DRAFT

Delft3D-WAVE Tutorial

Simulation of short-crested waves with SWAN

User Manual

Version: 4.05
Revision: 78359

17 April 2024

Delft3D-WAVE Tutorial, User Manual

DRAFT

Published and printed by:
Deltares
Boussinesqweg 1
2629 HV Delft
P.O. 177
2600 MH Delft
The Netherlands

telephone: +31 88 335 82 73
e-mail: Information
www: Deltares

For sales contact:
telephone: +31 88 335 81 88
e-mail: Sales
www: Sales & Support

For support contact:
telephone: +31 88 335 81 00
e-mail: Support
www: Sales & Support

Copyright © 2024 Deltares
All rights reserved. No part of this document may be reproduced in any form by print, photo print, photo copy, microfilm or any other means, without written permission from the publisher: Deltares.

Contents

List of Figures vii

1 Tutorials 1
1.1 Introduction . . . 1
1.2 Siu-Lam wave model (1 grid; 3 wave runs) . . . 1
1.2.1 Introduction . . . 1
1.2.2 WAVE Graphical User Interface . . . 2
1.2.3 Saving input data . . . 3
1.2.4 Data groups . . . 3
1.2.5 Description . . . 3
1.2.6 Hydrodynamics . . . 4
1.2.7 Grids . . . 4
1.2.7.1 Computational grid . . . 4
1.2.7.2 Bathymetry . . . 5
1.2.7.3 Spectral resolution . . . 5
1.2.7.4 Nesting . . . 7
1.2.7.5 Hydrodynamics . . . 7
1.2.8 Time frame . . . 7
1.2.9 Boundaries . . . 8
1.2.10 Obstacles . . . 9
1.2.11 Physical parameters . . . 10
1.2.11.1 Constants . . . 11
1.2.11.2 Wind . . . 12
1.2.11.3 Processes . . . 12
1.2.11.4 Various . . . 13
1.2.12 Numerical parameters . . . 13
1.2.13 Output curves . . . 14
1.2.14 Output parameters . . . 15
1.2.15 Additional parameters . . . 16
1.2.16 Executing the scenario . . . 16
1.2.17 Output files of Delft3D-WAVE . . . 17
1.2.18 Visualising results . . . 17
1.3 Nested wave model . . . 18
1.3.1 WAVE Graphical User Interface . . . 18
1.3.1.1 Description . . . 24
1.3.1.2 Hydrodynamics . . . 24
1.3.1.3 Grids . . . 24
1.3.1.4 Time frame . . . 24
1.3.1.5 Boundaries . . . 25
1.3.1.6 Obstacles . . . 25
1.3.1.7 Physical parameters . . . 25
1.3.1.8 Numerical parameters . . . 25
1.3.1.9 Output curves . . . 25
1.3.1.10 Output parameters . . . 25
1.3.1.11 Additional parameters . . . 26
1.3.2 Run and postprocessing . . . 26
1.4 Online-WAVE coupling (including morphology) . . . 26
1.4.1 Introduction . . . 26
1.4.2 Delft3D-FLOW model . . . 28
1.4.2.1 Description . . . 28
1.4.2.2 Domain . . . 28
1.4.2.3 Time frame . . . 28

DRAFT

1.4.2.4 Processes . . . 28
1.4.2.5 Initial conditions . . . 29
1.4.2.6 Boundaries . . . 29
1.4.2.7 Physical parameters . . . 30
1.4.2.8 Numerical parameters . . . 30
1.4.2.9 Operations . . . 30
1.4.2.10 Monitoring . . . 30
1.4.2.11 Additional parameters . . . 30
1.4.2.12 Output . . . 31
1.4.3 Delft3D-WAVE model . . . 31
1.4.3.1 Description . . . 31
1.4.3.2 Hydrodynamics . . . 32
1.4.3.3 Grids . . . 32
1.4.3.4 Time frame . . . 32
1.4.3.5 Boundaries . . . 32
1.4.3.6 Obstacles . . . 33
1.4.3.7 Physical parameters . . . 33
1.4.3.8 Numerical parameters . . . 33
1.4.3.9 Output curves . . . 33
1.4.3.10 Output parameters . . . 33
1.4.4 Run and postprocessing . . . 34
1.4.4.1 Foreground . . . 34
1.4.4.2 Background . . . 34
1.4.4.3 Output files . . . 35
1.5 FLOW-DD and Online WAVE . . . 35
1.5.1 Introduction . . . 35
1.5.2 Delft3D-FLOW models . . . 37
1.5.2.1 Model set-up outside FLOW domain . . . 37
1.5.2.2 Description . . . 37
1.5.2.3 Domain . . . 37
1.5.2.4 Time frame . . . 37
1.5.2.5 Processes . . . 38
1.5.2.6 Initial conditions . . . 38
1.5.2.7 Boundaries . . . 38
1.5.2.8 Physical parameters . . . 38
1.5.2.9 Numerical parameters . . . 38
1.5.2.10 Operations . . . 38
1.5.2.11 Monitoring . . . 39
1.5.2.12 Additional parameters . . . 39
1.5.2.13 Output . . . 39
1.5.2.14 Model set-up inside FLOW domain . . . 39
1.5.2.15 Description . . . 40
1.5.2.16 Domain . . . 40
1.5.2.17 Boundaries . . . 41
1.5.3 Delft3D-WAVE model . . . 41
1.5.3.1 Description . . . 41
1.5.3.2 Hydrodynamics . . . 41
1.5.3.3 Grids . . . 41
1.5.3.4 Time frame . . . 42
1.5.3.5 Boundaries . . . 42
1.5.3.6 Obstacles . . . 42
1.5.3.7 Physical parameters . . . 42
1.5.3.8 Numerical parameters . . . 42
1.5.3.9 Output curves . . . 42

DRAFT

1.5.3.10 Output parameters 43
1.5.4 Run and postprocessing 43
1.5.4.1 Foreground 43
1.5.4.2 Background 45
1.5.4.3 Output files 45

References **47**

DRAFT

DRAFT

List of Figures

1.1 Starting window of the WAVE Graphical User Interface . . . 2
1.2 Data Group: *Description* and sub-window . . . 3
1.3 Data Group *Grids* . . . 4
1.4 **Visualisation Area** window . . . 5
1.5 Sub-data Group *Bathymetry* . . . 6
1.6 Sub-data Group *Spectral resolution* . . . 6
1.7 Data Group *Time frame*. . . . 7
1.8 Data Group: *Boundaries* . . . 8
1.9 Space-varying boundary conditions . . . 9
1.10 Spectral space input parameters . . . 10
1.11 Data Group *Obstacles* . . . 11
1.12 Sub-data Group *Constants* . . . 11
1.13 Sub-data Group: *Wind* . . . 12
1.14 Sub-data Group *Processes* . . . 12
1.15 Sub-data Group *Various* . . . 13
1.16 Data Group *Numerical parameters* . . . 14
1.17 Data Group *Output parameters* . . . 15
1.18 **Output locations** window . . . 16
1.19 Select scenario to run . . . 17
1.20 Top panel: Siu Lam model area near Hong Kong area. Bottom panel: LAND BOUNDARY and curvilinear flow GRID . . . 19
1.21 Top panel: Model BATHYMETRY of Siu Lam model. Bottom panel: BATHYMETRY and GRID of Siu Lam model . . . 20
1.22 Top panel: Computed WAVE HEIGHT pattern on 1 Oct 2005, 18:00. Bottom panel: Computed MEAN WAVE PERIOD pattern on 1 Oct 2005, 18:00 . . . 21
1.23 Top panel: Computed ENERGY TRANSPORT on 1 Oct 2005, 18:00. Bottom panel: Computed DISSIPATION pattern on 1 Oct 2005, 18:00 . . . 22
1.24 Top panel: WAVE vector on 1 Oct 2005, 18:00. Bottom panel: Significant WAVE HEIGHT on 1 Oct 2005, 18:00 . . . 23
1.25 Data Group *Grids – Nesting* window . . . 25
1.26 Data group *Output parameters*, output for computational grids . . . 26
1.27 Development of a spit at the Head of Ameland: the Bornrif . . . 26
1.28 Measured 1989 and 1996 bathymetry . . . 27
1.29 Output restrictions for Online-WAVE . . . 29
1.30 Overview of active processes . . . 29
1.31 Overview of output parameters . . . 31
1.32 Overview of output parameters in Delft3D-WAVE . . . 34
1.33 Execute the Flow-Wave model . . . 35
1.35 Wind drag coefficients in Delft3D-FLOW for outside domain set-up . . . 38
1.36 Numerical parameters in Delft3D-FLOW for outside domain setup . . . 39
1.37 Overview of output parameters of the Delft3D-FLOW model for the outside domain . . . 40
1.38 Wave boundary conditions for 'Boundary North' in the WAVE model set-up . . . 43
1.39 Numerical parameters used in the WAVE model set-up . . . 44
1.40 Overview of out parameters in Delft3D WAVE. . . . 44

DRAFT

DRAFT

1 Tutorials

1.1 Introduction

In these tutorials we will guide you through the process of creating a simple example of a wave computation with SWAN. All the information for a wave computation, also called a scenario, is stored in an input file, also known as Master Definition Wave file (MDW-file). However, before starting this input definition process we want to explain in short the basics of a model definition, the structure of an MDW-file and the basic steps you are supposed to execute.

To execute a wave computation for a specific area we need various kinds of information, such as the extent of the model area, i.e. the boundary at which the incident waves are prescribed, the wind, the bathymetry, geometrical details of the area such as obstacles and a selection of the results that need to be stored for later inspection. Finally, a numerical grid must be defined onto which all location related parameters are defined. So, the basic steps that precede the definition of an input file can be summarised as:

- ◇ Selection of the extent of the area to be modelled.
- ◇ Definition of location and type of wave boundary.
- ◇ Generation of the bathymetry defined on the grid.
- ◇ Definition of many different options, such as wind speed and direction, water level field, current field, number and type of obstacles, etc.

Some of these activities (such as the generation of the bathymetry) must be done before starting the WAVE Graphical User Interface (GUI). In most cases they result in one or more files that are to be located in a project directory to be defined when starting the project. The project directory is also referred to as the working directory. The first two steps are based on experience in solving similar problems and on engineering judgement, no tools are available to support these steps others than (GIS-based) maps and (digitised) charts.

The data of the land boundary, the bathymetry (and the numerical flow grid if present) are stored in separate, so-called attribute files. In the MDW-file only a reference is made to these files instead of including all data in the MDW-file itself. The advantage of using attribute files is that the data can be used in many scenarios but it is stored only once on the system disks. However, the user himself must keep some administration on the use of the same attribute files in different scenarios.

For these tutorials the files, which are created outside the GUI, are provided.

1.2 Siu-Lam wave model (1 grid; 3 wave runs)

1.2.1 Introduction

In this tutorial we provide an existing MDW-file with attribute files for a specific example called ‘Siu-Lam’. The area modelled concerns an estuary called *Siu-Lam* near Hong Kong. We use this basic example to guide you through most of the input definition part of a wave simulation. It is noted that the wind, wave and other parameters that are used do not represent realistic conditions for that area. Therefore the presented results have no practical use.

The input data is located on the directory

<.../tutorials/wave/1_Siu-Lam/input_siu_lam>

and need to be copied first to the directory

DRAFT

<...\tutorials\wave\1_Siu-Lam\>.

The files used in the case of *Siu-Lam* are:

- <hongkong.ldb>: land boundary file
- <siu_lam.grd>: grid file
- <siu_lam.dep>: bathymetry file

The land boundaries <*.ldb> and the (Delft3D-FLOW) grid <*.grd> and bathymetry <*.dep> files can be helpful to design the computational grids for the wave model.

In this tutorial we will use area averaged values for the water level in stead of hydrodynamic results of a Delft3D-FLOW calculation.

1.2.2 WAVE Graphical User Interface

To start the WAVE Graphical User Interface (GUI), execute the following commands (see **??** for details):

- Click the Delft3D-MENU icon on the desktop (PC) or execute the command `delft3d-menu` on the command line (Linux).
- Select the item *Wave*.
- Change to the working directory; in this tutorial <...\tutorial\wave\1_Siu-lam>.
- Select *Wave input* in the **Waves (standalone)** selection window to start the WAVE-GUI.

The start-up window of the WAVE-GUI will be displayed (see Figure 1.1).

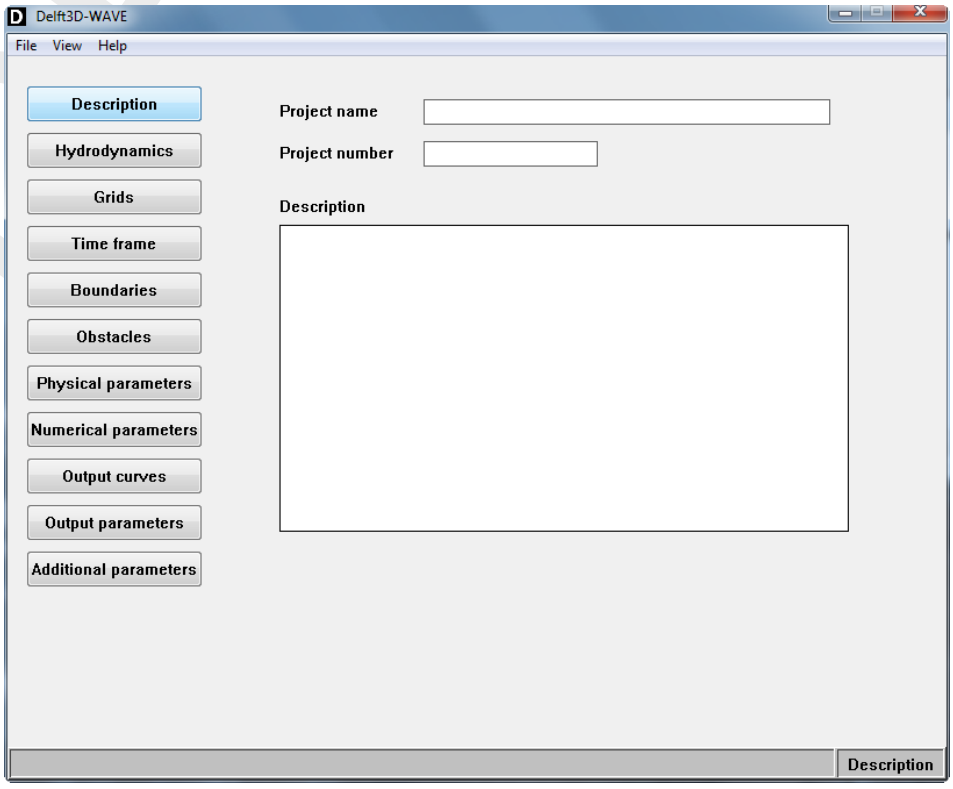

Figure 1.1: *Starting window of the WAVE Graphical User Interface*

Now you are in the main window of the WAVE-GUI. We have not selected an existing MDW-

file, because by doing so we would automatically have loaded all the attribute files referred to in the MDW-file. Instead, you are going to define yourself all the input that is part of the tutorial scenario Siu-Lam.

You are now ready to start defining your own scenario.

1.2.3 Saving input data

Initially, this tutorial may be somewhat tedious to work on. Rather than going on until the end, you may want to stop somewhere halfway the exercise. To prevent that you have to enter all the data again when restarting the exercise, you should save the data you have entered. In this case:

➢ Go to the *File* item in the menu bar of the WAVE-GUI window.
➢ Click *Save As.*

Go to the working directory and save the MDW-file under a new or under the same name (overwrite).

Remark:

◇ Upon saving the data, the GUI checks its integrity and will show you a message window if needed. Adjust your data untill there are no warnings/errors anymore before saving the MDW-file.

1.2.4 Data groups

The Delft3D-WAVE input is divided into several data groups. By selecting a button you get access to a data group. Each of these data groups will be described in the following sections.

1.2.5 Description

In the Data Group *Description* you can identify this MDW-file by giving a comprehensive description of the project, the application domain and the specific selections to be made in this scenario. The description is only used for identification and has no influence on the simulation itself.

➢ Type the description as displayed in Figure 1.2.

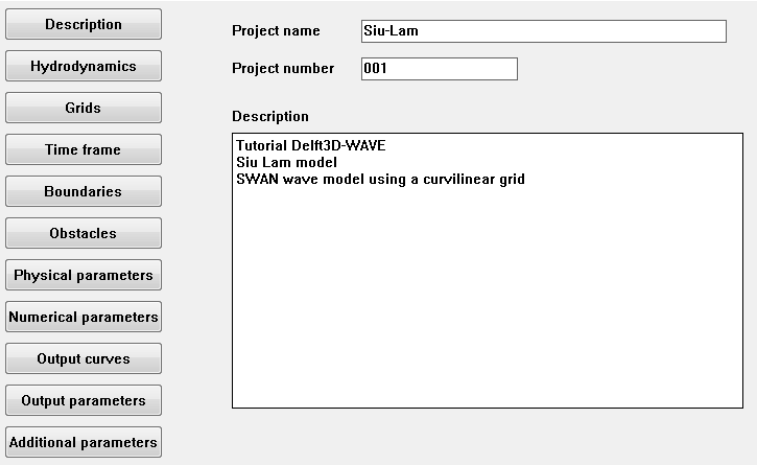

Figure 1.2: *Data Group:* Description *and sub-window*

1.2.6 Hydrodynamics

With Delft3D-WAVE you can run a wave computation that uses results from the FLOW module but also a standalone wave computation. This tutorial will not use FLOW results, in stead area averaged hydrodynamic values will be specified in Data Group *Time frame*.

