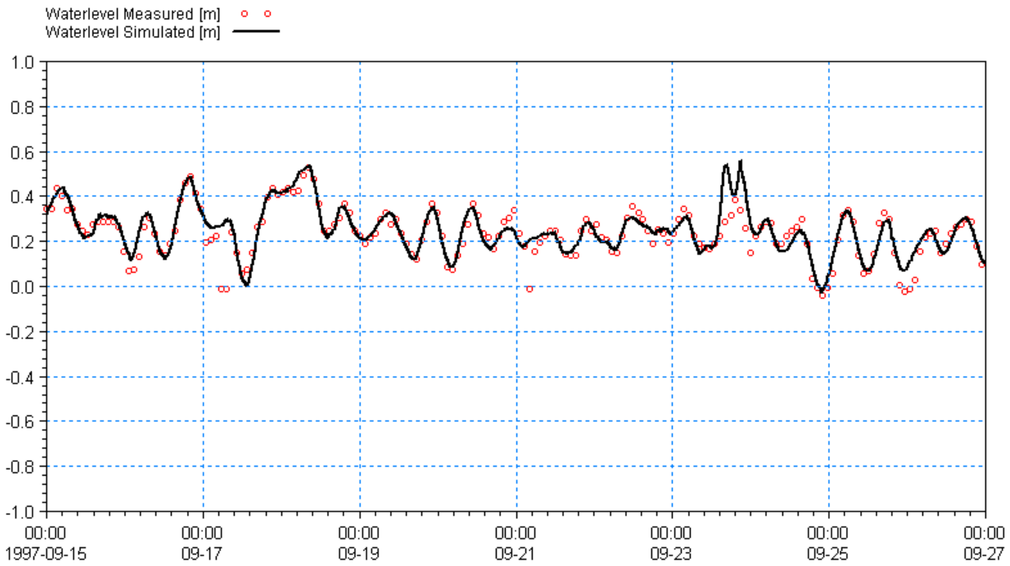


Figure 6.34 Water level comparison at Ndr. Roese. Period 2

The Current comparison for Ndr. Roese is shown in Figure 6.35 and Figure 6.36. For Period 2, the model seems to be 'hot' and the results look good.

