



Simulation packages for rivers, estuaries and seas

D-HYDRO SUITE



Dutch Delta Systems

D-Flow Flexible Mesh

User Manual

D-Flow Flexible Mesh

D-Flow FM in Delta Shell

User Manual

Version: 1.1.148
Revision: 41928

September 14, 2015

D-Flow Flexible Mesh, User Manual

Published and printed by:

Deltares
Boussinesqweg 1
2629 HV Delft
P.O. 177
2600 MH Delft
The Netherlands

telephone: +31 88 335 82 73
fax: +31 88 335 85 82
e-mail: info@deltares.nl
www: <https://www.deltares.nl>

For sales contact:

telephone: +31 88 335 81 88
fax: +31 88 335 81 11
e-mail: sales@deltaressystems.nl
www: <http://www.deltaressystems.nl>

For support contact:

telephone: +31 88 335 81 00
fax: +31 88 335 81 11
e-mail: support@deltaressystems.nl
www: <http://www.deltaressystems.nl>

Copyright © 2015 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

1 A guide to this manual	1
1.1 Introduction	1
1.2 Overview	1
1.3 Manual version and revisions	2
1.4 Typographical conventions	2
1.5 Changes with respect to previous versions	3
2 Introduction to D-Flow Flexible Mesh	5
2.1 Areas of application	5
2.2 Standard features	5
2.3 Special features	6
2.4 Important differences compared to Delft3D-FLOW	6
2.5 Coupling to other modules	7
2.6 Installation and computer configuration	7
2.7 Examples	8
3 Getting started	9
3.1 Introduction	9
3.2 Overview of D-Flow FM GUI	9
3.2.1 Project window	10
3.2.2 Central (map) window	11
3.2.3 Map window	11
3.2.4 Messages window	12
3.2.5 Time navigator window	12
3.3 Dockable views	12
3.3.1 Docking tabs separately	12
3.3.2 Multiple tabs	13
3.4 Ribbons and toolbars	15
3.4.1 Ribbons (hot keys)	15
3.4.2 File	16
3.4.3 Home	17
3.4.4 View	17
3.4.5 Tools	18
3.4.6 Map	18
3.4.7 Scripting	19
3.4.8 Shortcuts	20
3.4.9 Quick access toolbar	20
3.5 Basic steps to set up a D-Flow FM model	21
3.5.1 Add a D-Flow FM model	21
3.5.2 Set up a D-Flow FM model	23
3.5.3 Converting a Delft3D-FLOW model into D-Flow FM	23
3.5.4 Validate D-Flow FM model	23
3.5.5 File tree	24
3.5.6 Run D-Flow FM model	24
3.5.7 Inspect model output	24
3.5.8 Import/export or delete a D-Flow FM model	25
3.5.9 Save project	27
3.5.10 Exit Delta Shell	27
3.6 Important differences compared to Delft3D-FLOW GUI	27
3.6.1 Project vs model	27
3.6.2 Load/save vs import/export	27
3.6.3 Working from the map	27

3.6.4	Coordinate conversion	28
3.6.5	Model area	28
3.6.6	Integrated models (model couplings)	29
3.6.7	Ribbons (hot keys)	29
3.6.8	Context menus	30
3.6.9	Scripting	30
4	All about the modelling process	31
4.1	Introduction	31
4.2	MDU-file and attribute files	31
4.3	Filenames and conventions	32
4.4	Setting up a D-Flow FM model	32
4.4.1	General	33
4.4.1.1	Depth layer specification	33
4.4.1.2	Model coordinate system	34
4.4.1.3	Angle of latitude	35
4.4.2	Area 2D	36
4.4.2.1	Grid snapped features	37
4.4.2.2	Observation points	38
4.4.2.3	Observation cross-sections	39
4.4.2.4	Thin dams	40
4.4.2.5	Fixed weirs	41
4.4.2.6	Land boundaries	43
4.4.2.7	Dry points and dry areas	43
4.4.2.8	Dry areas	46
4.4.2.9	Pumps	46
4.4.2.10	Weirs	47
4.4.2.11	Gates	49
4.4.3	Computational grid	49
4.4.4	Bathymetry	49
4.4.5	Time frame	50
4.4.6	Processes	51
4.4.7	Initial conditions	52
4.4.8	Boundary conditions	54
4.4.8.1	Specification of boundary locations (support points)	54
4.4.8.2	Boundary data editor (forcing)	56
4.4.8.3	Import/export boundary conditions from the project tree	70
4.4.8.4	Overview of boundary conditions in attribute table (non-editable)	72
4.4.9	Physical parameters	73
4.4.9.1	Constants	73
4.4.9.2	Roughness	73
4.4.9.3	Viscosity	74
4.4.9.4	Wind	75
4.4.9.5	Heat Flux model	75
4.4.9.6	Tidal forces	75
4.4.10	Sources and sinks	75
4.4.11	Numerical parameters	77
4.4.12	Output parameters	79
4.4.13	Miscellaneous	81
4.5	Save project, MDU file and attribute files	81
5	Running a model	83
5.1	Running a simulation	83
5.2	Parallel calculations using MPI	83

5.2.1	Introduction	83
5.2.2	Partitioning the model	84
5.2.2.1	Partitioning the mesh	84
5.2.3	Partitioning the MDU file	86
5.2.3.1	Remaining model input	87
5.2.4	Running a parallel job	87
5.2.5	Visualizing the results of a parallel run	88
5.2.5.1	Plotting all partitioned map files with Delft3D-QUICKPLOT	88
5.2.5.2	Merging multiple map files into one	89
5.3	Running a scenario using Delta Shell	89
5.4	Running a scenario using a batch script	90
5.5	Run time	91
5.5.1	Multi-core performance improvements by OpenMP	91
5.6	Files and file sizes	92
5.6.1	History file	92
5.6.2	Map file	92
5.6.3	Restart file	93
5.7	Command-line arguments	93
5.8	Frequently asked questions	94
6	Visualize results	97
6.1	Introduction	97
6.2	Visualization with Delta Shell	97
6.3	Visualization with Quickplot	98
6.4	Visualization with Muppet	99
6.5	Visualization with Matlab	99
6.6	Visualization with Python	99
7	Hydrodynamics	101
7.1	Introduction	101
7.2	General background	101
7.2.1	Range of applications of D-Flow FM	101
7.2.2	Physical processes	102
7.2.3	Assumptions underlying D-Flow FM	102
7.3	Hydrodynamic processes	104
7.3.1	Topological conventions	105
7.3.2	Conservation of mass and momentum	108
7.3.2.1	Continuity equation	108
7.3.2.2	Momentum equations in horizontal direction	109
7.3.2.3	Vertical velocities	109
7.3.3	The hydrostatic pressure assumption	109
7.3.4	The Coriolis force	110
7.3.5	Diffusion of momentum	110
7.4	Hydrodynamics boundary conditions	111
7.4.1	Open boundary conditions	111
7.4.1.1	The location of support points	112
7.4.1.2	Physical information	113
7.4.1.3	Example	118
7.4.1.4	Miscellaneous	120
7.4.2	Vertical boundary conditions	121
7.4.3	Shear-stresses at closed boundaries	121
7.5	Artificial mixing due to sigma-coordinates	122
7.6	Secondary flow	126
7.6.1	Definition	127

7.6.2	Depth-averaged continuity equation	127
7.6.3	Momentum equations in horizontal direction	127
7.6.4	Effect of secondary flow on depth-averaged momentum equations	128
7.6.5	The depth averaged transport equation for the spiral motion intensity	129
7.7	Drying and flooding	130
7.7.1	Definitions	130
7.7.1.1	Piecewise constant approach for the bed level	131
7.7.1.2	Piecewise linear approach for the bed levels	132
7.7.1.3	Hybrid bed level approach	133
7.7.2	Specification in Delta Shell	133
7.8	Intakes, outfalls and coupled intake-outfalls	134
7.9	Equations of state for the density	135
7.10	Tide generating forces	136
8	Transport of matter	139
8.1	Introduction	139
8.2	Some words about suspended sediment transport	140
8.3	Transport processes	140
8.3.1	Advection	140
8.3.2	Diffusion	141
8.3.3	Sources and sinks	142
8.3.4	Forester filter	142
8.4	Transport boundary and initial conditions	143
8.4.1	Open boundary conditions	143
8.4.2	Closed boundary conditions	143
8.4.3	Vertical boundary conditions	143
8.4.4	Thatcher-Harleman boundary conditions	144
8.4.5	Initial conditions	144
9	Turbulence	147
9.1	<i>k</i> - ε turbulence model	149
10	Heat transport	153
10.1	Heat balance	155
10.2	Solar radiation	156
10.3	Atmospheric radiation (long wave radiation)	158
10.4	Back radiation (long wave radiation)	159
10.5	Effective back radiation	159
10.6	Evaporative heat flux	159
10.7	Convective heat flux	161
11	Wind	163
11.1	Definitions	163
11.1.1	Nautical convention	163
11.1.2	Drag coefficient	163
11.2	File formats	166
11.2.1	Defined on the computational grid	167
11.2.1.1	Specification of uniform wind through velocity components	167
11.2.1.2	Specification of uniform wind through magnitude and direction	168
11.2.2	Defined on an equidistant grid	168
11.2.3	Defined on a curvilinear grid	170
11.2.4	Defined on a spiderweb grid	171
11.2.5	Combination of several wind specifications	173
11.3	Masking of points in the wind grid from interpolation ('land-sea mask')	174

12 Hydraulic structures	177
12.1 Introduction	177
12.1.1 Model input	177
12.2 Structures	177
12.2.1 Fixed weirs	177
12.2.2 Adjustable weir	178
12.2.2.1 Barriers	178
12.2.3 Thin dams	178
12.2.4 Gates	178
12.2.5 Pumps	179
13 Bedforms and vegetation	181
13.1 Bedform heights	181
13.2 Trachytopes	181
13.2.1 Trachytype classes	181
13.2.2 Averaging and accumulation of trachytopes	188
13.3 (Rigid) three-dimensional vegetation model	189
14 Coupling with D-Waves (SWAN)	191
14.1 Getting started	191
14.1.1 Input D-Flow FM	191
14.1.2 Input D-Waves	192
14.1.3 Input d_hydro	193
14.1.4 Online process order	195
14.1.5 Related files	196
14.2 Forcing by radiation stress gradients	196
14.3 Stokes drift and mass flux	198
14.4 Streaming	198
14.5 Enhancement of the bed shear-stress by waves	199
15 Coupling with D-RTC (RTC-Tools)	203
15.1 Introduction	203
15.2 Getting started	203
15.2.1 User interface: the first steps	203
15.2.2 Input D-Flow FM	204
15.2.3 Input D-RTC	204
15.2.4 Input d_hydro	205
15.2.5 Online process order	205
16 Coupling with D-Water Quality (Delwaq)	207
16.1 Introduction	207
16.2 Offline versus online coupling	207
16.3 Creating output for D-Water Quality	207
16.4 Current limitations	208
17 Sediment transport and morphology	209
17.1 General formulations	209
17.1.1 Introduction	209
17.1.2 Suspended transport	209
17.1.3 Effect of sediment on fluid density	210
17.1.4 Sediment settling velocity	210
17.1.5 Dispersive transport	211
17.1.6 Three-dimensional wave effects	211
17.1.7 Initial and boundary conditions	211
17.1.7.1 Initial condition	212

17.1.7.2	Boundary conditions	212
17.2	Cohesive sediment	213
17.2.1	Cohesive sediment settling velocity	213
17.2.2	Cohesive sediment dispersion	214
17.2.3	Cohesive sediment erosion and deposition	214
17.2.4	Interaction of sediment fractions	215
17.2.5	Influence of waves on cohesive sediment transport	215
17.2.6	Inclusion of a fixed layer	215
17.2.7	Inflow boundary conditions cohesive sediment	216
17.3	Non-cohesive sediment	216
17.3.1	Non-cohesive sediment settling velocity	216
17.3.2	Non-cohesive sediment dispersion	217
17.3.2.1	Using the algebraic or $k - L$ turbulence model	217
17.3.2.2	Using the $k - \varepsilon$ turbulence model	217
17.3.3	Reference concentration	218
17.3.4	Non-cohesive sediment erosion and deposition	218
17.3.5	Inclusion of a fixed layer	222
17.3.6	Inflow boundary conditions non-cohesive sediment	222
17.4	Bedload sediment transport of non-cohesive sediment	223
17.4.1	Basic formulation	223
17.4.2	Suspended sediment correction vector	224
17.4.3	Interaction of sediment fractions	224
17.4.4	Inclusion of a fixed layer	225
17.4.5	Calculation of bedload transport at open boundaries	225
17.4.6	Bedload transport at velocity points	225
17.4.7	Adjustment of bedload transport for bed-slope effects	226
17.5	Transport formulations for non-cohesive sediment	229
17.5.1	Van Rijn (1993)	229
17.5.2	Engelund-Hansen (1967)	234
17.5.3	Meyer-Peter-Muller (1948)	234
17.5.4	General formula	235
17.5.5	Bijker (1971)	235
17.5.5.1	Basic formulation	235
17.5.5.2	Transport in wave propagation direction (Baird-approach)	237
17.5.6	Van Rijn (1984)	239
17.5.7	Soulsby/Van Rijn	240
17.5.8	Soulsby	242
17.5.9	Ashida-Michiue (1974)	244
17.5.10	Wilcock-Crowe (2003)	245
17.5.11	Gaeuman et al. (2009) laboratory calibration	246
17.5.12	Gaeuman et al. (2009) Trinity River calibration	246
17.6	Morphological updating	247
17.6.1	Bathymetry updating including bedload transport	249
17.6.2	Erosion of (temporarily) dry points	250
17.6.3	Dredging and dumping	250
17.6.4	Bed composition models and sediment availability	251
17.7	Specific implementation aspects	252
18	Tutorial	255
18.1	Introduction	255
18.1.1	Setup of the tutorial	255
18.1.2	Basic grid concepts	255
18.2	Tutorial 1: Creating a curvilinear grid	257
18.3	Tutorial 2: Creating a triangular grid	260

18.4 Tutorial 3: Coupling multiple separate grids	262
18.5 Tutorial 4: Inserting a bathymetry	264
18.6 Tutorial 5: Imposing boundary conditions	267
18.7 Tutorial 6: Defining output locations	270
18.8 Tutorial 7: Defining computational parameters	271
18.9 Tutorial 8: Running a model simulation	273
18.10 Tutorial 9: Viewing the output of a model simulation	275
19 Calibration and data assimilation	279
20 Calibration and data assimilation	281
20.1 Introduction	281
20.2 Getting started with OpenDA	281
20.3 The OpenDA black box model wrapper for D-Flow FM	282
20.4 OpenDA configuration	282
20.4.1 Main configuration file and the directory structure	282
20.4.2 The algorithm configuration	284
20.4.3 The stochObserver configuration	284
20.4.3.1 NoosTimeSeriesStochObserver	284
20.4.3.2 IoObjectStochObserver	285
20.4.4 The stochModel configuration	286
20.4.5 D-Flow FM files and the OpenDA dataObjects configuration	286
20.4.5.1 Start and end time in the model definition file (.mdu)	287
20.4.5.2 External forcings (.xyz)	287
20.4.5.3 Boundary time series (.tim)	289
20.4.5.4 Meteorological boundary conditions (.amu, .amv, .amp)	289
20.4.5.5 Result time series (_his.n)	289
20.4.5.6 Restart file (_map.nc)	290
20.5 Generating noise	290
20.6 Examples of the application of OpenDA for D-Flow FM	292
20.6.1 Example 1: Calibration of the roughness parameter	292
20.6.2 Example 2: EnKF with uncertainty in the tidal components	294
20.6.3 Example 3: EnKF with uncertainty in the inflow velocity	295
20.6.4 Example 4: EnKF with uncertainty in the inflow condition for salt	295
20.6.5 Example 5: EnKF with uncertainty on the wind direction	296
20.6.6 Example 6: EnKF with the DCSM v5 model and uncertainty on the wind direction	296
References	297
A The master definition file	303
B Attribute files	309
B.1 Introduction	309
B.2 Polyline/polygon file	309
B.3 Sample file	310
B.4 Time series file (ASCII)	310
B.5 The external forcings file	311
B.5.1 Old style external forcings	311
B.5.2 New style external forcing (boundary conditions only)	312
B.5.3 Accepted quantity names	312
B.6 Trachytopes	313
B.6.1 ARL-file (Area Roughness on Links)	314
B.6.1.1 Example	314
B.6.1.2 Conversion from Delft3D input files	314

B.6.2	TTD-file	315
B.6.2.1	General format	315
B.6.2.2	Example	315
B.6.2.3	Discharge dependent format	315
B.6.2.4	Water level dependent format	316
B.6.2.5	Supported roughness formulations	317
B.7	Sources and sinks	318
B.8	Dry points and areas	318
B.9	Structure INI file	318
B.10	Space varying wind and pressure	319
B.10.1	Meteo on equidistant grids	319
B.10.2	Curvilinear data	322
B.10.3	Spiderweb data	324
C	Initial conditions and spatially varying input	331
C.1	Introduction	331
C.2	Supported quantities	331
C.2.1	Water levels	331
C.2.2	Initial velocities	331
C.2.3	Salinity	331
C.2.4	Temperature	331
C.2.5	Tracers	331
C.2.6	Sediment	332
C.2.7	Physical coefficients	332
C.3	Supported file formats	332
C.3.1	Inside-polygon option	332
C.3.2	Sample file	333
C.3.3	Vertical profile file	333
C.3.4	Map file	333
C.3.5	Restart file	333
D	Boundary conditions specification	335
D.1	Supported boundary types	335
D.1.1	Open boundaries	335
D.1.2	Astronomic boundary conditions	335
D.1.3	Astronomic correction factors	335
D.1.4	Harmonic flow boundary conditions	335
D.1.5	QH-relation flow boundary conditions	336
D.1.6	Time-series flow boundary conditions	336
D.1.7	Time-series transport boundary conditions	336
D.1.8	Time-series for the heat model parameters	336
D.2	Boundary signal file formats	336
D.2.1	The <cmp> format	336
D.2.2	The <qh> format	337
D.2.3	The <bc> format	337
D.2.4	The NetCDF-format for boundary condition time-series	339
E	Output files	341
E.1	Diagnostics file	341
E.2	Demanding output	342
E.2.1	The MDU-file	342
E.2.2	Observation points	342
E.2.3	Moving observation points	342
E.2.4	Cross-sections	343

E.3	NetCDF output files	343
E.3.1	Timeseries as NetCDF his-file	343
E.3.2	Spatial fields as NetCDF map-file	345
E.3.3	Restart files as NetCDF rst-file	347
F	Spatial editor	349
F.1	Introduction	349
F.2	General	349
F.2.1	Overview of spatial editor	349
F.2.2	Import/export dataset	350
F.2.3	Activate (spatial) model quantity	351
F.2.4	Colorscale	351
F.2.5	Render mode	352
F.2.6	Context menu	354
F.3	Quantity selection	355
F.4	Geometry operations	356
F.4.1	Polygons	357
F.4.2	Lines	357
F.4.3	Points	358
F.5	Spatial operations	359
F.5.1	Import point cloud	359
F.5.2	Crop	361
F.5.3	Erase	362
F.5.4	Set value	362
F.5.5	Contour	363
F.5.6	Gradient	365
F.5.7	Interpolate	366
F.5.8	Smoothing	368
F.5.9	Overwrite (single) value	370
F.6	Operation stack	371
F.6.1	Stack workflow	371
F.6.2	Edit operation properties	372
F.6.3	Enable/disable operations	372
F.6.4	Delete operations	375
F.6.5	Refresh stack	376
F.6.6	Quick links	377
F.6.7	Import/export	377



List of Figures

3.1	Start-up lay-out Delta Shell	9
3.2	Project tree of D-Flow FM plugin	10
3.3	Central map with contents of the D-Flow FM plug-in	11
3.4	Map tree controlling map contents	11
3.5	Log of messages, warnings and errors in message window	12
3.6	Time navigator in Delta Shell	12
3.7	Docking windows on two screens within the Delta Shell framework.	12
3.8	Bringing the Undo/Redo window to the front	13
3.9	Docking the Undo/Redo window.	14
3.10	Auto hide the Undo / Redo window	15
3.11	Perform operations using the hot keys	15
3.12	The <i>File</i> ribbon.	16
3.13	The Delta Shell options dialog.	17
3.14	The <i>Home</i> ribbon.	17
3.15	The <i>View</i> ribbon.	17
3.16	The <i>Tools</i> ribbon.	18
3.17	The <i>Map</i> ribbon.	18
3.18	The ribbon with minimized categories.	19
3.19	The scripting <i>ribbon</i> within Delta Shell.	19
3.20	The quick access toolbar.	21
3.21	Adding a new model from the ribbon	21
3.22	Adding a new model using the Right Mouse Button on “project1” in the project tree	22
3.23	Select “D-Flow FM model”	22
3.24	Validate model	23
3.25	Validation report	24
3.26	Run model	24
3.27	Output of wave model in project tree	25
3.28	Import wave model from project tree	26
3.29	Import wave model from file ribbon	26
3.30	Set map coordinate system using RMB	28
3.31	Select a coordinate system using the quick search bar	28
3.32	Perform operations using the hot keys	29
4.1	Select model wizard	32
4.2	Overview of general tab	33
4.3	Depth layers specification window	34
4.4	Coordinate system wizard	35
4.5	Overview of geographical features	36
4.6	Overview of map ribbon. Left red box highlights “FM Region 2D / 3D” menu containing icons used to add features. Right red box highlights “Edit” menu which contains icons used to edit geographical features (move/add)	36
4.7	Example of expanded grid snapped features attribute in map tree	37
4.8	Example of grid snapped features displayed on the central map	38
4.9	Geographical and grid snapped representation of an observation point	38
4.10	Attribute table with observation points	39
4.11	Geographical and grid snapped representation of a cross section	39
4.12	Attribute table with observation cross sections	40
4.13	Geographical and grid snapped representation of a thin dam	40
4.14	Attribute table with thin dams	41
4.15	Geographical and grid snapped representation of a fixed weir	41
4.16	Schematic representation of a fixed weir	41

4.17	Attribute table with fixed weirs	42
4.18	Fixed weir editor	42
4.19	Geographical representation of a land boundary	43
4.20	Attribute table with land boundaries	43
4.21	Geographical and grid snapped representation of several dry points	44
4.22	Attribute table with dry points	44
4.23	Geographical and grid snapped representation of a dry area	45
4.24	Attribute table with dry areas	45
4.25	Polygon for pump (a) and adjustment of physical properties (b).	46
4.26	Selection of the pumps	47
4.27	Polygon for adjustable weir (a) and adjustment of geometrical and temporal conditions (b).	47
4.28	Time series for crest level.	48
4.29	Time series for crest level.	48
4.30	Polygon for gate (a) and adjustment of geometrical and temporal conditions (b).	49
4.31	Bathymetry activated in the spatial editor	50
4.32	Overview time frame tab	50
4.33	Overview processes tab	52
4.34	Initial conditions in the project tree	52
4.35	The 'Initial Conditions' tab where you can specify the uniform values and the layer distributions of the active physical quantities.	53
4.36	Initial water levels activated in the spatial editor	53
4.37	Selecting 3 dimensional initial fields from the dropdown box in the 'Map' ribbon to edit them in the spatial editor	53
4.38	Restart files in output states folder	54
4.39	Restart file in initial conditions attribute	54
4.40	Adding a boundary support point on a polyline in the central map. By double clicking on the polyline in the map, the boundary condition editor will open to edit the forcing data on the polyline.	55
4.41	Polyline added in project tree under 'Boundary Conditions'. By double clicking on the name of the polyline, the boundary condition editor will open to edit the forcing data on the polyline.	55
4.42	Geometry edit options in 'Map' ribbon	56
4.43	Edit name of polyline/boundary in Boundaries tab	56
4.44	Overview of the boundary data editor	57
4.45	Process and quantity selection in the boundary data editor	58
4.46	Activate a support point	59
4.47	Specification of time series in the boundary data editor (left panel)	59
4.48	Window for generating series of time points	60
4.49	Csv import wizard: csv file selection	61
4.50	Clipboard/csv import wizard: specification of how data should be parsed into columns	62
4.51	Clipboard/csv import wizard: specification of how values should be parsed and columns should be mapped	63
4.52	Window for entering input to download boundary data from WPS	64
4.53	Specification of harmonic components in boundary data editor	65
4.54	Selection of astronomical components from list (after pressing 'select components')	66
4.55	Suggestions for astronomical components in list	67
4.56	Editing harmonic/astronomic components and their corrections	67
4.57	Specification of a Q-h relationship	68
4.58	Selection of vertically uniform or varying boundary conditions in case of a 3D model	68

4.59	Overview of the layer view component of the boudary conditions editor. In the table the user can edit the vertical positions of the boundary conditions as a percentage from the bed. In the view left of the table, the user can see the vertical positions of the boundary conditions (indicated by number corresponding to the table) relative to the model layers.	69
4.60	Specification of boundary forcing data (in this example for salinity) at 3 positions in the vertical	69
4.61	Example of active and total signal for multiple water level data series on one support point	70
4.62	Importing or exporting boundary features - both polylines (*.pli) and forcing (*.bc) - from the project tree using the RMB	70
4.63	Import or export a *.pli file as is or with coordinate transformation.	71
4.64	Import or export a *.pli file as is or with coordinate transformation.	71
4.65	Import or export a *.pli file as is or with coordinate transformation.	72
4.66	Overview of all boundary conditions in attribute table	72
4.67	The physical parameters in the project tree	73
4.68	The section of the 'Physical Parameters' tab where you can specify roughness related parameters and formulations.	73
4.69	Roughness activated in the spatial editor to create/edit a spatially varying field	74
4.70	The section of the 'Physical Parameters' tab where you can specify (uniform) values for the horizontal and vertical eddy viscosity and diffusivity.	74
4.71	Viscosity activated in the spatial editor to create/edit a spatially varying field	75
4.72	Overview of parameters in sub-tab Wind	75
4.73	Activate the sources and sinks editing icon in the 'Map' ribbon	76
4.74	Add sources and sinks in the central map using the 'Sources and sinks' icon.	76
4.75	Sources and sinks appearing in the project tree	77
4.76	Specifying time series for sources and sinks in the sources and sinks editor	77
4.77	Output parameters tab	79
4.78	Overview output parameters tab	80
4.79	Model/data import wizard	81
5.1	Partitioning exporter dialog	84
5.2	Domain selector in Delft3D-QUICKPLOT for partitioned map files.	88
5.3	Selecting the model you want to run in project tree	89
5.4	Group Run in Home ribbon	90
5.5	Run console Delta Shell	90
6.1	Example of output folders in the map project tree	97
6.2	Useful map visualization options in the Delft3D-QUICKPLOT	98
6.3	Example of the Muppet visualization of the D-Flow FM map output file	99
7.1	Example of σ -grid (left) and Z-grid (right).	105
7.2	Flexible mesh topology	106
7.3	Two conventional definitions of the cell center of a triangle: the circumcenter and the mass center.	107
7.4	Nearly perfect orthogonality and smoothness along the edge connecting two triangles. Black lines/dots are network links/nodes, blue lines/dots are flow links/nodes.	107
7.5	Poor mesh properties due to violating either the smoothness or the orthogonality at the edge connecting two triangles. Black lines/dots are network links/nodes, blue lines/dots are flow links/nodes.	108
7.6	Input for map projection for specifying Coriolis parameter on the grid.	110
7.7	Input parameters for horizontal and vertical eddy viscosities.	111

7.8	Fictitious boundary "cells" near the shaded boundary; \vec{x}_{Lj} is the fictitious "cell" center near boundary face j ; $\vec{x}_{R(j)}$ is the inner-cell center; \vec{b}_j is the point on face j that is nearest to the inner-cell center	112
7.9	Delta-Shell view of a simple channel covered by a straightforward Cartesian grid. Boundary conditions are prescribed at the left hand side and the right hand size of the domain.	118
7.10	Example of a hydrostatic consistent and inconsistent grid; (a) $H\delta\sigma > \sigma \frac{\partial H}{\partial x} \delta x$, (b) $H\delta\sigma < \sigma \frac{\partial H}{\partial x} \delta x$	123
7.11	Finite Volume for diffusive fluxes and pressure gradients	123
7.12	Left and right approximation of a strict horizontal gradient	124
7.13	Vertical profile secondary flow (v) in river bend and direction bed stress	126
7.14	Definition of the water levels, the bed levels and the velocities in case of two adjacent triangular cells.	131
7.15	Specification of the conveyance option in Delta Shell.	133
7.16	Specification of the bed level treatment type in Delta Shell.	133
7.17	Specification of the hybrid bed options (with keywords <code>blminabove</code> and <code>blmeanbelow</code>).	134
10.1	Overview of the heat exchange mechanisms at the surface	154
10.2	Co-ordinate system position Sun δ : declination; θ : latitude; ωt : angular speed	158
11.1	Nautical conventions for the wind.	163
11.2	Prescription of the dependency of the wind drag coefficient C_d on the wind speed is achieved by means of at least 1 point, with a maximum of 3 points.	164
11.3	Grid definition of the spiderweb grid for cyclone winds.	171
12.1	Selection of structures in the toolbar.	177
14.1	Schematic view of non-linear interaction of wave and current bed shear-stresses (from Soulsby <i>et al.</i> (1993b, Figure 16, p. 89))	200
14.2	Inter-comparison of eight models for prediction of mean and maximum bed shear-stress due to waves and currents (from Soulsby <i>et al.</i> (1993b, Figure 17, p. 90))	201
15.1	An <i>Integrated model</i> in the Project window	203
15.2	Example of a <i>Control flow</i> in D-RTC	204
17.1	Selection of the k_{mx} layer; where a is Van Rijn's reference height	219
17.2	Schematic arrangement of flux bottom boundary condition	219
17.3	Approximation of concentration and concentration gradient at bottom of k_{mx} layer	220
17.4	Setting of bedload transport components at velocity points	226
17.5	Morphological control volume and bedload transport components	249
18.1	Topology and definitions for a grid as used in D-Flow FM.	256
18.2	Nearly perfect orthogonality and smoothness along the edge connecting two triangles. Black lines/dots are network links/nodes, blue lines/dots are flow links/nodes.	256
18.3	Poor grid properties due to violating either the smoothness or the orthogonality at the edge connecting two triangles. Black lines/dots are network links/nodes, blue lines/dots are flow links/nodes.	257
18.4	Delta Shell after opening a D-Flow FM model.	257
18.5	Settings for the 'grow grid from splines' procedure.	258
18.6	Generated curvilinear grid after the new 'grow grid from splines' procedure.	259

18.7 Orthogonality of the generated curvilinear grid after the new 'grow grid from splines' procedure after some orthogonalisation iterations.	259
18.8 Polygon that envelopes the area in which an unstructured grid is aimed to be established.	261
18.9 Unstructured grid, after having refined the polygon.	262
18.10 Connection of the river grid and the unstructured grid. The red lines show the inserted grid lines used to couple the two grids manually.	263
18.11 Orthogonality of the integrated grid, containing the curvilinear part, the triangular part and the coupling between the two grids.	264
18.12 Project menu.	265
18.13 Spatial Operations menu bar (located at the upper right part of the Delta Shell).	265
18.14 Interpolated bottom levels values at the grid.	266
18.15 Interpolated bottom levels values at the grid.	266
18.16 Line along the sea boundary.	268
18.17 Boundary conditions seaside.	268
18.18 Boundary condition riverside.	270
18.19 Overview cross sections and observation points.	271
18.20 The time frame of the simulation.	272
18.21 Relevant processes incorporated within the simulation.	272
18.22 Imposed initial conditions for the simulation.	272
18.23 Optional output parameters for the computation.	272
18.24 Menu that appears if one would like to save the project.	274
18.25 View of Delta Shell when running a model.	275
18.26 View of Delta Shell in combination with Openstreet maps.	276
18.27 View of Delta Shell in combination with Openstreet maps, available to select a location for timeseries in.	277
 20.1 Visualisation of the EnKF computation results from OpenDA for a certain observation point. The dots represent the observed data, the black line represents the original computation with D-Flow FM (without Kalman filtering) and the red line represents the D-Flow FM computation with Kalman filtering.	295
 B.1 Illustration of the data to grid conversion for meteo input on a separate curvilinear grid	325
B.2 Wind definition according to Nautical convention	326
B.3 Spiderweb grid definition	327
 F.1 Overview of spatial editor functionality in Map ribbon	349
F.2 Importing a point cloud into the project using the context menu on "project" in the project tree	350
F.3 Activate the imported point cloud in the spatial editor by double clicking it in the project tree	350
F.4 Activate the imported point cloud in the spatial editor by selecting it from the dropdown box in the Map ribbon	351
F.5 Activate the colorscale using the button in the map ribbon (top) and the colorscale will become visible in the map window (left). You can adjust/decrease the range of the colorscale using the sliders at the top and bottom of the colorbar (right).	351
F.6 Edit the colorscale properties using the context menu on the active layer in the Map Tree	352
F.7 Select the rendermode for the active layer in the property grid.	353
F.8 Example of a coverage rendered as colored numbers.	354
F.9 Selecting a smoothing operation for a polygon geometry from the context menu (using context menu)	355

F.10	Activating a spatial quantity by double clicking it in the project tree (in this example 'Initial Water Level')	356
F.11	Activating a spatial quantity by selecting it from the dropdown box in the 'Map' ribbon	356
F.12	Overview of the available geometry operations in the 'Map' ribbon	356
F.13	Activating the polygon operation and drawing polygons in the central map.	357
F.14	Activating the line operation and drawing lines in the central map.	358
F.15	Activating the 'Add points' operation, drawing them in the central map and assigning a value to them.	359
F.16	Overview of the available spatial operations in the 'Map' ribbon	359
F.17	Importing a point cloud using the 'Import' operation from the 'Map' ribbon	360
F.18	Option to perform a coordinate transformation on the imported point cloud	360
F.19	Appearance of import point cloud operation in the operations stack	361
F.20	Performing a crop operation on a point cloud with a polygon using 'Crop' from the 'Map' ribbon	361
F.21	Appearance of crop operation in the operations stack	361
F.22	Performing an erase operation on a point cloud with a polygon using 'Erase' from the 'Map' ribbon	362
F.23	Appearance of erase operation in the operations stack	362
F.24	Performing a set value operation (e.g. overwrite) on a point cloud with a polygon using 'Set Value' from the 'Map' ribbon	363
F.25	Appearance of set value operation in the operations stack	363
F.26	Import a nautical chart as a georeferenced tiff file	364
F.27	Set the right map coordinate system for the geotiff	364
F.28	Performing a contour operation on a nautical chart using lines to define the contours and 'Contour' from the 'Map' ribbon	364
F.29	Bring the sample set to the front if it appears behind the nautical chart	365
F.30	Appearance of contour operation in the operations stack	365
F.31	Performing a gradient operation on a point cloud with a polygon using 'Gradient' from the 'Map' ribbon	366
F.32	Appearance of gradient operation in the operations stack	366
F.33	Performing an interpolation operation on a single sample set (without using a polygon) using 'Interpolate' from the 'Map' ribbon	367
F.34	Appearance of interpolation of 'set1' to the coverage 'Bathymetry' in the operations stack	367
F.35	Performing an interpolation operation on multiple sample sets (without using a polygon) using 'Interpolate' from the 'Map' ribbon	368
F.36	Appearance of interpolation of 'set1' and 'set2' to the coverage 'Bathymetry' in the operations stack	368
F.37	Performing a smoothing operation on a point cloud with a polygon using 'Smoothing' from the 'Map' ribbon	369
F.38	Appearance of smoothing operation in the operations stack	369
F.39	The cursor for the overwrite operation showing the value of the closest coverage point	370
F.40	Performing an overwrite operation on a coverage point using 'Single Value' from the 'Map' ribbon	371
F.41	Appearance of overwrite operation in the operations stack	371
F.42	The 'Operations' panel with the operations stack. In this example 'Bathymetry' is the coverage (e.g. trunk) that is edited. The point clouds 'set 1' and 'set 2' (e.g. branches) are used to construct the 'Bathymetry' coverage.	372
F.43	Input for the operation (top panel), mask for the operation (middle panel) and output of the operation (bottom panel)	373
F.44	Editing the value or 'Pointwise operation' of a 'Set Value' operation using the properties panel	374

F.45	Disabling an operation using the boxed cross icon in the stack menu. The operation will become grey. Note the exlamation marks marking the stack 'out of sync'.	375
F.46	Removing an operation from the stack using the cross icon in the stack menu	375
F.47	Removing an operation from the stack using the context menu on the selected operation	376
F.48	Refresh the stack using the 'Refresh' button so that all operation are (re-)evaluated	376
F.49	Quick link to the original dataset before performing any spatial operations	377
F.50	Quick link to the output after performing all (enabled) operations	377

List of Tables

3.1	Functions and their descriptions within the scripting <i>ribbon</i> of Delta Shell.	19
3.1	Functions and their descriptions within the scripting <i>ribbon</i> of Delta Shell.	20
3.2	Shortcut keys within the scripting editor of Delta Shell.	20
4.1	Overview and description of numerical parameters	78
4.2	Overview and description miscellaneous parameters	81
14.1	Fitting coefficients for wave/current boundary layer model	201
17.1	Additional transport relations	229
17.2	Overview of the coefficients used in the various regression models (Soulsby <i>et al.</i> , 1993a)	243
17.3	Overview of the coefficients used in the various regression models, <i>continued</i> (Soulsby <i>et al.</i> , 1993a)	244
20.1	Directory structure of the OpenDA Ensemble Kalman filtering configuration for the simple Waal D-Flow FM model.	283
20.2	D-Flow FM files that can be manipulated and the corresponding OpenDA class names to be used in the <code>dflowfmWrapper.xml</code> file.	287
A.1	Standard MDU-file with default settings.	303
B.1	List of accepted external forcing quantity names.	312

1 A guide to this manual

1.1 Introduction

This User Manual concerns the hydrodynamic module D-Flow Flexible Mesh (D-Flow FM).

This module is part of several Modelling suites, released by Deltares as Deltares Systems or Dutch Delta Systems. These modelling suites are based on the Delta Shell framework. The framework enables to develop a range of modeling suites, each distinguished by the components and — most significantly — the (numerical) modules, which are plugged in. The modules which are compliant with the Delta Shell framework are released as D-Name of the module, for example: D-Flow Flexible Mesh, D-Waves, D-Water Quality, D-Real Time Control, D-Rainfall Run-off.

Therefore, this user manual is shipped with several modelling suites. In the start-up screen links are provided to all relevant User Manuals (and Technical Reference Manuals) for that modelling suite. It will be clear that the Delta Shell User Manual is shipped with all these modelling suites. Other user manuals can be referenced. In that case, you need to open the specific user manual from the start-up screen in the central window. Some texts are shared in different user manuals, in order to improve the readability.

1.2 Overview

To make this manual more accessible we will briefly describe the contents of each chapter.

If this is your first time to start working with D-Flow FM we suggest you to read [Chapter 3, Getting started](#) and practice the tutorial of [Chapter 18](#). These chapters explain the user interface and guide you through the modeling process resulting in your first simulation.

[Chapter 2: Introduction to D-Flow Flexible Mesh](#), provides specifications of D-Flow FM, such as the areas of application, the standard and specific features provided, coupling to other modules and utilities.

[Chapter 3: Getting started](#), gives an overview of the basic features of the D-Flow FM GUI and will guide you through the main steps to set up a D-Flow FM model.

[Chapter 4: All about the modelling process](#), provides practical information on the GUI, setting up a model with all its parameters and tuning the model.

[Chapter 5: Running a model](#), discusses how to validate and execute a model run. Either in the GUI, or in batch mode and/or in parallel using MPI. It also provides some information on run times and file sizes.

[Chapter 6: Visualize results](#), explains in short the visualization of results within the GUI. It introduces the programs Quickplot and Muppet to visualize or animate the simulation results, and Matlab for general post-processing.

[Chapter 7: Hydrodynamics](#), gives some background information on the conceptual model of the D-Flow FM module.

[Chapter 8: Transport of matter](#), discusses the modeled tranport processes, their governing equations, boundary and initial conditions and user-relevant numerical and physical settings.

[Chapter 9: Turbulence](#) provides a detailed insight into the modelling of turbulence.

[Chapter 10: Heat transport](#), provides a detailed insight into (the modelling of) heat transport.

[Chapter 11: Wind](#), gives background information of how wind fields should be imposed, the relevant definitions and the supported file formats.

[Chapter 12: Hydraulic structures](#), gives background information of the available hydraulic structures in D-Flow FM, the relevant definitions and the supported file formats.

[Chapter 14: Coupling with D-Waves \(SWAN\)](#), provides guidance on the integrated modelling of hydrodynamics (D-Flow FM) and waves (D-Waves).

[Chapter 15: Coupling with D-RTC \(RTC-Tools\)](#), provides guidance on the integrated modelling of hydrodynamics (D-Flow FM) and real time control of hydraulic structures (D-RTC).

[Chapter 16: Coupling with D-Water Quality \(Delwaq\)](#), provides guidance on the integrated modelling of hydrodynamics (D-Flow FM) and water quality (D-Water Quality).

[Chapter 17: Sediment transport and morphology](#), describes the three-dimensional transport of suspended sediment, bedload transport and morphological updating of the bottom.

[Chapter 18: Tutorial](#), gives you some first hands-on experience in using the D-Flow FM GUI to define the input of a simple problem, in validating this input, in executing the simulation and in inspecting the results.

[Chapter 19: Calibration and data assimilation](#), describes the three-dimensional transport of suspended sediment, bedload transport and morphological updating of the bottom.

1.3 Manual version and revisions

This manual applies to:

- ◊ the D-HYDRO Suite, version 2016
- ◊ the Delft3D Flexible Mesh Suite, version 2016

1.4 Typographical conventions

Throughout this manual, the following conventions help you to distinguish between different elements of text.

Example	Description
Waves Boundaries	Title of a window or sub-window. Sub-windows are displayed in the Module window and cannot be moved. Windows can be moved independently from the Module window, such as the Visualisation Area window.

Example	Description
Save	<p>Item from a menu, title of a push button or the name of a user interface input field.</p> <p>Upon selecting this item (click or in some cases double click with the left mouse button on it) a related action will be executed; in most cases it will result in displaying some other (sub-)window.</p> <p>In case of an input field you are supposed to enter input data of the required format and in the required domain.</p>
<\tutorial\wave\swan-curvi> <siu.mdw>	<p>Directory names, filenames, and path names are expressed between angle brackets, <>. For the Linux and UNIX environment a forward slash (/) is used instead of the backward slash (\) for PCs.</p>
“27 08 1999”	<p>Data to be typed by you into the input fields are displayed between double quotes.</p> <p>Selections of menu items, option boxes etc. are described as such: for instance ‘select Save and go to the next window’.</p>
delft3d-menu	<p>Commands to be typed by you are given in the font Courier New, 10 points.</p>
	<p>User actions are indicated with this arrow.</p>
[m/s] [-]	<p>Units are given between square brackets when used next to the formulae. Leaving them out might result in misinterpretation.</p>

Command prompts and terminal output are shown in framed boxes with typewriter font:

```
> ./dflowfm --version
Deltares, D-Flow FM Version 1.1.149.41663, Sep 02 2015, 10:40:42
Compiled with support for:
IntGUI: no
OpenGL: no
OpenMP: yes
MPI : yes
PETSc : yes
METIS : yes
```

1.5 Changes with respect to previous versions

This is the first edition that was published.

2 Introduction to D-Flow Flexible Mesh

D-Flow Flexible Mesh (D-Flow FM) is the latest hydrodynamical simulation program by Deltares. It is part of Deltares's unique, fully integrated computer software suite for a multi-disciplinary approach and 1D, 2D and 3D computations for coastal, river and estuarine areas. It can carry out simulations of flows, waves, water quality and ecology. Sediment transport processes and morphological developments are currently well underway for an upcoming release.

It has been designed for experts and non-experts alike. The Delft3D Flexible Mesh Suite is composed of several modules, grouped around a mutual interface, while being capable to interact with one another. D-Flow FM, which this manual is about, is one of these modules. D-Flow FM is a multi-dimensional (1D, 2D and 3D) hydrodynamic (and transport) simulation program which calculates non-steady flow and transport phenomena that result from tidal and meteorological forcing on structured and unstructured, boundary fitted grids. The term Flexible Mesh in the name refers to the flexible combination of rectilinear or curvilinear grids and unstructured grids composed of triangles, quads a rectilinear or a curvilinear, boundary fitted grid. In 3D simulations, the vertical grid is defined following the σ co-ordinate approach. Fixed z layers and combinations with σ are under development for an upcoming release.

2.1 Areas of application

- ◊ Tide and wind-driven flows (i.e., storm surges).
- ◊ Stratified and density driven flows.
- ◊ River flow simulations.
- ◊ Rural channel networks.
- ◊ Rainfall runoff in urban environments.
- ◊ Simulation of tsunamis, hydraulic jumps, bores and flood waves.
- ◊ Fresh-water river discharges in bays.
- ◊ Salt intrusion.
- ◊ Cooling water intakes and waste water outlets.
- ◊ Transport of dissolved material and pollutants.

2.2 Standard features

- ◊ Tidal forcing.
- ◊ The effect of the Earth's rotation (Coriolis force).
- ◊ Density driven flows (pressure gradients terms in the momentum equations).
- ◊ Advection-diffusion solver included to compute density gradients.
- ◊ Space and time varying wind and atmospheric pressure.
- ◊ Advanced turbulence models to account for the vertical turbulent viscosity and diffusivity based on the eddy viscosity concept. Four options are provided: $k-\varepsilon$, $k-\tau$, algebraic and constant model.
- ◊ Time varying sources and sinks (e.g., river discharges).
- ◊ Simulation of the thermal discharge, effluent discharge and the intake of cooling water at any location and any depth.
- ◊ Robust simulation of drying and flooding of inter-tidal flats and river winter beds.

2.3 Special features

- ◊ Built-in automatic switch converting 2D bottom-stress coefficient to 3D coefficient.
- ◊ Built-in anti-creep correction to suppress artificial vertical diffusion and artificial flow due to σ -grids.
- ◊ Heat exchange through the free water surface.
- ◊ Wave induced stresses and mass fluxes.
- ◊ Influence of waves on the bed shear stress.
- ◊ Optional facility to calculate the intensity of the spiral motion phenomenon in the flow (e.g., in river bends) which is especially important in sedimentation and erosion studies (for depth averaged — 2DH — computations only).
- ◊ Non-linear iterations in the solver can be enabled for accurate flooding results.
- ◊ Optional facility for tidal analysis of output parameters.
- ◊ Optional facility for special structures such as pumping stations, fixed weirs, controllable barriers (1D, 2D and 3D)
- ◊ Default advection scheme suitable for various flow regimes, from bore propagation to eddy shedding.
- ◊ Domain partitioning for parallelized runs on MPI-based High Performance Computing clusters.

2.4 Important differences compared to Delft3D-FLOW

The most noticeable difference between Delft3D-FLOW and D-Flow FM is the use of unstructured grids. Large regions with curvilinear grids can be coupled with much greater freedom than before, using triangles, pentagons and hexagons. Grid refinement (and coarsening) without DD-coupling is now possible in one and the same model grid. Also, 1D networks can be coupled to 2D networks, either adjacent to each other or the 1D network overlying the 2D network. The resulting system is solved implicitly. Finally, many of Delft3D-FLOW's grid restrictions are now gone: since there are no true grid 'rows' and 'columns' anymore, rows of grid cells may be coupled to columns, in any direction and at any position.

In addition to the unstructured grid files, all geometric model input is now specified in spatial coordinates, either in Cartesian or spherical coordinates (x,y or lat,lon), i.e., not in grid-indices. This so-called model-independent coordinates input allows for easy change of a model grid, after which the remaining model input can remain the same.

The new Delta Shell graphical user interface provides a much more powerful and integrated environment for setting up D-Flow FM models and inspecting model input such as time-dependent forcings (boundary conditions and barrier control). Another popular feature within Delta Shell is the use of scripting for running and live interaction with your model.

Coupled running of D-Flow FM with other modules has been extended with real time control of hydraulic structures, as listed in the following section.

D-Flow FM employs some different methods for handling bathymetry values and computing water depths. This can be a cause of differences with Delft3D-FLOW results, so existing users of Delft3D-FLOW are encouraged to read [section 7.7](#) on these topics.

Like Delft3D-FLOW, D-Flow FM implements a finite volume solver on a staggered grid. However, since there is no concept of grid 'rows' and 'columns', there is also no ADI-solver possible. The continuity equation is solved implicitly for all points in a single combined system. Time integration is done explicitly for part of the advection term, and the resulting dynamic time-step limitation is automatically set based on the Courant criterium. The possible performance penalty that may result from this approach can often be remedied by refining and coarsening the computational grid in the right locations.

The advection scheme in Delft3D-FLOW can be either the flooding scheme for flows that are potentially super critical, or the so called higher order cyclic scheme for all other flows. In D-Flow FM, we use the same advection scheme in both cases. The scheme is 'shockproof', capable of reproducing correct bore propagation velocities, and because of its higher order implementation also applicable in more gentle varying or possibly eddy shedding flows.

In Delft3D-FLOW, a velocity point is either wet or dry. In D-Flow FM, a velocity point may be partly wet. This leads to a more gradual flooding and drying process. Several options are implemented to compute the hydraulic radius associated with a velocity point that has a non uniform water depth. In a 2D model, the most complete option is called the 2D analytic conveyance approach, that shows fast grid convergence for friction dominated flows. This method is an efficient alternative for subgrid modelling.

2.5 Coupling to other modules

The hydrodynamic conditions (velocities, water elevations, density, salinity, vertical eddy viscosity and vertical eddy diffusivity) calculated in the D-Flow FM module are used as input to the other modules of the Delft3D Flexible Mesh Suite, which are:

module	description
D-Waves (SWAN)	short wave propagation, see also chapter 14
D-Water Quality (Delwaq)	far-field water quality, see also chapter 16
D-Real Time Control	flow-triggered control of hydrodynamic structures

Module couplings that are not yet available are summarized below:

module	description
D-Waq PART	mid-field water quality and particle tracking
Delft3D-SED	cohesive and non-cohesive sediment transport

For using D-Flow FM the following utilities are important:

module	description
Delta Shell	for complete model set-up and model runs, see chapter 3 and chapter 4
RGFGRID	for generating curvilinear and unstructured grids
Delft3D-QUICKPLOT	for visualisation and animation of simulation results
OpenEarthTools	set of MATLAB scripts for postprocessing of output files, see http://www.openearth.eu
DFMOUTPUT	for merging partitioned map files into one, see Section 5.2.5 .

For details on using these utility programs you are referred to the respective User Manuals.

2.6 Installation and computer configuration

A separate Installation Manual is provided.

2.7 Examples

An extensive set of example models is available as part of this D-Flow FM release. Throughout this User Manual, references to these testcases are made via the directory names as follows:

- f05_boundary_conditions/c019_waterlevel_bc_cmp_varying/
- f17_sources_sinks/c020_sourcesink_3D/

3 Getting started

3.1 Introduction

The D-Flow FM plugin is part of the Delta Shell framework. For an introduction to the general look-and-feel and functionalities of the Delta Shell framework you are referred to the Delta Shell User Manual. This chapter gives an overview of the basic features of the D-Flow FM plugin and will guide you through the main steps to set up a D-Flow FM model. For a more detailed description of the GUI features you are referred to [chapter 4](#). For technical documentation you are referred to [D-Flow FM Technical Reference Manual \(2015\)](#).

3.2 Overview of D-Flow FM GUI

Delta Shell is only available for Windows operating systems. You can either install the msi-version or copy the zip-version. For the msi-version first follow the steps in the installation programme. Consequently, start the application from *Windows(Start) / All programs / Deltares* or by double-clicking the short-cut on your desktop. For the zip-version you don't have to install anything. First unpack the zip, consequently go to bin and double-click *DeltaShell.Gui.x64.exe* to start the application.

When you start the application for the first time the lay-out will look like [Figure 3.1](#). The basic lay-out consists of the following items:

- ◊ **Project** window - up left
- ◊ **Map and Data window** - down left
- ◊ central (map) window - up centre
- ◊ **Messages** window and **Time navigator** - down centre
- ◊ **Region** and **Chart** window - up right
- ◊ **Properties** and **Undo/redo** window - down right

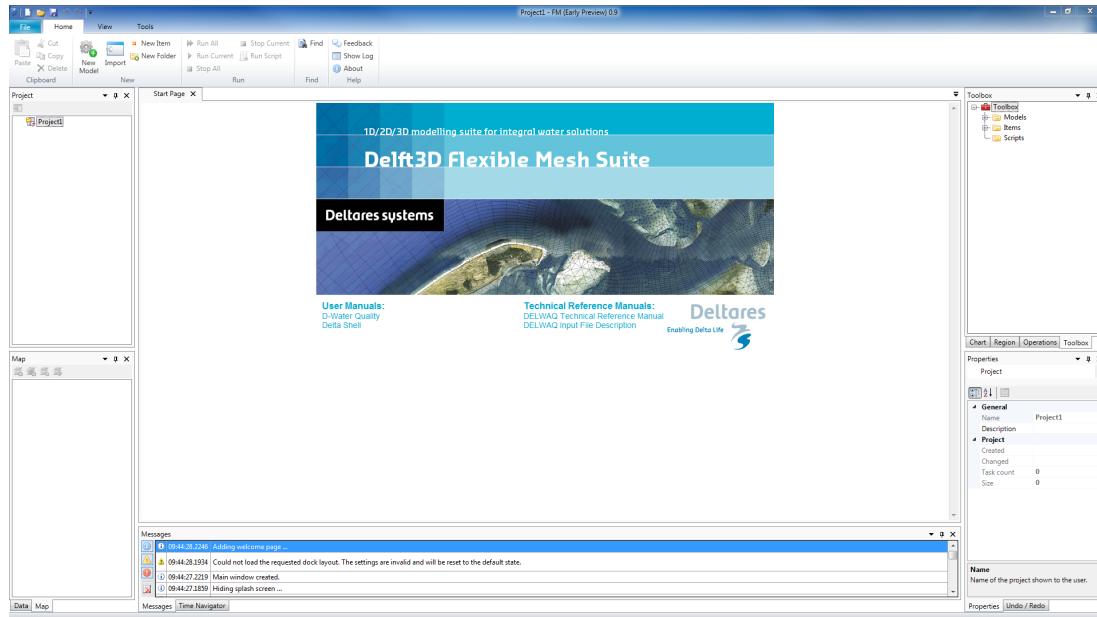


Figure 3.1: Start-up lay-out Delta Shell

All the windows can be customized/hidden according to your own preferences. These settings will be automatically saved for the next time you open the application. The most important windows

for the D-Flow FM plugin are the Project, central (map), Map, Messages and Time navigator windows. The contents of these windows are briefly discussed in the subsections below.

3.2.1 Project window

After adding or importing a D-Flow FM model (see [section 3.5.1](#) and [section 3.5.8](#)), the project tree will be extended with D-Flow FM specific features (see [Figure 3.2](#)). The project tree provides you with the basic steps to set up a D-Flow FM model.

The project tree consists of the following features:

<i>General</i>	general model information such as depth layer specification, model coordinate system and angle of latitude
<i>Area</i>	geographical (GIS based) features, such as observation points, structures, dry points and land boundaries
 <i>Domain</i>	computational grid (Note: still to be implemented, for now under general)
<i>Bathymetry</i>	model bathymetry
<i>Time Frame</i>	model time frame and time step
<i>Processes</i>	active physical processes in the model such as salinity, temperature, wind and tide generating forces
<i>Initial conditions</i>	initial conditions for water levels and other physical processes
<i>Boundary conditions</i>	model boundaries and boundary condition specification
<i>Physical parameters</i>	physical settings for processes such as roughness, viscosity, wind and temperature
<i>Sources and sinks</i>	location and time series specification for point sources and sinks
<i>Numerical parameters</i>	numerical simulation settings
<i>Output parameters</i>	output specification
<i>Output</i>	output after running the simulation

Upon clicking the items in the project tree the corresponding tab (in case of non-geographic model settings), attribute table (in case of geographic model settings) or editor view (in case of advanced editing options) will open. Using the right mouse button (RMB) gives options such as importing/exporting model data.

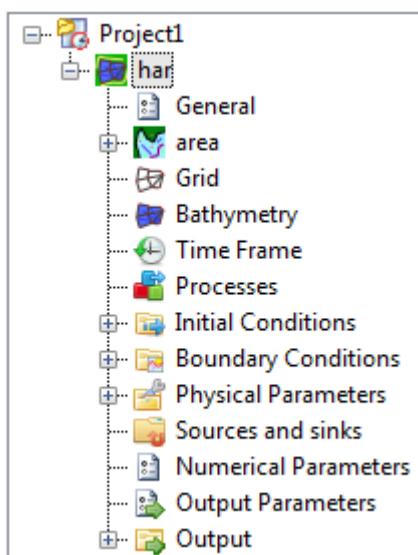


Figure 3.2: Project tree of D-Flow FM plugin

3.2.2 Central (map) window

The central window shows the contents of the main editor you are working with. In most cases this will be the central map with tabulated input fields (see Figure 3.3). The map is used to edit geographic model data, the tabulated input fields to edit overall model settings. Moreover, the contents of the central window can also be a specific editor such as the time point editor or the boundary condition editor. Each of these editors will open as a separate view.



Figure 3.3: Central map with contents of the D-Flow FM plug-in

3.2.3 Map window

The map tree allows the user to control the visibility of the contents of the central map using checkboxes. Furthermore, the user can add (wms) layers, such as satellite imagery or open street maps (see Figure 3.4).

Note: Please note that the map usually has a different coordinate system than the model. In rendering the model attributes they are transformed to the map coordinate system (for visual inspection on the map), but the model will be saved in the model coordinate system.

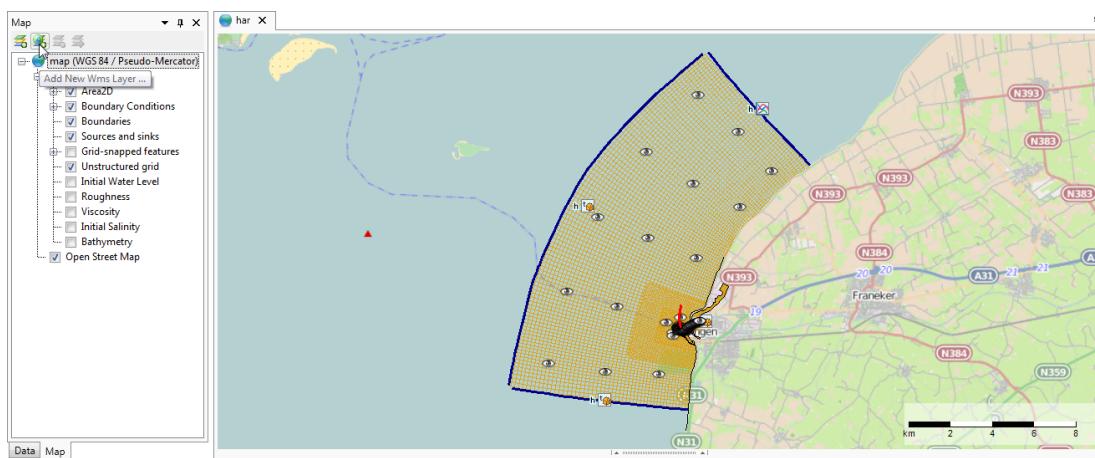


Figure 3.4: Map tree controlling map contents

3.2.4 Messages window

The message window ([Figure 3.5](#)) provides a log of information on the recent activities in Delta Shell. It also provides warning and error messages.

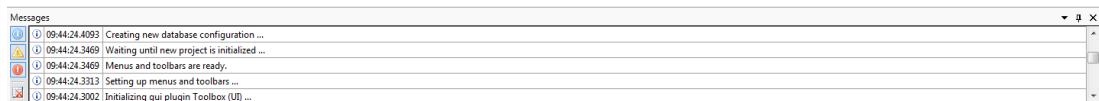


Figure 3.5: Log of messages, warnings and errors in message window

3.2.5 Time navigator window

The time navigator ([Figure 3.6](#)) can be used to step through time dependent model output and other time dependent geographic features on the map.



Figure 3.6: Time navigator in Delta Shell

3.3 Dockable views

The Delta Shell framework offers lots of freedom to customize dockable views, which are discussed in this section.

3.3.1 Docking tabs separately

Within the Delta Shell framework the user can dock the separate windows according to personal preferences. These preferences are then saved for future use of the framework. An example of such preferences is presented in [Figure 3.7](#), where windows have been docked on two screens.

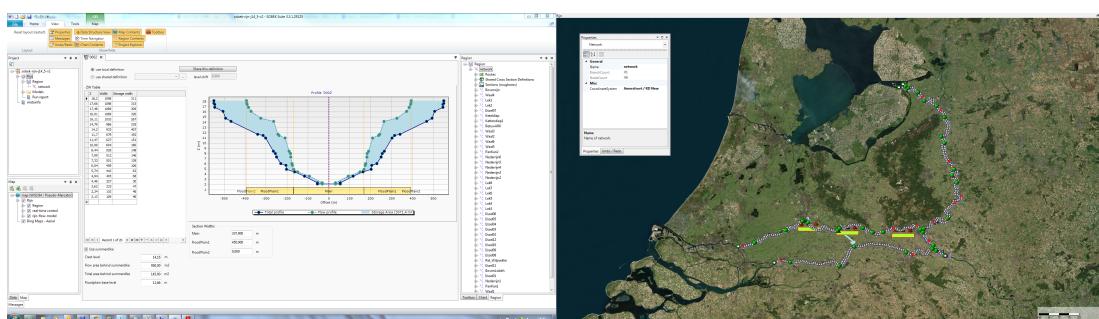


Figure 3.7: Docking windows on two screens within the Delta Shell framework.

3.3.2 Multiple tabs

In case two windows are docked in one view, the underlying window (tab) can be brought to the front by simply selecting the tab, as is shown here.

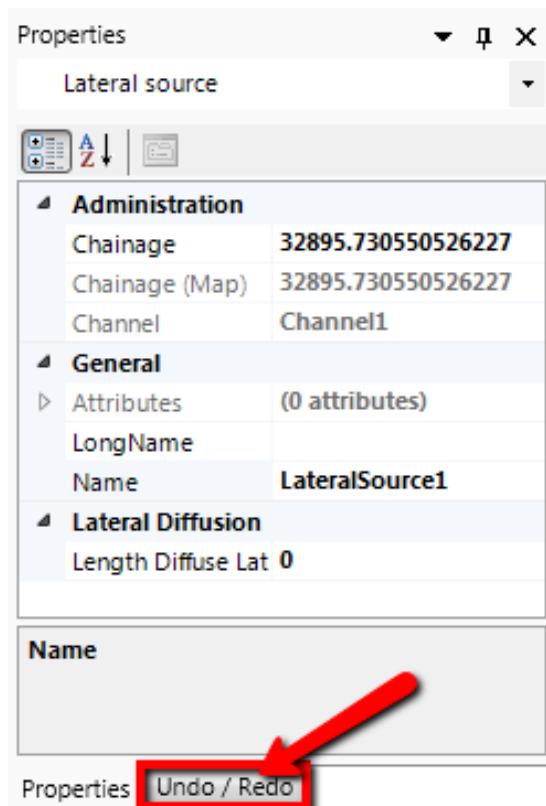


Figure 3.8: Bringing the **Undo/Redo** window to the front

By dragging dockable windows with the left mouse button and dropping the window left, right, above or below another one the graphical user interface can be customized according to personal preferences. Here an example of the **Undo/Redo** window being docked above the **Properties** window.

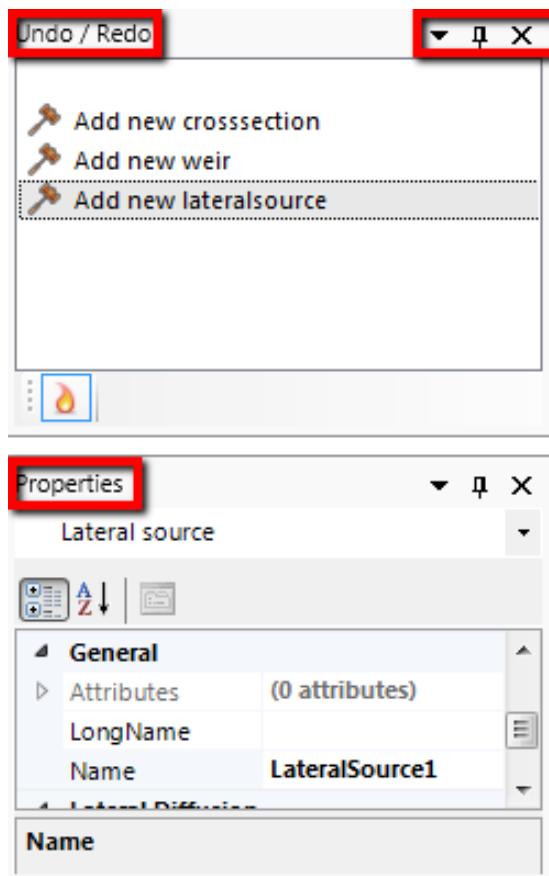


Figure 3.9: Docking the **Undo/Redo** window.

Additional features are the possibility to remove or (auto) hide the window (top right in [Figure 3.9](#)). In case of removal, the window can be retrieved by a mouse-click on *Undo/Redo* in the *View* ribbon. Hiding the **Undo/Redo** window results in:

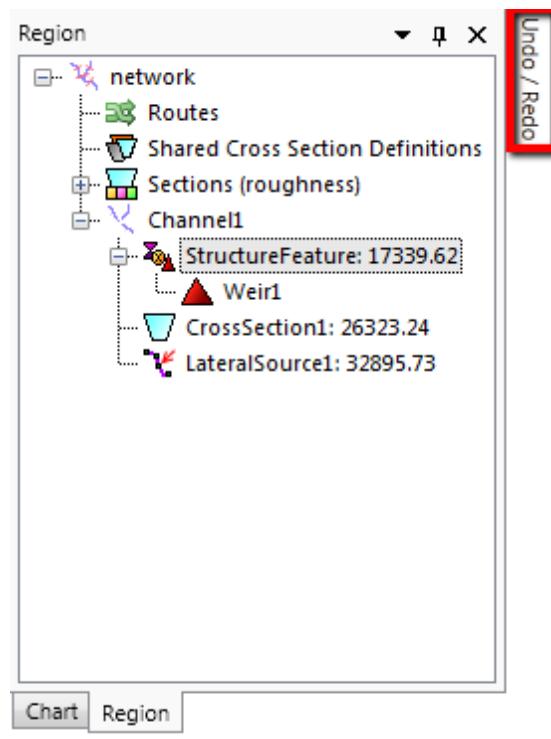


Figure 3.10: Auto hide the **Undo / Redo** window

3.4 Ribbons and toolbars

The user can access the toolbars arranged in *ribbons*. Model plug-ins can have their own model specific *ribbon*. The *ribbon* may be auto collapsed by activating the *Collapse the Ribbon* button when right-mouse-clicking on the *ribbon*.

3.4.1 Ribbons (hot keys)

Delta Shell makes use of ribbons, just like Microsoft Office. You can use these ribbons for most of the operations. With the ribbons comes hot key functionality, providing shortcuts to perform operations. If you press “ALT”, you will see the letters and numbers to access the ribbons and the ribbon contents (i.e. operations). For example, “ALT” + “H” will lead you to the “Home”-ribbon (Figure 3.11).

Note: Implementation of the hot key functionality is still work in progress.

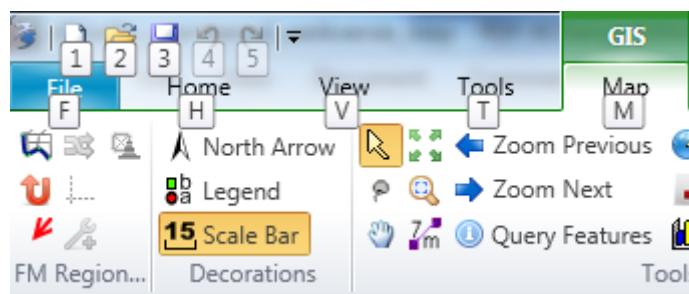


Figure 3.11: Perform operations using the hot keys

3.4.2 File

The left-most *ribbon* is the *File* ribbon. It has menu-items comparable to most Microsoft applications. Furthermore, it offers users import and export functionality, as well as the *Help* and *Options* dialogs, as shown in [Figure 3.12](#) and [Figure 3.13](#).

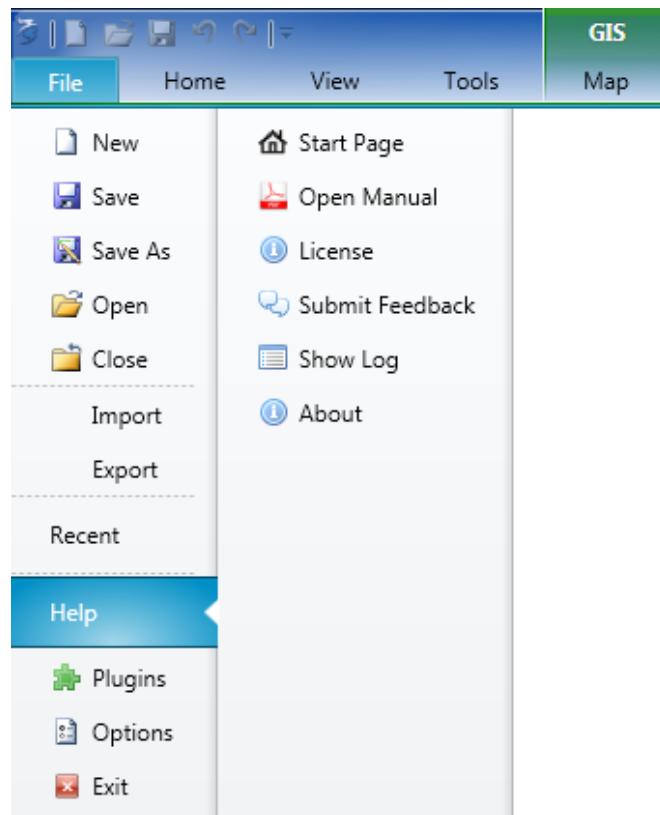


Figure 3.12: The File ribbon.

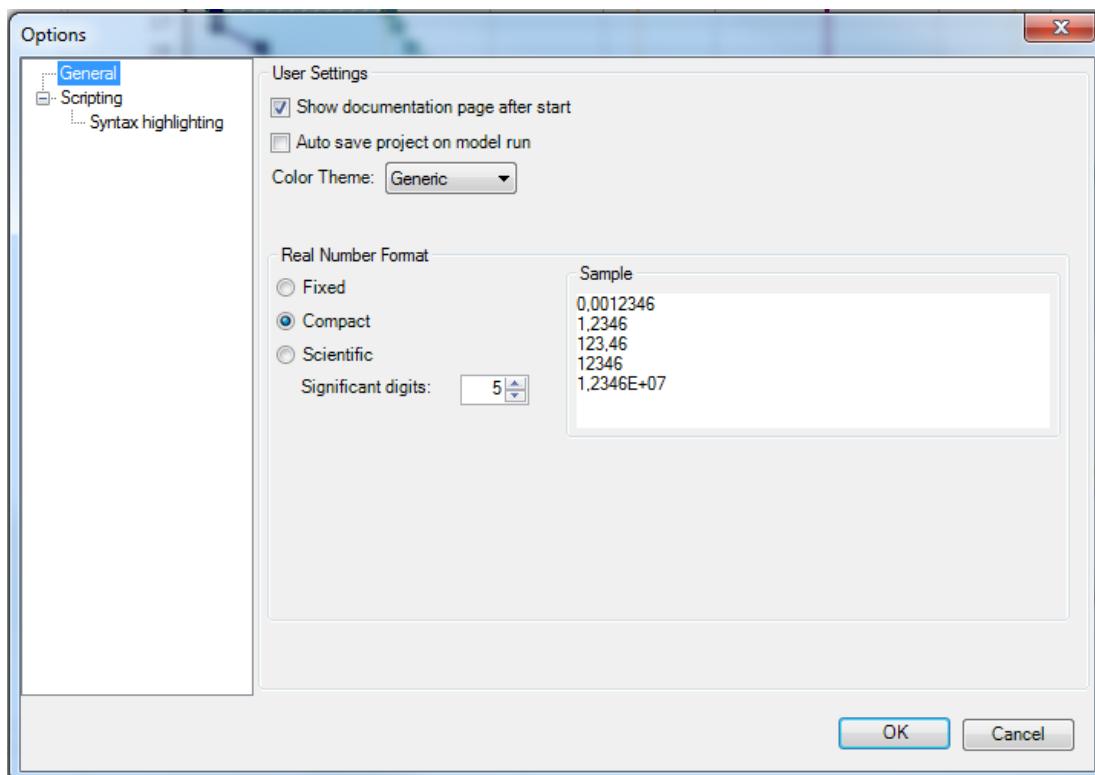


Figure 3.13: The Delta Shell options dialog.

3.4.3 Home

The second *ribbon* is the *Home* ribbon (Figure 3.14). It harbours some general features for clipboard actions, addition of items, running models, finding items within projects or views, and help functionality.

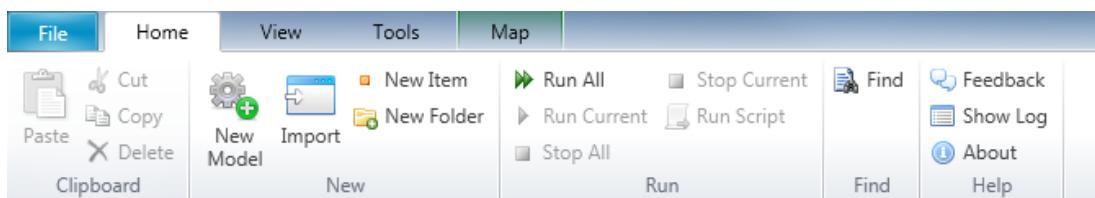


Figure 3.14: The Home ribbon.

3.4.4 View

The third *ribbon* is the *View* ribbon (Figure 3.15). Here, the user can show or hide windows.

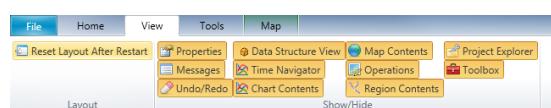


Figure 3.15: The View ribbon.

3.4.5 Tools

The fourth *ribbon* is the *Tools* ribbon (Figure 3.16). By default, it contains only the *Open Case Analysis View* tool. Some model plug-ins offer the installation of extra tools that may be installed. These are documented within the user documentation of those model plug-ins.

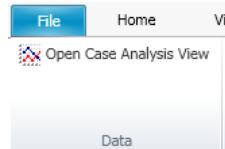


Figure 3.16: The Tools ribbon.

3.4.6 Map

The last *ribbon* is the *Map* ribbon (Figure 3.17).

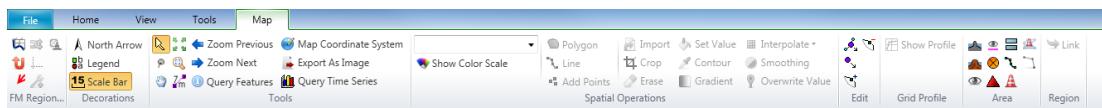


Figure 3.17: The Map ribbon.