1.2.7 Grids

In the Data Group *Grids* you define the computational grid(s) with the corresponding bathymetry file(s) (see Figure 1.3). In addition, the spectral grid on which SWAN performs the computation has to be specified per computational grid. When importing more than one grid, the nesting relations should be specified.

The grids can be defined in a common Cartesian co-ordinate system or in a spherical co-ordinate system, described in **??**. The choice of co-ordinate system should already be made when the grid is generated using RGFGRID.

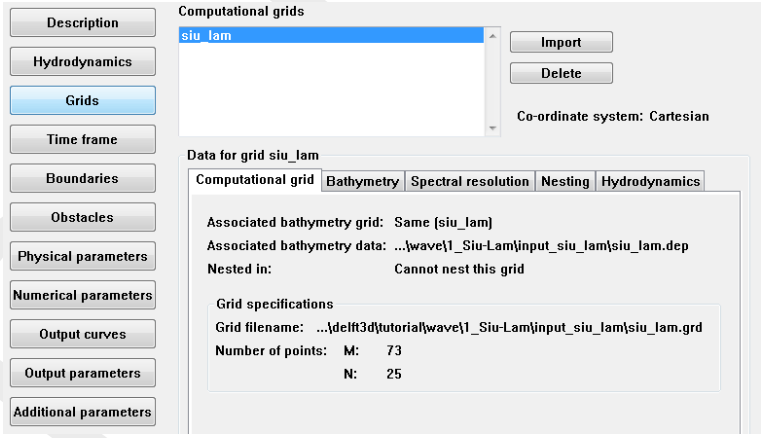

Figure 1.3: *Data Group* Grids

A computational grid is a grid on which SWAN solves the wave action balance equation. Within Delft3D-WAVE, SWAN wave computations can only be made on a curvilinear grid (which can still be rectangular, but created with RGFGRID). Each computational grid has its own corresponding bathymetry file <∗.dep>, created with QUICKIN. This file should be selected under the tab **Bathymetry**.

1.2.7.1 Computational grid

In Delft3D-WAVE you can specify several grids in one run; you have to point out which grid is nested in which. The idea of nesting is to have a coarse grid for a large area and one or more finer grids for smaller areas. The coarse grid computation is executed first and the finer grid computations use these results to determine their boundary conditions. Nesting can be repeated on ever decreasing scales. Additional information on this topic is given in **??** (**??**) and **??** (**??**).

- Select the *Grids* Data Group (Figure 1.3).
- Select *Import* to load a computational grid. Select from the browse screen the desired file: <siu_lam.grd>. Click *Open* to confirm the operation.

The steps above can be repeated when more grids need to be imported.

The grids can be displayed in the **Visualisation Area** window by selecting *View* → *Visualisation Area* from the menubar.

➢ Select *File* → *Open* → *Landboundary file* from the pull down menu in the **Visualisation Area** to open the land boundary. Select from the browse screen the desired file: <hongkong.ldb>.
➢ Click *Open* to confirm the operation. The land boundary will be displayed in the **Visualisation Area** window.
➢ Select *Zoom* → *Zoom Box* from the pull down menu in the **Visualisation Area**. Push and hold the mouse button, drag a box and release the button to zoom in on the concerning area (see Figure 1.4).

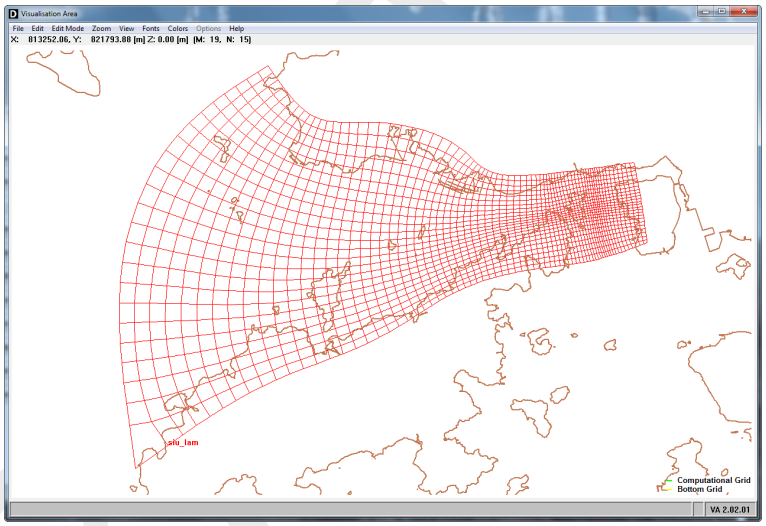

***Figure 1.4: Visualisation Area** window*

1.2.7.2 Bathymetry

The bathymetry data can be defined on the corresponding computational grid, but the bathymetry can also be provided on another grid; this grid must be rectangular. This grid should again be generated using RGFGRID. The bathymetry data should then be provided based on this rectangular grid using QUICKIN.

➢ From the tab **Bathymetry** (see Figure 1.5) click *Select bathymetry data* to open the corresponding bathymetry.
➢ Select from the browse screen the desired file: <siu_lam.dep>. Click *Open* to confirm the operation.

1.2.7.3 Spectral resolution

The computational grids have now been defined for SWAN. In addition to the computational grids in geographical space, SWAN also calculates wave propagation in the spectral space (see **??**). To that end, for each geographical grid the spectral grid has to be specified using the **Spectral resolution** tab.

➢ Click the tab **Spectral resolution** (see Figure 1.6) to edit the spectral grid for each computational grid (i.e. coarse and nested grids).

In the canvas *Directional space* you can define the range and the resolution in directional space for SWAN. In the present example the *Circle* option is considered (this means the full circle of 360° is taken into account). Edit the box *Number of directions* to specify the number of spectral directions.

➢ Enter the value “36” ($\Delta\theta$ = 360°/36).

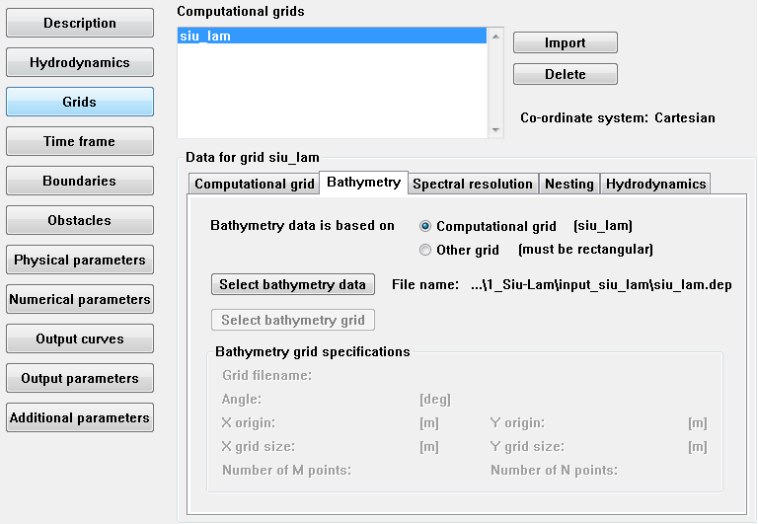

Figure 1.5: *Sub-data Group* Bathymetry

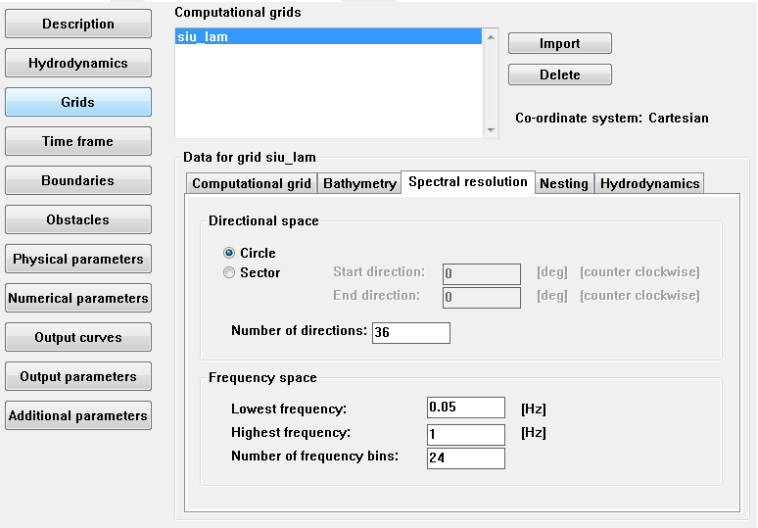

Figure 1.6: *Sub-data Group* Spectral resolution

In the canvas **Frequency space** you can define the resolution and the range in frequency space. The *Numbers of frequency bins* are the numbers of meshes in the frequency-space (one less than the number of grid points in frequency space). This defines the grid resolution in frequency space between the *Lowest frequency* and the *Highest frequency*. This resolution is not constant since the frequencies are distributed logarithmic (see **??**).

◇ For the current computation you can leave the values as default:

Lowest frequency: 0.05 Hz
Highest frequency: 1 Hz
Numbers of frequency bins: 24

◇ Select (if available) the other computational grids and specify the spectral space resolutions for all grids as for the first grid.

Remarks:

◇ If you want to consider only wave directions in a limited directional sector, the option *Sector* may be chosen. The range in Cartesian degrees of this directional sector is

specified giving the *Start direction* and the *End direction*.

◇ SWAN has the option to perform computations on a nested grid. In such cases, the spectral resolution of the nested grid does not need to be equal to the spectral grid of the coarse grid.

1.2.7.4 Nesting

When you want to make nested runs, you first have to import all considered grids. In *Grids* → *Nesting* you must prescribe in which grid the selected grid should be nested. An example of a nested wave model can be found in section 1.3.

Remarks:

◇ The first grid cannot be nested in another one. For this grid, boundary conditions must be specified in the Data Group *Boundaries*.

◇ A grid cannot be nested into itself.

◇ If land points remain dry during the computations, then these points will be ignored for the SWAN computation.

1.2.7.5 Hydrodynamics

For this tutorial the default settings of *Hydrodynamics* will be used.

1.2.8 Time frame

In the Data Group *Time frame* a number of times are specified at which wave computations must be carried out. If the hydrodynamics results of a FLOW simulation are used (see Data Group *Hydrodynamics*) then the time points can be selected at which these results are available. In this tutorial we will specify time steps and use a default (uniform) water level and velocity.

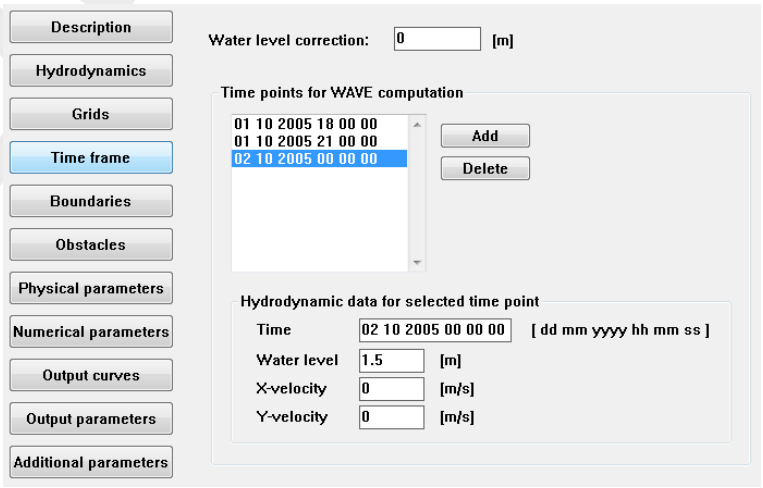

Figure 1.7: *Data Group* Time frame.

➢ Select *Time frame* to enter the Data Group (see Figure 1.7). No *Water level correction* is applied (0 m).

➢ Press *Add* to define a time point for a wave simulation. Specify in the *Time* input field “01 10 2005 18 00 00”.

➢ For this time point enter for the *Water level* “-1.0” and for the *velocities* “0”.

➢ Press *Add* to define a time point for a wave simulation. Specify in the *Time* input field “01 10 2005 21 00 00”.

➢ For this time point enter for the *Water level* “0.0” and for the *velocities* “0”.

DRAFT

- Press *Add* to define a time point for a wave simulation. Specify in the *Time* input field “02 10 2005 00 00 00”.
- For this time point enter for the *Water level* “1.5” and for the *velocities* “0”.

You can click *Delete* to remove a selected time point from the *Time points for WAVE computation* list.

1.2.9 Boundaries

In the Data Group *Boundaries* the incident wave conditions at the boundary of the first computational grid are prescribed (see Figure 1.8). All other computational grids (i.e. the nested grids) obtain their boundary information from other grids.

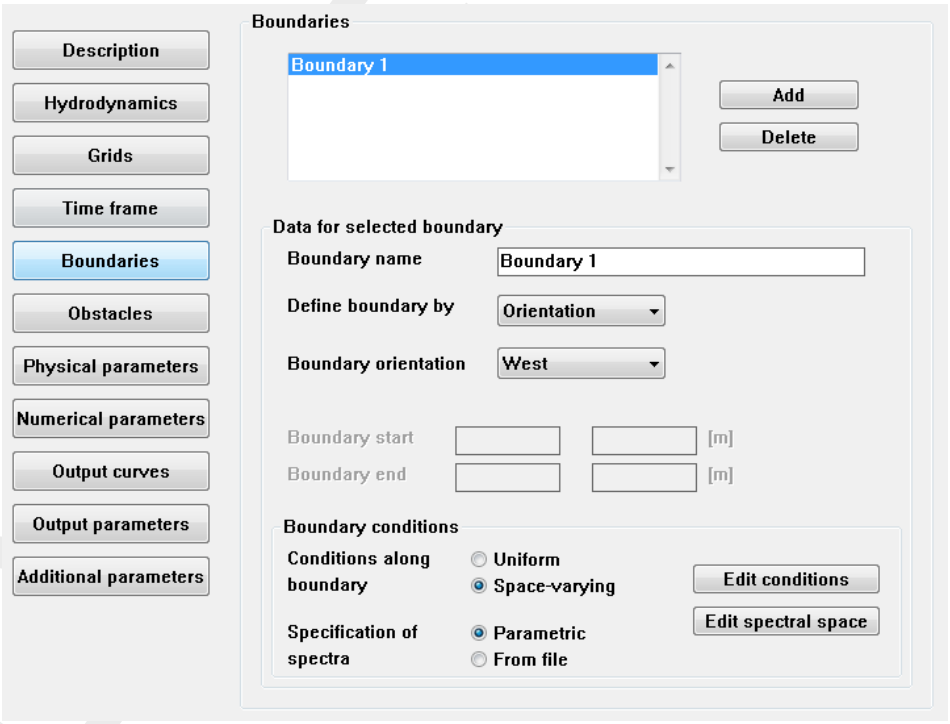

Figure 1.8: *Data Group:* Boundaries

In the SWAN computations, wave boundary conditions may be specified at each side of the computational grid (i.e. maximum of 4 sides). The number of sides at which boundary conditions are provided is zero by default.

The up-wave boundary in the Siu-Lam example is the boundary along the west side of the first computational grid. The wave conditions can vary along this up-wave boundary.

The boundary conditions in SWAN can be defined by specifying the integral wave parameters or can be read from an external file (i.e. results of other model runs or field observations).

- Select the *Boundaries* Data Group.
- Click *Add* to create a boundary. We will use the default name ‘Boundary 1’.
- Set *Define boundary by* to *Orientation* which is default. This means that the boundary is considered along a full side of the computational grid. Specify the boundary orientation in the box *Boundary orientation* indicating on which side the boundary condition is applied. Select boundary orientation: *West*.
- Set the *Conditions along boundary* to *Space-varying* to indicate that the wave conditions vary along the up-wave boundary (see Figure 1.8).

➢ Click *Edit conditions* to specify the incident wave parameters at the selected boundary (note that the mean wave direction has to be in agreement with the convention specified in the sub-data group *Physical parameters – Constants* (i.e. Cartesian or Nautical convention)). Enter the wave parameters for the first section (see Figure 1.9):

Distance from corner point: 1500 m
Significant wave height: 0.0 m
Peak period T_p: 5.0 s
Direction (nautical): 255 degrees
Directional spreading: 4 [-]
Select *Counter clockwise*

➢ Click *Add* for a new section. Enter the wave parameters for the second section:

Distance from corner point: 9000 m
Significant wave height: 1.0 m
Peak period T_p: 5.0 s
Direction (nautical): 255 degrees
Directional spreading: 4 [-]
Select *Counter clockwise*

➢ Click *OK* to close the **Space-varying boundary conditions** window.