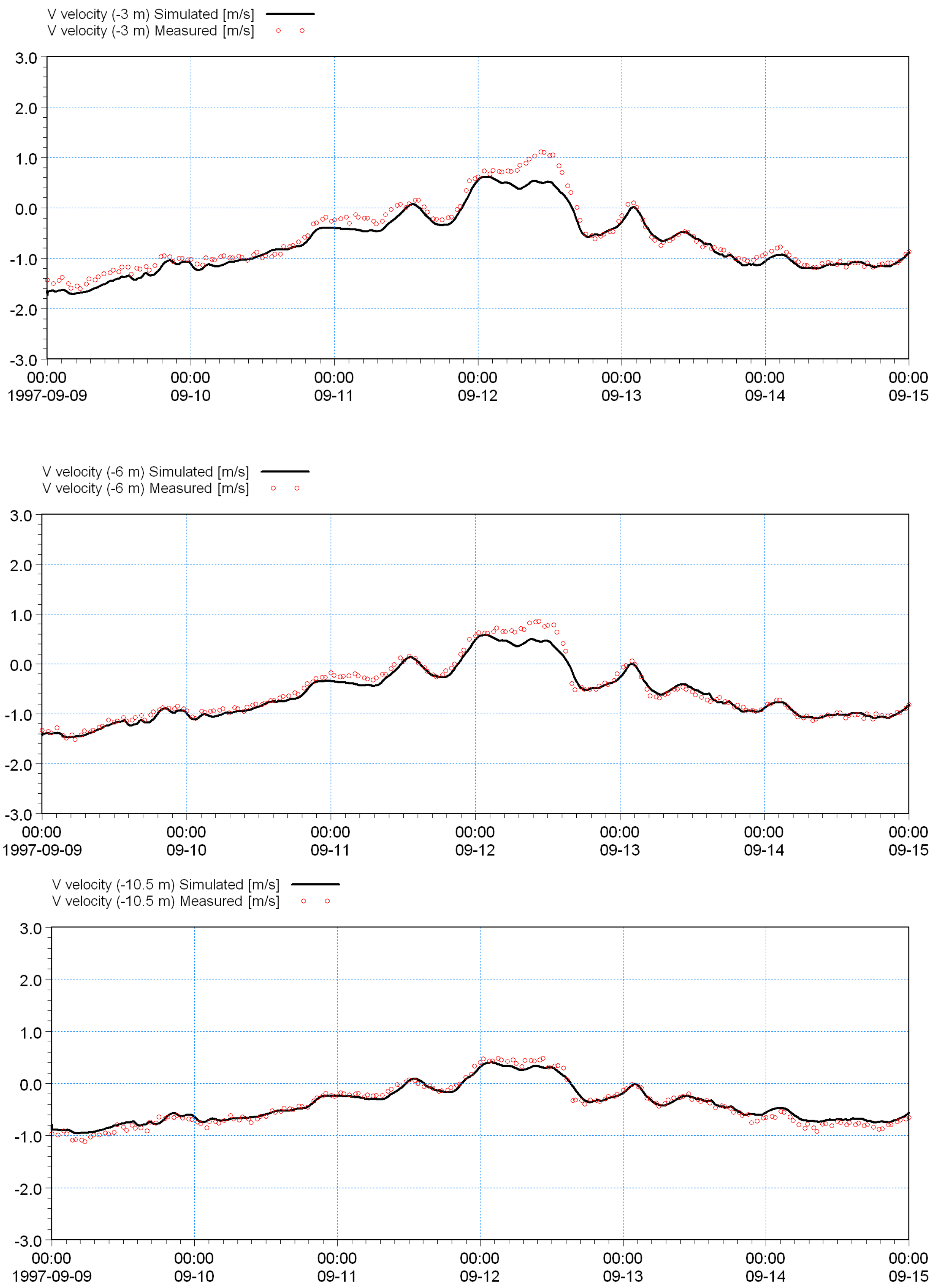


Figure 6.35 Current Velocity North comparison at Ndr. Roese. Period 1

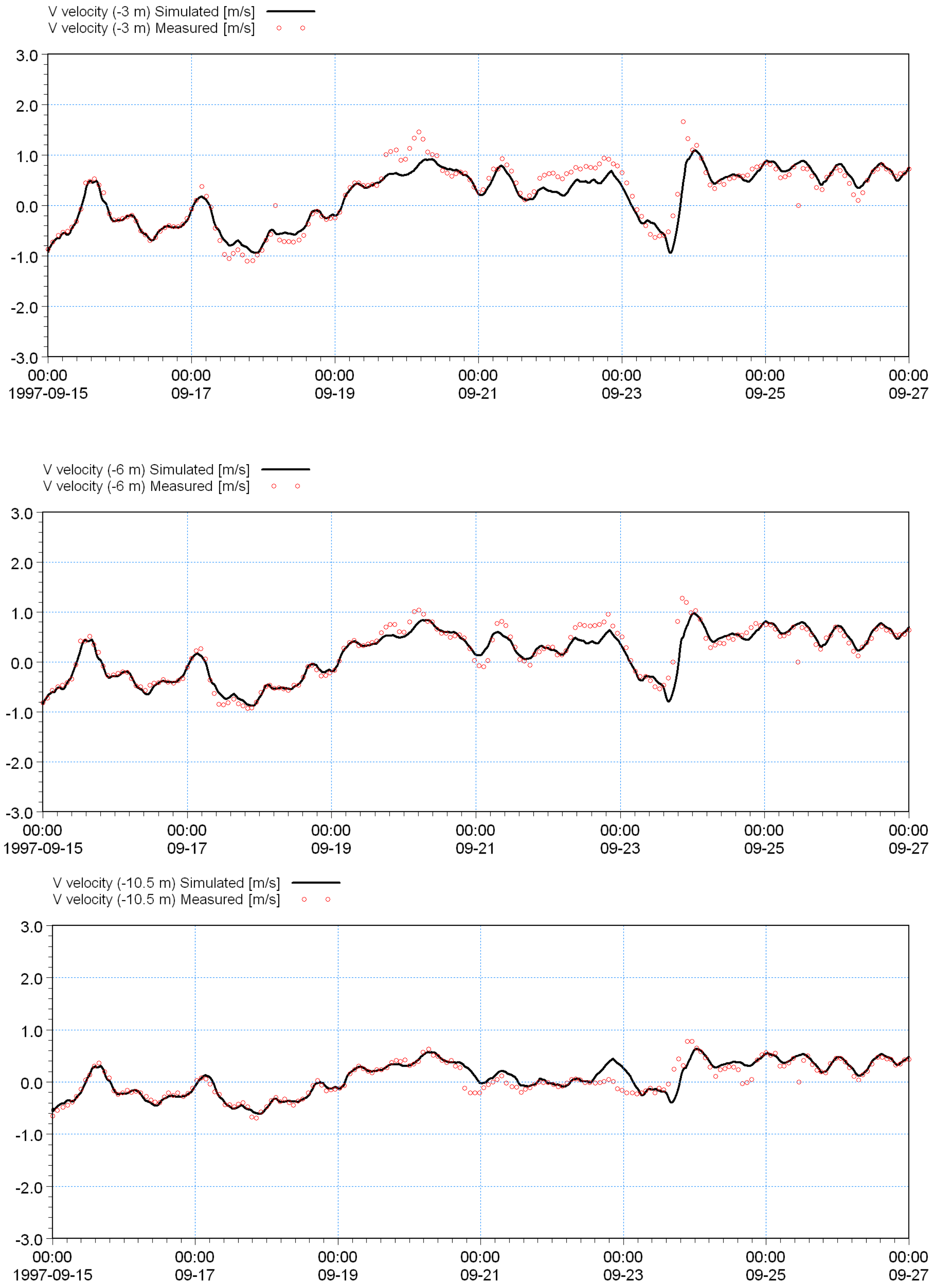


Figure 6.36 Current Velocity North comparison at Ndr. Roese. Period 2

For the temperature results comparison at Ndr. Roese, see Figure 6.37 and Figure 6.38, the model results look good for Period 2, in particular.