This will be used heavily, while it harbours all *Geospatial* functions, like:

- ◊ *Decorations* for the map
 - North arrow
 - Scale bar
 - Legend
 - ...
- ◊ *Tools* to customize the map view
 - Select a single item
 - Select multiple items by drawing a curve
 - Pan
 - Zoom to Extents
 - Zoom by drawing a rectangle
 - Zoom to Measure distance
 - ...
- ◊ *Edit* polygons, for example within a network, basin, or waterbody
 - Move geometry point(s)
 - Add geometry point(s)
 - Remove geometry point(s)
- ◊ Creation of a model *Network*, for example for D-Flow 1D
 - Add new Branch
 - Split Branch
 - Add Cross section
 - Add Weir
 - Add Pump
 - ...



Note: The *ribbons* adjust to the size of the application window. If, for what reason, the user wants to minimize the window, the ribbons might look like as shown in [Figure 3.18](#). Some of the *ribbon* categories have been condensed into a single drop-down panel.

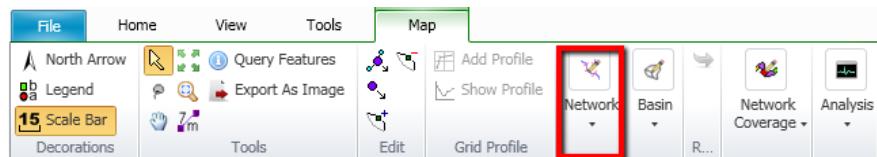


Figure 3.18: The ribbon with minimized categories.

Still, all functions of the category can be activated as they will appear in the drop-down panel.

3.4.7 Scripting

When you open the scripting editor in Delta Shell, a Scripting *ribbon* category will appear. This ribbon has the following additional options (see also [Figure 3.19](#)), which are described in [Table 3.1](#):



Figure 3.19: The scripting ribbon within Delta Shell.

Table 3.1: Functions and their descriptions within the scripting ribbon of Delta Shell.

Function	Description
Run script	Executes the selected text. If no text is selected then it will execute the entire script
Clear cached variables	Clears all variables and loaded libraries from memory
Debugging	Enables/Disables the debug option. When enabled you can add breakpoint to the code (using F9 or clicking in the margin) and the code will stop at this point before executing the statement (use F10 (step over) or F11 (step into) for a more step by step approach)
Python variables	Show or hide python variables (like _var_) in code completion
Insert spaces/tabs	Determines if spaces or tab characters are added when pressing tab
Tab size	Sets the number of spaces that are considered equal to a tab character
Save before run	Saves the changes to the file before running
Create region	Creates a new region surrounding the selected text
Comment selection	Comments out the selected text
Convert to space indenting	Converts all tab characters in the script to spaces. The number of spaces is determined by Tab size
Convert to tab indenting	Converts all x number of space characters (determined by Tab size) in the script to tabs
Python (documentation)	Opens a link to the python website, showing you the python syntax and standard libraries

Table 3.1: Functions and their descriptions within the scripting ribbon of Delta Shell.

Function	Description
Delta Shell (documentation)	Opens a link to the Delta Shell documentation website (generated documentation of the Delta Shell api)

3.4.8 Shortcuts

The shortcut keys of the scripting editor within Delta Shell are documented in Table 3.2.

Table 3.2: Shortcut keys within the scripting editor of Delta Shell.

Shortcut	Function
Ctrl + Enter	Run selection (or entire script with no selection)
Ctrl + Shift + Enter	Run current region (region where the cursor is in)
Ctrl + X	Cut selection
Ctrl + C	Copy selection
Ctrl + V	Paste selection
Ctrl + S	Save script
Ctrl + -	Collapse all regions
Ctrl + +	Expand all regions
Ctrl + "	Comment or Uncomment current selection
Ctrl + W	Add selection as watch variable
Ctrl + H	Highlight current selection in script (press esc to cancel)
F9	Add/remove breakpoint (In debug mode only)
F5	Continue running (In debug mode only — when on breakpoint)
Shift + F5	Stop running (In debug mode only — when on breakpoint)
F10	Step over current line and break on next line (In debug mode only - when on breakpoint)
F11	Step into current line if possible, otherwise go to next line (In debug mode only — when on breakpoint). This is used to debug functions declared in the same script (that have already runned)

3.4.9 Quick access toolbar



Note: The user can make frequently used functions available by a single mouse-click in the *Quick Access Toolbar*, the top-most part of the application-window. Do this by right-mouse-clicking a ribbon item and selecting *Add to Quick Access Toolbar*.

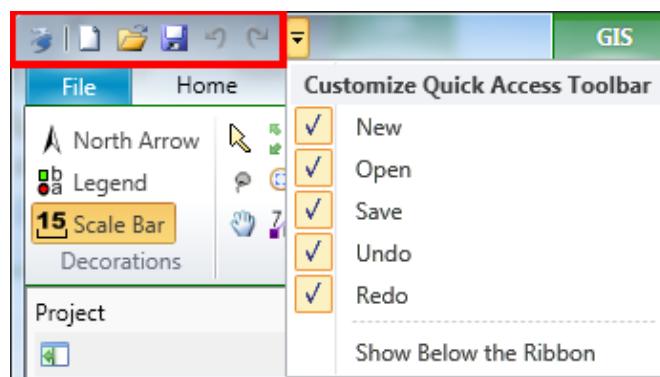


Figure 3.20: The quick access toolbar.

3.5 Basic steps to set up a D-Flow FM model

This section shows the basic steps to set up a D-Flow FM model. For a more detailed description of the steps and GUI features you are referred to Chapter 4.

3.5.1 Add a D-Flow FM model

After starting the application for the first time, the start page will open with a default project (i.e. "project1", see [Figure 3.1](#)). To add a D-Flow FM model to the project you have the following options:

- ◊ click "New Model" in the "Home"-ribbon ([Figure 3.21](#))
- ◊ use the Right Mouse Button (RMB) on "project1" in the project tree, go to "Add" and "New Model" ([Figure 3.22](#))

From the list of available models (which can vary depending on your installation), select "D-Flow FM model" ([Figure 3.23](#)).

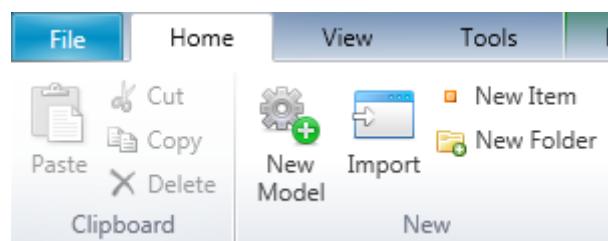


Figure 3.21: Adding a new model from the ribbon

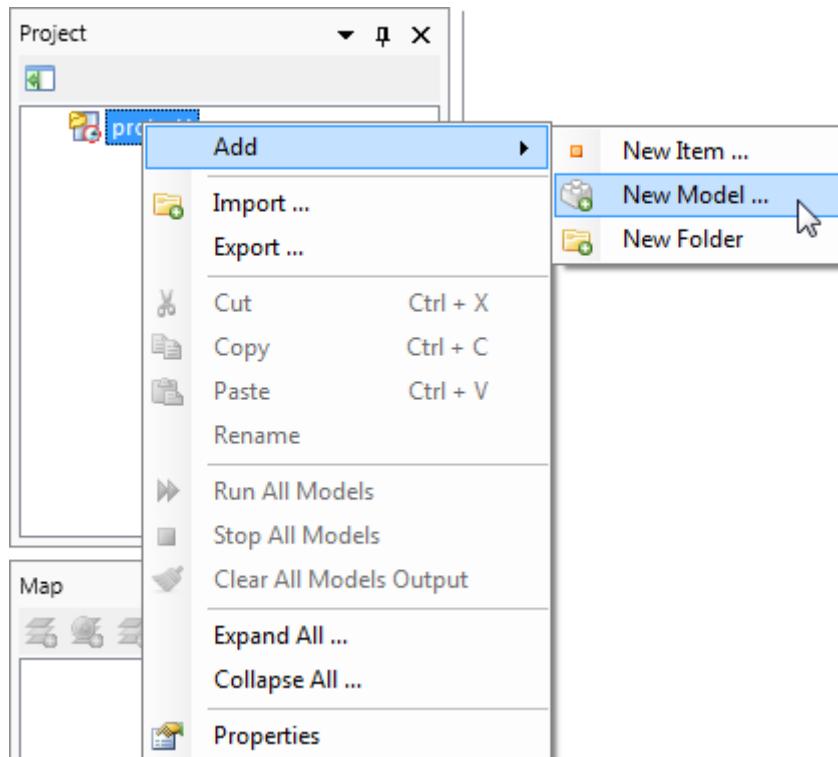


Figure 3.22: Adding a new model using the Right Mouse Button on “project1” in the project tree

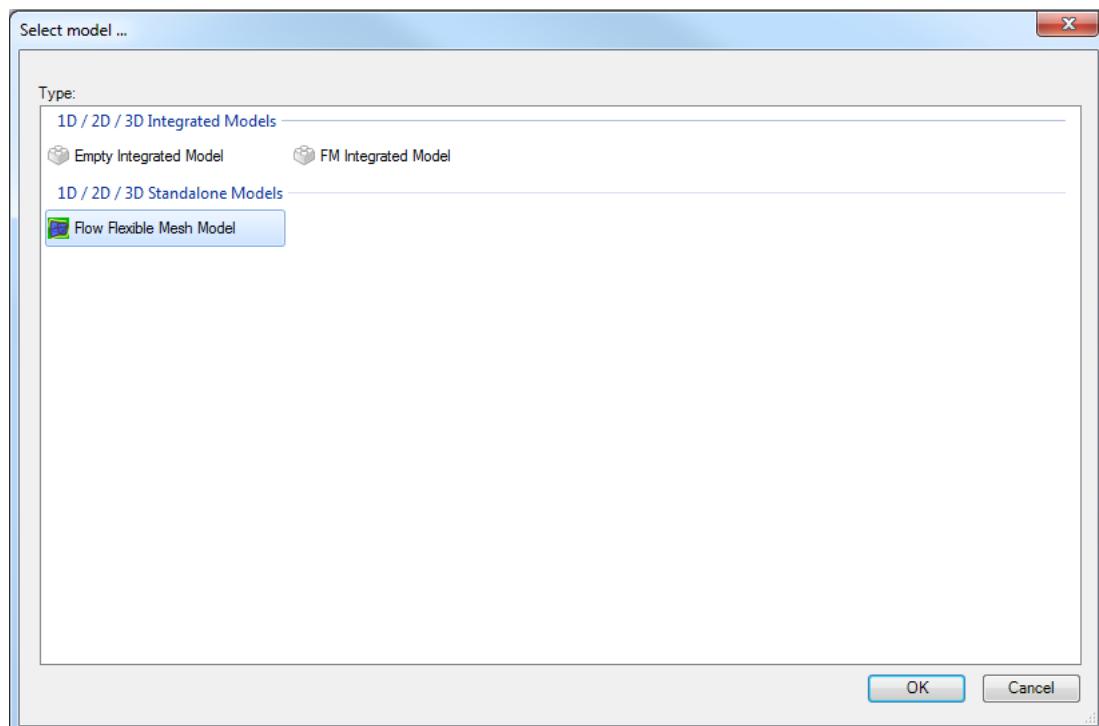


Figure 3.23: Select “D-Flow FM model”

3.5.2 Set up a D-Flow FM model

To set up the D-Flow FM model follow the steps in the project tree. For a more detailed description, see Chapter 4.

3.5.3 Converting a Delft3D-FLOW model into D-Flow FM

Existing Delft3D-FLOW models can be converted into a set of input files suitable for D-Flow FM (to a large extent). The conversion can be done by some Matlab utilities, which are described in [section B.6.1.2](#).

3.5.4 Validate D-Flow FM model

You can check whether your model setup is valid by using the RMB in the project tree and select “Validate” ([Figure 3.24](#)). This will produce a validation report ([Figure 3.25](#)). Red exclamation marks indicate the parts of the model that are still invalid. By clicking the hyperlink you will be automatically redirected to the invalid step in the model setup, so that you can correct it.

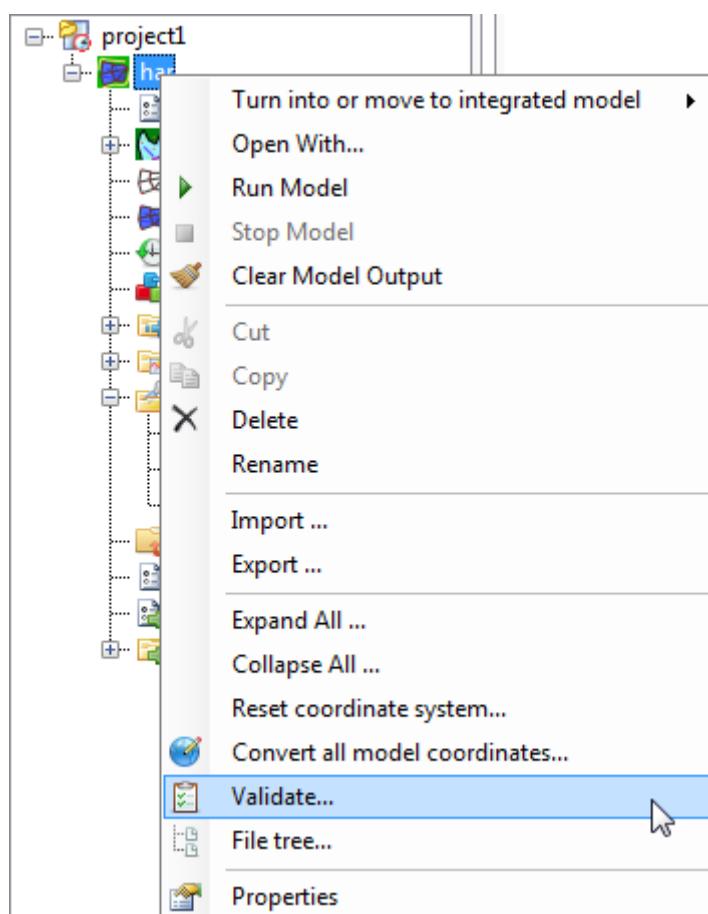
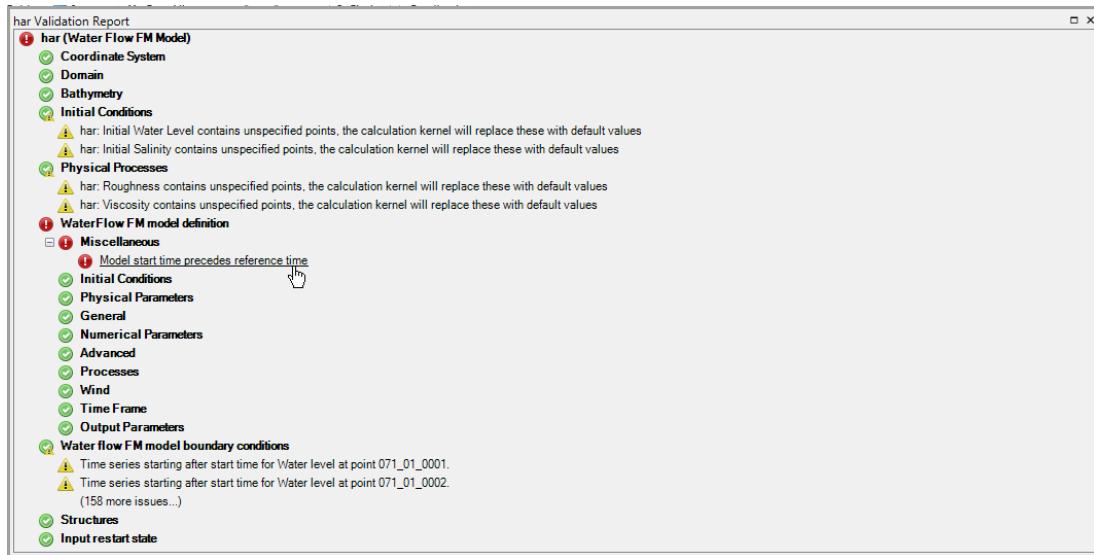


Figure 3.24: Validate model

**Figure 3.25:** Validation report

3.5.5 File tree

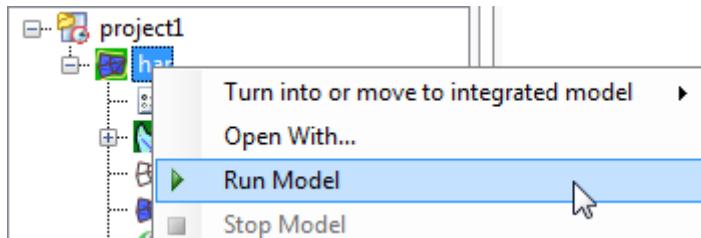
To check the file paths and names of the attribute files which are linked to your model, you can select “File tree” using the RMB on your model in the project tree.

3.5.6 Run D-Flow FM model

If you are satisfied with the model setup, you can run it from Delta Shell using the RMB on model and select “Run model” ([Figure 3.26](#)).



Note: it is also possible to run D-Flow FM outside Delta Shell using the command line.

**Figure 3.26:** Run model

3.5.7 Inspect model output

The simulation will start and the output will be stored in the output folder in the project tree ([Figure 3.27](#)). Delta Shell provides some basic tools to inspect the model output. For more extensive and advanced options you are referred to Quickplot and Muppet.

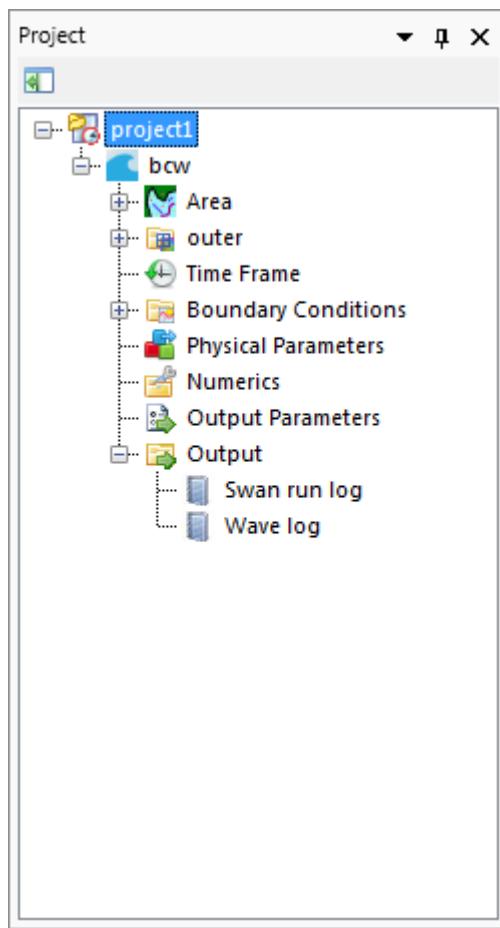


Figure 3.27: Output of wave model in project tree

3.5.8 Import/export or delete a D-Flow FM model

To import an existing D-Flow FM model either use the RMB on the project level in the project tree ([Figure 3.28](#)) or go to the “File”-ribbon and press the “Import” ([Figure 3.29](#)). Likewise you can export a model or delete a model.

For the steps in the project tree that are linked to attribute files (observation points, grid, bathymetry, etc.) you can use the RMB to import or export these attribute files.

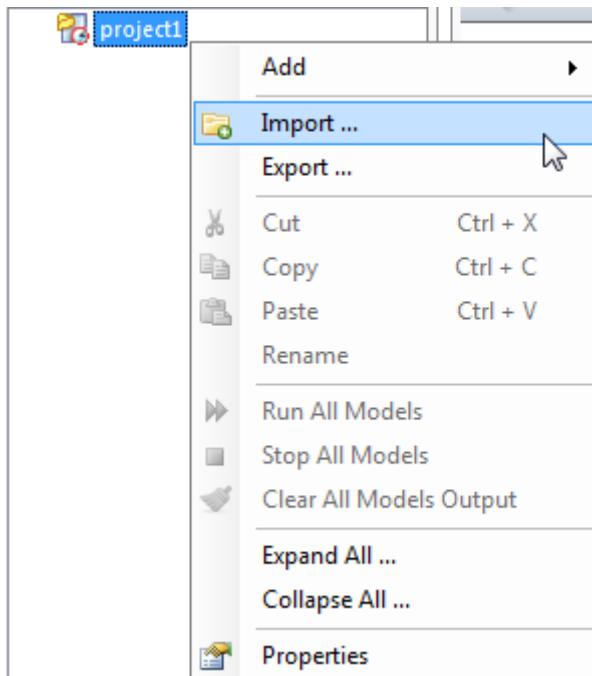


Figure 3.28: Import wave model from project tree

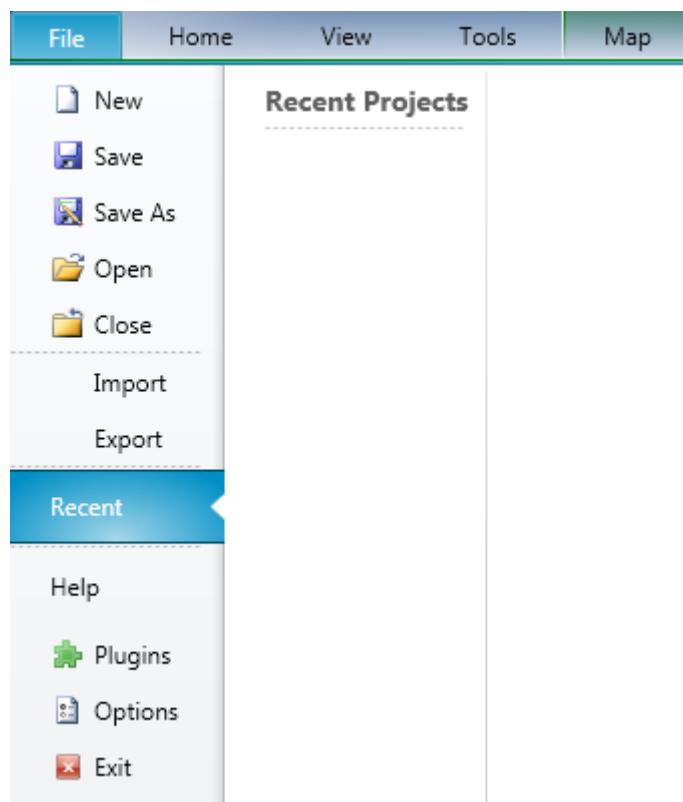


Figure 3.29: Import wave model from file ribbon

3.5.9 Save project

To save the project (and, hence, the model) use the disk-icon on the Quick Access Toolbar or the “File”-ribbon ([Figure 3.29](#)). If you would like to save the project under a different name use “Save as”.

3.5.10 Exit Delta Shell

If you are finished you can exit Delta Shell using the red cross or pressing the “Exit” button in the “File”-ribbon ([Figure 3.29](#)).

3.6 Important differences compared to Delft3D-FLOW GUI

The differences between the former Delft3D-FLOW GUI and the D-Flow FM plugin in Delta Shell in lay-out and functionality are numerous. Here, we address only the most important differences in the workflow.

3.6.1 Project vs model

The entity “project” is new in the Delta Shell GUI. In the hierarchy the entity “project” is on a higher level than the entity “model”. A project can contain multiple models, which can either run standalone or coupled. The user can run all models in the project at once (on project level) or each model separately (on model level). When the user saves the project, the project settings will be saved in a *.dsproj configuration file and the project data in a *.dsproj_data folder. The *.dsproj_data folder contains folders with all input and output files for the models within the project. There is no model intelligence in the *.dsproj configuration file, meaning that the models can also be run outside the GUI from the *.dsproj_data folder.

3.6.2 Load/save vs import/export

The user can load an existing Delta Shell project, make changes in the GUI and, consequently, save all the project data. Loading and saving means working on the original project data, i.e. the changes made by the user overwrite the original project data. Alternatively, use “save as” to keep the original project data and save the changes project data at another location (or with another name).

Import/export functionality can be used to copy data from another location into the project (import) or, vice versa, to copy data from the project to another location (export). Import/export is literally copying, e.g.:

- ◊ import: changes on the imported data will only affect the data in the project and not the source data (upon saving the project)
- ◊ export: the model data is copied to another location “as is”, changes made afterwards will only affect the data in the project not the exported data (upon saving the project)

3.6.3 Working from the map

One of the most important differences with the former GUI is the central map. The central map is comparable with the former “visualization area”, but with much more functionality and flexibility. The map helps you to see what you are doing and inspect the model at all times. You can use the “Region” and “Map” ribbons to add/edit model features in the map.

3.6.4 Coordinate conversion

With the map as a central feature, functionality to convert model and map coordinates is an indispensable feature. In the “General” tab you can set the model coordinate system. In the map tree you can set the map coordinate system using the RMB (Figure 3.30). The coordinate systems are subdivided in geographic and projected systems. Use the quick search bar to find the coordinate system you need either by name or EPSG code (Figure 3.31).

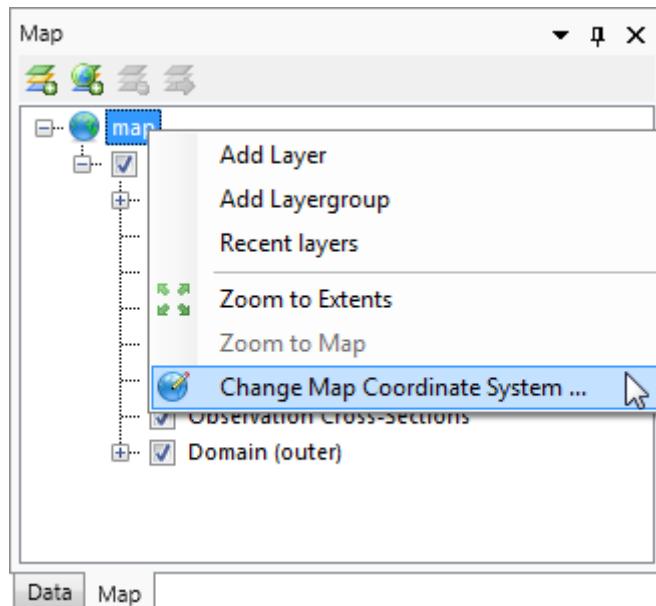


Figure 3.30: Set map coordinate system using RMB

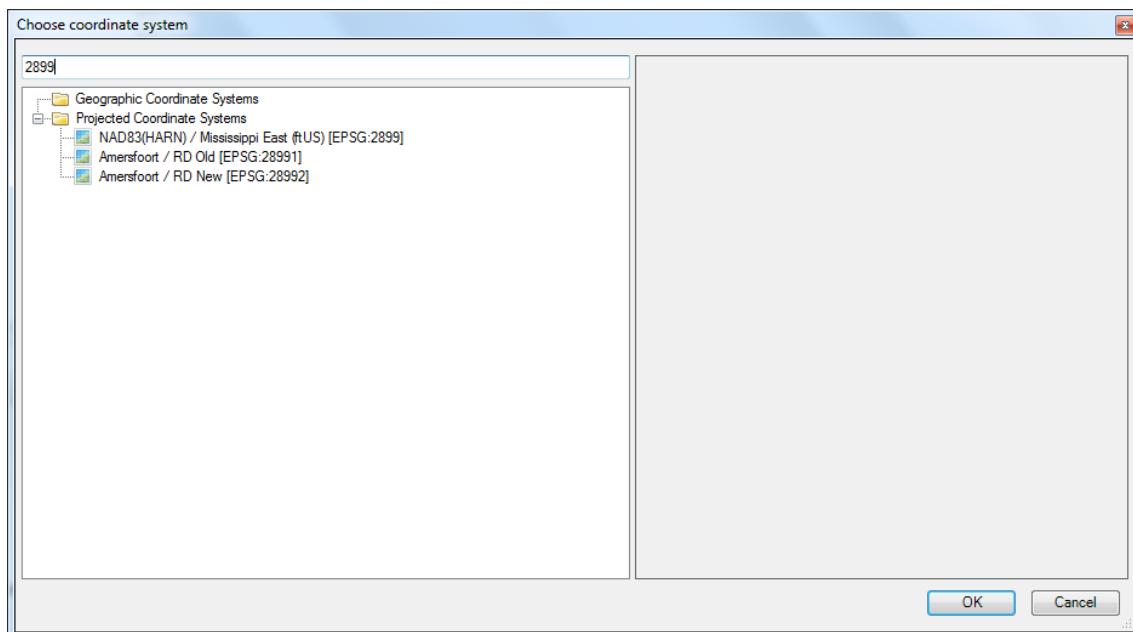


Figure 3.31: Select a coordinate system using the quick search bar

3.6.5 Model area

The model area contains geographical features, such as observation points & curves and obstacles. In contrast to the former GUI, these features can even exist without a grid or outside

the grid and they are not based on grid coordinates, implying that their location remains the same when the grid is changed (for example by (de-)refining).

Finally, for the computations, the SWAN computational core interpolates the features to the grid. In the future we would like to show to which grid points the features are snapped before running the computation. However, this requires some updates in the SWAN computational core.

3.6.6 Integrated models (model couplings)

The Delta Shell framework knows the concept of an *Integrated model* in order to couple different models, such as: hydrodynamics coupled with controlling of structures, waves, morphology and/or water quality.

Two types of coupling are distinguished: *offline* and *online* coupling. An *offline* coupling runs the entire hydrodynamic simulation first, i.e., *separately* from the second simulation. The file-based hydrodynamic output serves as input for the second simulation. As such, the hydrodynamic flow drives the controlling of structures, or the simulation of waves or water quality. In this case there is no feedback on the hydrodynamic simulation. For many applications, this is good practice.

An *online* coupling, on the other hand, exchanges data *every time step*. This tight coupling allows for direct feedback of the various processes on one another. This is crucial for controlling structures.

Note: *Offline* is also referred to as *Sequential* coupling and *online* as *Parallel* coupling.



3.6.7 Ribbons (hot keys)

Delta Shell makes use of ribbons, just like Microsoft Office. You can use these ribbons for most of the operations. With the ribbons comes hot key functionality, providing shortcuts to perform operations. If you press “ALT”, you will see the letters and numbers to access the ribbons and the ribbon contents (i.e. operations). For example, “ALT” + “H” will lead you to the “Home”-ribbon (Figure 3.32).

Note: Implementation of the hot key functionality is still work in progress.

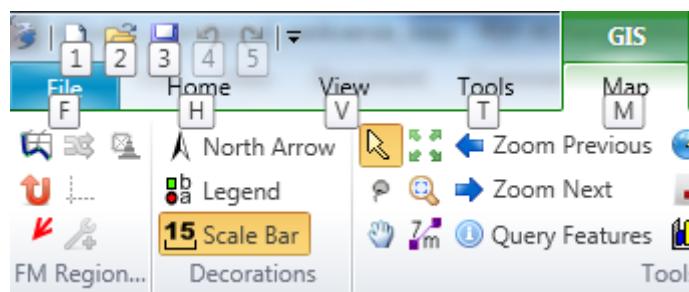


Figure 3.32: Perform operations using the hot keys

3.6.8 Context menus

Context menus are the menus that pop up using the right mouse button. These context menus provide you with some handy functionality and shortcuts specific for the selected item. The functionality is available in all Delta Shell windows and context dependent. You can best try it yourself to explore the possibilities.

3.6.9 Scripting

Delta Shell has a direct link with scripting in Iron Python (NB: this is not the same as C-Python). This means that you can get and set data, views and model files by means of scripting instead of having to do all manually. Scripting can be a very powerful tool to automate certain steps of your model setup or to add new functionality to the GUI. You can add a new script by adding a new item, either in the "Home"-ribbon or through the RMB.

4 All about the modelling process

4.1 Introduction

In order to set up a hydrodynamic model you must prepare an input file. All parameters to be used originate from the physical phenomena being modelled. Also from the numerical techniques being used to solve the equations that describe these phenomena, and finally, from decisions being made to control the simulation and to store its results. Within the range of realistic values, it is likely that the solution is sensitive to the selected parameter values, so a concise description of all parameters is required. This input data is collected into the Master Definition Unstructured file or MDU-file.

In [Section 4.2](#) we discuss some general aspects of the MDU-file and its attribute files. The sections thereafter describe how the actual modelling process can be done in the GUI.

4.2 MDU-file and attribute files

The Master Definition Unstructured file (MDU-file) is the input file for the hydrodynamic simulation program. It contains all the necessary data required for defining a model and running the simulation program. In the MDU-file you can define attribute files in which relevant data (for some parameters) is stored. This will be particularly the case when parameters contain a large number of data (e.g., time-dependent or space varying data). The MDU-file and all possible user-definable attribute files are listed and described in [Appendix A](#).

Although you are not supposed to work directly on the MDU-file it is useful to have some ideas on it as it reflects the idea of the designer on how to handle large amounts of input data and it might help you to gain a better idea on how to work with this file.

The basic characteristics of an MDU-file are:

- ◊ It is an ASCII file.
- ◊ Each line contains a maximum of 256 characters.
- ◊ Each (set of) input parameter(s) is preceded by a (set of) *keyword(s)*.

The results of all modules are written to platform independent binary (NetCDF-)files, so also these result files you can transfer across hardware platforms without any conversion.

The MDU-file contains several sections, denoted by square brackets, below are the most relevant ones:

- ◊ [model] – this section contains the program name and its version.
- ◊ [geometry] – in this section, the main entry comprises the specification of the grid (i.e. the netcdf network file). In addition, thin dams and thin dykes can be specified.
- ◊ [numerics] – this section contains the settings of specific parts of the flow solver, such as limiters and the iterative solver type.
- ◊ [physics] – in this field, physical model parameters can be inserted, for instance related to friction modeling and turbulence modeling.
- ◊ [wind] – the wind section prescribed the dependency of the wind drag coefficient to the wind velocity through 2 or 3 breakpoints. This field also contains pressure information.
- ◊ [time] – in this section, the start time and the stop time of the simulation are specified in hours, minutes or seconds. The other times specified are specified in seconds.
- ◊ [restart] – in this section, the restart file can be specified, either as a `_map.nc`-file or as an `_rst.nc` file.
- ◊ [external forcing] – this section only contains the name of the external forcings file.

- ◊ [output] – in this section, the writing frequency of output data can be prescribed.

[Appendix A](#) contains the full list of MDU sections and keywords.

4.3 Filenames and conventions

Filenames and file extensions hardly have any strict requirements in D-Flow FM, but we do advise to use the suggested file naming patterns below:

file pattern	description
mdident.mdu	MDU-file
*_net.nc	Unstructured grid (network) file
*.xyz	Sample file (for spatial fields)
*.ldb	Landboundary file (polyline file format)
*_thd.pli	Thin dam file (polyline file format)
*_fxw.pliz	Fixed weir file (polyline file format with <i>z</i> values)
*_part.pol	Partitioning polygon file (polyline file format)
*.ext	External forcings file
pliname.pli	Boundary condition location file (polyline file format)
pliname_000X.tim	Timeseries boundary data file at point #X
pliname_000X.cmp	Astronomic/harmonical component boundary data file at point #X
*.bc	BC-format boundary data file with polyline and point labels in file
*.xyn	Observation station file
*_crs.pli	Observation cross-sections file
mdident_map.nc	Output map file
mdident_his.nc	Output his file
mdident.dia	Output diagnostics (log) file

4.4 Setting up a D-Flow FM model

This chapter describes how to set up a D-Flow FM model in an empty Delta Shell project. When you open the GUI, an empty project is automatically created. Starting from scratch, you have to create an empty D-Flow FM model in a Delta Shell project:

- ◊ In the ribbon menu items, go to “Home” and click on “New Model”. The Select model wizard appears ([Figure 4.79](#)). Click on D-Flow FM model and click “OK”. Alternatively, you can also double click on D-Flow FM model to open it directly.
- ◊ Or use the RMB on the name of your project (project1 by default). In the context menu that appears, select “Add” and click “New Model”. Again, the Select model wizard appears allowing you to add an empty D-Flow FM model.

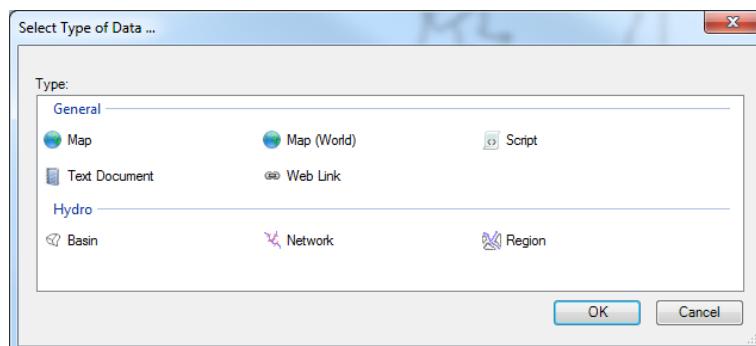


Figure 4.1: Select model wizard

In the project tree, an empty model has appeared (Water Flow FM Model (1) by default). Click the plus sign before the name of the model to expand all model attributes in the project tree; Area2D, Domain, Processes, Time Frame, Bathymetry Definitions, Initial Conditions, Boundary Conditions, Physical Parameters, Numerical Parameters, Output Parameters and Output. In the following paragraphs, all model attributes are treated separately.

4.4.1 General

When you double click on “General” in the project tree, tabulated input fields appear underneath the central map ([Figure 4.2](#)). The general tab contains general model information such as the model grid (will be moved to domain in later release), the type and number of model depth layers, the model coordinate system and the angle of latitude.

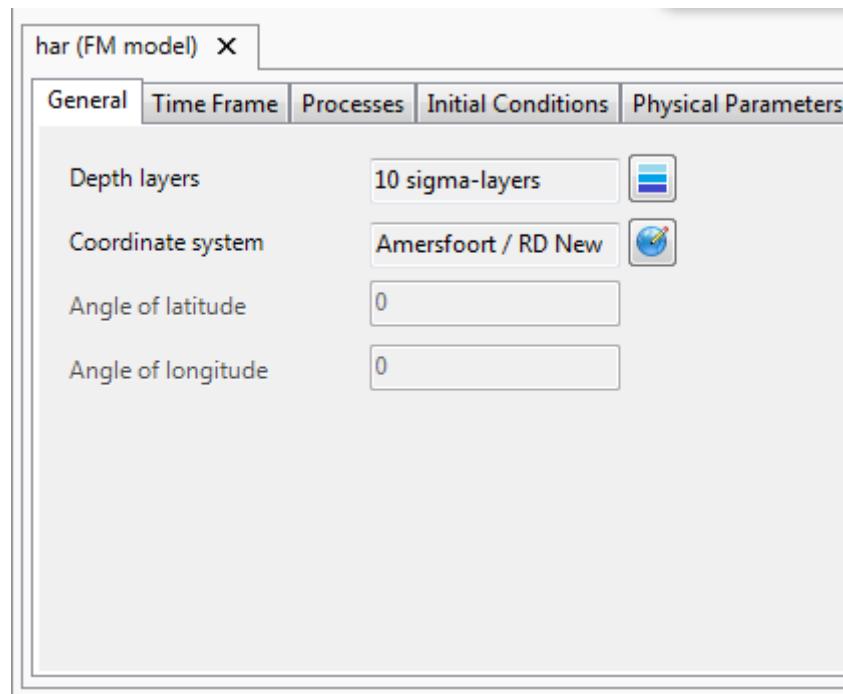


Figure 4.2: Overview of general tab

4.4.1.1 Depth layer specification

When you click on the blue striped icon next to the depth layers text box, the edit depth layers window appears ([Figure 4.3](#)). This window allows you to choose the type of layering and the corresponding number and distribution of vertical layers. The drop down menu contains three distinct layering types:

- ◊ Single
- ◊ Z
- ◊ Sigma

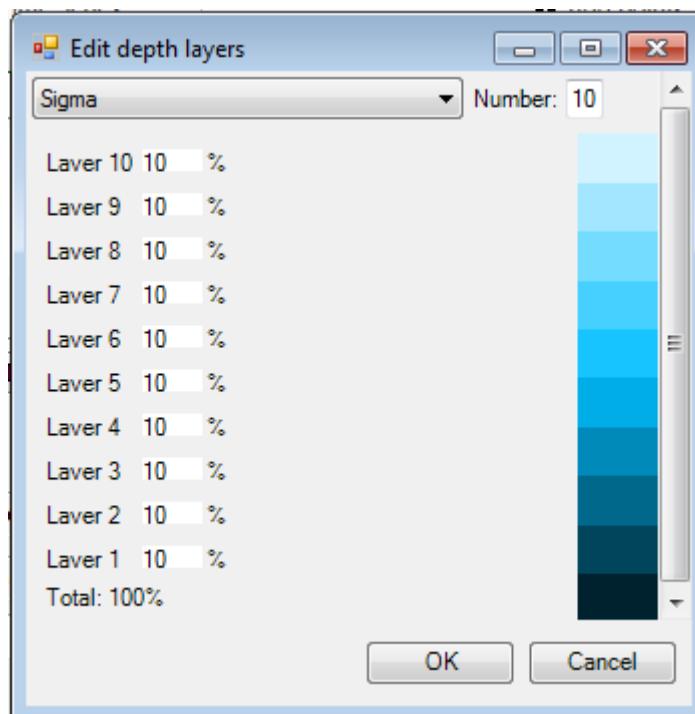


Figure 4.3: Depth layers specification window

The recommended type of layering differs depending on the model application and the processes that you are interested in. Sigma layers increase or decrease in height as the water depth in the model increases or decreases. The relative height distribution of the different layers however remains fixed. Z layers have a fixed height, which does not change as the water depth in the model varies. If the water depth drops below the cumulative height of all z-layers, layer(s) will fall dry. When Single is chosen, the model contains only 1 vertical sigma layer. (An extensive description of sigma and z type layering is found in [section 7.3](#).) (Note: Currently, only the Single and Sigma type layering are supported by the computational core).



If your model contains more than 1 layer, you have to specify the vertical layer height distribution. The layer height specification is dependent on the type of layering that you choose. In case of sigma layering, each layer is specified as a percentage of the total water depth. All percentages have to add up to 100%. In case of Z layers, absolute values in meters have to be prescribed for each layer (Note: Currently, only equidistant spacing (sigma layers) is supported by the computational core).



4.4.1.2 Model coordinate system

A very important property of your model is the coordinate system in which it is specified. Within the interface, there is a clear distinction between the coordinate system of your model and all of its attributes and the coordinate system of the central map and all of its items. Both coordinate systems can be set independent from each other. Keep in mind, that the coordinate system of your model is saved and used when you run your model.

The model coordinate system can be set using the globe icon next to the Coordinate system text box. After clicking this button, the coordinate system wizard is opened ([Figure 4.4](#)). This wizard allows you to choose one of many possible coordinate systems and apply it to your model. You can use the search bar to browse the various coordinate systems (searching

possible by name and EPSG code). If your model is specified in a certain coordinate system already, it is possible to convert the model coordinate system using the same button. After clicking “OK”, All model attributes are converted to the system of choice. Note that you have to close and re-open all map views for the changes to take effect in these views.

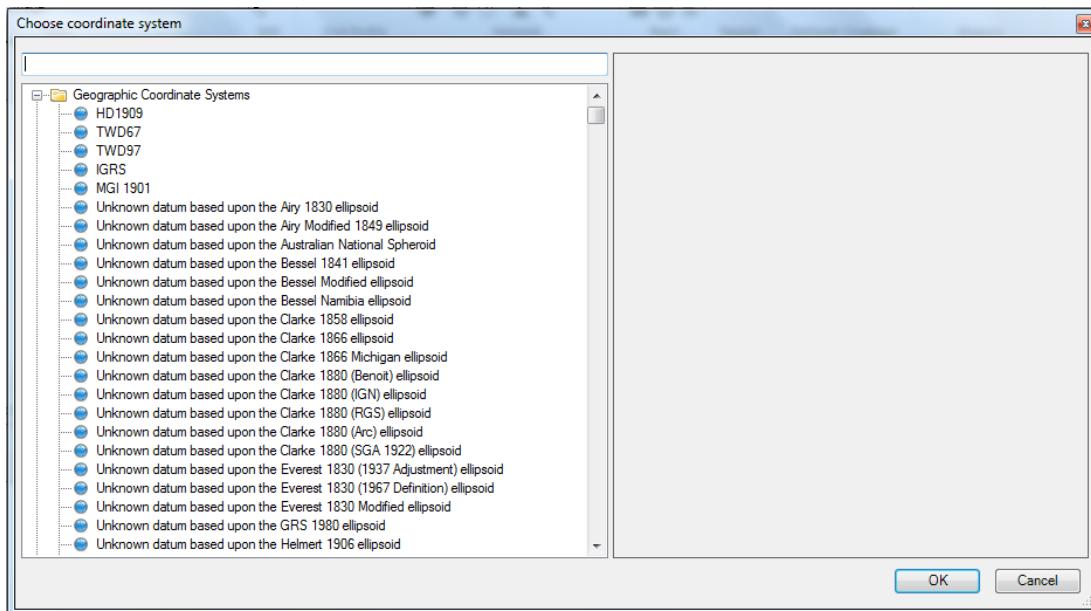


Figure 4.4: Coordinate system wizard

The map coordinate system applies to all items in the map project tree. To change the map coordinate system, navigate the menu ribbons to “Map” and click on “Map coordinate system”. Alternatively, right mouse click on “map” in the map project tree and select “Change Map Coordinate System”. The coordinate system wizard appears, allowing you to set the map coordinate system of your choice. When the original model coordinate system differs from the selected map coordinate system, all map items are automatically converted to the specified map coordinate system.

4.4.1.3 Angle of latitude

For a Cartesian grid you have to specify the latitude of the model area; this is used to calculate a fixed Coriolis force for the entire area. For a spherical grid the Coriolis force is calculated from the latitude coordinates in the grid file and thus varies in the latitude direction. Typically, you use spherical co-ordinates for large areas, such as a regional model. When a value of 0 is entered, the Coriolis force is not taken into account.

4.4.2 Area 2D

The model area contains all geographical features, such as the observation points, structures and land boundaries. These features can exist without a grid or outside the grid as they are not based on grid coordinates but xy-coordinates. This means that the location of the model area features remains the same when the grid is changed (for example by (de-)refining). When you expand the model attribute “Area 2D” in the project tree, a list of possible geographical features is displayed (Figure 4.5).

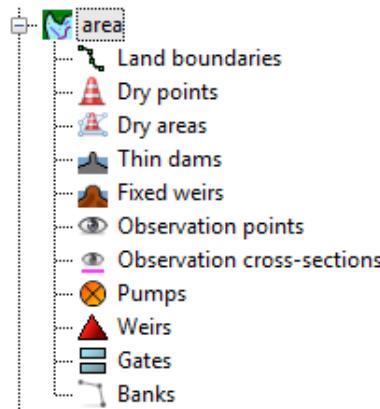


Figure 4.5: Overview of geographical features

Figure 4.6 displays an overview of the map ribbon. The left red box indicates the “FM Region 2D / 3D” of the map ribbon. To add a geographical feature, click the corresponding item in this box and use your mouse to indicate the location of the desired feature on the central map. Importing and exporting of all model features is done via the context menu by using your right mouse button on the different features. The red box on the right highlights the various editing buttons available to edit the locations of the geographical features. The specifics for each feature are discussed separately in the following sections.

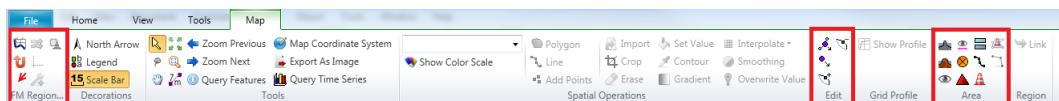


Figure 4.6: Overview of map ribbon. Left red box highlights “FM Region 2D / 3D” menu containing icons used to add features. Right red box highlights “Edit” menu which contains icons used to edit geographical features (move/add)

4.4.2.1 Grid snapped features

All geographical features of your model that are described by x-, and y-coordinates have to be interpolated to your computational grid when you run your model. The computational core of D-Flow FM automatically assigns these features to the corresponding parts of your grid. The graphical user interface allows you to inspect the interpolated locations of these features.

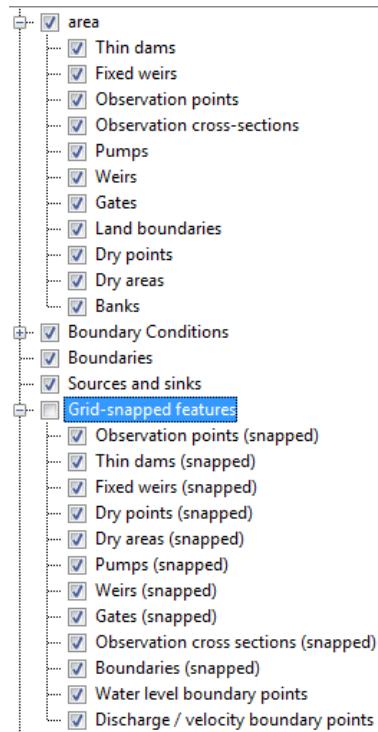


Figure 4.7: Example of expanded grid snapped features attribute in map tree

Figure 4.7 shows a part of the map tree, showing the Area2D and Grid-snapped features attributes. The x-, and y-locations of all spatial model features are shown within the Area2D attribute. You can hide or show any of these attributes by means of clicking the check boxes in front of the attributes. When you enable the Grid-snapped features, all items within the Area2D attribute, as well as all boundaries are interpolated to their corresponding locations on the computational grid. The interpolation is performed instantaneously by the computational core of D-Flow FM, which enables you to directly inspect the numerical interpretation of all features on the computational grid. Figure Figure 4.8 shows an example of four observation points and one thin dam in the central map, showing both the x-, and y-locations of these features as well as their representation on the computational grid.

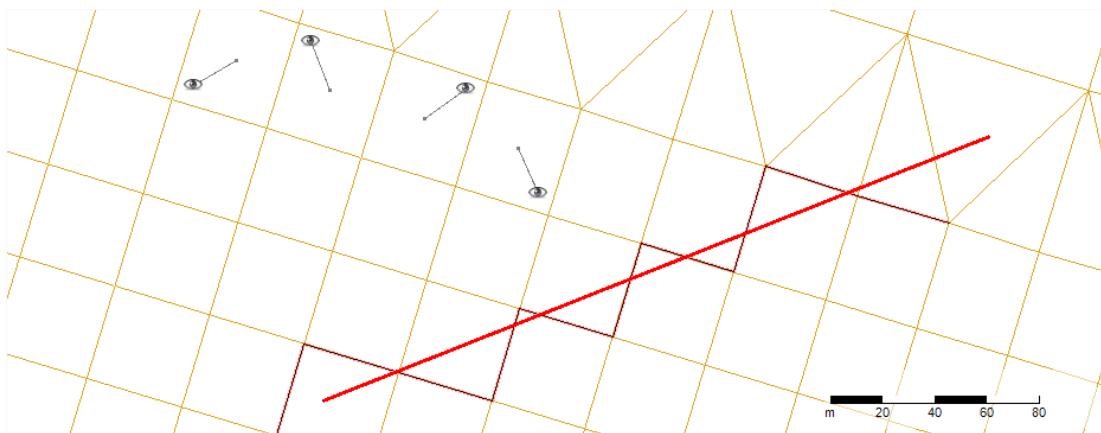


Figure 4.8: Example of grid snapped features displayed on the central map

4.4.2.2 Observation points

Observation points are used to monitor the time-dependent behaviour of one or all computed quantities as a function of time at a specific location, i.e. water elevations, velocities, fluxes, salinity, temperature and concentration of the constituents (**Note: Currently, sediment transport not supported by the computational core**). Observation points represent an Eulerian viewpoint at the results.



To add an observation point, click the corresponding icon from the “FM Region 2D / 3D” menu in the map ribbon (Figure 4.6). By clicking in the central map, observation points are placed in your model. Selected observation points (first click the “Select” icon from the “Tools” menu in the map ribbon) can be deleted using backspace or directly from the attribute table (explained below). The grid snapped representation (Figure 4.9) is indicated by a line linking the observation points to the closest cell center, indicating that output will be stored of this cell. Importing and exporting of observation points is possible via the context menu of “Observation points” in the project tree (right mouse button).

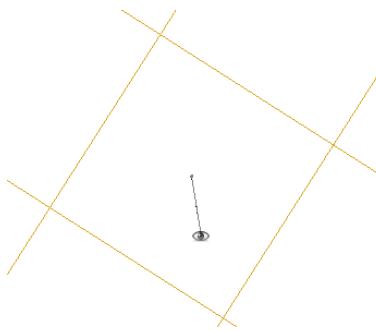


Figure 4.9: Geographical and grid snapped representation of an observation point

When you double click the “Observation points” attribute in the project tree, the observation points tab is displayed underneath the central map (Figure 4.10). This tab shows an attribute table with the names, x-, and y-locations (in the model coordinate system) of the various observation points within the model. When one of the entries is selected, the corresponding observation point is highlighted in the central map.

Name	X	Y
1	1,4993E+05	5,7492E+05
3	1,5227E+05	5,8183E+05
4	1,5458E+05	5,8489E+05
5	1,5702E+05	5,8758E+05
6	1,5264E+05	5,745E+05
7	1,532E+05	5,7758E+05
Inlet_1	1,5466E+05	5,8084E+05
Inlet_2	1,5517E+05	5,7438E+05
Inlet_3	1,5551E+05	5,7687E+05

Figure 4.10: Attribute table with observation points

4.4.2.3 Observation cross-sections

Cross-sections (Figure 4.11) are used to store the sum of computed fluxes (hydrodynamic), flux rates (hydrodynamic), fluxes of matter (if existing) and transport rates of matter (if existing) sequentially in time at a prescribed interval.

To add a cross section, click the corresponding icon from the “FM Region 2D / 3D” menu in the map ribbon (Figure 4.6). By clicking in the central map, cross section points are added. Note that a cross section consists of a minimum of two points, but an arbitrary amount of intermediate points can be added. The last point is indicated by double clicking the left mouse button. The distance in meters (independent of the local coordinate system) in between the last point and the mouse pointer is indicated in pink; in case more than two points are used, the cumulative length of the entire cross section is shown in black. Once highlighted in the central map, a cross section is deleted with backspace or directly from the attribute table described below. The positive direction through the cross section is indicated by a pink arrow. The direction of this arrow is dependent on the order in which the cross section points are drawn (to change the direction, flip the start and end points). Importing and exporting of cross sections is possible via the context menu of “Observation cross-sections” in the project tree (right mouse button).

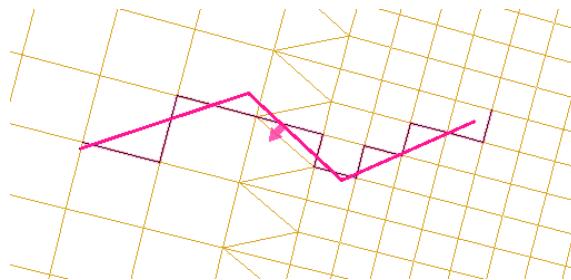


Figure 4.11: Geographical and grid snapped representation of a cross section

Double clicking the “Observation cross-sections” attribute in the project tree enables the Observation cross-sections tab in the central map view (Figure 4.12). Alternatively, you can double click on any cross section in the map. The attribute table displayed in the tab contains the names of the various cross sections of your model. When one of the entries is selected, the corresponding cross section is highlighted in the central map. A cross section entry can be deleted from the table via the context menu (right mouse button).

Name
► (83,97)..(97,97)
(83,77)..(96,77)
sluice
Inlet_1
Inlet_2
Inlet_3
ObservationC...
ObservationC...
ObservationC...

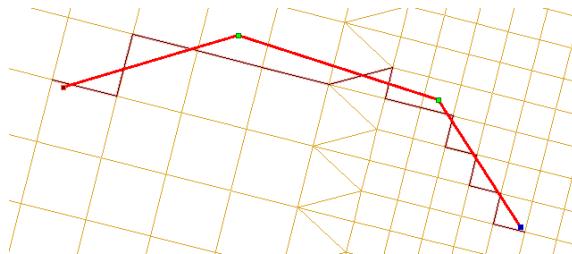
Record 1 of 9 [Navigation Buttons]

Figure 4.12: Attribute table with observation cross sections

4.4.2.4 Thin dams

Thin dams ([Figure 4.13](#)) are infinitely thin objects defined at the velocity points which prohibit flow exchange between the two adjacent computational cells without reducing the total wet surface and the volume of the model. The purpose of a thin dam is to represent small obstacles (e.g. breakwaters, dams) in the model which have sub-grid dimensions, but large enough to influence the local flow pattern. A thin dam is assumed to have an infinite height in the model; no water will ever overflow a thin dam.

To add a thin dam, click the corresponding icon from the “FM Region 2D / 3D” menu in the map ribbon ([Figure 4.6](#)). Adding, deleting, importing and exporting of a line feature such as a thin dam is discussed in more detail in [Section 4.4.2.3](#) on cross sections.

**Figure 4.13:** Geographical and grid snapped representation of a thin dam

When you double click the “Thin dams” attribute in the project tree, the corresponding Thin dams tab appears underneath the central map ([Figure 4.14](#)). Alternatively, you can also double click on any thin dam in the central map. Within this tab, an attribute table is shown which displays the names of all thin dams within your model. When one of the entries is selected, the corresponding item is highlighted in the central map. A thin dam entry can be deleted from the table via the context menu (right mouse button).

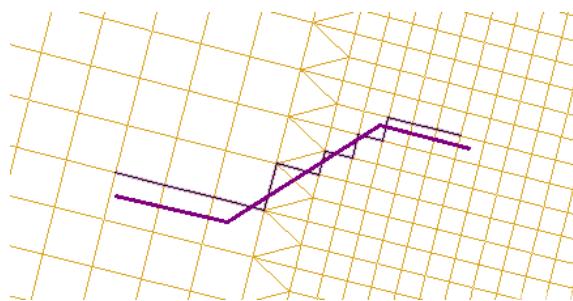
Thin dams X	
	Name
	ThinDam01
	ThinDam02
	ThinDam03
	ThinDam04
►	ThinDam05

Record 5 of 5

Figure 4.14: Attribute table with thin dams

4.4.2.5 Fixed weirs

A fixed weir (Figure 4.15) has the same function as a thin dam (Section 4.4.2.4). However, unlike a thin dam, a fixed weir can be assigned both xy- and z-values. Furthermore, a fixed weir can be assigned a crest length (a thin dam is infinitely thin). The z-values correspond to the height of the fixed weir at the corresponding x- and y-locations; the height can vary in space, but is constant in time (Figure 4.16). Consequently, a fixed weir can overflow if the water level exceeds the height of the fixed weir. The height is specified with regards to the same vertical reference level as all other model items with height specifications (e.g. bathymetrical values and initial water levels).

**Figure 4.15:** Geographical and grid snapped representation of a fixed weir**Figure 4.16:** Schematic representation of a fixed weir

To add a fixed weir, click the corresponding icon from the “FM Region 2D / 3D” menu in the map ribbon (Figure 4.6). Adding, deleting, importing and exporting of a line feature such as a fixed weir is discussed in more detail in Section 4.4.2.3 on cross sections.

When you double click the “Fixed weirs” attribute in the project tree, the corresponding Fixed

weirs tab appears underneath the central map ([Figure 4.17](#)). Alternatively, you can also double click on any fixed weir in the central map. Within this tab, an attribute table is shown which displays the names of all fixed weirs within your model. When one of the entries is selected, the corresponding item is highlighted in the central map. A fixed weir entry can be deleted from the table via the context menu (right mouse button).

Fixed weirs
Name
FixedWeir01
FixedWeir02
FixedWeir03
FixedWeir04
► FixedWeir05

Figure 4.17: Attribute table with fixed weirs

When you double click on a fixed weir in the central map, the fixed weir editor opens in a separate view ([Figure 4.18](#)). On the right, a graphic representation (top view) of the fixed weir is displayed. The support point that is selected in the table is highlighted by means of a blue circle. On the left, a table is displayed showing the following properties of each support point of the fixed weir under consideration:

- ◊ X: x-location of the support point
 - ◊ Y: y-location of the support point
 - ◊ Crest level: height of fixed weir (spatially varying but fixed in time)
 - ◊ Crest length: length of the crest (instead of an infinitely thin crest)
 - ◊ Left ground level: ground level to the left of the crest; together with crest height determines slope of crest on the left side
 - ◊ Right ground level: ground level to the right of the crest; together with crest height determines slope of crest on the right side

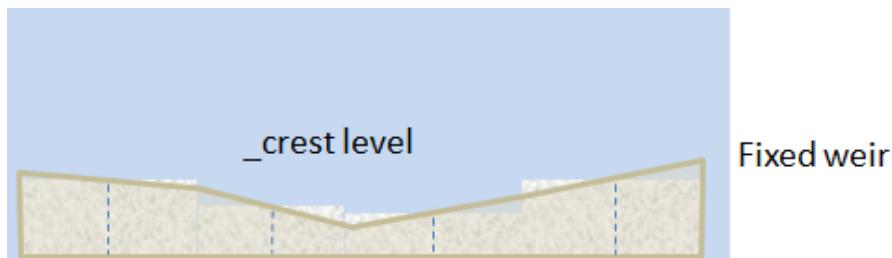


Figure 4.18: Fixed weir editor

4.4.2.6 Land boundaries

A land boundary ([Figure 4.19](#)) encloses the main geographic features surrounding your model and indicates the intersection of the water and land masses. When you set up your computational grid in RGFGRID, a land boundary determines the onshore extent of your model. When you open RGFGRID to edit your grid, the land boundary is automatically transferred and displayed. For more details on grid generation, you are referred to the User Manual of RGFGRID.

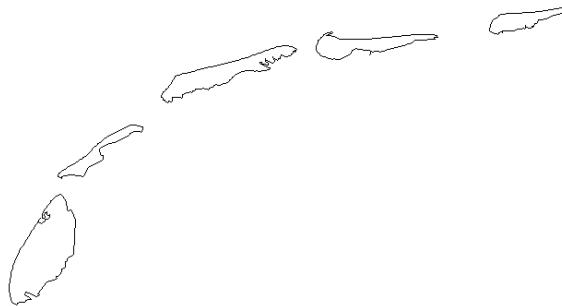


Figure 4.19: Geographical representation of a land boundary

To add a land boundary, click the corresponding icon from the “FM Region 2D / 3D” menu in the map ribbon ([Figure 4.6](#)). Adding, deleting, importing and exporting of a line feature such as a land boundary is discussed in more detail in [Section 4.4.2.3](#) on cross sections.

When you double click the “Land boundaries” attribute in the project tree, the corresponding land boundaries tab appears underneath the central map ([Figure 4.20](#)). Alternatively, you can also double click on any land boundary in the central map. Within this tab, an attribute table is shown which displays the names of all land boundaries within your model. When one of the entries is selected, the corresponding item is highlighted in the central map. A land boundary entry can be deleted from the table via the context menu (right mouse button).

Land boundaries X	
	Name
►	Texel
	Vlieland
	Terschelling
	Ameland
	Schiermonnikoog

Record 1 of 5

Figure 4.20: Attribute table with land boundaries

4.4.2.7 Dry points and dry areas

Dry points are grid cells centred around a water level point that are permanently dry during a computation, irrespective of the local water depth and without changing the water depth as seen from the wet points. Dry areas are the same, but provide an easy way of defining many grid points as a single dry area at once.

Note that the flexibility of unstructured grids makes the use of dry points less necessary than

with structured grid models, such as Delft3D-FLOW. In the interior of unstructured grids, some or more grid cells can easily be deleted during grid manipulation, e.g., in RGFGRID. Still, a dry points file can be used to explicitly mark locations or regions inside the grid as dry cells.

Dry points in the GUI

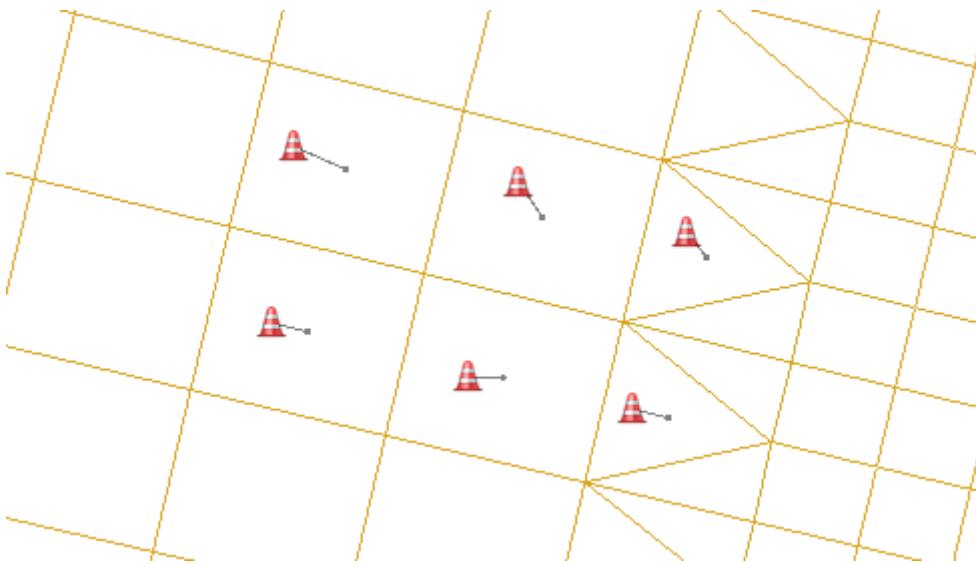


Figure 4.21: Geographical and grid snapped representation of several dry points

To add a dry point, click the corresponding icon from the “FM Region 2D / 3D” menu in the map ribbon ([Figure 4.6](#)). Adding, deleting, importing and exporting of a point feature such as a dry point is discussed in more detail in [Section 4.4.2.2](#) on observation points. The grid snapped representation of a dry point ([Figure 4.21](#)) is indicated by a line linking the dry point to the closest cell center.

When you double click the “Dry points” attribute in the project tree, the corresponding Dry points tab appears underneath the central map ([Figure 4.22](#)). Alternatively, you can also double click on any dry point in the central map. Within this tab, an attribute table is shown which displays the names of all dry points within your model. When one of the entries is selected, the corresponding item is highlighted in the central map. A dry point entry can be deleted from the table via the context menu (right mouse button).

Dry points X	
X	Y
1.5374E+05	5.7661E+05
1.537E+05	5.7644E+05
► 1.5352E+05	5.7649E+05
1.5354E+05	5.7665E+05
1.539E+05	5.7657E+05
1.5385E+05	5.7641E+05

◀◀ ◀ ◀ Record 3 of 6 ▶ ▶▶ ▶▶ + - ▲ ▼ ✖ ◀

Figure 4.22: Attribute table with dry points

Dry areas in the GUI

Dry areas ([Figure 4.23](#)), like dry points, indicate areas that permanently dry during a computation. Instead of adding many separate dry points, you can draw a polygon that encloses all required computational cells. Only cells which centers are strictly inside the polygon are taken into account. The grid snapped representation of the dry area clearly indicates which cells are considered within the dry area.

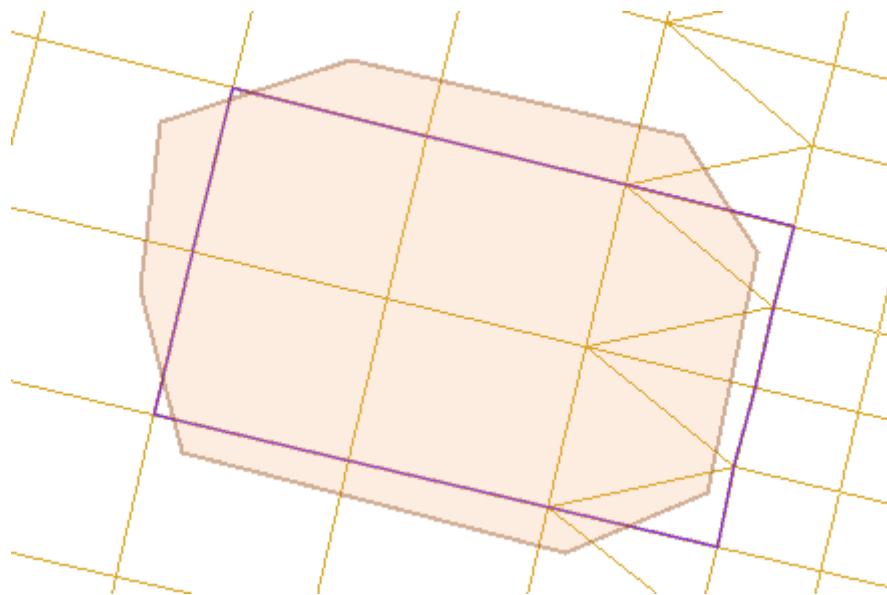


Figure 4.23: Geographical and grid snapped representation of a dry area

When you double click the “Dry areas” attribute in the project tree, the corresponding Dry areas tab appears underneath the central map ([Figure 4.24](#)). Alternatively, you can also double click on any dry area in the central map. Within this tab, an attribute table is shown which displays the names of all dry areas within your model. When one of the entries is selected, the corresponding item is highlighted in the central map. A dry area entry can be deleted from the table via the context menu (right mouse button).

Dry areas X	
	Name
	DryArea01
	DryArea02
	DryArea03
	DryArea04
►	DryArea05

◀◀	◀◀	◀	▶▶	▶▶	▶	+	-	▲	▼	✖	✖	◀
Record 5 of 5												

Figure 4.24: Attribute table with dry areas

Dry points file input

Dry points are defined by a sample file <*.xyz>, dry areas are defined by a polygon file <*.pol>. Add the filename to the MDU as below:

```
[geometry]
# ...
DryPointsFile = <filename.xyz> # Dry points file *.xyz, third column dummy z values,
# or polygon file *.pol.
```

The format of the sample file is defined in [Section B.3](#). The format of the polygon file is defined in [Section B.2](#). All grid cells that contain a sample point are removed from the model grid, and as a result do not appear in any of the output files. Alternatively, for a polygon file, all grid cells whose mass center lies within the polygon will be removed from the model grid.

Finally, an optional flag can be set in the polygon file to invert the masking behavior of the polygon. That is: all points *outside* of the polygon will be marked as dry and therefore removed. To realize this, the polygon block should have three columns, and the first point should have a 'z' value of -1 in the third column.

4.4.2.8 Dry areas

4.4.2.9 Pumps

Pumps are a type of structures in D-Flow FM. Unlike the other structures, pumps can force the flow only on one direction. This direction is determined by arrow in D-Flow FM. The direction of pump can be reverted by mouse right-click and selecting "Reverse line(s)".

Like all other structures in D-Flow FM, the pump can be defined by a polygon. The input data of the pumps can be given by selecting and editing the pump polygon (see [Figure 4.25](#)). Right click on the pump polygon and selecting "Delete Selection" leads to deletion of the selected pump. Double clicking the pump polygons (or right click the pump in the list and select "Open view"), it opens a tab for editing the pump properties. The tab includes pump capacity. If the pump capacity is time dependent, it can be given by time series data ([Figure 4.25](#)).

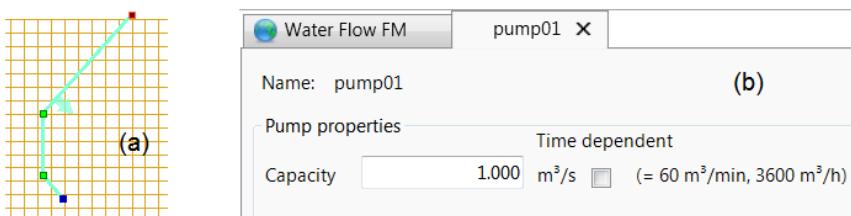


Figure 4.25: Polygon for pump (a) and adjustment of physical properties (b).

Right clicking the pumps attribute in the project tree opens a pop-down window on which you can select to import or export pumps. The pumps can be imported as polygon by a .pli file or by a structure file, and they can be exported as .pli file, structure file, or shapefile.

Double clicking the pumps attribute in the project tree opens the pumps tab underneath the central map. The attribute table in this tab shows all pumps with their corresponding proper-

ties. When one of the pumps is selected, the corresponding item is highlighted in the central map. Double clicking any of the pumps in the central map opens the Structure Editor as a new map view in which all parameters related to the pump can be set (Figure 4.26).

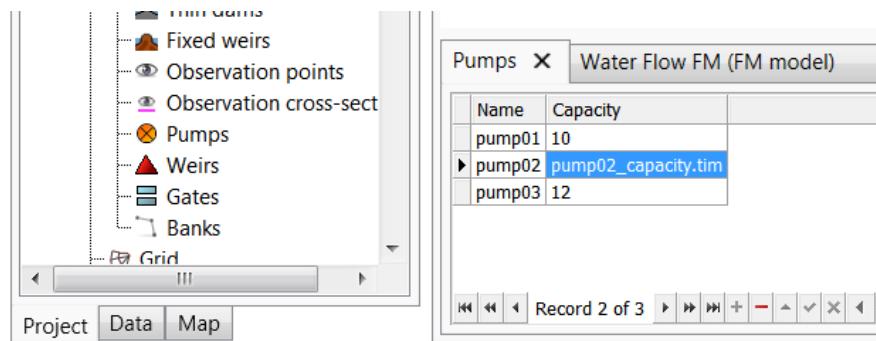


Figure 4.26: Selection of the pumps

4.4.2.10 Weirs

Unlike the fixed weir, weir (or adjustable weir) can be adjusted based on the user requirements. To set an adjustable weir in the computational domain, you can select the icon "weir" from the toolbar, and draw a line by mouse. This line includes direction, which defines the sign of total flux passing above the weir (positive flux in the direction of weir, otherwise negative). This direction can be inverted by mouse right-click and selecting "Reverse line(s)". By double-click on the weir, you can add the geometrical and time-dependent parameters such as "Crest level", "Crest width", "Crest level time series" and "Lateral concentration coefficient" (See Figure 4.27).

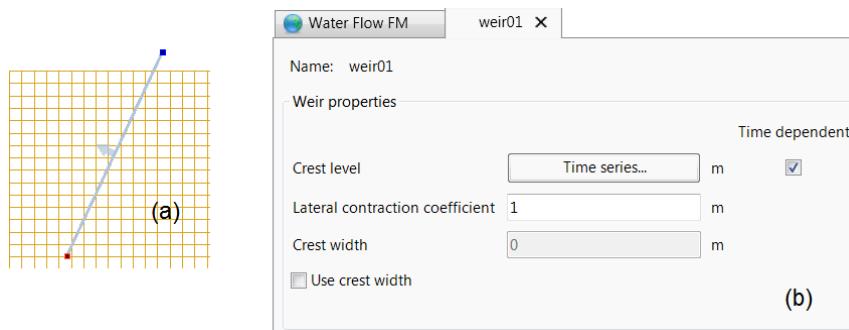
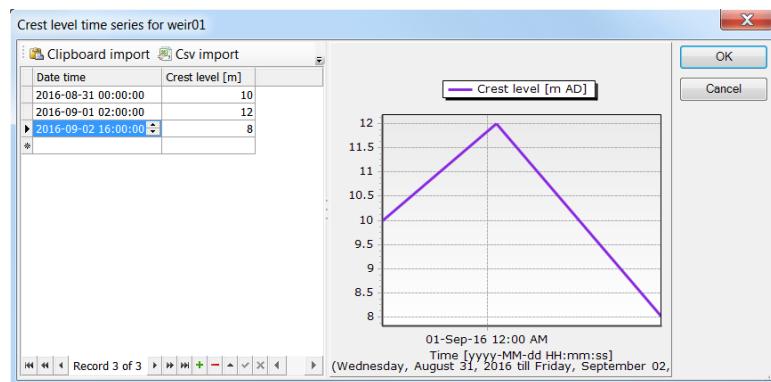


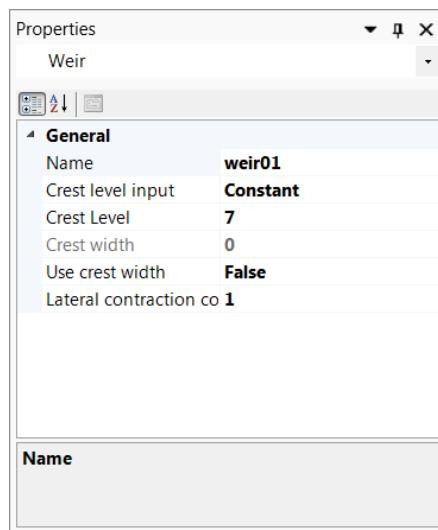
Figure 4.27: Polygon for adjustable weir (a) and adjustment of geometrical and temporal conditions (b).