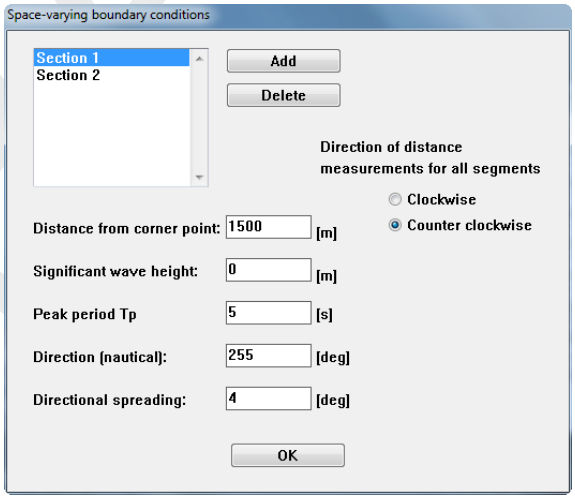

Figure 1.9: *Space-varying boundary conditions*

For the *Specification of spectra*, select *Parametric* to give the boundary conditions in a form of parametric input. To specify the spectral parameters that will be used:

➢ Click on *Edit spectral space*

The window **Spectral Space** will appear. A *JONSWAP* type spectrum will be used with the peak enhancement factor *Peak enh. fact.* set to the default value of "3.3". In the present tutorial the peak period *Peak* and the directional spreading expressed in *Degrees* are considered as input integral waves parameters. Select these options in the present window; see Figure 1.10.

➢ Click *OK* to confirm.

1.2.10 Obstacles

Within the Data Group *Obstacles* you can specify the characteristics of (a line of) sub-grid obstacles. The location of the obstacle is defined by a sequence of corner points of a line. The obstacles interrupt the propagation of the waves from one grid point to the next wherever this

DRAFT

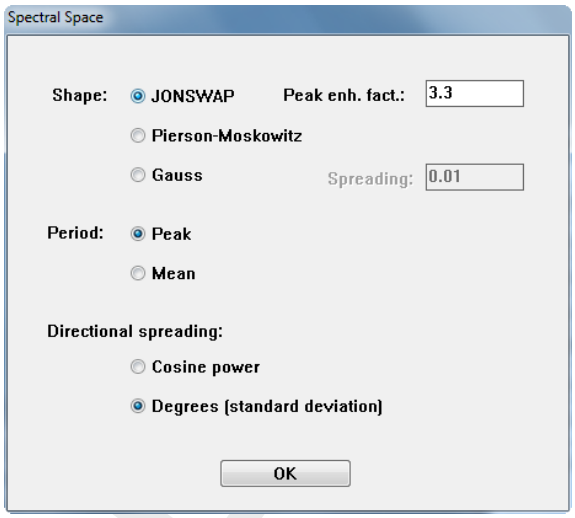

Figure 1.10: *Spectral space input parameters*

obstacle line is located between two neighbouring grid points of the computational grid (the resolution of transmission or blockage is therefore equal to the computational grid spacing).

- Click the Data Group *Obstacles*.
- Select *Add* to specify that an obstacle is present (this is the first obstacle).

Add may be used more than once to define more obstacles.

- Select *Dam* as obstacle type to specify that the transmission coefficient depends on the incident wave conditions at the obstacle and on the obstacle height (which may be submerged). The default values are used for reflection ("no") and for the *Height* of the dam (with respect to the reference level) and the coefficients *Alpha* and *Beta*.
- Select *Add* from the *Obstacle segment* item.
- Enter the co-ordinates of the first corner point of the obstacle in the *X-start* and *Y-start* boxes. Enter x = "814800" [m] and y = "818000" [m].
- Enter the co-ordinates of the second corner point of the obstacle in the *X-end* and *Y-end* boxes (x = "814800" [m] and y = "820000" [m]) and click on any edit box to confirm. The first segment has now been specified, see Figure 1.11.

The button *Add* may be used more than once to include for more segments. You can click *Delete* to remove a selected obstacle or a segment from the list.

1.2.11 Physical parameters

In the Data Group *Physical parameters* you can specify a number of physical parameters. The following sub-data groups are available.

Constants	In this sub-data group you can assign values to some general parameters.
Wind	Here you can specify the wind conditions.
Processes	In this sub-data group you can select the physical processes in SWAN (i.e. type of formulation, dissipation processes, non-linear wave-wave interactions, diffraction).
Various	Here you can switch on or off wave propagation in spectral space and several physical processes in SWAN.

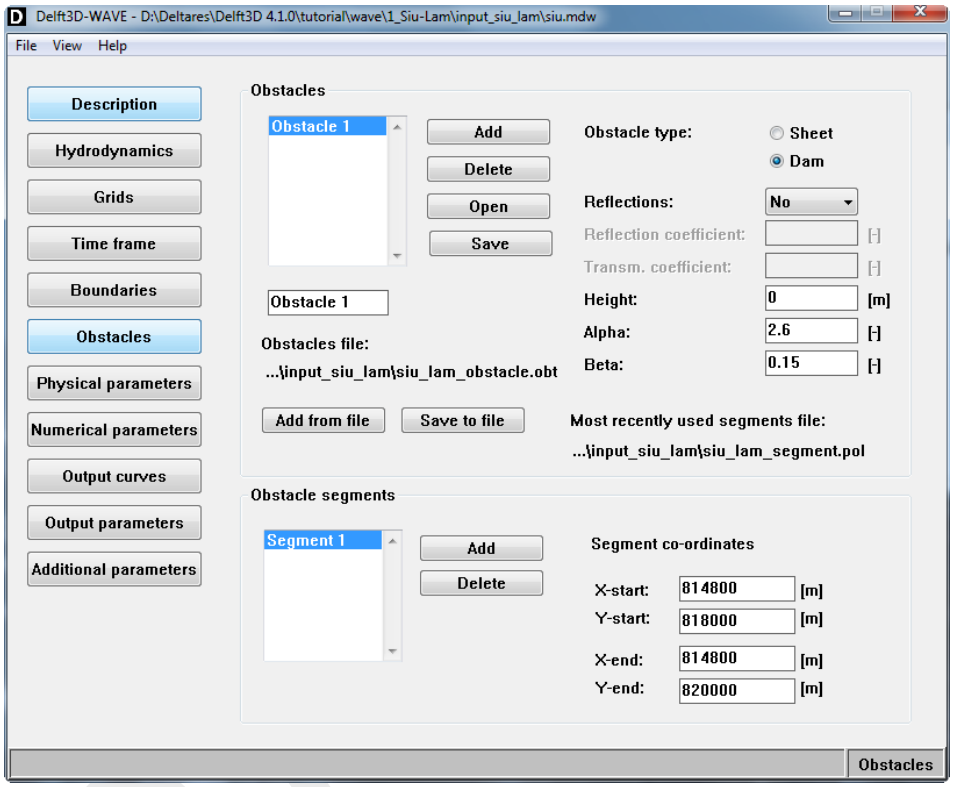

Figure 1.11: *Data Group* Obstacles

1.2.11.1 Constants

➢ Click the Data Group *Physical parameters* to show the sub-data groups.
➢ Select *Constants* in order to assign values to various general input parameters, see Figure 1.12.

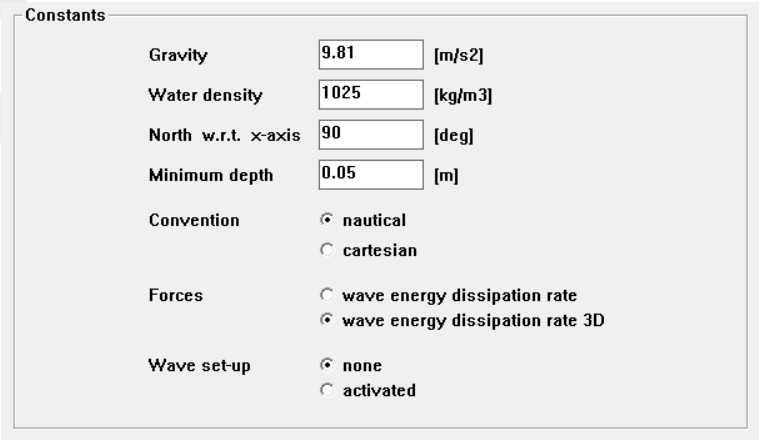

Figure 1.12: *Sub-data Group* Constants

The standard values for the gravitational acceleration *Gravity*, the *Water density*, the direction of *North* with respect to the x-axis and the threshold depth (in m) *Minimum depth* will be used. The *nautical* convention for wind and wave direction (button input and output) will be adopted in this tutorial. *Wave set-up* (within the SWAN model!) is de-activated. The *Forces* will be based on the *wave energy dissipation rate*.

1.2.11.2 Wind

➢ Select *Wind* to specify the wind conditions, see Figure 1.13.

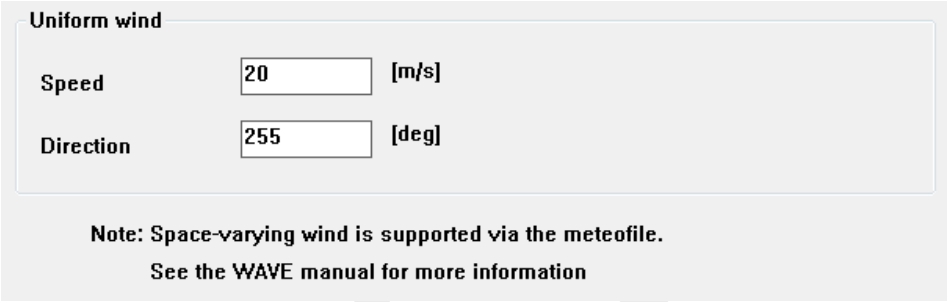

Figure 1.13: *Sub-data Group:* Wind

Here use will be made of a constant wind field (wind speed and direction). The wind direction applied is the same direction as the incident wave direction at the up-wave boundary.

➢ Enter the uniform wind parameters:
Speed: “20” m/s.
Direction: “255” degrees (conform the convention activated: Nautical)
➢ Select the Sub-data Group *Processes*. Next the window in Figure 1.14 is displayed.

1.2.11.3 Processes

In this sub-data group the physical processes to be activated in SWAN can be selected. Type of formulations: you can specify the mode in which SWAN can operate (first-, second-, third-generation mode).

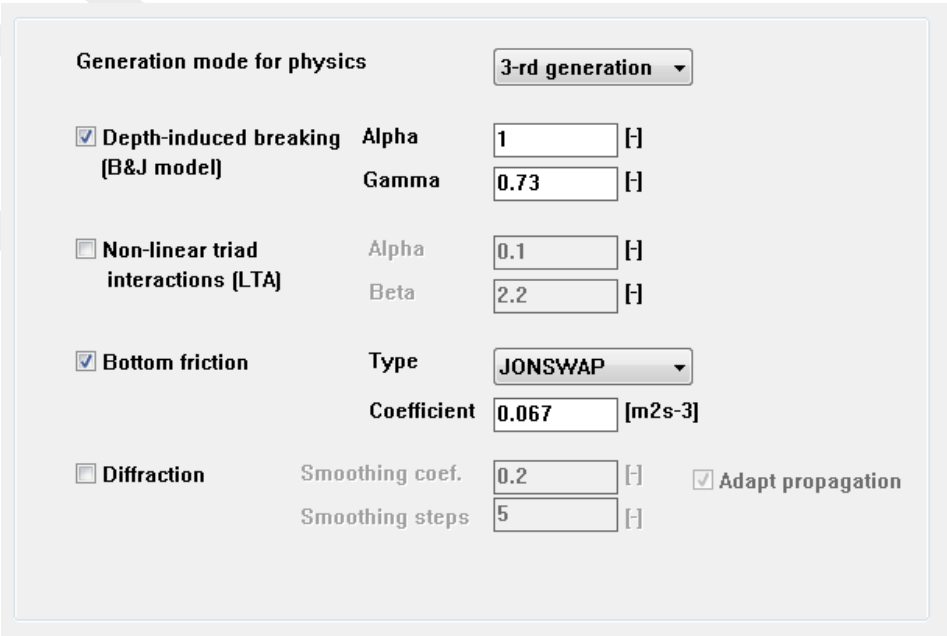

Figure 1.14: *Sub-data Group* Processes

➢ Select *3–rd generation* mode.

This means that SWAN will use third-generation formulations for the representation of the deep water physical processes.

Within the **Depth-induced breaking** sub-window you can activate depth-induced wave breaking using B&J model. Here the default values are used (*Alfa* = 1.0 and *Gamma* = 0.73).

Within **Non-linear triad interactions (LTA)** sub-window you can activate the triad wave-wave interactions based on the *LTA* (i.e. Lumped Triad Approximation, see **??** and **??**) with default values for *Alfa* = 0.10 and *Beta* = 2.2. De-activate the *Non-linear triad interactions (LTA)*.

➢ To activate dissipation by bottom friction, check *Bottom friction* (see **??** and **??**). Select the *JONSWAP* bottom friction formulation with its default value 0.067 m^2/s^3.

Within the **Diffraction** sub-window you can activate diffraction. De-activate *Diffraction*.

1.2.11.4 Various

Within the Sub-data Group *Various* (see Figure 1.15) you can de-activate or activate several physical processes in order to perform, e.g. a sensitivity study. Keep all processes activated.

◇ When wind is present, the quadruplets are activated when using the third-generation mode for physics.

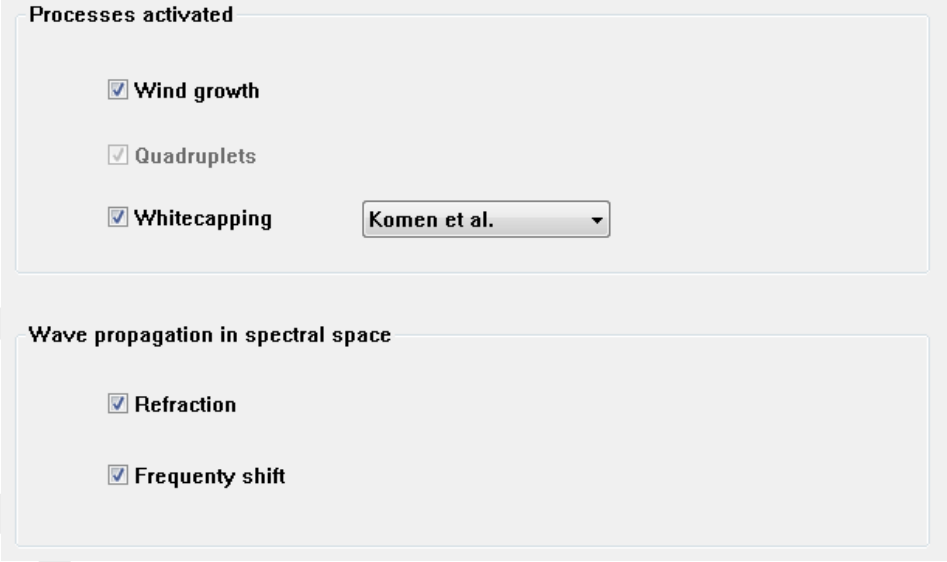

Figure 1.15: *Sub-data Group* Various

Remarks:

◇ For initial SWAN runs, it is strongly advised to use the default values of the model coefficients.

◇ Switching off depth-induced breaking is usually not recommended, since this leads to unacceptably high wave heights near beaches (the computed wave heights 'explode' due to shoaling effects).

1.2.12 Numerical parameters

In the Data Group *Numerical parameters* you can modify parameters that affect the stability and accuracy of the numerical computation.

➢ Click the Data Group *Numerical parameters*: Next the window in Figure 1.16 is displayed.

In the **Spectral space** canvas you can control the amount of diffusion of the implicit scheme in the directional space through the parameter for the *Directional space (CDD)* and frequency

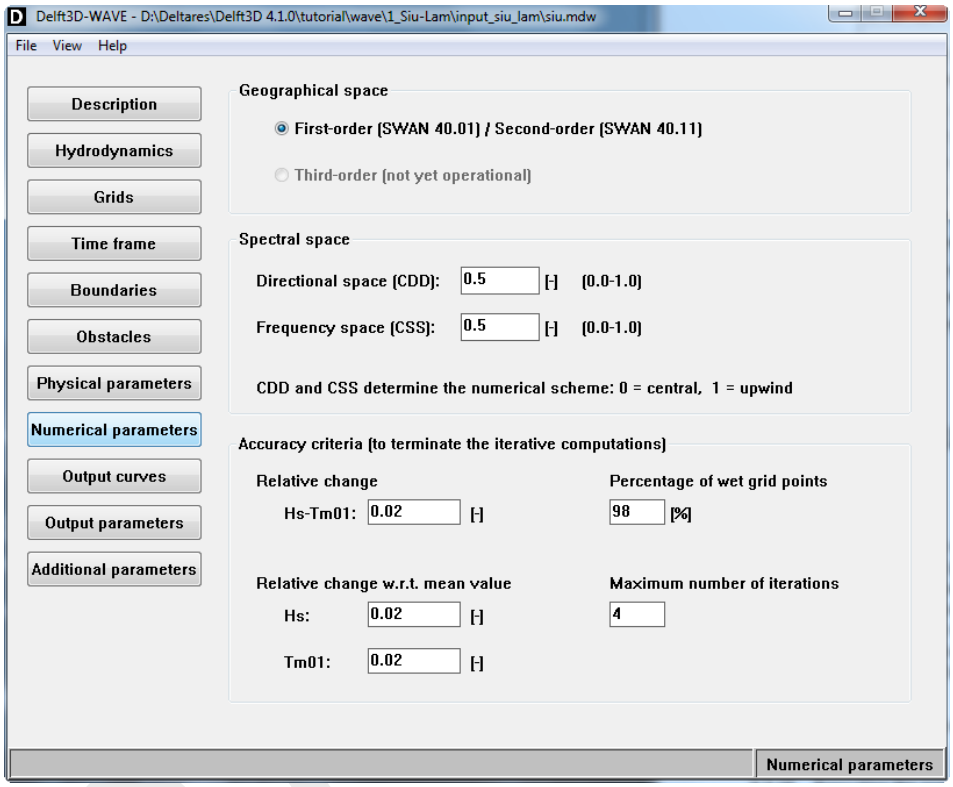

Figure 1.16: *Data Group* Numerical parameters

space through the parameter for the *Frequency space (CSS)*. The default values will be used here.