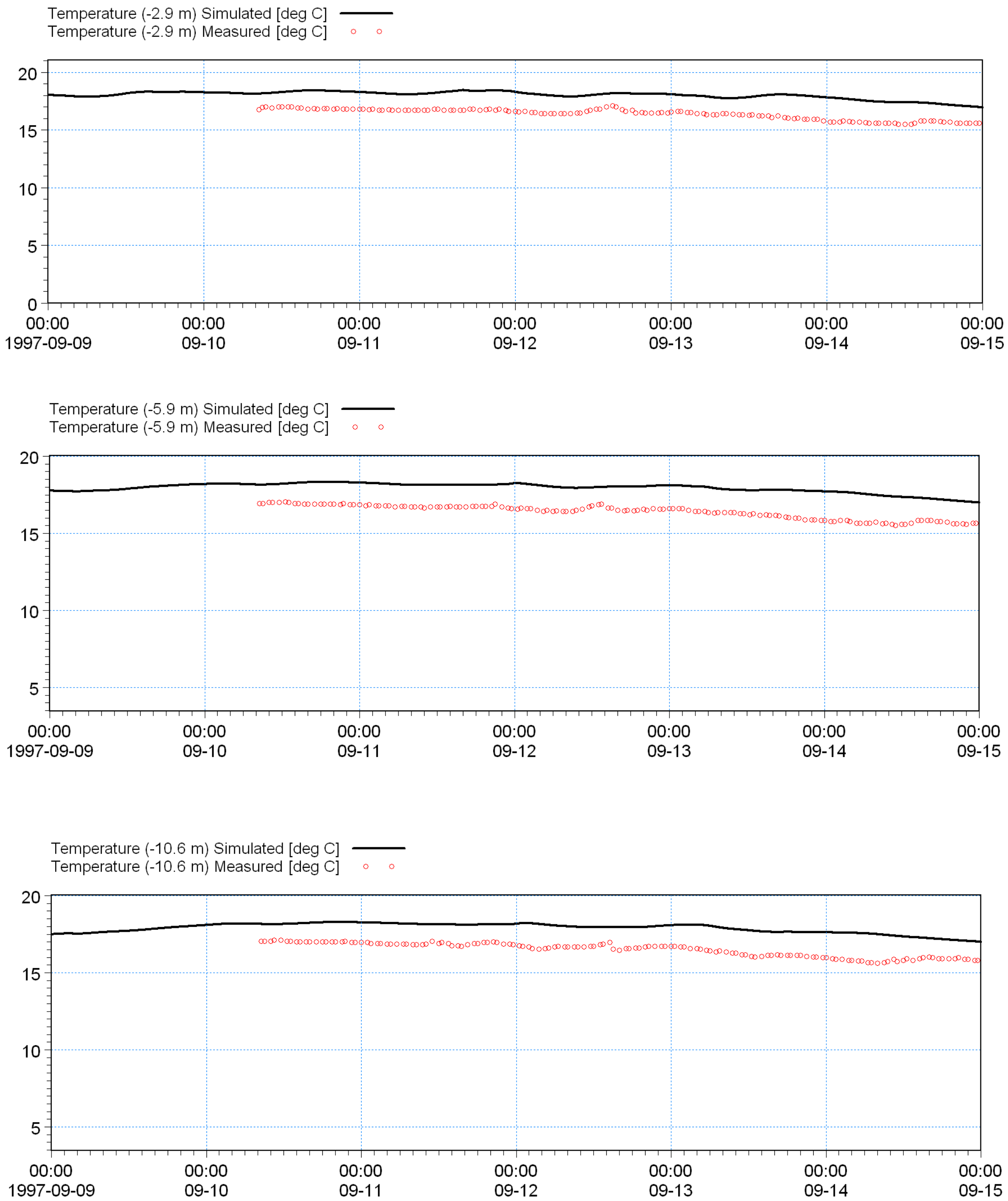


Figure 6.37 Temperature comparison at Ndr. Roese. Period 1

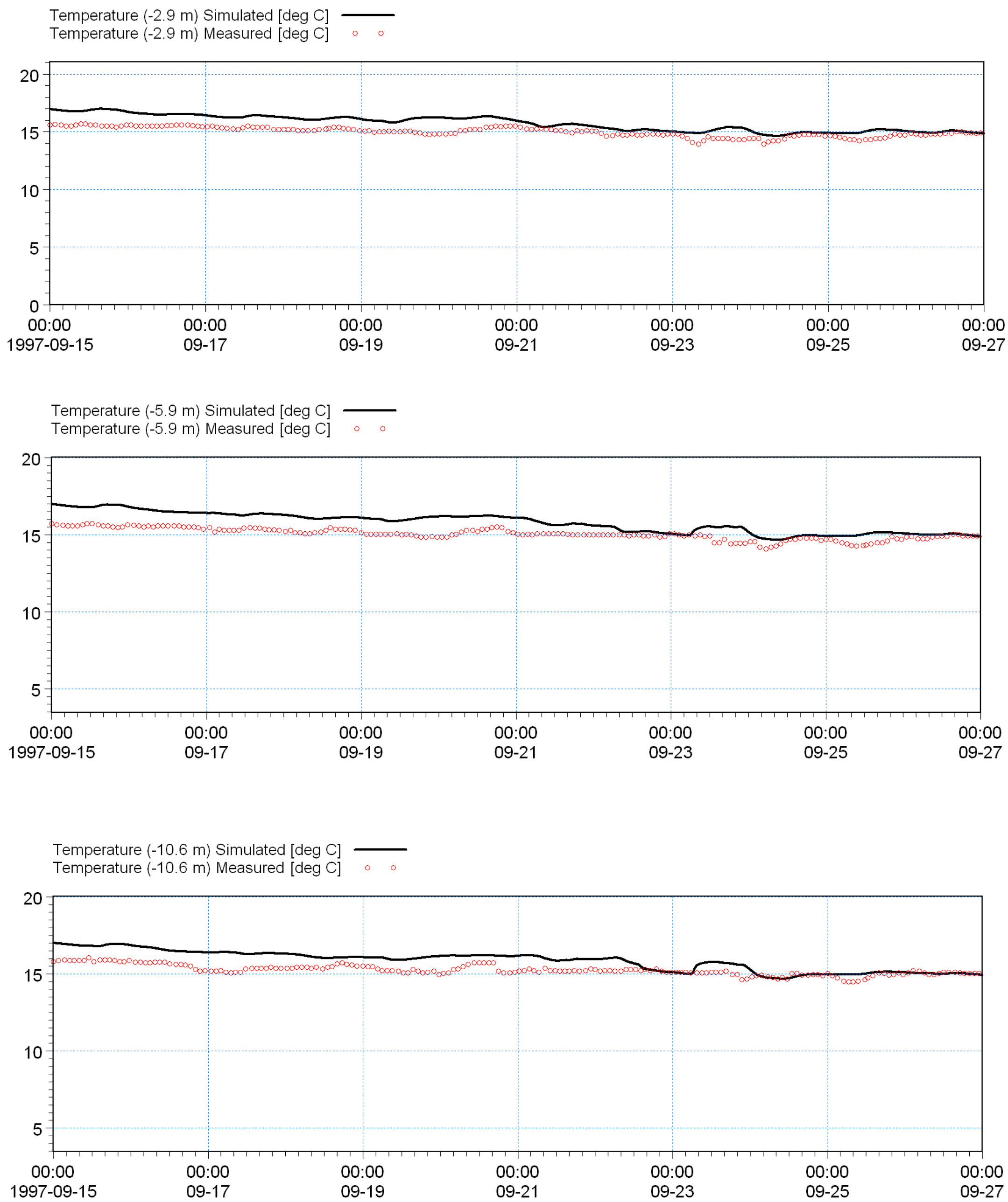


Figure 6.38 Temperature comparison at Ndr. Roese. Period 2

For the salinity results comparison at Ndr. Roese, see Figure 6.39 and Figure 6.40, the model results look reasonable for Period 1. However, for Period 2, where the paradigm changes from a well-mixed water column with salinities around 21 PSU to a well-mixed water column with salinities around 11 PSU, the transition from one system to the other is not described with great accuracy. The end levels are correct though.