Moreover, the time series of the crest level can be set in the case the crest level is time dependent. The time dependency diagram can be defined by the help of time series diagram as shown in Figure 4.28. The time series can also be imported (and exported) from external "csv" file.

**Figure 4.28:** Time series for crest level.

The weirs can be deleted, imported and exported. By right clicking on the weir polygon, and selecting "Delete Selection" from the pop-down window, you can delete the selected weir. Right clicking the "Weirs" attribute in the project tree opens the "Weirs" tab opens the options for import and export. You can import weirs as polygon (*.pli file) or as a structure by structure file. The weirs can also be exported to a polygon file, to a structure data file, or by the help of shapefile.

Double clicking the "Weirs" attribute in the project tree opens the "Weirs" tab underneath the central map. The attribute table in this tab shows all weirs with their corresponding properties. When one of the weirs is selected, the corresponding item is highlighted in the central map. Double clicking any of the weirs in the central map opens the Structure Editor as a new map view in which all parameters related to the weir can be set ([Figure 4.29](#)).

**Figure 4.29:** Time series for crest level.

4.4.2.11 Gates

In D-Flow FM the gates can be imposed by polygon, and can be edited in a similar way as the other structures (see [Figure 4.30](#)). Like the other structures, mentioned above, the gates can be imported and exported by means of structure file or .pli file.

[Figure 4.30](#) shows the edit tab of the gate properties. The gate can be opened horizontally, as well as vertically.

Note: Unlike the weirs, the flow cannot flow above the gate in the present release of D-Flow FM.

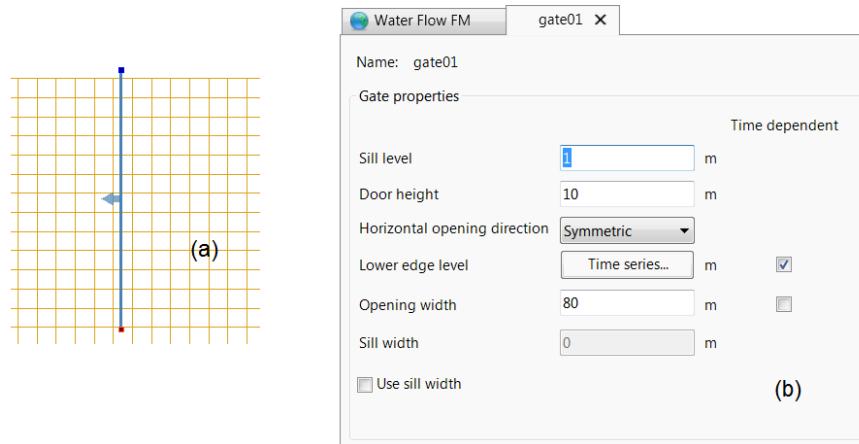


Figure 4.30: Polygon for gate (a) and adjustment of geometrical and temporal conditions (b).

4.4.3 Computational grid

To set up your grid, click on the “Edit grid” button which opens the program RGFGRID. All features of grid setup in RGFGRID are treated separately in [Appendix A](#). If a land boundary is present in your project, this is exported to RGFGRID automatically. Once you have setup your grid in RGFGRID, click “Save Project” and close the program. The grid will now be visible within the central map. Editing of the grid remains possible at any point in time during the setup of your model by means of clicking the “edit grid” button. Any changes you make are always saved after clicking “Save Project” and loaded back into the central map.

4.4.4 Bathymetry

When you double click on ‘Bathymetry’ in the project tree or select ‘Bathymetry’ from the dropdown box in the spatial editor section of the ‘Map’ ribbon, the spatial editor is activated ([Figure 4.31](#)). This editor can be used to generate a bathymetry for your computational grid. How to work with the spatial editor is described in [Appendix F](#). Be aware that the bathymetry in D-Flow FM is defined as the bed level (e.g. positive upward), implying that all bed levels below the reference plane are negative. By default the bed levels are defined on the net nodes.

Note: Please note that, currently, other bed level definition types (e.g. BedlevTypes) are not visually supported by the GUI. If you would like to switch the bed level definitions to another type, you have to set the BedlevType in the ‘Physical Parameters’ tab. However, the bed level locations will not be updated accordingly in the central map.



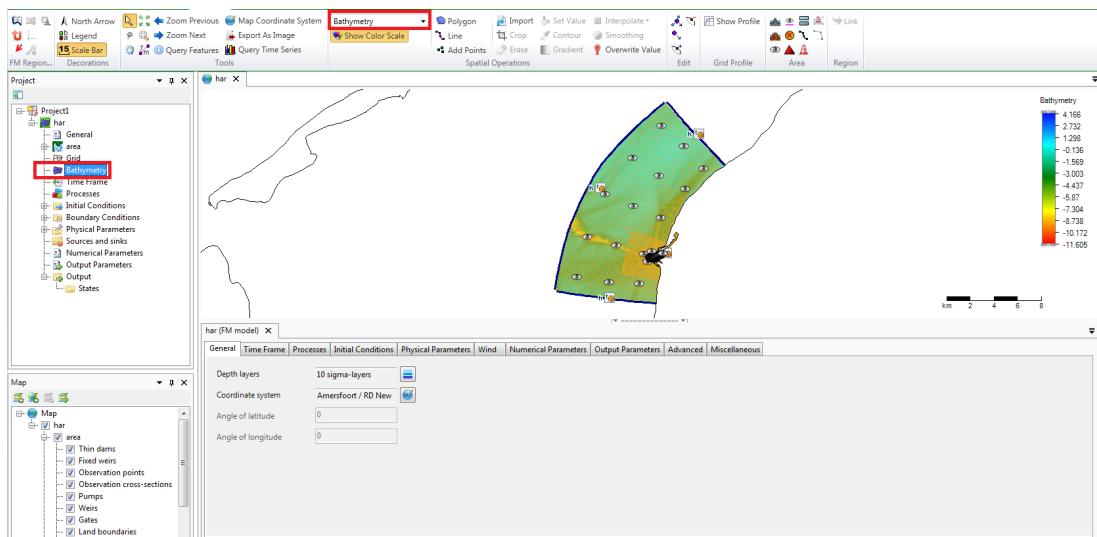


Figure 4.31: Bathymetry activated in the spatial editor

4.4.5 Time frame

In the settings tab, in the sub-tab time frame (Figure 4.32), you can specify everything related to the time frame in which your model will run.

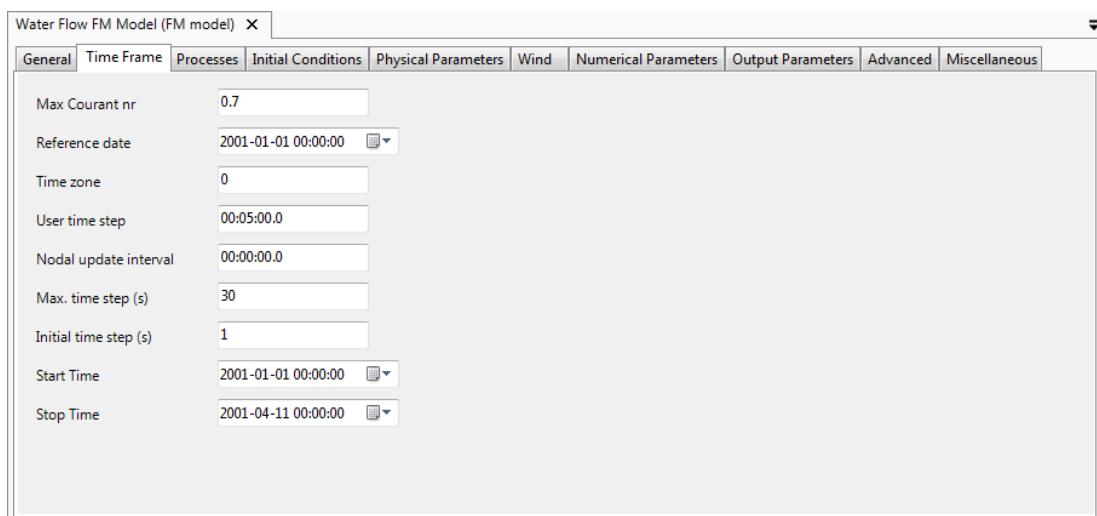


Figure 4.32: Overview time frame tab

In general, the time frame is defined by a reference date and a start and stop time. The time step size of your model is automatically limited (every time step) based on a Courant condition. In more detail, you must define the following input data:

Max Courant nr

The maximum allowed Courant number, which is used to compute the time step size from the CFL criterium. D-Flow FM uses an explicit advection scheme, therefore a value of 0.7 or lower is advised.

Remark:

◊ You should check the influence of the time step on your results at least once.

Reference date

The reference date and time of the simulation. It defines the (arbitrary) $t = 0$ point for all time-series as used in the simulation. In the

	GUI, time-specifications are always absolute, but in the underlying model input files, these are stored as time values relative to the reference date. Typically, input time-series files are specified in minutes after this $t = 0$ point.
<i>Time zone</i>	The time difference between local time and UTC. The time zone is defined as the time difference (in hours) between the local time (normally used as the time frame for D-Flow FM) and Coordinated universal time (UTC). The local Time Zone is used for two processes: <ul style="list-style-type: none"> ◊ To determine the phases in local time of the tidal components when tide generating forces are included in the simulation, see Section 7.10. ◊ To compare the local time of the simulation with the times at which meteo input is specified, e.g., wind velocities and atmospheric pressure. These can be specified in a different time zone.
<i>User time step</i>	If the <i>Time Zone</i> = 0 then the simulation time frame will refer to UTC. The interval that is highest in the hierarchy. It specifies the interval with which the meteorological forcings are updated. The <i>Max. time step</i> cannot be larger than the <i>User time step</i> , and it will automatically be set back if it is. Also, the output intervals should be a multiple of this <i>User time step</i> , see Appendix E . Finally the computational time steps will be fitted to end up exactly at each <i>User time step</i> , such that proper equidistant output time series are produced.
<i>Nodal update interval</i>	When using astronomic boundary conditions, the nodal factors can be updated with certain intervals, see section 7.10 .
<i>Max. time step</i>	The <i>Max. time step</i> is the upper limit for the computational time step. The automatic time step can not be switched off explicitly. (If you want to enforce a fixed time step anyway, set the parameter <i>Max. time step (s)</i> to the desired step size, and the parameter <i>Max. Courant nr.</i> to an arbitrary high value.)
<i>Initial time step</i>	the initial time step of the model; there is no data available yet during the first time step to compute the time step automatically based on a Courant condition. The computational time step then gradually increases from <i>Initial time step</i> to the CFL-number limited time step (assuming that <i>Initial time step</i> is relatively small).
<i>Start Time</i>	The start date and time of the simulation.
<i>Stop Time</i>	The stop date and time of the simulation. Always make sure that the model <i>Stop Time</i> is larger than the model <i>Start Time</i> to avoid errors during your calculation.

4.4.6 Processes

In the processes tab ([Figure 4.33](#)) you can specify which processes you want to incorporate into your model. You can choose whether or not to include tidal forcing and salinity by means of check boxes. In addition, you can specify which *heat flux model* and which *Wave model* you want to use.

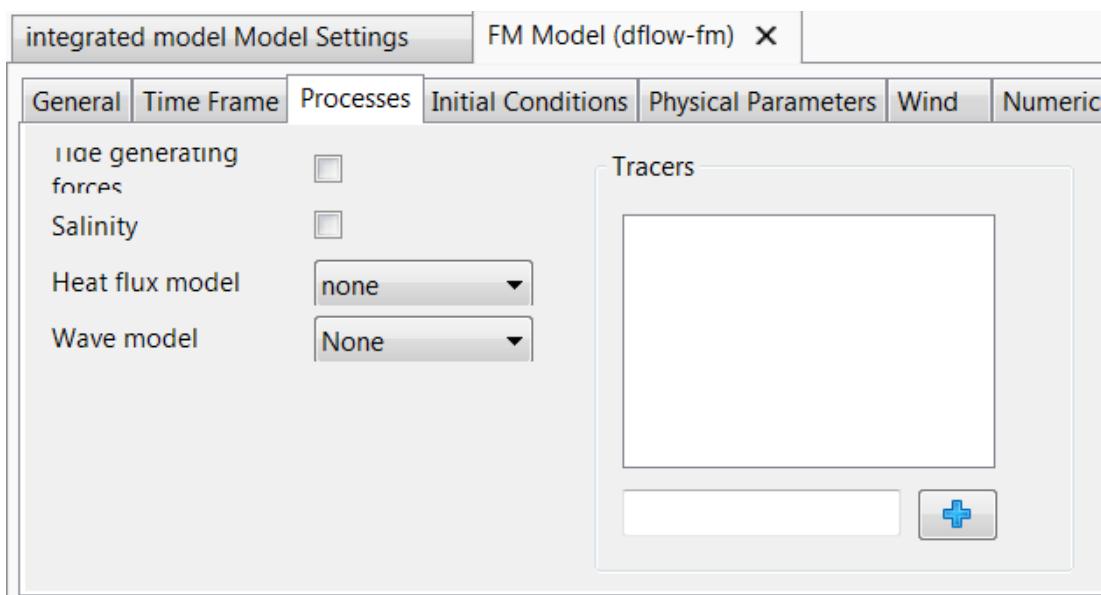


Figure 4.33: Overview processes tab

4.4.7 Initial conditions

When expanding the initial conditions in the project tree, all quantities requiring an initial state are shown (Figure 4.34). The number of quantities depends on the activated physical processes in the ‘Processes’ tab (see Section 4.4.6). The initial conditions for each quantity can be specified as a uniform value or as a coverage (e.g. a spatially varying field).

The uniform values can be edited in the ‘Initial Conditions’ tab, which opens upon double clicking ‘Initial Conditions’ in the project tree (Figure 4.35). In this tab you can also specify the layer distribution for the initial condition specification in case of a 3 dimensional quantity (i.e. salinity). **Note:** Please note that for 3 dimensional initial conditions currently only the option ‘top-bottom’ is supported.



In case of spatially varying initial conditions you can double click the quantity in the project tree or select it from the dropdown box in the spatial editor section of the /emphSpecial Operations ribbon (Figure 4.36). Then the spatial editor is activated, which you can use to edit spatially varying fields. For more information on how to use the spatial editor you are referred to Appendix F. In case of 3 dimensional initial conditions, you can select the layer from the quantity dropdown box in the ‘Map’ ribbon (Figure 4.37).

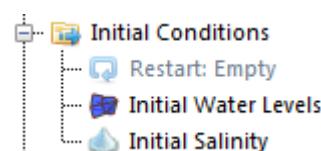


Figure 4.34: Initial conditions in the project tree

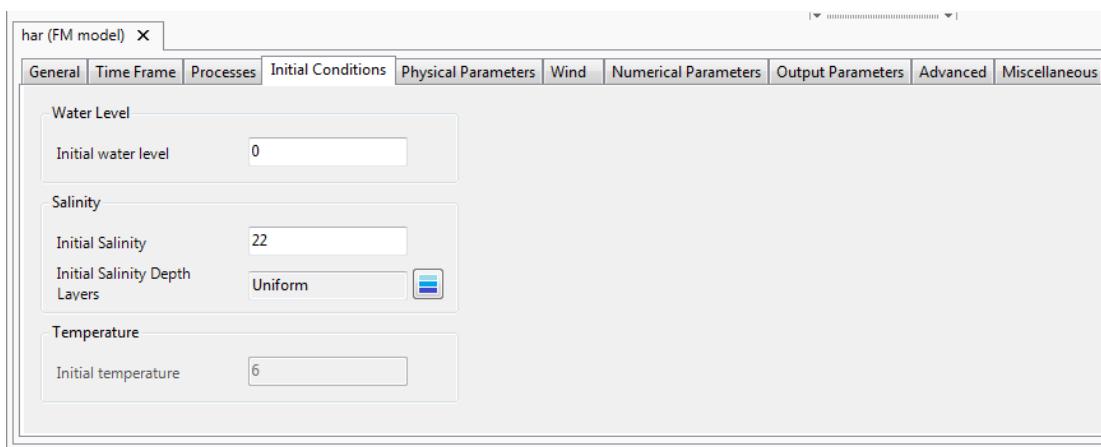


Figure 4.35: The 'Initial Conditions' tab where you can specify the uniform values and the layer distributions of the active physical quantities.

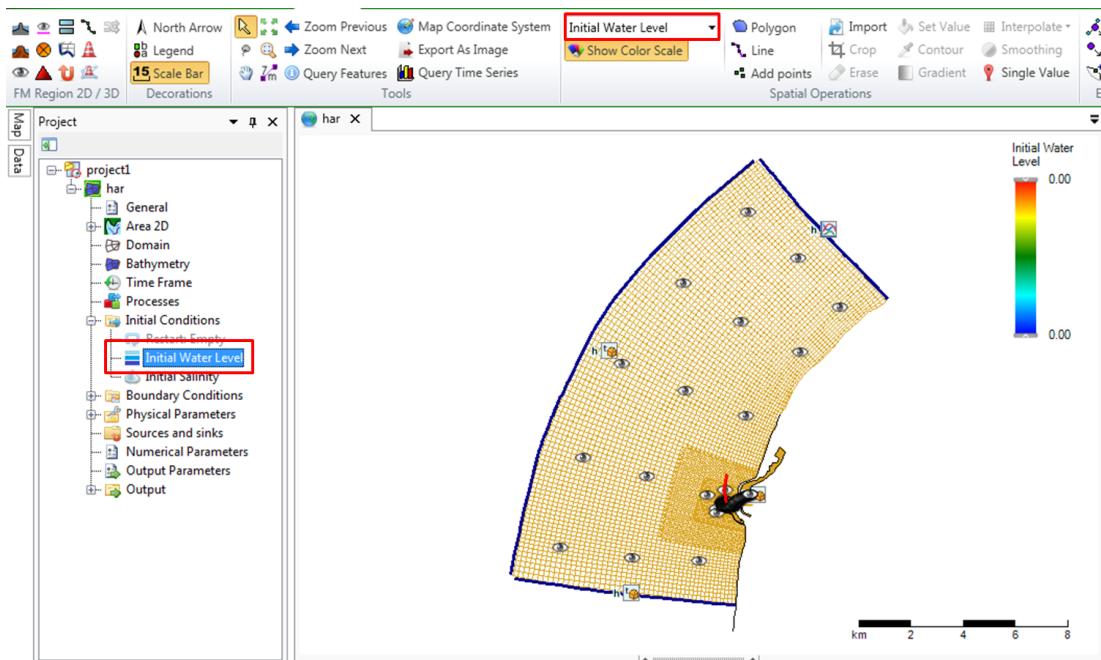


Figure 4.36: Initial water levels activated in the spatial editor

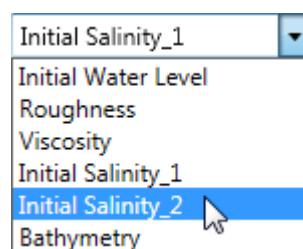


Figure 4.37: Selecting 3 dimensional initial fields from the dropdown box in the 'Map' ribbon to edit them in the spatial editor

Instead of defining initial conditions from scratch you can also import fields from a previous computation (using restart files). When you run a model using the Delta Shell GUI which

is writing restart files, the restart states will appear in the “Output” folder in the project tree ([Figure 4.38](#)). For a description of the specification of restart files, please refer to paragraph [Section 4.4.12](#). To use a restart file as initial conditions use the RMB and select “Use as initial state”. The file will now appear under “Initial Conditions” in the project tree ([Figure 4.39](#)). To activate the restart file use the RMB and select “Use restart”. The file will no longer be grey, but is now highlighted in black. The model will now restart from this file.

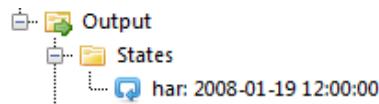


Figure 4.38: Restart files in output states folder

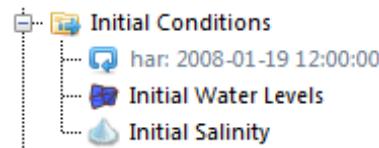


Figure 4.39: Restart file in initial conditions attribute

4.4.8 Boundary conditions

Boundary conditions consist of a location specification ('support points') and a forcing for that location. [Section 4.4.8.1](#) describes how support points can be specified in Delta Shell. The boundary forcing can be specified in the boundary data editor. [Section 4.4.8.2](#) describes the functionality of the boundary data editor. Finally, [section 4.4.8.4](#) describes how the user can get an overview of the boundary locations and forcing in the attribute table.

4.4.8.1 Specification of boundary locations (support points)

In D-Flow FM the boundary locations are defined as ‘support points’ on a polyline (*.pli). The user can add a boundary polyline (*.pli) in the central map by selecting the ‘Add Boundary’ icon in ‘FM Region 2D / 3D’ of the ‘Map’ ribbon (see [Figure 4.40](#)). The number of individual mouse clicks determines the number of support points on the polyline. The polyline is closed by a double click. Once the polyline is added it becomes visible in the project tree under ‘Boundary Conditions’ (see [Figure 4.41](#)). The polyline can be edited by the general edit operations in the ‘Map’ ribbon (i.e. add/delete/move individual geometry points or the complete geometry – see [Figure 4.42](#)). The name of the polyline (or ‘boundary’) can be edited in the Boundaries tab, which can be opened by double clicking ‘Boundaries’ in the ‘Map Tree’ (see [Figure 4.43](#)).

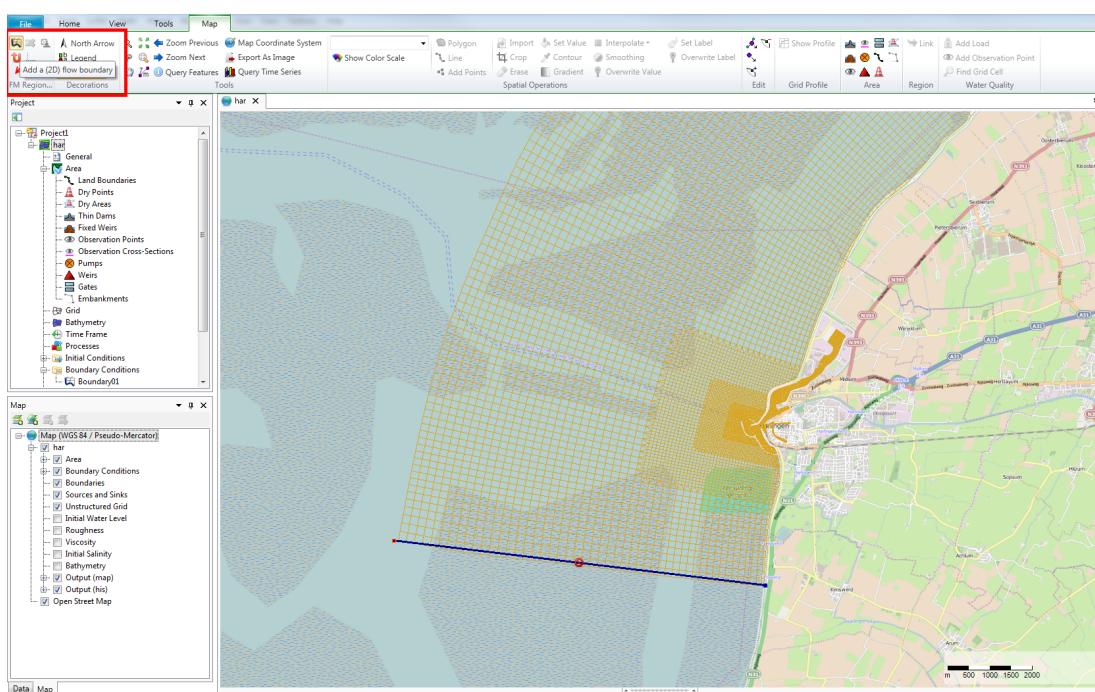


Figure 4.40: Adding a boundary support point on a polyline in the central map. By double clicking on the polyline in the map, the boundary condition editor will open to edit the forcing data on the polyline.

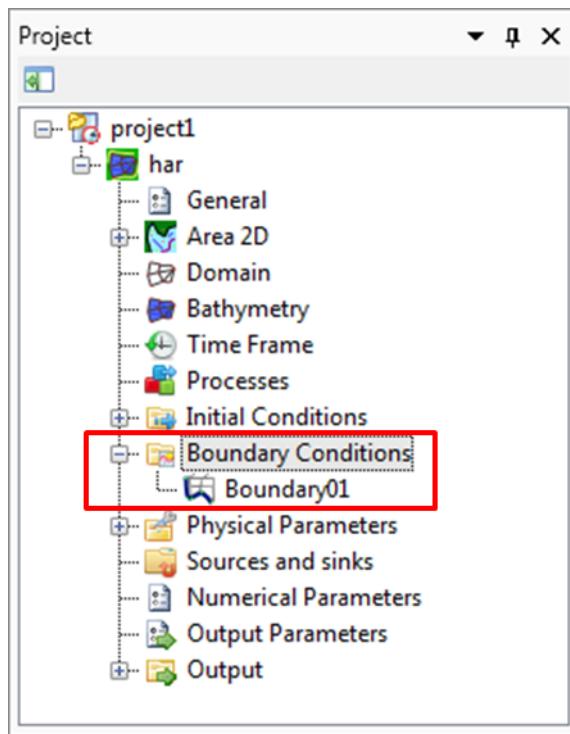


Figure 4.41: Polyline added in project tree under 'Boundary Conditions'. By double clicking on the name of the polyline, the boundary condition editor will open to edit the forcing data on the polyline.

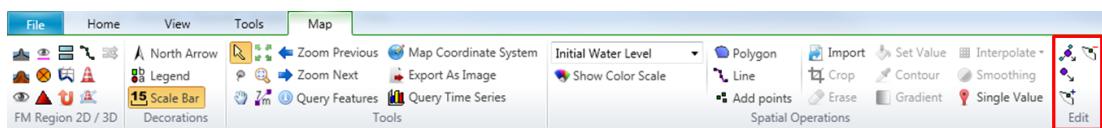


Figure 4.42: Geometry edit options in 'Map' ribbon

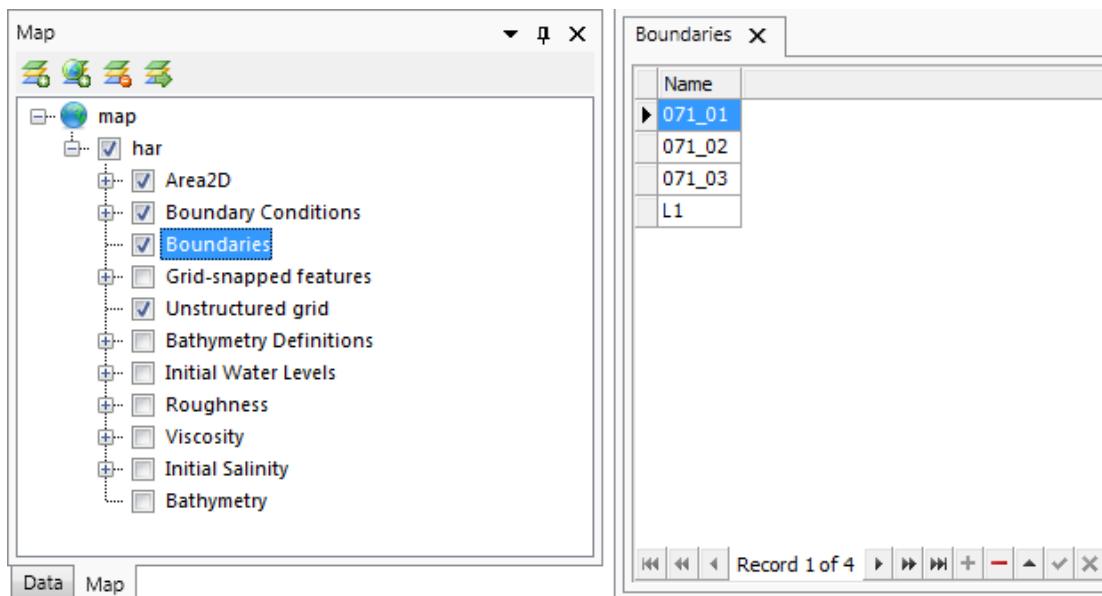


Figure 4.43: Edit name of polyline/boundary in Boundaries tab

4.4.8.2 Boundary data editor (forcing)

The boundary condition editor can be opened either by double clicking on the boundary name in the project tree (Figure 4.41) or by double clicking the boundary polyline in the map window (Figure 4.40). An overview of the boundary data editor is given in Figure 4.44. This editor can be used to specify the boundary forcing for different quantities (i.e. water level, velocity, discharge, salinity, etc.) corresponding to different processes (i.e. flow, salinity, temperature, tracers). The user can select the processes and quantities in the upper left corner. In the upper centre panel the user can select the forcing function for the selected quantity (i.e. time series, harmonic components, astronomical components or Q-h relation). The upper right corner contains a list of the support points on the polyline. The geometry view shows the location of the selected support point on the polyline (*.pli). In the middle panel are some handy buttons to generate, import and export forcing data. In the lower left panel the user can specify the boundary data for the selected support point. The lower right corner shows the signal of the boundary data. The following sections describe the features of the boundary data editor in more detail.

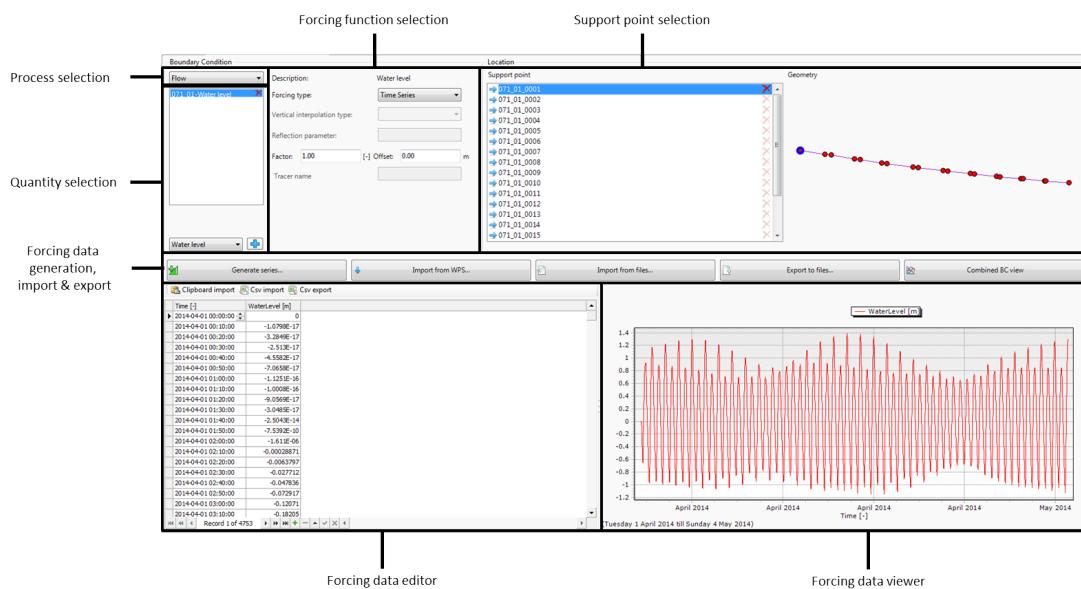


Figure 4.44: Overview of the boundary data editor

Process and quantity selection:

Currently, D-Flow FM supports the processes flow, salinity, temperature and tracers (**Note: tracers are not yet fully editable**). The processes available in the boundary condition editor depend on the selected processes in the processes tab (see [Section 4.4.6](#)). After selecting the process from the dropdown box the user can select one of the corresponding quantities, as illustrated in [Figure 4.45](#). For the process flow the user can choose from five principal quantities: water level, velocity, Riemann invariant, Neuman gradient and discharge. All quantities are specified per support point, except for discharges which are specified per polyline (*.pli). A support point can have multiple forcing quantities of the same type (i.e. water level, velocity, Riemann, Neumann or discharge). These quantities are added up. This can be relevant to add a surge level to an astronomical water level for example. Furthermore, the user can apply normal and tangential velocities as 'add on' quantities. The following combinations of quantities are allowed:

- ◊ Water level + normal velocity
- ◊ Water level + tangential velocity
- ◊ Water level + normal velocity + tangential velocity
- ◊ Velocity (= normal) + tangential velocity
- ◊ Riemann + tangential velocity



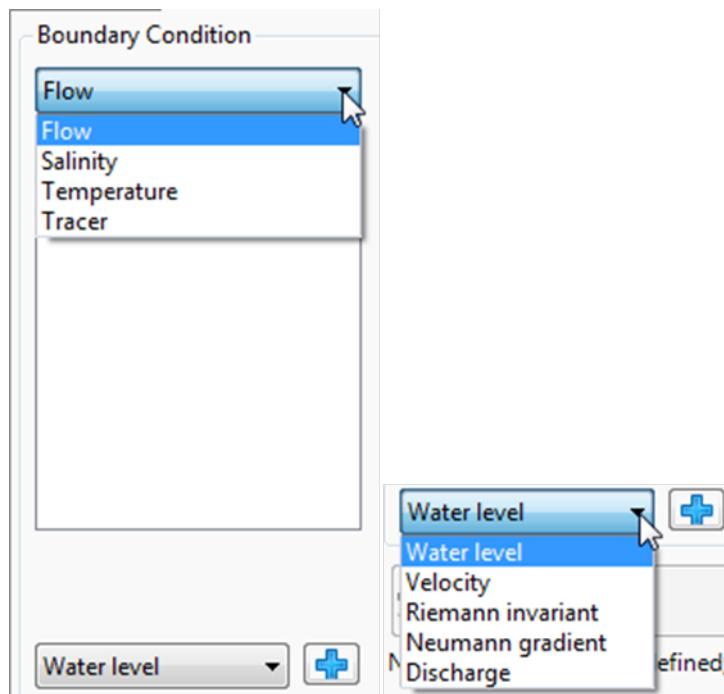


Figure 4.45: Process and quantity selection in the boundary data editor

Forcing function selection:

The user can select one of the following forcing functions:

- ◊ Time series
- ◊ Harmonic components
- ◊ Harmonic components + correction
- ◊ Astronomic components
- ◊ Astronomic components + correction
- ◊ Q-h relation (only for water levels)

The next section describes how data can be added for these types of forcing functions.

Add forcing data to a support point or polyline:

By default all support points on the polyline are deactivated. To add forcing data to a support point the user first needs to activate it by pressing the green 'add'-symbol in the list of support points (see [Figure 4.46](#)). Consequently, the user can specify the forcing data in the lower left panel based on the selected forcing function. Of which a preview is shown in the lower right panel. **Note: Please note that once a support point containing forcing data is made inactive, all data on the support point is lost!**



The user can choose from the forcing functions time series, harmonic components (+ correction), astronomic components (+ correction) and Q-h relation. Examples of the boundary data specifications for these different forcing functions are given below. **Note: Please note that once the user changes the forcing function of a polyline, all data on the polyline is lost!**



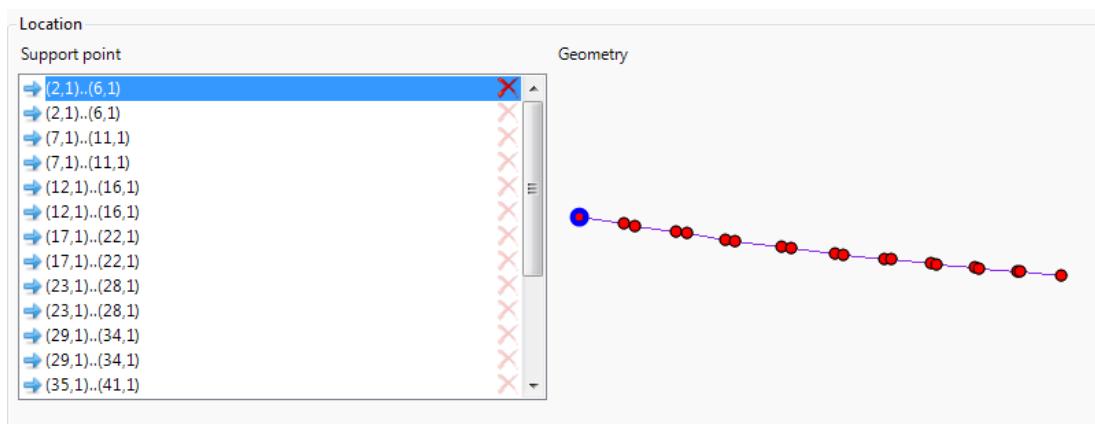


Figure 4.46: Activate a support point

Time series

The time format for time series is yyyy-mm-dd HH:MM:SS. There are multiple ways to specify time series for the selected quantity:

- ◊ Specify the time series step by step in the table (Figure 4.47): the user can add or delete rows with the “plus”- and “minus”-signs below the table.

	Time [-]	WaterLevel [m]
	2010-12-15 00:00:00	0
	2010-12-16 00:00:00	1

Record 3 of 3

Figure 4.47: Specification of time series in the boundary data editor (left panel)

- ◊ Generate time series using the ‘Generate time series’ button (Figure 4.48): the user can specify start time, stop time and time step.

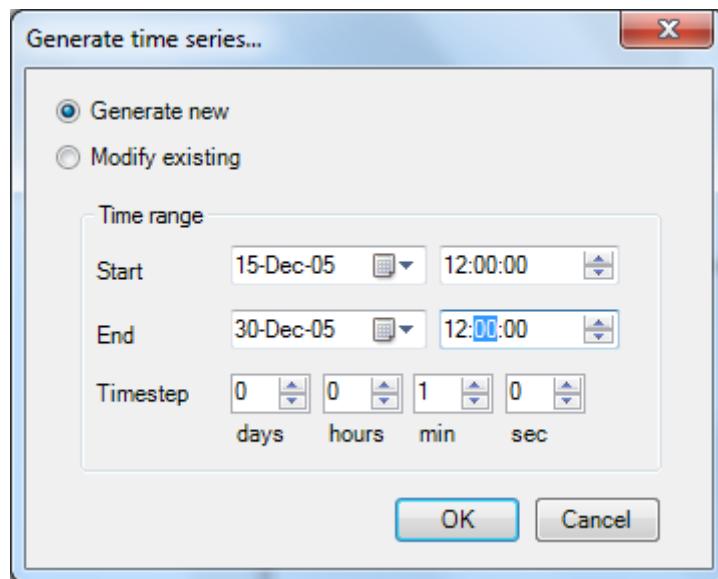


Figure 4.48: Window for generating series of time points

- ◊ Import from csv using the 'Csv import' button: a wizard will open in which the user can (1) select a csv-file ([Figure 4.49](#)), (2) specify how data should be parsed into columns ([Figure 4.50](#)) and (3) how the values should be parsed and mapped into columns ([Figure 4.51](#)).

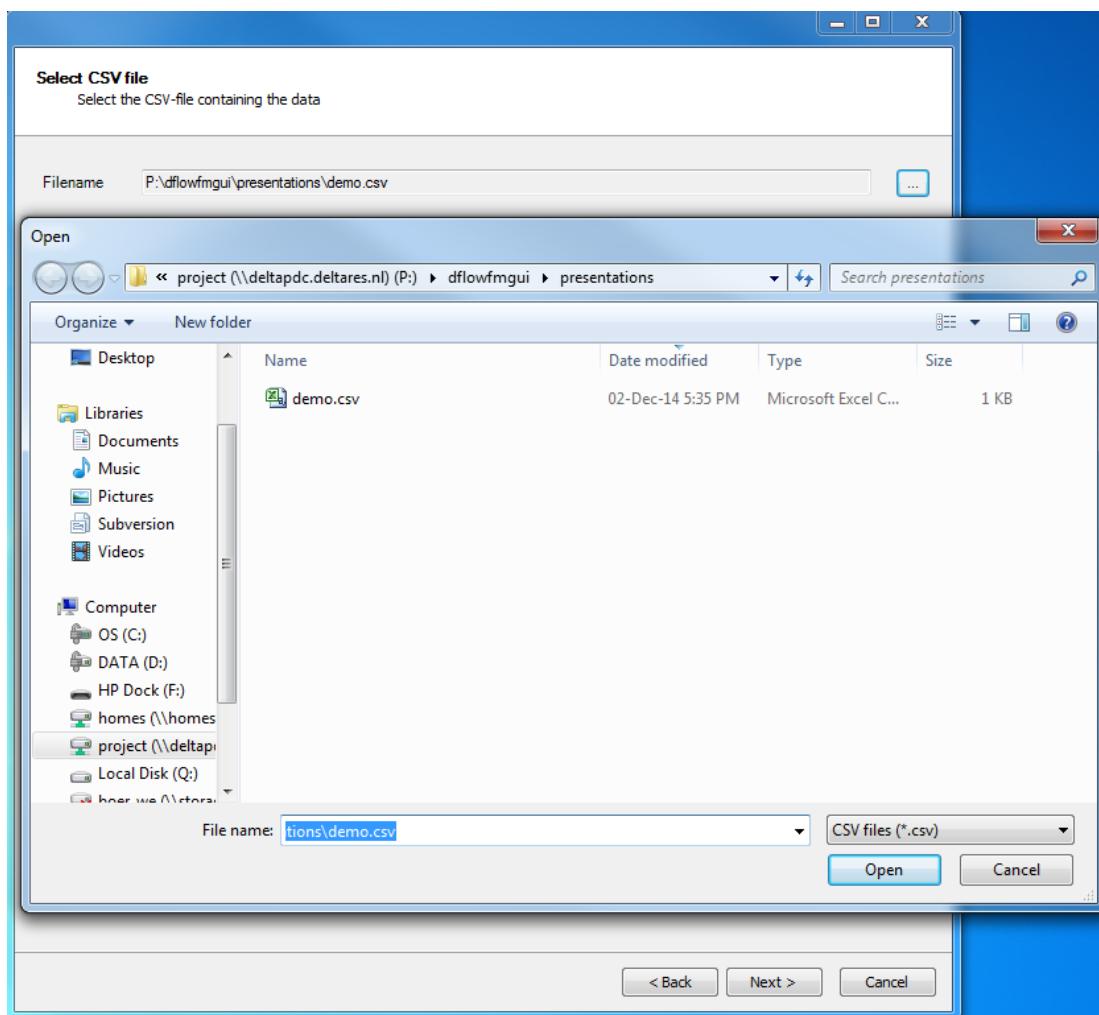


Figure 4.49: Csv import wizard: csv file selection

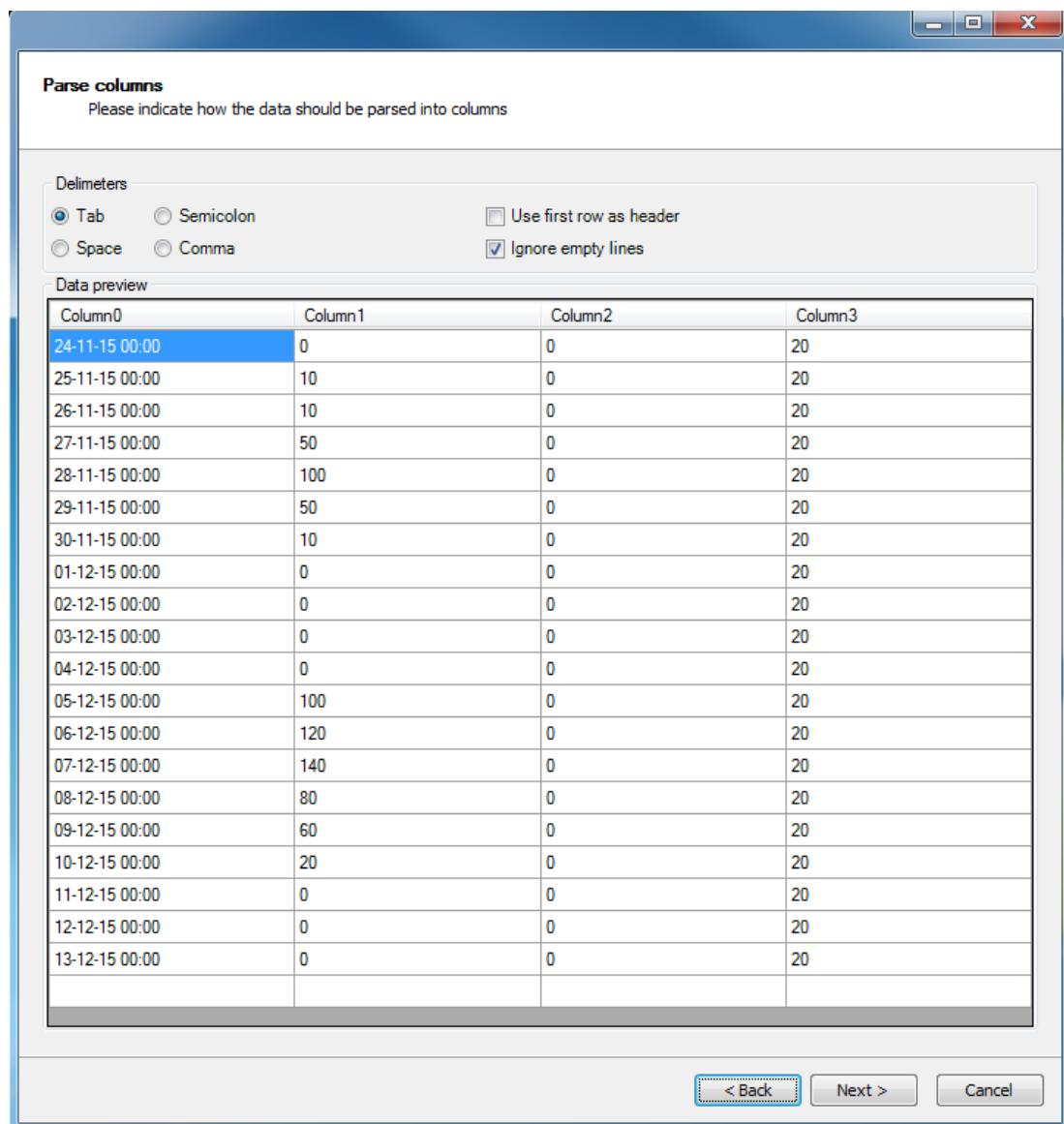


Figure 4.50: Clipboard/csv import wizard: specification of how data should be parsed into columns

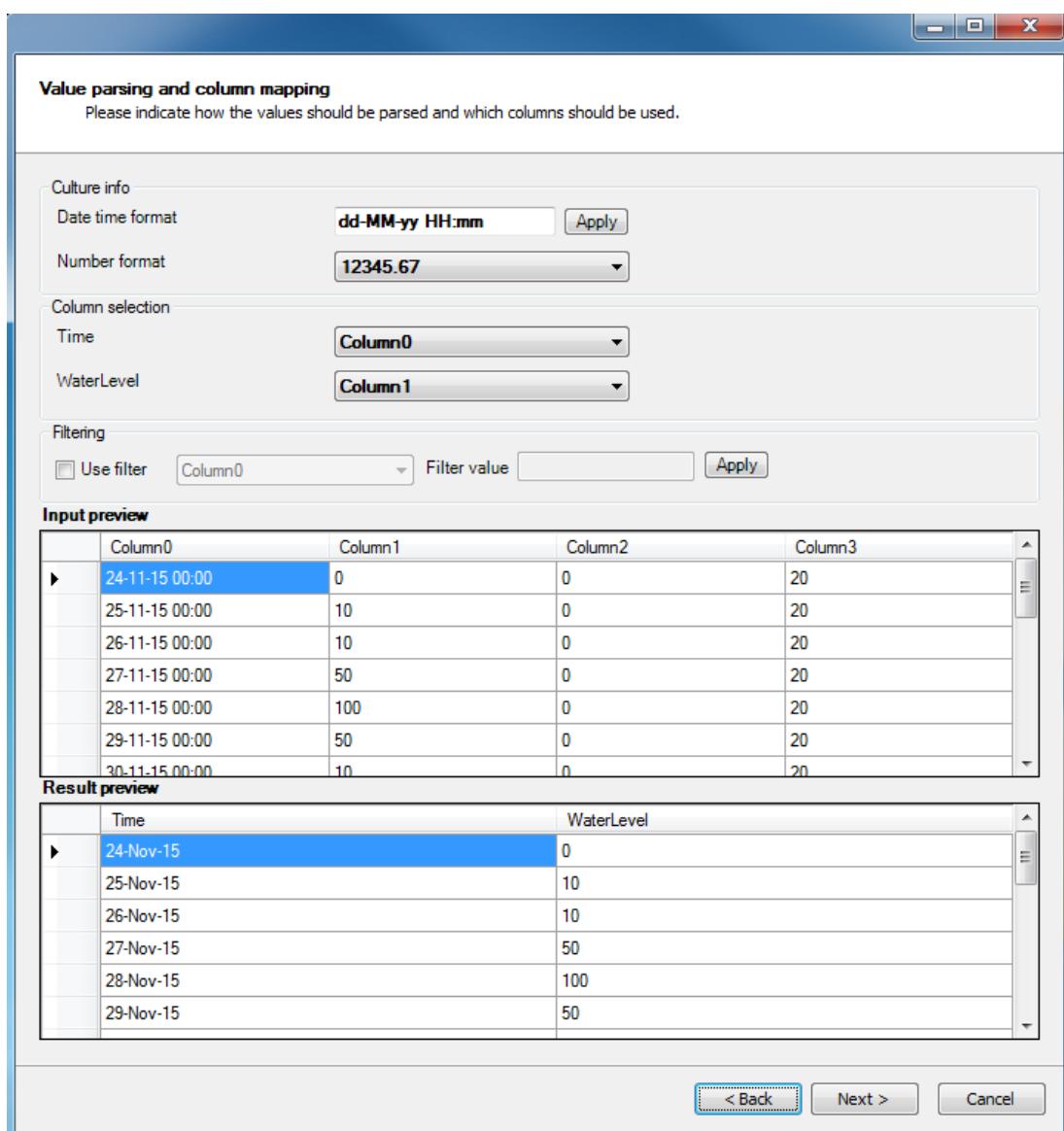


Figure 4.51: Clipboard/csv import wizard: specification of how values should be parsed and columns should be mapped

- ◊ Import from clipboard using the ‘Clipboard import’ button: a wizard will open in which the user can specify (1) how data should be parsed into columns (Figure 4.50) and (2) how the values should be parsed and mapped into columns (Figure 4.51).
- ◊ Import from Web Processing Service (WPS): with this service the user can download boundary forcing data (for now only water level time series) for a selected support point from an online database (Topex/Poseidon 7.2). Upon pressing the button ‘Import from WPS’ a window will pop up as depicted in Figure 4.52. Here, the user can specify the time interval and time step for downloading the data. (Note: Please note that this service is only available with an internet connection!)



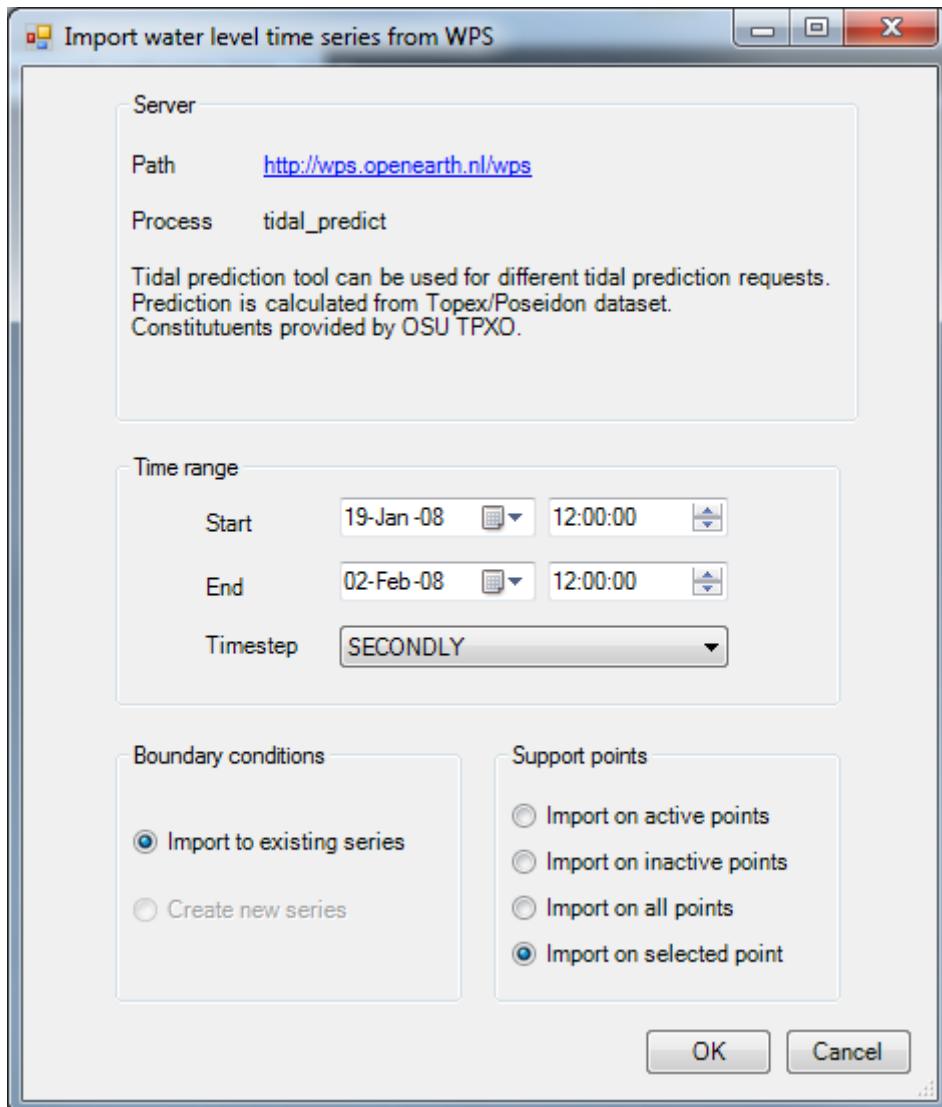


Figure 4.52: Window for entering input to download boundary data from WPS

- ◊ Import from attribute file (*.bc): with this button the user can import data from existing boundary conditions (*.bc) file. The user has three options for importing:
 - Overwrite where matching (replace): only overwrites the forcing data for matching support points in the bc-file and GUI input.
 - Overwrite where missing (extend): only overwrites the forcing data for matching support points in the bc-file and in the GUI input that did not contain data.
 - Overwrite all: overwrites the forcing data for all support points in the GUI (meaning that the forcing data for non-matching support points is emptied).

Harmonic components

The harmonic components are defined by a frequency, amplitude and phase (see [Figure 4.53](#)). By default the forcing data viewer shows the harmonic component for the time frame specified for the model simulation. The options to define the harmonic components are similar to the options for time series:

- ◊ Specify components step by step: the user can add or delete rows with the “plus”- and “minus”-signs below the table.
- ◊ Select (astronomical) components using the ‘Select components’ button ([Figure 4.54](#)): the

user can select astronomical components which will be transformed in the corresponding frequencies.

- ◊ Import from csv using the 'Csv import' button: a wizard will open in which the user can (1) select a csv-file, (2) specify how data should be parsed into columns and (3) how the values should be parsed and mapped into columns.
- ◊ Import from clipboard using the 'Clipboard import' button: a wizard will open in which the user can specify (1) how data should be parsed into columns and (2) how the values should be parsed and mapped into columns.
- ◊ Import from attribute file (*.bc): with this button the user can import data from existing boundary conditions (*.bc) file. The user has three options for importing:
 - Overwrite where matching (replace): only overwrites the forcing data for matching support points in the bc-file and GUI input.
 - Overwrite where missing (extend): only overwrites the forcing data for matching support points in the bc-file and in the GUI input that did not contain data.
 - Overwrite all: overwrites the forcing data for all support points in the GUI (meaning that the forcing data for non-matching support points is emptied).

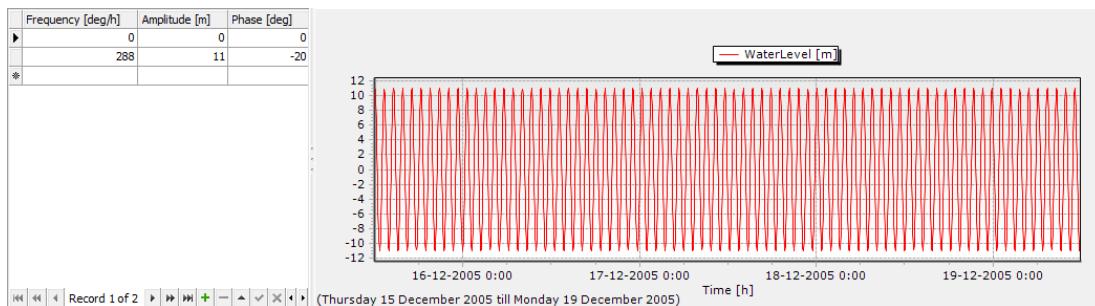


Figure 4.53: Specification of harmonic components in boundary data editor

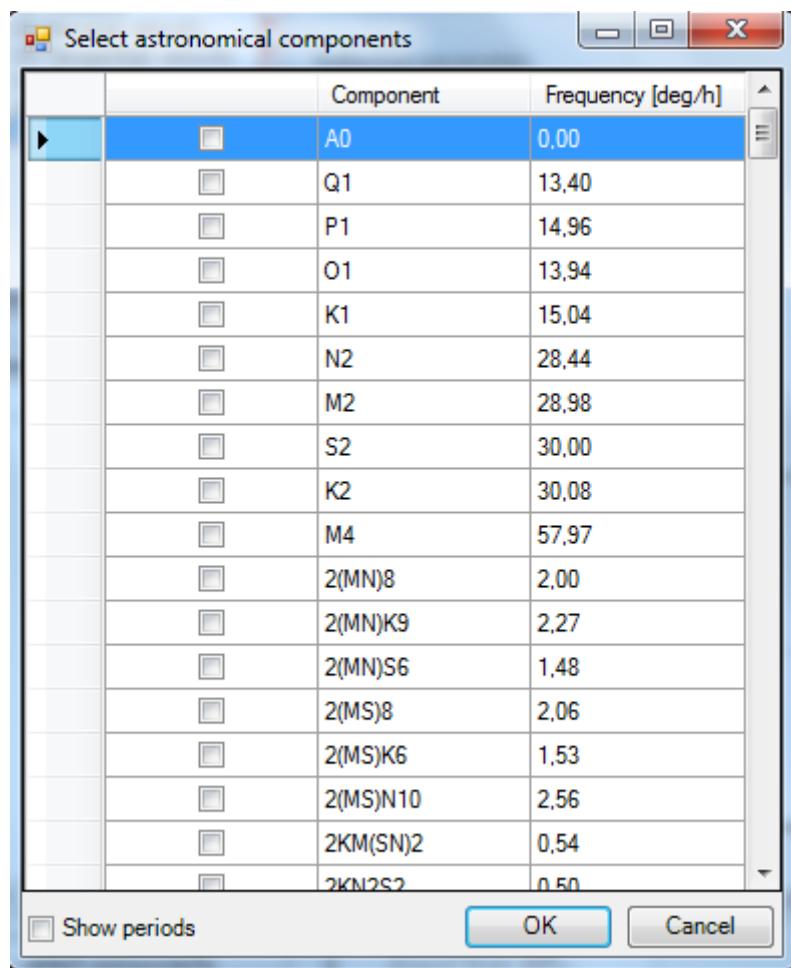


Figure 4.54: Selection of astronomical components from list (after pressing 'select components')

Astronomic components

Astronomical components are similar to harmonic components, with the exception that the frequency is prescribed. Instead of specifying the frequency the user can select astronomical components by name. Upon editing the component field in the table the user will get suggestions for components in a list (Figure 4.55). The most frequently used components (A0, Q1, P1, O1, K1, N2, M2, S2, K2 and M4) are put on top of the list, the other components are listed in alphabetic order. Instead of defining each component individually, the user can also make a selection of components by pressing the button 'select components' (Figure 4.54).

	Component [-]	Amplitude [m]	Phase [deg]
A0			
Q1			
P1			
O1			
K1			
N2			
M2			

Record 1 of 1

◀◀
◀
▶
▶▶
+/-
▲
▼
✓
✗

Figure 4.55: Suggestions for astronomical components in list

Harmonic or astronomic components with corrections

For calibration purposes the user can combine harmonic or astronomic components with corrections. The corrections are defined in terms of an amplitude (multiplication) factor and a phase difference (see [Figure 4.56](#)). This allows the user to keep track of both the original signal and the calibration coefficients. The effects of the corrections on the resulting signal are directly visualized in the forcing data viewer. **Note: Please note that the import functionality is not (yet) working properly for astronomic/harmonic boundary conditions with corrections.**



	Component [-]	Amplitude [m]	Phase [deg]
A0		0.2	0
► M2		1	55
*			

Figure 4.56: Editing harmonic/astronomic components and their corrections



Q-h relation (only for water level)

The user can force the boundary with a Q-h relationship, but only for the quantity water level. This is a relationship between discharge and water level (see [Figure 4.57](#)). The relationship can only be prescribed per polyline, not per support point. (**Note: this functionality has not been tested extensively yet**)

	Q [m^3/s]	h [m]	
	1000	7	
	1500	85	
▶			

Record 3 of 3

Figure 4.57: Specification of a Q-h relationship

Exporting boundary conditions

With the “Export to files” button the user can export all boundary forcing data for the given polyline to a *.bc-file.

4.4.8.2.1 3D boundary conditions

When the model is 3D (i.e. the number of layers is larger than 1), the user can specify 3 dimensional boundary conditions for relevant quantities such as velocity, salinity, temperature and tracer concentration. The user can choose from vertically uniform or vertically varying boundary conditions (Figure 4.58), where the latter are defined as a percentage from the bed. In the boundary condition editor the layer view will appear (Figure 4.59). Here, the user can specify the vertical positions of the boundary conditions and view their position relative to the model layers. Please note that the number and position of vertical boundary conditions does not necessarily have to match the number and/or the exact position of the model layers. The computational core will interpolate the boundary forcing position to the number of model layers.

In case of non-uniform boundary conditions over the vertical, the number of columns in the boundary forcing data editor will increase correspondingly (see Figure 4.60). In this way the user can specify the conditions for all vertical positions in the same table and view the resulting signals in the forcing data viewer.

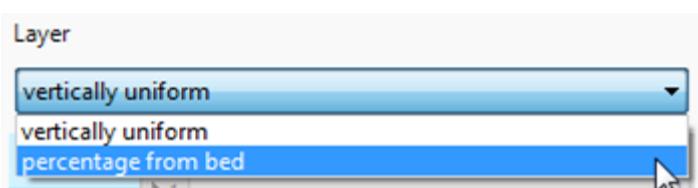


Figure 4.58: Selection of vertically uniform or varying boundary conditions in case of a 3D model

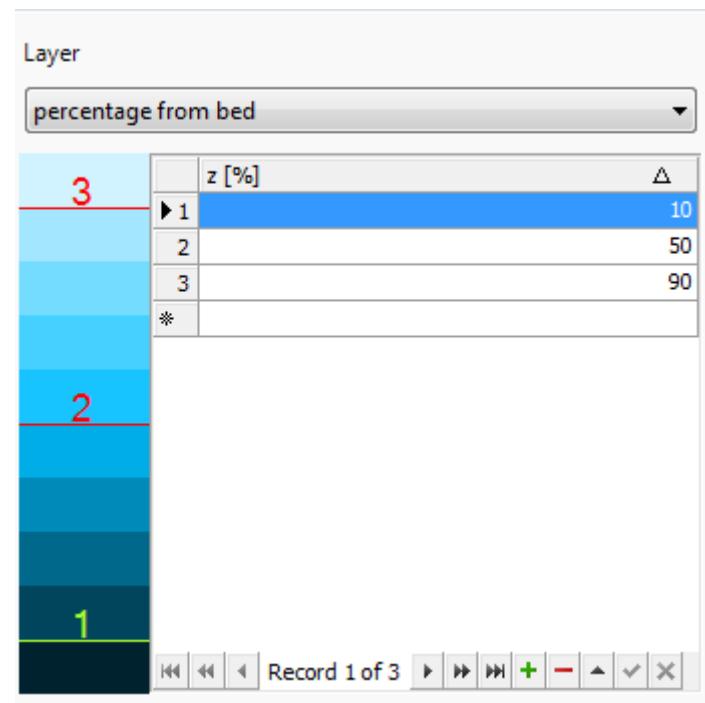


Figure 4.59: Overview of the layer view component of the boudary conditions editor. In the table the user can edit the vertical positions of the boundary conditions as a percentage from the bed. In the view left of the table, the user can see the vertical positions of the boundary conditions (indicated by number corresponding to the table) relative to the model layers.

	Time [-]	Salinity(1) [ppt]	Salinity(2) [ppt]	Salinity(3) [ppt]
	2014-04-11 12:00:00	30	25	20
	2014-04-25 12:00:00	30	25	20
▶				

Figure 4.60: Specification of boundary forcing data (in this example for salinity) at 3 positions in the vertical

View boundary data

All boundary data of the same quantity on a support point can be (pre-)viewed in the boundary data view in the lower-right panel. If multiple signals of the same quantity have been entered, the viewer will show the active signal in red and the total signal of all datasets in grey (see Figure 4.61). In the view the user can zoom-in by dragging a box from top-left to bottom-right and zoom-out by dragging a box from bottom-right to top-left.

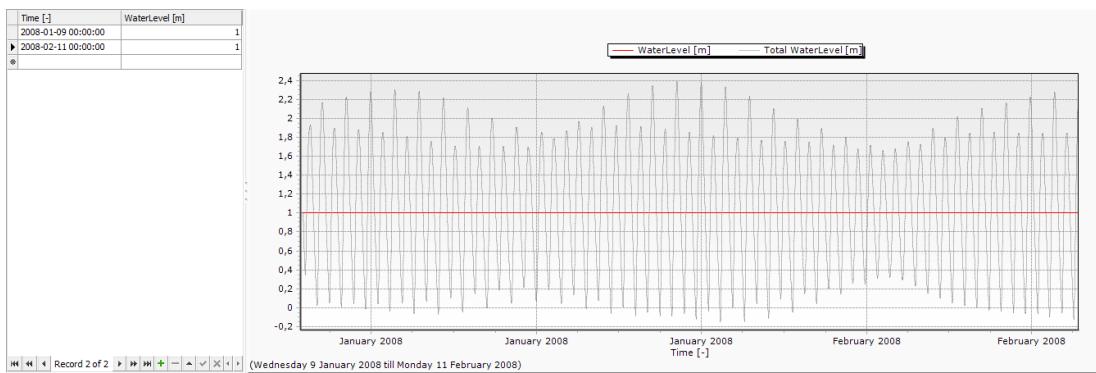


Figure 4.61: Example of active and total signal for multiple water level data series on one support point

To inspect multiple quantities at a support point at the same time (for example water level and salinity) the user can use the combined boundary data viewer by pressing the button 'combined BC view'. **Note: This does not work properly yet.**



4.4.8.3 Import/export boundary conditions from the project tree

Apart from import and export functionality per individual boundary polyline in the boundary condition editor, the GUI offers the opportunity to import and export boundary locations (*.pli) and forcing (*.bc) on a higher level. Hereto, you have to click the RMB on "Boundary Conditions" in the project tree and select "import" or "export" (Figure 4.62). Imports and exports on "Boundary Conditions" apply to all the boundary conditions whereas import and exports on a boundary polyline apply only to that boundary condition.

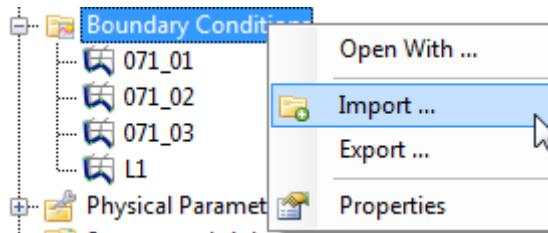


Figure 4.62: Importing or exporting boundary features - both polylines (*.pli) and forcing (*.bc) - from the project tree using the RMB

Import and export polylines

Upon importing a *.pli file with the same filename and the same polyline name(s) as the existing polyline names in the GUI, the existing polyline(s) will be replaced and all forcing data thereon will be deleted. Upon importing a polyline(s) with a different name(s), the polyline(s) will be added to the project tree without any forcing data on it/them. The user will be asked to import the data "as is" or to perform a coordinate transformation before the import (see Figure 4.63).

Alternatively, the user can export created polylines to a *.pli-file. Upon export the user will be asked to export the data "as is" or to perform a coordinate transformation before the export (see Figure 4.63).

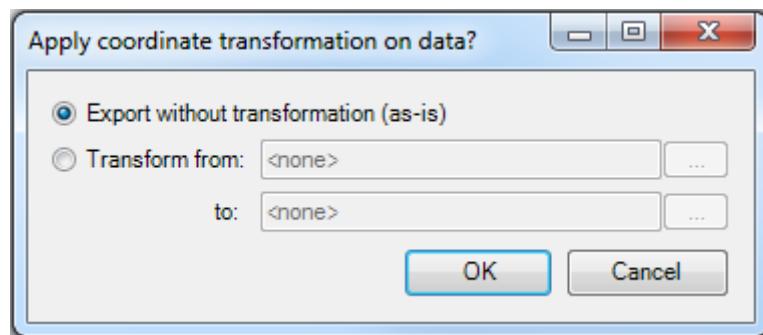


Figure 4.63: Import or export a *.pli file as is or with coordinate transformation.

Import and export boundary forcing data

Similar to the polylines, you can import and export boundary forcing data from/to a *.bc-file. To import forcing data the existence of a polyline with at least one matching support point is a prerequisite. Upon importing *.bc data you can select which quantities and forcing types from the *.bc-file should be imported and with which overwrite options (see Figure 4.64). Similarly, you can export boundary forcing data. As an additional exporting feature you can select whether you would like to export: (1) all forcing data into one file, (2) as separate files per boundary, (3) as separate files per process or (4) as separate files per quantity (see Figure 4.65).

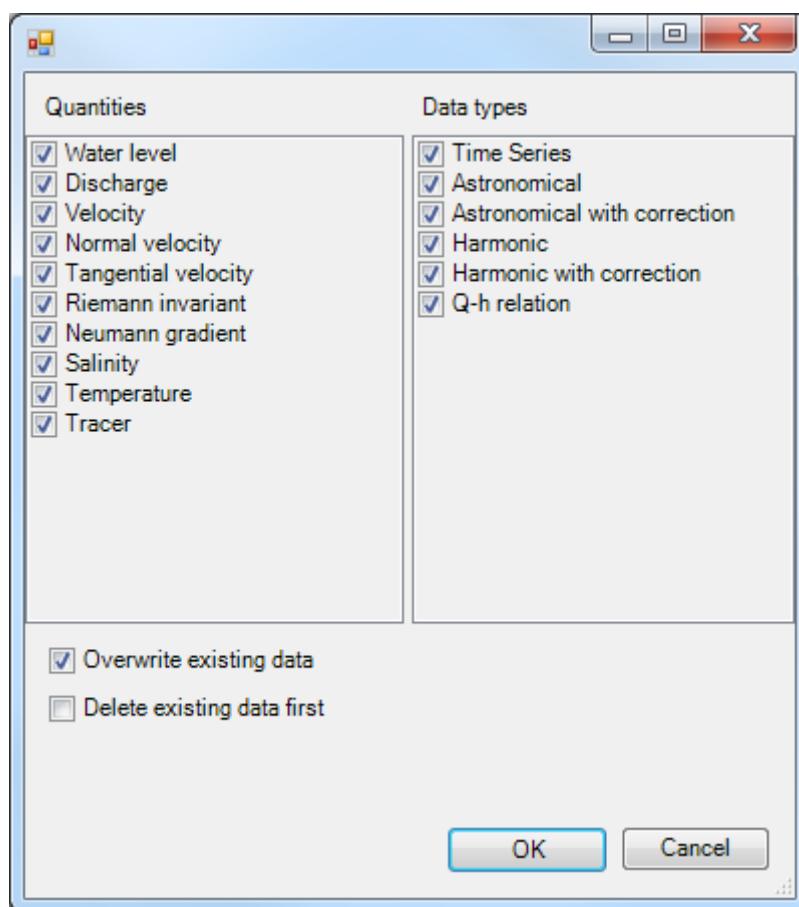


Figure 4.64: Import or export a *.pli file as is or with coordinate transformation.

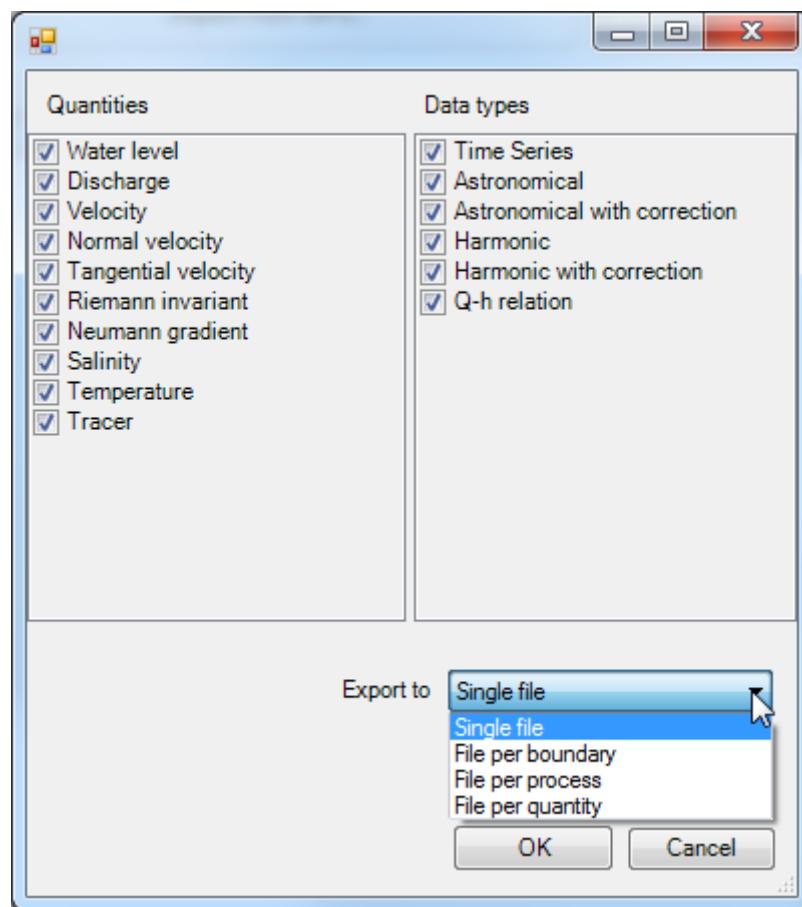


Figure 4.65: Import or export a *.pli file as is or with coordinate transformation.