In the canvas **Accuracy criteria (to terminate the iterative computations)**, you can influence the criteria for terminating the iterative procedure in the SWAN computations (for convergence criteria of SWAN see **??**). Here the default values are used for the *Relative change*, the *Relative change w.r.t. mean value* and the *Percentage of wet grid points*.

You can also specify the *Maximum number of iterations* at which the computation stops.

➢ In this tutorial change the default number of “15” iterations into “4” iterations.

1.2.13 Output curves

Within the Data Group *Output curves* you can specify an output curve at which wave output should be generated by Delft3D-WAVE. Actually the curve is piecewise linear. In this tutorial no output curves will be defined.

1.2.14 Output parameters

Within the Data Group *Output parameters* (see Figure 1.17) you can determine to which grid (i.e. wave or flow grid) output is written and to which extent the computations should be monitored. The latter option can be used to specify that Delft3D-WAVE should produce intermediate (model) results during a SWAN run (test output) if the program produces unexpected results.

Within this data group it is also possible to select output locations for which Delft3D-WAVE produces wave output that is directly obtained from SWAN, e.g. 2D wave spectra.

➢ Select *Output parameters* to enter the Data Group (Figure 1.17).

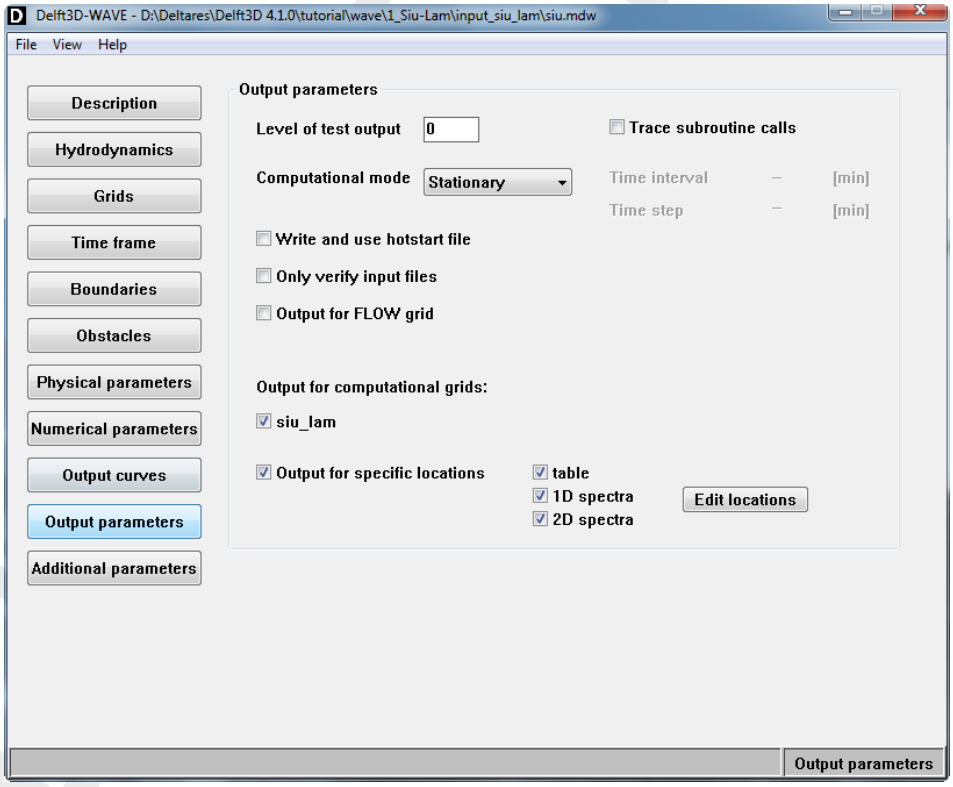

Figure 1.17: *Data Group* Output parameters

The default values for *Level of test output* and *Debug level* will be used. No hotstart file will be written and used.

Sometimes we want as much results as possible, Delft3D-WAVE offers to save the results of the calculation: on the communication file <com-$*$.dat> and on a SWAN output file <wavm-$*$.dat>. An overview of these output files is given in **??**.

➢ Check *Output for computational grids* button *siu_lam* to save the results on the <wavm-$*$.dat> output file
➢ and check *Output for specific locations* to indicate that SWAN output should be generated at some locations.
➢ Click on *Add* to edit the x- and y-co-ordinates of the output locations. Enter the location with co-ordinates (826000, 823000), see Figure 1.18.
➢ Save file by pressing the *Save* button.
➢ Press the *Close* button.

For the selected location you can have three types of output: *table*, *1D spectra*, *2D spectra*.

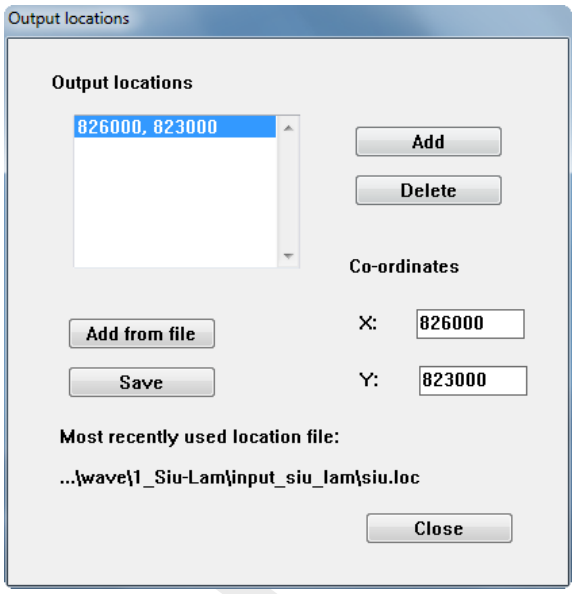

***Figure 1.18: Output locations** window*

➢ Select all these options for the Siu-Lam case.
Remarks:
- ◇ The Table output for specific locations is stored in <waveh-run-id.nc> for the overall computational grid and the i–th nested grid.
- ◇ The 1D spectra output for specific locations is stored in <*case*.sp1> for the overall computational grid, <*case*ni.sp1> for the i–th nested grid.
- ◇ Similar for the 2D spectra output in <∗.sp2> files.

➢ After the input is completed, select *File* → *Save As* to save the input as <siu.mdw> file.
➢ Select *File* → *Exit* to close the WAVE-GUI.

Now the scenario is ready to be executed.

1.2.15 Additional parameters

Description of *Additional Parameters* is under construction

1.2.16 Executing the scenario

A wave scenario is stored in an <∗.mdw> file. To execute a wave scenario, the <∗.mdw> file must be selected.

➢ To start in foreground, select *Start* in the **Wave (standalone)** menu.
➢ Select the WAVE input file <siu.mdw> (see Figure 1.19).

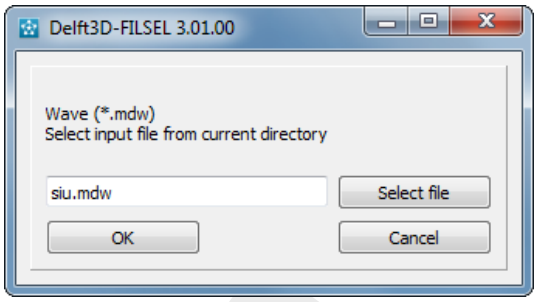

Figure 1.19: *Select scenario to run*

➢ Confirm by *OK* and the wave computation will be carried out.

In foreground the status of the simulation and possible messages are displayed in the active window. The simulation will start. After the simulation has finished check the results with the postprocessing program.

After the simulation is finished you are strongly advised to inspect at least some of the report files generated during the simulation to check if all went according to plan. This concerns especially the <swn-diag> file. To see this report:

➢ Select *Report* in the Delft3D-MENU window to inspect the report file of the wave model SWAN.

Especially the end of the <swn-diag.siu> file is of importance as it summarises errors, warnings and information of the computation.

1.2.17 Output files of Delft3D-WAVE

The result files of the calculation are of the NEFIS file format. The result files are:

<wavm-siu.dat> and <wavm-siu.def>

The results of the calculation can be visualised using either GPP or Delft3D-QUICKPLOT postprocessors. A description of the output parameters available on the output files is given in **??**.

1.2.18 Visualising results

The results presented in this section are generated using the GPP postprocessor.

In Figure 1.20 to Figure 1.24 some results are shown of the computed wave pattern near Siu-Lam. To reproduce these plots you should start the postprocessing program GPP.

➢ Select GPP either in the **Waves** window or in the **Utilities** window of Delft3D-MENU.
➢ In the main window of GPP select *Session - Open*.
➢ In the file selection menu select and open the session file <tutorial_swan_siu_lam.ssn>.
➢ In the main window of GPP select *Plots* and select from the list of possible plots the one you would like to inspect.

In an <∗.ssn> file the references are stored to data sets, in this case the result files of the siu-scenario, and the definition of earlier defined plots and their layout. By calling this scenario file you can inspect the same plots after repeating the simulation with (other input data) of the WAVE scenario “siu”. For details of using GPP you are referred to the User Manual of GPP.

To return to the main window of GPP while viewing a plot:

➢ Select *Plot* → *Close*.

You can select another plot as described above.

To close GPP and return to Delft3D-MENU:

➢ Select in the main window of GPP *Session* → *Exit*.

To close Delft3D-MENU:

➢ Select *Return*.
➢ Select *Exit*.

1.3 Nested wave model

This tutorial discusses the set-up of a wave model in which a nesting procedure takes place for a specific example called 'Friesian Inlet'. The modelled area covers an area in the north of The Netherlands called the 'Wadden Sea', which is an open sea protected by a series of barrier islands. Most of the input definition is already discussed in Tutorial 1 and therefore we will mention those steps briefly. Only the additional steps needed for the wave simulation are presented in this tutorial.

It is noted that the used wind, wave and other parameters do not represent realistic conditions for that area. Therefore the presented results have no practical use.

The input data is located on the directory

<.../tutorial/wave/2_Nested_wave_model/input_nested_wave>

and need to be copied first to the directory

<.../tutorial/wave/2_Nested_wave_model/>.

The files used in the case of *Friesian Inlet* are:

<netherlands.ldb>	land boundary file
<wadden_sea.grd>	Delft3D grid file
<wadden_sea.enc>	Delft3D enclosure file
<wadden_sea.dep>	Delft3D depth file
<inlet.grd>	Delft3D grid file
<inlet.end>	Delft3D enclosure file
<inlet.dep>	Delft3D depth file
<detailed.grd>	Delft3D grid file
<detailed.enc>	Delft3D enclosure file
<detailed.dep>	Delft3D depth file

1.3.1 WAVE Graphical User Interface

Start the WAVE-GUI on the directory <.../tutorial/wave/2_Nested_wave_model/> (see **??** for details).

DRAFT

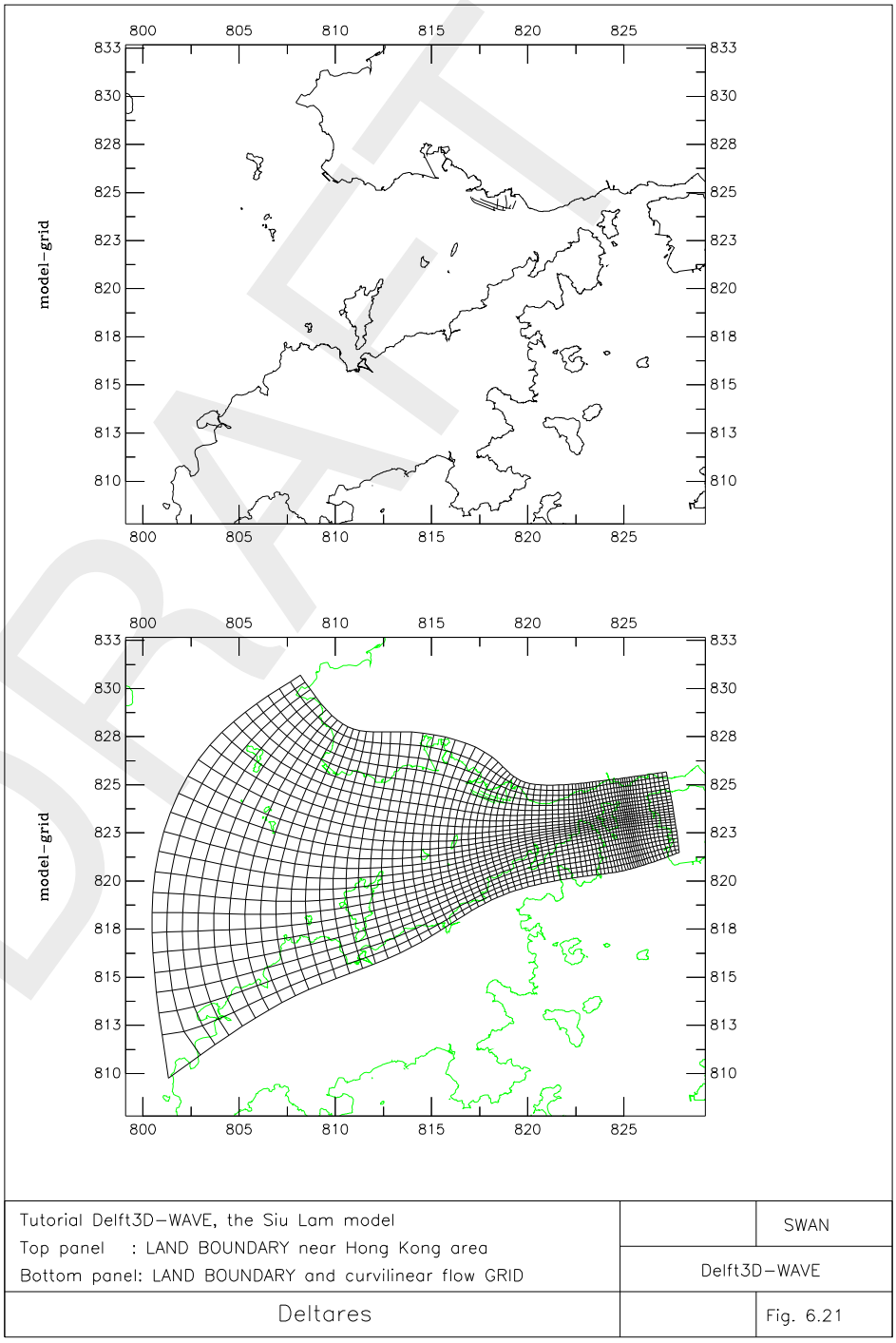

Figure 1.20: *Top panel: Siu Lam model area near Hong Kong area.*
Bottom panel: LAND BOUNDARY and curvilinear flow GRID