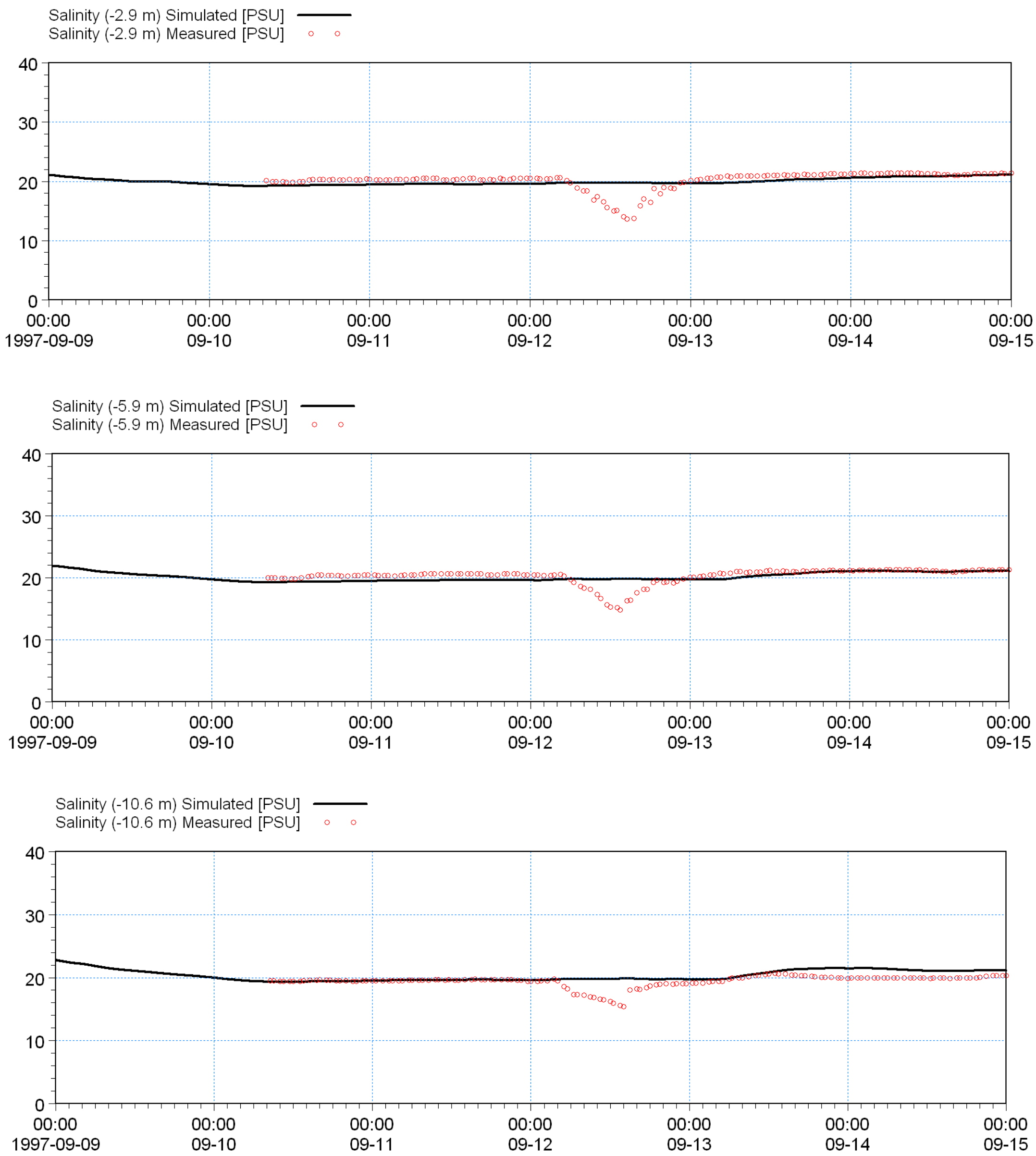


Figure 6.39 Salinity comparison at Ndr. Roese. Period 1

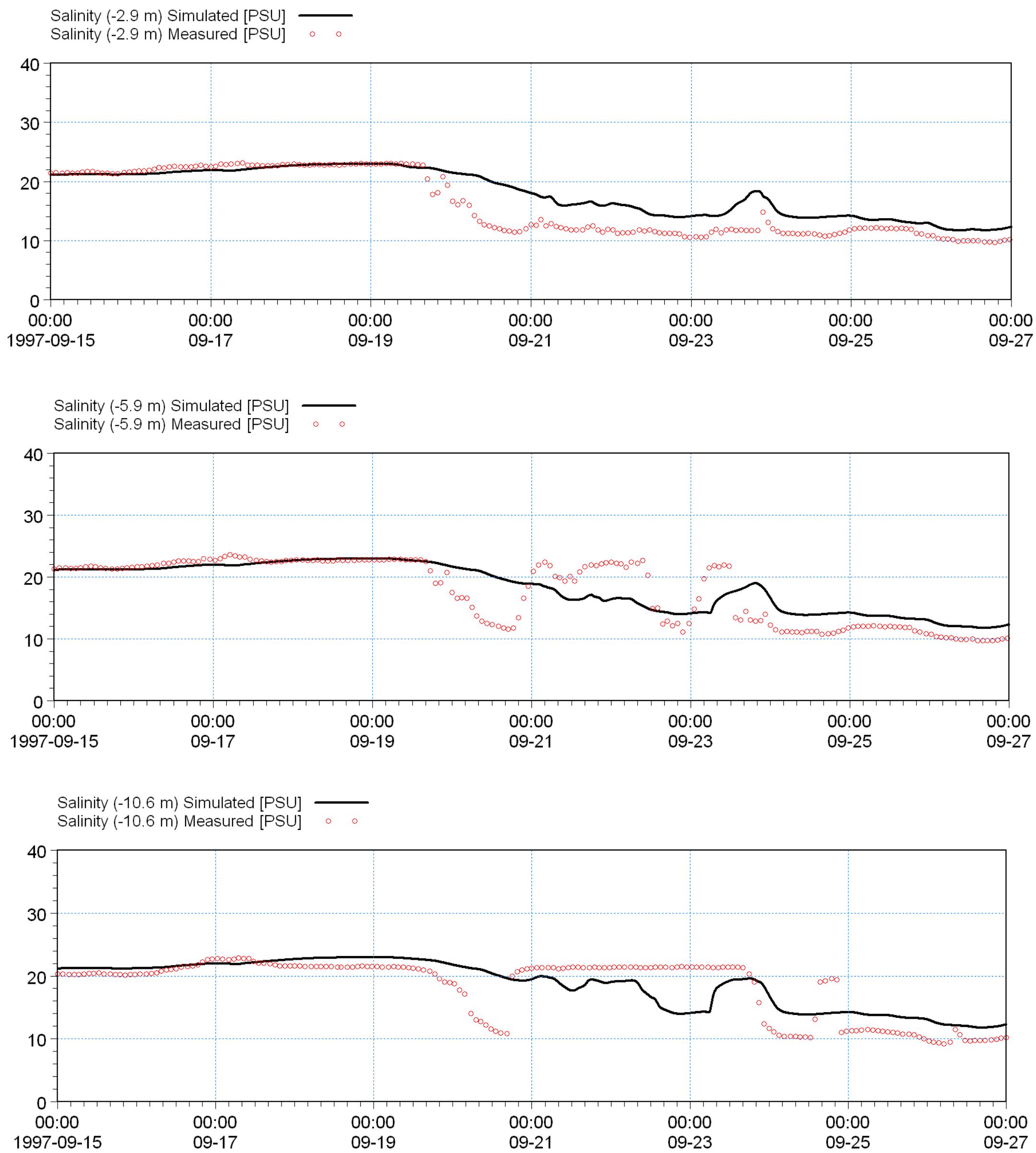


Figure 6.40 Salinity comparison at Ndr. Roese. Period 2

You could try calibrating the Dispersion properties of salinity to get a better agreement of the salinity in Period 2.

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.

Note the way that the folders are organised in the example: You can simply copy the Calibration_2 folder and rename it to Calibration_3. This way the Plot Composer file will also work for the new calibration simulation, because the paths to the files in the Plot Composer file are relative to the present folder. If you are making many simulations this trick can save a lot of time.