4.4.8.4 Overview of boundary conditions in attribute table (non-editable)

The attribute table of the boundary conditions gives an overview of all specified boundary polylines and corresponding forcing (see [Figure 4.66](#)). This attribute table can be opened by double clicking 'Boundary Conditions' from the project or map tree. Most of the features in the attribute table are non-editable, except for the optional (multiplication) factor and offset per quantity per polyline. With these settings you can integrally multiply all data on a polyline with a factor and/or add an offset to it.

Boundary Conditions X					
Boundary	Quantity	Forcing type	Factor	Offset	
► 071_01	WaterLevel	Time Series	1	0	
071_01	Salinity	Time Series	1	0	
071_02	WaterLevel	Time Series	1	0	
071_02	Salinity	Time Series	1	0	
071_03	WaterLevel	Harmonic	1	0	
071_03	Salinity	Time Series	1	0	
L1	Discharge	Time Series	1	0	
L1	Salinity	Time Series	1	0	

Figure 4.66: Overview of all boundary conditions in attribute table

4.4.9 Physical parameters

The physical parameters attribute is used to set all physical parameters of your model. When it is expanded in the project tree, it shows three attributes; roughness, viscosity and wind.

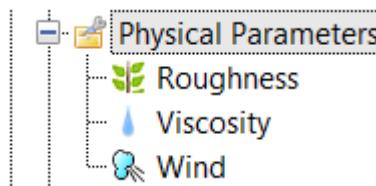


Figure 4.67: The physical parameters in the project tree

4.4.9.1 Constants

In the *Physical Parameters* tab, under *Constants*, the user can adjust the value of *Gravity* (m/s^{**2}) and *Default water density* ($\text{g/m}^{**3}/1000$).

4.4.9.2 Roughness

Bed roughness can be specified as a uniform value or as a coverage (e.g. a spatially varying field). The uniform values as well as the roughness formulation (i.e. Chezy, Manning, WhiteColeBrook or Z0) can be edited in the ‘Physical Parameters’ tab, which opens upon double clicking ‘Roughness’ in the project tree (Figure 4.68). **Note:** The latter is not yet working. In this tab you can also specify the linear friction coefficient, linear friction Umod, wall behaviour (free slip or partial slip) and wall ks for partial slip.



In case of spatially varying bed roughness you can double click the quantity in the project tree or select it from the dropdown box in the *Spatial Operations* ribbon (Figure 4.69). Then the spatial editor is activated, which you can use to edit spatially varying fields. For more information on how to use the spatial editor you are referred to Appendix F.

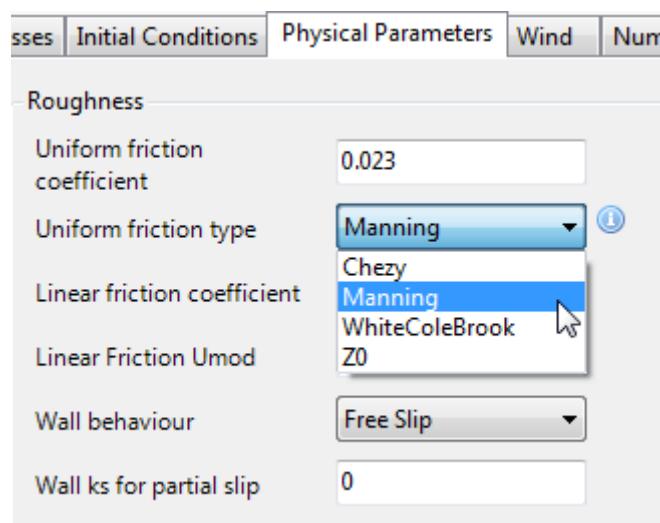


Figure 4.68: The section of the ‘Physical Parameters’ tab where you can specify roughness related parameters and formulations.

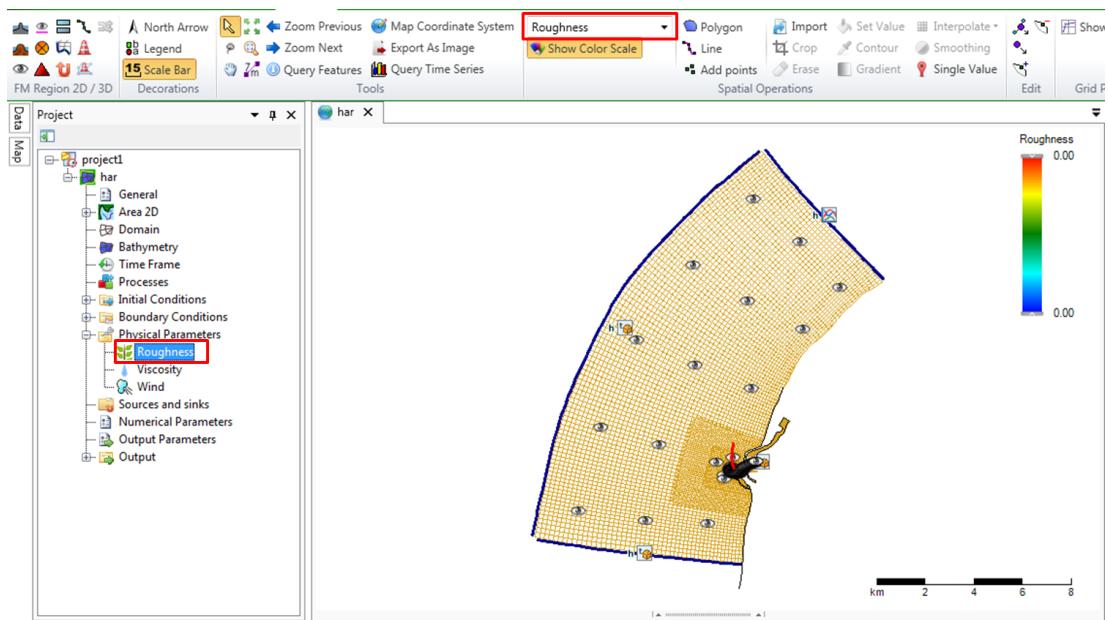


Figure 4.69: Roughness activated in the spatial editor to create/edit a spatially varying field

4.4.9.3 Viscosity

The eddy viscosity can be specified as a uniform value or as a coverage (e.g. a spatially varying field). In the ‘Physical parameters’ tab you can specify the uniform values for the horizontal and vertical (in case of 3D simulations) eddy viscosity and diffusivity (Figure 4.70).

In case of spatially varying viscosity you can double click the quantity in the project tree or select it from the dropdown box in the *Spatial Operations* ribbon (Figure 4.71). Then the spatial editor is activated, which you can use to edit spatially varying fields. For more information on how to use the spatial editor you are referred to Appendix F.

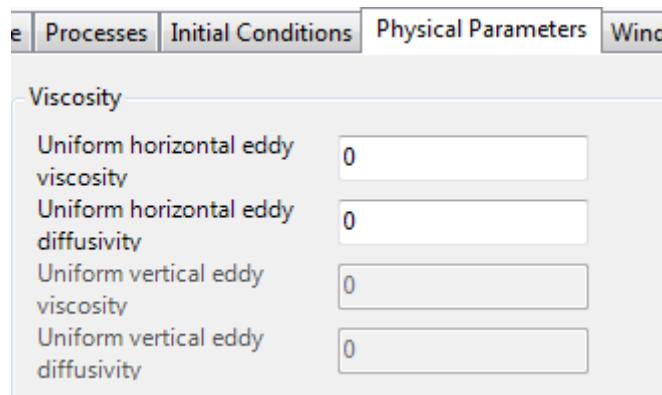


Figure 4.70: The section of the ‘Physical Parameters’ tab where you can specify (uniform) values for the horizontal and vertical eddy viscosity and diffusivity.

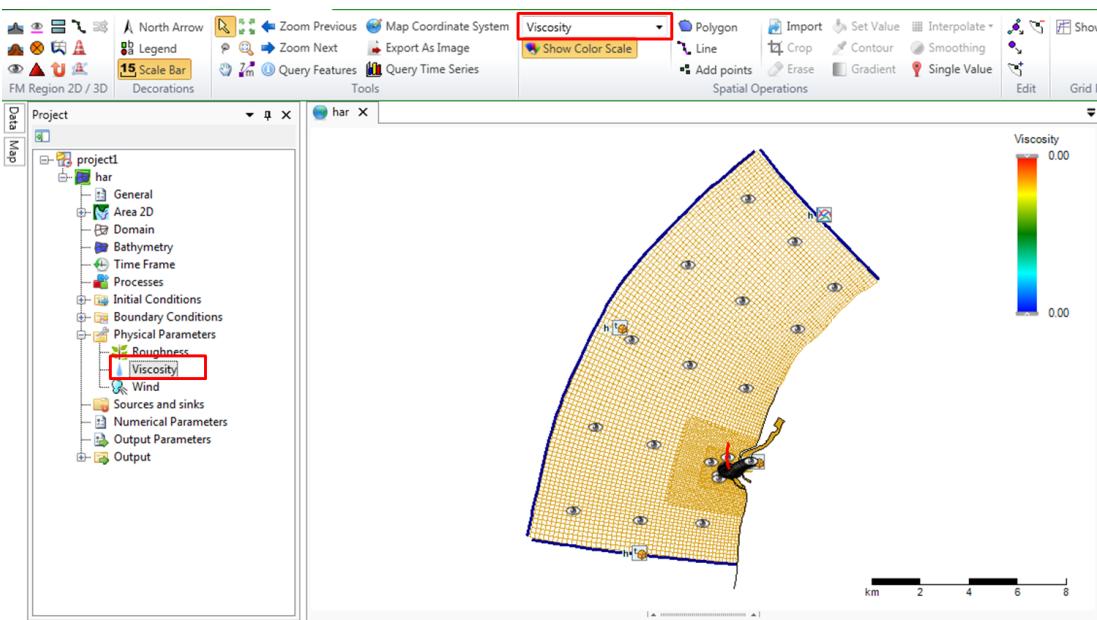


Figure 4.71: Viscosity activated in the spatial editor to create/edit a spatially varying field

4.4.9.4 Wind

All relevant parameters, described in [Chapter 11](#) can be adjusted in the following sub-tab.

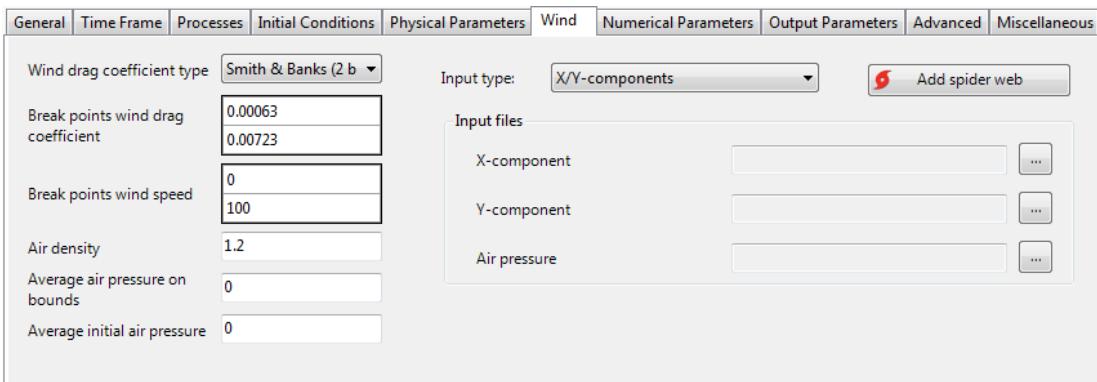


Figure 4.72: Overview of parameters in sub-tab Wind

4.4.9.5 Heat Flux model

The Heat flux model, described in [Chapter 10](#), needs only specification of which model to use. See the drop-down selection in [Figure 4.33](#).

4.4.9.6 Tidal forces

The Tide generating forces, described in [Section 7.10](#), can be enabled in the *Processes* tab, see [Figure 4.33](#).

4.4.10 Sources and sinks

Sources and sinks (or: intake/outfall facilities) can be used to add/extract a discharge to/from the model or to redistribute water and constituents (such as temperature and salinity) within the model. Sources and sinks consists of a location (defined by a *.pli-file) and time series

describing the discharges (defined by a *.tim-file). All the hydrodynamical considerations behind sources and sinks are discussed in [Section 7.8](#)?

Sources and sinks locations

Sources and sinks can be added to the model using the corresponding icon from the ‘Map’ ribbon ([Figure 4.73](#)). When the sources/sinks icon is active you can add them as polyline elements in the central map using the LMB.

-  Each polyline element is closed by double clicking the LMB. **Note:** Please note that the length of the polyline elements is not taken into account in the handling of sources and sinks (e.g. it is modeled as an instant redistribution of water and constituents without delays and friction losses). Polyline elements starting outside the model domain and going inward are sources and, vice versa, polyline elements starting inside the model domain and going outward are sinks. Polyline elements starting and ending within the model are intake/outfall type of discharges. The drawing direction determines the direction of the discharge indicated by an arrow ([Figure 4.74](#)). In case of a source the direction of the last polyline element determines the direction of flow momentum into the model. **Note:** The ‘reverse’ line option - to switch the discharge direction without having to redraw the source/sink - is yet to be implemented.
- 

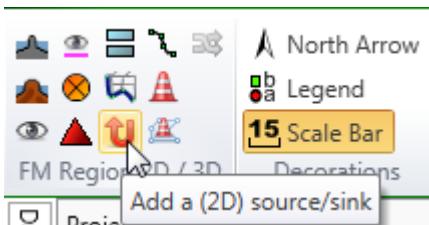


Figure 4.73: Activate the sources and sinks editing icon in the ‘Map’ ribbon

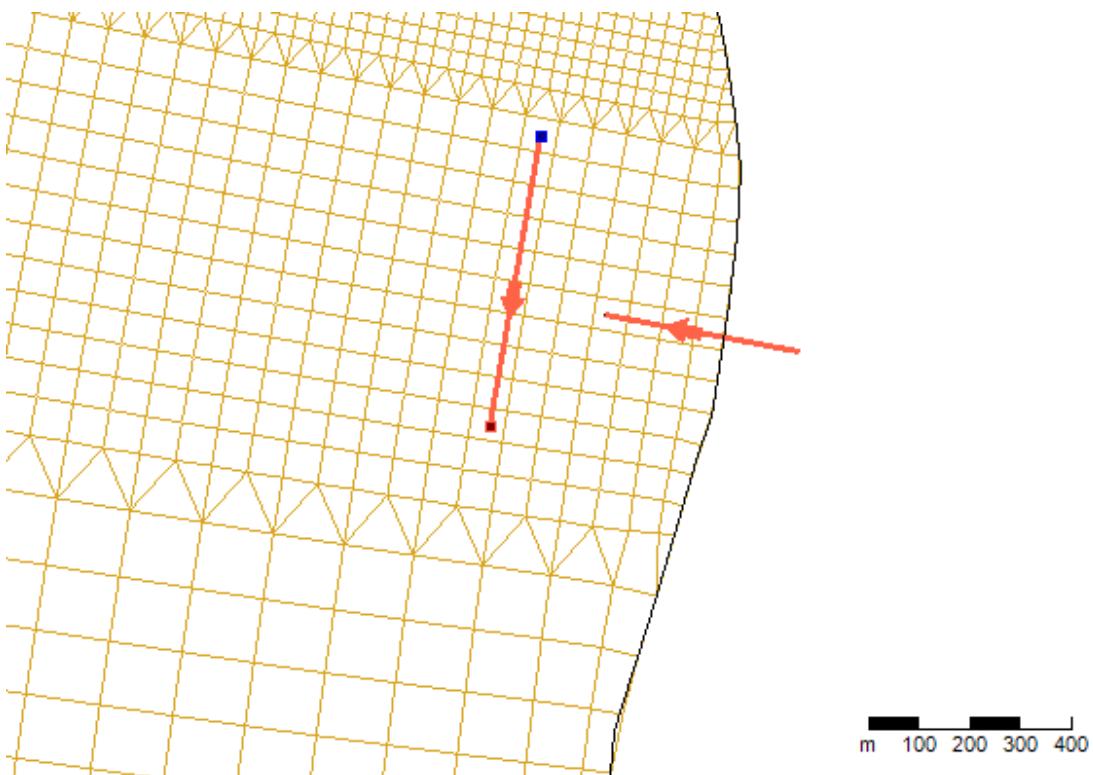


Figure 4.74: Add sources and sinks in the central map using the ‘Sources and sinks’ icon.

Sources and sinks time series

After drawing the sources/sinks locations in the central map, they appear under ‘Sources and sinks’ in the project tree (Figure 4.75). Either by double clicking the sources/sinks in the project tree or the line element in the central map, you can open the sources and sinks editor (Figure 4.76). In this editor you can specify the discharge time series as well as the corresponding constituents (for example salinity and temperature, depending on the active physical processes). The time series of water discharges are always defined as absolute values. For the time series of constituents the following applies:

- ◊ For sources the constituent time series are absolute values
- ◊ For sinks the constituent time series are determined by the modeled values
- ◊ For sources and sinks (intake/outfall relationships) the constituent time series are the excess values (e.g. on top of the modeled values)

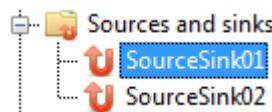


Figure 4.75: Sources and sinks appearing in the project tree

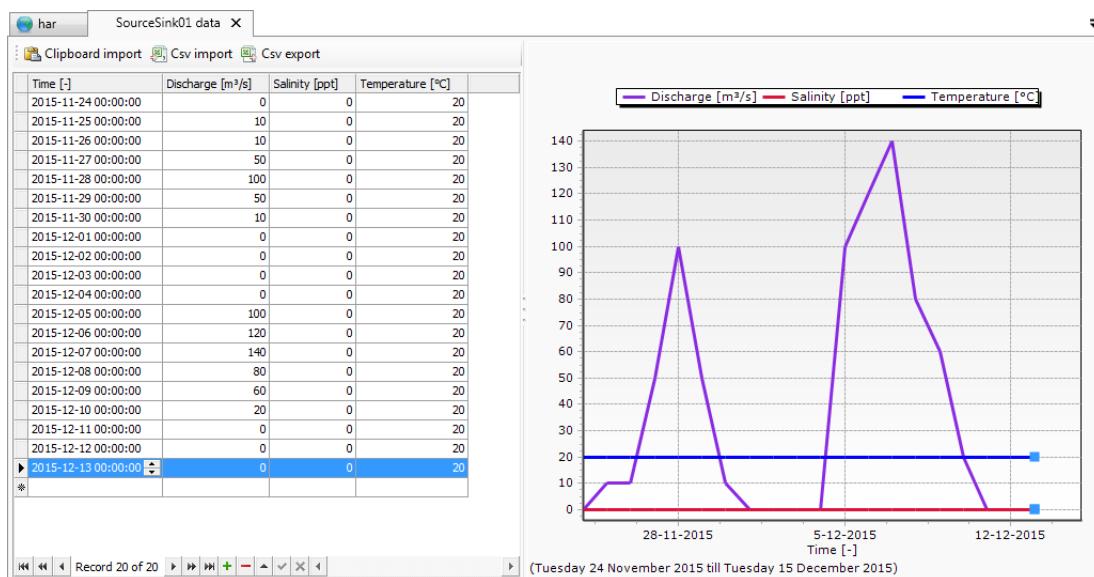


Figure 4.76: Specifying time series for sources and sinks in the sources and sinks editor

4.4.11 Numerical parameters

In the numerical parameters tab, all numerical parameters related to your computation can be set. The parameters that can be set are described in Table 4.1.

Table 4.1: Overview and description of numerical parameters

Parameter	Description
Max Courant number	The size of the time step, Δt , is computed by the computational kernel automatically, each time step by means of the maximum tolerable Courant number. By default, this value is 0.7.
Wave velocity fraction	The wave velocity fraction is related to stability of the computation. By using this fraction, the velocity of the flow is enlarged with the wave velocity times this fraction. The value of this fraction is 0.1 by default.
Advection type	This key depicts the ID of the advection scheme. By default this value is 33.
Water depth limiter type	The limiter type for waterdepth in continuity equation: 0 means no limiter (default), 1 is the minmod method, 2 the Van Leer method, 3 the Koren method and 4 the monotone central method.
Advection velocity limiter type	The limiter type for the cell center advection velocity: 0 means no limiter, 1 the minmod method, 2 the Van Leer method, 3 the Koren method and 4 the monotone central method (default).
Salinity transport limiter type	The limiter type for salinity transport: 0 means no limiter, 1 the minmod method, 2 the Van Leer method, 3 the Koren method and 4 the monotone central method (default).
Solver type	Specification of the Poisson equation type solver for the pressure: 1 = sobekGS_OMP, 2 = sobekGS_OMPthreadsafe, 3 = sobekGS, 4 = sobekGS + Saadilud (default), 5 = parallel/global Saad, 6 = parallel/Petsc, 7 = parallel/GS.
Max degree in Gaussian elimination	The maximum degree in the Gauss elimination, part of the pressure solver. This key has the value 6 by default.
Thin dike scheme	Or: fixed weir scheme. 0: none, 1: compact stencil, 2: whole tile lifted, full subgrid weir + factor.
Thin dike contraction	Or: fixed weir contraction. This is the fixed weir flow width contraction factor, being the flow width = flow width times the fixed weir contraction.
Boundary smoothing time	Fourier smoothing time on waterlevel boundaries (s).
Linear continuity	Default 0; Set to 1 for linearizing $d(H_u)/dx$; link to AdvecType.
Threshold for drop losses	Apply droplosses only if local bed slope is larger than this specific value.
Theta of time integration	Compromise in explicit / implicit time integration.
Downwind cell H on Q boundaries	Specifies the way of the upstream discharge boundary: 0 is original hu on qbnd, 1 is downwind hs on qbnd

4.4.12 Output parameters

Model runs can produce various types of output files. The map tree shows the two most-used types: map output and his(tory) output (Figure 4.77).

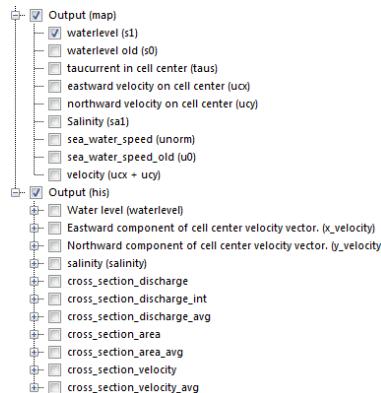


Figure 4.77: Output parameters tab

The history file contains output on specific locations: time series data on observation points (Section 4.4.2.2), cross-sections (Section 4.4.2.3) and structures (Sections 4.4.2.9 – 4.4.2.11). The map file contains flow quantities on the entire grid at specified time intervals, and can later be used for 2D and 3D visualizations of entire flow fields. The map file can typically turn out much larger than the history file, and it is therefore advised to use larger time intervals for map files than for his files.

Additionally, three more types of output can be requested: restart files, WAQ output, and timing statistics. Restart files are a special type of map file that can later be used as initial states in other runs. Restart files contain several additional flow quantities and are written at specified intervals into one file per each restart time. Typically, one selects a large restart interval in order not to waste disk space. WAQ output files are written by D-Flow FM and are intended to be used as input files to subsequent D-Water Quality runs. More details on water quality modelling can be found in Chapter 16. Timing statistics can be produced both in the diagnostics file (via *Statistics output interval*, for viewing basic simulation progress), and in a separate detailed timings file (via *Timing statistics output interval*, for detailed performance analysis).

When double clicking the output parameters tab in the project tree, the output parameters sub-tab of the settings tab is highlighted in the central map. All parameters related to the output of your model run are specified here (Figure 4.78). The most common output parameters to set are the parameters related to the history files, map files and restart files.

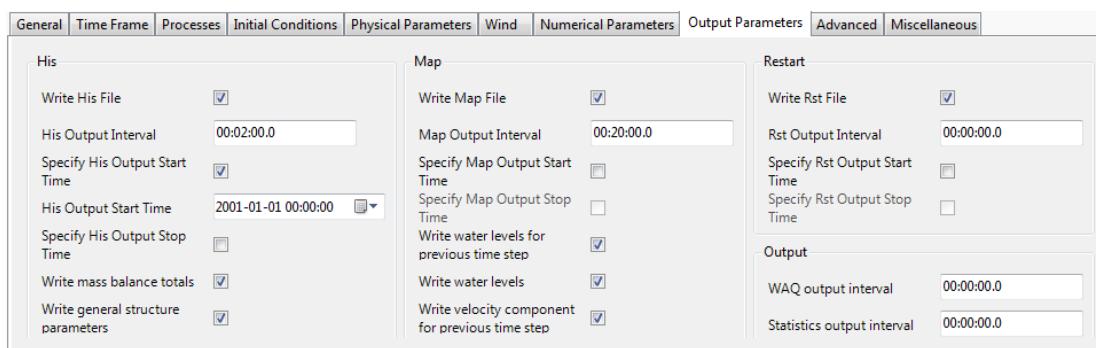


Figure 4.78: Overview output parameters tab

For each of the three files, his, map and restart, the input parameters are specified in the same manner (Figure 4.78). Taking his output as example: when *Write His File* is checked, the output of history file is enabled. *His Output Interval* determines the interval at which the output data is stored in the file; the smaller the interval, the more detailed the output and the larger the file. By default, history output is written from the start until the end of the simulation. Optionally, output can be restricted to a certain time window: to specify different output start and/or stop times, check the box next to *Specify His Output Start Time* and/or *Specify His Output Stop Time* and enter the desired times next to the parameters *His Output Start Time* and *His Output Stop Time*.

Below, the various output options are described in greater detail.

Write his file

<i>His Output Interval</i>	Time interval for history time series.
<i>His Output Start/Stop time</i>	Restrict history output to a specified time window.
<i>Write mass balance totals</i>	Enable detailed mass balance time series output in the his file.
<i>Write (misc.) structure parameters</i>	Enable time series output across general structures, pumps, weirs and gates in the his file.

Write map file

<i>Map Output Interval</i>	Time interval for map field time series.
<i>Map Output Start/Stop time</i>	Restrict map output to a specified time window.
<i>Specific Map Output Times</i>	File containing specific time values at which to produce additional map output snapshots.
<i>Write water levels, etc.</i>	Several optionals for enabling/disabling certain quantities output in the map file.

Write restart file

<i>Restart interval</i>	Time interval for restart files.
<i>Rst Output Start/Stop time</i>	Restrict restart output to a specified time window.

Other output options

<i>WAQ Output Interval</i>	Time interval for D-Water Quality files in <DFM_DELWAQ_mdident*.hyd>, etc.
----------------------------	---

Table 4.2: Overview and description miscellaneous parameters

Parameter	Description
Time step type	Leave at default.
Turbulence model	See chapter 9 .
Turbulence advection	Leave at default.
Water level threshold	Max allowed water level difference between old and new time step in any cell. Run will abort if exceeded. (0 means disabled)
Velocity threshold	Max allowed velocity difference between old and new time step in any cell. Run will abort if exceeded. (0 means disabled)
Dry cell threshold	Flooding threshold at velocity points. Used in wetting and drying.
<i>Simulation statistics output interval</i>	Interval for simulation progress output (on standard out and diagnostics file).
<i>Timing statistics output interval</i>	Interval for detailed timings output into <mdident_timings.txt> for expert performance analysis.

4.4.13 Miscellaneous

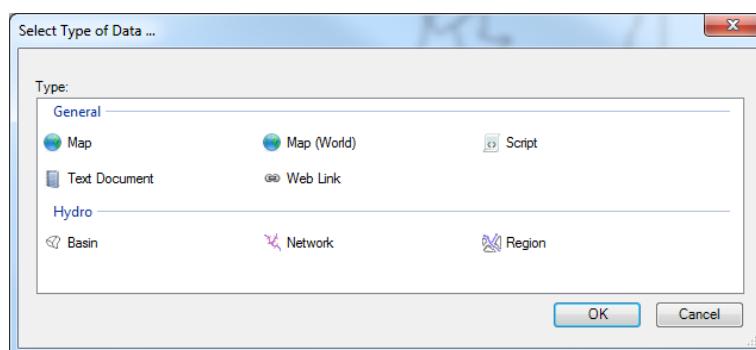
Within the miscellaneous sub-tab, various parameters in relation to waves and equatorial settings can be adjusted. [Table 4.2](#) gives an overview and description of these parameters.

4.5 Save project, MDU file and attribute files

To save your Delta Shell project, navigate the menu ribbons to “File” and click “Save as”. Choose a location, specify a name and click “Save”. Your project will now be saved in a folder called #yourname#.dsproj_data and a #yourname#.dsproj file is written. Within this folder you will find all input ASCII input files of your model, output files of your model (if the model was run using the GUI) and zip folders containing your restart files. Be aware that the output files are stored within a separate folder in which the input files of your model are stored. The output folder on the same level as the folder containing model input files is empty.

To open a project, navigate the menu ribbons to “File” and click “Open”. Select the *.dsproj file of choice and click “Open”.

Importing model or data within a Delta Shell project can be achieved in two ways. Navigate the menu ribbons to “File” and click “Import”. The import wizard appears, allowing you to select what you want to import ([Figure 4.79](#)).

**Figure 4.79:** Model/data import wizard

Alternatively, you can also right mouse click on the name of your project in the project tree and select “Import”.

Exporting your model can be achieved in the same fashion as importing your model. All model input files will be written to the folder you select. Be aware that model files exported to a folder in which other model files with the same names are present will be overwritten.

5 Running a model

5.1 Running a simulation

After defining the input for the D-Flow FM hydrodynamic simulation, the computation can be executed either via Delta Shell or using batch scripts. Via Delta Shell, the status of the computation and possible messages are displayed in a separate window. When using a batch script, all messages are written to the diagnostics file ([section E.1](#)) and you can continue working in the current window. Not all functionality is available when using Delta Shell to start a calculation. Use a batch script (see [section 5.4](#)) in the following cases:

- 1 Using MPI to run in parallel
- 2 Using some queueing mechanism on a cluster
- 3 Running some unattended simulations, while continuing to work in Delta Shell.

Note that currently we have two different names of the D-Flow FM executables, for Windows `dflowfm-cli.exe` and for Linux `dflowfm`.

5.2 Parallel calculations using MPI

5.2.1 Introduction

This section describes parallel computing with D-Flow FM based on the *Message Passing Interface* system (MPI). This can be run both on computing clusters with distributed memory as well as shared memory machines with multiple processors and/or multiple CPU cores. The goal of parallelization of D-Flow FM is twofold. We aim for much faster computations on shared- or distributed-memory machines and the ability to model problems that do not fit on a single machine. A less powerful, yet possibly attractive performance improvement is offered by D-Flow FM's OpenMP-based parallelization ([section 5.5.1](#)).

Technical backgrounds on the parallel algorithms in D-Flow FM are described in the Technical Reference Manual [D-Flow FM Technical Reference Manual \(2015\)](#).

Workflow of a parallel run

A parallel run divides the work between multiple processes. To this end we partition the model and let separate processes solve the submodels and generate partitioned output. Given a whole model, the workflow is as follows:

- 1 partition the model (mesh and model definition file),
- 2 submit the parallel job to a queue on a computing cluster, and
- 3 visualize the results from partitioned output files.

5.2.2 Partitioning the model

In D-Flow FM a model is defined by the model definition file, the mesh file and external forcing/boundary condition files, et cetera. The latter are shared by all submodels and do *not* need to be partitioned, they should only be available to all processes. So, partitioning the model concerns partitioning of:

- ◊ the mesh. This is achieved through the graphical user interface or by a command line option. Mesh files for every subdomain will be created, as well as one so-called partitioning polygon file that contains the subdomain bounding polygons;
- ◊ the model definition file. The partitioned model definition files will contain references to the subdomain mesh and the partitioning polygon file, all other information equals its sequential counterpart. The model definition files are partitioned with a script as will be explained later.

5.2.2.1 Partitioning the mesh

The mesh can be automatically partitioned with the METIS software package, see the Technical Reference Manual [D-Flow FM Technical Reference Manual \(2015\)](#) and references mentioned therein, or manually by supplying polygons that define the subdomains.

The mesh partitioning is available through Delta Shell or on the command line. The METIS partitioner or, alternatively, the user-supplied polygons, produce a cell coloring of the unpartitioned mesh. The coloring is used to decompose the mesh into subdomain meshes, which are augmented with ghost cells and saved to file. The coloring itself is not used directly in the parallel computations. Instead subdomain bounding polygons are used that are saved to file while partitioning the mesh. There exists only one such polygon file and we will refer to it as the *partitioning polygon* file. It is read by all parallel processes during the initialization of the (parallel) computations.

The partitioning of the mesh will thus generate:

- ◊ subdomain mesh files, e.g. <example_NNNN_net.nc>, and
- ◊ the partitioning polygon file, e.g. <example_part.pol>.

Partitioning the mesh with METIS via Delta Shell

We will firstly focus on the METIS partitioner. Partitioning the mesh from within Delta Shell is achieved by the following steps. In the project tree, select the model you want to run by means of clicking on the desired model ([Figure 6.1](#)). A right-mouse click will open the context menu, then select *Export....* In the window **Select Type of Data...**, choose *Partition exporter* and the partitioning dialog will appear ([Figure 5.1](#)).

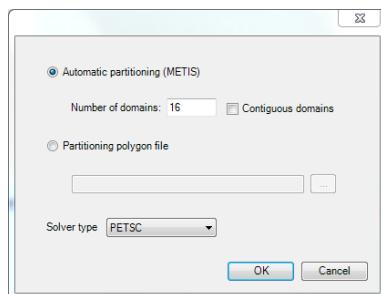


Figure 5.1: Partitioning exporter dialog

Enter the desired amount of subdomains, and typically leave the *contiguous* option switched off and the solver type at its default. After pressing *OK*, a file dialog will appear. Enter the name of the MDU file, *without* any trailing '_000x' partition numbers: these will be added automatically.

Partitioning the mesh manually

Alternatively, the mesh can be partitioned manually with user-supplied partitioning polygons. The partitioning obeys the following rules:

- ◊ if the polygons have a *z*-value specified, it is considered a subdomain number,
- ◊ if the polygons have no *z*-value specified, its order determines the corresponding subdomain number,
- ◊ if a cell is not inside at least one polygon, it is assigned to subdomain 0,
- ◊ if a cell is inside only one polygon, it is assigned to the subdomain defined by that polygon,
- ◊ if a cell is inside more than one polygon, it is assigned to the subdomain with the highest number.

In other words, the polygons may be overlapping and the largest subdomain number is taken in the overlapping regions. If the polygons have no *z*-value, the polygon order determines the corresponding subdomain number, i.e. the first polygon corresponds to subdomain 1 et cetera and there is no polygon defining subdomain 0.

The resulting polygon file may be used from the command line (discussed hereafter), or may be selected in the Delta Shell partitioning dialog ([Figure 5.1](#)).

Partitioning the mesh from the command line

Apart from using the graphical user interface, it is possible to perform the partitioning on the command line:

```
> dflowfm --partition[:ndomains=n[:contiguous=j]] <meshfile> [polygonfile]
```

where *n* is the number of subdomains and *j* is 1 to enforce contiguous subdomains or 0 otherwise (default). If the number of subdomains is omitted, the polygons in the polygon file are used for manual partitioning. For the basename of the partitioning files the input mesh filename is used. For example, if we want to decompose the meshfile <example_net.nc> into 8 subdomain meshes and do not need contiguous domains:

```
> dflowfm --partition:ndomains=8 example_net.nc
```

This will generate:

```
example_part.pol      example_0002_net.nc  example_0005_net.nc
example_0000_net.nc  example_0003_net.nc  example_0006_net.nc
example_0001_net.nc  example_0004_net.nc  example_0007_net.nc
```

The command to partition manually, based on a user-specified polygon file <userpols.pol>, is

```
> dflowfm --partition example_net.nc userpols.pol
```

This will generate files with the same names as before.

5.2.3 Partitioning the MDU file

Having partitioned the mesh, the model definition file needs to be partitioned, as every sub-model requires its own definition file with references to

- ◊ the partitioned mesh file, e.g. <example_0000_net.nc>,
- ◊ the partitioning polygon file, e.g. <example_part.pol>,
- ◊ an appropriate parallel Krylov solver, can be
 - 6: PETSc solver, recommended, or
 - 7: parallel CG with MILU block preconditioning,
- ◊ a unique snapshot directory, and
- ◊ optionally, a partitioned restart file.

The `generate_parallel_mdu.sh` script partitions a sequential model definition file automatically:

```
> generate_parallel_mdu.sh <mdu-file> <nprocs> <partpol-file> <Icgsolver>
```

with

`mdu-file` sequential model definition file,
`nprocs` number of subdomains/parallel processes,
`partpol-file` partitioning polygon file,
`Icgsolver` parallel Krylov solver, can be 6 or 7.

Note that the partitioned mesh filenames are derived from the mesh filename specified in the sequential model definition file. If the sequential model is never run, the sequential meshfile itself does not necessarily has to exist.

For example, if in the sequential mdu-file <example.mdu> contains a reference to meshfile <example_net.nc>, then the command

```
> generate_parallel_mdu.sh example.mdu 8 example_part.pol 6
```

will generate eight partitioned mdu-files, namely <example_0000.mdu> to <example_0007.mdu> and the different entries in e.g. <example_0000.mdu> with respect to the sequential model definition file are

```
[geometry]
NetFile      = example_0000_net.nc
PartitionFile = example_part.pol

[numerics]
Icgsolver    = 6

[output]
SnapshotDir  = snapshots_0000
```

5.2.3.1 Remaining model input

A parallel run of a D-Flow FM model needs only partitioned <.mdu> and <.net.nc> files and a partitioning polygon file <.part.pol>. All other model input is the same as for a standalone run, e.g., meteo forcings, boundary conditions, observation stations and more. These are generally copied to the working directory by the parallel job submission script.

5.2.4 Running a parallel job

To run a parallel job with D-Flow FM model on a cluster you have to prepare the submission script. The submission script should be prepared with respect to the specific options that job scheduler on your cluster requires.

The simple example of the D-Flow FM submission script on the cluster with the Grid Engine:

```
#!/bin/bash
#$ -V
#$ -q test
#$ -cwd
#$ -N My_DflowFM_JOB
#$ -m bea
#$ -M my.email@provider.net

export LD_LIBRARY_PATH=$DFLOWFM/lib:$LD_LIBRARY_PATH
export PATH=$DFLOWFM/bin:$PATH

mpixexec -np 4 dflowfm --autostartstop YOUR_MDU_FILE.mdu > out.txt 2> err.txt
```

In this simplified example above we submit the D-Flow FM simulation that was partitioned into 4 domains and is going to use only 1 node. The options used above are:

- ◊ -V Specify that all environment variables active within the qsub utility be exported to the context of the job.
- ◊ -q Specify the queue 'test' to be used for this job, if absent default queue is used.
- ◊ -cwd Execute the job from the current working directory.
- ◊ -N Specify the name of the job.
- ◊ -m Specifies which message type should be emailed (b=beginning of job, e=end of job, a=abort of job).
- ◊ -M Specifies the email address to send the notification.

In order to submit more complicated, e.g. multi-node simulations, additional options that are scheduler depended have to be added.

5.2.5 Visualizing the results of a parallel run

The map and history output files (as introduced in [section 4.4.12](#)) deserve special attention in parallel runs.

The history file — with time series for observation points, structures and more — is written only by process #0, and all model-global data has already been aggregated into that single file: <mdident_0000_his.nc>.

The map file — with full-grid output of flow fields — is written for each domain separately as <mdident_000X_map.nc>. This saves communication overhead during the parallel run. The partitioned map files contain duplicate points, since each file also contains the domain's ghost nodes. For postprocessing these map files, two options are now available:

- 1 Direct plotting of the set of all map files in Delft3D-QUICKPLOT: the partitioned file series will be recognized automatically, and the partition results will be drawn on top of each other. For water levels this gives good results.
- 2 Merging the partitioned map files into a single global map file with the DFMOUTPUT tool. The resulting map file can then be loaded again in Delft3D-QUICKPLOT and other post-processing utilities.

5.2.5.1 Plotting all partitioned map files with Delft3D-QUICKPLOT

When opening one of the partitioned map files into Delft3D-QUICKPLOT, it will automatically detect that the map file is part of a series. An additional select list *Domain* appears, see [Figure 5.2](#). Select either “all partitions”, or a partition of your choice, and proceed with the plotting as normal ([section 6.3](#)).

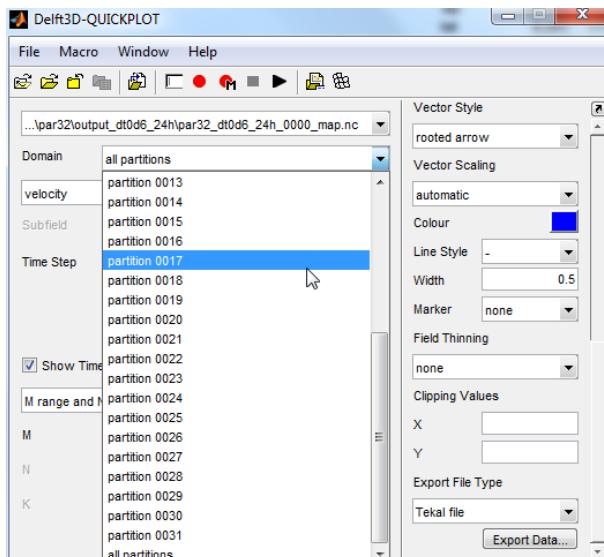


Figure 5.2: Domain selector in Delft3D-QUICKPLOT for partitioned map files.

5.2.5.2 Merging multiple map files into one

The partitioned map files of a parallel model run can be merged into a single global map file with the DFMOUPUT tool. It cuts off ghost nodes, and concatenates all grid points, taking care of correct global renumbering. Usage:

```
> dfmoutput mapmerge [--infile FILE1 [FILE2 FILE3...]] [--outfile DSTFILE]
```

where FILE1/2/3 are the input files, e.g., <mdident_0000_map.nc>, ..., <mdident_0031_map.nc> and DSTFILE is an optional output file name (the default is <model>_merged_map.nc>).

The built-in help gives a list of more advanced options:

```
> dfmoutput mapmerge --help
Merge multiple map files from parallel run into one.

Optional switches:
--infile FILE1 [FILE2...], -i FILE1 [FILE2...]
    default value
    One or more input files.
--listfile LISTFILE, -F LISTFILE
    Pass contents of LISTFILE as input files.
--outfile DSTFILE, -o DSTFILE
    Write output to file DSTFILE. Default: <model>_merged_map.nc
--force, -f
    default value .false.
    Force overwriting of existing output file.
--help, -h
    Print this help message
--version, -v
    Print version

Examples:
dfmoutput mapmerge model_0000_map.nc model_0001_map.nc
dfmoutput extract --station='stat A' model_his.nc
```

5.3 Running a scenario using Delta Shell

In the project tree, select the model you want to run by means of clicking the first attribute of the desired model ([Figure 6.1](#)).

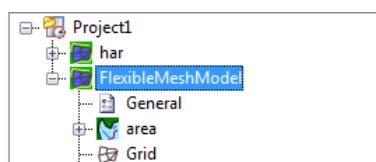


Figure 5.3: Selecting the model you want to run in project tree

Starting the calculation can be achieved in two ways. You can navigate the menu ribbons; go to "Home" and in the group Run click on the button "Run Current". To run all models that are opened in the project tree, click on "Run All" ([Figure 5.4](#)).



Figure 5.4: Group Run in Home ribbon

Alternatively, you can right mouse click the first attribute in the project tree of the model you want to run. Next, click “Run Model” to start the calculation. When you select the first of the model you want to run, the properties window (“View” “Properties”) shows several properties of the model you selected. If you set “ShowModelRunConsole” on true, the model run console of the computational core will be showed during the calculation, providing you with additional information during the model run ([Figure 5.5](#)).

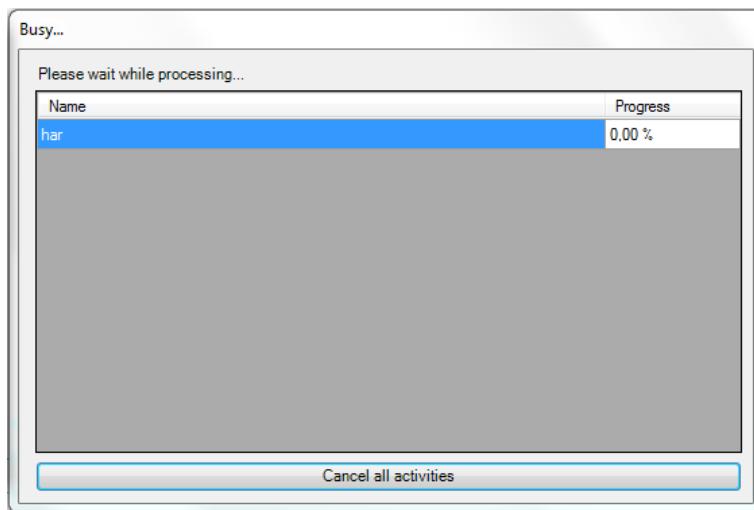


Figure 5.5: Run console Delta Shell

When you cancel the run by clicking “Cancel all activities”, model results are stored up to the point where you cancel the run.

5.4 Running a scenario using a batch script

Separate scripts are needed for Windows (with the extension <.bat>) and Linux (with the extension <.sh>). See [section 5.7](#) for the command-line arguments.

In this section we refer to the variable %DFLOWFM% for Windows and \$DFLOWFM for Linux as a variable that stores path to the directory that contains the dflowfm-cli package.

The easiest Windows script (assuming you have downloaded the Windows dflowfm-cli package, assuming you are in the directory of a configured test-case):

```
set PATH=%DFLOWFM%\bin;%PATH%
dflowfm-cli.exe --autostartstop YOUR_MDU_FILE.mdu
```

The easiest Linux script (assuming you have downloaded the Linux dflowfm-cli package, assuming you are in the directory of a configured test-case):

```
export LD_LIBRARY_PATH=$DFLOWFM/lib:$LD_LIBRARY_PATH
export PATH=$DFLOWFM/bin:$PATH
dflowfm --autostartstop YOUR_MDU_FILE.mdu
```

5.5 Run time

The actual run time of a model can vary considerably depending on a variety of factors such as:

- ◊ The problem being solved, characterised by the number of active grid points, the number of layers in the vertical or the number of processes taken into account.
- ◊ The length of the simulation in time and the time step being used.
- ◊ The hardware configuration that is used and the work load of the processor.

For this reason, only some general considerations are given to determine the run time of a hydrodynamic simulation. On a PC or a workstation without separate I/O-processors the CPU time is the sum of the processor time and the I/O time.

The *processor time* required for a simulation is primarily determined by:

- ◊ The model definition, i.e., the number of active grid points and the number and type of the processes taken into account.
- ◊ The length of the simulated period in terms of the number of time steps executed.

The *I/O time* is determined by:

- ◊ The number of times the computed data are written to history, map, restart files and other communication files for water quality or wave model couplings.
- ◊ The number of observation points, cross-sections and the number of output parameters.

The simulation performance is defined as the CPU time per grid point per time step per constituent:

$$\text{simulation performance} = \frac{\text{CPU time}}{\text{Dnt} \cdot \text{Ndx}} \quad [\text{system seconds}]$$

where:

- | | |
|-----|--------------------------------------|
| Dnt | is the number of time steps executed |
| Ndx | is the number of flow nodes |

The simulation performance is written to the diagnostic file at the end of the simulation.

5.5.1 Multi-core performance improvements by OpenMP

D-Flow FM has built-in support for multi-core parallelism using OpenMP¹. This speeds up calculations by employing multiple processor cores in a single (shared-memory) computer,

¹<http://www.openmp.org>

e.g., a modern-day notebook. OpenMP-parallelism in D-Flow FM does not scale as well as MPI-parallelism ([section 5.2](#)), but it comes for free (not any change to model input necessary) and can give a welcome performance improvement (approximately double speed on an Intel quadcore CPU). It is strongly advised to limit the number of OpenMP-threads to one less than the number of physical cores in your machine, thus also ignoring any hyperthreading. An example on Linux for an i7 quadcore CPU machine:

```
export OMP_NUM_THREADS=3
dflowfm --autostartstop YOUR_MDU_FILE.mdu
```

5.6 Files and file sizes

For estimating the required disk space the following files are important:

- ◊ history file
- ◊ map file
- ◊ restart file

5.6.1 History file

The size of the history file is determined by:

- ◊ The number of monitoring points (observation points + cross-sections): H1.
- ◊ The number of quantities stored: H2.
- ◊ The number of additional process parameters, such as salinity, temperature, constituents and turbulence quantities, taken into account in the simulation: H3.
- ◊ The number of time the history data is written to the history file: H4.

You can estimate the size of a history file (in bytes) from the following equation:

$$\text{size history file} = H1 \cdot (H2 + H3) \cdot H4 \cdot 8 \text{ bytes.}$$

As a first approximation you can use H2 = 8.

Example

For a 2D simulation with density driven currents (salinity and temperature), a simulated period of 12 hrs 30 min, a time integration step of 5 minutes, 30 monitoring points and each time step being stored, the size of the history file will be of the order of 384 kBytes. For the same model but now with 10 layers in the vertical the file size will increase to about 4 MBytes. These estimates show that history files are rather small. Unless the number of monitoring points is excessively large the history files are typically much smaller than the map output files.

5.6.2 Map file

The size of the map file is determined by:

- ◊ The size of the model, i.e. the number of grid cells multiplied by the number of layers ($N_{dx} \cdot K_{max}$): M1n, and the number of flow links (open grid cell edges) multiplied by the number of layers ($L_{nx} \cdot K_{max}$): M1l.
- ◊ The number of quantities stored on grid cells and flow links: M2n, M2l, respectively.
- ◊ The number of process parameters taken into account, such as salinity, temperature,

- constituents and turbulence quantities: M3.
 ◇ The number of time steps for which the map file is written: M4.

Remark:

- ◇ For a more refined estimate you should distinguish between parameters that depend or not on the number of layers used (such as the water level). For a 3D simulation the latter quantities can be neglected, for a 2D simulation they must be accounted for. As a first estimate we double the number of quantities M2 in a 2D simulation.



As a first approximation you can use $M2n = 5$, $M2l = 5$ for a 3D simulation and $M2n = 8$, $M2l = 5$ for a 2D simulation.

You can estimate the size of a map file (in bytes) from the following equation:

$$\text{size map file} = (M1n \cdot (M2n + M3) + M1l \cdot M2l) \cdot M4 \cdot 8 \text{ bytes.}$$

Example

For a 2D simulation with 6800 grid cells and 13000 flow links, simulation results stored for a period of 7 days, and the file is written with an interval of 60 minutes the size of the map file will be about 161 MBytes. For larger models the map file can easily become excessively large, as result the map file is less frequently written, for instance every 2 or 3 hours.

5.6.3 Restart file

A restart file is a special type of map file, where only one time snapshot per file is saved (i.e., $M4 = 1$). No grid geometry information is stored in a restart file, but since this independent data also was not included in the above map file size estimates, the equation for map files equally applies to restart files.

5.7 Command-line arguments

A complete model schematisation can be run from the command line using the D-Flow FM Command Line Interface (CLI), `dflowfm-cli.exe` (`dflowfm` on Linux). A basic non-interactive run is started by:

```
> dflowfm-cli --autostartstop mdident.mdu
```

In the box below, a full list of command-line options and arguments is shown:

```
> dflowfm-cli --help
Usage: dflowfm-cli [OPTIONS] [FILE]...
Options:
  --autostart MDUFILE
    Auto-start the model run, and wait upon completion.

  --autostartstop MDUFILE
    Auto-start the model run, and exit upon completion.

  --noautostart MDUFILE
    Disable any AutoStart option in the MDU file (if any).
```

```
--partition:OPTS [POLFILE] NETFILE
    Partitions the unstructured grid in NETFILE into multiple files.

    POLFILE is an optional polygon file which defines the partitions.
    Only used when ndomain in OPTS is undefined or 0.

    OPTS is a colon-separated list opt1=val1:opt2=val2:...
        ndomains=N           Number of partitions.
        contiguous=[01]       Enforce contiguous grid cells in each domain.

-t N, --threads N
    Set maximum number of OpenMP threads. N must be a positive integer.

--refine:OPTS NETFILE
    Refine the unstructured grid in NETFILE from commandline.
    OPTS is a colon-separated list opt1=val1:opt2=val2:...
        hmin=VAL
        dtmax=VAL
        maxlevel=M
        connect=[01]
        directional=[01]
        outsidecell=[01]

-q, --quiet
    Minimal output: Only (fatal) errors are shown.

--verbose:[level_stdout[:level_dia]], e.g., --verbose:INFO:DEBUG
    Set verbosity level of output on standard out and in diagnostics file.
    where level is in: {DEBUG|INFO|WARNING|ERROR|FATAL}
    Levels are optional, default is INFO on screen, DEBUG in dia file.

-h, --help
    Display this help information and exit.

-v, --version
    Output version information and exit.
```

5.8 Frequently asked questions

This chapter aims to help you with common questions that may arise while using D-Flow FM.

1 Question

My model does not run/crashes. What's wrong?

Answer

The diagnostics file is the starting point for finding out what went wrong. See [section E.1](#) for a detailed description of the contents of this file, and the order of model run output. Globally, ask yourself the following questions:

- ◊ Was the MDU file found?
- ◊ If yes, was the model successfully loaded? Common mistakes are missing boundary or meteorological forcings file.
- ◊ If yes, was the time loop successfully started? Possible errors are non-writable output files.
- ◊ If yes, does the dia file contain any messages from during the time loop? Possible errors are solver convergence errors.
- ◊ If not, did the run end successfully?

2 Question

I get a warning that my network is non-orthogonal. Can I loosen the orthogonality thresh-

old?

Answer

Unfortunately, no. Orthogonality is very important for accuracy: advised orthogonality values for your grid are around 0.01, preferably lower. The current threshold is already very high at a value of 0.5. Use RGFGRID to improve your grid orthogonality (and smoothness).

6 Visualize results

6.1 Introduction

A model run will produce 2 types of output:

- ◊ a graph or history (*.his-file) for a specific quantity on a location
- ◊ a map (*.map-file) for a specific quantity

Both are stored in files within the project, as described in [section 5.6](#).

D-Flow FM provides basic visualization of the model and the model results. Advanced and tailor-made visualization is possible by the export of the his- and/or map-files, and inspection with dedicated visualization applications. Some are provided and described below.

6.2 Visualization with Delta Shell

When your model run has finished, a new folder called “Output” has appeared at the bottom of the attributes of your model in the project tree. Inside this folder, all output quantities of the his-, and map-files can be found, as well as a folder called “States”, in which you find all restart files written during the model run. Output folders have also appeared in the map project tree; “Output (map)” and “Output (his)”. Both the results of your map and his files can be viewed in the central map by means of enabling the desired quantities from the map project tree ([Figure 6.1](#)).

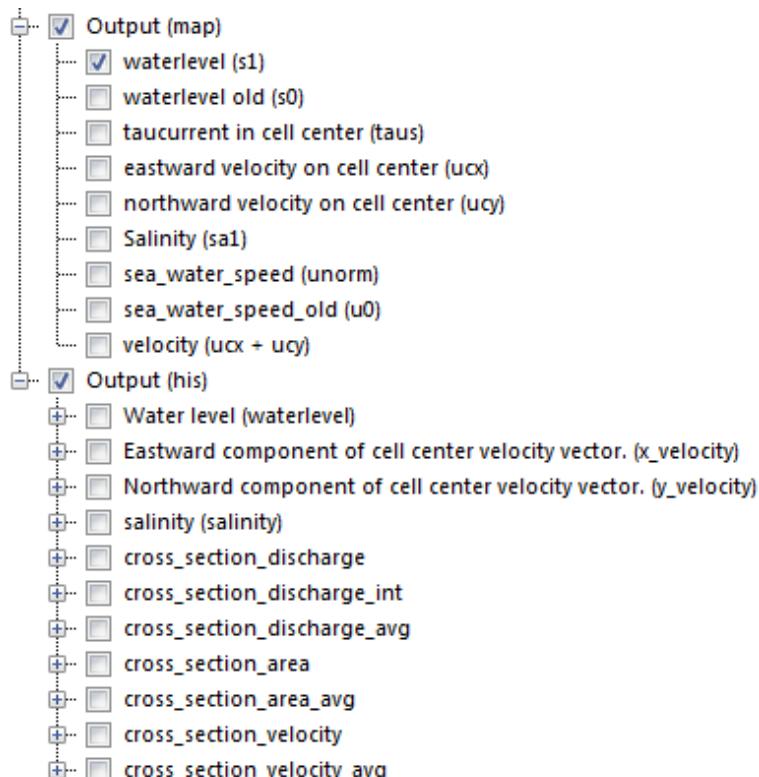


Figure 6.1: Example of output folders in the map project tree

The time navigator can be used to slide through the different time steps in the output files. If your model is relatively large, the drawing performance of the interface can be improved

dramatically by enabling QuadTree visualizations in the properties window of the layer you want to visualize. To this end, select the layer you want to visualize in the ...

6.3 Visualization with Quickplot

The interface of the Delft3D-QUICKPLOT allows to open the NetCDF output files from a D-Flow FM simulation. In the interface you can select data fields, make a selection of time steps, stations or specify a preferred figure presentation options. After selection of a certain options you can visualize your data by using the “Quick View” button. To add another plot to an existing figure the “Add to Plot” button should be used.

When plotting the map output files from a D-Flow FM simulation, by default the presentation type “markers” will be selected. To smoothly visualize your results it is recommended to change presentation type into “polygons” and then select the option “Fill Polygons” (see the example on the [Figure 6.2](#)).

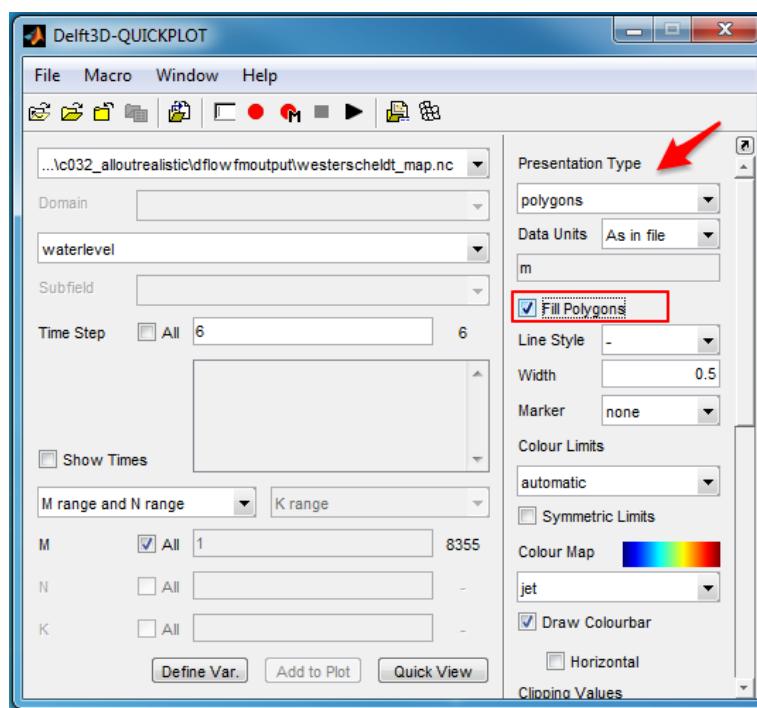


Figure 6.2: Useful map visualization options in the Delft3D-QUICKPLOT

See the Delft3D-QUICKPLOT User Manual for full details and a description of the routines and their use.

6.4 Visualization with Muppet

The interface of the Muppet allows to “Add Dataset” from a NetCDF output file from a D-Flow FM simulation. It is possible to create a timeseries for a selected variable in from the history files. Additionally Muppet offers the option to visualize the map output files from a D-Flow FM simulation (see [Figure 6.3](#)).

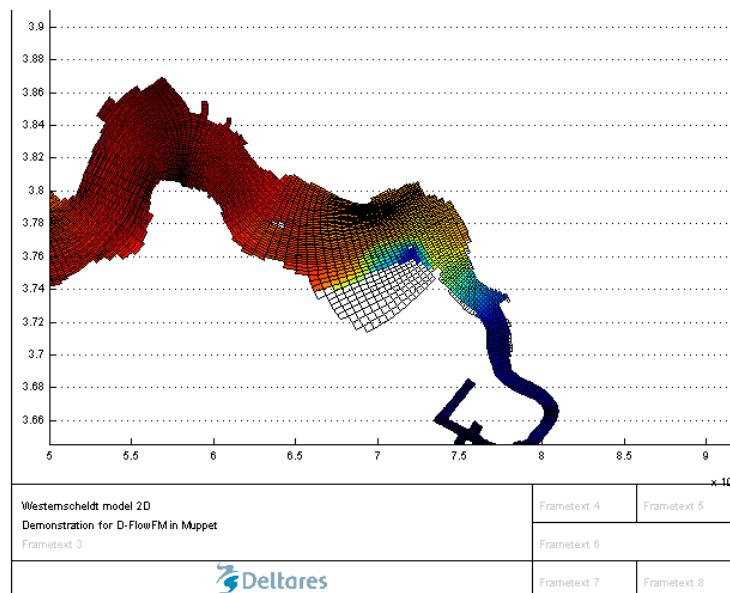


Figure 6.3: Example of the Muppet visualization of the D-Flow FM map output file

In the Muppet interface it is possible to specify figure settings, plot descriptions, labels and many more. See the Muppet User Manual for full details and a description of the routines and their use.

6.5 Visualization with Matlab

In the OpenEarthTools you can find two example scripts that allow you to load and visualize the D-Flow FM output. Note that in the Matlab scripts for the D-Flow FM the UGRID notation is used.

The <plotMap.m> script plots a D-Flow FM unstructured map and optionally the handles h are returned. By modifying the script you can change the plotted variable, layout or the plot style options.

The <plotNet.m> script plots a D-Flow FM unstructured net, optionally the handles h are returned. This script can visualize nodes, links and cell circumcenters of a network. By modifying the script you can choose the preferred plotting settings and variables to plot (by default nodes, edges and faces are plotted).

6.6 Visualization with Python

Currently there are no examples of Python scripts for the D-Flow FM output visualization in the OpenEarthTools. However, if such a need arises it is possible to load netcdf Python libraries, and create your own simple visualization of the variables present in the NetCDF output files.



7 Hydrodynamics

7.1 Introduction

Increasing awareness of environmental issues has focused the attention of scientists and engineers on the problem of predicting the flow and dispersion of contaminants in water systems. Reliable information on water flow, waves, water quality, sediment transport and morphology can be obtained from appropriate mathematical models. In general the first step in such modelling activities concerns the simulation of the flow itself. Whether the problem is related, for example, to the stability of a hydraulic structure, to salt intrusion, to the dispersion of pollutants or to the transport of silt and sediment, flow simulations usually form the basis of the investigations to be carried out.

The *Delft3D Flexible Mesh Suite* is the integrated modelling system of Deltares for the aquatic environment. D-Flow Flexible Mesh, the flow module of this system, provides the hydrodynamic basis for other modules such as water quality, ecology, waves and morphology. For steady and non-steady modelling of the far-field water quality and ecology, it is coupled with the far-field water quality module D-Water Quality. For the interaction between waves and currents the flow module may be coupled with the short-waves model D-Waves. To control structures, the flow module is coupled to the D-Real Time Control module.

D-Flow FM is flexible by using an unstructured grid in the horizontal plane. In the vertical direction D-Flow FM offers two different vertical grid systems: a so-called σ co-ordinate system (σ -model) introduced by [Phillips \(1957\)](#) for ocean models and the Cartesian Z co-ordinate system (Z-model).

This chapter gives some background information on the conceptual model of the D-Flow FM module. Most of the concepts and algorithms are applicable to both the σ -model and Z-model.

7.2 General background

7.2.1 Range of applications of D-Flow FM

The hydrodynamic module D-Flow FM simulates two-dimensional (2DH, depth-averaged) or three-dimensional (3D) unsteady flow and transport phenomena resulting from tidal and/or meteorological forcing, including the effect of density differences due to a non-uniform temperature and salinity distribution (density-driven flow). The flow model can be used to predict the flow in shallow seas, coastal areas, estuaries, lagoons, rivers and lakes. It aims to model flow phenomena of which the horizontal length and time scales are significantly larger than the vertical scales.

If the fluid is vertically homogeneous, a depth-averaged approach is appropriate. D-Flow FM is able to run in two-dimensional mode (one computational layer), which corresponds to solving the depth-averaged equations. Examples in which the two-dimensional, depth-averaged flow equations can be applied are tidal waves, storm surges, tsunamis, harbor oscillations (seiches) and transport of pollutants in vertically well-mixed flow regimes.

Three-dimensional modelling is of particular interest in transport problems where the horizontal flow field shows significant variation in the vertical direction. This variation may be generated by wind forcing, bed stress, Coriolis force, bed topography or density differences. Examples are dispersion of waste or cooling water in lakes and coastal areas, upwelling and downwelling of nutrients, salt intrusion in estuaries, fresh water river discharges in bays and thermal stratification in lakes and seas.

7.2.2 Physical processes

The numerical hydrodynamic modelling system D-Flow FM solves the unsteady shallow water equations in two (depth-averaged) or in three dimensions. The system of equations consists of the horizontal equations of motion, the continuity equation, and the transport equations for conservative constituents. The equations are formulated in orthogonal curvilinear co-ordinates or in spherical co-ordinates on the globe. In D-Flow FM models with structured grid are considered as a simplified form of an unstructured grid. In Cartesian co-ordinates, the free surface level and bathymetry are related to a flat horizontal plane of reference, whereas in spherical co-ordinates the reference plane follows the Earth curvature.

The flow is forced by tide at the open boundaries, wind stress at the free surface, pressure gradients due to free surface gradients (barotropic) or density gradients (baroclinic). Source and sink terms are included in the equations to model the discharge and withdrawal of water.

The D-Flow FM model includes mathematical formulations that take into account the following physical phenomena:

- ◊ Free surface gradients (barotropic effects).
- ◊ The effect of the Earth rotation (Coriolis force).
- ◊ Water with variable density (equation of state).
- ◊ Horizontal density gradients in the pressure (baroclinic effects).
- ◊ Turbulence induced mass and momentum fluxes (turbulence closure models).
- ◊ Transport of salt, heat and other conservative constituents.
- ◊ Tidal forcing at the open boundaries.
- ◊ Space and time varying wind shear-stress at the water surface.
- ◊ Space varying shear-stress at the bed.
- ◊ Space and time varying atmospheric pressure on the water surface.
- ◊ Time varying sources and sinks (e.g. river discharges).
- ◊ Drying and flooding of tidal flats.
- ◊ Heat exchange through the free surface.
- ◊ Evaporation and precipitation.
- ◊ Tide generating forces.
- ◊ Effect of secondary flow on depth-averaged momentum equations.
- ◊ Lateral shear-stress at wall.
- ◊ Vertical exchange of momentum due to internal waves.
- ◊ Influence of waves on the bed shear-stress (2D and 3D).
- ◊ Wave induced stresses (radiation stress) and mass fluxes.
- ◊ Flow through hydraulic structures.
- ◊ Wind driven flows including tropical cyclone winds.

7.2.3 Assumptions underlying D-Flow FM

In D-Flow FM the 2D (depth-averaged) or 3D non-linear shallow water equations are solved. These equations are derived from the three dimensional Navier-Stokes equations for incompressible free surface flow. The following assumptions and approximations are applied:

- ◊ In the σ co-ordinate system the depth is assumed to be much smaller than the horizontal length scale. For such a small aspect ratio the shallow water assumption is valid, which means that the vertical momentum equation is reduced to the hydrostatic pressure relation. Thus, vertical accelerations are assumed to be small compared to the gravitational acceleration and are therefore not taken into account.
- ◊ The effect of variable density is only taken into account in the pressure term (Boussinesq approximation).
- ◊ In the σ co-ordinate system, the immediate effect of buoyancy on the vertical flow is

not considered. In D-Flow FM vertical density differences are taken into account in the horizontal pressure gradients and in the vertical turbulent exchange coefficients. So the application of D-Flow FM is restricted to mid-field and far-field dispersion simulations of discharged water.

- ◊ For a dynamic online coupling between morphological changes and flow the 2D sediment and morphology feature is available.
- ◊ In a Cartesian frame of reference, the effect of the Earth curvature is not taken into account. Furthermore, the Coriolis parameter is assumed to be uniform unless specifically specified otherwise.
- ◊ In spherical co-ordinates the inertial frequency depends on the latitude.
- ◊ At the bed a slip boundary condition is assumed, a quadratic bed stress formulation is applied.
- ◊ The formulation for the enhanced bed shear-stress due to the combination of waves and currents is based on a 2D flow field, generated from the velocity near the bed using a logarithmic approximation.
- ◊ The equations of D-Flow FM are capable of resolving the turbulent scales (large eddy simulation), but usually the hydrodynamic grids are too coarse to resolve the fluctuations. Therefore, the basic equations are Reynolds-averaged introducing so-called Reynolds stresses. These stresses are related to the Reynolds-averaged flow quantities by a turbulence closure model.
- ◊ In D-Flow FM the 3D turbulent eddies are bounded by the water depth. Their contribution to the vertical exchange of horizontal momentum and mass is modelled through a vertical eddy viscosity and eddy diffusivity coefficient (eddy viscosity concept). The coefficients are assumed to be proportional to a velocity scale and a length scale. The coefficients may be specified (constant) or computed by means of an algebraic, $k - \tau$ or $k - \epsilon$ turbulence model, where k is the turbulent kinetic energy, L is the turbulent time scale and ϵ is the dissipation rate of turbulent kinetic energy.
- ◊ In agreement with the aspect ratio for shallow water flow, the production of turbulence is based on the vertical (and not the horizontal) gradients of the horizontal flow. In case of small-scale flow (partial slip along closed boundaries), the horizontal gradients are included in the production term.
- ◊ The boundary conditions for the turbulent kinetic energy and energy dissipation at the free surface and bed assume a logarithmic law of the wall (local equilibrium).
- ◊ The eddy viscosity is an-isotropic. The horizontal eddy viscosity and diffusivity coefficients should combine both the effect of the 3D turbulent eddies and the horizontal motions that cannot be resolved by the horizontal grid. The horizontal eddy viscosity is generally much larger than the vertical eddy viscosity.
- ◊ For large-scale flow simulations, the tangential shear-stress at lateral closed boundaries can be neglected (free slip). In case of small-scale flow partial slip is applied along closed boundaries.
- ◊ For large-scale flow simulations, the horizontal viscosity terms are reduced to a bi-harmonic operator along co-ordinate lines. In case of small-scale flow the complete Reynold's stress tensor is computed. The shear-stress at the side walls is calculated using a logarithmic law of the wall.
- ◊ In the σ co-ordinate system, D-Flow FM solves the so-called long wave equation. The pressure is hydrostatic and the model is not capable of resolving the scales of short waves. Therefore, the basic equations are averaged in analogy with turbulence introducing so-called radiation stresses. These stresses are related to the wave quantities of Delft3D-WAVE by a closure model.
- ◊ It is assumed that a velocity point is set dry when the actual water depth is below half of a user-defined threshold. If the point is set dry, then the velocity at that point is set to zero. The velocity point is set wet again when the local water depth is above the threshold. The hysteresis between drying and flooding is introduced to prevent drying and flooding in two consecutive time steps. The drying and flooding procedure leads to a discontinuous

movement of the closed boundaries at tidal flats.

- ◊ A continuity cell is set dry when all surrounding velocity points at the grid cell faces are dry or when the actual water depth at the cell centre is below zero (negative volume).
- ◊ The flux of matter through a closed wall and through the bed is zero.
- ◊ Without specification of a temperature model, the heat exchange through the free surface is zero. The heat loss through the bed is always zero.
- ◊ If the total heat flux through the water surface is computed using a temperature excess model the exchange coefficient is a function of temperature and wind speed and is determined according to [Sweers \(1976\)](#). The natural background temperature is assumed constant in space and may vary in time. In the more advanced heat flux formulation the fluxes due to solar radiation, atmospheric and back radiation, convection, and heat loss due to evaporation are modeled separately.
- ◊ The effect of precipitation on the water temperature is accounted for.

7.3 Hydrodynamic processes

D-Flow FM solves the Navier Stokes equations for an incompressible fluid, under the shallow water and the Boussinesq assumptions. In the vertical momentum equation the vertical accelerations are neglected, which leads to the hydrostatic pressure equation. In 3D models the vertical velocities are computed from the continuity equation. The set of partial differential equations in combination with an appropriate set of initial and boundary conditions is solved on an unstructured finite volume grid.

In the horizontal direction D-Flow FM uses orthogonal unstructured grids. Two coordinate references are supported:

- ◊ Cartesian co-ordinates
- ◊ Spherical co-ordinates

The boundaries of a river, an estuary or a coastal sea are in general curved and are not smoothly represented on a structured grid. The boundary becomes irregular and may introduce significant discretization errors. To reduce these errors unstructured grids are used. The unstructured grids also allow local grid refinement in areas with large horizontal gradients.

In the vertical direction D-Flow FM offers two different vertical grid systems: the σ coordinate system (σ -model) and the Cartesian Z co-ordinate system (Z-model). In the σ model, the vertical grid consists of layers bounded by two σ planes, which are not strictly horizontal but follow the bed topography and the free surface. Because the σ -grid is boundary fitted both to the bed and to the moving free surface, a smooth representation of the topography is obtained. The number of layers over the entire horizontal computational area is constant, irrespective of the local water depth. The distribution of the relative layer thickness is usually non-uniform. This allows for more resolution in the zones of interest such as the near surface area (important for e.g. wind-driven flows, heat exchange with the atmosphere) and the near bed area (sediment transport). Please note that in D-Flow FM, unlike Delft3D, the σ coordinate is equal to zero on the bed and 1 on the water surface.

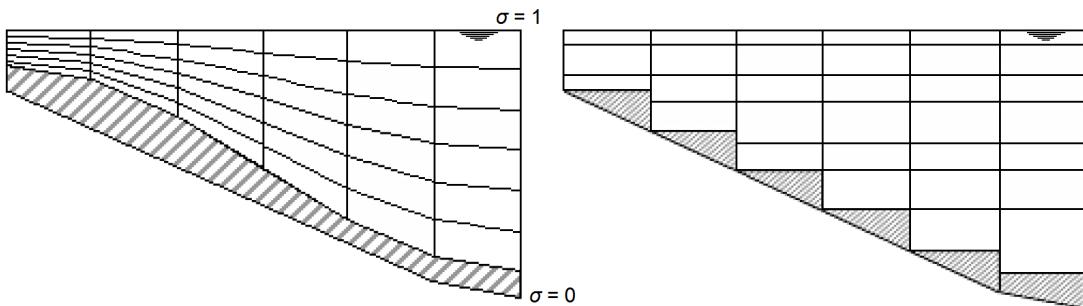


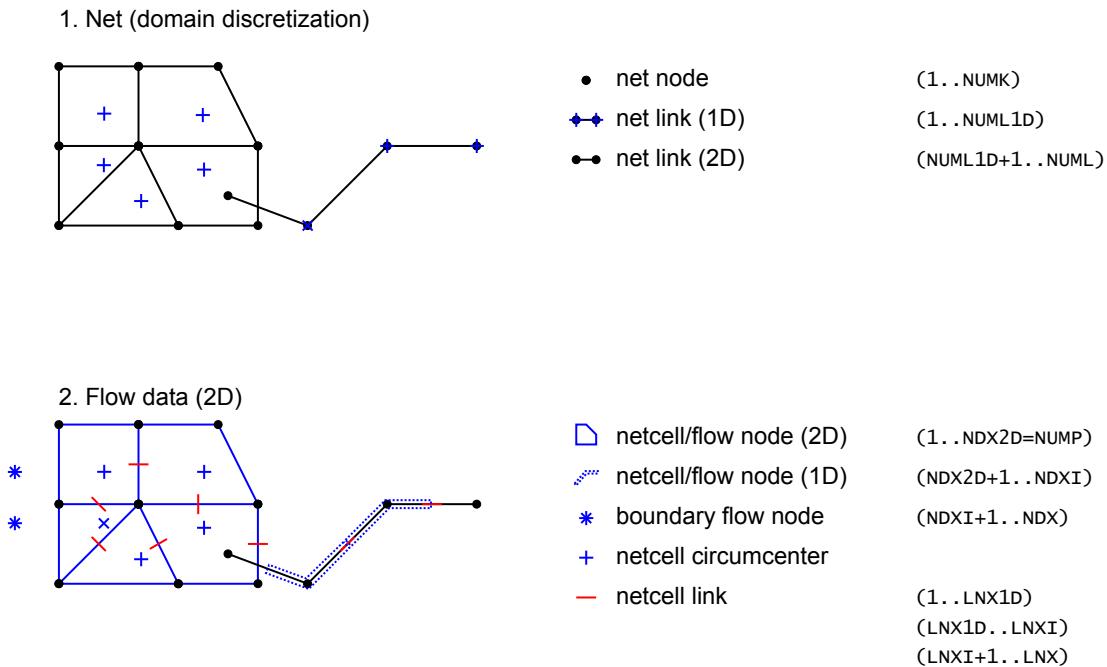
Figure 7.1: Example of σ -grid (left) and Z -grid (right).