DRAFT

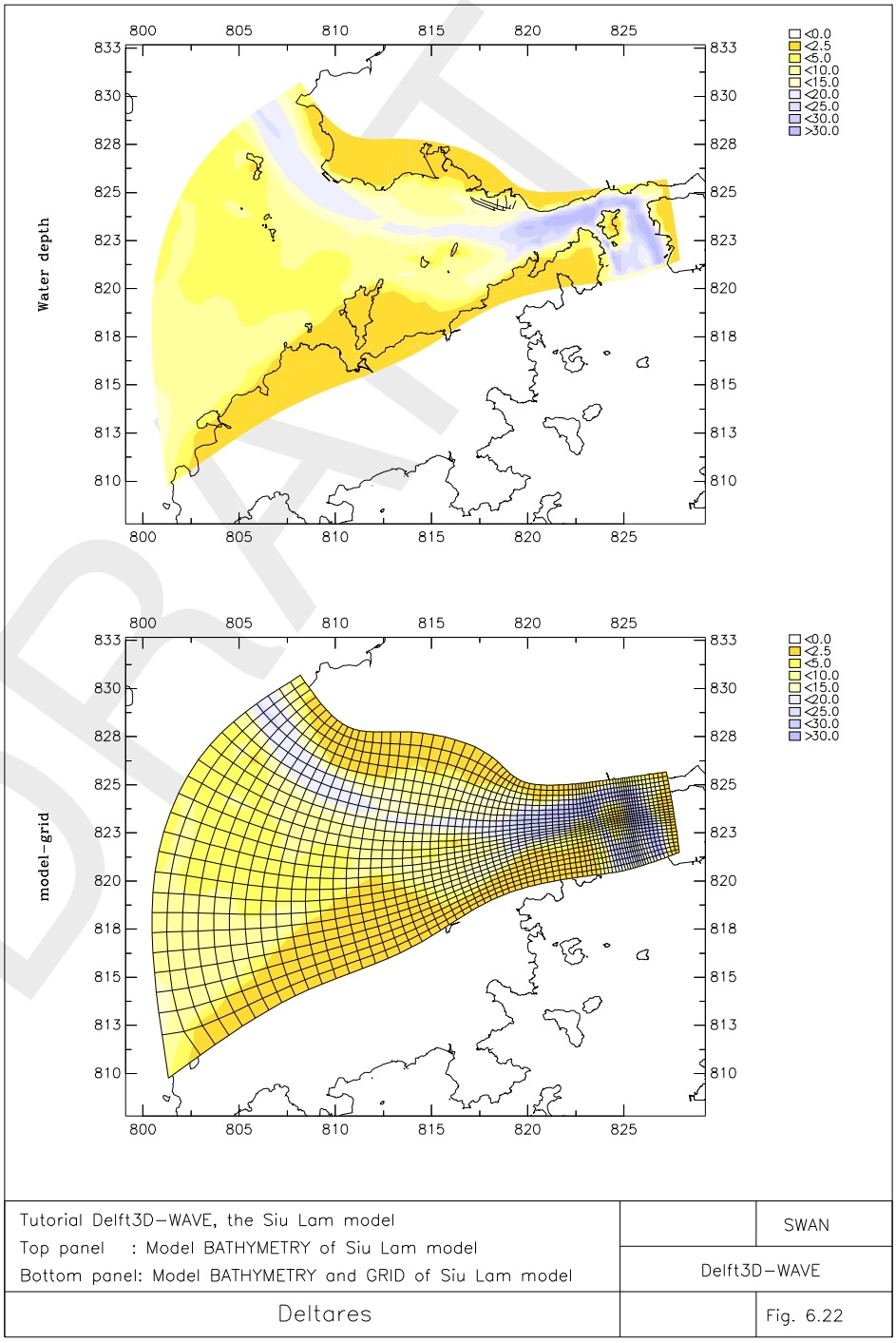

Figure 1.21: *Top panel: Model BATHYMETRY of Siu Lam model.*
Bottom panel: BATHYMETRY and GRID of Siu Lam model

DRAFT

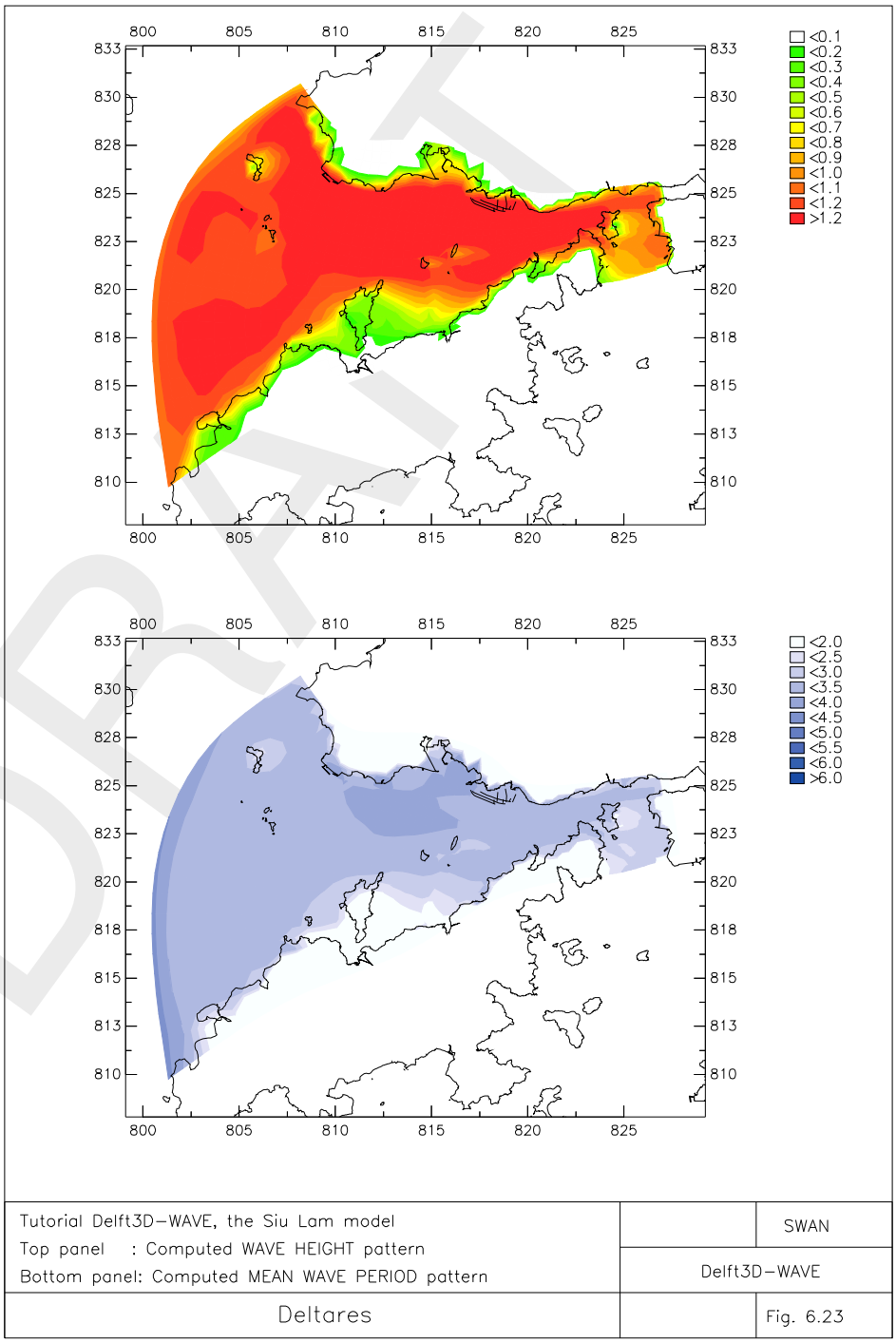

Figure 1.22: *Top panel: Computed WAVE HEIGHT pattern on 1 Oct 2005, 18:00. Bottom panel: Computed MEAN WAVE PERIOD pattern on 1 Oct 2005, 18:00*

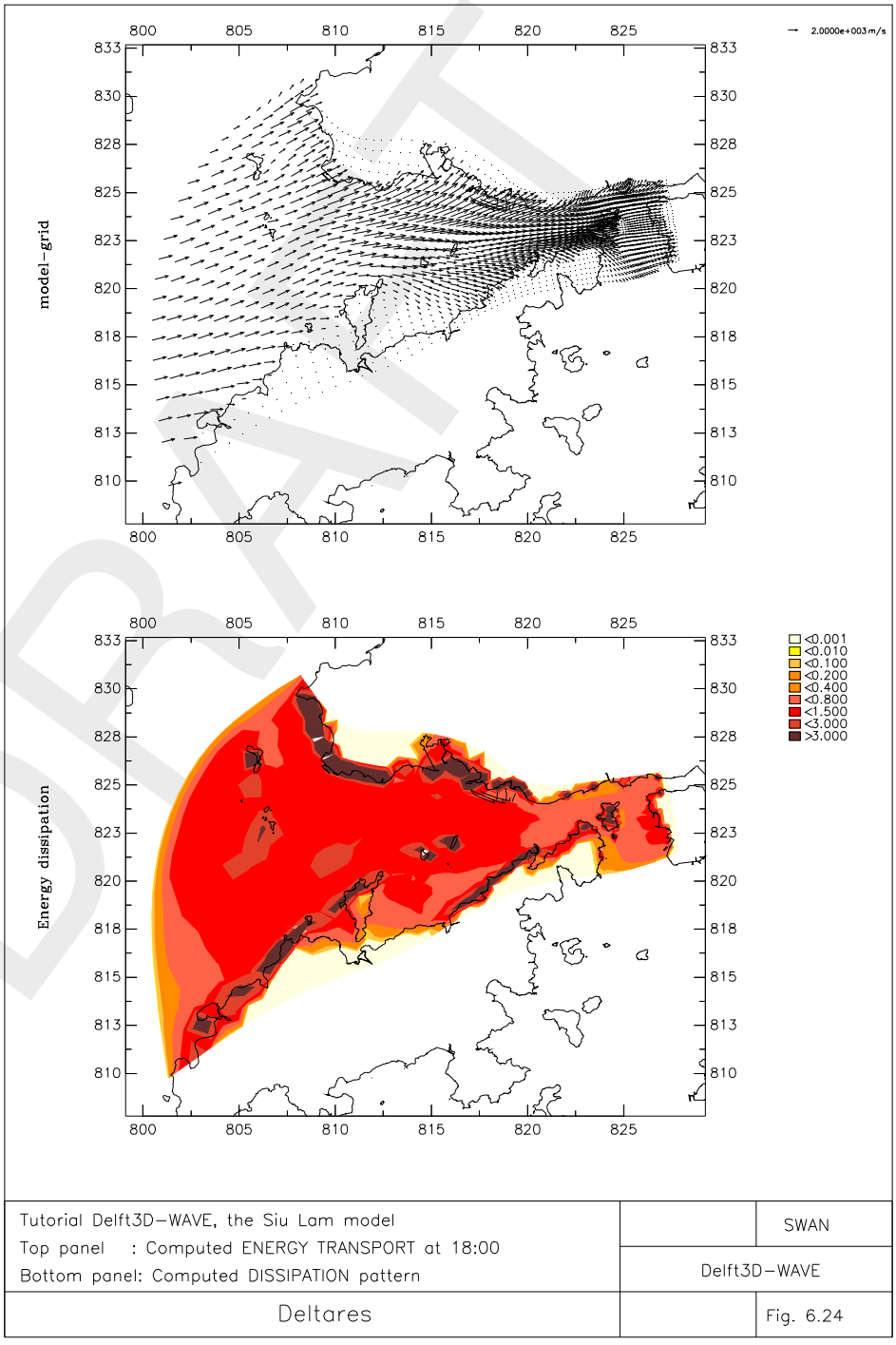

Figure 1.23: *Top panel: Computed ENERGY TRANSPORT on 1 Oct 2005, 18:00. Bottom panel: Computed DISSIPATION pattern on 1 Oct 2005, 18:00*

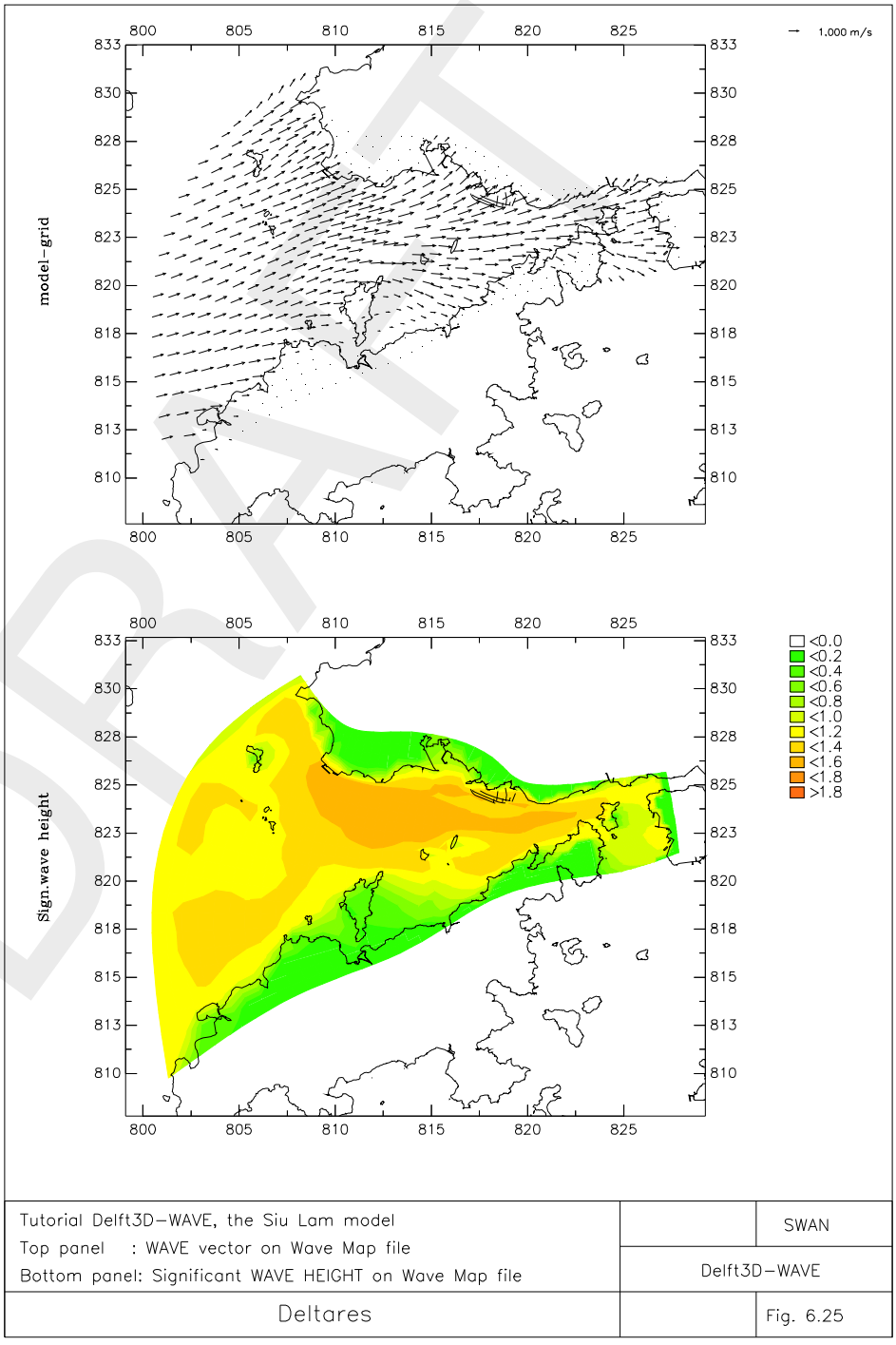

Figure 1.24: *Top panel: WAVE vector on 1 Oct 2005, 18:00.*
Bottom panel: Significant WAVE HEIGHT on 1 Oct 2005, 18:00

DRAFT

1.3.1.1 Description

Type the description:

- ➢ Project name: “Friesian Inlet”
- ➢ Project: “f01”
- ➢ Description: “Tutorial Delft3D-WAVE”
  “Friesian Inlet”
  “Standalone Wave model with nesting”

1.3.1.2 Hydrodynamics

No information from a flow model is used in this tutorial.

1.3.1.3 Grids

When you want to make nested runs, you first have to import all considered grids. In this canvas three computational grids must be imported.

- ➢ Start with importing the coarsest grid called <wadden_sea.grd>.

For this grid, a corresponding depth file must be imported, where the bathymetry data is based on the selected computational grid.

- ➢ Go to the tab **Bathymetry** and open the file <wadden_sea.dep>.
- ➢ Import the <inlet.grd> and the corresponding <inlet.dep> file.
- ➢ Finally, import the <detailed.grd> and its depth file <detailed.dep>.

In the tab **Nesting** you must define from which grid the selected grid must obtain its boundary conditions.

- ➢ First select the *detailed* grid in the list box for *Computational grids*.

You can select grids from the computational grid window by clicking on the presented grids. The line of that grid will become dark-blue (see Figure 1.25).

Note that for the last grid, a choice can be made between two grids (see Figure 1.25). It is possible to nest the <detailed.grd> in the <wadden_sea.grd> or in the <inlet.grd>.

- ➢ Select the <inlet.grd>.

Also check if the <inlet.grd> is nested in the <waddensea.grd>.

Remarks:

- ◇ The first grid cannot be nested in another one. For this grid, boundary conditions must be specified in the Data Group *Boundaries*.
- ◇ A grid cannot be nested in itself.
- ◇ If land points remain dry during the computations, then these points will be ignored for the SWAN computation.

1.3.1.4 Time frame

- ➢ Add one time point for the wave computation: “04 08 2005 00 00 00” [dd mm yyyy hh mm ss]

DRAFT

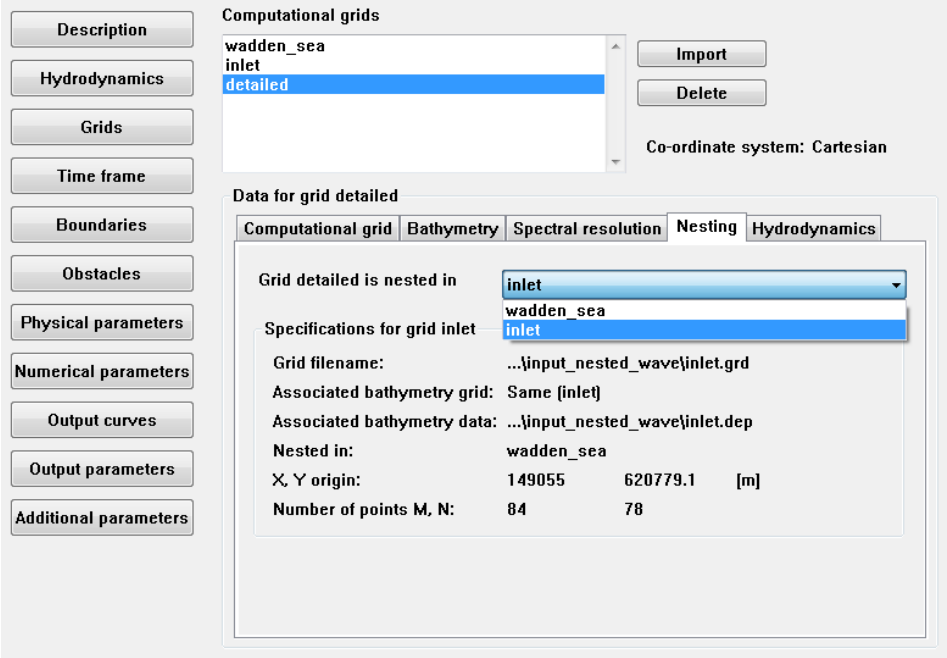

Figure 1.25: *Data Group Grids – Nesting window*

1.3.1.5 Boundaries

➢ Add one boundary called “Boundary West” with orientation ‘West’.
➢ Specify uniform conditions along the boundary with:

Significant wave height: “2.0” m
Peak period T_p: “6.3” s
Direction (nautical): “270” degrees
Directional spreading: “4” [-]

1.3.1.6 Obstacles

No obstacles are applied.

1.3.1.7 Physical parameters

➢ Press button *Wind*
➢ Define a uniform wind with a wind speed of “10” m/s coming from the southwest (225°).

1.3.1.8 Numerical parameters

Keep the default values.

1.3.1.9 Output curves

No output curves are defined.

1.3.1.10 Output parameters

➢ Select to write output to all three computational grids (see Figure 1.26).

➢ Save the input as <wad.mdw>.

Figure 1.26: *Data group* Output parameters*, output for computational grids*

1.3.1.11 Additional parameters

No additional parameters are defined.

1.3.2 Run and postprocessing

Execute this scenario and check the results with a postprocessing program.

You will see that SWAN will make three computations: one computation for each computational grid. The result files will be named as follows:

<wavm-wad-waddensea.dat> and <wavm-wad-waddensea.def>
<wavm-wad-inlet.dat> and <wavm-wad-inlet.def>
<wavm-wad-detailed.dat> and <wavm-wad-detailed.def>

The wave map files include the grid name in the name, so you can directly select the correct output file.

1.4 Online-WAVE coupling (including morphology)

1.4.1 Introduction

Figure 1.27: *Development of a spit at the Head of Ameland: the Bornrif*

This tutorial discusses the set-up of a model with flow-wave interaction. The coupling of the FLOW and WAVE module is discussed based on an example called *Bornrif*. The modelling area concerns the Ameland inlet, which is a part of the Wadden Sea. The model was made (Wilkens, 1999) to reproduce the development of a spit at the head of Ameland: the *Bornrif* (see Figure 1.27). It appeared to be possible to reproduce the spit forming, though not with the schematisations of the real acted hydraulic conditions. Only wind and waves from the west or north-west were applied. The simulated time span amounted five years and took three days of computation time.

The morphological development agreed partly with the observations. The spit is formed, but not with the correct shape. The severe erosion of the ebb delta and the sedimentation north and east of it are less extensive in reality. The migration of the channels is not reproduced very well.

The driving forces of the bathymetry development appeared to be the waves. This is correct to a certain extent. In this model, however, the sediment transport of waves seemed to be overestimated with respect to the currents. In areas sheltered from waves the morphological activity is too small. It was also concluded that the transport over the flats is probably too high with respect to the transport through the channels.

DRAFT

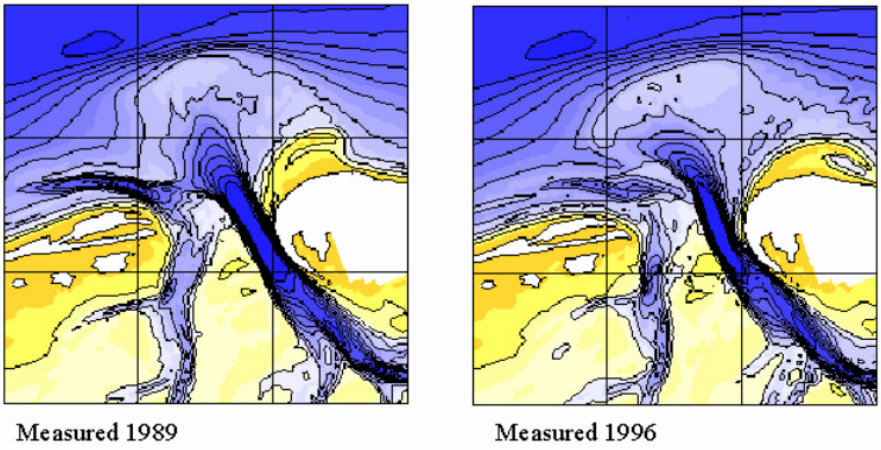

Figure 1.28: *Measured 1989 and 1996 bathymetry*

This tutorial covers a part of the above simulation focussing only on one wave condition. Also the morphodynamic aspects are taken into account in this tutorial.

The input data is located on the directory

<.../tutorial/wave/3_bornrif/input_bornrif>

and need to be copied first to the directory

<.../tutorial/wave/3_bornrif/>.

The files used in the *Bornrif* case are:

Delft3D-FLOW:

<netherlands.ldb>	Landboundary file
<rif.grd>	Delft3D grid file
<rif.enc>	Delft3D enclosure file
<rif.dep>	Delft3D depth file
<rif_neu.bnd>	Delft3D open boundaries file
<rif.bch>	Delft3D boundary condition file (harmonic)
<rif.bcc>	Delft3D boundary condition file (concentration)
<rif.wnd>	Delft3D wind file
<rif_200.sed>	Delft3D file containing sediment data
<rif_200.mor>	Delft3D file containing morphological data
<rif.obs>	Delft3D Observation points
<rif.crs>	Delft3D Cross-sections

Delft3D-WAVE:

<wave_overall.grd>	Delft3D grid file
<wave_overall.enc>	Delft3D enclosure file
<wave_overall.dep>	Delft3D depth file
<wave_detail.grd>	Delft3D grid file
<wave_detail.enc>	Delft3D enclosure file
<wave_detail.dep>	Delft3D depth file

To make a coupling between the FLOW and WAVE module, both flow- and wave-models

should be set up, before the coupling can be accomplished. First a description of the set-up of the FLOW model is discussed.

1.4.2 Delft3D-FLOW model

Start the FLOW-GUI on the directory <.../tutorial/wave/3_bornrif/> (see **??** for details).

1.4.2.1 Description

Type a description of the project name, project identifier and a descriptive text:

- Name: "Friesian Inlet"
- Identifier: "f01"
- Text: "Tutorial Delft3D-WAVE"
  "Ameland Tidal Inlet"
  "Coupling of the FLOW and WAVE module"

1.4.2.2 Domain

- Import the flow grid <rif.grd>,
- the corresponding enclosure file <rif.enc> and
- Specify the latitude "52".
- related bathymetry file <rif.dep>.

No dry points and thin dams are specified here.

1.4.2.3 Time frame

- Choose as a reference date: "01 01 1996"
- Simulation start time: "01 01 1996 04 12 00"
- Simulation stop time: "02 01 1996 01 00 00"
- Time step: "1" minute

1.4.2.4 Processes

- In the FLOW datagroup *Processes*, the process *Wave* must be activated.

A window will appear with instructions related to the flow-wave coupling (see Figure 1.29). Read these instructions.

- Click *Go to Output* (see Figure 1.31).

It is also possible to do this in the end.

The FLOW module now expects to read wave data from the communication file at certain time points during the computation. The process *Online Delft3D-WAVE* should be selected to couple a Delft3D-FLOW computation directly with a Delft3D-WAVE computation. This feature is called the *Online-WAVE* option.

- Check *Online Delft3D-WAVE*

Two other processes (Sediments and wind) must be activated as well, which are flow related (see Figure 1.30):

DRAFT

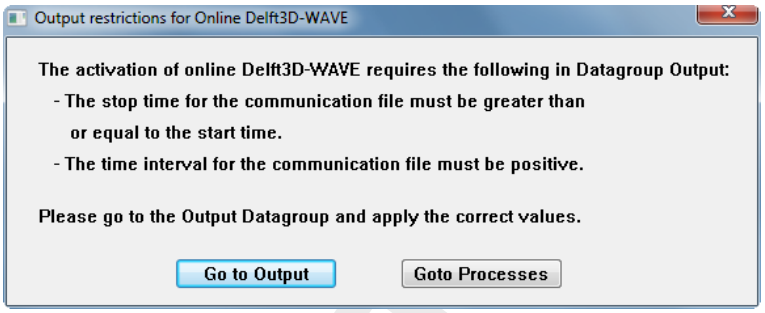

Figure 1.29: *Output restrictions for Online-WAVE*

- Check the buttonSediments
- Enter the name of non-cohesive sediment: “Sediment sand”
- Click on button *Add*
- To close the window click on button *Close*
- Check the checkbox *Wind*.

Now the window looks like Figure 1.30.

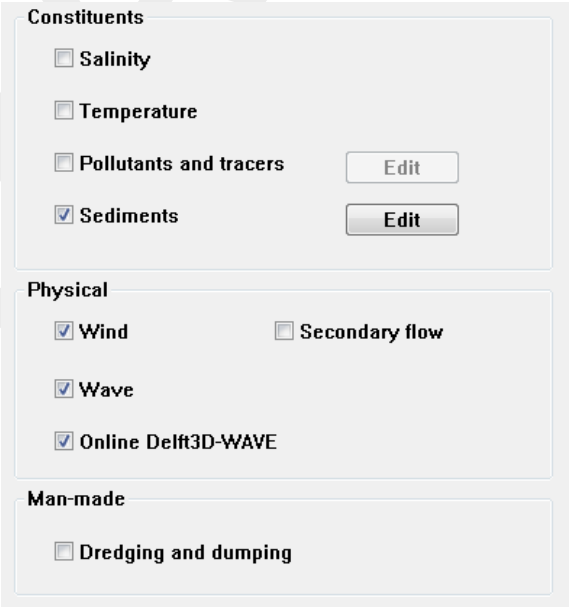

Figure 1.30: *Overview of active processes*

1.4.2.5 Initial conditions

- Enter “-0.45” m as the uniform value for water level.
- Enter “0” kg/m^3 for sediment sand.

1.4.2.6 Boundaries

- Press the button *Open*
- Import the following files:

  <rif_neu.bnd> boundary definitions,
  <rif.bch> harmonic boundary conditions and
  <rif.bcc> boundary conditions for constituents (Transport conditions).
- Press the *Close* button to close the window **Open/Save Boundaries**.

1.4.2.7 Physical parameters

Use all default values for the constants, but adapt wind drag coefficients

➢ Select tab *Constants*

Enter the following values for the *Wind drag coefficients*

➢ Breakpoint A: Enter coefficient “0.0025”, wind speed “0” m/s
➢ Breakpoint B: Enter coefficient “0.0025”, wind speed “100” m/s
➢ Breakpoint B: Enter coefficient “0.0025”, wind speed “100” m/s

➢ Select tab *Roughness*
➢ Set the uniform Manning roughness coefficient for both velocity components (U and V) to “0.026”.

Use the default values for viscosity.

➢ Select tab *Sediment*
➢ Press button *Open* and select the file <rif_200.sed>.

➢ Select tab *Morphology*
➢ Press button *Open* and select the file <rif_200.mor>.

➢ Select tab *Wind*
➢ Press button *Open* and select the file <rif.wnd>.

1.4.2.8 Numerical parameters

➢ Adapt the *Threshold depth* = “0.35” m

Use default for the other options

1.4.2.9 Operations

No operations (like dredge and dump) are specified in this case.

1.4.2.10 Monitoring

Import the files for observation points and cross sections. No drogues are specified in this case.

➢ Select button *Observations*
➢ Press button *Open* and select the file <rif.obs>.

➢ Select button *Cross-sections*
➢ Press button *Open* and select the file <rif.crs>.

1.4.2.11 Additional parameters

➢ Add the following keyword (see Deltares (2024)): “Cstbnd=#Yes#”

DRAFT

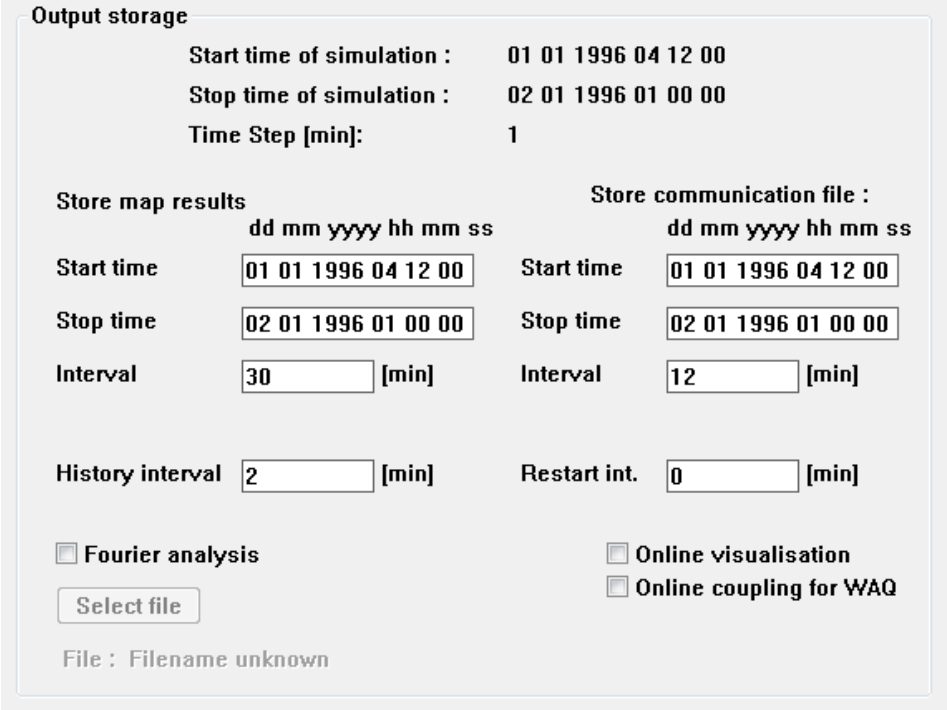

Figure 1.31: *Overview of output parameters*

1.4.2.12 Output

Store map results and prescribe the history interval as presented in Figure 1.31.

Essential for the flow-wave coupling is the storing interval of the communication file.

- At each interval (“12” minutes), the FLOW module will be updated with wave data starting at “01 01 1996 04 12 00” till the Stop time “02 01 1996 01 00 00”.
- Save this file as <rif.mdf>.

Exit the FLOW-GUI:

- Click *File* → *Exit.*

1.4.3 Delft3D-WAVE model

Start the WAVE-GUI on the directory <. . . /tutorial/wave/3_bornrif/> (see **??** for details).

1.4.3.1 Description

Type the description:

- Project name: “Bornrif”
- Project: “003”
- Description: “Tutorial Delft3D-WAVE”
  “FLOW-3DMOR and Online WAVE simulation”

1.4.3.2 Hydrodynamics

Select the hydrodynamic results from Delft3D-FLOW

- Check checkbox *Run WAVE together with FLOW*
- Select FLOW file <rif.mdf>.

1.4.3.3 Grids

In this exercise, the detailed grid is nested in an overall grid as shown in the section 1.3.

Computational grid

- Import the overall grid file: <wave_overall.grd>.
- Select tab *Bathymetry*.
- Import the related bathymetry file <wave_overall.dep>.
- Import the detailed grid file: <wave_detail.grd>.
- Import the related bathymetry file <wave_detail.dep>.

This grid must be nested in the overall grid. Use for both grids the spectral resolution as default values.

Hydrodynamics

- Select tab *Hydrodynamics*.

Select for both grids the items as follows:

- **Water level** option “Use but don’t extend”.
- **Current** option “Use but don’t extend”.
- **Bathymetry** option “Use but don’t extend”.
- **Wind** option “Don’t use”.