Although the σ -grid is boundary fitted (in the vertical), it will not always have enough resolution around the pycnocline. The co-ordinate lines intersect the density interfaces that may give significant errors in the approximation of strictly horizontal density gradients (Leendertse, 1990; Stelling and Van Kester, 1994). Therefore, Z -grid was introduced in D-Flow FM for 3D simulations of weakly forced stratified water systems. The Z -grid model has horizontal co-ordinate lines that are (nearly) parallel with density interfaces (isopycnals) in regions with steep bed slopes. This is important to reduce artificial mixing of scalar properties such as salinity and temperature. The Z -grid is not boundary-fitted in the vertical. The bed (and free surface) is usually not a co-ordinate line and is represented as a staircase (zig-zag boundary).

7.3.1 Topological conventions

A computational cell in a D-Flow FM grid (sometimes referred to as a 'network') consists of corner nodes and edges connecting the corner nodes. Such a grid cell should contain at least three corner nodes and at most six corner nodes. The following topological conventions are used:

- ◊ netnodes: corners of a cell (triangles, quadrangles, ...),
- ◊ netlinks: edges of a cell, connecting netnodes,
- ◊ flownodes: the cell circumcentre, in case of triangles the exact intersection of the three perpendicular bisectors and hence also the centre of the circumscribing circle,
- ◊ flowlinks: a line segment connecting two flownodes.

**Figure 7.2:** Flexible mesh topology

This mesh topology is illustrated in [Figure 7.2](#). The 'center' of a cell can be defined in multiple ways. To illustrate this, two conventional cell center definitions for a triangle are highlighted in [Figure 7.3](#). The two displayed cell centers have different properties:

- 1 the circumcenter is the location within the triangle which is the center point of a circle that intersects the triangle at each corner node of the triangle; as a consequence, the orthogonal projection of the center point to each face of the triangle divides each face into two exactly equidistant pieces,
- 2 the mass center (or centroid) is the center of gravity; as a consequence, a line through a corner node and the mass center divides a face into two exactly equidistant pieces under an angle not necessarily equal to 90° .

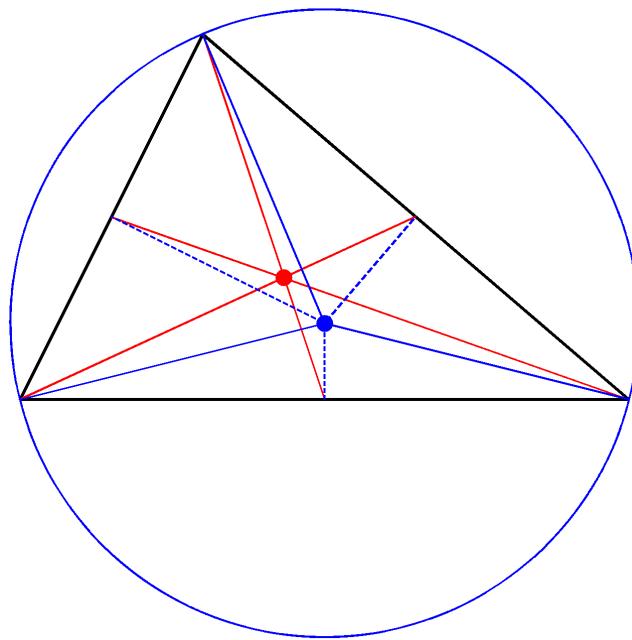


Figure 7.3: Two conventional definitions of the cell center of a triangle: the *circumcenter* and the *mass center*.

D-Flow FM utilizes the **circumcenter** as the basis of the definition of the elementary flow variables 'water level' and 'flow velocity'. The water level is defined at the circumcenter, whereas the face normal flow velocity is defined at the orthogonal projection of the circumcenter onto the cell face, i.e., the midpoint of the cell face.

Important properties of the mesh are the *orthogonality* and *smoothness*. The *orthogonality* is defined as the cosine of the angle φ between a flowlink and a netlink. The *smoothness* of a mesh is defined as the ratio of the areas of two adjacent cells. Ideally, both parameters equal 1, i.e. the angle $\varphi = 90^\circ$ and the areas of the cells are equal to each other. A nearly ideal setup is shown in [Figure 7.4](#).

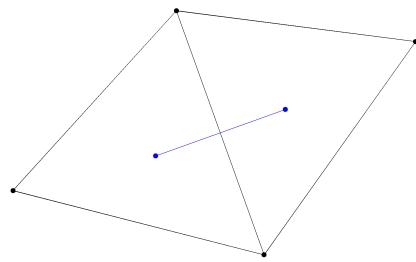


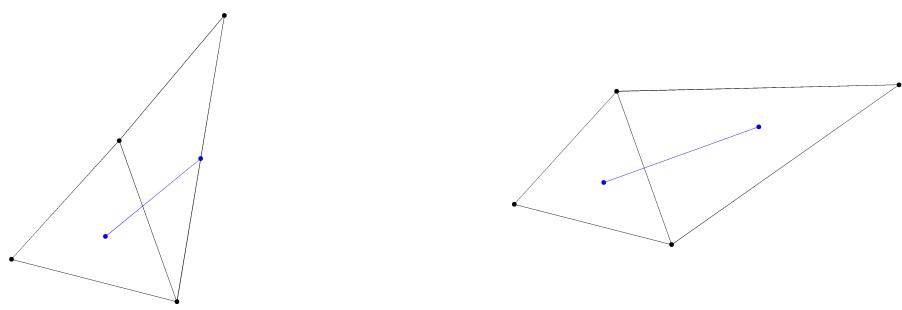
Figure 7.4: Nearly perfect orthogonality and smoothness along the edge connecting two triangles. Black lines/dots are network links/nodes, blue lines/dots are flow links/nodes.

It is quite easy (and therefore dangerous) to generate meshes that violate the orthogonality

and smoothness requirements. In [Figure 7.5](#), two different setups of two gridcells are shown with different mesh properties.

The left picture of [Figure 7.5](#) shows how orthogonality can be deteriorated by skewing the right triangle with respect to the left triangle. While having the same area (perfect smoothness), the mutually oblique orientation results in poor orthogonality. In this particular case, the centre of the circumscribing circle is in principle located outside the right triangle. Such a triangle is denoted as an 'open' triangle, which is bad for computations.

The opposite is shown in the right picture of [Figure 7.5](#) in which the right triangle has strongly been elongated, disturbing the smoothness property. However, the orthogonality is nearly perfect. Nonetheless, both meshes need to be improved to assure accurate model results.



(a) Perfect smoothness, but poor orthogonality. (b) Perfect orthogonality, but poor smoothness

Figure 7.5: Poor mesh properties due to violating either the smoothness or the orthogonality at the edge connecting two triangles. Black lines/dots are network links/nodes, blue lines/dots are flow links/nodes.

7.3.2 Conservation of mass and momentum

In this section, we will present in detail the governing equations for mass and momentum conservation. These equations are indicated by a continuity equation and momentum equations.

7.3.2.1 Continuity equation

D-Flow FM solves the depth-averaged continuity equation, derived by integrating the continuity equation, for incompressible fluids ($\nabla \cdot \vec{u} = 0$) over the total depth, taken into account the kinematic boundary conditions at water surface and bed level, and is given by:

$$\frac{\partial h}{\partial t} + \frac{\partial Uh}{\partial x} + \frac{\partial Vh}{\partial y} = hQ \quad (7.1)$$

with U and V the depth averaged velocities. Q is representing the contributions per unit area due to the discharge or withdrawal of water, precipitation and evaporation:

$$Q = \int_0^h (q_{in} - q_{out}) dz + P - E \quad (7.2)$$

with q_{in} and q_{out} the local sources and sinks of water per unit of volume [1/s], respectively, P the non-local source term of precipitation and E non-local sink term due to evaporation. We remark that the intake of, for example, a power plant is a withdrawal of water and should be modelled as a sink. At the free surface there may be a source due to precipitation or a sink due to evaporation.

7.3.2.2 Momentum equations in horizontal direction

The momentum equations in x - and y -direction are given by:

$$\frac{\partial u}{\partial t} + u \frac{\partial u}{\partial x} + v \frac{\partial u}{\partial y} + \frac{w}{h} \frac{\partial u}{\partial z} - fv = -\frac{1}{\rho_0} \frac{\partial P}{\partial x} + F_x + \frac{1}{h^2} \frac{\partial}{\partial z} \left(\nu_V \frac{\partial u}{\partial z} \right) + M_x \quad (7.3)$$

$$\frac{\partial v}{\partial t} + u \frac{\partial v}{\partial x} + v \frac{\partial v}{\partial y} + \frac{w}{h} \frac{\partial v}{\partial z} - fu = -\frac{1}{\rho_0} \frac{\partial P}{\partial y} + F_y + \frac{1}{h^2} \frac{\partial}{\partial z} \left(\nu_V \frac{\partial v}{\partial z} \right) + M_y \quad (7.4)$$

Where ν_V is the vertical eddy viscosity coefficient. Density variations are neglected, except in the baroclinic pressure terms, $\partial P/\partial x$ and $\partial P/\partial y$ represent the pressure gradients. The forces F_x and F_y in the momentum equations represent the unbalance of horizontal Reynolds stresses. M_x and M_y represent the contributions due to external sources or sinks of momentum (external forces by hydraulic structures, discharge or withdrawal of water, wave stresses, etc.). The effects of surface waves on the flow as modelled in D-Flow FM are described in section [section 14.2](#).

7.3.2.3 Vertical velocities

The vertical velocity w in the adapting σ co-ordinate system is computed from the continuity equation:

$$\frac{\partial h}{\partial t} + \frac{\partial uh}{\partial x} + \frac{\partial vh}{\partial y} + \frac{\partial w}{\partial z} = h (q_{in} - q_{out}) \quad (7.5)$$

At the surface the effect of precipitation and evaporation is taken into account. The vertical velocity w is defined at the iso σ -surfaces. w is the vertical velocity relative to the moving σ -plane. It may be interpreted as the velocity associated with up- or downwelling motions.

7.3.3 The hydrostatic pressure assumption

Under the shallow water assumption, the vertical momentum equation is reduced to a hydrostatic pressure equation. Vertical accelerations due to buoyancy effects and due to sudden variations in the bed topography are not taken into account. So:

$$\frac{\partial P}{\partial z} = -\rho g H \quad (7.6)$$

For water of constant density and taking into account the atmospheric pressure, it includes gradients of the free surface level, called barotropic pressure gradients. The atmospheric pressure is included in the system for storm surge simulations. The atmospheric pressure gradients dominate the external forcing at peak winds during storm events. Space and time varying wind and pressure fields are especially important when simulating storm surges.

In case of a non-uniform density the pressure gradients includes not only barotropic pressure gradient, but also vertical pressure gradient, the so called baroclinic pressure gradient. The baroclinic pressure gradient is the result of variable distribution of density and temperature in the vertical direction.

In the horizontal gradient a vertical derivative is introduced by the σ co-ordinate transformation. In estuaries and coastal seas the vertical grid may deteriorate strongly in case of steep bed slopes. In order to avoid artificial flow the numerical approximation of the baroclinic pressure terms requires a special numerical approach. The treatment of D-Flow FM to avoid the artificial mixing due to σ co-ordinates are discussed in [section 7.5](#), see also [Stelling and Van Kester \(1994\)](#).

7.3.4 The Coriolis force

The Coriolis parameter f depends on the geographic latitude and the angular speed of rotation of the earth, Ω : $f = 2\Omega \sin \phi$. For a grid the user should specify the space varying Coriolis parameter, using a suitable projection. This can be done by selecting *Coordinate System* in RGFGRID, and selection of the option for *Spherical Coordinate*. The parameters for translation and rotation can be given as shown in [Figure 7.6](#).

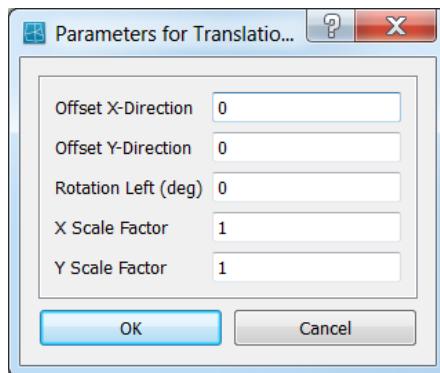


Figure 7.6: Input for map projection for specifying Coriolis parameter on the grid.

7.3.5 Diffusion of momentum

The forces F_x and F_y in the horizontal momentum equations represent the unbalance of horizontal Reynolds stresses. The Reynolds stresses are modelled using the eddy viscosity concept, (for details e.g. [Rodi \(1984\)](#)). This concept expresses the Reynolds stress component as the product between a flow as well as grid-dependent eddy viscosity coefficient and the corresponding components of the mean rate-of-deformation tensor. The meaning and the order of the eddy viscosity coefficients differ for 2D and 3D, for different horizontal and vertical turbulence length scales and fine or coarse grids. In general the eddy viscosity is a function of space and time.

For 3D shallow water flow the stress tensor is an-isotropic. The horizontal eddy viscosity coefficient, ν_H , is much larger than the vertical eddy viscosity ν_V ($\nu_H \gg \nu_V$). The horizontal viscosity coefficient may be a superposition of three parts:

- 1 a part due to "sub-grid scale turbulence",
- 2 a part due to "3D-turbulence" see [Uittenbogaard et al. \(1992\)](#) and
- 3 a part due to dispersion for depth-averaged simulations.

In simulations with the depth-averaged momentum and transport equations, the redistribution of momentum and matter due to the vertical variation of the horizontal velocity is denoted as dispersion. In 2D simulations this dispersion is not simulated as the vertical profile of the horizontal velocity is not resolved. Then this dispersive effect may be modelled as the product of a viscosity coefficient and a velocity gradient. The dispersion term may be estimated by the Elder formulation.

If the vertical profile of the horizontal velocity is not close to a logarithmic profile (e.g. due to stratification or due to forcing by wind) then a 3D-model for the transport of matter is recommended.

The horizontal eddy viscosity is mostly associated with the contribution of horizontal turbulent motions and forcing that are not resolved by the horizontal grid ("sub-grid scale turbulence") or

by (a priori) the Reynolds-averaged shallow-water equations. In 3D, in the vertical direction, ν_V is referred to as the three-dimensional turbulence and in it is computed following a 3D-turbulence closure model.

Therefore, in addition to all turbulence closure models in D-Flow FM a constant (space and time) background mixing coefficient may be specified by the user, which is a background value for the vertical eddy viscosity in the momentum [Equation 7.3](#) and [Equation 7.4](#) consequently.

The horizontal and vertical eddy viscosities can be set by user defined value under Physical Parameters shown in [Figure 7.7](#).

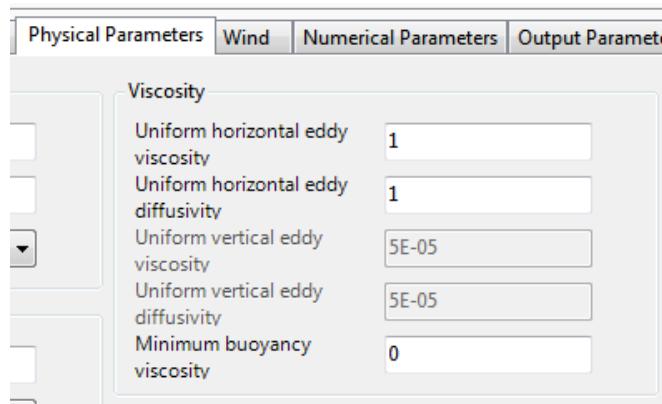


Figure 7.7: Input parameters for horizontal and vertical eddy viscosities.

7.4 Hydrodynamics boundary conditions

In [section 4.4.8](#), the boundary conditions are discussed from the viewpoint of the user interface. In the user interface, the user can specify the locations at which particular boundary conditions are to be imposed. Using [section 4.4.8](#) as a backdrop, the present section discusses the underlying files and fileformats and the way these are interpreted by the computational kernel. Three types of boundary conditions are discussed in this section, namely open boundaries (in [section 7.4.1](#)), vertical boundaries (in [section 7.4.2](#)) and closed boundaries (in [section 7.4.3](#)).

7.4.1 Open boundary conditions

The proper prescription of an open boundary condition along a certain (part of the) rim of the grid can be achieved by considering four elements of the model:

- 1 a polyline file (extension .pli), containing the *locations* at which the boundary conditions should be imposed,
- 2 a boundary conditions file (extension .bc), containing the key *physical information*, such the time dependent information of the quantity under consideration and the physical nature of the quantity itself,
- 3 an external forcing file (extension .ext), in which the connection is laid between the polyline file and the boundary conditons file,
- 4 the prescription of the single external forcing file, containing the connection between location and physical information, in the master definition file (extension .mdu).

These four elements are subsequently discussed in the following.

7.4.1.1 The location of support points

The flow engine needs the specification of support points for the boundary conditions (also see [section 4.4.8.1](#)). By default, these support points are a means to construct a series of fictitious cell centers along the boundary rim of the grid. [Figure 7.8](#) provides an image of this concept.

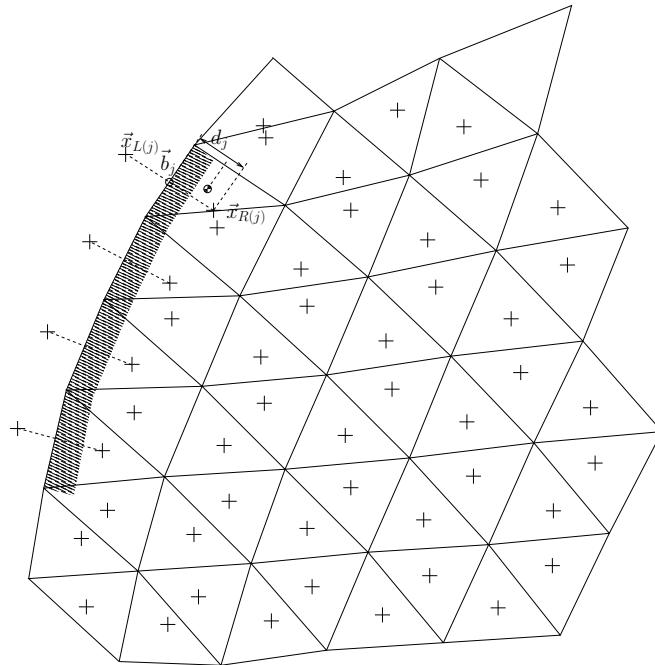


Figure 7.8: Fictitious boundary "cells" near the shaded boundary; \vec{x}_{Lj} is the fictitious "cell" center near boundary face j ; $\vec{x}_{R(j)}$ is the inner-cell center; \vec{b}_j is the point on face j that is nearest to the inner-cell center

The support points are stored as one single polyline per boundary condition, marking the rim along which the boundary conditions should hold. The polyline should be drawn in the vicinity of the rim. The user can specify the size of this ‘vicinity’ by means of the MDU-file keyword `OpenBoundaryTolerance`. The keyword specifies the search tolerance factor between the boundary polyline and the grid cells. The unit of this keyword is the cell size unit (i.e., not metres). By default, this value is 3, which loosely means that in the vicinity of $3\Delta x$ of the grid rim is searched of a boundary condition polyline.

The actual location of a specific boundary location point can be computed in three different ways, dependent on the user’s choice for the keyword `izbndpos` in the MDU-file:

- ◊ `izbndpos = 0`: construction of the boundary condition point by means of orthogonal mirroring of the closest cell center,
- ◊ `izbndpos = 1`: construction of the boundary condition point as the orthogonal projection of the closest cell center onto the grid rim,
- ◊ `izbndpos = 2`: construction of the boundary condition point as the actual location of the support points spanning the polyline.

Only the option `izbndpos = 0` is discussed; the other two options are not yet fully operational. The mirroring of the closest cell center is conducted as follows. First, the orthogonal distance from the boundary cell center to the actual rim of the grid is computed: d_j in [Figure 7.8](#). Second, the cell center of the outside fictitious cell is defined at a distance d_j outside of the

grid. However, the flat area of the cell, say A , at the rim can give rise to an adaptation of this distance. If $\frac{1}{2}\sqrt{A} > d_j$, then the center of the fictitious cell is located at a distance $\frac{1}{2}\sqrt{A}$ away from the grid.

7.4.1.2 Physical information

In the bc-file, multiple types of boundary conditions can be prescribed. In this section, the boundary conditions for the flow motion are briefly reflected on.

Water level

A water level signal is applied at the cell center of the fictitious cell outside the grid (ghost cells or mirror cells). Water levels can be imposed as a timeseries or as a harmonic signal. In case of a harmonic signal, the period of the signal can be given as an astronomic component acronym.

If a timeseries is applied to the first support point of a polyline named `arbitraryname`, prescribed in some polyline file, then the header of the bc-file is:

```
[forcing]
Name          = arbitraryname_0001
Function      = timeseries
Time-interpolation = linear
Quantity       = time
Unit           = minutes since YYYY-MM-DD
Quantity       = waterlevelbnd
Unit           = m
[two-column data]
```

The data is to be inserted as a two-column array containing the time (in minutes) and the water level itself (in meters w.r.t. the reference level). In case of harmonic components, the header of the bc-file is:

```
[forcing]
Name          = arbitraryname_0001
Function      = harmonic
Quantity       = harmonic component
Unit           = minutes
Quantity       = waterlevelbnd amplitude
Unit           = m
Quantity       = waterlevelbnd phase
Unit           = degrees
[three-column data]
```

The data is to be inserted as a three-column array containing the period (in minutes), the amplitude (in meters) and the phase (in degrees). If the period is T minutes, the amplitude is A meters and the phase is φ degrees, then the signal that is applied reads $h(t) = A \cdot \cos(2\pi t/T + \varphi) + B$. The remaining parameter B can be prescribed by an additional signal with a period specified equal to 0 minutes. As an example, the dataseries:

0.0 0.5 0.0

745.0 2.0 0.0

represents the signal $h(t) = 2.0 \cdot \cos(2\pi t/745) + 0.5$, with the time t in minutes.

Discharge

A discharge boundary condition is applied at the face-center of the fictitious boundary cell. For this, the face-normal velocity is used in combination with the face-center water depth. The face-based water depth in the evaluation of the flow area can optionally be set to a downwind approximation (for an inflowing discharge boundary) with the option `jbasqbnndownwindhs = 1` (default value).

Discharges can be imposed as a timeseries or as a harmonic signal. In case of a harmonic signal, the period of the signal can be given as an astronomic component acronym.

If a timeseries is applied to a the first support point of a polyline named `arbitraryname`, prescribed in some polyline file, then the header of the bc-file is:

```
[forcing]
Name          = arbitraryname_0001
Function      = timeseries
Time-interpolation = linear
Quantity       = time
Unit           = minutes since YYYY-MM-DD
Quantity       = dischargebnd
Unit           = m3/s
[two-column data]
```

The data is to be inserted as a two-column array containing the time (in minutes) and the water level itself (in meters w.r.t. the reference level). In case of harmonic components, the header of the bc-file is:

```
[forcing]
Name          = arbitraryname_0001
Function      = harmonic
Quantity       = harmonic component
Unit           = minutes
Quantity       = dischargebnd amplitude
Unit           = m3/s
Quantity       = dischargebnd phase
Unit           = degrees
[three-column data]
```

The data is to be inserted as a three-column array containing the period (in minutes), the amplitude (in cubic meters per second) and the phase (in degrees).

Velocity

A velocity boundary condition is applied at the face-center of the fictitious boundary cell. Values provided are interpreted as face-normal velocities. Velocities can be imposed as a time-series or as a harmonic signal. In case of a harmonic signal, the period of the signal can be given as an astronomic component acronym.

If a timeseries is applied to a the first support point of a polyline named `arbitraryname`, prescribed in some polyline file, then the header of the bc-file is:

```
[forcing]
Name          = arbitraryname_0001
Function      = timeseries
Time-interpolation = linear
Quantity      = time
Unit          = minutes since YYYY-MM-DD
Quantity      = velocitybnd
Unit          = m/s
[two-column data]
```

The data is to be inserted as a two-column array containing the time (in minutes) and the water level itself (in meters w.r.t. the reference level). In case of harmonic components, the header of the bc-file is:

```
[forcing]
Name          = arbitraryname_0001
Function      = harmonic
Quantity      = harmonic component
Unit          = minutes
Quantity      = velocitybnd amplitude
Unit          = m/s
Quantity      = velocitybnd phase
Unit          = degrees
[three-column data]
```

The data is to be inserted as a three-column array containing the period (in minutes), the amplitude (in meters per second) and the phase (in degrees).

Water level gradient

Besides the option to prescribe actual water levels as a boundary condition, D-Flow FM facilitates the prescription of water level *gradients*. Such a Neumann-type boundary condition for the water level can be assigned through the keyword `neumannbnd`. The value of the water level gradient is applied at the face-center.

Water level gradients can be imposed as a timeseries or as a harmonic signal. In case of a harmonic signal, the period of the signal can be given as an astronomic component acronym. If a timeseries is applied to a the first support point of a polyline named `arbitraryname`, prescribed in some polyline file, then the header of the bc-file is:

```
[forcing]
Name          = arbitraryname_0001
```

```

Function      = timeseries
Time-interpolation = linear
Quantity      = time
Unit          = minutes since YYYY-MM-DD
Quantity      = neumannbnd
Unit          = -
[two-column data]

```

The data is to be inserted as a two-column array containing the time (in minutes) and the water level itself (in meters w.r.t. the reference level). In case of harmonic components, the header of the bc-file is:

```

[forcing]
Name          = arbitraryname_0001
Function      = harmonic
Quantity      = harmonic component
Unit          = minutes
Quantity      = neumannbnd amplitude
Unit          = -
Quantity      = neumannbnd phase
Unit          = degrees
[three-column data]

```

The data is to be inserted as a three-column array containing the period (in minutes), the amplitude (in meters per second) and the phase (in degrees). The water level gradient is interpreted positive in the outward-normal direction.

Riemann invariant

At a Riemann boundary we do not allow any outgoing perturbation with respect to some reference boundary state to reflect back from the boundary. This is achieved by prescribing the incoming Riemann invariant. Using the convention of a positive inward normal at the boundary, this can be put as $u_b + 2\sqrt{gH_b}$, with u_b the velocity at the boundary and H_b the total water depth at the boundary. In this expression, we take the boundary values u_b and H_b as the reference boundary state. While applying Riemann boundaries, directional effects are disregarded.

Using linearization and the assumption of a flow field that is initially at rest, the Riemann invariant is rewritten such that it takes the form:

$$\zeta = 2\zeta_b - \sqrt{\frac{H}{g}}u - \zeta_0 \quad (7.7)$$

with ζ the surface level elevation, H the total water depth, g the gravitational acceleration, u the velocity (positive inward) and ζ_0 the initial surface level elevation. Instead of prescribing a combination of the velocity and the water level, we prefer to prescribe only the water level at the boundary, i.e. ζ_b . The value for ζ_b is supposed to be provided by the user (as well as, obviously, the initial surface level elevation ζ_0).

A Riemann invariant can be imposed as a timeseries or as a harmonic signal. In case of a harmonic signal, the period of the signal can be given as an astronomic component acronym. If a timeseries is applied to the first support point of a polyline named `arbitraryname`, prescribed in some polyline file, then the header of the bc-file is:

```
[forcing]
Name          = arbitraryname_0001
Function      = timeseries
Time-interpolation = linear
Quantity       = time
Unit           = minutes since YYYY-MM-DD
Quantity       = riemannbnd
Unit           = m
[two-column data]
```

The data is to be inserted as a two-column array containing the time (in minutes) and the Riemann variant itself (in meters). In case of harmonic components, the header of the bc-file is:

```
[forcing]
Name          = arbitraryname_0001
Function      = harmonic
Quantity       = harmonic component
Unit           = minutes
Quantity       = riemannbnd amplitude
Unit           = m
Quantity       = riemannbnd phase
Unit           = degrees
[three-column data]
```

The data is to be inserted as a three-column array containing the period (in minutes), the amplitude (in meters per second) and the phase (in degrees).

Suppose, as an example, that we have:

- ◊ a flow domain with a bathymetry at a bed level equal to 0 m w.r.t. the reference level,
- ◊ an initial water level equal to 10 m w.r.t. the reference level,
- ◊ a local initial disturbance of the initial water level,
- ◊ and an initial flow field at rest,

and that we aim to prevent reflections at the boundary. In that case, it follows from [Equation 7.7](#) that $\zeta_b = 10$ m w.r.t. the reference level is to be prescribed. Remark that this application only holds for small disturbances of ζ from ζ_b .

Discharge-water level dependency

If a relation between the discharge and the local water level is known on beforehand, then this relation can be provided to the flow model as a table by means of the keyword qhtable. This table, provides the water level ζ as a function of the computed discharge Q .

If a Qh-table is applied to a the first support point of a polyline named arbitraryname, prescribed in some polyline file, then the header of the bc-file is:

```
[forcing]
Name          = arbitraryname_0001
```

```

Function      = qhtable
Quantity     = qhbnd discharge
Unit         = m3/s
Quantity     = qhbnd waterlevel
Unit         = m
[two-column data]

```

The data is to be inserted as a two-column array containing the discharge (in cubic meters per second) and the water *level* (in meters w.r.t. the reference level).

The user is able to apply a weighting parameter to compromise between the water level computed by the Qh-relation and the water level computed at the previous time step level. This parameter is accessible as the keyword `Qhrelax` in the MDU-file. By default, this parameter is 0.01. As a consequence, the newly computed water level consists of the Qh-relation result for 1% and for 99% of the previous time step water level.

7.4.1.3 Example

Recalling the four elementary actions from the beginning of [section 7.4.1](#), the administration in files can be illustrated by means of an example. Suppose, we have a channel that is geometrically modelled by means of the grid shown in [Figure 7.9](#). The domain is 10000 m long and 500 m wide. The bottom left corner coincides with the origin of the geometrical frame of reference.

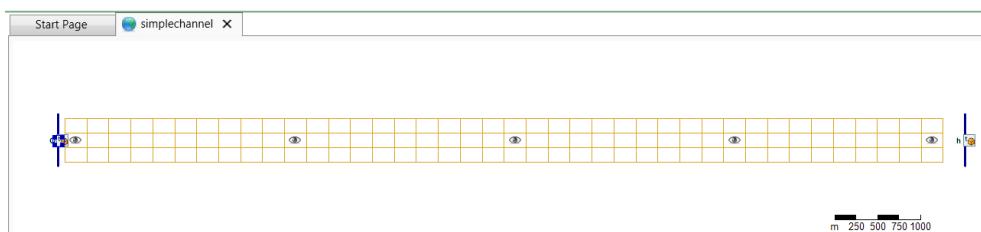


Figure 7.9: Delta-Shell view of a simple channel covered by a straightforward Cartesian grid. Boundary conditions are prescribed at the left hand side and the right hand size of the domain.

The domain shown in [Figure 7.9](#) contains two boundaries: on the left hand, a discharge boundary condition is present to present flow entering the domain, on the right hand, a water level boundary condition is set. Having two boundaries, we have two polyline files, in this example: `left.pli` and `right.pli`.

The contents of `left.pli` are:

```

boundaryleft
 2      2
 -80    -50
 -80    550

```

whereas the contents of `right.pli` are:

```

boundaryright
 2      2
10250    -50
10250    550

```

Notice that both polyline files contain the name of the polyline, namely `boundaryleft` and `boundaryright`, respectively. In this example, both the polylines contain *two* support points. For each of these support points, timeseries can be prescribed. In this example, we restrict ourselves to homogeneous boundary conditions. This means that we have to prescribe physical information for the first support points, i.e. for `boundaryleft_0001` and `boundaryright_0001`.

The physical information should be provided in the `bc`-file. In this example, we use the following `bc`-file:

```

[forcing]
Name          = boundaryleft_0001
Function       = timeseries
Time-interpolation = linear
Quantity        = time
Unit           = minutes since 2001-01-01
Quantity        = dischargebnd
Unit           = m3/s
 0.000000  2500.0
120.000000 3000.0
240.000000 2500.0
360.000000 2000.0
480.000000 2500.0
600.000000 3000.0
720.000000 2500.0
840.000000 2000.0
960.000000 2500.0
1080.000000 3000.0
1200.000000 2500.0
1320.000000 2000.0
1440.000000 2500.0

[forcing]
Name          = boundaryright_0001
Function       = timeseries
Time-interpolation = linear
Quantity        = time
Unit           = minutes since 2001-01-01
Quantity        = waterlevelbnd
Unit           = m
 0.000000  2.50
1440.000000 2.50

```

This file couples the timeseries for the discharge to the support point named `boundaryleft_0001`. Likewise, the water level boundary, being constant in time, is coupled to the support point named `boundaryright_0001`.

The final specification of the boundary conditions is wrapped up in the external forcing file, with extension `.ext`. In this file, the connection is laid between the quantity of the boundary condition, the name of the polyline file (in which the name of the polyline itself is given)

and the forcing file (in which the physical information is provided). In our example, the file `simplechannel.ext` has the following contents:

```
[boundary]
quantity      = dischargebnd
locationfile = left.pli
forcingfile  = simplechannel.bc

[boundary]
quantity      = waterlevelbnd
locationfile = right.pli
forcingfile  = simplechannel.bc
```

The final step is to let the model know that the external forcings file `simplechannel.ext` is the one that should be used. This is to be achieved in the MDU-file:

```
[external forcing]
ExtForceFile           =
ExtForceFileNew        = simplechannel.ext
```

Notice that the name is specified at the keyword `ExtForceFileNew`. The other keyword, `ExtForceFile`, should be kept empty, unless *deprecated* .tim-files and/or .cmp-files are used to prescribe the physical information for the boundaries.

7.4.1.4 Miscellaneous

In the previous sections, the most essential information on the application of boundary conditions is described. Some remaining aspects are discussed in this section.

Artificial boundary layers

Advection terms at the offshore boundary may generate an artificial boundary layer along the boundary. The advection terms containing normal gradients have to be switched off. This is done by utilizing the keyword `jacstbnd` in the MDU-file. By default this keyword `jacstbnd = 0`, keeping the functionality inactive. The keyword can be set to `jacstbnd = 1` to do otherwise.

Smoothing parameter boundary conditions

The solution of the shallow water equations is uniquely determined by a set of initial and boundary conditions. The boundary conditions represent the external forcing and determine the steady state solution. The deviation between the initial condition and the steady state solution generates a transient (mass spring system analogy).

In D-Flow FM, the initial conditions for the water level and velocities are obtained from:

- ◊ The results of a previous run (warm start).
- ◊ User-prescribed (space varying or uniform) input fields (cold start).

The initial values are usually inconsistent with the boundary conditions at the start time of the simulation. This will generate a transient solution consisting of waves with eigen frequencies

of the model domain. These waves may be reflected at the boundaries and generate a standing wave system. The waves should be dissipated completely by bottom friction and viscosity terms or leave the domain through the open boundaries. The damping of the transient solution determines the spin-up time of the numerical model.

To reduce the amplitude of the transient wave and the spin-up time of a model, D-Flow FM has an option to switch on the boundary forcing gradually by use of a smoothing period (parameter T_{smo}). With $F_i(t)$ the initial value at the boundary, $F_b(t)$ the boundary condition signal and $F_b^{smo}(t)$ the boundary condition after smoothing, the boundary forcing is given by:

$$F_b^{smo}(t) = \alpha F_i(t) + (1 - \alpha) F_b(t), \quad (7.8)$$

with:

$$\alpha = \frac{T_{smo} - t}{T_{smo}} \quad (7.9)$$

if $t < T_{smo}$. In case $t \geq T_{smo}$, then the smoothing parameter is set to zero: $\alpha = 0$.

Smoothing is possible both for a warm and a cold start. If the initial conditions are consistent with the boundary conditions at the start time of the simulation then the smoothing time should be set to zero.

Secondary boundary conditions

In [section 7.4.1.2](#), several types of boundary conditions are discussed. In addition, two types of boundary conditions can be applied on top of the canonical types: *normal* velocities and *tangential* velocities. The associated keyword are `normalvelocitybnd` and `tangentialvelocitybnd`, respectively.

7.4.2 Vertical boundary conditions

Vertical boundary conditions close the system of shallow water equations at the bed level and the free surface level. Please refer to the Delft3D-FLOW User Manual, Section 9.4.1.1.

7.4.3 Shear-stresses at closed boundaries

A closed boundary is situated at the transition between land and water. At a closed boundary, two boundary conditions have to be prescribed. One boundary condition has to do with the flow normal to the boundary and the other one with the shear-stress along the boundary. A closed sidewall is always considered as impermeable. For the shear stress along the boundary, the possible conditions to be prescribed are free-slip (zero tangential shear-stress), partial-slip and full-slip.

For large-scale simulations, the influence of the shear-stresses along closed boundaries can be neglected. Free slip is then applied for all closed boundaries. For simulations of smallscale flow (e.g. laboratory scale), the influence of the side walls on the flow may no longer be neglected. This option has been implemented for closed boundaries and dry points *but not for thin dams*. The reason is that the shear stress at a thin dam is dependent on the side (double valued).

Along the side walls, the tangential shear stress is calculated based on a logarithmic law of the wall. The friction velocity u_* is determined by the logarithmic law for a rough wall, with side wall roughness y_0 and the velocity in the first grid point near the side wall. Let Δy_s be the

grid size normal to the sidewall, then:

$$|\vec{u}_{\text{sidewall}}| = \frac{u_*}{\kappa} \ln \left(1 + \frac{\Delta y_s}{2y_0} \right) \quad (7.10)$$

The choice for the way tangial shear stress are computed can be inserted through the key `irov` in the MDU-file (under [physics]):

- ◊ the case `irov` = 0 treats the side walls as free-slip walls (default),
- ◊ the case `irov` = 1 treats the side walls as partial-slip walls,
- ◊ the case `irov` = 2 treats the side walls as no-slip walls.

If the choice for partial-slip walls is made, then a specific wall roughness value should be prescribed in the MDU-file. For this, the keyword `wall_ks` should be used (under [physics] in the MDU-file). The computational engine interpretes this value as Nikuradse roughness k_s . The value for y_0 is computed as $y_0 = k_s/30$.

7.5 Artificial mixing due to sigma-coordinates

The σ -transformation is boundary-fitted in the vertical. The bottom boundary and free surface are represented smoothly. The water column is divided into the same number of layers independent of the water depth. In a σ -model, the vertical resolution increases automatically in shallow areas.

For steep bottom slopes combined with vertical stratification, σ -transformed grids introduce numerical problems for the accurate approximation of horizontal gradients both in the baroclinic pressure term and in the horizontal diffusion term. Due to truncation errors artificial vertical mixing and artificial flow may occur, [Leendertse \(1990\)](#) and [Stelling and Van Kester \(1994\)](#). This artificial vertical transport is sometimes called "creep".

Let z_b be the position of the bed and H the total water depth. If we consider the transformation from Cartesian co-ordinates to σ co-ordinates, defined by:

$$x = x^*, \quad y = y^*, \quad \sigma = \frac{z - z_b}{H} \quad (H = \zeta - z_b) \quad (7.11)$$

as result $\sigma = 0$ at the bed level and $\sigma = 1$ at the free surface. The horizontal pressure gradient reads:

$$\frac{\partial p}{\partial x} = \frac{\partial p^*}{\partial x^*} \frac{\partial x^*}{\partial x} + \frac{\partial p^*}{\partial \sigma} \frac{\partial \sigma}{\partial x} = \frac{\partial p^*}{\partial x^*} - \frac{1}{H} \left(\frac{\partial z_b}{\partial x} + \sigma \frac{\partial H}{\partial x} \right) \frac{\partial p^*}{\partial \sigma}. \quad (7.12)$$

In case of vertical stratification near steep bottom slopes, small pressure gradients at the left-hand side may be the sum of relatively large terms with opposite sign at the right-hand side. Small truncation errors in the approximation of both terms result in a relatively large error in the pressure gradient. This artificial forcing produces artificial flow. The truncation errors depend on the grid sizes Δx and Δz . Observations of this kind has led to the notion of "hydrostatic consistency", see also Figure 7.10. In the notation used by [Haney \(1991\)](#) this consistency relation is given by:

$$\left| \frac{\sigma}{H} \frac{\partial H}{\partial x} \right| < \left| \frac{\partial \sigma}{\partial x} \right| \quad (7.13)$$

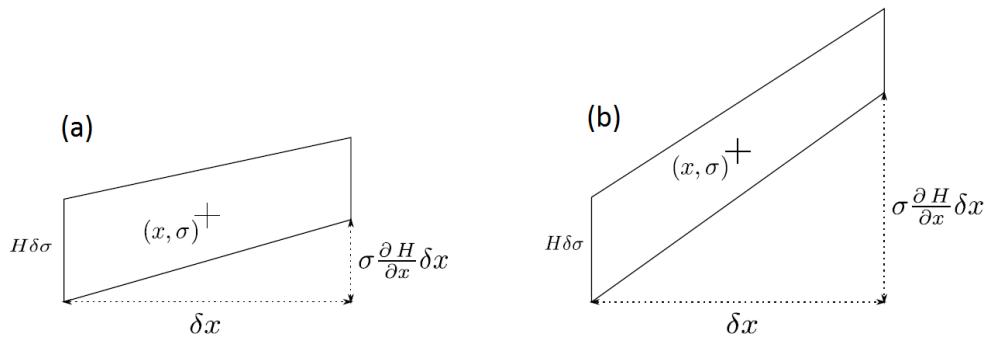


Figure 7.10: Example of a hydrostatic consistent and inconsistent grid; (a) $H\delta\sigma > \sigma\frac{\partial H}{\partial x}\delta x$, (b) $H\delta\sigma < \sigma\frac{\partial H}{\partial x}\delta x$

From this equation, it can be seen that by increasing the number of σ -levels the consistency condition will eventually be violated.

Similarly, for the horizontal diffusion term, the transformation from Cartesian co-ordinates to σ co-ordinates leads to various cross derivatives. For example, the transformation of a simple second order derivative leads to:

$$\frac{\partial^2 c}{\partial x^2} = \frac{\partial^2 c^*}{\partial x^{*2}} + \left(\frac{\partial \sigma}{\partial x}\right)^2 - \frac{\partial^2 c^*}{\partial \sigma^2} + 2\frac{\partial \sigma}{\partial x} - \frac{\partial^2 c^*}{\partial x^*\partial \sigma} + \frac{\partial^2 \sigma}{\partial x^2} - \frac{\partial c^*}{\partial \sigma} \quad (7.14)$$

For such a combination of terms it is difficult to find a numerical approximation that is stable and positive, see [Huang and Spaulding \(1996\)](#). Near steep bottom slopes or near tidal flats where the total depth becomes very small, truncations errors in the approximation of the horizontal diffusive fluxes in σ -co-ordinates are likely to become very large, similar to the horizontal pressure gradient.

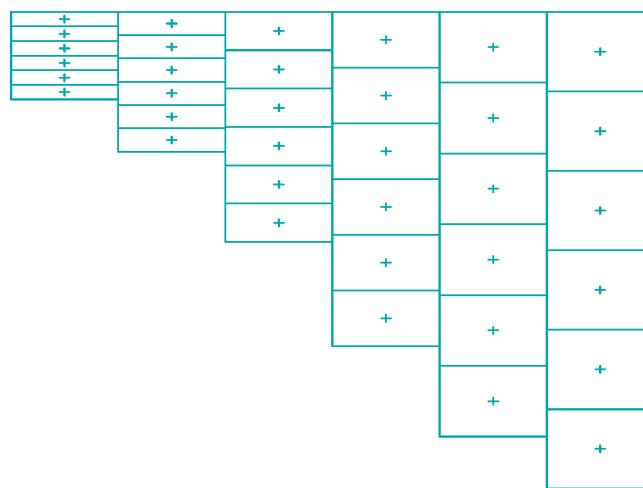


Figure 7.11: Finite Volume for diffusive fluxes and pressure gradients

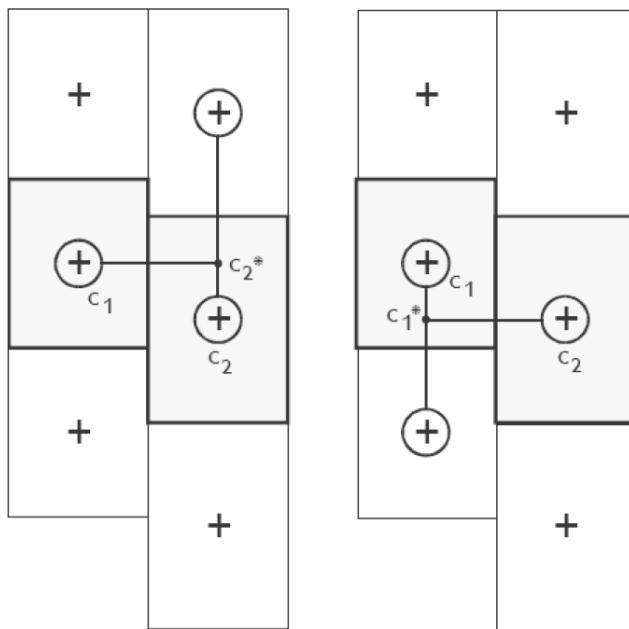


Figure 7.12: Left and right approximation of a strict horizontal gradient

In D-Flow FM the stress tensor is redefined in the σ co-ordinate system assuming that the horizontal length scale is much larger than the water depth (Blumberg and Mellor, 1985) and that the flow is of boundary-layer type. The horizontal gradients are taken along σ -planes. This approach guarantees a positive definite operator, also on the numerical grid (Beckers et al., 1998).

If the same approach is used for the horizontal diffusion operator in the transport equation:

$$\frac{\partial^2 c}{\partial x^2} \approx \frac{\partial^2 c^*}{\partial x^{*2}} \quad (7.15)$$

Horizontal diffusion will lead to vertical transport of matter through vertical stratification interfaces (pycnocline) which is nonphysical. A more accurate, strict horizontal discretization is needed.

In D-Flow FM an option is available that minimises artificial vertical diffusion and artificial flow due to truncation errors. A method has been implemented which gives a consistent, stable and monotonic approximation of both the horizontal pressure gradient and the horizontal diffusion term, even when the hydrostatic consistency condition equation is not fulfilled. This "anti-creep" option is based upon a Finite Volume approach; see Figure 7.11. The horizontal diffusive fluxes and baroclinic pressure gradients are approximated in Cartesian co-ordinates by defining rectangular finite volumes around the σ -co-ordinate grid points. Since these boxes are not nicely connected to each other, see Figure 7.12, an interpolation in z co-ordinates is required to compute the fluxes at the interfaces.

Since the centres of the finite volumes on the left-hand side and right-hand side of a vertical interval are not at the same vertical level, a z -interpolation of the scalar concentration c is needed to compute strictly horizontal derivatives. The values obtained from this interpolation are indicated by c_1^* and c_2^* respectively in Figure 7.12. (Stelling and Van Kester, 1994) apply a non-linear filter to combine the two consistent approximations of the horizontal gradient,

$$s_1 = (c_2^* - c_1) / \Delta x \text{ and } s_2 = (c_2 - c_1^*) / \Delta x$$

```

If  $s_1 \times s_2 < 0$  Then
   $\frac{\Delta c}{\Delta x} = 0$ 
Else
   $\frac{\Delta c}{\Delta x} = sign(s_1) \times \min(|s_1|, |s_2|)$ 
Endif

```

(7.16)

If an interval has only grid boxes at one side, the derivative is directly set to zero. The horizontal fluxes are summed for each control volume to compute the diffusive transport. The integration of the horizontal diffusion term is explicit with time step limitation:

$$\Delta t \leq \frac{1}{D_H} \left(\frac{1}{\Delta x^2} + \frac{1}{\Delta y^2} \right) \quad (7.17)$$

The derivatives are used in the integral for the baroclinic pressure force in the momentum equation:

$$P_x(x, z) = \frac{1}{\rho_0} \int_z^{\zeta} g \frac{\partial \rho(x, s)}{\partial x} ds \quad (7.18)$$

Originally, this approach was implemented in Delft3D-FLOW. [Slørdal \(1997\)](#) stated that the above approximation may sometimes produce errors of the same sign which leads to a systematic underestimation of the baroclinic pressure term. This underestimation can be ascribed to the non-linear filter, which selects the minimum of the two gradients under consideration. This limiter is fully analogous to the min-mod limiter used for the construction of monotone advection schemes ([Hirsch, 1990](#)). Since the same approximation of the horizontal gradient is used for the horizontal diffusion flux, it is important to ensure that the difference operator is positive definite in order to get physically realistic solutions. The maximum and minimum of a variable being transported by diffusion do not increase or decrease (min-max principle). By taking the minimum of the gradients, [Stelling and Van Kester \(1994\)](#) show that, the min-max principle is fulfilled. [Beckers et al. \(1998\)](#) show that any nine-point consistent linear discretization of the horizontal diffusion on the σ -grid does not fulfil the min-max principle. From numerical tests [Slørdal \(1997\)](#) concluded that the underestimation is reduced by increasing the vertical resolution, but is sometimes enhanced by increasing the horizontal resolution.

Let s_4 be a consistent approximation of the horizontal gradient $s_4 = (s_1 + s_2)/2$. [Slørdal \(1997\)](#) suggested to take s_4 as approximation of the horizontal gradient. He calls his approach the "modified Stelling and Van Kester scheme". It is equivalent to linear interpolation at a certain z -level before taking the gradient. It is more accurate than taking the minimum of the absolute value of the two slopes s_1 and s_2 but it does not fulfil the min-max principle for the diffusion operator. It may introduce wiggles and a small persistent artificial vertical diffusion (except for linear vertical density distributions). Due to the related artificial mixing, stratification may disappear entirely for long term simulations, unless the flow is dominated by the open boundary conditions.

By introducing an additional approximation of the horizontal gradient in the filter algorithm defined by $s_3 = (c_2 - c_1)/\Delta x$, the stringent conditions of the minimum operator can be relaxed somewhat. The drawback of underestimation of the baroclinic pressure force reported by [Slørdal \(1997\)](#) can be minimised without loosing that the method fulfils the min-max principle. This third gradient s_3 , which is consistent for $\min(|s_1|, |s_2|) < s_3 < \max(|s_1|, |s_2|)$, has point-to-point transfer properties and therefore leads to a positive scheme for sufficiently small time steps. The following non-linear approach presently available in D-Flow FM is both

consistent and assures the min-max principle:

```

If  $s_1 \times s_2 < 0$  Then
   $\frac{\Delta c}{\Delta x} = 0$ 
Elseif  $|s_4| < |s_3|$  Then
   $\frac{\Delta c}{\Delta x} = s_4$ 
Elseif  $\min(|s_1|, |s_2|) < |s_3| < \max(|s_1|, |s_2|)$  Then
   $\frac{\Delta c}{\Delta x} = s_3$ 
Else
   $\frac{\Delta c}{\Delta x} = \text{sign}(s_1) \min(|s_1|, |s_2|)$ 
Endif

```

(7.19)

The method requires a binary search to find the indices of neighbouring grid boxes, which is time consuming. The increase in computation time is about 30%.

If the streamlines are strictly horizontal, transport of matter discretised on a σ co-ordinate grid may still generate some numerical vertical diffusion by the discretisation of the advection terms.

7.6 Secondary flow

The flow in a river bend is basically three-dimensional. The velocity has a component in the plane perpendicular to the river axis. This component is directed to the inner bend near the riverbed and directed to the outer bend near the water surface, see Figure 7.13.

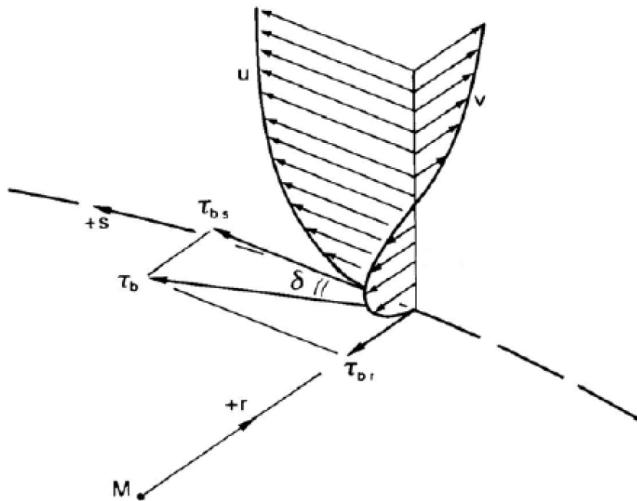


Figure 7.13: Vertical profile secondary flow (v) in river bend and direction bed stress

This so-called "secondary flow" (spiral motion) is of importance for the calculation of changes of the riverbed in morphological models and the dispersion of matter. In a 3D model the secondary flow is resolved on the vertical grid, but in 2D depth-averaged simulations the secondary flow has to be determined indirectly using a secondary flow model. It strongly varies over the vertical but its magnitude is small compared to the characteristic horizontal flow velocity.

7.6.1 Definition

The secondary flow will be defined here as the velocity component $v(\sigma)$ normal to the depth-averaged main flow. The spiral motion intensity of the secondary flow I is a measure for the magnitude of this velocity component along the vertical:

$$I = \int_0^1 |v(\sigma)| d\sigma \quad (7.20)$$

A vertical distribution for a river bend is given in [Figure 7.13](#). The spiral motion intensity I leads to a deviation of the direction of the bed shear stress from the direction of the depth-averaged flow and thus affects the bedload transport direction. This effect can be taken into account in morphological simulations.

The component of the bed shear stress normal to the depth-averaged flow direction τ_{br} reads:

$$\tau_{br} = -2\rho\alpha^2 \left(1 - \frac{\alpha}{2}\right) |\vec{u}| I \quad (7.21)$$

where α is defined in [Equation 7.32](#) and $|\vec{u}|$ is the magnitude of the depth-averaged velocity. To take into account the effect of the secondary flow on the depth-averaged flow, the depth-averaged shallow water equations have to be extended with:

- ◊ An additional advection-diffusion equation to account for the generation and adaptation of the spiral motion intensity.
- ◊ Additional terms in the momentum equations to account for the horizontal effective shear stresses originating from the secondary flow.

7.6.2 Depth-averaged continuity equation

The depth-averaged continuity equation is given by:

$$\frac{\partial h}{\partial t} + \frac{\partial hu}{\partial x} + \frac{\partial hv}{\partial y} = Q \quad (7.22)$$

where u and v indicate the depth-averaged velocities along Cartesian axis.

7.6.3 Momentum equations in horizontal direction

The momentum equations in x - and y -direction are given by:

$$\frac{\partial u}{\partial t} + u \frac{\partial u}{\partial x} + v \frac{\partial u}{\partial y} - fu = -\frac{1}{\rho_0} P_x - \frac{gu\sqrt{u^2 + v^2}}{C_{2D}^2 h} + F_x + F_{sx} + M_x \quad (7.23)$$

$$\frac{\partial v}{\partial t} + u \frac{\partial v}{\partial x} + v \frac{\partial v}{\partial y} - fv = -\frac{1}{\rho_0} P_v - \frac{gv\sqrt{u^2 + v^2}}{C_{2D}^2 h} + F_y + F_{sy} + M_y \quad (7.24)$$

The fourth term in the right-hand side represents the effect of the secondary flow on the depth-averaged velocities (shear stresses by depth-averaging the non-linear acceleration terms).

7.6.4 Effect of secondary flow on depth-averaged momentum equations

To account for the effect of the secondary flow on the depth-averaged flow, the momentum equations have to be extended with additional shear stresses. To close the equations these stresses are coupled to parameters of the depth-averaged flow field. The main flow is assumed to have a logarithmic velocity profile and the secondary flow originates from a multiplication of a universal function with the spiral motion intensity, see [Kalkwijk and Booij \(1986\)](#). Depth averaging of the 3D equations leads to correction terms in the depth-averaged momentum equations for the effect of spiral motion:

$$F_{sx} = \frac{1}{h} \left(\frac{\partial h T_{xx}}{\partial x} + \frac{\partial h T_{xy}}{\partial y} \right) \quad (7.25)$$

$$F_{sy} = \frac{1}{h} \left(\frac{\partial h T_{xy}}{\partial x} + \frac{\partial h T_{yy}}{\partial y} \right) \quad (7.26)$$

with the shear-stresses, resulting from the secondary flow, modelled as:

$$T_{xx} = -2\beta uv \quad (7.27)$$

$$T_{xy} = T_{yx} = \beta(u^2 - v^2) \quad (7.28)$$

$$T_{yy} = 2\beta uv \quad (7.29)$$

and:

$$\beta = \beta^* \frac{h}{R_s^*} \quad (7.30)$$

$$\beta^* = \beta_c (5\alpha - 15.6\alpha^2 + 37.5\alpha^3) \quad (7.31)$$

$\beta_c \in [0, 1]$, correction coefficient specified by you,

$$\alpha = \frac{\sqrt{g}}{\kappa C_{2D}} < \frac{1}{2} \quad (7.32)$$

with R_s^* the effective radius of curvature of a 2D streamline to be derived from the intensity of the spiral motion and κ the Von Kármán constant. The spiral motion intensity is computed by Equation 7.33. The limitation on α , Equation 7.32, is set to ensure that the length scale L_a in Equation 7.40 is always positive. For $\beta_c = 0$, the depth-averaged flow is not influenced by the secondary flow.

Remark:

- ◊ Equation 7.32 effectively means a lower limit on C_{2D} .

7.6.5 The depth averaged transport equation for the spiral motion intensity

The variation of the spiral motion intensity I in space and time, is described by a depth-averaged advection-diffusion equation:

$$\frac{\partial hI}{\partial t} + \frac{\partial uhI}{\partial x} + \frac{\partial vhI}{\partial y} = h \frac{\partial}{\partial x} \left(D_H \frac{\partial I}{\partial x} \right) + h \frac{\partial}{\partial x} \left(D_H \frac{\partial I}{\partial y} \right) + hS \quad (7.33)$$

with:

$$S = -\frac{I - I_e}{T_a} \quad (7.34)$$

$$I_e = I_{be} - I_{ce} \quad (7.35)$$

$$I_{be} = \frac{h}{R_s} |\vec{u}| \quad (7.36)$$

$$I_{ce} = f \frac{h}{2} \quad (7.37)$$

$$|\vec{u}| = \sqrt{u^2 + v^2} \quad (7.38)$$

$$T_a = \frac{L_a}{|\vec{u}|} \quad (7.39)$$

$$L_a = \frac{(1 - 2\alpha) h}{2\kappa^2 \alpha} \quad (7.40)$$

and R_s the radius of curvature of the stream-line defined by:

$$\frac{u_s}{R_s} = -\frac{\partial u_r}{\partial s} \quad (7.41)$$

with u_s and u_r the components along and perpendicular to the streamline. The effective radius of curvature to be used for the evaluation of the coefficient β , Equation 7.31, reads:

$$R_s^* = \frac{h |\vec{u}|}{I} \quad (7.42)$$

To guarantee stability the effective radius of curvature is bounded by the following empirical relation:

$$R_s^* \geq 10h \quad (7.43)$$

The above formulas account for two sources of secondary flow:

- ◊ The centrifugal force in case of curved streamlines, I_{be} .
- ◊ The effect of the Coriolis force, I_{ce} .

7.7 Drying and flooding

Estuaries and coastal embayments contain large, shallow, and relatively flat areas separated and interleaved by deeper channels and creeks. When water levels are high, the entire area is covered by water. But as tide falls, the shallow areas are exposed, and ultimately the flow is confined only to the deeper channels. The dry tidal flats may occupy a substantial fraction of the total surface area. The accurate reproduction of covering or uncovering of the tidal flats is an important feature of numerical flow models based on the shallow water equations.

Many rivers have compound channels, consisting of a main channel that always carries flow (the summer bed) and one or two flood plains which only convey flow during extreme river discharges (the winter bed). The summer bed is surrounded by low dykes, which could be overtapped if the river discharge increases. The winter-bed is surrounded by much higher dykes, which are designed to protect the polders against water levels due extreme river discharges. The flooding of the flood plains increases the drainage capacity of the river and reduces the local water level gradients.

In a numerical model, the process of drying and flooding is represented by removing grid points from the flow domain that become *dry* when the tide falls and by adding grid points that become *wet* when the tide rises. Drying and flooding is constrained to follow the sides of grid cells. In this section, we specify the algorithms which have been used to determine the moment when a grid cell (water level point) or cell boundary (velocity point) becomes dry or wet. Drying and flooding gives a discontinuous movement of the closed boundaries and may generate small oscillations in water levels and velocities. The oscillations introduced by the drying and flooding algorithm are small if the grid sizes are small and the bottom has smooth gradients.

Essential elements of the wetting and drying algorithm are the definition of the water level, the definition of the bed level and the criteria for setting a velocity and/or water level point wet or dry. In the following subsections, these three items will be discussed.

7.7.1 Definitions

In [section 7.3.1](#), the locations of the primary flow variables (water level and flow velocity) have been described. Through [Figure 7.3](#) it has been clarified that the water level is computed at the location of the circumcenter (cell center), whereas the face normal flow velocity is computed at the midpoint of each cell face (face center).

For the computation of these two primary variables, the level of the bed must be known both at the cell center and at the face center. The user can specify the way these values are interpreted from the available bathymetry by means of the MDU-file keywords `bedlevtyp` and `conveyance2D`. For a proper understanding of the possibilities of D-Flow FM, [Figure 7.14](#) is provided with a three-dimensional representation of two adjacent triangular cells.

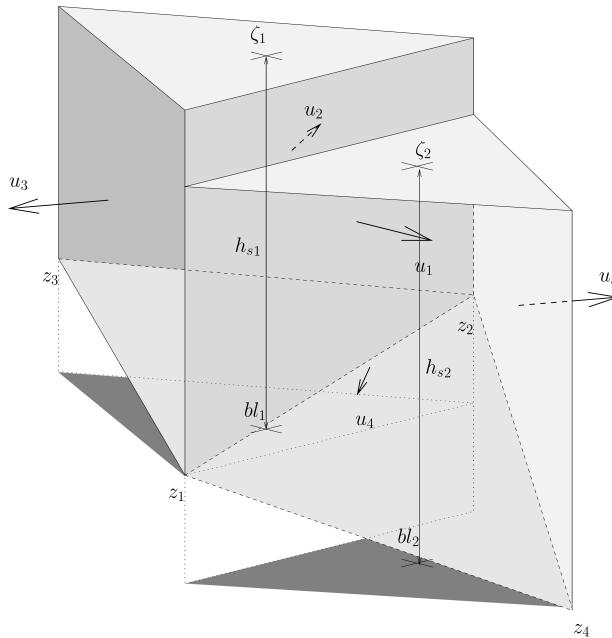


Figure 7.14: Definition of the water levels, the bed levels and the velocities in case of two adjacent triangular cells.

To the keyword `bedlevtyp`, the values 1, 2, 3, 4, 5 and 6 can be assigned. For the keyword `conveyance2D`, the values -1, 0, 1, 2 and 3 are available.

The most common case is the definition of the bed levels at the corner nodes of the cell and a choice for the keyword `bedlevtyp = 3`. Depending on the choice for `conveyance2D`, the face center bed levels and cell center bed levels are computed.

7.7.1.1 Piecewise constant approach for the bed level

In principle, the bed levels are considered as piecewise constant in space. This approach is followed if `conveyance2D < 1`. The keyword `bedlevtyp` can be used in combination with the piecewise constant approach as highlighted in the table below. Note that the bed level is defined with respect to the reference level (not to be confused with a water depth given as a positive value downwards). With `conveyance2D < 1`, we have:

Keyword	Value	Cell center bed level	Face center bed level
bedlevtyp	1	user specified	the highest cell center bed level considering the two cells next to the face
bedlevtyp	2	the lowest face center bed level considering all the faces of the cell	user specified
bedlevtyp	3	the lowest face center bed level considering all the faces of the cell	the mean corner bed level considering the two corner nodes the face is connecting
bedlevtyp	4	the lowest face center bed level considering all the faces of the cell	the minimum corner bed level considering the two corner nodes the face is connecting
bedlevtyp	5	the lowest face center bed level considering all the faces of the cell	the maximum corner bed level considering the two corner nodes the face is connecting
bedlevtyp	6	the mean corner bed level considering all the corners of the cell	the highest cell center bed level considering the two cells next to the face

The former Delft3D-FLOW version, running on curvilinear meshes, has utilized the piecewise constant approach for bed levels as well. The treatment of the bed level itself is, however, different from D-Flow FM. In Delft3D, the user can distinctly specify the treatment type for the cell center bed level and the face center bed level. In D-Flow FM, the choice for the one implies the choice for the other. A strict, *exact* match of settings for which Delft3D-FLOW and D-Flow FM treat the bed similarly, is not facilitated. Hence, the user himself should take care in comparing the Delft3D-FLOW results and D-Flow FM results when it comes to the bed level treatment settings.

7.7.1.2 Piecewise linear approach for the bed levels

A piecewise linear bed level approach can be chosen for through setting `conveyance2D ≥ 1`. In this case, only the approach for `bedlevtyp` equal to 3, 4 and 5 is affected. With `conveyance2D ≥ 1`, we have:

Keyword	Value	Cell center bed level	Face center bed level
bedlevtyp	3, 4, 5	the lowest corner node bed level considering all the nodes of the cell	linearly varying from the bed level at the one corner node to the bed level at the other corner node

Notice that the choice for either `bedlevtyp` equal to 3, 4 or 5 does not imply different bed level treatment approaches. For the case `conveyance2D ≥ 1`, the bed is assumed linearly

varying within a face only to compute the wet cross-sectional area of the vertical face; it should, however, be remarked that for the computation of the water column volume in a cell, this linear variation of the bed is *not* taken into account.

7.7.1.3 Hybrid bed level approach

In addition to the previously described bed level treatment approaches, D-Flow FM facilitates a hybrid approach for the computation of the cell center bed level by means of the keywords `blminabove` and `blmeanbelow`. In this approach, the cell center bed level is computed as the mean of the associated face center bed levels, be it only below the user specified level `blmeanbelow`. For levels above the user specified level `blminabove`, the minimum value of the associated face center bed levels is used. In between these two user specified levels, the bed levels are constructed by means of a linear interpolation between the two approaches' results.

7.7.2 Specification in Delta Shell

The specification of the treatment of the bed level locations can be achieved through the tab fields 'Numerical Parameters' and 'Physical Parameters'.

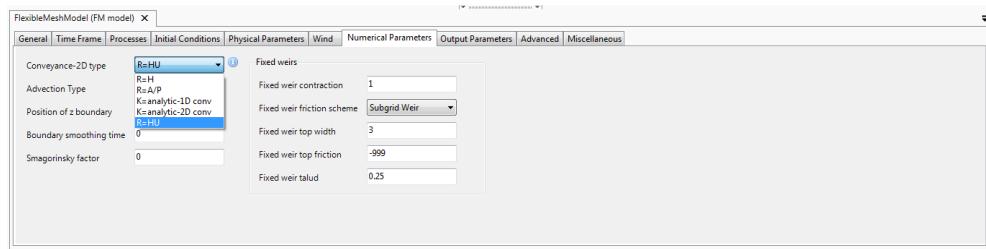


Figure 7.15: Specification of the conveyance option in Delta Shell.

The tab field 'Numerical Parameters' contains the choice for the conveyance options: see [Figure 7.16](#), present in the center of the screen.

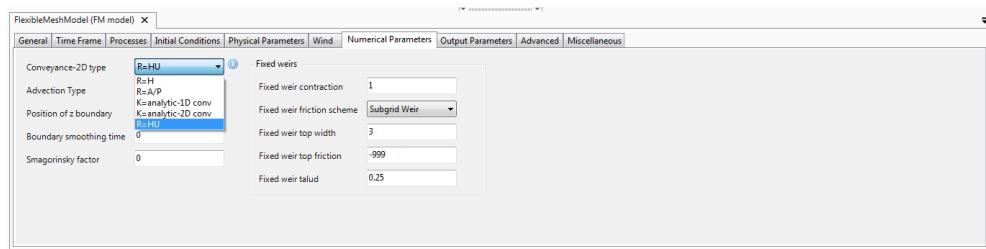


Figure 7.16: Specification of the bed level treatment type in Delta Shell.

In the user interface, five options are facilitated: `bedlevtyp` numbers 1 up to 5; the option 6 is not facilitated in the user interface.

By means of the tab field 'Miscellaneous', the hybrid options with the keywords `blminabove` and `blmeanbelow` can be enabled.

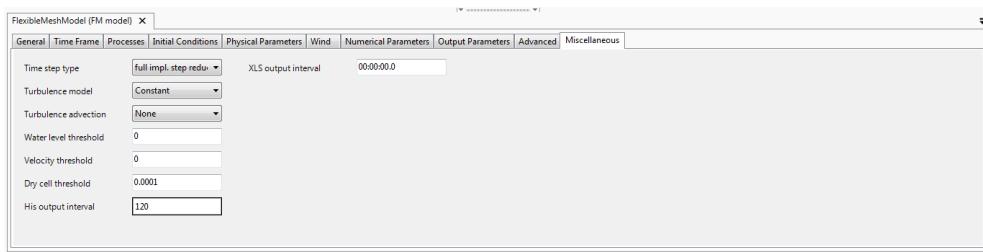


Figure 7.17: Specification of the hybrid bed options (with keywords `b1minabove` and `b1meanbelow`).

7.8 Intakes, outfalls and coupled intake-outfalls

Many engineering studies concern the design of intakes and outfalls. For instance, studies about positioning of waste water diffusers or coupled intake-outfall design for cooling water at power plants. Intakes and outfalls can be modeled using sinks and sources. A source is a point in the model where a discharge Q in [m^3/s] is prescribed by a time series ASCII file ([section D.1.6](#)) with four columns. The first column is the time in minutes, the second is the discharge in [m^3/s], the third column contains the salinity in [ppt] and the fourth contains the temperature in [$^\circ\text{C}$]. The four columns are always required in the file, even if salinity or temperature are not used in the computation.

The location of the source or sink is specified in a polyline file ([section B.2](#)), containing a polyline with either multiple points or just one point. If two or more points are specified, a coupled source sink pair is made, the first point is the FROM (or sink) point, the last point is the TO (or source) point. Three variants may occur:

- 1 Sink point side lies inside a grid cell A, source point also lies inside a grid cell B (A may equal B, but that is rarely useful): water is extracted from cell A and transported to B.
- 2 Sink point lies outside of the grid, source point lies in a grid cell B: water is discharged into B, a bit like an inflow discharge boundary condition.
- 3 Sink side lies inside a grid cell A, source side lies outside of the grid: water is extracted from A, a bit like an outflow discharge boundary condition.

Specifying a negative discharge value effectively interchanges the role of the source and sink points. If only one point is in the `<*.pli>` file, it is assumed to be a source point. Specifying a negative discharge turns the source into a sink.

For 3D computations, the polyline file should have a third column with z-values. It is good practice to change the `.pli` file into a `.pliz` file. The z-values are used to determine in which vertical grid cell the source and/or sink lie. The layer number can vary in time in sigma models.

In the case of a coupled pair of sink source points (variant 1), the third and fourth column with the salinity and temperature specification are interpreted as delta salinity and delta temperature. So the values at the source point become the values at the sink point plus the specified delta values.

The sources and sinks are specified in the `*.ext` file in a way similar to the boundary conditions:

```
QUANTITY=discharge_salinity_temperature_sorsin
FILENAME=chan1_westeast.pli # A file 'chan1_westeast.tim', with same basename
# as the polyline should be present
```

```
FILETYPE=9
METHOD =1
OPERAND =0
AREA    =1.5
```

The specified area in the last line of this file determines (together with the specified discharge) the amount of momentum $u \cdot Q$ that is released at the source point. This is currently only implemented in advection scheme 33 (cf. D-Flow FM Technical Reference Manual [D-Flow FM Technical Reference Manual \(2015\)](#) on Momentum advection). Omitting this line or specifying a zero area switches off the release of momentum at the source point. The direction of the discharged momentum is in direction of the last two points of the polyline. So a one point polyline is momentumless. Both in 2D and 3D, the momentum can only be directed in horizontal direction.

Sources and sinks are treated explicitly in the numerical scheme. This implies that the actual discharged or extracted amounts of water are limited by the velocity Courant condition $Q\Delta t/V < 1$. Not doing so could lead to severe timestep restrictions. Consider for instance a specified extraction in case the extraction point has fallen dry. In that case, a real pump would not be able to extract water and the specified extraction is more likely an input error than an actual description of a physically feasible situation. So, we limit the specified discharges to the local velocity Courant restriction. By specifying observation cross-sections ([section 4.4.2.3](#)), one can compare prescribed discharges to the discharges that were actually realised in the model. In case of large differences, the discharge should probably be distributed over a larger number of gridcells, or the extraction channel that feeds the extraction point should be dredged.

When outfalls are adjacent to a closed model boundary, one might consider specifying a discharge boundary instead of a point source. These are treated more implicitly in the numerical scheme and are preferable for that reason.

When modeling freshwater inflow from a river into a sea, the vertical distribution of the discharge that one should specify depends on the amount of detail available for modeling the saline water - fresh water interface. If the river is modeled with sufficient detail, and the river is well mixed at the upstream model boundary, the mixing process can be part of the modeling and the river can be discharged over the whole vertical. If the mixing process cannot be resolved in the model, for instance if the river is modeled as a point discharge adjacent to a closed boundary, make sure that the whole river is discharged into the top layer or in the top layers, depending on the estimated thickness of the fresh water plume at the discharge cell. If one would have distributed the river discharge over the whole vertical, the size of the fresh water plume would be underestimated because of too much mixing at the river discharge cell.

For example input files see example directories:

- f17_sources_sinks/c010_sourcesink_2D/
- f17_sources_sinks/c020_sourcesink_3D/

7.9 Equations of state for the density

The density of water ρ is a function of salinity (s) and temperature (t).

In D-Flow FM we copied the implementation from Delft3D of the formulation derived by [Eckart \(1958\)](#) that is based on a limited number of measurements dating from 1910 (only two salinities at 5 temperatures). In the original equation the pressure is present, but at low pressures

the effect on density can be neglected.