1.4.3.4 Time frame

The time frame is automatically read from the file <rif.mdf>. The storing interval to the communication file (i.e. 12 minutes) determines when Delft3D-WAVE is executed. Leave the water level correction on its default value.

1.4.3.5 Boundaries

Create the three boundaries North, East and West. First the North boundary.

- Press the *Add*
- Set *Boundary name* to “Boundary North”
- Set *Define boundary by* to “Orientation”
- Set *Boundary orientation* to “North”
- Set *Conditions along boundary* to “Uniform”
- Set *Specification of spectra* to “Parametric”
- Press button *Edit conditions*

- Significant wave height: “2.0” m
- Peak period T_p: “7.0” s
- Direction (nautical): “330” degrees
- Directional spreading: “4” [-]

DRAFT

The boundaries East and West use the same values, except for the word 'North' (0°).

1.4.3.6 Obstacles

No obstacles are defined.

1.4.3.7 Physical parameters

- Press button *Wind*
- Specify a uniform wind speed of "6" m/s and
- specify a wind direction of "330" degrees.

Use default settings for all other processes.

1.4.3.8 Numerical parameters

Keep the default values.

1.4.3.9 Output curves

No output curves are defined.

1.4.3.10 Output parameters

- Select *Write and use the hotstart file*.
- Select *Output for FLOW grid*, for writing the wave data to the communication file.
- Select *Output for computational grids* to write output for the computational grid <wave_overall.grd> and
- Select *Output for computational grids* to write output for the computational grid<wave_detail.grd> (see Figure 1.32).
- Specify the interval for output of the computational grids, "12".

DRAFT

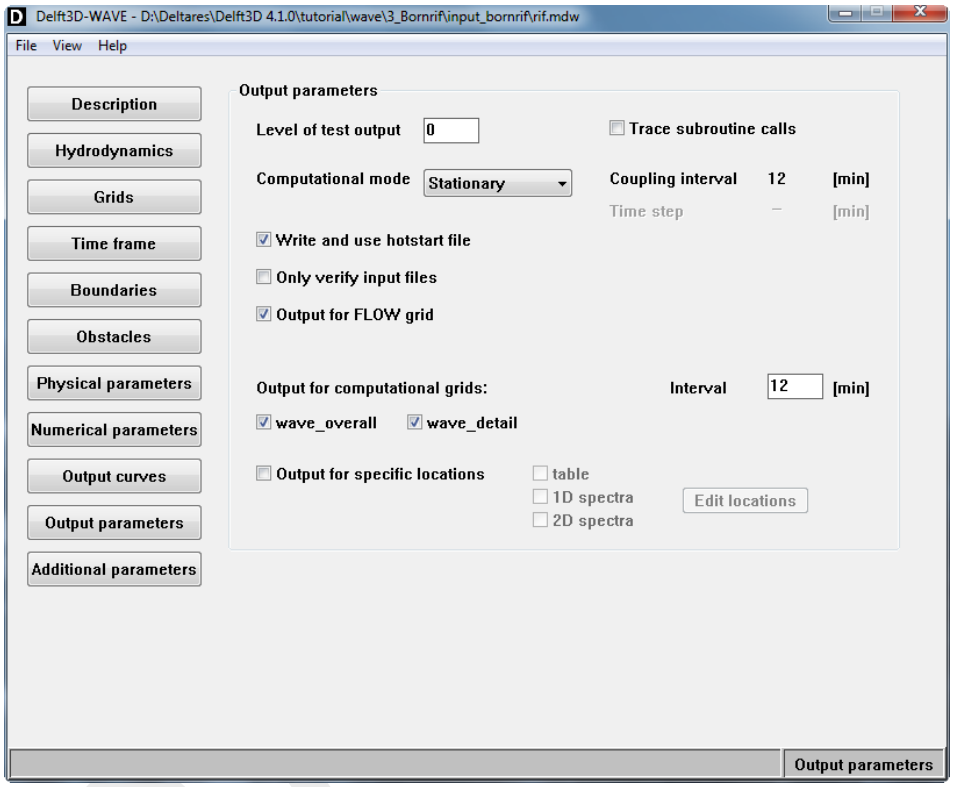

Figure 1.32: *Overview of output parameters in Delft3D-WAVE*

➢ Save the wave input to file <rif.mdw>.

Exit the WAVE-GUI:

➢ Click *File* → *Exit.*

1.4.4 Run and postprocessing

Executing this flow-wave model (including sediment and morphology) can be done by in foreground or background.

1.4.4.1 Foreground

To start in foreground

➢ Select *Start* in the **Hydrodynamics (including morphology)** menu.
➢ Select the FLOW input file <rif.mdf>.
➢ Confirm the selection by pressing *OK*.
➢ Select the WAVE input file <rif.mdw>.
➢ Confirm by *OK* and the flow-wave computation will be carried out.

The simulation will start. After the simulation has finished check the results with the postprocessing program.

1.4.4.2 Background

Go back to the main Delft3D menu

➢ Click *Batch* (see Figure 1.33).
➢ Click *Prepare*.

➢ Click *Start*.

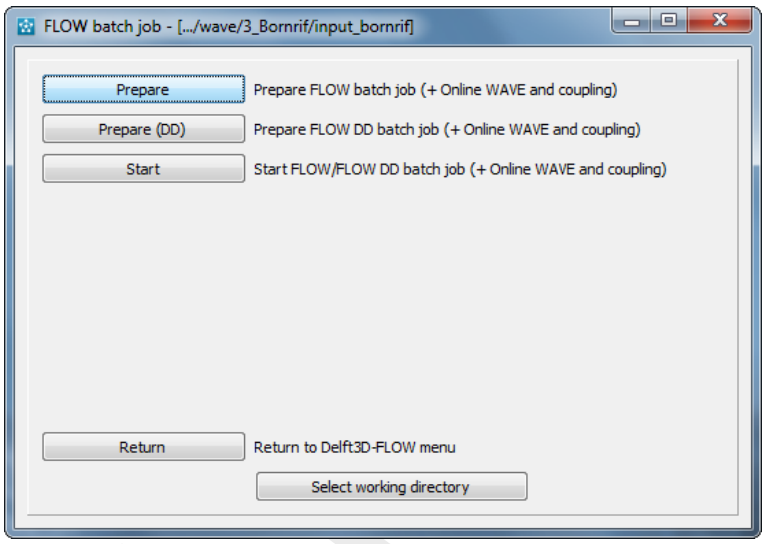

Figure 1.33: *Execute the Flow-Wave model*

1.4.4.3 Output files

The FLOW module will create the following files:

<trim-rif.dat> and <trim-rif.def>
<trih-rif.dat> and <trih-rif.def>
<com-rif.dat> and <com-rif.def>.

The WAVE module will create the files:

<wavm-rif-wave_overall.dat> and <wavm-rif-wave_overall.def>
<wavm-rif-wave_detail.dat> and <wavm-rif-wave_detail.def>

The wave map files include the grid name in the file name, so you can directly select the correct output file.

1.5 FLOW-DD and Online WAVE

1.5.1 Introduction

In this tutorial the set-up of a Domain Decomposition model in combination with a Delft3D-FLOW-WAVE simulation is discussed, based on an example called Bornrif. With Domain Decomposition it is possible to divide the large domain into several smaller sub-domains. More information on Domain Decomposition can be found in Appendix B.13 of the Delft3D-FLOW user manual. The intention of this tutorial is only to illustrate the set-up of a domain decomposition model in combination with Delft3D-FLOW and Delft3D-WAVE. There is no physical functionality of domain decomposition in the Bornrif example. The domain decomposition is applied in the tidal channel between Terschelling and Ameland in order to obtain a higher resolution (factor 5) at that specific location. See Figure 1.34.

The input data is located on the directory

<.../tutorial/wave/4_bornrif_dd/input_bornrif>

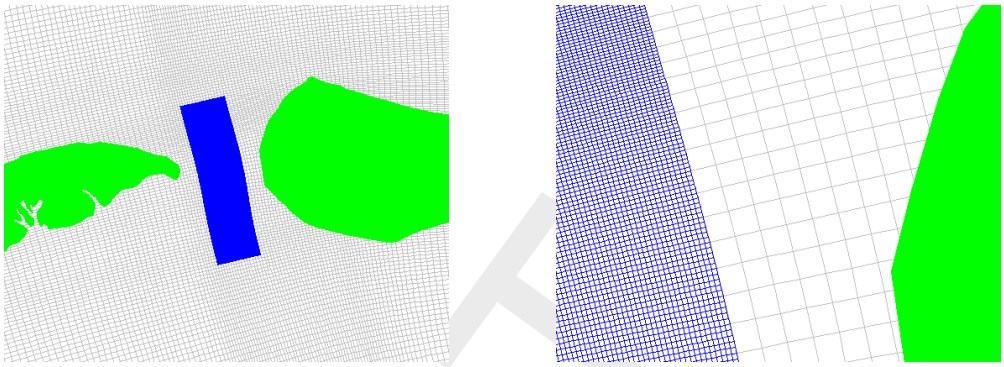

Figure 1.34: *Location of grids of both domains between Terschelling and Ameland (left panel) and detail of both domains close to Ameland (right panel)*

and need to be copied first to the directory

<. . . /tutorial/wave/4_bornrif_dd/>.

The following files are used in the *Bornrif DD* case:

Delft3D-FLOW:

<netherlands.ldb>	Landboundary file
<rif_inside.grd>	Delft3D grid file
<rif_inside.enc>	Delft3D enclosure file
<rif_inside.dep>	Delft3D depth file
<rif_outside.grd>	Delft3D grid file
<rif_outside.enc>	Delft3D enclosure file
<rif_outside.dep>	Delft3D depth file
<rif_neu.bnd>	Delft3D open boundaries file
<rif.bch>	Delft3D boundary condition file (harmonic)
<rif.bcc>	Delft3D boundary condition file (concentration)
<rif.wnd>	Delft3D wind file
<rif_200.sed>	Delft3D file containing sediment data
<rif_200.mor>	Delft3D file containing morphological data

Delft3D-WAVE:

<wave_overall.grd>	Delft3D grid file
<wave_overall.enc>	Delft3D enclosure file
<wave_overall.dep>	Delft3D depth file
<wave_detail.grd>	Delft3D grid file
<wave_detail.enc>	Delft3D enclosure file
<wave_detail.dep>	Delft3D depth file

DD-boundaries:

<inside_outside.ddb>	DD-boundary file for coupling of both grids

Other files are generated during the tutorial. For the simulation it is advised to store all the files in the same directory.

The complete model set-up of a FLOW-DD and Online WAVE simulation is discussed below. A description about the construction and coupling of sub-domain grids can be found in the

RGFGRID and FLOW user manuals (RGFGRID UM, 2016; Deltares, 2024). The model set-up starts with a description of the Delft3D-FLOW models.

1.5.2 Delft3D-FLOW models

For each sub-domain an mdf-file must be specified. First the set-up of the outside domain is discussed, followed by the set-up of the inside domain

1.5.2.1 Model set-up outside FLOW domain

Start the FLOW-GUI on the directory <.../tutorial/wave/4_bornrif_dd/>.

1.5.2.2 Description

➢ Type the description:

"Tutorial Delft3D-WAVE"
"Ameland Tidal Inlet"
"Combining FLOW-DD and WAVE"
"Outside model"

1.5.2.3 Domain

➢ Select tab *Grid*
➢ Import grid file <rif_outside.grd> and grid enclosure file <rif_outside.enc>
➢ Set the latitude to "52" degrees.
➢ Select tab *Bathymetry*
➢ Import the corresponding bathymetry file <rif_outside.dep>.

Dry points and thin dams are not specified in this case

1.5.2.4 Time frame

Set the following timings:

➢ Reference date: "01 01 1996"
➢ Simulation start time: "01 01 1996 04 12 00"
➢ Simulation stop time: "02 01 1996 01 00 00"
➢ Time step: "0.1" minute

Remark:

◇ The time step of the outside domain should equal the time step of the inside domain. Because the inside domain has a five times higher resolution the resolution of the inside domain is leading in setting the time step.

DRAFT

1.5.2.5 Processes

The following processes need to be activated:

- Check *Sediments* and define a non-cohesive sediment fraction 'sand'
- Check *Wind*
- Check *Wave*
- Check *Online Delft3D-WAVE*

For remarks on the wave processes see the Delft3D-WAVE user manual.

1.5.2.6 Initial conditions

Specify Uniform values as initial conditions

- Set the initial water level at "-0.45" m,
- Set the sand sediment initially to "0" kg/m^3.

1.5.2.7 Boundaries

- Press button *Open/Save*
- Import the boundary files <rif_neu.bnd>, <rif.bch> and <rif.bcc>.

1.5.2.8 Physical parameters

- Use all default values for the constants, but adapt the wind drag coefficient of the first breakpoint to "0.0025" (see Figure 1.35).

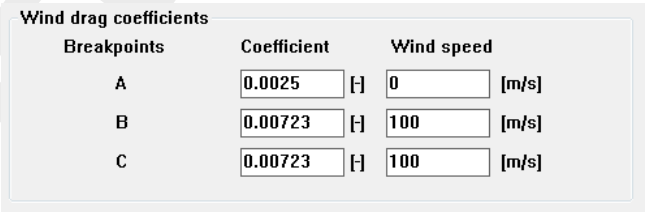

Figure 1.35: *Wind drag coefficients in Delft3D-FLOW for outside domain set-up*

- Set the uniform Manning roughness coefficient for both velocity components (U and V) to "0.026".
- Use the default values for viscosity.
- Select tab *Sediment*
- Import file <rif_200.sed>
- Select tab *Morphology*
- Import file <rif_200.mor> files.
- Select tab *Wind*
- Import the <rif.wnd> file.

1.5.2.9 Numerical parameters

- Adapt the threshold depth to "0.35" m and use default values for other parameters (see Figure 1.36).

1.5.2.10 Operations

No operations are specified in this case.

DRAFT

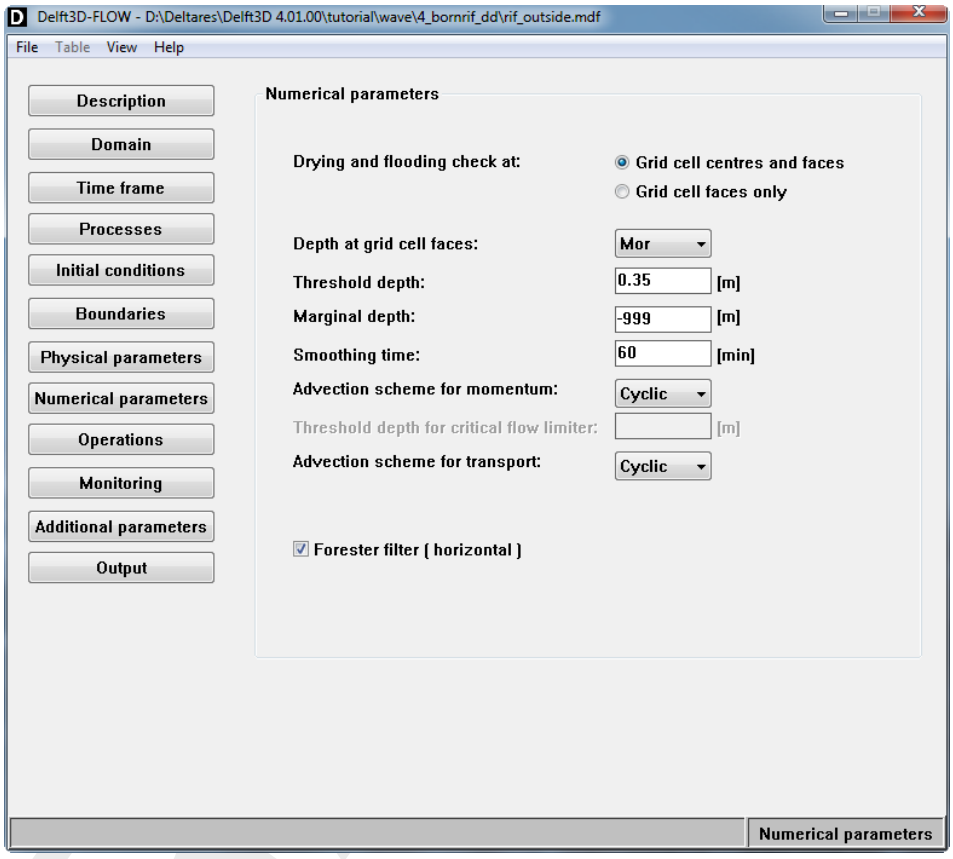

Figure 1.36: *Numerical parameters in Delft3D-FLOW for outside domain setup*

1.5.2.11 Monitoring

No observation points, drogues or cross-sections are specified in this case.

1.5.2.12 Additional parameters

Add the following additional keyword (see FLOW manual Deltares (2024)).

- In column *keyword*: “Cstbnd” and
- in column *value*: “#Yes#”

1.5.2.13 Output

- Store results as specified in the figure below, see Figure 1.37.
- Save the file as <rif_outside.mdf>,

Exit the FLOW-GUI:

- Click *File* → *Exit.*

1.5.2.14 Model set-up inside FLOW domain

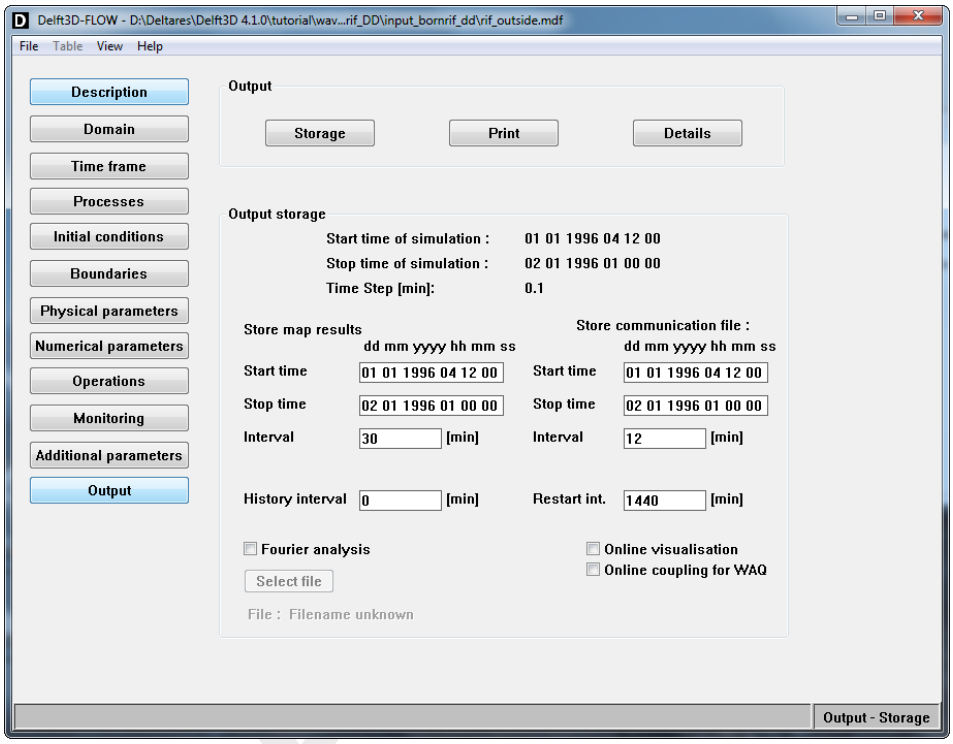

Figure 1.37: *Overview of output parameters of the Delft3D-FLOW model for the outside domain*

1.5.2.15 Description

Type the description:

➢ “Tutorial Delft3D-WAVE”
“Ameland Tidal Inlet”
“Combining FLOW-DD and WAVE”
“Inside model”

1.5.2.16 Domain

➢ Select tab *Grid*
➢ Import grid file <rif_inside.grd> and grid enclosure file <rif_inside.enc>
➢ Set the latitude to “52” degrees.
➢ Select tab *Bathymetry*
➢ Import the corresponding bathymetry file <rif_inside.dep>.

Dry points and thin dams are not specified in this case

Set the Data Groups *Time frame*, *Processes* and *Initial conditions* the same as the ‘Outside domain FLOW model’.

1.5.2.17 Boundaries

Do not specify any boundary and boundary condition! Boundary conditions are coming from the 'Outside domain FLOW model'.

Set the Data Groups *Physical parameters*, *Numerical parameters*, *Operations*, *Monitoring*, *Additional parameters* and *Output* the same as the 'Outside domain FLOW model', see section 1.5.2.1.

➢ Save the file as <rif_inside.mdf>.

Exit the FLOW-GUI:

➢ Click *File* → *Exit.*

1.5.3 Delft3D-WAVE model

1.5.3.1 Description

Type the description:

➢ Project name: "Bornrif"
➢ Project: "004"
➢ Description: "Tutorial Delft3D-WAVE"
"FLOW-DD with Online WAVE"

1.5.3.2 Hydrodynamics

Select the hydrodynamic results from Delft3D-FLOW

➢ Check checkbox *Run WAVE together with FLOW*
➢ Select FLOW file <rif_outside.mdf>.

Remark:
◇ Any mdf-file from the sub-domains can be selected. During the simulation Delft3D-WAVE will search for all mdf-files in case of a DD-simulation.

1.5.3.3 Grids

In this exercise, the detailed grid is nested in an overall grid.

Computational grid

➢ Import the overall grid file: <wave_overall.grd>.
➢ Select tab *Bathymetry*.
➢ Import the related bathymetry file <wave_overall.dep>.
➢ Import the detailed grid file: <wave_detail.grd>.
➢ Import the related bathymetry file <wave_detail.dep>.

This grid must be nested in the overall grid. Use for both grids the spectral resolution as default values.

Hydrodynamics

➢ Select tab *Hydrodynamics*.

DRAFT

Select for both grids the items as follows:

- **Water level** option “Use but don’t extend”.
- **Current** option “Use but don’t extend”.
- **Bathymetry** option “Use but don’t extend”.
- **Wind** option “Don’t use”.

1.5.3.4 Time frame

Communication between FLOW and WAVE will not be determined by the imported time points, but by the communication time settings specified in the FLOW-file.

1.5.3.5 Boundaries

Create the three boundaries North, East and West. First the North boundary, see Figure 1.38.

- Press the *Add*
- Set *Boundary name* to “Boundary North”
- Set *Define boundary by* to “Orientation”
- Set *Boundary orientation* to “North”
- Set *Conditions along boundary* to “Uniform”
- Set *Specification of spectra* to “Parametric”
- Press button *Edit conditions*

- Significant wave height: “2.0” m
- Peak period T_p: “7.0” s
- Direction (nautical): “330” degrees
- Directional spreading: “4” [-]

The boundaries East and West use the same values, accept for the word ’North’.

1.5.3.6 Obstacles

No obstacles are defined

1.5.3.7 Physical parameters

- Press button *Wind*
- Specify a uniform wind speed of “6” m/s and
- specify a wind direction of “330” degrees.

Use default settings for all other processes.

1.5.3.8 Numerical parameters

Keep the default values (see Figure 1.39)

1.5.3.9 Output curves

No output curves are specified

DRAFT

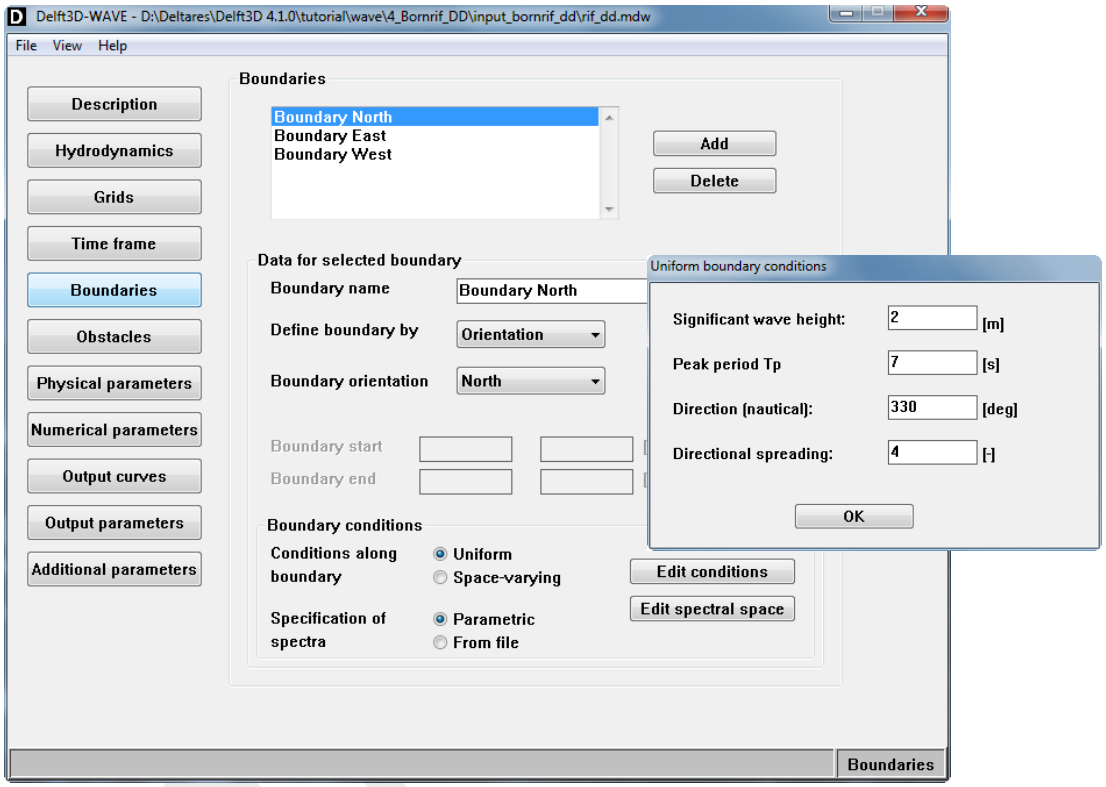

Figure 1.38: *Wave boundary conditions for 'Boundary North' in the WAVE model set-up*

1.5.3.10 Output parameters

➢ Set the output parameters as specified in Figure 1.40.

The name of the mdw-file does not have to be the same as the name of the coupled mdf-file. During the simulation Delft3D-WAVE will search for all available mdf-files in the directory.

➢ Save the wave input file as <rif_dd.mdw>.

Exit the WAVE-GUI:

➢ Click *File* → *Exit.*

1.5.4 Run and postprocessing

1.5.4.1 Foreground

To start in foreground

➢ Select *Start DD* in the **Hydrodynamics (including morphology)** menu.
➢ Select the FLOW input file <inside_outside.ddb>.
➢ Confirm the selection by pressing *OK*.
➢ Select the WAVE input file <rif_dd.mdw>.
➢ Confirm by *OK* and the flow-wave computation will be carried out.

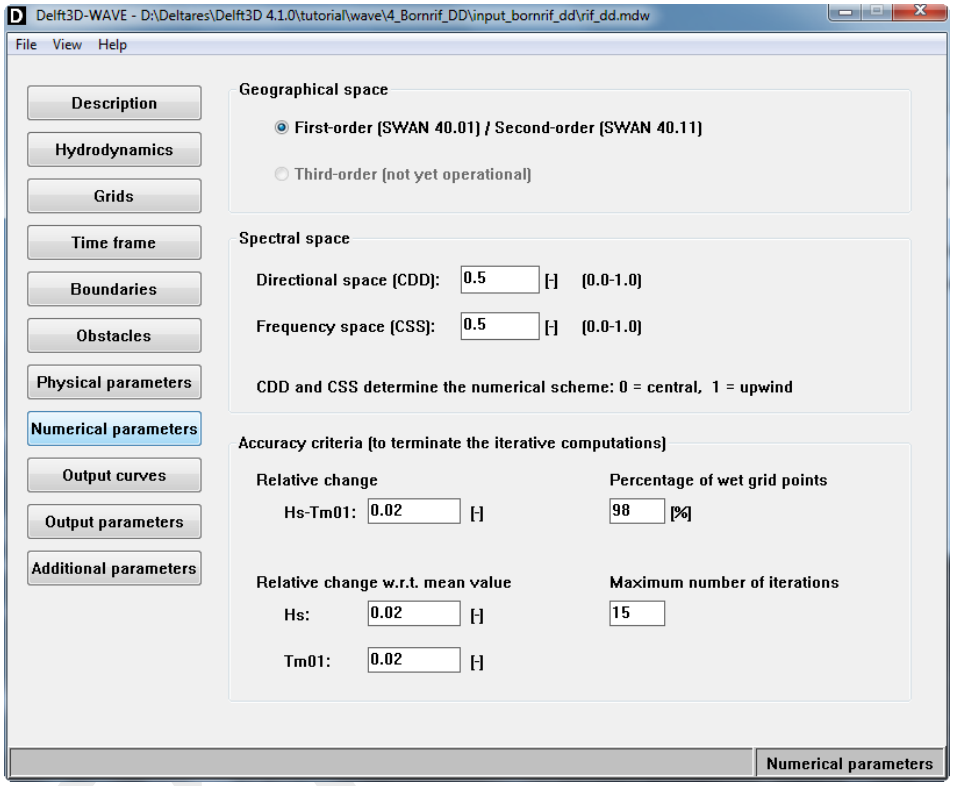

Figure 1.39: *Numerical parameters used in the WAVE model set-up*

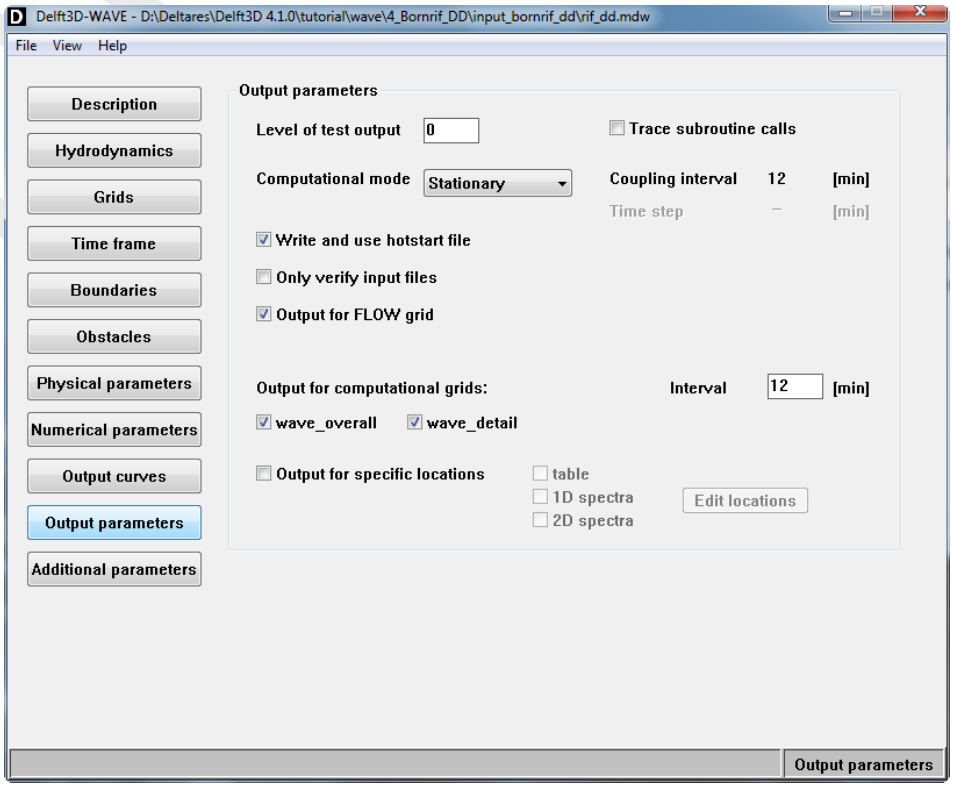

Figure 1.40: *Overview of out parameters in Delft3D WAVE.*

1.5.4.2 Background

➢ Go back to the main Delft3D menu and click *Batch* (see Figure 1.33).
➢ Click *Prepare DD*.
➢ Click *Start*.

Or the complete simulation can be started with the following batch-file:

```
@ echo off
rem ============ set exedir for FLOW executable
set exedirflow=d:\delft3d\win32\flow\bin

rem ============ set exedir for WAVE executable
set exedirwave=d:\delft3d\win32\wave\bin

rem ============ set ddb-file
set ddb-file=inside_outside.ddb
echo -c %ddb-file% > delftflow.inp

rem ============ remove old output files
del runid
del TMP*.*
del trid*.*
del trih*.*
del trim*.*
del *.msg
del com*.*
del fourier*.*
del md-diag*.*

echo === start wave.exe ===
start %exedirwave%\wave.exe rif_dd.mdw 1

echo === start delftflow.exe ===
%exedirflow%\delftflow.exe delftflow.inp delftflow.out delft3d

echo === end of the simulation ===

rem ============ break before deleting output and closing window
pause

rem ============ remove new output files
rem del runid
rem del TMP*.*
rem del *.msg
rem del com*.*
rem del fourier*.*
rem del md-diag*.*
```

1.5.4.3 Output files

The simulation results are stored in:

◇ <trim-rif_inside.dat> and <trim-rif_inside.def>
◇ <trim-rif_outside.dat> and <trim-rif_outside.def>
◇ <wavm-rif_dd-wave_detail.dat> and <wavm-rif_dd-wave_detail.def>
◇ <wavm-rif_dd-wave_overall.dat> and <wavm-rif_dd-wave_overall.def>

With the use of MATLAB or Delft3D-QUICKPLOT, the results can be visualised.

DRAFT

DRAFT

References

Deltares, 2024. *Delft3D-FLOW User Manual*. Deltares, 3.14 ed.

RGFGRID UM, 2016. *Delft3D-RGFGRID User Manual*. Deltares, 5.00 ed.

Wilkens, 1999. *Bar Morphology Bornrif: modelling the evolution from 1982 to 1987*. Tech. rep., WL | Delft Hydraulics, Delft, The Netherlands. M.Sc.Thesis Univesity of Twente.

DRAFT

DRAFT