Eckart formulation

The Eckart formulation is given by ([Eckart, 1958](#)):

Range: $0 < t < 40 \text{ }^{\circ}\text{C}$, $0 < s < 40 \text{ ppt}$

$$\rho = \frac{1,000P_0}{\lambda + \alpha_0 P_0}, \quad (7.44)$$

where:

$$\lambda = 1779.5 + 11.25t - 0.0745t^2 - (3.80 + 0.01t) s, \quad (7.45)$$

$$\alpha_0 = 0.6980, \quad (7.46)$$

$$P_0 = 5890 + 38t - 0.375t^2 + 3s. \quad (7.47)$$

with the salinity s in [ppt] and the water temperature t in [$^{\circ}\text{C}$].

The keyword that selects the equation of state in the mdu file is called Idensform. The Eckart formulation is set as default (1). The influence of salinity and or temperature on water motion through the baroclinic pressure can be switched off by setting Idensform=0

7.10 Tide generating forces

Numerical models of tidal motion in coastal seas generally do not account for the direct local influence of the tide generating forces. The amount of water mass in these models is relatively small and the effect of these forces on the flow can be neglected. In that case, tidal motion can often be reproduced with sufficient accuracy just by prescribing the tidal forcing along open model boundaries.

In models covering larger seas or oceans, the contribution of the gravitational forces on the water motion increases considerably and can no longer be neglected. In a global model, open boundaries are absent and the tidal motion can only be induced by including tide generating forces. Because tidal forces only play a significant role in the larger models, and because the exact position of each computational point on earth must be known, we have implemented these forces only in combination with spherical coordinates. By default, tide generating forces is switched on in spherical models. You can switch it off by setting in the mdu file : Tidalforging=0

The tide generating forces originate from the Newtonian gravitational forces of the terrestrial system (Sun, Moon and Earth) on the water mass. The equilibrium tide is the tide that would result from the gravitational forces if a solid earth would be completely covered by an ocean of about 21.5 km deep, such that the wave propagation speed would match the speed of the celestial forces over the ocean surface. The interacting frequencies that result can be grouped as diurnal, semi diurnal and long periodic. The total number of tide generating frequencies that is evaluated in the computation is a tradeoff between required accuracy and computational cost. Per default, we use a set of 60 frequencies, similar to TRIWAQ.

A smaller set of 11 main frequencies as used in Delft3D can also be selected by setting in the MDU file :

```
Doodsonstart = 57.555
```

```
Doodsonstop = 275.555
Doodsoneps   = 0.030
```

The full set of 484 components can be selected by specifying:

```
Doodsonstart = 55.565
Doodsonstop  = 375.575
Doodsoneps   = 0.000
```

For the set of 60 components these numbers are:

```
Doodsonstart = 55.565
Doodsonstop  = 375.575
Doodsoneps   = 0.030
```

The earth itself is deformed by the celestial forces, this is called the solid earth tide. An estimate of this influence is based on the work of A.E.H. Love (1927), is included in the total tide generating potential.

We have gratefully applied the Fortran subroutines written by E.J.O. Schrama. Details on the implementation of tide generating forces can be found in his lecture notes [Schrama \(2007\)](#).

In order to save computation cost, the tidal potential is not computed for each computational point, but on a grid with a resolution of 1 degree in both the South-North and West-East direction.

The tidal and surge motions of the seas and ocean also deform the earth , which is called tidal loading. Next to that, the water at some point is attracted by all moving water elsewhere on the globe. This is called self attraction. The combined influence of these two processes is implemented in a beta version.

8 Transport of matter

8.1 Introduction

In D-Flow FM, transport is formulated as

$$\frac{d}{dt} \int_{V(t)} c dV + \int_{\partial V(t)} c(\vec{u} - \vec{v}) \cdot \vec{n} dS = \int_{\partial V(t)} (K \nabla c) \cdot \vec{n} dS + \int_{V(t)} s dV, \quad (8.1)$$

where $V(t)$ is a three-dimensional control volume, c is a concentration, \vec{u} the flow velocity field, \vec{v} the velocity of the (vertically) moving control volume, K is a diagonal matrix $K = \text{diag}(\kappa, \kappa, \kappa_z)$ with diffusion coefficients and s a source term. For two-dimensional (depth-averaged) flow, we obtain

$$\frac{\partial hc}{\partial t} + \nabla \cdot (h \vec{u} c) = \nabla \cdot (h \kappa \nabla c) + hs, \quad (8.2)$$

where h is the water depth. In case of three-dimensional (layer-averaged) flow, with Δz a layer height from $z_0(x, y, t)$ to $z_1(x, y, t)$, we obtain

$$\begin{aligned} \frac{\partial \Delta z c}{\partial t} + \nabla \cdot (\Delta z \vec{u} c) + [\omega_{z_1} c]_{z=z_1} - [\omega_{z_0} c]_{z=z_0} &= \nabla \cdot (\Delta z \kappa \nabla c) + \\ \left[\kappa_z \frac{\partial c}{\partial z} - \kappa \nabla z_1 \cdot \nabla c \right]_{z=z_1} - \left[\kappa_z \frac{\partial c}{\partial z} - \kappa \nabla z_0 \cdot \nabla c \right]_{z=z_0} &+ \Delta z s, \end{aligned} \quad (8.3)$$

where by \vec{u} and ∇ still the horizontal components are meant, i.e. $\vec{u} = (u, v)^t$ and $\nabla = (\frac{\partial}{\partial x}, \frac{\partial}{\partial y})^t$ and κ_z is the vertical diffusion coefficient. Furthermore ω_{z_0} and ω_{z_1} are the velocity component normal and relative to the moving $z = z_0$ and $z = z_1$ layer interfaces respectively. Note that taking $c = 1$ yields

$$\omega_{z_1} + \frac{\partial z_1}{\partial t} = \omega_{z_0} - \nabla \cdot (\Delta z \vec{u} c) + \frac{\partial z_0}{\partial t}, \quad (8.4)$$

which, combined with a zero-flux condition at the bed, recursively defines ω_{z_0} and ω_{z_1} for all layers.

We apply Eqns. (8.2) and (8.3) to transport of

- ◊ salinity,
- ◊ temperature,
- ◊ suspended sediment,
- ◊ tracers,
- ◊ other, intentionally not mentioned,

and ultimately to the water itself (the continuity equation) to obtain the relative layer interface velocities as expressed by Eqn. (8.4).

In the following we will highlight various physical and numerical settings in D-Flow FM. First and foremost, it is important to understand that there are *two* numerical implementations available, that differ slightly. These are selected with the keyword `transportmethod` in the `mdufile`:

```
transportmethod = 0
```

or

```
transportmethod = 1 # default.
```

The difference mainly concerns the treatment of vertical advection as explained in the next section. Note that tracers are only available with `transportmethod '1'`.

8.2 Some words about suspended sediment transport

In D-Flow FM two sediment models are available. The first is a genuine D-Flow FM implementation, while the second is adopted from Delft3D-SED and is only available in combination with `transportmethod '1'`. For a description of the latter, see Chapter 17 in this manual. We will not discuss the specifics of either of the two sediment models further and confine ourselves to mentioning the relevant settings that concern suspended sediment transport of the one of Chapter 17.

8.3 Transport processes

Looking at the equations that govern transport of matter, we can identify three processes, namely advection, diffusion and sources/sinks. The next sections will describe the relevant user settings.

8.3.1 Advection

Advection of matter is expressed by the term $\nabla \cdot (h\vec{u}c)$ in Eqn. (8.2) in the case of two-dimensional modeling. In three dimensions, we make a distinction between horizontal advection $\nabla \cdot (\Delta z \vec{u}c)$ and vertical advection $[\omega_{z_1} c]_{z=z_1} - [\omega_{z_0} c]_{z=z_0}$ in Eqn. (8.3). A higher-order numerical approximation of these terms can be obtained by setting their "limiter type" to an appropriate value. Setting the limiter type to "0" reduces the numerical approximation to a low-order upwind method (i.e. severe limiting). Although not restricted to the advection of salinity only, the choices for *all* matter are set with keyword `limtypsa` in the MDU file, for example

```
[numerics]
limtypsa = 0 # first-order upwind, or
```

or

```
[numerics]
limtypsa = 4 # MC limiter
```

For `transportmethod=1`, only the monotonized central (MC) limiter is available and used when `limtypsa>0`, for example 1 or 2 or 6, but not -1. For `transportmethod=0` other limiters are available as well and the MC limiter is selected by setting `limtypsa` to '4', which is its default value. We will not discuss the non-default limiters.

In three-dimensional modeling horizontal advection is similarly as in two dimension, but now vertical advection can be selected with the keyword `Vertadvtypsal` in the MDU file, i.e.

```
[numerics]
Vertadvtypsal = 0 # no vertical advection
```

or

```
[numerics]
Vertadvtypsal = 5 # default
```

The various combinations of `transportmethod` and `Vertadvtypsal` have the following meaning:

Vertadvtypsal	transport method		time
	space	0	
0	none	none	
1	1 st order upwind	forward Euler	
2	central	forward Euler	
3	1 st order upwind	θ -method	salt and temperature: central, other: higher-order
4	central	θ -method	θ -method, other: forward Euler
5	neg. stratification or water depth < <code>chkadvd</code> : 1 st order upwind, otherwise: central	θ -method	limited upwind
6	as 5	as 5	temperature: central, other: higher-order limited upwind
			temperature: θ -method, other: forward Euler

For the limited higher-order upwind discretization the MC limiter is used if `transportmethod=1` if `limtypsa>0`, regardless of its specific value, and 1st-order upwind if `limtypsa=0`.

8.3.2 Diffusion

Diffusion of matter is expressed by the term $\nabla \cdot (h\kappa \nabla c)$ for two-dimensional modeling and a similar and additional terms in three dimensions, see Eqn. (8.3). Clearly, we have horizontal diffusivity κ and vertical diffusivity κ_z . The horizontal diffusivity κ is a summation of

- ◊ the molecular diffusivity κ_l , only for `transportmethod=1`. We use the following values:

$$\kappa_l = \begin{cases} \frac{1}{700}\nu_l, & \text{salt,} \\ \frac{1}{6.7}\nu_l, & \text{temperature,} \\ 0\nu_l, & \text{sediment,} \\ 0\nu_l, & \text{tracers,} \end{cases} \quad (8.5)$$

where $\nu_l = 10^{-6} m^2/s$ is the kinematic viscosity,

- ◊ a background diffusivity, user-specified as

- spatially varying values by `horizontaleddydiffusivitycoefficient` in the extfile, or
- a uniform value `Dicouv` in the mdfile,

in that order of precedence, and

- ◊ a contribution from turbulent transport expressed as $\frac{1}{\sigma_t}\nu_t$, where ν_t is the eddy viscosity coefficient and σ_t is the turbulent Prandtl-Schmidt number for which the following values are used:

$$\sigma_t = \begin{cases} 0.7, & \text{salt,} \\ 0.7, & \text{temperature,} \\ 1.0, & \text{sediment,} \\ 1.0, & \text{tracers.} \end{cases} \quad (8.6)$$

Horizontal diffusion is turned off when $Dicouv=0$.

The user has to be aware of the following. The explicit nature of the horizontal diffusion terms in the time-integration method imposes a condition on the time step for numerical stability. Recall that we have a similar condition due to explicit horizontal advection. Although we always decrease the time step to satisfy the advection-related time step criterion, we do not do so for diffusion. Instead, diffusion is limited such that our time step is restricted by advection only, or by a user-specified maximum time step if it is smaller. For further details, consult the D-Flow FM Technical Reference Manual [D-Flow FM Technical Reference Manual \(2015\)](#).

Be aware that the modelled diffusion may be smaller than anticipated. However, the actual effective diffusion encountered may be (much) larger than anticipated due to numerical diffusion of the advection scheme.

Not considering suspended sediment, and similar to the horizontal diffusivity, the vertical diffusivity κ_z is a summation of

- ◊ the molecular diffusivity κ_l (for both transport methods), see Eqn. (8.5), not added for transport method '1' if $Dicoww=0$,
- ◊ a background vertical diffusivity, user-specified as a uniform value $Dicoww$ in the mdfile, and
- ◊ a contribution from turbulent transport, similar to its horizontal counterpart, but with the horizontal viscosity coefficient now replaced by the vertical (eddy) viscosity coefficient.

Time integration is performed with a method that does not impose an additional constraint on the time step. Unlike horizontal diffusion which is limited to ensure numerical stability while maintaining the time-step size, vertical diffusion is *not* limited.

Vertical diffusion is turned off when, again not considering suspended sediment:

- ◊ for transport method '0': $Dicoww=0$ or $Vertadvtypsal=1$ or $Vertadvtypsal=2$,
- ◊ for both transport methods: the water depth is smaller than a threshold $10^{-2} m$.

For the vertical diffusivity of suspended sediment, see Chapter 17.

8.3.3 Sources and sinks

Sources and sinks may be provided by an entry in the extfile as follows:

```
quantity=discharge_salinity_temperature_sorsin
```

This simultaneously prescribes sources and sinks of

- ◊ water volume itself (i.e. discharge),
- ◊ salinity, and
- ◊ temperature.

See Section 7.8 for more details.

8.3.4 Forester filter

The central vertical advection schemes of transport method '0' may cause nonphysical oscillations, or wiggles, especially near regions of large gradients. The user has the option to

suppress salt and temperature wiggles with a filter inspired by the so-called Forester filter. This filter also penalizes physically unstable stratification. The maximum number of iterations in the filter is controlled with keywords `Maxitverticalforestersal` for salt (default 100) and `Maxitverticalforestertem` for temperature (default 0, i.e. no filtering). Note that the filter is unavailable when using transport method '1'.

8.4 Transport boundary and initial conditions

The equations that govern transport of matter are complemented with boundary and initial conditions. We make the distinction between "horizontal" boundaries, that are either "open" or "closed", and vertical boundaries, only relevant for three-dimensional modeling.

8.4.1 Open boundary conditions

At "horizontal" open boundaries the following boundary conditions are applied:

- ◊ salt, temperature and tracers:
 - inflow: user-specified Dirichlet condition,
 - outflow: homogeneous Neumann condition,
- ◊ suspended sediment: see Chapter 17.

The user-specified Dirichlet conditions are supplied in the usual manner through the extfile, i.e.

```
quantity=salinitybnd
```

for salt and

```
quantity=temperaturebnd
```

for temperature. Boundary conditions for multiple tracers may be defined by appending their name to the `tracerbnd` keyword, for example

```
quantity=tracerbndMY_FIRST_TRACER
```

and

```
quantity=tracerbndMY_SECOND_TRACER
```

8.4.2 Closed boundary conditions

At "horizontal" closed boundaries zero-flux conditions are applied.

8.4.3 Vertical boundary conditions

At the "vertical" boundaries, i.e. at the bed and at the water surface, zero-flux conditions are applied, except for temperature that is, see Chapter 10 for further details on that matter.

8.4.4 Thatcher-Harleman boundary conditions

Consider (a part of) an open boundary where the flow reverts from outflowing to inflowing. According to Section 8.4.1, at that very moment a Dirichlet condition becomes effective and a user-specified boundary value is prescribed. This value does in general not reflect the true condition at the boundary and causes a discontinuous temporal behaviour. The so-called Thatcher-Harleman boundary condition is intended to regularize this behaviour. The actual value applied at the boundary is smoothly transformed from the last value under outflow conditions to the user-specified value under inflow conditions within a user-specified *return time*. This return time is prescribed with the keyword `return_time` in the “new style” external forcings file for boundary condition, for example

```
[boundary]
quantity      = salinitybnd
locationfile  = tfl_01.pli
forcingfile   = tfl.bc
return_time   = 250
```

See Section B.5 for more details on this format. Note that the Thatcher-Harleman boundary conditions are only available currently for two-dimensional modeling.

8.4.5 Initial conditions

We will only consider initial conditions for salinity, temperature and tracers. For initial sediment concentrations, see Chapter 17.

Initial conditions are specified in three possible ways, namely

- ◊ a horizontally spatially varying field in the usual way through the extfile,
- ◊ a vertical profile in three dimensions, horizontally uniformly distributed, for salinity and temperature only in the extfile, and
- ◊ uniform values for salinity and temperature in the mdufile.

In the extfile we have

```
quantity      = initialsalinity
```

for the initial salinity, and

```
quantity      = initialsalinitytop
```

for the initial salinity in the top layer in case of three-dimensional modeling. When specified, the initial salinity field is linearly distributed from the "initialsalinity" in the lowest layer to the "initialsalinitytop" in the top layer. When *not* specified, the initial salinity field is vertically uniformly distributed.

The initial temperature field is prescribed with

```
quantity      = initialtemperature
```

which in three dimensions is vertically uniformly distributed.

A horizontally uniformly distributed vertical profile of salinity and temperature can be prescribed with `initialverticalsalinityprofile` and `initialverticaltemperatureprofile` respectively, for example for salinity

```
QUANTITY=initialverticalsalinityprofile
FILENAME=inisal.pol
FILETYPE=10
METHOD=4
OPERAND=0
```

The polygon file contains $(z, \text{salinity})$ value pairs, where z is the vertical coordinate in meters. For example for linearly varying salinity from 30 to 20 ppt from -10 to 0 m:

```
L1
2 2
-10 30
0 20
```

The initial field of an arbitrary number of tracers are prescribed in the same way, except for the user-specified tracername that is added to the keyword `initialtracer`, similar to the tracer boundary conditions, for example

```
quantity=initialtracerMY_FIRST_TRACER
```

and

```
quantity=initialtracerMY_SECOND_TRACER
```

Default values for salinity and temperature are defined in the mdfile with `Initialsalinity` and `Initialtemperature` respectively.

9 Turbulence

This chapter is an almost integral copy of the Delft3D user manual.

The main difference is that in Delft3D many turbulence models are discussed, whereas in D-Flow FM we only implemented the $k-\varepsilon$ turbulence closure model.

D-Flow FM solves the Navier-Stokes equations for an incompressible fluid. Usually the grid (horizontal and/or vertical) is too coarse and the time step too large to resolve the turbulent scales of motion. The turbulent processes are “sub-grid”. The primitive variables are space- and time-averaged quantities. Filtering the equations leads to the need for appropriate closure assumptions.

For 3D shallow water flow the stress and diffusion tensor are an-isotropic. The horizontal eddy viscosity coefficient ν_H and eddy diffusivity coefficient D_H are much larger than the vertical coefficients ν_V and D_V , i.e. $\nu_H \gg \nu_V$ and $D_H \gg D_V$. The horizontal coefficients are assumed to be a superposition of three parts:

- 1 a part due to molecular viscosity.
- 2 a part due to “2D-turbulence”,
- 3 a part due to “3D-turbulence” see [Uittenbogaard et al. \(1992\)](#) and

The 2D part is associated with the contribution of horizontal motions and forcings that cannot be resolved (“sub-grid scale turbulence”) by the horizontal grid (Reynolds averaged or eddy resolving computations). The 3D part is referred to as the three-dimensional turbulence and is computed following one of the turbulence closure models, described in this section. For 2D depth-averaged simulations, the horizontal eddy viscosity and eddy diffusivity coefficient should also contain a contribution due to the vertical variation of the horizontal flow (Taylor shear dispersion).

The background horizontal viscosity coefficient ν_H^{back} and eddy diffusivity coefficient D_H^{back} (constant or space-varying) can be specified in the Delta Shell GUI and D-Flow FM’s MDU file. In Delft3D-FLOW a sub-grid scale model, HLES for 2D-turbulence is implemented. (see Delft3D-FLOW User Manual). In D-Flow FM we have not yet achieved this, but we implemented a simple horizontal model, the so called Smagorinsky model, so that we can at least cope with possibly very large grid size variations.

The horizontal eddy coefficients are typically an order of magnitude larger than the vertical coefficients determined by the turbulence closure model.

In D-Flow FM, two turbulence closure models have been implemented to determine ν_V and D_V :

- 1 $k-\varepsilon$ turbulence closure model.
- 2 $k-\tau$ turbulence closure model. (in beta version)

The $k-\tau$ model is a mathematical variant of the $k-\varepsilon$ model. Both models solve equations for production, dissipation and transport of turbulent kinetic energy k . In the $k-\varepsilon$ model the dissipation rate of turbulent kinetic energy ε , is modeled, whereas in the $k-\tau$ model the dissipation timescale τ is modeled.

Both models are based on the so-called eddy viscosity concept of [Kolmogorov \(1942\)](#) and [Prandtl \(1945\)](#).

A brief description of each of these turbulence closure model will be given further on in this section, for more details we refer to [Uittenbogaard et al. \(1992\)](#). The k - ε model has been used in tidal simulations by [Baumert and Radach \(1992\)](#) and [Davies and Gerritsen \(1994\)](#), for stratified flow of a river plume by [Postma et al. \(1999\)](#) and for the evolution of a thermocline by [Burchard and Baumert \(1995\)](#).

For strongly stratified flows it is important to introduce suitably chosen constant ambient (background) mixing coefficients, because the mixing coefficients computed by turbulence models with shear production only, reduce to zero. In the latter situation the vertical layers are completely de-coupled (frictionless). Disturbances are hardly damped and also the erosion of the vertical stratification is reduced to molecular diffusion.

Based on our experience with highly stratified flows we suggest applying an ambient or background vertical eddy viscosity in the order of 10^{-4} m²/s for the vertical exchange of momentum. This value corresponds with field measurements in the Rotterdam Waterway, The Netherlands.

Eddy diffusivity

The vertical eddy diffusivity is a scaled form of the eddy viscosity according to:

$$D_{3D} = \frac{\nu_{3D}}{\sigma_c}. \quad (9.1)$$

Parameter σ_c is the Prandtl-Schmidt number. Its numerical value depends on the substance c .

In Delft3D-FLOW the following settings of σ_c are used:

- ◊ In all cases, regardless the turbulence closure model, $\sigma_c = 0.7$ for the transport of heat, salinity, and tracer concentrations. For suspended sediment concentrations in online sediment transport computations, $\sigma_c = 1.0$.
- ◊ For the transport of turbulent kinetic energy k in the k - L model and k - ε model $\sigma_c = 1.0$, and for the transport of turbulent kinetic energy dissipation ε in the k - ε model $\sigma_c = 1.3$.

In the mathematical formulation, the fluxes are instantaneously influenced by changes in the vertical gradients of velocity and density. A physical adjustment time of the turbulence to the variations of the vertical gradients, is not taken into account. The fluxes are not a monotone function of the gradients. For the transport equation of heat, for small temperature gradients the heat flux increases when the temperature gradient increases but for large temperature gradients the heat flux decreases because the vertical eddy diffusivity is damped. For large values of the density gradients and small values of the velocity gradients, the vertical diffusion equation becomes mathematically ill-posed [Barenblatt et al. \(1993\)](#), and the computed vertical profiles may become discontinuous (stepwise). The number of “steps” is dependent on the vertical grid.

The numerical scheme for the vertical advection of heat and salt (central differences) may introduce small vertical oscillations. This computational noise may enhance the turbulent mixing. D-Flow FM has a vertical filtering technique to remove this noise and to reduce the undesirable mixing. For more details, see [section 8.3.4](#).

In strongly-stratified flows, the turbulent eddy viscosity at the interface reduces to zero and the vertical mixing reduces to molecular diffusion. To account for the vertical mixing induced by shearing and breaking of short and random internal gravity waves, we suggest to apply an

ambient eddy diffusivity in the order of 10^{-4} to 10^{-5} m²/s dependent on the Prandtl-Schmidt number. In Delft3D-FLOW for stable stratified flows, the minimal eddy diffusivity may be based on the Ozmidov length scale L_{oz} , specified by you and the Brunt-Väisälä frequency of internal waves:

$$D_V = \max \left(D_{3D}, 0.2 L_{oz}^2 \sqrt{-\frac{g}{\rho} \frac{\partial \rho}{\partial z}} \right). \quad (9.2)$$

This feature is still to be implemented in D-Flow FM

For a detailed description of the turbulence closure models of Delft3D-FLOW we refer to [Rodijns et al. \(1984\)](#) and [Uittenbogaard et al. \(1992\)](#).

9.1 $k-\varepsilon$ turbulence model

In the $k-\varepsilon$ turbulence model, transport equations must be solved for both the turbulent kinetic energy k and for the energy dissipation ε . The mixing length L is then determined from ε and k according to:

$$L = c_D \frac{k \sqrt{k}}{\varepsilon}. \quad (9.3)$$

In the transport equations, the following two assumptions are made:

- ◊ The production, buoyancy, and dissipation terms are the dominating terms.
- ◊ The horizontal length scales are larger than the vertical ones (shallow water, boundary layer type of flows).

Because of the first assumption, the conservation of the turbulent quantities is less important and the transport equation is implemented in a non-conservation form.

The transport equations for k and ε are non-linearly coupled by means of their eddy diffusivity D_k , D_ε and the dissipation terms. The transport equations for k and ε are given by:

$$\begin{aligned} \frac{\partial k}{\partial t} + \frac{u}{\sqrt{G_{\xi\xi}}} \frac{\partial k}{\partial \xi} + \frac{v}{\sqrt{G_{\eta\eta}}} \frac{\partial k}{\partial \eta} + \frac{\omega}{\zeta - z_b} \frac{\partial k}{\partial \sigma} &= + \\ + \frac{1}{(\zeta - z_b)^2} \frac{\partial}{\partial \sigma} \left(D_k \frac{\partial k}{\partial \sigma} \right) + P_k + P_{kw} + B_k - \varepsilon, \end{aligned} \quad (9.4)$$

$$\begin{aligned} \frac{\partial \varepsilon}{\partial t} + \frac{u}{\sqrt{G_{\xi\xi}}} \frac{\partial \varepsilon}{\partial \xi} + \frac{v}{\sqrt{G_{\eta\eta}}} \frac{\partial \varepsilon}{\partial \eta} + \frac{\omega}{\zeta - z_b} \frac{\partial \varepsilon}{\partial \sigma} &= \\ + \frac{1}{(\zeta - z_b)^2} \frac{\partial}{\partial \sigma} \left(D_\varepsilon \frac{\partial \varepsilon}{\partial \sigma} \right) + P_\varepsilon + P_{\varepsilon w} + B_\varepsilon - c_{2\varepsilon} \frac{\varepsilon^2}{k}. \end{aligned} \quad (9.5)$$

with

$$D_k = \frac{\nu_{mol}}{\sigma_{mol}} + \frac{\nu_{3D}}{\sigma_k} \quad \text{and} \quad D_\varepsilon = \frac{\nu_{3D}}{\sigma_\varepsilon} \quad (9.6)$$

In the production term P_k of turbulent kinetic energy, the horizontal gradients of the horizontal velocity and all the gradients of the vertical velocities are neglected. The production term is given by:

$$P_k = \nu_{3D} \frac{1}{(\zeta - z_b)^2} \left[\left(\frac{\partial u}{\partial \sigma} \right)^2 + \left(\frac{\partial v}{\partial \sigma} \right)^2 \right]. \quad (9.7)$$

For small-scale applications (e.g. simulation of laboratory flume), you can switch on a more extended production term P_k of turbulent kinetic energy (option “partial slip”, rough side wall) given by:

$$\begin{aligned} P_k = & 2\nu_{3D} \left[\frac{1}{2(\zeta - z_b)^2} \left\{ \left(\frac{\partial u}{\partial \sigma} \right)^2 + \left(\frac{\partial v}{\partial \sigma} \right)^2 \right\} + \left(\frac{1}{\sqrt{G_{\xi\xi}}} \frac{\partial u}{\partial \xi} \right)^2 \right] + \\ & + 2\nu_{3D} \left[\frac{1}{2} \left(\frac{1}{\sqrt{G_{\eta\eta}}} \frac{\partial u}{\partial \eta} + \frac{1}{\sqrt{G_{\xi\xi}}} \frac{\partial v}{\partial \xi} \right)^2 + \left(\frac{1}{\sqrt{G_{\eta\eta}}} \frac{\partial v}{\partial \eta} \right)^2 \right]. \end{aligned} \quad (9.8)$$

In this expression, ν_{3D} is the vertical eddy viscosity, prescribed by [Equation 9.16](#). In Eqs. 9.7 and 9.8 it has been assumed that the gradients of the vertical velocity w can be neglected with respect to the gradients of the horizontal velocity components u and v . The horizontal and vertical (σ -grid) curvature of the grid has also been neglected.

The turbulent energy production due to wave action is given by P_{kw} , but has not been implemented yet in D-Flow FM: Wave forcing in 3D models is being prepared for an upcoming release.

Near the closed walls the normal derivative of the tangential velocity is determined with the law of the wall:

$$\frac{\partial u}{\partial y} = \frac{u_*}{\kappa y}. \quad (9.9)$$

In stratified flows, turbulent kinetic energy is converted into potential energy. This is represented by a buoyancy flux B_k defined by:

$$B_k = \frac{\nu_{3D}}{\rho \sigma_\rho} \frac{g}{H} \frac{\partial \rho}{\partial \sigma} \quad (9.10)$$

with the Prandtl-Schmidt number $\sigma_\rho = 0.7$ for salinity and temperature and $\sigma_\rho = 1.0$ for suspended sediments.

The production term P_ε and the buoyancy flux B_ε are defined by:

$$P_\varepsilon = c_{1\varepsilon} \frac{\varepsilon}{k} P_k, \quad (9.11)$$

$$B_\varepsilon = c_{1\varepsilon} \frac{\varepsilon}{k} (1 - c_{3\varepsilon}) B_k, \quad (9.12)$$

with L prescribed by [Equation 9.3](#) and the calibration constants by ([Rodi, 1984](#)):

$$c_{1\varepsilon} = 1.44, \quad (9.13)$$

$$c_{2\varepsilon} = 1.92, \quad (9.14)$$

$$c_{3\varepsilon} = \begin{cases} 0.0 & \text{unstable stratification} \\ 1.0 & \text{stable stratification} \end{cases} \quad (9.15)$$

In D-Flow FM in the ε -equation for stable stratification the buoyancy flux is switched off, so $c_{3\varepsilon} = 1.0$ and for unstable stratification the buoyancy flux is switched on $c_{3\varepsilon} = 0.0$.

The energy production and energy dissipation due to waves, the terms P_{kw} and $P_{\varepsilon w}$ in Eqs. 9.4 and 9.5, have not been implemented yet in D-Flow FM: Wave forcing in 3D models is being prepared for an upcoming release.

The coefficients of the 3D k - ε turbulence closure model as implemented in D-Flow FM are not the same as in the depth-averaged k - ε turbulence closure model (Rodi, 1984), therefore for depth-averaged simulations, the k - ε turbulence closure model is not available for you.

The vertical eddy viscosity ν_{3D} is determined by:

$$\nu_{3D} = c'_\mu L \sqrt{k} = c_\mu \frac{k^2}{\varepsilon}, \quad (9.16)$$

with:

$$c_\mu = c_D c'_\mu. \quad (9.17)$$

To solve the transport equation, boundary conditions must be specified. A local equilibrium of production and dissipation of kinetic energy is assumed at the bed which leads to the following Dirichlet boundary condition:

$$k|_{\sigma=-1} = \frac{u_{*b}^2}{\sqrt{c_\mu}}. \quad (9.18)$$

The friction velocity u_{*b} at the bed is determined from the magnitude of the velocity in the grid point nearest to the bed, under the assumption of a logarithmic velocity profile. The bed roughness (roughness length) may be enhanced by the presence of wind generated short crested waves.

In case of wind forcing, a similar Dirichlet boundary condition is prescribed for the turbulent kinetic energy k at the free surface:

$$k|_{\sigma=0} = \frac{u_{*s}^2}{\sqrt{c_\mu}}. \quad (9.19)$$

In the absence of wind, the turbulent kinetic energy k at the surface is set to zero.

At open boundaries, the turbulent energy k is computed using the equation for k without horizontal advection. For a logarithmic velocity profile this will approximately lead to the following linear distribution based on the shear-stress at the bed and at the free surface:

$$k(z) = \frac{1}{\sqrt{c_\mu}} \left[u_{*b}^2 \left(1 - \frac{z+d}{H} \right) + u_{*s}^2 \frac{z+d}{H} \right]. \quad (9.20)$$

The k - ε turbulence model was successfully applied for the simulation of stratified flow in the Hong Kong waters (Postma et al., 1999) and verified for the seasonal evolution of the thermocline (Burchard and Baumert, 1995).

10 Heat transport

This chapter is an almost integral copy of the Delft3D-FLOW manual. The difference is that in Delft3D-FLOW five heat flux models are implemented, whereas in D-Flow FM only two models are implemented. These are the most complete heat flux model, the so called Ocean heat flux model (model nr 5 in Delft3D) and the most simple model, the Excess temperature model (model nr 3 in Delft3D). In D-Flow FM, the parameter that sets the temperature model is called Temperaturemodel in the mdu file. We kept the numbering of Delft3D. When specifying Temperaturemodel=1, the temperature is taken into account in the transport solver and in the equation of state, but heat fluxes through the water surface are not taken into account. This may be useful when mixing is the primary factor that determines the temperature distribution.

The heat radiation emitted by the sun reaches the earth in the form of electromagnetic waves with wavelengths in the range of 0.15 to 4 μm . In the atmosphere the radiation undergoes scattering, reflection and absorption by air, cloud, dust and particles. On average neither the atmosphere nor the earth accumulates heat, which implies that the absorbed heat is emitted back again. The wavelengths of these emitted radiations are longer (between 4 and 50 μm) due to the lower prevailing temperature in the atmosphere and on Earth. Schematically the radiation process, along with the heat flux mechanisms at the water surface, is shown in [Figure 10.1](#).

Legend for [Figure 10.1](#):

Q_{sc}	radiation (flux) for clear sky condition in [J/m ² s]
Q_{co}	heat loss due to convection (sensible) in [J/m ² s]
Q_{sr}	reflected solar radiation in [J/m ² s]
Q_s	solar radiation (short wave radiation) in [J/m ² s]
Q_{sn}	net incident solar radiation (short wave), = $Q_s - Q_{sr}$
Q_a	atmospheric radiation (long wave radiation) in [J/m ² s]
Q_{an}	net incident atmospheric radiation (long wave)
Q_{ar}	reflected atmospheric radiation in [J/m ² s]
Q_{br}	back radiation (long wave radiation) in [J/m ² s]
Q_{ev}	heat loss due to evaporation (latent) in [J/m ² s]

In DFM the heat exchange at the free surface is modeled by taking into account the separate effects of solar (short wave) and atmospheric (long wave) radiation, and heat loss due to back radiation, evaporation and convection. The heat losses due to evaporation and convection are functions of the wind speed. In absence of wind, these terms become zero. However, since water vapor is lighter than air, water may be cooled by evaporation and convection even in a no wind situation. The terms are called Q_{evfree} and Q_{cofree} respectively.

Excess temperature model - heat flux model 3

In the Excess temperature model the heat exchange flux at the air-water interface is computed based upon the prescribed background air temperature, the computed water temperature of the top layer and the prescribed wind speed. This relatively simple model is sometimes used in intake-outfall design studies. It can be applied when the temperature mixing process itself is more relevant than the actual heat loss through the air water interface. The applied heat exchange coefficient is mainly a function of the windspeed and water surface temperature.

The excess temperature model 3 is based on [Sweers \(1976\)](#), the heat exchange flux is represented by a bulk exchange formula:

$$Q_{tot} = -\lambda (T_s - T_{back}), \quad (10.1)$$

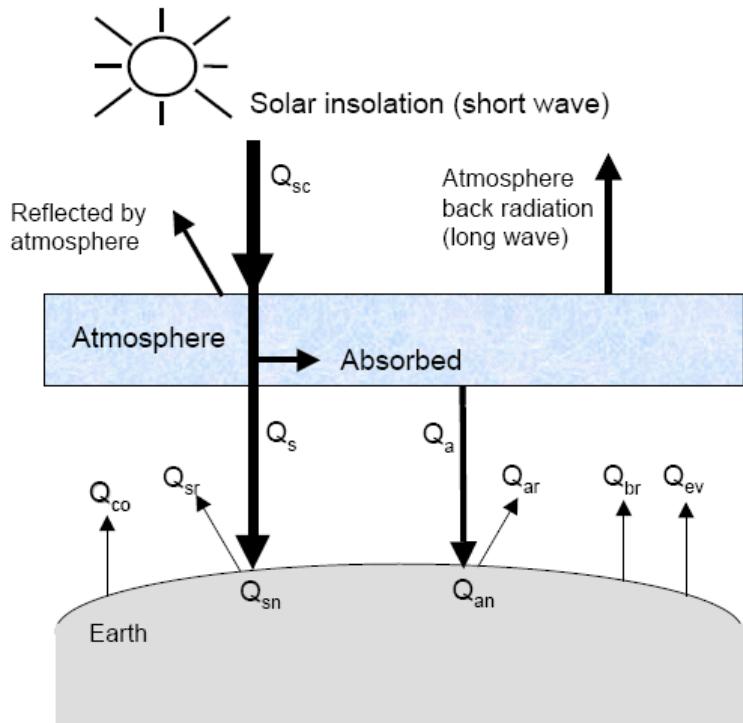


Figure 10.1: Overview of the heat exchange mechanisms at the surface

with T_s the water temperature at the free surface and T_{back} the natural background temperature, both in $^{\circ}\text{C}$.

The heat exchange coefficient λ is a function of the surface temperature T_s and the wind speed U_{10} . It is derived by linearization of the exchange fluxes for back radiation, evaporation and convection. The following relation was derived by [Sweers \(1976\)](#):

$$\lambda = 4.48 + 0.049T_s + f(U_{10}) (1.12 + 0.018T_s + 0.00158T_s^2). \quad (10.2)$$

Ocean model - heat flux model 5

The heat flux model 5 following [Gill \(1982\)](#) and [Lane \(1989\)](#) was calibrated for the North Sea and successfully applied for great lakes.

In the Ocean heat flux model, the relative humidity in [%], air temperature in $^{\circ}\text{C}$ and cloudiness in [%] are prescribed. These quantities may be either uniform or specially varying. In the external forcingsfile one may have:

```
QUANTITY =humidity_airtemperature_cloudiness
FILENAME =meteo.hac
FILETYPE =6
METHOD =3
OPERAND =0
```

For example input files see example directories:

➤ f20_heat_flux/**/

The effective back radiation and the heat losses due to evaporation and convection are computed by the model. Additionally, when air and water densities and/or temperatures are such that free convection occurs, free convection of latent and sensible heat is computed by the model.

Normally, solar radiation is computed based upon time of day, position on earth and cloudiness. However, if solar radiation was measured, it can also be prescribed (in [W/m²]), e.g.:

```
QUANTITY =humidity_airtemperature_cloudiness_solarradiation
FILENAME =meteo.hacs
FILETYPE =6
METHOD =3
OPERAND =0
```

In both heat flux models, the wind forcing may be uniform or spatially varying.

If wind is uniform, the wind speed and direction are prescribed, wind speed is in m/s, and direction follows nautical convention: 0 means wind coming from North, 90 means wind is coming from East. In the external forcings file specify, e.g.:

```
QUANTITY=windxy
FILENAME=zeg99-10.wnd
FILETYPE=2
METHOD=1
OPERAND=0
```

If wind is spatially varying, the air pressure is also prescribed:

```
QUANTITY =airpressure_windx_windy
FILENAME =CSM_2015.apwxwy
FILETYPE =6
METHOD =3
OPERAND =0
```

Air pressure is in Pa, wind -x and -y components are given m/s.

For the physical background of the heat exchange at the air-water interface and the definitions, we refer to [Sweers \(1976\)](#) for the Excess temperature model (Temperaturemodel=3), and to [Gill \(1982\)](#) and [Lane \(1989\)](#) for the Ocean heat flux model (Temperaturemodel=5).

10.1 Heat balance

The total heat flux through the free surface reads:

$$Q_{tot} = Q_{sn} + Q_{an} - Q_{br} - Q_{ev} - Q_{co} - Q_{evfree} - Q_{cofree}, \quad (10.3)$$

with:

Q_{sn}	net incident solar radiation (short wave)
Q_{an}	net incident atmospheric radiation (long wave)
Q_{br}	back radiation (long wave)
Q_{ev}	evaporative heat flux (latent heat)
Q_{co}	convective heat flux (sensible heat)
Q_{evfree}	evaporative heat flux (free convection latent heat)
Q_{cofree}	convective heat flux (free convection sensible heat).

The subscript n refers to a net contribution. Each of the heat fluxes in [Equation 10.3](#) will be discussed in detail.

The change in temperature in the top layer T_s (in $^{\circ}\text{C}$) is given by:

$$\frac{\partial T_s}{\partial t} = \frac{Q_{tot}}{\rho_w c_p \Delta z_s}, \quad (10.4)$$

where Q_{tot} [$\text{J/m}^2\text{s}$] is the total heat flux through the air-water surface, c_p ($= 3930 \text{ J/kgK}$) is the specific heat capacity of sea water, ρ_w is the specific density of water (in kg/m^3) and Δz_s (in m) is the thickness of the top layer. As in Delft3D-FLOW, the heat exchange at the bed is assumed to be zero. This may lead to over-prediction of the water temperature in shallow areas. Also the effect of precipitation on the water temperature is not taken into account.



Remarks:

- ◊ The temperature T is by default expressed in $^{\circ}\text{C}$. However, in some formulas the absolute temperature \bar{T} in K is used. They are related by:

$$\bar{T} = T + 273.15. \quad (10.5)$$

- ◊ In [Equation 10.4](#) the total incoming heat flux is absorbed with exponential decay as a function of depth. See the parameter Secchidepth in the mdu file.

10.2 Solar radiation

The short-wave radiation emitted by the sun that reaches the earth surface under a clear sky condition can be evaluated by means of:

- ◊ Applying Stefan-Boltzmann's law for radiation from a black-body:

$$Q = \sigma \bar{T}^4 \quad (10.6)$$

with σ = Stefan-Boltzmann's constant $= 5.67 \cdot 10^{-8} \text{ J/(m}^2\text{s K}^4\text{)}$ and \bar{T} the (absolute) temperature in K.

- ◊ Direct measurements.

Not all of the radiation is absorbed at the water surface. A part is transmitted to deeper water. Short waves can penetrate over a distance of 3 to 30 meters, depending on the clarity of the water, while the relatively longer waves are absorbed at the surface. Therefore, it is convenient to separate the incoming solar insolation into two portions:

- 1 βQ_{sn} , the longer wave portion, which is absorbed at the surface and
- 2 $(1 - \beta) Q_{sn}$, the remainder part, which is absorbed in deeper water.

The absorption of heat in the water column is an exponential function of the distance H from

the water surface:

$$(1 - \beta) Q_{sn} = \int_0^H e^{-\gamma z} dz \Rightarrow \quad (10.7)$$

$$Q_{sn}(h) = \frac{\gamma e^{-\gamma h}}{1 - e^{-\gamma H}} (1 - \beta) Q_{sn}, \quad (10.8)$$

with:

β	part of Q_{sn} absorbed at the water surface which is a function of the wavelength. The default value of β in Delft3D-FLOW is 0.06.
γ	extinction coefficient (measured) in m^{-1} , also related to the so-called Secchi-depth $\gamma = \frac{1.7}{H_{Secchi}}$
h	distance to the water surface in meters.
H	total water depth.

The incoming energy flux at the water surface depends on the angle (declination) between the incoming radiation and the Earth's surface. This declination depends on the geographical position on the Earth and the local time. The Earth axis is not perpendicular to the line connecting the Sun with Earth. This tilting (angle δ) varies with the time of the year and it leads to a seasonal variation of the radiation flux. At June 21, the declination is maximal, 23.5 degrees. Of course, by the rotation of the Earth the solar radiation also varies during the day. Near twelve o'clock local time, the sun elevation above the horizon is maximal. For an overview of the angles used to determine the solar elevation angle γ , see [Figure 10.2](#).

The temporal and latitude-dependent solar elevation angle γ is estimated by:

$$\sin(\gamma) = \sin(\delta) \sin\left(\frac{\pi\phi}{180}\right) - \cos(\delta) \cos\left(\frac{\pi\phi}{180}\right) \cos(\omega_1 t) \quad (10.9)$$

with:

$$\delta = \frac{23.5\pi}{180} \cos(\omega_0 t - 2.95), \quad (10.10)$$

where ω_0 is the frequency of the annual variation and ω_1 the frequency of the diurnal variation; ϕ is the latitude.

The incoming short-wave solar radiation through a clear sky at ground level Q_{sc} is about 0.76 of the flux incident at the top of the atmosphere ([Gill, 1982](#)):

$$Q_{sc} = \begin{cases} 0.76S \sin(\gamma), & \sin(\gamma) \geq 0, \\ 0.0, & \sin(\gamma) < 0. \end{cases} \quad (10.11)$$

The solar constant $S = 1368 \text{ J}/(\text{m}^2\text{s})$ or W/m^2 . This is the average energy flux at the mean radius of the Earth.

A part of the radiation that reaches the water surface is reflected. The fraction reflected or scattered (surface albedo) is dependent on latitude and season. Cloud cover will reduce the magnitude of the radiation flux that reaches the sea surface. The cloudiness is expressed by a cloud cover fraction F_c , the fraction of the sky covered by clouds. The correction factor for cloud cover is an empirical formula. The absorption of solar radiation is calculated ([Gill, 1982](#)) as the product of the net downward flux of short wave-radiation in cloudless conditions and factors correcting for reflection and cloud cover:

$$Q_{sn} = Q_s - Q_{sr} = (1 - \alpha) Q_{sc} (1.0 - 0.4F_c - 0.38F_c^2), \quad (10.12)$$

with:

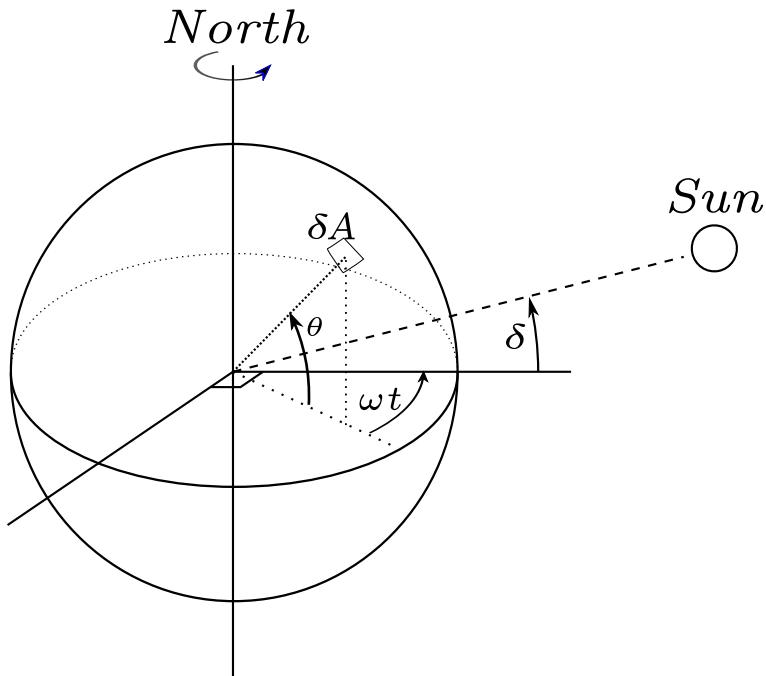


Figure 10.2: Co-ordinate system position Sun
 δ : declination; θ : latitude; ωt : angular speed

Q_{sn}	net heat radiation (flux) from the Sun
Q_s	solar radiation (short wave radiation) in $[J/m^2 s]$
Q_{sr}	reflected solar radiation in $[J/m^2 s]$
Q_{sc}	radiation (flux) for clear sky condition
α	albedo (reflection) coefficient ($=0.06$)
F_c	fraction of sky covered by clouds (user-defined input)

10.3 Atmospheric radiation (long wave radiation)

Atmospheric radiation is primarily due to emission of absorbed solar radiation by water vapour, carbon dioxide and ozone in the atmosphere. The emission spectrum of the atmosphere is highly irregular. The amount of atmospheric radiation that reaches the earth is determined by applying the Stefan-Boltzmann's law that includes the emissivity coefficient of the atmosphere ε . Taking into account the effect of reflection by the surface and reflection and absorption by clouds, the relation for the net atmospheric radiation Q_{an} reads (Octavio et al., 1977):

$$Q_{an} = (1 - r) \varepsilon \sigma \bar{T}_a^4 g(F_c), \quad (10.13)$$

where \bar{T}_a is the air temperature (in K) and the reflection coefficient $r = 0.03$. The emissivity factor of the atmosphere ε may depend both on vapour pressure and air temperature. The emissivity of the atmosphere varies between 0.7 for clear sky and low temperature and 1.0. The presence of clouds increases the atmospheric radiation. This is expressed in the cloud function $g(F_c)$.

with T_a the air temperature (in $^{\circ}\text{C}$). The cloud function $g(F_c)$ in (10.13) is given by:

$$g(F_c) = 1.0 + 0.17F_c^2. \quad (10.14)$$

The linearisation of Equation 10.13 is carried out around $T_a = 15 \text{ } ^{\circ}\text{C}$.

① **Remark:**

- ◊ The atmospheric radiation is part of the total long-wave radiation flux, the so-called effective back radiation, see [section 10.5](#).

10.4 Back radiation (long wave radiation)

Water radiates as a near black body, so the heat radiated back by the water can be described by Stefan-Boltzmann's law of radiation, corrected by an emissivity factor $\varepsilon = 0.985$ of water ([Sweers, 1976](#); [Octavio et al., 1977](#)) and the reflection coefficient for the air-water interface $r = 0.03$:

$$Q_{br} = (1 - r) \varepsilon \sigma \bar{T}_s^4, \quad (10.15)$$

with \bar{T}_s the (absolute) water surface temperature in K.

10.5 Effective back radiation

The total net long wave radiation flux is computed. This is called the effective back radiation:

$$Q_{eb} = Q_{br} - Q_{an}. \quad (10.16)$$

The atmospheric radiation depends on the vapour pressure e_a , see [section 10.6](#), the air temperature T_a and the cloud cover F_c . The back radiation depends on the surface temperature T_s .

The effective back radiation Q_{eb} is computed following:

$$Q_{eb} = \varepsilon \sigma \bar{T}_s^4 (0.39 - 0.05\sqrt{e_a}) (1.0 - 0.6F_c^2), \quad (10.17)$$

with the actual vapour pressure e_a given by [Equation 10.22](#).

10.6 Evaporative heat flux

Evaporation is an exchange process that takes place at the interface between water and air and depends on the conditions both in the water near the surface and the air above it. The evaporation depends on meteorological factors (wind-driven convection) and vapour pressures.

Forced convection of latent heat

The latent heat flux due to forced convection for the ocean heat flux model reads:

$$Q_{ev,forced} = L_V \rho_a f(U_{10}) \{q_s(T_s) - q_a(T_a)\}, \quad (10.18)$$

with q_s and q_a the specific humidity of respectively saturated air and remote air (10 m above water level):

$$q_s(T_s) = \frac{0.62e_s}{P_{atm} - 0.38e_s}, \quad (10.19)$$

$$q_a(T_a) = \frac{0.62e_a}{P_{atm} - 0.38e_a}. \quad (10.20)$$

The saturated and remote vapour pressures e_s and e_a are given by:

$$e_s = 10^{\frac{0.7859+0.03477T_s}{1.0+0.00412T_s}}, \quad (10.21)$$

$$e_a = r_{hum} 10^{\frac{0.7859+0.03477T_a}{1.0+0.00412T_a}}. \quad (10.22)$$

With L_v the latent heat of vaporisation in J/kg water:

$$L_v = 2.5 \cdot 10^6 - 2.3 \cdot 10^3 T_s. \quad (10.23)$$

The wind function in [Equation 10.18](#) is defined as:

$$f(U_{10}) = c_e U_{10}, \quad (10.24)$$

Without the influence of free convection, the Dalton number c_e in the Ocean heat flux model was calibrated for the North Sea to be $c_e = 0.0015$. This value should be close to the Cd coefficient that is used in the computation of wind stresses. The exchange coefficients of latent heat and momentum transfer are closely related. Specifying a negative Dalton number in the mdu file forces the use of the specified Cd coefficient, thus taking into account the specified dependency between windspeed and the Cd coefficient.

Here r_{hum} is the relative humidity in [-].



Remarks:

- ◊ The relative humidity r_{hum} is specified in the input files in percentages.
- ◊ When the computed E is negative, it is replaced by zero, assuming that it is caused by modelling misfit and not by the actual physical process of water condensation out of the air into the water. The same applies to the part associated with free convection.

For the excess temperature model, the wind speed function $f(U_{10})$ following [Sweers \(1976\)](#) is used:

$$f(U_{10}) = (3.5 + 2.0U_{10}) \left(\frac{5.0 \cdot 10^6}{S_{area}} \right)^{0.05}, \quad (10.25)$$

where S_{area} is the exposed water surface in m^2 , defined in the input and fixed for the whole simulation. The coefficients calibrated by Sweers were based on the wind speed at 3 meter above the free surface; the coefficients in [Equation 10.25](#) are based on the wind speed 10 meter above the water level.

Free convection of latent heat

Loss of heat due to evaporation occurs not only by forced convection, wind driven, but also by free convection. Free convection is driven by buoyant forces due to density differences (by temperature and/or water vapour content) creating unstable conditions in the atmospheric boundary layer (ABL). Evaporation due to free convection is important in circumstances where inverse temperature/density gradients are present and wind speeds are almost negligible so that the amount of forced convection is small. Neglecting free convection in this situation will lead to underestimating the heat loss. ([Ryan et al., 1974](#)) developed a correction to the wind function, accounting for free convection. The derivation of evaporation by just free convection is based on the analogy of heat and mass transfer.

The latent heat flux due to free convection reads:

$$Q_{ev,free} = k_s L_V \bar{\rho}_a (q_s - q_a), \quad (10.26)$$

with the average air density:

$$\bar{\rho}_a = \frac{\rho_{a0} + \rho_{a10}}{2}, \quad (10.27)$$

and with the heat transfer coefficient defined as:

$$k_s = \begin{cases} 0 & \text{if } \rho_{a10} - \rho_{a0} \leq 0 \\ C_{fr.\text{conv}} \left\{ \frac{g\alpha^2}{\nu_{air}\bar{\rho}_a} (\rho_{a10} - \rho_{a0}) \right\}^{1/3} & \text{if } \rho_{a10} - \rho_{a0} > 0 \end{cases} \quad (10.28)$$

where the coefficient of free convection $C_{fr.\text{conv}}$ was calibrated to be 0.14, see (Ryan *et al.*, 1974). The viscosity of air ν_{air} is assumed to have the constant value $16.0 \cdot 10^{-6} \text{ m}^2/\text{s}$. The molecular diffusivity of air $\alpha \text{ m}^2/\text{s}$ is defined as

$$\alpha = \frac{\nu_{air}}{\sigma}, \quad (10.29)$$

with $\sigma = 0.7$ (for air) the Prandtl number. In [Equation 10.26](#), the saturated air density is given by:

$$\rho_{a0} = \frac{\frac{100P_{atm}-100e_s}{R_{dry}} + \frac{100e_s}{R_{vap}}}{T_s + 273.15}, \quad (10.30)$$

the remote air density (10 m above the water level):

$$\rho_{a10} = \frac{\frac{100P_{atm}-100e_a}{R_{dry}} + \frac{100e_a}{R_{vap}}}{T_{air} + 273.15}, \quad (10.31)$$

where R_{dry} is the gas constant for dry air: 287.05 J/(kg K) and R_{vap} is the gas constant for water vapour: 461.495 J/(kg K). The specific humidity of respectively saturated air and remote air (10 m above the water level), q_s and q_a are given by Eqs. [10.19](#) and [10.20](#). The saturated and remote vapour pressure e_s and e_a are defined in Eqs. [10.21](#) and [10.22](#).

The total heat flux due to evaporation then results from adding the forced convection of latent heat in [Equation 10.18](#) and the free convection of latent heat in [Equation 10.26](#):

$$Q_{ev} = Q_{ev,\text{forced}} + Q_{ev,\text{free}}. \quad (10.32)$$

10.7 Convective heat flux

In the Ocean heat flux model, the convective heat flux is split into two parts, just as the evaporative heat flux. The convective heat flux is divided into a contribution by forced convection and a contribution by free convection.

Forced convection of sensible heat

The sensible heat flux due to forced convection is computed by:

$$Q_{co,\text{forced}} = \rho_a c_p g (U_{10}) (T_s - T_a), \quad (10.33)$$

with c_p the specific heat of air. It is considered constant and taken to be 1 004.0 J/(kg K). The wind-speed function $g (U_{10})$ is defined following Gill (1982):

$$g (U_{10}) = c_H U_{10}, \quad (10.34)$$

with c_H the so-called Stanton number. Without the influence of free convection, the Stanton number was calibrated for the North Sea to be $c_H = 0.00145$.

Free convection of sensible heat

$$Q_{co,\text{free}} = k_s \bar{\rho}_a c_p (T_s - T_a), \quad (10.35)$$

with the heat transfer coefficient k_s given by [Equation 10.28](#).

The total heat flux due to convection then results from adding the forced convection of sensible heat in [Equation 10.33](#) and the free convection of sensible heat in [Equation 10.35](#):

$$Q_{co} = Q_{co,\text{forced}} + Q_{co,\text{free}}. \quad (10.36)$$

11 Wind

Various external influences can exert a force on the flow field. One of these influences is the wind. The force exerted by the wind is coupled to the flow equations as a shear stress. The magnitude is determined by the following widely used quadratic expression:

$$|\vec{\tau}_s| = \rho_a C_d U_{10}^2 \quad (11.1)$$

where:

- ρ_a the density of air.
 U_{10} the wind speed 10 meter above the free surface (time and space dependent).
 C_d the wind drag coefficient, dependent on U_{10} .

In order to specify the wind shear stress, a drag coefficient is required as well as the wind field in terms of velocity magnitude and wind direction. In this chapter, the backgrounds are provided of how wind fields should be imposed, in addition to [section 4.4.9.4](#). Relevant definitions are addressed in [section 11.1](#), whereas supported file formats are addressed in [section 11.2](#).



11.1 Definitions

When imposing wind conditions, two definitions are respected: a definition for the wind direction (see [section 11.1.1](#)) and a definition regarding the drag coefficient (see [section 11.1.2](#)).

11.1.1 Nautical convention

The wind direction is defined according to the nautical definition, i.e. relative to true North and positive measured clockwise. In [Figure 11.1](#) the wind direction is about +60 degrees, i.e. an East-North-East wind.

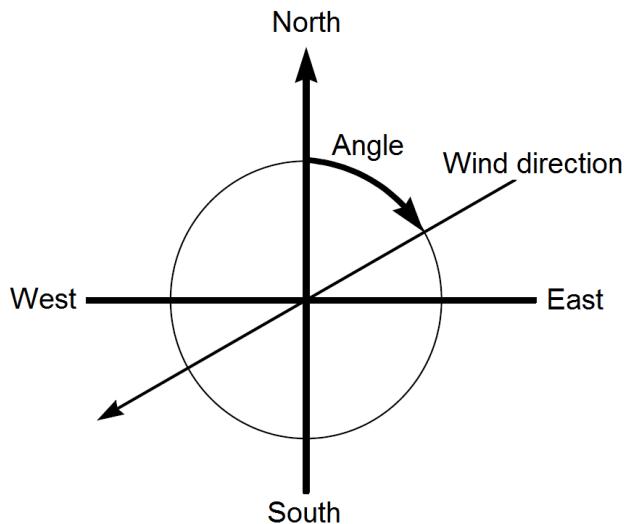


Figure 11.1: Nautical conventions for the wind.

11.1.2 Drag coefficient

The dependency of the drag coefficient on the wind speed should be specified by the user. The user can choose between the following concepts:

- ◊ a constant drag coefficient,
- ◊ a dependency according to [Smith and Banke \(1975\)](#),
- ◊ a dependency according to [Charnock \(1955\)](#),
- ◊ a dependency according to [Hwang \(2005a\)](#) and [Hwang \(2005b\)](#).

The specification of the type of wind drag formulation should be accomplished in the MDU-file. For this purpose, the keyword `ICdtyp` can be utilized. For this keyword `ICdtyp`, five options could be demanded for:

- ◊ `ICdtyp = 1` – constant drag coefficient,
- ◊ `ICdtyp = 2` – linearly varying drag coefficient (cf. [Smith and Banke \(1975\)](#)),
- ◊ `ICdtyp = 3` – piecewise linearly varying drag coefficient (cf. [Smith and Banke \(1975\)](#)),
- ◊ `ICdtyp = 4` – [Charnock \(1955\)](#) formulation (no breakpoints),
- ◊ `ICdtyp = 5` – [Hwang \(2005a\)](#) and [Hwang \(2005b\)](#) formulation (no breakpoints).

If a Smith & Banke type dependency is chosen for, the additional entries `Cdbreakpoints` and `Windspeedbreakpoints` come into play. In the following sections, the specification of either of these options are depicted.

Smith & Banke type formulation

When specifying a Smith & Banke type dependency, the definition as sketched in [Figure 11.2](#) should be kept in mind.

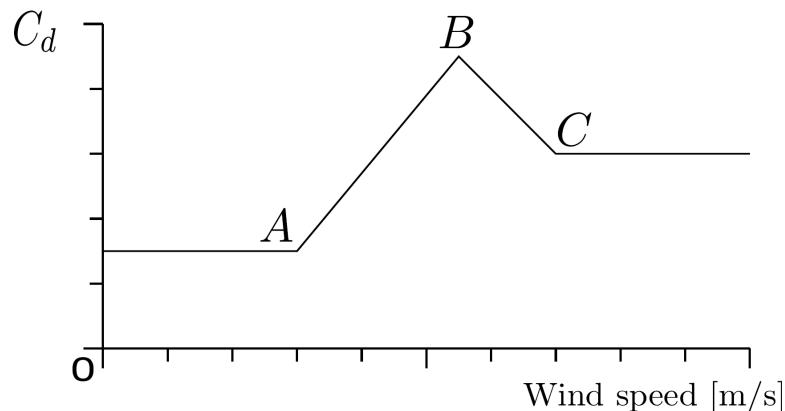


Figure 11.2: Prescription of the dependency of the wind drag coefficient C_d on the wind speed is achieved by means of at least 1 point, with a maximum of 3 points.

From this sketch, it can be seen that the wind drag is considered as dependent on the wind speed in a piecewise linear way. The options, that are facilitated in this respect, are:

- ◊ define *one* set of coordinates (breakpoint A), specifying a constant drag coefficient, valid for all wind speeds,
- ◊ define *two* sets of coordinates (breakpoints A and B), specifying a linearly varying dependency for one range of wind speeds,
- ◊ define *three* sets of coordinates (breakpoints A, B and C), specifying a piecewise linear dependency for two ranges of wind speeds.

Remark that for the latter two options, the drag coefficient is taken constant for wind speeds lower/higher than the lowest/highest specified wind speed, with a drag coefficient equal to the drag coefficient associated with the lowest/highest specified lowest/highest wind speed. In case of three breakpoints, the expression reads:

$$C_d(U_{10}) = \begin{cases} C_d^A, & U_{10} \leq U_{10}^A, \\ C_d^A + (C_d^B - C_d^A) \frac{U_{10} - U_{10}^A}{U_{10}^B - U_{10}^A}, & U_{10}^A \leq U_{10} \leq U_{10}^B, \\ C_d^B + (C_d^C - C_d^B) \frac{U_{10} - U_{10}^B}{U_{10}^C - U_{10}^B}, & U_{10}^B \leq U_{10} \leq U_{10}^C, \\ C_d^C, & U_{10}^C \leq U_{10}, \end{cases} \quad (11.2)$$

By means of the entries `Cdbreakpoints` and `Windspeedbreakpoints`, the coordinates of the breakpoints (see [Figure 11.2](#)) can be specified. Typical values associated with the [Smith and Banke \(1975\)](#) formulation are $C_d = 6.3 \cdot 10^{-4}$ for $U = 0$ m/s and $C_d = 7.23 \cdot 10^{-3}$ for $U = 100$ m/s. In this case, the entries in the MDU-file should be specified as follows:

[wind]	
ICdtyp	= 2
Cdbreakpoints	= 0.00063 0.00723
Windspeedbreakpoints	= 0.00000 100.00000

Charnock formulation

The Charnock formulation (see [Charnock \(1955\)](#)) is based on the assumption of a fully developed turbulent boundary layer of the wind flow over the water surface. The associated wind speed profile follows a logarithmic shape. In the Charnock formulation, the wind speed is considered at 10 meters above the free water surface, hence yielding the following expression:

$$\frac{U_{10}}{u_*} = \frac{1}{\kappa} \ln \left(\frac{z_{10}}{z_0} \right) \quad (11.3)$$

with κ the Von Kármán constant, z_{10} the distance to the water surface (equal to 10 m), u_* the friction velocity and U_{10} the wind speed at 10 m above the water surface. The drag coefficient C_d is defined as:

$$C_d = \frac{u_*^2}{U_{10}^2}. \quad (11.4)$$

[Charnock \(1955\)](#) has proposed to represent the friction of the water surface as z_0 according to:

$$z_0 = \frac{b \cdot u_*^2}{g}, \quad (11.5)$$

with g the gravitation acceleration and b a specific constant. [Charnock \(1955\)](#) has proposed $b = 0.012$. The value of the constant b can be specified in the MDU-file by the user by means of one single value for `Cdbreakpoints`. Since the above relation yields an implicit relation for u_* , the system is solved for iteratively. The user should be aware of interpretation of the specified wind field as the wind field at 10 m above the water surface.

Hwang formulation

The dynamic roughness could also be related to the steady state wave conditions of the flow field under consideration. The connection of the wave parameters with the drag coefficient as elaborated by [Hwang \(2005a\)](#) is available within D-Flow FM through `ICdtyp = 5`, given a wave field. The Hwang-formulation interprets the user defined wind speed as the wind speed at 10 m above the water surface.

The drag coefficient is computed as:

$$C_d = \left[\frac{1}{\kappa} \ln \left(\frac{k_p z_{10}}{k_p z_0} \right) \right]^{-2} \quad (11.6)$$

with $z_{10} = 10$ m, κ the Von Kármán constant. With wavelength scaling, $k_p z_0$ is the natural expression of the dimensionless roughness, where k_p is the wave number of the spectral peak, computed on the basis of the actual water depth and the provided peak period T_p as wave field. Further following [Hwang \(2005a\)](#),

$$k_p z_0 = \pi \exp \left(-\kappa C_{\lambda/2}^{-0.5} \right) \quad (11.7)$$

in which $C_{\lambda/2}$ is the drag coefficient at half the wavelength above surface. This parameter $C_{\lambda/2}$ is computed as:

$$C_{\lambda/2} = A_{10} \left(\frac{\omega_p U_{10}}{g} \right)^{a_{10}} \quad (11.8)$$

in which $A_{10} = 1.289 \cdot 10^{-3}$, $a_{10} = 0.815$, U_{10} the wind speed at 10 m above the water surface and ω_p the wave peak frequency ($\omega_p = 2\pi/T_p$). Thus, the drag coefficient C_d is defined.

11.2 File formats

The wind field should be provided by means of an ascii-type file. This file should contain the grid on which the wind field is defined as well as the wind velocity vector(s).

D-Flow FM currently supports four types of wind field prescriptions, i.e. four grid types on which the wind field can be given. This wind grid does not need to be the same as the computational grid. The grid options to provide the wind data on are:

- 1 the computational grid – in this case, no specific wind grid is provided. The provided wind field is considered to be uniform over the entire model area. The wind field can be time dependent.
- 2 an equidistant grid – in this case, a wind field can be prescribed that varies both in space and in time. A Cartesian arcinfo-type grid should be provided on which the wind field is defined.
- 3 a curvilinear grid – this case is conceptually similar to the previous type (the equidistant grid) in the sense that a wind field can be imposed that both varies in space and time. However, a separate file should be provided in which a curvilinear grid is defined (a classic .grd-type file as known from Delft3D) on which the wind field is defined.
- 4 a spiderweb grid – this type of wind specification is specially devoted to cyclone winds and is only available in combination with computational grids that are of spherical type. In this case, a cyclone wind field is given on a polar grid with the center ('eye') of the cyclone being the origin of the polar coordinate system. The location of this eye and the associated wind field usually varies in time.

Each of these filetypes can be assigned through the entry in the external forcings file (the .ext-file) named FILETYPE. In this chapter, the various types of wind field specifications are highlighted subsequently. Each of the options is illustrated by means of an example.

11.2.1 Defined on the computational grid

In D-Flow FM, the specification of the wind on the computational grid is equivalent to the specification of a uniform wind, since no separate wind grid is provided to the model. The specification of a uniform wind field can be done in two ways:

- 1 componentwise: as velocity in the longitudinal x -direction (m/s) and in the latitudinal y -direction (m/s) – the associated FILETYPE in the external forcings file is depicted as `uniform`, which has FILETYPE=1.
- 2 by magnitude (m/s) and direction (degN) (see [Figure 11.1](#)) – the associated FILETYPE in the external forcings file is depicted as `unimagdir`, which has FILETYPE=2.

These two types are treated below separately.

11.2.1.1 Specification of uniform wind through velocity components

Since no particular wind grid is used, only timeseries for the x -component and the y -component of the wind need to be specified. The specification of these timeseries can be done separately (one single file for the x -component and one single file for the y -component) or jointly (one single file containing the x -component and the y -component of the wind).

Uniform wind should be provided as an .wnd-file containing either 2 columns (in case of separate specification of the x -component and y -component of the wind) or 3 columns (in case of joint specification of the velocity components). In either case, the first column contains the time in minutes with respect to the overall reference time.

Example

As an example, a uniform wind field is applied to a certain model. The uniform wind is provided in a file named `windxdirydir.wnd`. The contents of this wind file are:

0.00000 10.00000 10.00000
60.00000 -10.00000 -10.00000

The first column denotes the time in minutes with respect to the reference date (specified in the MDU-file). The second column denotes the wind velocity in x -direction, whereas the third column denotes the wind velocity in y -direction; both wind components are provided in one single file.

The connection with the flow model itself is laid through the external forcings file. The actual specification of the wind is in this case:

QUANTITY =windxy
FILENAME =windxdirydir.wnd
FILETYPE =1

```
METHOD    =1
OPERAND   =0
```

Since the two components are given in one single file, the QUANTITY is set to `windxy`. If two separate files would have been provided, the QUANTITY would have been set to `windx` and `windy` over two separate datablocks in the external forcings file.

11.2.1.2 Specification of uniform wind through magnitude and direction

Instead of specifying the separate components of the wind field, the uniform wind vector can also be prescribed through its magnitude and direction. This kind of specification should be done by means of one single file, containing three columns, representing the time (in minutes with respect to the reference date), the velocity magnitude (in m/s, not necessarily positive) and the wind direction (in degN).

Example

As an example, the previous uniform wind case is reformulated as a case with magnitude and direction of the wind field prescribed. The unimagdir wind is provided in a file named `windinput.wnd`. The contents of this file are:

```
0.00000  14.14213562373095  225.00000
60.00000 -14.14213562373095  225.00000
```

The first column denotes the time in minutes with respect to the reference date (specified in the MDU-file). The second column denotes the wind velocity magnitude, whereas the third column denotes the wind direction. Note that there is a clear difference between the above case and a case in which the magnitude is kept positive (14.1421 m/s) and the direction varies (and hence *rotates!*) from 225 degN to 45 degN.

The connection with the flow model itself is laid through the external forcings file. The actual specification of the wind is in this case:

```
QUANTITY =windxy
FILENAME =windinput.wnd
FILETYPE =2
METHOD    =1
OPERAND   =0
```

11.2.2 Defined on an equidistant grid

The vector components of the velocity vectors can also be specified on a distinct grid, either of equidistant type or of curvilinear type. In both cases, the characteristics of the grid should be provided. In case of an equidistant grid, the grid is specified in arcinfo-style. That means, the constant grid sizes Δx and Δy should be specified such that a grid is spanned with respect to the location of the lower left corner of the grid (either the center of the lower left cel or the lower left corner of the lower left cell).

Example

As an example, a grid with $\Delta x = \Delta y = 100$ m is spanned, based on the center of the lower left cell, located at $x = y = 60$ m with respect to the origin. The input data for the x -component and the y -component should be specified separately, in two distinct files. The input of the x -component data should be given in an .amu-type file, such as `windxdir.amu` as an example:

```
### START OF HEADER
### This file is created by Deltares
### Additional comments
FileVersion      = 1.03
filetype        = meteo_on_equidistant_grid
NODATA_value   = -9999.0
n_cols          = 5
n_rows          = 4
grid_unit       = m
x_llcenter     = 60
y_llcenter     = 60
dx              = 110
dy              = 110
n_quantity     = 1
quantity1       = x_wind
unit1           = m s-1
### END OF HEADER
TIME = 0 hours since 2006-01-01 00:00:00 +00:00
10 10 10 10 10
10 10 10 10 10
10 10 10 10 10
10 10 10 10 10
TIME = 1 hours since 2006-01-01 00:00:00 +00:00
-10 -10 -10 -10 -10
-10 -10 -10 -10 -10
-10 -10 -10 -10 -10
-10 -10 -10 -10 -10
```

For the y -component data, a similar file (e.g. `windydir.amv`) should be provided. In addition, the pressure could be specified in a similar file (e.g. `pressure.amp`). Note that `x_llcorner` and `y_llcorner`, instead of `x_llcenter` and `y_llcenter`, are also supported.

Wind on an equidistant grid has been provided a filetype specification as FILETYPE=4. The connection with the flow model itself is laid through the external forcings file. The actual specification of the wind is in this case:

```
QUANTITY =windx
FILENAME =windxdir.amu
FILETYPE =4
METHOD   =2
OPERAND  =0

QUANTITY =windy
FILENAME =windydir.amv
FILETYPE =4
METHOD   =2
OPERAND  =0

QUANTITY =atmosphericpressure
```

```
FILENAME =pressure.amp
FILETYPE =4
METHOD =2
OPERAND =0
```

11.2.3 Defined on a curvilinear grid

In analogy with the wind specification on an equidistant grid, the wind can be specified on a curvilinear grid. This curvilinear grid should be provided as a classic .grd-file as known from Delft3D. A difference with the equidistant grid wind is the necessity to compile all data blocks (i.e. x -component, y -component and pressure) in one single file. This file should have the extension .apwxwy. The sequence of this datablock is: pressure — x -velocity component — y -velocity component.

Example

As an example, a curvilinear grid named meteo.grd is present, providing the underlying coordinates of the wind data field. The input data, comprising the atmospheric pressure, the x -velocity component *and* the y -velocity component, are given in one single file (as is compulsory). The contents of the example meteo.apwxwy-file is:

```
### START OF HEADER
### This file is created by Deltares
### Additional comments
FileVersion      =    1.03
filetype        =    meteo_on_curvilinear_grid
NODATA_value   =    -9999.0
grid_file       =    meteo.grd
first_data_value =    grid_llcorner
data_row        =    grid_row
n_quantity     =    3
quantity1      =    apwxwy
unit1          =    Pa
### END OF HEADER
TIME = 0.0 hours since 2006-01-01 00:00:00 +00:00
101325 101325 101325 101325 101325
101325 101325 101325 101325 101325
101325 101325 101325 101325 101325
101325 101325 101325 101325 101325
101325 101325 101325 101325 101325
10 10 10 10 10
10 10 10 10 10
10 10 10 10 10
10 10 10 10 10
10 10 10 10 10
10 10 10 10 10
10 10 10 10 10
10 10 10 10 10
10 10 10 10 10
10 10 10 10 10
TIME = 1.0 hours since 2006-01-01 00:00:00 +00:00
101325 101325 101325 101325 101325
101325 101325 101325 101325 101325
101325 101325 101325 101325 101325
101325 101325 101325 101325 101325
-10 -10 -10 -10 -10
-10 -10 -10 -10 -10
-10 -10 -10 -10 -10
-10 -10 -10 -10 -10
-10 -10 -10 -10 -10
-10 -10 -10 -10 -10
```

```
-10 -10 -10 -10 -10
```

Note that `grid_llcenter`, instead of `grid_llcorner`, is also supported. On the contrary, `grid_column` is *not* supported instead of `grid_row`.

Wind on a curvilinear grid has been provided a filetype specification as `FILETYPE=6`. The connection with the flow model itself is laid through the external forcings file. The actual specification of the wind is in this case:

```
QUANTITY =airpressure_windx_windy
FILENAME =meteo.apwxwy
FILETYPE =6
METHOD =3
OPERAND =0
```

Notice that `METHOD=3` is chosen for wind on a curvilinear grid, instead of `METHOD=2` in case of wind on an equidistant grid.

11.2.4 Defined on a spiderweb grid

Cyclone winds can be imposed by means of a polar grid that is spanned around the cyclone eye. In addition to the cyclone eye, quite probably varying in both space and time, the number of rows (discretisation in radial direction) and the number of columns (discretisation in angular direction) should be given, as well as the radius of the grid (in meters). The definitions of the cyclone wind grid (also depicted as spiderweb grid) is illustrated in [Figure 11.3](#).

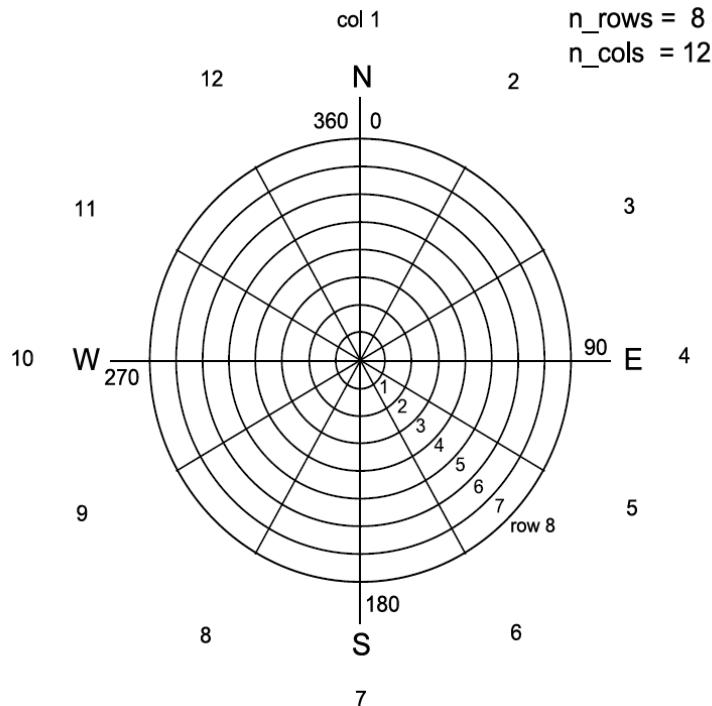


Figure 11.3: Grid definition of the spiderweb grid for cyclone winds.

Cyclone winds can only be used in combination with a spherical computational grid. The location of the cyclone eye should be given as longitude (for x_{eye}) and latitude (for y_{eye}). At the eye the pressure drop should be prescribed. This pressure drop is taken relative to the atmospheric pressure as prescribed in the MDU-file. The extension of the spiderweb grid file is .spw. The contents of the spiderweb wind file should comprise the local atmospheric pressure drop, the wind velocity magnitude and the wind direction.

Example

As an example, a spiderweb grid named spwsimple.spw is present, providing the underlying coordinates of the wind data field. The input data, comprising the atmospheric pressure drops, the wind velocity magnitudes (in m/s) and the wind directions (in degN), are given in one single file (as is compulsory). The contents of the example spwsimple.spw-file is:

```
### Spiders web derived from TRACK file: gonu.trk
### This file is created by Deltires
### All text on a line behind the first # is parsed as commentary
### Additional comments
FileVersion      = 1.03
filetype        = meteo_on_spiderweb_grid
### Spiders web derived from TRACK file: gonu.trk
### This file is created by Deltires
### All text on a line behind the first # is parsed as commentary
### Additional comments
NODATA_value   = -1001
n_cols          = 4
n_rows          = 4
grid_unit       = degree
spw_radius     = 600000.0
spw_rad_unit   = m
n_quantity     = 3
quantity1       = wind_speed
quantity2       = wind_from_direction
quantity3       = p_drop
unit1           = m s-1
unit2           = degree
unit3           = Pa
### END OF HEADER
TIME            = 3400000.00 minutes since 2005-01-01 00:00:00 +00:00
x_spw_eye       = 265.00
y_spw_eye       = 33.00
pdrop_spw_eye   = 7000.000
  5.000000 5.000000 5.000000 5.000000
 10.000000 10.000000 10.000000 10.000000
 15.000000 15.000000 15.000000 15.000000
 20.000000 20.000000 20.000000 20.000000
  270.00    0.00    90.00   180.00
  270.00    0.00    90.00   180.00
  270.00    0.00    90.00   180.00
  270.00    0.00    90.00   180.00
 4000.00   4000.00   4000.00   4000.00
 3000.00   3000.00   3000.00   3000.00
 2000.00   2000.00   2000.00   2000.00
 1000.00   1000.00   1000.00   1000.00
TIME            = 3800000.00 minutes since 2005-01-01 00:00:00 +00:00
x_spw_eye       = 275.00
y_spw_eye       = 18.00
pdrop_spw_eye   = 8000.000
  5.000000 5.000000 5.000000 5.000000
 10.000000 10.000000 10.000000 10.000000
 15.000000 15.000000 15.000000 15.000000
 20.000000 20.000000 20.000000 20.000000
```

270.00	0.00	90.00	180.00
270.00	0.00	90.00	180.00
270.00	0.00	90.00	180.00
270.00	0.00	90.00	180.00
4000.00	4000.00	4000.00	4000.00
3000.00	3000.00	3000.00	3000.00
2000.00	2000.00	2000.00	2000.00
1000.00	1000.00	1000.00	1000.00

Note that in the header of the file, only the entry `unit3` could be chosen freely, i.e. either the unit Pa or the unit mbar could be chosen. The other entries are frozen and hence not available for free choices.

Wind on a spiderweb grid has been provided a filetype specification as `FILETYPE=5`. The connection with the flow model itself is laid through the external forcings file. The actual specification of the wind is in this case:

```
QUANTITY =airpressure_windx_windy
FILENAME =spwsimple.spw
FILETYPE =5
METHOD   =1
OPERAND  =0
```

Notice that `METHOD=1` is chosen for wind on a spiderweb grid, instead of `METHOD=2` in case of wind on an equidistant grid and `METHOD=3` in case of wind on a curvilinear grid.

Note that the spiderweb file contains a pressure *drop* rather than the actual pressure. This, hence, invokes the need to prescribe the pressure reference value. This value should be prescribed in the MDU-file, in Pascals, for both the boundaries as well as the interior field. As an example:

```
[wind]
PavBnd          = 101325.0000000
Gapres          = 101325.0000000
```

11.2.5 Combination of several wind specifications

The combination of the various wind specification types can *only* be achieved if the `QUANTITY` of the winds to be combined is the same, for instance `QUANTITY=windx`. That means that the combination of a uniform wind with a cyclone cannot be combined, at the moment. The option `OPERAND=+` can be used to add a wind field to an existing wind field.

Example

If the uniform wind is to be combined with a wind specified on an equidistant grid, then the wind field could be assigned in the external forcings file as follows:

```
QUANTITY =windx
FILENAME =windxdir wnd
```

```

FILETYPE =1
METHOD   =1
OPERAND  =0

QUANTITY =windy
FILENAME =windydir.wnd
FILETYPE =1
METHOD   =1
OPERAND  =0

QUANTITY =windx
FILENAME =windxdir.amu
FILETYPE =4
METHOD   =2
OPERAND  =+

QUANTITY =windy
FILENAME =windydir.amv
FILETYPE =4
METHOD   =2
OPERAND  =+

```

11.3 Masking of points in the wind grid from interpolation ('land-sea mask')

A mask can be supplied by the user to prevent selected points in the wind grid from contributing to the wind interpolation on velocity points, e.g. to exclude land points. This feature was included to conform to SIMONA and therefore implemented in the same way.

For each individual grid point for which to interpolate from the wind grid:

- ◊ Masked wind points are excluded from the interpolation.
- ◊ The total of the weight factors for the remaining wind points is determined.
- ◊ If this total falls below 1E-03, the mask is ignored and the original bilinear weights are used.
- ◊ Otherwise, the weights for the remaining wind points are normalised again.

The effect of the mask, when applied as a land-sea mask, is that for velocity points close to shore the interpolated wind is no longer influenced by the wind over land (which would otherwise yield a zone of points with reduced wind near the shore).

Specification and format of the mask file

The name of the mask file, if any, is specified in the .EXT file, labelled SOURCEMASK, directly following the FILENAME specification, e.g.:

```

QUANTITY =windxy
FILENAME =meteo.wxwy
SOURCEMASK =meteo_mask.asc
FILETYPE =6
METHOD   =3
OPERAND  =0

```

The mask file itself has the same layout as the wind file, though the number of required header fields is reduced, e.g.:

```
FileVersion      = 1.03
.
unit1          = Pa
### END OF HEADER
1      1      1      1      1
1      1      1      0      0
1      1      1      0      0
1      1      0      0      0
```

The lines in the header are ignored. The number of columns and rows in the matrix of ones and zeros should match those of a block (for a single variable and a single timestep) in the meteo files. Zeros signify the position of rejected points (and ones those of the accepted points) in the wind grid.

12 Hydraulic structures

12.1 Introduction

Obstacles in the flow may generate sudden transitions from flow contraction to flow expansion. The grid resolution is often low compared to the gradients of the water level, the velocity and the bathymetry. The hydrostatic pressure assumption may locally be invalid. Examples of these obstacles in civil engineering are: gates, barriers, dams, groynes and weirs. The obstacles generate energy losses and may change the direction of the flow.

The forces due to obstacles in the flow which are not resolved (sub-grid) on the horizontal grid, should be parameterised. The obstacles are denoted in D-Flow FM as hydraulic structures. In the numerical model the hydraulic structures should be located at velocity points of the grid. The direction of the forces should be specified at input. To model the force on the flow generated by a hydraulic structure, a quadratic energy or linear loss term is added to the momentum equation.

12.1.1 Model input

Structure definitions can be made in Delta Shell, and will then be saved into a structures <.ini> file ([section B.9](#)).

12.2 Structures

The user can insert the hydraulic structures by the means of polygons on the grid. By selecting the required structure and drawing a polygon on the computational grid, the location of structure can be defined (see [Figure 12.1](#)). The supported structures in D-Flow FM are

- ◊ weirs
- ◊ Thin dams
- ◊ Gates
- ◊ Pumps

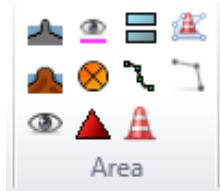


Figure 12.1: Selection of structures in the toolbar.

12.2.1 Fixed weirs

In D-Flow FM, a fixed weir is a fixed non-movable construction generating energy losses due to constriction of the flow. They are commonly used to model sudden changes in depth (roads, summer dikes) and groynes in simulation models of rivers.

Weirs are implemented in D-Flow FM by means of polygons on the velocity points of the computational grids. For input of the computational conditions of weirs, we refer you to [section 4.4.2.10](#).

12.2.2 Adjustable weir

Unlike the fixed weir (with fixed crest level), an adjustable weir can have some of its geometric parameters adjusted in time. The controllable weirs (simply called weirs in D-Flow FM) are weirs which can be controlled with a predefined time series or get controlled based on the water level or other conditions.

The structure parameters for a controllable weir can be defined via the <structures.ini> file:

```
type      = weir
id       = weir02
polylinefile = weir02.pli
crest_level = weir02_crest_level.tim
lat_cont_coeff = 1
```

12.2.2.1 Barriers

A barrier is modelled as a gate or a weir in combination with a so-called Control Group (D-Real Time Control model).

12.2.3 Thin dams

Thin dams are similar to fixed weirs. The only difference between the thin dams and fixed weirs are in their crest levels. Thin dams, in principle, include infinitely high crest levels and hence, they do not allow water flux. Similar to the other structures, the thin dams can be selected from the toolbar and drawn by a polygon. D-Flow FM adjusts the polygon to the nearest velocity points. The input data for a thin dam is identical to those for fixed-weir, except for the crest level.

12.2.4 Gates

Constructions which partially block the horizontal flow can be modelled as so-called "gates". Its horizontal and vertical position can be specified. Upstream of the gate the flow is accelerated due to contraction and downstream the flow is decelerated due to expansion. A gate may include two type of openings, namely, in horizontal and in vertical directions. In two-dimensional simulations, the vertical effect is parameterized by a quadratic energy loss term.

The horizontal effect are mimicked by setting the velocities of the computational faces (at position of the gate) to zero. This generates structure of the horizontal flow around the gate which is more realistic. There is no transport of salt or sediment through the blocked computational faces of a gate. The width of a gate is assumed to be zero, so it has no influence on the water volume.

In D-Flow FM the gates can be imposed by polygons, and can be edited in a similar way as the other structures. For more details on gates in Delta Shell, we refer you to [section 4.4.2.11](#).

The structure of the input file for the gates is as follows:

```
type      = gate
id       = gate01
polylinefile = gate01.pli
```

```

lower_edge_level      = 15
opening_width        = gate01_opening_width.tim
sill_level           = 7
door_height          = 5
horizontal_opening_direction = symmetric

```

12.2.5 Pumps

Pumps are another type of structures in D-Flow FM. Unlike the other structures, a pump can force the flow only on one direction. However, pumps can be defined by polygons, like all other structures in D-Flow FM.

The pump includes specific capacity, and pumps the water by its capacity, as long as the water level is sufficient. In the case, the water level is lower than a required value, pump will not pump any flow, despite of their capacity. The structure of the input file for the pumps is as follows:

```

type                  = pump
id                   = pump01
polylinefile         = pump01.pli
capacity             = pump01_capacity.tim
start_level_delivery_side = 0
stop_level_delivery_side = 0
start_level_suction_side = 3
stop_level_suction_side = 2
reduction_factor_no_levels = 0

```


13 Bedforms and vegetation

The terrain and vegetation exert shear stresses on the passing flow. The magnitude of the shear stress of the bed is often characterised by means of roughness coefficient of type Chézy, Manning or White-Colebrook. Within the main stream flow the shear stresses are largely determined by the local conditions of the alluvial bed (bed composition and bedform characteristics). In other areas, such the floodplains of rivers and in the intertidal areas of estuaries, the flow resistance is determined by a combination of vegetation and an alluvial bedforms or even a non-alluvial bed. To accurately represent such conditions in the numerical model, D-Flow FM has been extended with a vegetation model. Another related feature known from Delft3D-FLOW is the bedform roughness predictors; these are not available in D-Flow FM yet. These types of flow resistance may be resolved in a 2D numerical model using the trachytope approach (see [section 13.2](#)).

13.1 Bedform heights

The dune height and [Van Rijn \(2007\)](#) bedform roughness predictors, known from Delft3D-FLOW, are not available yet in D-Flow FM. They will be in an upcoming release.

13.2 Trachytopes

This functionality allows you to specify the bed roughness and flow resistance on a sub-grid level by defining and using various land use or roughness/resistance classes, further referred to as trachytopes after the Greek word *τραχύτης* for roughness. The input parameters and files to use the trachytopes functionality are described in [section B.6](#).

At every time step (or less frequent as requested by the user) the trachytopes are converted into a representative bed roughness C , k or n and optional linear flow resistance coefficient $\lambda_{m,n}$ per velocity point.

$$M_\xi = -\frac{1}{2}\lambda_{m,n}u\sqrt{u^2 + v^2} \quad (13.1)$$

$$M_\eta = -\frac{1}{2}\lambda_{m,n}v\sqrt{u^2 + v^2} \quad (13.2)$$

To save computational time the user may choose to update the computed bed roughness and resistance coefficients less frequently than every time step. See [section B.6](#) for a description of the keywords and input files associated with this feature.

The following two sections describe the various classes of trachytopes distinguished and the way in which they are combined, respectively.

13.2.1 Trachytope classes

Three base classes of trachytopes are distinguished: area classes, line classes and point classes. The area classes (type range 51–200) basically cover the whole area, therefore, they are generally the dominant roughness factor. The line classes (type range 201–250) may be used to represent hedges and similar flow resistance elements; it will add anisotropy to the roughness field. The point class (type range 251–300) represents a set of point flow resistance elements. The next six sections provide an overview of the various trachytope formulae implemented.

Special classes (1–50)

In addition to the three base classes two special trachytype classes have been defined: a flood protected area and a composite trachytype class. The first class represents a sub-grid area that is protected from flooding and thus does not contribute to the bed roughness; however, the effect on the flow resistance should be taken into account. The second class can be used to make derived trachytype classes that are a combination of two other trachytypes: an area fraction α of trachytype type T_1 and an area fraction β (often equal to $1 - \alpha$) of trachytype type T_2 .

FormNr	Name	Formula
Special classes (1–50)		
1	flood protected area	area fraction shows up as f_b in Eqs. 13.54 and 13.57
2	composite trachytype	fraction α of type T_1 and fraction β (generally $\beta = 1 - \alpha$) of type T_2

Area trachytype classes (51–200)

The class of area trachytypes is subdivided into three types: simple (51–100), alluvial (101–150) and vegetation (151–200). Four simple area trachytypes have been implemented representing the four standard roughness types of flow module.

FormNr	Name	Formula
51	White-Colebrook value	k
52	Chézy value	C
53	Manning value	$C = \sqrt[6]{h/n}$
54	z_0 value	$k = 30z_0$

Six alluvial trachytypes have been implemented.

FormNr	Name	Formula
101	simplified Van Rijn power relation	Equation 13.3
102	Van Rijn (1984c)	Equation 13.4
103	Struiksma	Equations 13.5 to 13.13
104	bedforms quadratic	Equations 13.14 to 13.17
105	bedforms linear	Equation 13.18
106		Equation 13.19

The first alluvial roughness formula is a simplified version of the [Van Rijn \(1984c\)](#) alluvial roughness predictor

$$k = Ah^{0.7} \left[1 - e^{-Bh^{-0.3}} \right] \quad (13.3)$$

it is obtained from [Equation 13.5](#) by noting that $h_b \propto h^{0.7}$ and $L_b \propto h$ and ignoring the grain related roughness. The parameters A and B can be calibrated by the user. The second

formula implemented is a straightforward general power law

$$C = Ah^B \quad (13.4)$$

where A and B are calibration coefficients. The [Van Rijn \(1984c\)](#) alluvial roughness predictor reads

$$k = k_{90} + 1.1h_b (1 - e^{-25h_b/L_b}) \quad (13.5)$$

where the bedform height h_b and length L_b are given by

$$h_b = 0.11h \left(\frac{D_{50}}{h} \right)^{0.3} (1 - e^{-T/2}) (25 - T) \quad (13.6)$$

$$L_b = 7.3h \quad (13.7)$$

where h is the local water depth and the transport stage parameter T is given by

$$T = \frac{u'^2_* - u_{*,cr}^2}{u_{*,cr}^2} \quad (13.8)$$

where u'_* is the bed shear velocity given by

$$u'^2_* = gu^2/C_{g,90}^2 \quad (13.9)$$

where

$$C_{g,90} = 18^{10} \log(12h/k_{90}) \text{ and } k_{90} = 3D_{90} \quad (13.10)$$

and $u_{*,cr}$ is the critical bed shear velocity according Shields given by

$$u_{*,cr}^2 = g\Delta D_{50}\theta_c \quad (13.11)$$

given

$$\theta_c = \begin{cases} 0.24/D_* & \text{if } D_* \leq 4 \\ 0.14D_*^{-0.64} & \text{if } 4 < D_* \leq 10 \\ 0.04D_*^{-0.10} & \text{if } 10 < D_* \leq 20 \\ 0.013D_*^{0.29} & \text{if } 20 < D_* \leq 150 \\ 0.055 & \text{if } 150 < D_* \end{cases} \quad (13.12)$$

where

$$D_* = D_{50} \left(\frac{g\Delta}{\nu^2} \right)^{1/3} \quad (13.13)$$

This predictor does not contain any calibration coefficients but requires D_{50} and D_{90} data from the morphology module. It does not include the advective and relaxation behaviour that is available by explicitly simulating the dune height as described in [Section 13.1](#) combined with trachytone number 106.

The second alluvial roughness predictor proposed by (Struiksma, pers. comm.) allows for a lot of adjustments, it reads

$$\frac{1}{C^2} = (1 - \xi) \frac{1}{C_{90}^2} + \xi \frac{1}{C_{min}^2} \quad (13.14)$$

where

$$C_{90} = A_1^{10} \log(A_2 h / D_{90}) \quad (13.15)$$

and

$$\xi = \frac{\max(0, \theta_g - \theta_c)}{\theta_m - \theta_c} \frac{\theta_m^2 - \theta_c \theta_g}{(\theta_m - \theta_c) \theta_g} \quad (13.16)$$

which varies from 0 at $\theta_g \leq \theta_c$ to 1 at $\theta_g = \theta_m$ where

$$\theta_g = \frac{u^2}{C_{90}^2 \Delta D_{50}} \quad (13.17)$$

and $A_1, A_2, \theta_c, \theta_m, C_{\min}$ are coefficients that the user needs to specify. This formula requires also D_{50} and D_{90} data from the morphology module. The fifth formula is based on [Van Rijn \(2007\)](#) and reads

$$k = \min(\sqrt{k_{s,r}^2 + k_{s,mr}^2 + k_{s,d}^2}, \frac{h}{2}) \quad (13.18)$$

It uses the roughness heights of ripples k_r , mega-ripples k_{mr} and dunes k_d . These roughness heights are based on [Van Rijn \(2007\)](#) formulae as described in [section 13.1](#); these formulae depend on D_{50} and D_{90} data which may be either specified as part of the roughness type or obtained from the morphology module. The sixth formula is similar, but uses a linear addition

$$k = \min(k_{s,r} + k_{s,mr} + k_{s,d}, \frac{h}{2}) \quad (13.19)$$

Four vegetation based area trachytopes have been implemented. Two formulae (referred to as ‘Barneveld’) are based on the work by [Klopstra et al. \(1996, 1997\)](#) and two on the work by [Baptist \(2005\)](#).

FormNr	Name	Formula
151	Barneveld 1	Eqs. 13.20 – 13.29, $C_D = 1.65$
152	Barneveld 2	Eqs. 13.20 – 13.26, 13.30 – 13.32
153	Baptist 1	Eqs. 13.33 and 13.34
154	Baptist 2	Eqs. 13.35, 13.37 and 13.38

The formula by [Klopstra et al. \(1997\)](#) reads

$$C = \frac{1}{h^{3/2}} \left\{ \begin{array}{l} \frac{2}{\sqrt{2A}} \left(\sqrt{C_3 e^{h_v \sqrt{2A}} + u_{v0}^2} \right) + \\ \frac{u_{v0}}{\sqrt{2A}} \ln \left(\frac{(\sqrt{C_3 e^{h_v \sqrt{2A}} + u_{v0}^2} - u_{v0})(\sqrt{C_3 + u_{v0}^2} + u_{v0})}{(\sqrt{C_3 e^{2\sqrt{2A}} + u_{v0}^2} + u_{v0})(\sqrt{C_3 + u_{v0}^2} - u_{v0})} \right) + \\ \frac{\sqrt{g(h-(h_v-a))}}{\kappa} \left((h - (h_v - a)) \ln \left(\frac{h - (h_v - a)}{z_0} \right) - a \ln \left(\frac{a}{z_0} \right) - (h - h_v) \right) \end{array} \right\} \quad (13.20)$$

where

$$A = \frac{n C_D}{2\alpha} \quad (13.21)$$

$$C_3 = \frac{2g(h - h_v)}{\alpha\sqrt{2A}(e^{h_v\sqrt{2A}} + e^{-h_v\sqrt{2A}})} \quad (13.22)$$

$$a = \frac{1 + \sqrt{1 + \frac{4E_1^2\kappa^2(h - h_v)}{g}}}{\frac{2E_1^2\kappa^2}{g}} \quad (13.23)$$

and

$$z_0 = ae^{-F} \quad (13.24)$$

where

$$E_1 = \frac{\sqrt{2A}C_3e^{h_v\sqrt{2A}}}{2\sqrt{C_3e^{h_v\sqrt{2A}} + u_{v0}^2}} \quad (13.25)$$

and

$$F = \frac{\kappa\sqrt{C_3e^{h_v\sqrt{2A}} + u_{v0}^2}}{\sqrt{g(h - (h_v - a))}} \quad (13.26)$$

Here, h is the water depth, h_v is the vegetation height, and $n = mD$ where m is the number of stems per square metre and D is the stem diameter. For the first implementation the parameter α in Equation 13.22 is given by

$$\alpha = \max(0.001, 0.01\sqrt{hh_v}) \quad (13.27)$$

and the velocity within the vegetation is approximated by $u_{v0}\sqrt{i}$ where

$$u_{v0}^2 = \frac{2g}{C_Dn} \quad (13.28)$$

and i is the water level gradient. For emerged vegetation the first implementation reads

$$\frac{1}{C^2} = \frac{C_Dnh}{2g} \quad (13.29)$$

The second implementation based on Klopstra *et al.* (1996) is identical except for the following modifications to Eqs. 13.27 – 13.29. The main difference between the two implementations is the inclusion of the roughness C_b of the bed itself (without vegetation). The parameter α in Equation 13.22 is now given by

$$\alpha = 0.0227h_v^{0.7} \quad (13.30)$$

and the velocity within the vegetation is approximated by $u_{v0}\sqrt{i}$ where

$$u_{v0}^2 = \frac{h_v}{\frac{C_Dhvn}{2g} + \frac{1}{C_b^2}} \quad (13.31)$$

and i is the water level gradient. For emerged vegetation the second implementation reads

$$\frac{1}{C^2} = \frac{C_D nh}{2g} + \frac{1}{C_b^2} \quad (13.32)$$

For large values of C_b the latter two equations simplify to the corresponding equations of the first implementation. The first implementation requires vegetation height h_v and density n as input parameters (the drag coefficient C_D is equal to 1.65); for second implementation you'll also need to specify the drag coefficient C_D and the alluvial bed roughness k_b (C_b in [Equation 13.32](#) is computed as $18^{10} \log(12h/k_b)$).

The first implementation of the roughness predictor by Baptist ([Baptist, 2005](#)) reads for the case of submerged vegetation

$$C = \frac{1}{\sqrt{\frac{1}{C_b^2} + \frac{C_D nh_v}{2g}}} + \frac{\sqrt{g}}{\kappa} \ln\left(\frac{h}{h_v}\right) \quad (13.33)$$

where n is the vegetation density ($n = mD$ where m is the number of stems per square metre and D is the stem diameter). The second term goes to zero at the transition from submerged to emerged vegetation. At that transition the formula changes into the formula for non-submerged vegetation which reads

$$C = \frac{1}{\sqrt{\frac{1}{C_b^2} + \frac{C_D nh}{2g}}} \quad (13.34)$$

which is identical to the non-submerged case of the second implementation of the work by [Klopstra et al. \(1996\)](#) (see [Equation 13.32](#)).

The drawback of the three vegetation based formulations above is that they parameterize the flow resistance by means of the bed roughness. Consequently, the presence of vegetation will lead to a higher bed roughness and thus to a higher bed shear stress and larger sediment transport rates in case of morphological computations. Therefore, we have included a $-\frac{\lambda}{2} u^2$ term in the momentum equation where λ represents the flow resistance of the vegetation. For the case of non-submerged vegetation $h < h_v$ the flow resistance and bed roughness are strictly separated

$$C = C_b \quad \text{and} \quad \lambda = C_D n \quad (13.35)$$

In the case of submerged vegetation $h > h_v$ the two terms can't be split in an equally clean manner. However, we can split the terms such that the bed shear stress computed using the depth averaged velocity u and the net bed roughness C equals the bed shear stress computed using the velocity u_v within the vegetation layer and the real bed roughness C_b .

$$\frac{u^2}{C^2} = \frac{u_v^2}{C_b^2} \quad (13.36)$$

With this additional requirement we can rewrite [Equation 13.33](#) as

$$C = C_b + \frac{\sqrt{g}}{\kappa} \ln\left(\frac{h}{h_v}\right) \sqrt{1 + \frac{C_D nh_v C_b^2}{2g}} \quad (13.37)$$

and

$$\lambda = C_D n \frac{h_v}{h} \frac{C_b^2}{C^2} \quad (13.38)$$

which simplify to [Equation 13.35](#) for $h = h_v$. Both formulae by Baptist require vegetation height h_v , density n , drag coefficient C_D and alluvial bed roughness C_b as input parameters.

Linear trachytope classes (201–250)

Two formulae have been implemented for linear trachytopes such as hedges or bridge piers.

FormNr	Name	Formula
201	hedges 1	Eqs. 13.39 to 13.41
202	hedges 2	Eqs. 13.42 to 13.44

The first implementation reads

$$\frac{1}{C^2} = \frac{h}{2g} \frac{L_{hedge}}{W_{cell} L_{cell}} \frac{1 - \mu^2}{\mu^2} \quad (13.39)$$

where L_{hedge} is the projected length of the hedge, W_{cell} and L_{cell} are the width and length of the grid cell. The ratio L_{hedge}/W_{cell} may be interpreted as the number of hedges that the flow encounters per unit width. The second ratio is thus the inverse of the average distance between these hedges within the grid cell. The last term may be loosely referred to as the drag of the hedge, which is determined by the hedge pass factor μ given by

$$\mu = 1 + 0.175n \left(\frac{h}{h_v} - 2 \right) \quad (13.40)$$

if the hedge extends above the water level ($h_v > h$) and is given by

$$\mu = 1 - 0.175n \left(\frac{h}{h_v} \right) \quad (13.41)$$

if the hedge is fully submerged ($h > h_v$) where n is a dimensionless hedge density. The second implementation reads

$$\frac{1}{C^2} = \frac{C_D n L_{hedge} h}{2g L_{cell} W_{cell}} \quad (13.42)$$

or equivalently

$$C = \sqrt{\frac{2gL_{cell}W_{cell}}{hL_{hedge}}} \left(\sqrt{\frac{1}{C_D n}} \right) \quad (13.43)$$

for non-submerged conditions and

$$C = \sqrt{\frac{2gL_{cell}W_{cell}}{hL_{hedge}}} \left(\frac{h_v}{h} \sqrt{\frac{1}{C_D n}} + m_0 \sqrt{\frac{\left(\frac{h-h_v}{h}\right)^2}{1 - \left(\frac{h-h_v}{h}\right)^2}} \right) \quad (13.44)$$

for submerged conditions. We recognize the same ratio $L_{cell}W_{cell}/L_{hedge}$ that represents the average distance between hedges. [Equation 13.42](#) can be directly compared to similar equations for area trachytopes ([Equation 13.29](#)), point trachytopes ([Equation 13.45](#)). Note that the formula for computing the loss coefficient for a bridge explicitly includes the reduction in the flow area and the resulting increase in the effective flow velocity, whereas the above mentioned trachytope formulae don't.

Point trachytype classes: various (251–300)

One formula for point trachytypes has been implemented. It may be used to represent groups of individual trees or on a smaller scale plants.

FormNr	Name	Formula
251	trees	Eqn. 13.45

The implemented formula reads

$$C = \sqrt{\frac{2g}{C_D n \min(h_v, h)}} \quad (13.45)$$

where $n = mD$ with m the number of trees per unit area and D the characteristic tree diameter, h_v is the vegetation height and h is the local water depth. The formula is identical to [Equation 13.34](#) except for the fact that the point trachytype formula has no bed roughness and area associated with it. The generalization of [Equation 13.45](#) to the submerged case ($h > h_v$) lacks the extra term in [Equation 13.33](#).

13.2.2 Averaging and accumulation of trachytypes

Point and linear roughnesses are accumulated by summing the inverse of the squared Chézy values C_i .

$$\frac{1}{C_{pnt}^2} = \sum_i \frac{1}{C_{pnt,i}^2} \quad (13.46)$$

$$\frac{1}{C_{lin}^2} = \sum_i \frac{1}{C_{lin,i}^2} \quad (13.47)$$

The area roughnesses are accumulated weighted by the surface area fraction f_i . These roughnesses are accumulated as White-Colebrook roughness values and as Chézy values; for the latter values both the linear sum (“parallel”) and the sum of inverse of squared values (“serial”) are computed. Roughness values are converted into each other as needed based on the local water depth.

$$k_{area} = \sum_i f_i k_i \quad (13.48)$$

$$\frac{1}{C_{area,s}^2} = \sum_i f_i \frac{1}{C_i^2} \quad (13.49)$$

$$C_{area,p} = \sum_i f_i C_i \quad (13.50)$$

For the fraction of the grid cell area for which no roughness class is specified the default roughness is used.

The flow resistance coefficients are also accumulated proportionally to the surface area fraction f_i associated with the trachytype considered. For the fraction of the grid cell area for which no flow resistance is specified, obviously none is used.

$$\lambda = \sum_i f_i \lambda_i \quad (13.51)$$

The final effective bed roughness of the grid cell may be computed by either one of the following two methods.

Method 1

The total mean roughness is computed by summing the White-Colebrook values for the areas and line and point resistance features.

$$k_m = k_{area} + k_{lin} + k_{pnt} \quad (13.52)$$

where $k_{lin} = 12h10^{-C_{lin}/18}$ and $k_{pnt} = 12h10^{-C_{pnt}/18}$. The effect of the water free area fraction f_b is taken into account by means of the following empirical relation in which $C_m = 18^{10}\log(12h/k_m)$ is the mean Chézy value corresponding to the total mean White-Colebrook roughness value obtained from [Equation 13.52](#).

$$f_b = \max(\min(0.843, f_b), 0.014) \quad (13.53)$$

$$C_{total} = C_m \left(1.12 - 0.25f_b - 0.99\sqrt{f_b} \right) \quad (13.54)$$

The resulting C_{total} value is used in the computation. This method together with trachytope classes 1, 51, 101, 151 and 201 corresponds to the NIKURADSE option of the WAQUA/TRI-WAQ flow solver.

Method 2

The total roughness is computed by first averaging over the serial and parallel averages of the Chézy values according

$$C_{area} = \alpha_s C_{area,s} + (1 - \alpha_s) C_{area,p} \quad (13.55)$$

where $\alpha_s = 0.6$ by default. Subsequently the effect of the water free area fraction f_b is taken into account by means of the following empirical relation (identical to [Equation 13.54](#) of method 1).

$$f_b = \max(\min(0.843, f_b), 0.014) \quad (13.56)$$

$$C_{area,corr} = C_{area} \left(1.12 - 0.25f_b - 0.99\sqrt{f_b} \right) \quad (13.57)$$

Finally the Chézy value representing the total bed roughness is computed by accumulating the inverses of the squared Chézy values.

$$\frac{1}{C_{total}^2} = \frac{1}{C_{area,corr}^2} + \frac{1}{C_{lin}^2} + \frac{1}{C_{pnt}^2} \quad (13.58)$$

The resulting C_{total} value is used in the computation. This method together with trachytope classes 1, 51, 52, 53, 101, 152, 202 and 251 corresponds to the ROUGHCOMBINATION option of the WAQUA/TRIWAQ flow solver.

13.3 (Rigid) three-dimensional vegetation model

The (rigid) 3D Vegetation model ([Winterwerp and Uittenbogaard \(1997\)](#)), as known from Delft3D-FLOW, is not available yet in D-Flow FM.

14 Coupling with D-Waves (SWAN)

This chapter is on the *coupling* of hydrodynamics and waves. Full documentation on D-Waves is available in its own User Manual; this chapter is limited to the details of running coupled flow-wave models, and the physical interaction processes between the two of them.

14.1 Getting started

The Delta Shell framework knows the concept of an *Integrated model* in order to couple different models, such as: hydrodynamics coupled with controlling of structures, waves, morphology and/or water quality.

Two types of coupling are distinguished: *offline* and *online* coupling. An *offline* coupling runs the entire hydrodynamic simulation first, i.e., *separately* from the second simulation. The file-based hydrodynamic output serves as input for the second simulation. As such, the hydrodynamic flow drives the controlling of structures, or the simulation of waves or water quality. In this case there is no feedback on the hydrodynamic simulation. For many applications, this is good practice.

An *online* coupling, on the other hand, exchanges data *every* time step. This tight coupling allows for direct feedback of the various processes on one another. This is crucial for controlling structures.

Note: *Offline* is also referred to as *Sequential* coupling and *online* as *Parallel* coupling.



There are two possible wave modes:

- 1 Offline: A communication file containing wave data is prepared, only D-Flow FM runs, using the wave data from the com-file.
- 2 Online: D-Flow FM computations are alternated with D-Waves calculations. D-Flow FM writes flow data to the com-file, D-Waves uses this flow data for the wave calculation and writes wave data to the com-file. D-Flow FM uses the updated wave data.

All modes are started by executing <d_hydro.exe> with a <d_hydro_config.xml> file as argument. The <d_hydro_config.xml> file prescribes the mode and when the D-Flow FM computation should be paused to perform a wave calculation. From within the working directory, the following run-script in the installation directory can be called: <run_dhydro.bat> on Windows systems and <run_dhydro.bat> on Linux systems. The scripts with base name <run_dhydro_parallel> are variants to start a parallel calculation.

14.1.1 Input D-Flow FM

Optionally add the following lines to the .mdu file:

```
[numerics]
Epshu      = 0.05 # Input for threshold water depth for wet and dry cells

[waves]
WavemodeLnr = 3    # Wave model nr, 0=no, 1=fetch/depth limited hurdlesteive,
                    2=youngverhagen, 3 = D-Waves, 4=wave group forcing
Rouwav     = FR84
Gammax     = 0.5   # Maximum wave height/water depth ratio

[output]
```

```
EulerVelocities = 1      # 0 (default): GLM, 1: Euler. Only relevant when using waves
```

Description:

Wavemodelnr	Key switch to enable wave modelling. Use “3” for wave data from D-Waves (online or offline) and passing hydrodynamic data to D-Waves (online only). A file will be generated automatically named <“runid”_com.nc> to exchange data.
Rouwav	Necessary to include bed shear-stress enhancement by waves. See also Delft3D-FLOW manual.
EulerVelocities	Optional flag to write Eulerian velocities to the D-Flow FM map file. Currently, only Eulerian values will be written for the cell centre velocity x-component and y-component (parameters ucx and ucy). Check that the “long_name” contains the word “Eulerian”.
Epshu	Optionally overwrite the default value of “1.0e-4”. Depending on your model, the default value of Epshu in combination with modelling waves may lead to huge local velocities near (almost) dry points. This will result in very small time steps. Increasing Epshu might be a reasonable workaround.
Gammax	Optionally overwrite the default value of “1.0”. Depending on your model, the default value of Gammax may lead to huge local velocities in shallow water. Decreasing Gammax might be a reasonable workaround.

14.1.2 Input D-Waves

Optionally add the following lines to the .mdw file:

```
[Output]
MapWriteNetCDF      = true
COMFile            = ../fm/dflowfmoutput/fff_com.nc
NetCDFSinglePrecision = false
locationFile       = ../fm/loo_obs.xyn
writeTable          = true
```

Description:

MapWriteNetCDF	Default value “false”, resulting in no output written in NetCDF format. The coupling with D-Flow FM is only implemented for NetCDF format, so this flag must be set to “true” when being coupled with D-Flow FM.
COMFile	Necessary reference to the file being used to communicate data to and from D-Flow FM. The name must exactly match with the name of the com-file being generated by D-Flow FM.
NetCDFSinglePrecision	Optional flag to write data in single precision instead of double precision. Default value is “false”. Might be useful to decrease the size of the com-file or to compare with a Delft3D-FLOW computation.
locationFile	Optional reference to observation points in D-Flow FM. When in combination with the “writeTable” flag, D-Waves will produce a history file in NetCDF format for these observation points.

writeTable	Optional flag to force SWAN to produce an output file in table format for each set of locations specified in a locationFile. Default value is "false".
------------	--

14.1.3 Input d_hydro

Both D-Flow FM and D-Waves are used as dynamic libraries (DLL's on Windows, so's on Linux). d_hydro is a small executable steering both dynamic libraries. Its input file, usually called "d_hydro_config.xml", looks like this:

```

<?xml version="1.0" encoding="iso-8859-1"?>
<deltaresHydro xmlns="http://schemas.deltares.nl/deltaresHydro"
    xmlns:xsi="http://www.w3.org/2001/XMLSchema-instance"
    xsi:schemaLocation="http://schemas.deltares.nl/deltaresHydro
        http://content.oss.deltares.nl/schemas/d_hydro-1.00.xsd">
    <documentation>
        <fileVersion>1.00</fileVersion>
        <createdBy>Deltares, Coupling team</createdBy>
        <creationDate>2015-05-20T07:56:32+01</creationDate>
    </documentation>
    <control>
        <parallel>
            <startGroup>
                <time>0.0 60.0 99999999.0</time>
                <coupler name="flow2rtc"/>
                <start name="myNameRTC"/>
                <coupler name="rtc2flow"/>
            </startGroup>
            <startGroup>
                <time>0.0 3600.0 99999999.0</time>
                <start name="myNameWave"/>
            </startGroup>
            <start name="myNameDFlowFM"/>
        </parallel>
    </control>

    <component name="myNameDFlowFM">
        <library>dflowfm</library>
        <process>0 1 2</process>
        <mpiCommunicator>DFM_COMM_DFMWORLD</mpiCommunicator>
        <workingDir>fm</workingDir>
        <inputFile>weirtimeseries.mdu</inputFile>
    </component>
    <component name="myNameWave">
        <library>wave</library>
        <process>0</process>
        <workingDir>wave</workingDir>
        <!-- component specific -->
        <inputFile>weir.mdw</inputFile>
    </component>
    <component name="myNameRTC">
        <library>RTCTools_BMI</library>
        <process>0</process>
        <workingDir>rtc</workingDir>
        <!-- component specific -->
        <inputFile>.</inputFile>
    </component>

    <coupler name="flow2rtc">
        <sourceComponent>myNameDFlowFM</sourceComponent>
        <targetComponent>myNameRTC</targetComponent>
        <item>
            <sourceName>observations/Upstream/water_level</sourceName>
            <targetName>input_ObservationPoint01_water_level</targetName>
        </item>
    </coupler>

```

```
</coupler>
<coupler name="rtc2flow">
    <sourceComponent>myNameRTC</sourceComponent>
    <targetComponent>myNameDFlowFM</targetComponent>
    <item>
        <sourceName>output_weir_crest_level</sourceName>
        <targetName>weirs/weir01/crest_level</targetName>
    </item>
</coupler>
</deltasHydro>
```

Description:

- <**control**> Specifies the workflow of the deltasHydro executable. It indicates which components are started in which order. If the data transfer is to be arranged by the main program d_hydro, then a coupler should be included. The main <control> block is a sequential block; this means that each component is initialized, time stepped, and finalized before the next component starts. For each component/coupler listed inside the <control> block there will be a corresponding component/ coupler specification block defined below.
- <**parallel**> Within a <parallel> tag the components are started concurrently (if the mpi process ids listed per component don't overlap) or executed synchronously in sequence (first all initialize, then time stepping, and to conclude all finalization calls). The order of the components is retained.
- <**start**> A <parallel> block contains exactly one <start/> component, defining the start and end time of the simulation. This is the component inside the <parallel> block with the smallest time step and can be denoted as the "master-component". All other components must be defined with a <startGroup> and can be denoted as a "slave-component".
- <**startGroup**> A <startGroup> should be used if a component (possibly including couplers) should only be executed at a subset of simulation time steps.
- <**time**> Start-, step- and stop-time (in seconds) at which this slave-component should be executed. The times are relative to the times of the master-component. Thus a start-time of 0.0 always refers to the start time of the master-component and a stop-time of "infinity" always refers to the end time of the master-component.
- <**component name="myComponentName"**> Component specification block. "myComponentName" is free to be defined by the user. It must match exactly with the reference in the <control> block above. The name of a component must be unique.
- <**library**> Reference to the component to be executed. Currently "flowfm", "wave" and "RTCTools_BMI" are supported. The name must match exactly the name of the related dll/so (excluding prefixes (e.g. "lib") and suffixes (e.g. ".dll" or ".so")). The library should be located in the search path, or it may include an absolute or relative path. Multiple <component> blocks may refer to the same component to be executed.
- <**process**> Optional list of the ids of the mpi processes that should be used to run the component. If not specified, then the component will run only in process "0" (i.e. non-parallel). The processes may be specified as a space separated list with series compressed using colons e.g. "16:31" represents processes "16 17 18" up to "31".
- <**mpiCommunicator**> D-Flow FM specific flag. Mandatory only when this D-Flow FM component should run a parallel (partitioned) model. Note that this is *unrelated* to the <parallel> tag as introduced above.
- <**workingDir**> Specification of the working directory of this <component>, relative to the location of the "d_hydro_config.xml" file. The workingDir is the base/root directory of all input and output files for the component. All other files will be located

RELATIVE TO this folder. If not specified, then workingDir will be equal to the folder of the configuration file.

<inputFile> Specification of the input file of this <component>, relative to <workingDir>:
D-Flow FM: mdu-file, D-Waves: mdw-file, D-RTC: .

<coupler name="myCouplerName"> Coupler specification block. "myCouplerName" is free to be defined by the user. It must match exactly with the reference in the <control> block above. The name of a coupler must be unique.

<sourceComponent> Identifies which component provides the data.

<targetComponent> Identifies which component needs to receive the data. The coupler runs on the superset of the processes configured for the source and target components.

<item> For each quantity to be exchanged, the name in the source component and the name in the target component are specified. To support recursive components, a directory syntax is used with forward slashes. If a name includes a forward slash, then it needs to be escaped using a backward slash, like \; if a name includes a backward slash, then it needs to be escaped with a backward slash, like \\.

<sourceName> Identifies which parameter will be sent at the source component side.

<targetName> Identifies which parameter will be received at the target component side.

14.1.4 Online process order

When D-Flow FM runs online in alternation with D-Waves and D-RTC, process steps can be identified, where the order as prescribed in "d_hydro_config.xml" is respected as much as possible. The process order of the example "d_hydro_config.xml" file in the section above is:

- 1 D-Flow FM::Init
D-Flow FM is initialized. Grid- and Flow-data is written to the NetCDF com-file.
Even though D-Flow FM is not the first component in the example config-file, it must be the first component to be initialized, because the initialization results may be needed when other components are initialized. This is implemented as an exceptional rule: The master-component is always the first component to be initialized.
- 2 D-RTC::Init
D-RTC is initialized using (only) D-RTC input. Data from other components is not initialized yet; default values are used.
- 3 D-Waves::Init
D-Waves is initialized. Grid- and Flow-data created by D-Flow FM::Init is read from the com-file. Grid conversion factors are generated. The factors for the conversion from the (structured!) wave grid to the (unstructured) flow grid are calculated inside D-Waves. The factors for the conversion from the (unstructured) flow grid to the (structured) wave grid are calculated by "ESMF_RegridWeightGen" via the execution of script <ESMF_RegridWeightGen_in_Delft3D-WAVE.bat> (Windows) or <ESMF_RegridWeightGen_in_Delft3D-WAVE.sh> (Linux).
- 4 D-RTC::Step
A D-RTC computation is performed at time "0.0": Data is communicated (by d_hydro) from D-Flow FM to D-RTC, the actual D-RTC computation is executed, data is communicated (by d_hydro) from D-RTC to D-Flow FM.
- 5 D-Waves::Step
A D-Waves calculation is performed at time "0.0": Flow data is read (by D-Waves) from the com-file and converted to the wave grid(s). SWAN input files are written, a SWAN calculation is started by executing script <swan.bat> (Windows) or <swan.sh> (Linux), SWAN output files are read, wave-data is converted to the flow grid and written to the com-file (by D-Waves).

6 D-Flow FM::Step

A D-Flow FM calculation is performed: The com-file is checked for wave data to be used/updated. The simulation time will proceed from “0.0” to “60.0” (next time that another component should be executed). Flow-data is written to the com-file (by D-Flow FM).

7 D-RTC::Step, D-Flow FM::Step

A D-RTC computation is performed after every “60.0” seconds. Then a D-Flow FM computation is performed and the simulation time will proceed another “60.0” seconds. This continues until the simulation time has progressed for “3600.0” seconds.

8 D-RTC::Step, D-Waves::Step, D-Flow FM::Step

A D-RTC computation is performed, then a D-Waves computation, then a D-Flow FM computation. The simulation time will proceed another “60.0” seconds and the next D-RTC computation can start.

9 D-RTC::Finish

D-RTC is finished at the end of the simulation.

10 D-Waves::Finish

D-Waves is finished at the end of the simulation.

11 D-Flow FM::Finish

D-Flow FM is finished at the end of the simulation.

14.1.5 Related files

Below is an overview of the related files in the default directory structure. In this example the runid is set to foo.

```
<testcaseroot>
  d_hydro_config.xml Configuration file, input for d_hydro
  <fm>                 Work directory for D-Flow FM
    foo.mdu      D-Flow FM input file
    <dflowfmoutput> D-Flow FM output directory
      foo_com.nc Communication file, written/read by both D-Flow FM and D-Waves
  <rtc>                 Work directory for D-RTC
    settings.json D-RTC input file (must have exactly this name)
    <xml_dir>       D-RTC directory containing xml input/output files and csv output file
    <xsd_dir>       D-RTC directory containing xsd input files
  <wave>                 Work directory for D-Waves
    foo.mdw      D-Waves input file
    TMP_ESMF*_source.nc Temporary input file for ESMF_Regridder, created by D-Waves
    TMP_ESMF*_destination.nc Temporary input file for ESMF_Regridder, created by
                                  D-Waves
    TMP_ESMF*_weights.nc Resulting weights file, created by ESMF_Regridder, read by
                                  D-Waves
    PETO.RegridWeightGen.Log ESMF_Regridder log file
    swn-diag.foo   SWAN log file
```

14.2 Forcing by radiation stress gradients

The momentum equation in x -direction, averaged over the wave motion and expressed in GLM co-ordinates is given by:

$$\frac{\partial \bar{u}_j^L}{\partial t} + \bar{u}_i^L \frac{\partial \bar{u}_j^L}{\partial x_i} + \dots + g \frac{\partial \bar{\zeta}}{\partial x_j} - \frac{1}{\rho} \frac{\partial \bar{\tau}_{ij}^L}{\partial x_i} = F_j^L, \quad (14.1)$$

where for i and j the summation rule applies, $i, j = \{1, 2, 3\}$. As shown by [Groeneweg \(1999\)](#) the right-hand side of [Equation 14.1](#) contains a term related to a Stokes correction of the shear stresses. In the current implementation this term is neglected.

The wave-induced force, i.e. the right-hand side of [Equation 14.1](#), can be expressed in the wave parameters of the wave model that is being applied. For linear current refraction the expression can be derived analytically. To account for wave dissipation due to for instance bottom friction, wave breaking and whitecapping and wave growth due to wind one can rely on mild slope formulations with dissipation terms.

As shown by [Dingemans et al. \(1987\)](#), using the gradients of the radiation stresses in numerical models can result in spurious currents. [Dingemans et al. \(1987\)](#) showed that the divergence free part of the radiation stress is not capable of driving currents and can therefore be neglected if one is primarily interested in wave-driven currents. The remaining part of the radiation stress gradients is closely related to the wave energy dissipation, i.e. the right-hand side of [Equation 14.1](#) can be written as:

$$F_i = \frac{Dk_i}{\omega}, \quad (14.2)$$

where D is the total energy dissipation due to waves, k_i is the wave number in i -direction and ω is the wave frequency; see [Dingemans \(1997\)](#) for many details and discussions on this subject.

2D implementation

For a depth averaged model the momentum equations in x - and y -direction, leaving out most of the terms, can be written as:

$$\frac{\partial U}{\partial t} + \dots + \frac{gU\sqrt{U^2 + V^2}}{C_{2D}^2 h} + \dots = \dots + F_x, \quad (14.3)$$

$$\frac{\partial V}{\partial t} + \dots + \frac{gV\sqrt{U^2 + V^2}}{C_{2D}^2 h} + \dots = \dots + F_y, \quad (14.4)$$

where F_x and F_y are the depth averaged wave-induced forcings and given by the gradients of the radiation stress tensor S , or following [Dingemans et al. \(1987\)](#) approximated by wave energy dissipation:

$$F_x = -\frac{\partial S_{xx}}{\partial x} - \frac{\partial S_{yx}}{\partial y} = D \frac{k_x}{\omega}, \quad (14.5)$$

$$F_y = -\frac{\partial S_{xy}}{\partial x} - \frac{\partial S_{yy}}{\partial y} = D \frac{k_y}{\omega}. \quad (14.6)$$

The dissipation rate D (a negative quantity) is computed by the wave model and read from the communication file. In SWAN, the dissipation rate may be computed from the bottom friction (orbital motion), depth-induced breaking and whitecapping.

You can choose to apply the radiation stress or the dissipation rate to determine the wave-induced forces.

3D implementation

There is no coupling available yet for 3D D-Flow FM models and D-Waves.

14.3 Stokes drift and mass flux

In surface waves, fluid particles describe an orbital motion. The net horizontal displacement for a fluid particle is not zero. This wave induced drift velocity, the so-called Stokes-drift, is always in the direction of wave propagation. A particle at the top of the orbit beneath a wave crest moves slightly faster in the forward direction than it does in the backward direction beneath a wave trough. The mean drift velocity is a second order quantity in the wave height. The drift leads to additional fluxes in the wave averaged mass continuity equation.

The wave-induced mass fluxes M_x^S and M_y^S are found by integration of the components of the Stokes drift u^S and v^S over the wave-averaged total water depth:

$$M_x^S = \int_0^{\bar{h}} \rho_0 u^S dz = \frac{E}{\omega} k_x \quad (14.7)$$

$$M_y^S = \int_0^{\bar{h}} \rho_0 v^S dz = \frac{E}{\omega} k_y \quad (14.8)$$

with E the wave energy defined as:

$$E = \frac{1}{8} \rho_0 g H_{rms}^2. \quad (14.9)$$

The mass fluxes M_x^S and M_y^S are computed by an interface program and are written to the communication file.



Remarks:

- ◊ The mass flux effect is only taken into account when D-Flow FM is used from within Delft3D-MOR.
- ◊ The velocities written to the communication file for use in Delft3D-MOR, D-Waves, and D-Water Quality are based on the *total flux* velocities.
- ◊ The Eulerian velocities, which may be used in comparisons with measurements at a fixed location, are written to the hydrodynamic map and history files.

2D implementation

The depth-averaged Stokes drift is given by:

$$U^S = \frac{M_x^S}{\rho_0 \bar{h}}, \quad (14.10)$$

$$V^S = \frac{M_y^S}{\rho_0 \bar{h}}. \quad (14.11)$$

3D implementation

There is no coupling available yet for 3D D-Flow FM models and D-Waves.

14.4 Streaming

Streaming is a 3D feature. There is still no coupling available for 3D D-Flow FM models and D-Waves.

14.5 Enhancement of the bed shear-stress by waves

The boundary layers at the bed associated with the waves and the current interact non-linearly. This has the effect of enhancing both the mean and oscillatory bed shear-stresses. In addition the current profile is modified, because the extra turbulence generated close to the bed by the waves appears to the current as being equivalent to an enhanced bottom roughness. The bed shear-stress due to the combination of waves and current is enhanced beyond the value which would result from a linear addition of the bed shear-stress due to waves, $\vec{\tau}_w$, and the bed shear-stress due to current $\vec{\tau}_c$. For sediment transport modelling it is important to predict the maximum bed shear-stress, $\vec{\tau}_{\max}$, while the current velocity and the turbulent diffusion are determined by the combined wave-current bed shear-stress $\vec{\tau}_m$.

Various, often very complex, methods exist to describe the bottom boundary layer under combined current and wave action and the resulting virtual roughness. [Soulsby et al. \(1993a\)](#) developed a parameterisation of these methods allowing a simple implementation and comparison of various wave-current interaction models: [Fredsøe \(1984\)](#); [Myrhaug and Slaattelid \(1990\)](#); [Grant and Madsen \(1979\)](#); [Huynh-Thanh and Temperville \(1991\)](#); [Davies et al. \(1988\)](#); [Bijker \(1967\)](#); [Christoffersen and Jonsson \(1985\)](#); [O' Connor and Yoo \(1988\)](#); [Van Rijn et al. \(2004\)](#). All these methods have all been implemented in D-Flow FM and can be applied in 2D and 3D modelling. However, as there are minor, but specific differences in determining certain quantities, such as determining the shear-stress at the bottom, we prefer to discuss the 2D and 3D implementation separately.

2D implementation

Following [Soulsby et al. \(1993b\)](#), Figure 14.1 gives a schematic overview of the bed shear-stresses for wave current interaction.

[Soulsby et al. \(1993b\)](#) fitted one standard formula to all of the models, each model having its own fitting coefficients. The parameterisation of Soulsby for the time-mean bed shear-stress is of the form:

$$|\vec{\tau}_m| = Y (|\vec{\tau}_c| + |\vec{\tau}_w|), \quad (14.12)$$

with

$$Y = X \{1 + bX^p(1 - X)^q\}, \quad (14.13)$$

and for the maximum bed shear-stress:

$$|\vec{\tau}_{\max}| = Z (|\vec{\tau}_c| + |\vec{\tau}_w|), \quad (14.14)$$

with

$$Z = 1 + aX^m(1 - X)^n. \quad (14.15)$$

and:

$$X = \frac{|\vec{\tau}_c|}{|\vec{\tau}_c| + |\vec{\tau}_w|}, \quad (14.16)$$

The value of the parameters a , b , p , q , m and n depends on the friction model which is parameterised, and:

$ \vec{\tau}_c $	magnitude of the bed stress due to current alone
$ \vec{\tau}_w $	magnitude of the bed stress for waves alone
$ \vec{\tau}_m $	magnitude of the mean bed stress for combined waves and current

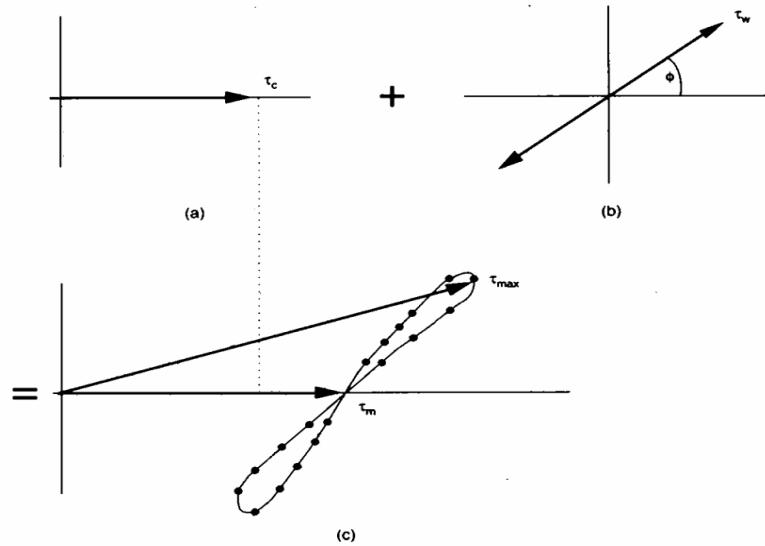


Figure 14.1: Schematic view of non-linear interaction of wave and current bed shear-stresses (from [Soulsby et al. \(1993b, Figure 16, p. 89\)](#))

$|\vec{\tau}_{\max}|$ magnitude of the maximum bed stress for combined waves and current.

Remark:

- ◊ The stresses $\vec{\tau}_m$ and $\vec{\tau}_{\max}$ are assumed to have the same direction as $\vec{\tau}_c$.



Following [Soulsby et al. \(1993b\)](#) the expressions for the parameters χ ($= a, b, p, q, m, n$) and \mathcal{J} ($= I, J$; also depending on the friction model) have the form:

$$\chi = \left(\chi_1 + \chi_2 |\cos \phi|^{\mathcal{J}} \right) + \left(\chi_3 + \chi_4 |\cos \phi|^{\mathcal{J}} \right)^{10} \log \left(\frac{f_w}{C_{2D}} \right), \quad (14.17)$$

in which:

- | | |
|----------|--|
| C_{2D} | drag coefficient due to current |
| f_w | wave friction factor |
| ϕ | the angle between the current direction and the direction of wave propagation. |

As the radiation stress is always in the wave direction, we can derive ϕ from:

$$|\cos \phi| = \frac{|UF_x + VF_y|}{|\vec{U}| |\vec{F}|}. \quad (14.18)$$

Values of the parameters a, b, p, q and J in [Equation 14.17](#) have been optimised by [Soulsby et al. \(1993b\)](#), see [Table 14.1](#) and [Figure 14.2](#).

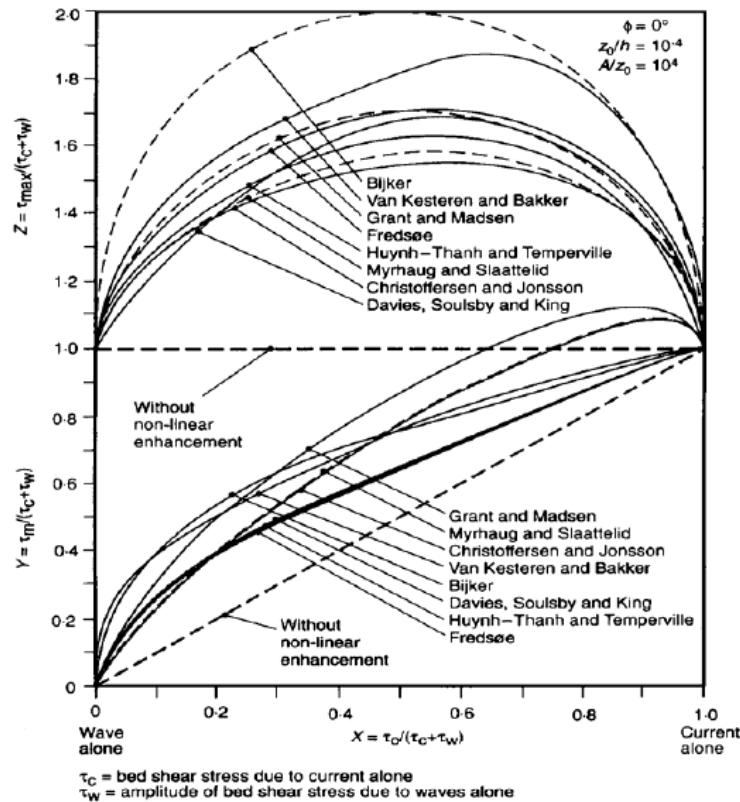
The bed shear-stress due to flow alone may be computed using various types of formulations like Chézy, Manning or White-Colebrook. The bed shear-stress due to current alone can be written in the form:

$$\vec{\tau}_c = \frac{g\rho_0 \vec{U} |\vec{U}|}{C_{2D}^2}. \quad (14.19)$$

Table 14.1: Fitting coefficients for wave/current boundary layer model

Model ¹	a_1	a_2	a_3	a_4	m_1	m_2	m_3	m_4	n_1	n_2	n_3	n_4	I
FR84	-0.06	1.70	-0.29	0.29	0.67	-0.29	0.09	0.42	0.75	-0.27	0.11	-0.02	0.80
MS90	-0.01	1.84	-0.58	-0.22	0.63	-0.09	0.23	-0.02	0.82	-0.30	0.19	-0.21	0.67
HT91	-0.07	1.87	-0.34	-0.12	0.72	-0.33	0.08	0.34	0.78	-0.23	0.12	-0.12	0.82
GM79	0.11	1.95	-0.49	-0.28	0.65	-0.22	0.15	0.06	0.71	-0.19	0.17	-0.15	0.67
DS88	0.05	1.62	-0.38	0.25	1.05	-0.75	-0.08	0.59	0.66	-0.25	0.19	-0.03	0.82
BK67	0.00	2.00	0.00	0.00	0.00	0.50	0.00	0.00	0.00	0.50	0.00	0.00	1.00
CJ85	-0.01	1.58	-0.52	0.09	0.65	-0.17	0.18	0.05	0.47	-0.03	0.59	-0.50	0.64
OY88	-0.45	2.24	0.16	-0.09	0.71	0.27	-0.15	0.03	1.19	-0.66	-0.13	0.12	0.77
	b_1	b_2	b_3	b_4	p_1	p_2	p_3	p_4	q_1	q_2	q_3	q_4	J
FR84	0.29	0.55	-0.10	-0.14	-0.77	0.10	0.27	0.14	0.91	0.25	0.50	0.45	3.00
MS90	0.65	0.29	-0.30	-0.21	-0.60	0.10	0.27	-0.06	1.19	-0.68	0.22	-0.21	0.50
HT91	0.27	0.51	-0.10	-0.24	-0.75	0.13	0.12	0.02	0.89	0.40	0.50	-0.28	2.70
GM79	0.73	0.40	-0.23	-0.24	-0.68	0.13	0.24	-0.07	1.04	-0.56	0.34	-0.27	0.50
DS88	0.22	0.73	-0.05	-0.35	-0.86	0.26	0.34	-0.07	-0.89	2.33	2.60	-2.50	2.70
BK67	0.32	0.55	0.00	0.00	-0.63	0.05	0.00	0.00	1.14	0.18	0.00	0.00	3.00
CJ85	0.47	0.29	-0.09	-0.12	-0.70	0.13	0.28	-0.04	1.65	-1.19	-0.42	0.49	0.60
OY88	-0.06	0.26	0.08	-0.03	-1.00	0.31	0.25	-0.26	0.38	1.19	0.25	-0.66	1.50
VR04	$Y = 0.0$ and $Z = 1.0$												

¹ FR84=Fredsøe (1984), MS90=Myrhaug and Slaattelid (1990),
 HT91=Huynh-Thanh and Temperville (1991), GM79=Grant and Madsen (1979), DS88=Davies *et al.* (1988),
 BK67=Bijker (1967), CJ85=Christoffersen and Jonsson (1985), OY88=O' Connor and Yoo (1988),
 VR04=Van Rijn *et al.* (2004)

**Figure 14.2:** Inter-comparison of eight models for prediction of mean and maximum bed shear-stress due to waves and currents (from Soulsby *et al.* (1993b, Figure 17, p. 90))

The magnitude of the wave-averaged bed shear-stress due to waves alone is related to the wave orbital velocity near the bottom \vec{u}_{orb} and the friction coefficient f_w :

$$|\vec{\tau}_w| = \frac{1}{2} \rho_0 f_w u_{orb}^2. \quad (14.20)$$

The orbital velocity is computed from the linear wave theory and is given by:

$$u_{orb} = \frac{1}{4} \sqrt{\pi} \frac{H_{rms}\omega}{\sinh(kH)}, \quad (14.21)$$

where the root-mean-square wave height H_{rms} and the wave period $T (= 2\pi/\omega)$ are read from the communication file. The variation of the wave friction factor with relative orbital excursion at the bed under purely oscillatory flow is given by [Swart \(1974\)](#) (\equiv Equations 17.69, 17.108 and 17.149):

$$f_w = \begin{cases} 0.00251 \exp \left[5.21 \left(\frac{A}{k_s} \right)^{-0.19} \right], & \frac{A}{k_s} > \frac{\pi}{2}, \\ 0.3, & \frac{A}{k_s} \leq \frac{\pi}{2}, \end{cases} \quad (14.22)$$

with:

$$A = \frac{u_{orb}}{\omega}, \quad (14.23)$$

k_s is the Nikuradse roughness length-scale and ω is the wave angular frequency.

As the bed is in rest and the equations are formulated in GLM co-ordinates, we must correct the bed shear-stress used in the momentum equations for the Stokes drift. The total or effective bed shear stress is given by:

$$\vec{\tau}_b = \frac{|\vec{\tau}_m|}{|\vec{U}|} \left(\vec{U} - \vec{U}^S \right), \quad (14.24)$$

where the components of the depth-averaged Stokes drift \vec{U}^S are given by Eqs. [14.10](#) and [14.11](#).

3D implementation

There is no coupling available yet for 3D D-Flow FM models and D-Waves.

15 Coupling with D-RTC (RTC-Tools)

This chapter is on the *coupling* of hydrodynamics and real-time control of hydraulic structures.

15.1 Introduction

The Delta Shell framework knows the concept of an *Integrated model* in order to couple different models, such as: hydrodynamics coupled with controlling of structures, waves, morphology and/or water quality.

Two types of coupling are distinguished: *offline* and *online* coupling. An *offline* coupling runs the entire hydrodynamic simulation first, i.e., *separately* from the second simulation. The file-based hydrodynamic output serves as input for the second simulation. As such, the hydrodynamic flow drives the controlling of structures, or the simulation of waves or water quality. In this case there is no feedback on the hydrodynamic simulation. For many applications, this is good practice.

An *online* coupling, on the other hand, exchanges data *every time step*. This tight coupling allows for direct feedback of the various processes on one another. This is crucial for controlling structures.

Note: *Offline* is also referred to as *Sequential* coupling and *online* as *Parallel* coupling.



A coupled flow-rtc model can be run either as an Integrated Model from within Delta Shell, or from the commandline using the d_hydro program.

In case of a flow-rtc coupling

- ◊ the flow model will exchange for example *Water levels*
- ◊ subsequently, the rtc model will evaluate for example the *Crest level* of a controlled structure, based on the exchanged *water levels*
- ◊ in case of an *online* coupling this *Crest level* will influence the simulation of the flow of water

15.2 Getting started

15.2.1 User interface: the first steps

First, the user must add a *real-time control* model to the *flow-fm* model. The **Project** window will look like this.

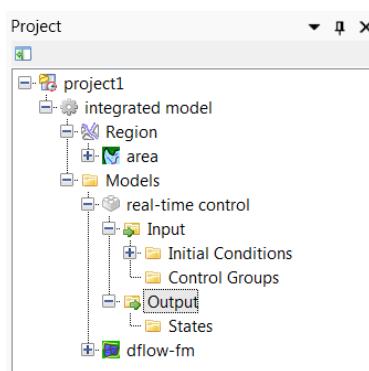


Figure 15.1: An Integrated model in the **Project** window

Secondly, the user will model the control flow for an hydraulic structure. This may look like this.

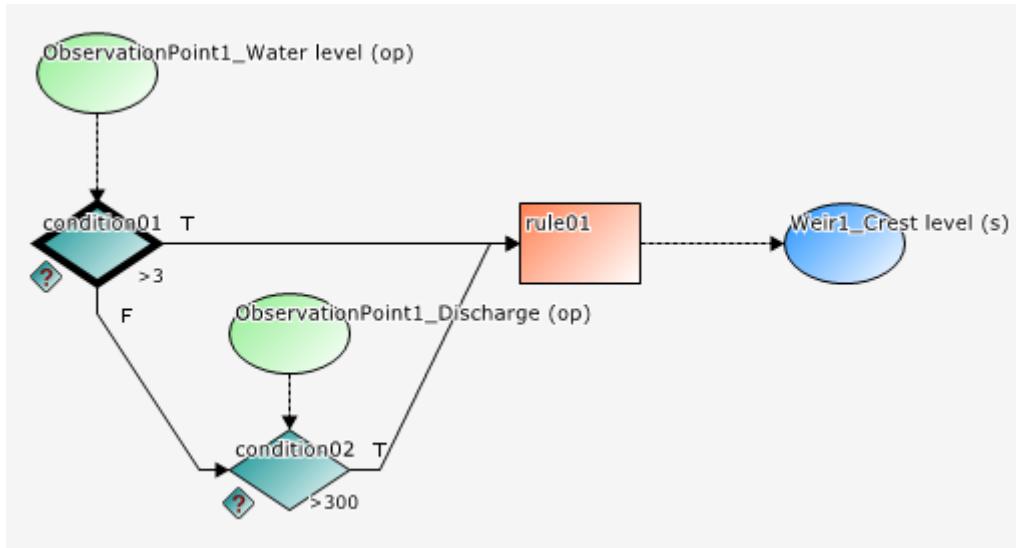


Figure 15.2: Example of a Control flow in D-RTC

Full documentation on D-RTC is available in its own User Manual. This chapter is limited to the details of running coupled flow-rtc models.

15.2.2 Input D-Flow FM

The hydraulic structures that are normally driven by time series in <.tim> files, will now be fed by D-RTC. Replace the timeseries file name by the REALTIME keyword in the <structures.ini> file ([section B.9](#)):

```

[structure]
type          = weir           # Type of structure
id            = weir01         # Name of the structure
polylinefile  = weir01.pli    # *.pli; Polyline geometry definition for 2D structure
crest_level   = REALTIME      # Crest height in [m]
  
```

Also, the MDU file may optionally contain an observation file and/or a cross section file, such that D-RTC's triggers can be set to the observed time series at these locations.

```

[output]
ObsFile       = obs.xyn
CrsFile       = river_crs.pli
  
```

15.2.3 Input D-RTC

The input of D-RTC consists of several <.xml> files and a toplevel <settings.json>. We refer to the D-RTC User Manual for further details.

15.2.4 Input d_hydro

In [Section 14.1.3](#), details on a coupled flow-rtc-wave model are presented.

15.2.5 Online process order

This is discussed in [Section 14.1.4](#).

16 Coupling with D-Water Quality (Delwaq)

16.1 Introduction

D-Water Quality is a multi-dimensional water quality model framework developed by Deltares over the past decades. It solves the advection-diffusion-reaction equation on a predefined computational grid and for a wide range of model substances. D-Water Quality offers flexible configuration of the substances to be included, as well as the processes to be evaluated. D-Water Quality is not a hydrodynamic model, so information on flow fields is obtained from hydraulic models such as D-Flow 1D (SOBEK 3) and D-Flow FM.

Here, only the *coupling* is discussed. Full documentation on D-Water Quality is available.

16.2 Offline versus online coupling

The Delta Shell framework knows the concept of an *Integrated model* in order to couple different models, such as: hydrodynamics coupled with controlling of structures, waves, morphology and/or water quality.

Two types of coupling are distinguished: *offline* and *online* coupling. An *offline* coupling runs the entire hydrodynamic simulation first, i.e., *separately* from the second simulation. The file-based hydrodynamic output serves as input for the second simulation. As such, the hydrodynamic flow drives the controlling of structures, or the simulation of waves or water quality. In this case there is no feedback on the hydrodynamic simulation. For many applications, this is good practice.

An *online* coupling, on the other hand, exchanges data *every time step*. This tight coupling allows for direct feedback of the various processes on one another. This is crucial for controlling structures.

Note: *Offline* is also referred to as *Sequential* coupling and *online* as *Parallel* coupling.



The *online* coupling is scheduled for a future release.

16.3 Creating output for D-Water Quality

Creating output for D-Water Quality by D-Flow FM can be enabled in Delta Shell in the main window under the tab 'Output Parameters'. Change the "WAQ output interval" to a nonzero value. The MDU-equivalent is the keyword [output], WaqInterval. This interval should be a whole number of seconds, and must be a multiple of the "User time step" (see tab "Time frame"). D-Flow FM will create a special output folder named <DFM_DELWAQ_<mdident>>. In this folder a <.hyd>-file will be created that gives an overview of the coupling with references to the files containing the hydrodynamic exchange data.

Load the <.hyd>-file in the D-Water Quality GUI to prepare the input for a water quality calculation. For further information on how to run D-Water Quality please consult its user manual.

16.4 Current limitations

The coupling between D-Flow FM and D-Water Quality has currently been tested with good results for 2D and 3D meshes without domain decomposition. A future release will contain some minor improvements for regions with wetting and drying mesh cells.

The D-Water Quality GUI fully supports coupling with results from D-Flow FM. Postprocessing can be done in Delft3D-QUICKPLOT.

It can be useful to model the water quality processes on a coarser mesh than the hydrodynamic grid, and for this grid aggregation can be used in the near future. An upcoming release of D-Flow FM will contain further details on this topic.

17 Sediment transport and morphology

Note: The implementation of sediment transport and morphological processes is currently not yet part of the standard release of D-Flow FM. Please contact support@deltaressystems.nl for more information.



17.1 General formulations

17.1.1 Introduction

The sediment transport and morphology module supports both bedload and suspended load transport of non-cohesive sediments and suspended load of cohesive sediments. For schematisation we distinguish "mud" (cohesive suspended load transport), "sand" (non-cohesive bedload and suspended load transport) and "bedload" (non-cohesive bedload only or total load transport) fractions. A model may contain a mixture of up to 99 suspended (i.e. "sand" and "mud") fraction and an arbitrary amount of "bedload" fractions if computer memory and simulation time allows. The only difference between "bedload" and "sand" fractions lies in the fact that the suspended load advection-diffusion equation is not solved for the "bedload" fraction. If the suspended load is known to be negligible (either due to sediment diameter or sediment transport formula chosen), the "bedload" approach is more efficient. Sediment interactions are taken into account although especially in the field of sand-mud interactions still a couple of processes are lacking.

17.1.2 Suspended transport

Three-dimensional transport of suspended sediment is calculated by solving the three-dimensional advection-diffusion (mass-balance) equation for the suspended sediment:

$$\frac{\partial c^{(\ell)}}{\partial t} + \frac{\partial u c^{(\ell)}}{\partial x} + \frac{\partial v c^{(\ell)}}{\partial y} + \frac{\partial (w - w_s^{(\ell)}) c^{(\ell)}}{\partial z} + \\ - \frac{\partial}{\partial x} \left(\varepsilon_{s,x}^{(\ell)} \frac{\partial c^{(\ell)}}{\partial x} \right) - \frac{\partial}{\partial y} \left(\varepsilon_{s,y}^{(\ell)} \frac{\partial c^{(\ell)}}{\partial y} \right) - \frac{\partial}{\partial z} \left(\varepsilon_{s,z}^{(\ell)} \frac{\partial c^{(\ell)}}{\partial z} \right) = 0, \quad (17.1)$$

where:

$c^{(\ell)}$	mass concentration of sediment fraction (ℓ) [kg/m ³]
u, v and w	flow velocity components [m/s]
$\varepsilon_{s,x}^{(\ell)}, \varepsilon_{s,y}^{(\ell)}$ and $\varepsilon_{s,z}^{(\ell)}$	eddy diffusivities of sediment fraction (ℓ) [m ² /s]
$w_s^{(\ell)}$	(hindered) sediment settling velocity of sediment fraction (ℓ) [m/s]

The local flow velocities and eddy diffusivities are based on the results of the hydrodynamic computations. Computationally, the three-dimensional transport of sediment is computed in exactly the same way as the transport of any other conservative constituent, such as salinity, heat, and constituents. There are, however, a number of important differences between sediment and other constituents, for example, the exchange of sediment between the bed and the flow, and the settling velocity of sediment under the action of gravity. These additional processes for sediment are obviously of critical importance. Other processes such as the effect that sediment has on the local mixture density, and hence on turbulence damping, can also be taken into account. In addition, if a net flux of sediment from the bed to the flow, or vice versa, occurs then the resulting change in the bathymetry should influence subsequent hydrodynamic calculations. The formulation of several of these processes (such as, settling velocity, sediment deposition and pickup) are sediment-type specific, this especially applies for sand and mud. Furthermore, the interaction of sediment fractions is important for many

processes, for instance the simultaneous presence of multiple suspended sediment fractions has implications for the calculation of the local hindered settling velocity of any one sediment fraction as well as for the resulting mixture density.

The following sections describe, at a conceptual level, the differences between the suspended transport of sediments and the transport of other conservative constituents. At the same time we discuss some of the differences in general terms and refer for the details of the mathematical formulations to Sections [section 17.2](#) and [section 17.3](#).



Remarks:

- ◊ The presence of multiple sediment fractions considerably complicates the calculation of the density of the bed and the availability of a particular class of sediment at the bed. See the sections on sediment interaction ([section 17.4.3](#)) and bed composition models ([section 17.6.4](#)).
- ◊ Small negative sediment concentrations ($-1 \cdot 10^{-3} \text{ kg/m}^3$) can be found in a computation. These negative concentrations can be suppressed by applying a horizontal Forester filter. However, this can result in a substantially larger computing time. It is suggested to accept small negative concentrations and to apply a Forester filter only when the negative concentrations become unacceptably large.
- ◊ A *vertical* Forester filter applied in a sediment transport computation will not affect the sediments. Since this filter smoothes the vertical profile and thus can have a strong influence on the vertical mixing processes, this vertical filter has been de-activated for sediments.

17.1.3 Effect of sediment on fluid density

The effect of sediment on fluid density is not implemented in D-Flow FM yet.

17.1.4 Sediment settling velocity

The settling velocity $w_s^{(\ell)}$ for sand and mud are strongly different in formulation; see Sections [17.2.1](#) and [17.3.1](#) for details. In high concentration mixtures, the settling velocity of a single particle is reduced due to the presence of other particles. In order to account for this hindered settling effect we follow [Richardson and Zaki \(1954\)](#) and determine the settling velocity in a fluid-sediment mixture as a function of the sediment concentration and the non-hindered settling fall velocity:

$$w_s^{(\ell)} = \left(1 - \frac{c_s^{tot}}{C_{soil}}\right)^5 w_{s,0}^{(\ell)}. \quad (17.2)$$

where C_{soil} is the reference density (input parameter), $w_{s,0}$ is the ‘basic’ sediment fraction specific settling velocity. The total mass concentration c_m^{tot} is the sum of the mass concentrations of the sediment fractions:

$$c_m^{tot} = \sum_{\ell=1}^{lsed} c_s^{(\ell)}. \quad (17.3)$$

As the fall velocity is now a function of the sediment fractions concentration, this implies that each sediment fraction has a fall velocity which is a function of location and time.



Remark:

- ◊ The process of sediment settling is computed with a first-order upwind numerical scheme. While use of the upwind settling formulation does slightly under-predict the mass of sed-

iment settling, the magnitude of this error has been shown to be rather small ([Lesser et al., 2000](#)).

17.1.5 Dispersive transport

The eddy diffusivities $\varepsilon_{s,x}^{(\ell)}$, $\varepsilon_{s,y}^{(\ell)}$ and $\varepsilon_{s,z}^{(\ell)}$ depend on the flow characteristics (turbulence level, taking into account the effect of high sediment concentrations on damping turbulent exchange processes) and the influence of waves (due to wave induced currents and enhanced bottom shear stresses). D-Flow FM supports four so-called “turbulence closure models”:

- ◊ Constant coefficient.
- ◊ Algebraic eddy viscosity closure model.
- ◊ $k-L$ turbulence closure model.
- ◊ $k-\varepsilon$ turbulence closure model.

The first is a simple constant value which is specified by you. A constant eddy viscosity will lead to parabolic vertical velocity profiles (laminar flow). The other three turbulence closure models are based on the eddy viscosity concept of [Kolmogorov \(1942\)](#) and [Prandtl \(1945\)](#) and offer zero, first, and second order closures for the turbulent kinetic energy (k) and for the mixing length (L). All three of the more advanced turbulence closure models take into account the effect that a vertical density gradient has on damping the amount of vertical turbulent mixing. See [chapter 9](#) for a full description of the available turbulence closure models.

The output of a turbulence closure model is the eddy viscosity at each layer interface; from this the vertical sediment mixing coefficient is calculated:

$$\varepsilon_s^{(\ell)} = \beta \varepsilon_f, \quad (17.4)$$

where:

$\varepsilon_s^{(\ell)}$	vertical sediment mixing coefficient for the sediment fraction (ℓ)
β	non-cohesive sediment: Van Rijn's 'beta' factor or effective 'beta' factor.
	cohesive sediment fractions and fine sand (< 150 μm): 1.0.
ε_f	vertical fluid mixing coefficient calculated by the selected turbulence model.

Remarks:

- ◊ For cohesive sediment fractions the extra turbulent mixing due to waves is not yet included in the eddy diffusivity. This is a limitation of the present implementation. See also [section 17.2.2](#).
- ◊ For non-cohesive sediment the effect of waves is accounted for by using a modified or effective 'beta' factor of Van Rijn ($k-\varepsilon$ model) or by using a separate formula to compute ε_f (algebraic or $k-L$) model. See also [section 17.3.2](#).



17.1.6 Three-dimensional wave effects

The three-dimensional sediment transport is not implemented in D-Flow FM yet.

17.1.7 Initial and boundary conditions

To solve [Equation 17.1](#) you need to prescribe initial and boundary conditions for each suspended sediment fraction.

17.1.7.1 Initial condition

The initial conditions for the sediment fractions are handled in exactly the same manner as those for any other conservative constituent, i.e. you can specify:

- ◊ One global initial concentration for each sediment fraction.
- ◊ Space-varying initial concentrations read from a restart file generated by a previous run.
- ◊ Space-varying initial concentrations read from a user-defined input file.

In these options cohesive and non-cohesive sediment fractions are treated in the same way.

In many practical applications the non-cohesive sediment concentrations adapt very rapidly to equilibrium conditions, so in the case of a cold start where the hydrodynamic model also takes some time to stabilise, a uniform zero concentration for the non-cohesive sediment fractions is usually adequate.

17.1.7.2 Boundary conditions

For each of the model boundaries you must prescribe the boundary condition for each sediment fraction. We discuss in short the general type of conditions and refer for the details to the sections to follow.

Water surface boundary

The vertical diffusive flux through the free surface is set to zero for all conservative constituents (except heat, which can cross this boundary). This is left unchanged for suspended sediment.

$$-w_s^{(\ell)} c^{(\ell)} - \varepsilon_{s,z}^{(\ell)} \frac{\partial c^{(\ell)}}{\partial z} = 0, \quad \text{at } z = \zeta \quad (17.5)$$

where $z = \zeta$ is the location of the free surface.

Bed boundary condition

The exchange of material in suspension and the bed is modelled by calculating the sediment fluxes from the bottom computational layer to the bed, and vice versa. These fluxes are then applied to the bottom computational layer by means of a sediment source and/or sink term in each computational cell. The calculated fluxes are also applied to the bed in order to update the bed level. The boundary condition at the bed is given by:

$$-w_s^{(\ell)} c^{(\ell)} - \varepsilon_{s,z}^{(\ell)} \frac{\partial c^{(\ell)}}{\partial z} = D^{(\ell)} - E^{(\ell)}, \quad \text{at } z = z_b \quad (17.6)$$

where:

- | | |
|--------------|--|
| $D^{(\ell)}$ | sediment deposition rate of sediment fraction (ℓ) . |
| $E^{(\ell)}$ | sediment erosion rate of sediment fraction (ℓ) . |

The formulations of $D^{(\ell)}$ and $E^{(\ell)}$ strongly differ for cohesive and non-cohesive sediment; for the details you are referred to Sections 17.2.3 and 17.3.4 respectively.

Open inflow boundaries

D-Flow FM requires you to specify boundary conditions for all conservative constituents at all open inflow boundaries. When modelling in three dimensions you may choose to specify boundary concentrations that have a uniform, linear, or step distribution over the vertical. You may also choose to specify a "Thatcher-Harleman" return time to simulate the re-entry of material that flowed out of the model after the flow reverses direction.

All of these options are also available for sediment constituents, although they are probably more appropriate for fine, cohesive sediment than for sand-sized particles. To assist with modelling coarser material an additional option has been included. This option allows you to specify that, at all open inflow boundaries, the flow should enter carrying all "sand" sediment fractions at their "equilibrium" concentration profiles. This feature has been implemented as a Neumann boundary condition, that is, zero concentration gradient at the boundary. By setting the sediment concentrations at the boundary equal to those just inside model domain, a near-perfectly adapted flow will enter the domain and very little accretion or erosion should be experienced near the model boundaries. This will generally be the desired situation if the model boundaries are well chosen. This feature can be activated for sand and mud fraction separately by setting NeuBcSand (previously, EqmBc) and/or NeuBcMud to true in the morphology input file.

Open outflow boundaries

No boundary condition is prescribed at outflow boundaries; effectively this means that the dispersive transport of sediment at the outflow boundary is neglected compared to the advective transport.

17.2 Cohesive sediment

The cohesive sediment is still not implemented in D-Flow FM. It will be add in the future.

17.2.1 Cohesive sediment settling velocity

In salt water cohesive sediment tends to flocculate to form sediment "flocs", with the degree of flocculation depending on the salinity of the water. These flocs are much larger than the individual sediment particles and settle at a faster rate. In order to model this salinity dependency you must supply two settling velocities and a maximum salinity. The first velocity, WS0, is taken to be the settling velocity of the sediment fraction in fresh water (salinity = 0). The second velocity, WSM, is the settling velocity of the fraction in water having a salinity equal to SALMAX. The settling velocity of the sediment flocs is calculated as follows:

$$w_{s,0}^{(\ell)} = \begin{cases} \frac{w_{s,\max}^{(\ell)}}{2} \left(1 - \cos\left(\frac{\pi S}{S_{\max}}\right)\right) + \frac{w_{s,f}^{(\ell)}}{2} \left(1 + \cos\left(\frac{\pi S}{S_{\max}}\right)\right), & \text{when } S \leq S_{\max} \\ w_{s,\max}^{(\ell)}, & \text{when } S > S_{\max} \end{cases} \quad (17.7)$$

where:

$w_{s,0}^{(\ell)}$	the (non-hindered) settling velocity of sediment fraction (ℓ)
$w_{s,\max}^{(\ell)}$	WSM, settling velocity of sediment fraction (ℓ) at salinity concentration SALMAX
$w_{s,f}^{(\ell)}$	WS0, fresh water settling velocity of sediment fraction (ℓ)
S	salinity
S_{\max}	SALMAX, maximal salinity at which WSM is specified

Remarks:



- ◊ Modelling turbulence induced flocculation or the break-up of sediment flocs is not yet implemented.
- ◊ The influence of flocculation is disregarded by setting WSM = WS0.

17.2.2 Cohesive sediment dispersion

The vertical mixing coefficient for sediment is equal to the vertical fluid mixing coefficient calculated by the selected turbulence closure model, i.e.:

$$\varepsilon_s^{(\ell)} = \varepsilon_f, \quad (17.8)$$

where:

$\varepsilon_s^{(\ell)}$	vertical sediment mixing coefficient for sediment fraction (ℓ)
ε_f	vertical fluid mixing coefficient calculated by the selected turbulence closure model

17.2.3 Cohesive sediment erosion and deposition

For cohesive sediment fractions the fluxes between the water phase and the bed are calculated with the well-known Partheniades-Krone formulations ([Partheniades, 1965](#)):

$$E^{(\ell)} = M^{(\ell)} S \left(\tau_{cw}, \tau_{cr,e}^{(\ell)} \right), \quad (17.9)$$

$$D^{(\ell)} = w_s^{(\ell)} c_b^{(\ell)} S \left(\tau_{cw}, \tau_{cr,d}^{(\ell)} \right), \quad (17.10)$$

$$c_b^{(\ell)} = c^{(\ell)} \left(z = \frac{\Delta z_b}{2}, t \right), \quad (17.11)$$

where:

$E^{(\ell)}$	erosion flux [$\text{kg m}^{-2} \text{s}^{-1}$]
$M^{(\ell)}$	user-defined erosion parameter EROUNI [$\text{kg m}^{-2} \text{s}^{-1}$]
$S \left(\tau_{cw}, \tau_{cr,e}^{(\ell)} \right)$	erosion step function:

$$S \left(\tau_{cw}, \tau_{cr,e}^{(\ell)} \right) = \begin{cases} \left(\frac{\tau_{cw}}{\tau_{cr,e}^{(\ell)}} - 1 \right), & \text{when } \tau_{cw} > \tau_{cr,e}^{(\ell)}, \\ 0, & \text{when } \tau_{cw} \leq \tau_{cr,e}^{(\ell)}. \end{cases} \quad (17.12)$$

$D^{(\ell)}$	deposition flux [$\text{kg m}^{-2} \text{s}^{-1}$]
$w_s^{(\ell)}$	fall velocity (hindered) [m/s]
$c_b^{(\ell)}$	average sediment concentration in the near bottom computational layer
$S \left(\tau_{cw}, \tau_{cr,d}^{(\ell)} \right)$	deposition step function:

$$S \left(\tau_{cw}, \tau_{cr,d}^{(\ell)} \right) = \begin{cases} \left(1 - \frac{\tau_{cw}}{\tau_{cr,d}^{(\ell)}} \right), & \text{when } \tau_{cw} < \tau_{cr,d}^{(\ell)}, \\ 0, & \text{when } \tau_{cw} \geq \tau_{cr,d}^{(\ell)}. \end{cases} \quad (17.13)$$

τ_{cw}	maximum bed shear stress due to current and waves as calculated by the wave-current interaction model selected by the user.
$\tau_{cr,e}^{(\ell)}$	user-defined critical erosion shear stress TCEUNI [N/m^2]

$\tau_{cr,d}^{(\ell)}$ user-defined critical deposition shear stress TCDUNI [N/m²]

! **Remark:**

- ◊ Superscript (ℓ) implies that this quantity applies to sediment fraction (ℓ) .

The calculated erosion or deposition flux is applied to the near bottom computational cell by setting the appropriate sink and source terms for that cell. Advection, particle settling, and diffusion through the bottom of the near bottom computational cell are all set to zero to prevent double counting these fluxes.

17.2.4 Interaction of sediment fractions

The following notes hold only in case of multiple sediment fractions. The formulations given in the previous section have been formulated for uniform cohesive sediment beds. However, often the bed will be made up of a range of sediment types and sizes. In such cases the erosion rate will be affected. If the bed stratigraphy is modelled in detail, it may be assumed that the erosion rate is proportional to the availability of the sediment fraction considered in the top-most layer of the bed stratigraphy. On the other hand if the bed stratigraphy is not explicitly included in the model and only the overall characteristics of the local bed composition is known, one must assume either that the bed composition is almost uniform (in which case the erosion rate can again be assumed to be proportional to the bed composition) or that the cohesive sediment fraction considered forms a layer that covers the other sediment fractions (in this case the erosion rate of the cohesive sediment will not be reduced). The former approach is nowadays the default approach for the online-morphology module, but the latter behaviour may be activated by setting the OldMudFrac keyword tot true in the morphology input file.

Remarks:

- ◊ Assuming an erosion rate proportional to the availability of the sediment fraction considered may result in a significant underestimation of the erosion rate if the bed is modelled as a single uniformly mixed layer (default setting) and the mud contents is low.
- ◊ Assuming that the erosion rate is independent of the availability of the sediment fraction considered will lead to an overestimation of the erosion rate. For instance, if the model includes two equal cohesive sediment fractions their total transport rate will be double that of the rate observed in an identical simulation carried out using the total amount of the two sediment fractions in the former simulation.



17.2.5 Influence of waves on cohesive sediment transport

The influence of wave on cohesive sediment is not implemented in D-Flow FM yet.

17.2.6 Inclusion of a fixed layer

If the thickness of the sediment layer becomes small then the erosion flux is reduced by a factor f_{FIXFAC} as defined in [section 17.4.4](#). This reduction factor is related to the formulations implemented for non cohesive sediment transport (see Sections [17.3.5](#) and [17.4.4](#) for suspended and bedload transport respectively).

17.2.7 Inflow boundary conditions cohesive sediment

Although it is general good advice to locate the open boundaries sufficiently far away from the area of interest, this is not always possible in long-term simulations. In such cases it is desirable to impose some kind of equilibrium boundary conditions. The mud concentrations are in general more loosely coupled to local morphology than the concentrations of coarser non-cohesive sediment fractions; a unique "equilibrium" concentration (profile) does often not exist due to differences in critical shear stresses for erosion and sedimentation. So, D-Flow FM allows for a different approach. For cohesive material you can specify that, at all open inflow boundaries, the flow should enter carrying the same mud concentration as computed in the interior of the model. This feature is enabled by setting `NeuBcMud` in the morphology input file to true (Neumann boundary condition: concentration gradient perpendicular to open boundary equal to zero). Although this option may sometimes be very useful, one must be careful when applying it: the sediment concentration of the incoming flow may take on any value that does not lead to significant deposition in the first grid cell.

By setting `NeuBcMud = false`, the concentrations to be applied at the inflow boundaries are read from the `<*.bcc>` file, which has to be created with the FLOW User Interface. If the parameter is set to true, the values specified in the `<*.bcc>` file are overruled.

17.3 Non-cohesive sediment

For the transport of non-cohesive sediment, [Van Rijn et al. \(2000\)](#) approach is followed by default. You can also specify a number of other transport formulations (see [section 17.5](#))

17.3.1 Non-cohesive sediment settling velocity

The settling velocity of a non-cohesive ("sand") sediment fraction is computed following the method of [Van Rijn \(1993\)](#). The formulation used depends on the diameter of the sediment in suspension:

$$w_{s,0}^{(\ell)} = \begin{cases} \frac{(s^{(\ell)} - 1)gD_s^{(\ell)2}}{18\nu}, & 65 \mu\text{m} < D_s \leq 100 \mu\text{m} \\ \frac{10\nu}{D_s} \left(\sqrt{1 + \frac{0.01(s^{(\ell)} - 1)gD_s^{(\ell)3}}{\nu^2}} - 1 \right), & 100 \mu\text{m} < D_s \leq 1000 \mu\text{m} \\ 1.1\sqrt{(s^{(\ell)} - 1)gD_s^{(\ell)}}, & 1000 \mu\text{m} < D_s \end{cases} \quad (17.14)$$

where:

$s^{(\ell)}$	relative density $\rho_s^{(\ell)} / \rho_w$ of sediment fraction (ℓ)
$D_s^{(\ell)}$	representative diameter of sediment fraction (ℓ)
ν	kinematic viscosity coefficient of water [m^2/s]

$D_s^{(\ell)}$ is the representative diameter of the suspended sediment given by the user-defined sediment diameter SEDDIA (D_{50} of bed material) multiplied by the user-defined factor `FACDSS` (see also remarks). This value of $D_s^{(\ell)}$ will be overruled if `IOPSUS=1` and the transport formula of [Van Rijn \(1993\)](#) has been selected, see [section 17.5.1](#) for details.



Remark:

- ◊ In the case of non-uniform bed material [Van Rijn \(1993\)](#) concluded that, on the basis of measurements, $D_s^{(\ell)}$ should be in the range of 60 to 100 % of D_{50} of the bed material.

If the bed material is very widely graded (well sorted) consideration should be given to using several sediment fractions to model its behaviour more accurately.

17.3.2 Non-cohesive sediment dispersion

The output of a turbulence closure model is the eddy viscosity at each layer interface; from this the vertical sediment mixing coefficient is calculated using the following expressions:

17.3.2.1 Using the algebraic or $k - L$ turbulence model

The $K - L$ model is not implemented in D-Flow FM yet.

17.3.2.2 Using the $k - \varepsilon$ turbulence model

In the case of the $k - \varepsilon$ turbulence closure model the vertical sediment mixing coefficient can be calculated directly from the vertical fluid mixing coefficient calculated by the turbulence closure model, using the following expression:

$$\varepsilon_s^{(\ell)} = \beta_{\text{eff}}^{(\ell)} \varepsilon_f, \quad (17.15)$$

where:

$\varepsilon_s^{(\ell)}$	vertical sediment mixing coefficient of sediment fraction (ℓ)
$\beta_{\text{eff}}^{(\ell)}$	the effective Van Rijn's 'beta' factor of sediment fraction (ℓ) As the beta factor should only be applied to the current-related mixing this is estimated as: $\beta_{\text{eff}}^{(\ell)} = 1 + (\beta^{(\ell)} - 1) \frac{\tau_c}{\tau_w + \tau_c}$, for non-cohesive sediment fractions
$\beta^{(\ell)}$	Van Rijn's 'beta' factor of the sediment fraction (ℓ), Equation 17.16
τ_c	bed shear stress due to currents
τ_w	bed shear stress due to waves
ε_f	vertical fluid mixing coefficient calculated by the $k - \varepsilon$ turbulence closure model

Van Rijn's 'beta' factor is calculated from ([Van Rijn, 1984b](#)):

$$\beta^{(\ell)} = 1 + 2 \left(\frac{w_s^{(\ell)}}{u_{*,c}} \right)^2. \quad (17.16)$$

Where $w_s^{(\ell)}$ is the settling velocity of the non-cohesive sediment fraction, and $u_{*,c}$ is the local bed shear stress due to currents.

This implies that the value of $\beta^{(\ell)}$ is space (and time) varying, however it is constant over the depth of the flow. In addition, due to the limited knowledge of the physical processes involved, we follow [Van Rijn \(1993\)](#) and limit $\beta^{(\ell)}$ to the range $1 < \beta^{(\ell)} < 1.5$.

Remarks:

- ◊ In a wave and current situation [Van Rijn \(1993\)](#) applies the β -factor to only the current-related turbulent mixing, whereas we apply it to the total turbulent mixing calculated by the selected turbulence closure model. However, little is known about the dependence of the β -factor on flow conditions; this discrepancy is expected to be of little importance in practical situations.
- ◊ The $k - \varepsilon$ turbulence closure model has been extended by [Walstra et al. \(2000\)](#) to include the three-dimensional effects of waves. However the effect of wave asymmetry on the bedload transport is not yet included.



17.3.3 Reference concentration

For non-cohesive sediment (e.g. sand), we follow the method of [Van Rijn \(1993\)](#) for the combined effect of waves and currents. The reference height is given by:

$$a = \min \left[\max \left\{ AKSFAC \cdot k_s, \frac{\Delta_r}{2}, 0.01h \right\}, 0.20h \right], \quad (17.17)$$

where:

a	Van Rijn's reference height
AksFac	user-defined proportionality factor (morphology input file)
k_s	user-defined current-related effective roughness height (see options below)
Δ_r	wave-induced ripple height, set to a constant value of 0.025 m
h	water depth



Remark:

- ◊ Van Rijn's reference height a is limited to a maximum of 20% of the water depth. This precaution is only likely to come into effect in very shallow areas.

With the keyword IOPKCW you have two options to calculate k_s (and k_w):

- ◊ k_s derived from current-related effective roughness height as determined in the D-Flow FM module (spatially varying) and $k_w = R\text{WAVE} \cdot \Delta r$.
- ◊ k_s and k_w specified by you (constant in space).

Calculation of the reference concentration

The reference concentration c_a is calculated directly by the sediment transport formula or it is derived from the suspended sediment transport rate given by the sediment transport formula as $c_a = S_s/H_u$. The default transport formula ([Van Rijn, 1993](#)) includes a formula for the reference concentration (see [section 17.5.1](#)). The reference concentration is adjusted proportional to the relative availability of the sediment fraction in the top-layer of the bed (see [section 17.6.4](#) on bed composition models).



Remark:

- ◊ The reference concentration and therefore the suspended load can be calibrated using the keyword S_{sus} in the morphology input file.

17.3.4 Non-cohesive sediment erosion and deposition

The transfer of sediment between the bed and the flow is modelled using sink and source terms acting on the near-bottom layer that is entirely above Van Rijn's reference height. This layer is identified as the reference layer and for brevity is referred to as the kmx -layer; see [Figure 17.1](#).

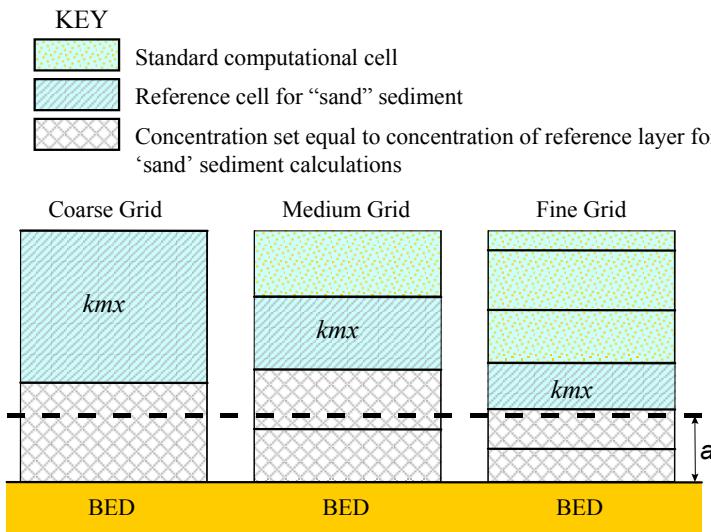


Figure 17.1: Selection of the kmx layer; where a is Van Rijn’s reference height

The sediment concentrations in the layer(s) that lie below the kmx layer are assumed to rapidly adjust to the same concentration as the reference layer.

Each half time-step the source and sink terms model the quantity of sediment entering the flow due to upward diffusion from the reference level and the quantity of sediment dropping out of the flow due to sediment settling. A sink term is solved implicitly in the advection-diffusion equation, whereas a source term is solved explicitly. The required sink and source terms for the kmx layer are calculated as follows.

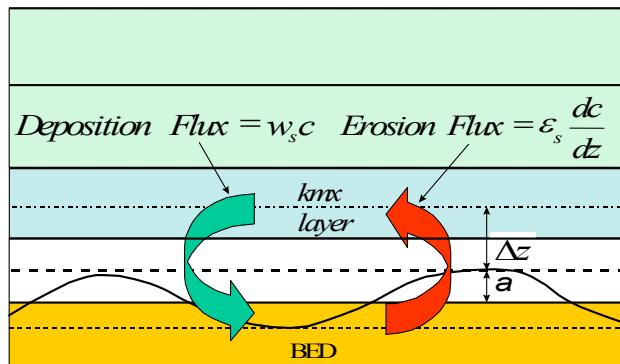


Figure 17.2: Schematic arrangement of flux bottom boundary condition

In order to determine the required sink and source terms for the kmx layer, the concentration and concentration gradient at the bottom of the kmx layer need to be approximated. We assume a standard Rouse profile between the reference level a and the centre of the kmx layer (see Figure 17.3).

$$c^{(\ell)} = c_a^{(\ell)} \left[\frac{a(h-z)}{z(h-a)} \right]^{A^{(\ell)}}, \quad (17.18)$$

where:

$c^{(\ell)}$ concentration of sediment fraction (ℓ)

$c_a^{(\ell)}$	reference concentration of sediment fraction (ℓ)
a	Van Rijn's reference height
h	water depth
z	elevation above the bed
$A^{(\ell)}$	Rouse number

As the reference concentration and the concentration in the centre of the k_{mx} layer $c_{k_{mx}}$ are known, the exponent $A(\ell)$ can be determined.

$$c_{k_{mx}}^{(\ell)} = c_a^{(\ell)} \left(\frac{a(h - z_{k_{mx}})}{z_{k_{mx}}(h - a)} \right)^{A(\ell)} \Rightarrow A^{(\ell)} = \frac{\ln \left(\frac{c_{k_{mx}}}{c_a} \right)}{\ln \left(\frac{a(h - z_{k_{mx}})}{z_{k_{mx}}(h - a)} \right)} \quad (17.19)$$

The concentration at the bottom of the k_{mx} layer is:

$$c_{k_{mx}(bot)}^{(\ell)} = c_a^{(\ell)} \left(\frac{a(h - z_{k_{mx}(bot)})}{z_{k_{mx}(bot)}(h - a)} \right)^{A(\ell)} \quad (17.20)$$

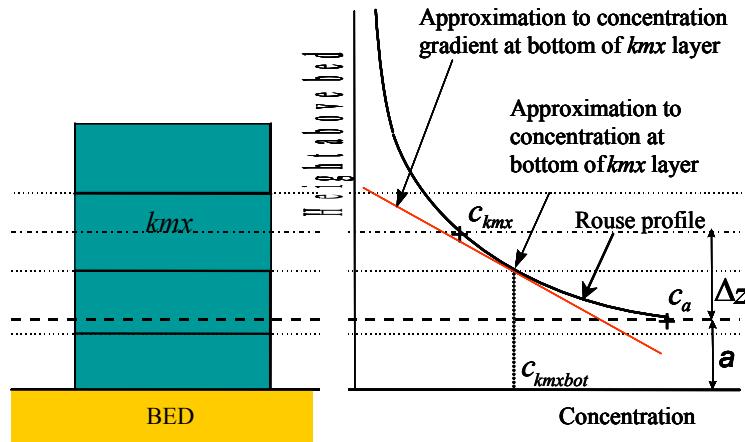


Figure 17.3: Approximation of concentration and concentration gradient at bottom of k_{mx} layer

We express this concentration as a function of the known concentration $c_{k_{mx}}$ by introducing a correction factor α_1 :

$$c_{k_{mx}(bot)}^{(\ell)} = \alpha_1^{(\ell)} c_{k_{mx}}^{(\ell)} \quad (17.21)$$

The concentration gradient of the Rouse profile is given by:

$$\frac{\partial c^{(\ell)}}{\partial z} = A^{(\ell)} c_a^{(\ell)} \left(\frac{a(h - z)}{z(h - a)} \right)^{A^{(\ell)} - 1} \cdot \left(\frac{-ah}{z^2(h - a)} \right) \quad (17.22)$$

The concentration gradient at the bottom of the k_{mx} layer is:

$$c'_{k_{mx}(bot)}^{(\ell)} = A^{(\ell)} c_a^{(\ell)} \left(\frac{a(h - z_{k_{mx}(bot)})}{z_{k_{mx}(bot)}(h - a)} \right)^{A^{(\ell)} - 1} \cdot \left(\frac{-ah}{z_{k_{mx}(bot)}^2(h - a)} \right) \quad (17.23)$$

We express this gradient as a function of the known concentrations c_a and c_{kmx} by introducing another correction factor α_2 :

$$c'_{kmx(bot)}^{(\ell)} = \alpha_2^{(\ell)} \left(\frac{c_{kmx}^{(\ell)} - c_a^{(\ell)}}{\Delta z} \right) \quad (17.24)$$

Erosive flux due to upward diffusion

The upward diffusion of sediment through the bottom of the k_{mx} layer is given by the expression:

$$E^{(\ell)} = \varepsilon_s^{(\ell)} \frac{\partial c^{(\ell)}}{\partial z}, \quad (17.25)$$

where $\varepsilon_s^{(\ell)}$ and $\frac{\partial c^{(\ell)}}{\partial z}$ are evaluated at the bottom of the k_{mx} layer.

We approximate this expression by:

$$E^{(\ell)} \approx \alpha_2^{(\ell)} \varepsilon_s^{(\ell)} \left(\frac{c_a^{(\ell)} - c_{kmx}^{(\ell)}}{\Delta z} \right), \quad (17.26)$$

where:

- $\alpha_2^{(\ell)}$ correction factor for sediment concentration
- $\varepsilon_s^{(\ell)}$ sediment diffusion coefficient evaluated at the bottom of the k_{mx} cell of sediment fraction(ℓ)
- $c_a^{(\ell)}$ reference concentration of sediment fraction(ℓ)
- $c_{kmx}^{(\ell)}$ average concentration of the k_{mx} cell of sediment fraction(ℓ)
- Δz difference in elevation between the centre of the k_{mx} cell and Van Rijn's reference height: $\Delta z = z_{k_{mx}} - a$

The erosion flux is split in a source and sink term:

$$E^{(\ell)} \approx \frac{\alpha_2^{(\ell)} \varepsilon_s^{(\ell)} c_a^{(\ell)}}{\Delta z} - \frac{\alpha_2^{(\ell)} \varepsilon_s^{(\ell)} c_{kmx}^{(\ell)}}{\Delta z}. \quad (17.27)$$

The first of these terms can be evaluated explicitly and is implemented as a sediment source term. The second can only be evaluated implicitly and is implemented as a (positive) sink term. Thus:

$$\text{Source}_{\text{erosion}}^{(\ell)} = \frac{\alpha_2^{(\ell)} \varepsilon_s^{(\ell)} c_a^{(\ell)}}{\Delta z} \quad (17.28)$$

$$\text{Sink}_{\text{erosion}}^{(\ell)} = \frac{\alpha_2^{(\ell)} \varepsilon_s^{(\ell)} c_{kmx}^{(\ell)}}{\Delta z} \quad (17.29)$$

Deposition flux due to sediment settling

The settling of sediment through the bottom of the k_{mx} cell is given by the expression:

$$D^{(\ell)} = w_s^{(\ell)} c_{kmx(bot)}^{(\ell)}, \quad (17.30)$$

where $w_s^{(\ell)}$ and $c_{kmx(bot)}^{(\ell)}$ are evaluated at the bottom of the k_{mx} layer.

We set:

$$c_{k_{mx}(bot)}^{(\ell)} = \alpha_1^{(\ell)} c_{k_{mx}}^{(\ell)}. \quad (17.31)$$

The deposition flux is approximated by:

$$D^{(\ell)} \approx \alpha_1^{(\ell)} c_{k_{mx}}^{(\ell)} w_s^{(\ell)}. \quad (17.32)$$

This results in a simple deposition sink term:

$$Sink_{deposition}^{(\ell)} = \alpha_1^{(\ell)} c_{k_{mx}}^{(\ell)} w_s^{(\ell)}. \quad (17.33)$$

The total source and sink terms is given by:

$$Source^{(\ell)} = \alpha_2^{(\ell)} c_a^{(\ell)} \left(\frac{\varepsilon_s^{(\ell)}}{\Delta z} \right), \quad (17.34)$$

$$Sink^{(\ell)} = \left[\alpha_2^{(\ell)} \left(\frac{\varepsilon_s^{(\ell)}}{\Delta z} \right) + \alpha_1^{(\ell)} w_s^{(\ell)} \right] c_{k_{mx}}^{(\ell)}. \quad (17.35)$$

These source and sink terms are both guaranteed to be positive.

17.3.5 Inclusion of a fixed layer

The bedload transport is reduced if the thickness of the sediment layer becomes small (see section 17.4.4). The same effect has been implemented as a reduction for the entrainment and deposition terms as well as the equilibrium concentration by a factor f_{FIXFAC} if erosion is expected to occur.

17.3.6 Inflow boundary conditions non-cohesive sediment

Although it is general good advice to locate the open boundaries sufficiently far away from the area of interest, this is not always possible in long-term simulations. In such cases it is desirable to impose some kind of equilibrium boundary conditions. Although equilibrium boundary conditions may be better defined for non-cohesive sediments than for cohesive sediments, we have implemented the open boundary condition in a consistent manner. For non-cohesive suspended material you can specify that, at all open inflow boundaries, the flow should enter carrying the same concentration of sediment as computed in the interior of the model. This feature is enabled by setting NeuBcSand in the morphology input file to true (Neumann boundary condition: concentration gradient perpendicular to open boundary equal to zero). This means that the sediment load entering through the boundaries will be near-perfectly adapted to the local flow conditions and very little accretion or erosion should be experienced near the model boundaries. This will generally be the desired situation if the model boundaries are well chosen. This method gives the correct results even when the turbulent mixing profile is clearly non-parabolic.

By setting NeuBcSand = false, the concentrations to be applied at the inflow boundaries are read from the <*.bcc> file, which has to be created with the FLOW User Interface. If the parameter is set to true, the values specified in the <*.bcc> file are overruled. This parameter used to be called EqmBc.

17.4 Bedload sediment transport of non-cohesive sediment

Bedload (or, for the simpler transport formulae, total load) transport is calculated for all “sand” and “bedload” sediment fractions by broadly according to the following approach: first, the magnitude and direction of the bedload transport at the cell centres is computed using the transport formula selected (See [section 17.5](#)), subsequently the transport rates at the cell interfaces are determined, corrected for bed-slope effect and upwind bed composition and sediment availability.

17.4.1 Basic formulation

For simulations including waves the magnitude and direction of the bedload transport on a horizontal bed are calculated using the transport formula selected assuming sufficient sediment and ignoring bed composition except for e.g. hiding and exposure effects on the critical shear stresses. The default sediment transport formula is [Van Rijn \(1993\)](#) as documented in [section 17.5.1](#).

Some of the sediment transport formulae prescribe the bedload transport direction whereas others predict just the magnitude of the sediment transport. In the latter case the initial transport direction will be assumed to be equal to the direction of the characteristic (near-bed) flow direction. In the case of a depth-averaged simulation, the secondary flow/spiral flow intensity I_s optionally computed by the flow module may be taken into account; the bedload transport direction φ_τ is given by the following formula:

$$\tan(\varphi_\tau) = \frac{v - \alpha_I \frac{u}{U} I_s}{u - \alpha_I \frac{v}{U} I_s} \quad (17.36)$$

in which

$$\alpha_I = \frac{2}{\kappa^2} E_s \left(1 - \frac{1}{2} \frac{\sqrt{g}}{\kappa C} \right) \quad (17.37)$$

where:

g	gravitational acceleration
κ	Von Kármán constant
C	Chézy roughness
U	the depth-averaged velocity
E_s	coefficient to be specified by you as <code>Espir</code> keyword in the morphology input file

The default value of E_s is 0, which implies that the spiral flow effect on the bedload transport direction is not included. The spiral flow effect is of crucial importance in a depth-averaged simulation to get pointbar formation in river bends. This effect is only included for transport formulae that return the bedload transport rate but not its direction, i.e. Engelund & Hansen, Meyer-Peter & Muller, General formula, Van Rijn (1984), Ashida & Michiue and optionally the user-defined formula.

The [Van Rijn \(1993\)](#) formula distinguishes the following transport components that are all treated like bed or total load, i.e without relaxation effects of an advection diffusion equation:

- ◊ bedload due to currents, S_{bc}
- ◊ bedload due to waves, S_{bw}
- ◊ suspended load due to waves, S_{sw} .

These three transport components can be calibrated independently by using the respective keywords `Bed`, `BedW` and `SusW` in the morphology input file.

17.4.2 Suspended sediment correction vector

The Suspended sediment correction vector is not implemented yet in D-Flow FM.

17.4.3 Interaction of sediment fractions

The following notes hold only in case of multiple sediment fractions. Sediment fractions may interacted in several ways:

- ◊ reference concentrations, erosion rates and sediment transport rates will be reduced proportional to the availability of sediment fraction considered in the bed (less of the fraction available for transport)
- ◊ sediment fractions of different sizes influence each other by means of hiding and exposure: fine sediments hide among coarse sediments and are thereby partly shielded from the main flow while the coarser sediments are more exposed than they would be among other sediments of the same size. This effect is taken into account by increasing the effective critical shear stress for fine sediments while lowering it for coarse sediments. This adjustment is carried out using a multiplicative factor ξ . The following formulations have been implemented:

- No hiding and exposure correction ($\xi = 1$)
- Egiazaroff formulation

$$\xi = \left(\frac{^{10}\log 19}{^{10}\log 19 + ^{10}\log(D_i/D_m)} \right)^2. \quad (17.38)$$

- Ashida & Michiue formulation

$$\xi = \begin{cases} 0.8429 \frac{D_m}{D_i} & \text{if } D_i/D_m < 0.38889 \\ \left(\frac{^{10}\log 19}{^{10}\log 19 + ^{10}\log(D_i/D_m)} \right)^2 & \text{otherwise} \end{cases}. \quad (17.39)$$

- Parker, Klingeman & McLean or Soehngen, Kellermann & Loy formulation

$$\xi = \left(\frac{D_m}{D_i} \right)^\alpha. \quad (17.40)$$

where α is given by the ASKLHE keyword in the morphology input file.

- Wu, Wang & Jia formulation

$$\varphi^{(\ell)} = \sum_i \eta^{(i)} \frac{D^{(i)}}{D^{(\ell)} - D^{(i)}} \quad (17.41)$$

$$\xi^{(\ell)} = \left(\frac{1 - \varphi^{(\ell)}}{\varphi^{(\ell)}} \right)^m \quad (17.42)$$

where m is given by the MWWJHE keyword in the morphology input file.

The hiding and exposure effect has been implemented for the following transport formulae containing a critical shear stress: Meyer-Peter & Muller, general formula, Ashida-Michiue and optionally the user-defined formula.

17.4.4 Inclusion of a fixed layer

Inclusion of a fixed layer implies that the quantity of sediment at the bed is finite and may, if excessive erosion occurs, become exhausted and be unavailable to supply sediment to suspended and bedload transport modes. In case the bed is covered by bedforms, the troughs of the bedforms will start to expose the non-erodible layer before sediment runs out completely. This results in a gradual reduction of the transport capacity over a certain sediment thickness indicated by THRESH. This effect is taken into account in the bedload formulations by comparing the thickness of the sediment layer available at the bed with a user-defined threshold value. If the quantity of sediment available is less than the threshold then the magnitude of the calculated bedload transport vector is reduced as follows:

$$S_b'' = f_{\text{FIXFAC}} S_b'', \quad (17.43)$$

where:

S_b''	magnitude of the bedload transport vector (before correction for bed slope effects)
f_{FIXFAC}	upwind fixed layer proximity factor: $f_{\text{FIXFAC}} = \frac{\text{DPSED}}{\text{THRESH}}$, limited to the range $0 \leq f_{\text{FIXFAC}} \leq 1$.
DPSED	depth of sediment available at the bed
THRESH	user-defined erosion threshold

The equilibrium suspended load concentration in the sediment pickup term is reduced by the same fixed layer proximity factor (in this case of course the local value and not some upwind value is used since suspended sediment pickup has no associated horizontal direction).

In effect, because of the upwind approach used to transfer the bedload transport components to the velocity points, this method limits the sediment that can leave a computational cell, if the quantity of the sediment at the bed is limited. One implication of the use of this rather simple approach is that a finite (although always less than the user-defined threshold) thickness of sediment is required at the bed if a non-zero magnitude of the bedload transport vector is required.

Remarks:

- ◊ Areas may be initially specified as containing zero bottom sediment if non-erodible areas are required. It is likely that these areas will accrete a little sediment in order to allow an equilibrium bedload transport pattern to develop.
- ◊ This effect has also been included for cohesive and non cohesive suspended sediment as indicated in Sections 17.2 and 17.3.5.



17.4.5 Calculation of bedload transport at open boundaries

Bedload transport at open boundaries is not implemented yet.

17.4.6 Bedload transport at velocity points

As the control volume for bed level change calculations is centred on the water level points, see Figure 17.4, the bedload transport vector components are actually required at the velocity points, rather than at the water level points where S_x and S_y are calculated. By default, we use a simple "upwind" numerical scheme to set the bedload transport components at the velocity points as this ensures that the bed will remain stable. For each active velocity point the upwind direction is determined by summing the bedload transport components at the water level points on either side of the velocity point and taking the upwind *direction* relative to the resulting net transport direction. The bedload transport component at the velocity point is then set equal to the component computed at the water level point immediately "upwind" (see

Figure 17.4). In the example shown in [Figure 17.4](#) the bedload transport normal component on the velocity point, S_u^1 , is set equal to S_n^1 , where S_n^1 is the projection of bedload flux vector at point 1 in the normal direction of the corresponding velocity face. Based on the same method, the component S_u^2 is set equal to S_n^2 . It is possible to switch from upwind to central approach by setting the UpwindBedload keyword in the morphology input file to false; although the central approach is more accurate, it is less stable (less damping).

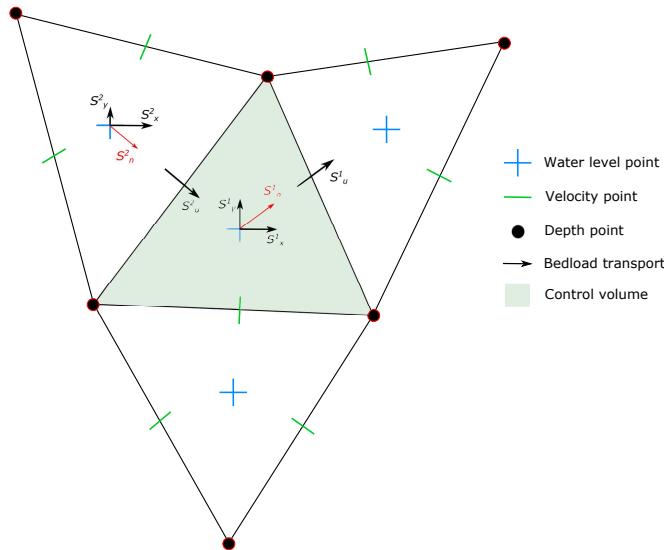


Figure 17.4: Setting of bedload transport components at velocity points

17.4.7 Adjustment of bedload transport for bed-slope effects

Bedload transport is affected by bed level gradients. Two bed slope directions are distinguished: the slope in the initial direction of the transport (referred to as the longitudinal bed slope) and the slope in the direction perpendicular to that (referred to as the transverse bed slope). The longitudinal bed slope results in a change in the sediment transport rate as given by:

$$\vec{S}'_b = \alpha_s \vec{S}''_b, \quad (17.44)$$

or, in vector component form:

$$S'_{b,x} = \alpha_s S''_{b,x}, \quad (17.45)$$

$$S'_{b,y} = \alpha_s S''_{b,y}, \quad (17.46)$$

whereas the primary effect of the transverse bed slope is a change in transport towards the downslope direction (this may be accomplished by either a pure rotation of the transport vector or by adding a transverse transport component). You may choose one of the following formulations for these effects.

- 1 no effect of bed slope on bedload transport
- 2 [Bagnold \(1966\)](#) for longitudinal slope and [Ikeda \(1982, 1988\)](#) as presented by [Van Rijn \(1993\)](#) for transverse slope. This is the default option for the bedload transport of all sediment transport formulae. In this case α_s is given by

$$\alpha_s = 1 + \alpha_{bs} \left(\frac{\tan(\phi)}{\cos(\tan^{-1}(\frac{\partial z}{\partial s})) (\tan(\phi) + \frac{\partial z}{\partial s})} - 1 \right), \quad (17.47)$$

where α_{bs} is a user-defined tuning parameter, ALFABS keyword in the morphology input file (default = 1.0). An additional bedload transport vector is subsequently calculated,

perpendicular to the main bedload transport vector. The magnitude of this vector is calculated using a formulation based on the work of Ikeda (1982, 1988) as presented by Van Rijn (1993). Van Rijn's equation (7.2.52) is modified to [Equation 17.48](#) by setting the reference co-ordinates s and n aligned with and perpendicular to the local characteristic flow direction respectively. This implies that there is no flow in the n direction: i.e. $u_{b,n} = 0$:

$$S_{b,n} = |S'_b| \alpha_{bn} \frac{u_{b,cr}}{|\vec{u}_b|} \frac{\partial z_b}{\partial n}, \quad (17.48)$$

where:

$S_{b,n}$	additional bedload transport vector. The direction of this vector is normal to the unadjusted bedload transport vector, in the down slope direction
$ S'_b $	magnitude of the unadjusted bedload transport vector (adjusted for longitudinal bed slope only): $ S'_b = \sqrt{(S'_{b,x})^2 + (S'_{b,y})^2}$.
α_{bn}	user-defined coefficient, ALFABN (default = 1.5)
$u_{b,cr}$	critical (threshold) near-bed fluid velocity
\vec{u}_b	near-bed fluid velocity vector
$\frac{\partial z_b}{\partial n}$	bed slope in the direction normal to the unadjusted bedload transport vector

To evaluate [Equation 17.48](#) we substitute:

$$\frac{u_{b,cr}}{|\vec{u}_b|} = \sqrt{\frac{\tau_{b,cr}}{|\vec{\tau}_b|}}, \quad (17.49)$$

where:

$\tau_{b,cr}$	critical bed shear stress
$\vec{\tau}_b$	bed shear stress due to current and waves: $\vec{\tau}_b = \mu_c \vec{\tau}_{b,cw} + \mu_w \vec{\tau}_{b,w}$.

resulting in:

$$S_{b,n} = |S'_b| f_{norm}, \quad (17.50)$$

where:

$$f_{norm} = \alpha_{bn} \sqrt{\frac{\tau_{b,cr}}{|\vec{\tau}_b|}} \frac{\partial z_b}{\partial n}. \quad (17.51)$$

The two components of this vector are then added to the two components of the bedload transport vector as follows:

$$\begin{aligned} S_{b,x} &= S'_{b,x} - S'_{b,y} f_{norm} \\ S_{b,y} &= S'_{b,y} + S'_{b,x} f_{norm} \end{aligned} \quad (17.52)$$

where $S_{b,x}$ and $S_{b,y}$ are the components of the required bedload transport vector, calculated at the water level points

- 3 Koch and Flokstra (1980) as extended by Talmon et al. (1995). In this case α_s is given by

$$\alpha_s = 1 - \alpha_{bs} \frac{\partial z}{\partial s}, \quad (17.53)$$

where α_{bs} is a user-defined tuning parameter, ALFABS keyword in the morphology input file (default = 1.0). The direction of the bedload is adjusted according to the following formulation:

$$\tan(\varphi_s) = \frac{\sin(\varphi_\tau) + \frac{1}{f(\theta)} \frac{\partial z_b}{\partial y}}{\cos(\varphi_\tau) + \frac{1}{f(\theta)} \frac{\partial z_b}{\partial x}}, \quad (17.54)$$

in which φ_τ is the original direction of the sediment transport and φ_s is the final direction and where $f(\theta)$ equals:

$$f(\theta) = A_{sh} \theta_i^{B_{sh}} \left(\frac{D_i}{H} \right)^{C_{sh}} \left(\frac{D_i}{D_m} \right)^{D_{sh}}, \quad (17.55)$$

where A_{sh} , B_{sh} , C_{sh} and D_{sh} are tuning coefficients specified by you in the morphology input file as keywords Ashld, Bshld, Cshld and Dshld.

- 4 [Parker and Andrews \(1985\)](#). The same formulae for α_s and φ_s hold as in the previous case except for $f(\theta)$ which now equals:

$$f(\theta) = \frac{c_L}{1 + \mu c_L} \sqrt{\frac{\theta}{\max(\frac{1}{10}\theta, \xi\theta_{cr})}}, \quad (17.56)$$

where Coulomb friction parameter c_L , lift-drag ratio μ and critical shields parameter θ_{cr} should be specified by you in the morphology input file as keywords CoulFri, FlFdRat and ThetaCr. Note that this formula includes the hiding and exposure factor ξ .

This completes the calculation of the bedload transport field. The transports at the velocity points are then stored for use in the computation of bed level changes, as described in the [section 17.6](#). In all cases the bed slope has been defined as follows.

Longitudinal bed slope

This bed slope is calculated as:

$$\frac{\partial z_b}{\partial s} = \frac{\partial z_{(u)}}{\partial x} \frac{S''_{b,x}}{|S''_b|} + \frac{\partial z_{(v)}}{\partial y} \frac{S''_{b,y}}{|S''_b|}, \quad (17.57)$$

$$\left(\frac{\partial z_b}{\partial s} \right)_{\max} = 0.9 \tan(\phi), \quad (17.58)$$

where:

$\frac{\partial z_b}{\partial s}$	bed slope in the direction of bedload transport
$\frac{\partial z_{(u)}}{\partial x}$	bed slope in the positive x -direction
$\frac{\partial z_{(v)}}{\partial y}$	bed slope in the positive y -direction
ϕ	internal angle of friction of bed material (assumed to be 30°)



Remarks:

- ◊ z_b is the depth down to the bed from a reference height (positive down), a downward bed slope returns a positive value).
- ◊ The bed slope is calculated at the velocity points as these are the locations at which the bedload transport vector components will finally be applied.

Transverse bed slope

This bed slope is calculated as:

$$\frac{\partial z_b}{\partial n} = -\frac{\partial z_{(u)}}{\partial x} \frac{S''_{b,y}}{|S''_b|} + \frac{\partial z_{(v)}}{\partial y} \frac{S''_{b,x}}{|S''_b|}. \quad (17.59)$$

Table 17.1: Additional transport relations

Formula	Bedload	Waves
17.5.1, Van Rijn (1993)	Bedload + suspended	Yes
17.5.2, Engelund-Hansen (1967)	Total transport	No
17.5.3, Meyer-Peter-Muller (1948)	Total transport	No
17.5.4, General formula	Total transport	No
17.5.5, Bijker (1971)	Bedload + suspended	Yes
17.5.6, Van Rijn (1984)	Bedload + suspended	No
17.5.7, Soulsby/Van Rijn	Bedload + suspended	Yes
17.5.8, Soulsby	Bedload + suspended	Yes
17.5.9, Ashida-Michiue (1974)	Total transport	No
17.5.10, Wilcock-Crowe (2003)	Bedload	No
17.5.11, Gaeuman et al. (2009) laboratory calibration	Bedload	No
17.5.12, Gaeuman et al. (2009) Trinity River calibration	Bedload	No

17.5 Transport formulations for non-cohesive sediment

This special feature offers a number of standard sediment transport formulations for non-cohesive sediment. [Table 17.1](#) gives a summary of the available additional formulae.

Additionally, you can implement your own sediment transport formula in a shared library (<dll> or <so>) and call it from D-Flow FM. Now, let us continue with a general description of the sediment transport formulae included by default.

17.5.1 Van Rijn (1993)

[Van Rijn \(1993\)](#) distinguishes between sediment transport below the reference height a which is treated as bedload transport and that above the reference height which is treated as suspended-load. Sediment is entrained in the water column by imposing a reference concentration at the reference height.

Reference concentration

The reference concentration is calculated in accordance with [Van Rijn et al. \(2000\)](#) as:

$$c_a^{(\ell)} = 0.015 \rho_s^{(\ell)} \frac{D_{50}^{(\ell)} \left(T_a^{(\ell)} \right)^{1.5}}{a \left(D_*^{(\ell)} \right)^{0.3}} \quad (17.60)$$

where:

$c_a^{(\ell)}$ mass concentration at reference height a

In order to evaluate this expression the following quantities must be calculated:

$D_*^{(\ell)}$ non-dimensional particle diameter:

$$D_*^{(\ell)} = D_{50}^{(\ell)} \left[\frac{(s^{(\ell)} - 1)g}{\nu^2} \right]^{1/3} \quad (17.61)$$

$T_a^{(\ell)}$ non-dimensional bed-shear stress:

$$T_a^{(\ell)} = \frac{(\mu_c^{(\ell)} \tau_{b,cw} + \mu_w^{(\ell)} \tau_{b,w}) - \tau_{cr}^{(\ell)}}{\tau_{cr}^{(\ell)}} \quad (17.62)$$

$\mu_c^{(\ell)}$ efficiency factor current:

$$\mu_c^{(\ell)} = \frac{f'_c^{(\ell)}}{f_c} \quad (17.63)$$

$f'_c^{(\ell)}$ gain related friction factor:

$$f'_c^{(\ell)} = 0.24 \left[{}^{10}\log \left(\frac{12h}{3D_{90}^{(\ell)}} \right) \right]^{-2} \quad (17.64)$$

$f_c^{(\ell)}$ total current-related friction factor:

$$f_c^{(\ell)} = 0.24 \left[{}^{10}\log \left(\frac{12h}{k_s} \right) \right]^{-2} \quad (17.65)$$

$\tau_{b,cw}$ bed shear stress due to current in the presence of waves. Note that the bed shear velocity u_* is calculated in such a way that Van Rijn's wave-current interaction factor α_{cw} is not required.

$$\tau_{b,cw} = \rho_w u_*^2 \quad (17.66)$$

$\mu_w^{(\ell)}$ efficiency factor waves:

$$\mu_w^{(\ell)} = \max \left(0.063, \frac{1}{8} \left(1.5 - \frac{H_s}{h} \right)^2 \right) \quad (17.67)$$

$\tau_{b,w}$ bed shear stress due to waves:

$$\tau_{b,w} = \frac{1}{4} \rho_w f_w \left(\hat{U}_\delta \right)^2 \quad (17.68)$$

f_w total wave-related friction factor:

$$f_w = \exp \left[-6 + 5.2 \left(\frac{\hat{A}_\delta}{k_{s,w}} \right)^{-0.19} \right] \quad (17.69)$$

To avoid the need for excessive user input, the wave related roughness $k_{s,w}$ is related to the estimated ripple height, using the relationship:

$$k_{s,w} = RWAVE \cdot \Delta_r, \text{ with } \Delta_r = 0.025 \text{ and } 0.01 \text{ m} \leq k_{s,w} \leq 0.1 \text{ m} \quad (17.70)$$

where:

RWAVE the user-defined wave roughness adjustment factor. Recommended to be in the range 1–3, default = 2.

$\tau_{cr}^{(\ell)}$ critical bed shear stress:

$$\tau_{cr}^{(\ell)} = (\rho_s^{(\ell)} - \rho_w) g D_{50}^{(\ell)} \theta_{cr}^{(\ell)} \quad (17.71)$$

$\theta_{cr}^{(\ell)}$ threshold parameter $\theta_{cr}^{(\ell)}$ is calculated according to the classical Shields curve as modelled by [Van Rijn \(1993\)](#) as a function of the non-dimensional grain size D_* . This avoids the need for iteration.



Note: for clarity, in this expression the symbol D_* has been used where $D_*^{(\ell)}$ would be more correct:

$$\theta_{cr}^{(\ell)} = \begin{cases} 0.24D_*^{-1}, & 1 < D_* \leq 4 \\ 0.14D_*^{-0.64}, & 4 < D_* \leq 10 \\ 0.04D_*^{-0.1}, & 10 < D_* \leq 20 \\ 0.013D_*^{0.29}, & 20 < D_* \leq 150 \\ 0.055, & 150 < D_* \end{cases} \quad (17.72)$$

a	Van Rijn's reference height
\hat{A}_δ	peak orbital excursion at the bed: $\hat{A}_\delta = \frac{T_p \hat{U}_\delta}{2\pi}$.
$D_{50}^{(\ell)}$	median sediment diameter
$D_{90}^{(\ell)}$	90 % sediment passing size: $D_{90}^{(\ell)} = 1.5D_{50}^{(\ell)}$
h	water depth
k_a	apparent bed roughness felt by the flow when waves are present. Calculated by D-Flow FM using the wave-current interaction formulation selected: $k_a \leq 10k_s$
k_s	user-defined current-related effective roughness height (space varying)
$k_{s,w}$	wave-related roughness, calculated from ripple height, see Equation 17.70
u_z	velocity magnitude taken from a near-bed computational layer. In a current-only situation the velocity in the bottom computational layer is used. Otherwise, if waves are active, the velocity is taken from the layer closest to the height of the top of the wave mixing layer δ .
\hat{U}_δ	peak orbital velocity at the bed: $\sqrt{2} \times RMS_{orbital\ velocity}$ at bed, taken from the wave module.
z_u	height above bed of the near-bed velocity (u_z) used in the calculation of bottom shear stress due to current
Δ_r	estimated ripple height, see Equation 17.70
δ_m	thickness of wave boundary mixing layer following Van Rijn (1993) : $3\delta_w$ (and $\delta_m \geq k_a$)
δ_w	wave boundary layer thickness: $\delta_w = 0.072\hat{A}_\delta \left(\frac{\hat{A}_\delta}{k_{s,w}} \right)^{-0.25}.$

We emphasise the following points regarding this implementation:

- ◊ The bottom shear stress due to currents is based on a near-bed velocity taken from the hydrodynamic calculations, rather than the depth-averaged velocity used by Van Rijn.
- ◊ All sediment calculations are based on hydrodynamic calculations from the previous half time-step. We find that this is necessary to prevent unstable oscillations developing.

The apparent roughness felt by the flow (k_a) is dependent on the hydrodynamic wave-current interaction model applied. At this time, Van Rijn's wave-current interaction model is not available in D-Flow FM. This means that it is not possible for a user to exactly reproduce results obtained using Van Rijn's full formulations for waves and currents.

Adjustment of the representative diameter of suspended sediment

The representative diameter of the suspended sediment $D_s^{(\ell)}$ generally given by the user-defined sediment diameter SEDDIA (D_{50} of bed material) multiplied by the user-defined factor FACDSS (see also remarks) can be overruled in case the [Van Rijn \(1993\)](#) transport formula is selected. This achieved by setting IOPSUS=1 the representative diameter of the suspended sediment will then be set to:

$$D_s^{(\ell)} = \begin{cases} 0.64 D_{50}^{(\ell)} & \text{for } T_A^{(\ell)} \leq 1 \\ D_{50}^{(\ell)} \left(1 + 0.015 \left(T_A^{(\ell)} - 25 \right) \right) & \text{for } 1 < T_A^{(\ell)} \leq 25 \\ D_{50}^{(\ell)} & \text{for } 25 < T_A^{(\ell)} \end{cases} \quad (17.73)$$

where $T_a^{(\ell)}$ is given by equation [17.62](#).

Bedload transport rate

For simulations including waves the magnitude and direction of the bedload transport on a horizontal bed are calculated using an approximation method developed by [Van Rijn et al. \(2003\)](#). The method computes the magnitude of the bedload transport as:

$$|S_b| = 0.006 \rho_s w_s D_{50}^{(\ell)} M^{0.5} M_e^{0.7} \quad (17.74)$$

where:

S_b	bedload transport [$\text{kg m}^{-1} \text{s}^{-1}$]
M	sediment mobility number due to waves and currents [-]
M_e	excess sediment mobility number [-]

$$M = \frac{v_{\text{eff}}^2}{(s-1) g D_{50}} \quad (17.75)$$

$$M_e = \frac{(v_{\text{eff}} - v_{cr})^2}{(s-1) g D_{50}} \quad (17.76)$$

$$v_{\text{eff}} = \sqrt{v_R^2 + U_{on}^2} \quad (17.77)$$

in which:

v_{cr}	critical depth averaged velocity for initiation of motion (based on a parameterisation of the Shields curve) [m/s]
v_R	magnitude of an equivalent depth-averaged velocity computed from the velocity in the bottom computational layer, assuming a logarithmic velocity profile [m/s]
U_{on}	near-bed peak orbital velocity [m/s] in onshore direction (in the direction of wave propagation) based on the significant wave height

U_{on} (and U_{off} used below) are the high frequency near-bed orbital velocities due to short waves and are computed using a modification of the method of [Isobe and Horikawa \(1982\)](#). This method is a parameterisation of fifth-order Stokes wave theory and third-order cnoidal wave theory which can be used over a wide range of wave conditions and takes into account the non-linear effects that occur as waves propagate in shallow water ([Grasmeijer and Van Rijn, 1998](#)).

The direction of the bedload transport vector is determined by assuming that it is composed of two parts: part due to current ($S_{b,c}$) which acts in the direction of the near-bed current, and

part due to waves ($S_{b,w}$) which acts in the direction of wave propagation. These components are determined as follows:

$$S_{b,c} = \frac{S_b}{\sqrt{1 + r^2 + 2|r|\cos\varphi}} \quad (17.78)$$

$$|S_{b,w}| = r |S_{b,c}| \quad (17.79)$$

where:

$$r = \frac{(|U_{on}| - v_{cr})^3}{(|v_R| - v_{cr})^3} \quad (17.80)$$

$S_{b,w} = 0$ if $r < 0.01$, $S_{b,c} = 0$ if $r > 100$, and φ = angle between current and wave direction for which [Van Rijn \(2003\)](#) suggests a constant value of 90° .

Also included in the “bedload” transport vector is an estimation of the suspended sediment transport due to wave asymmetry effects. This is intended to model the effect of asymmetric wave orbital velocities on the transport of suspended material within about 0.5 m of the bed (the bulk of the suspended transport affected by high frequency wave oscillations).

This wave-related suspended sediment transport is again modelled using an approximation method proposed by [Van Rijn \(2001\)](#):

$$S_{s,w} = f_{susw} \gamma U_A L_T \quad (17.81)$$

where:

$S_{s,w}$	wave-related suspended transport [kg/(ms)]
f_{susw}	user-defined tuning parameter
γ	phase lag coefficient (= 0.2)
U_A	velocity asymmetry value [m/s] = $\frac{U_{on}^4 - U_{off}^4}{U_{on}^3 + U_{off}^3}$
L_T	suspended sediment load [kg/m ²] = $0.007\rho_s D_{50} M_e$

The three separate transport modes are imposed separately. The direction of the bedload due to currents $S_{b,c}$ is assumed to be equal to the direction of the current, whereas the two wave related transport components $S_{b,w}$ and $S_{s,w}$ take on the wave propagation direction. This results in the following transport components:

$$S_{bc,u} = \frac{u_{b,u}}{|u_b|} |S_{b,c}| \quad (17.82)$$

$$S_{bc,v} = \frac{u_{b,v}}{|u_b|} |S_{b,c}| \quad (17.83)$$

$$S_{bw,u} = S_{b,w} \cos \phi \quad (17.84)$$

$$S_{bw,v} = S_{b,w} \sin \phi \quad (17.85)$$

$$S_{sw,u} = S_{s,w} \cos \phi \quad (17.86)$$

$$S_{sw,v} = S_{s,w} \sin \phi \quad (17.87)$$

where ϕ is the local angle between the direction of wave propagation and the computational grid. The different transport components can be calibrated independently by using the `Bed`, `BedW` and `SusW` keywords in the morphology input file.

17.5.2 Engelund-Hansen (1967)

The Engelund-Hansen sediment transport relation has frequently been used in rivers and estuaries. It reads:

$$S = S_b + S_{s,eq} = \frac{0.05\alpha q^5}{\sqrt{g}C^3\Delta^2 D_{50}} \quad (17.88)$$

where:

q	magnitude of flow velocity
Δ	the relative density $(\rho_s - \rho_w)/\rho_w$
C	Chézy friction coefficient
α	calibration coefficient ($O(1)$)

The transport rate is imposed as bedload transport due to currents S_{bc} . The following formula specific parameters have to be specified in the input files of the Transport module: calibration coefficient α and roughness height r_k .



Remarks:

- ◊ The D_{50} grain size diameter is based on the sediment fraction considered.
- ◊ A second formula specific input parameter (r_k) is required for the Engelund-Hansen formula. This parameter, which represents the roughness height for currents alone in [m], is only used to determine the C value when the Chézy friction in the flow has not been defined. Generally, this parameter can thus be treated as a dummy parameter.

17.5.3 Meyer-Peter-Muller (1948)

The Meyer-Peter-Muller sediment transport relation is slightly more advanced than the Engelund-Hansen formula, as it includes a critical shear stress for transport. It reads:

$$S = 8\alpha D_{50} \sqrt{\Delta g D_{50}} (\mu\theta - \xi\theta_{cr})^{3/2} \quad (17.89)$$

where:

α	calibration coefficient ($O(1)$)
Δ	the relative density $(\rho_s - \rho_w)/\rho_w$
μ	ripple factor or efficiency factor
θ_{cr}	critical mobility parameter ($= 0.047$)
ξ	hiding and exposure factor for the sediment fraction considered

and the Shields mobility parameter θ given by

$$\theta = \left(\frac{q}{C}\right)^2 \frac{1}{\Delta D_{50}} \quad (17.90)$$

in which q is the magnitude of the flow velocity [m/s]. The ripple factor μ reads:

$$\mu = \min \left(\left(\frac{C}{C_{g,90}} \right)^{1.5}, 1.0 \right) \quad (17.91)$$

where $C_{g,90}$ is the Chézy coefficient related to grains, given by:

$$C_{g,90} = 18^{10} \log \left(\frac{12(d + \zeta)}{D_{90}} \right) \quad (17.92)$$

with D_{90} specified in [m]. The transport rate is imposed as bedload transport due to currents S_{bc} . The following formula specific parameters have to be specified in the input files of the Transport module: calibration coefficient α and a dummy value.

Remark:

- ◊ The D_{50} is based on the sediment fraction considered, the D_{90} grain size diameters is based on the composition of the local sediment mixture.



17.5.4 General formula

The general sediment transport relation has the structure of the Meyer-Peter-Muller formula, but all coefficients and powers can be adjusted to fit your requirements. This formula is aimed at experienced users that want to investigate certain parameters settings. In general this formula should not be used. It reads:

$$S = \alpha D_{50} \sqrt{\Delta g D_{50}} \theta^b (\mu \theta - \xi \theta_{cr})^c \quad (17.93)$$

where ξ is the hiding and exposure factor for the sediment fraction considered and

$$\theta = \left(\frac{q}{C} \right)^2 \frac{1}{\Delta D_{50}} \quad (17.94)$$

in which q is the magnitude of the flow velocity.

The transport rate is imposed as bedload transport due to currents S_{bc} . The following parameters have to be specified in the input files of the Transport module: calibration coefficient α , powers b and c , ripple factor or efficiency factor μ , critical mobility parameter θ_{cr} .

17.5.5 Bijker (1971)

The Bijker formula sediment transport relation is a popular formula which is often used in coastal areas. It is robust and generally produces sediment transport of the right order of magnitude under the combined action of currents and waves. Bedload and suspended load are treated separately. The near-bed sediment transport (S_b) and the suspended sediment transport (S_s) are given by the formulations in the first sub-section. It is possible to include sediment transport in the wave direction due to wave asymmetry and bed slope following the Bailard approach, see [Bailard \(1981\)](#), [Stive \(1986\)](#). Separate expressions for the wave asymmetry and bed slope components are included:

$$\vec{S}_b = \vec{S}_{b0} + \vec{S}_{b,asymm} + \vec{S}_{s,asymm} + \vec{S}_{b,slope} + \vec{S}_{s,slope} \quad (17.95)$$

$$\vec{S}_s = \vec{S}_{s0} \quad (17.96)$$

where S_{b0} and S_{s0} are the sediment transport in flow direction as computed according to the formulations of Bijker in the first sub-section, and the asymmetry and bed slope components for bedload and suspended transport are defined in the second sub-section. Both bedload and suspended load terms are incorporated in the bedload transport for further processing. The transport vectors are imposed as bedload transport vector due to currents S_{bc} and suspended load transport magnitude S_s , from which the equilibrium concentration is derived, respectively.

17.5.5.1 Basic formulation

The basic formulation of the sediment transport formula according to Bijker is given by:

$$S_b = b D_{50} \frac{q}{C} \sqrt{g} (1 - \phi) \exp(A_r) \quad (17.97)$$

$$S_s = 1.83 S_b \left(I_1 \ln \left(\frac{33.0 h}{r_c} \right) + I_2 \right) \quad (17.98)$$

where

C	Chézy coefficient (as specified in input of D-Flow FM module)
h	water depth
q	flow velocity magnitude
ϕ	porosity

and

$$A_r = \max(-50, \min(100, A_{ra})) \quad (17.99)$$

$$b = BD + \max\left(0, \min\left(1, \frac{(h_w/h) - C_d}{C_s - C_d}\right)\right) (BS - BD) \quad (17.100)$$

$$I_1 = 0.216 \frac{\left(\frac{r_c}{h}\right)^{z_*-1}}{\left(1 - \frac{r_c}{h}\right)^{z_*}} \int_{r_c/h}^1 \left(\frac{1-y}{y}\right)^{z_*} dy \quad (17.101)$$

$$I_2 = 0.216 \frac{\left(\frac{r_c}{h}\right)^{z_*-1}}{\left(1 - \frac{r_c}{h}\right)^{z_*}} \int_{r_c/h}^1 \ln y \left(\frac{1-y}{y}\right)^{z_*} dy \quad (17.102)$$

where

BS	Coefficient b for shallow water (default value 5)
BD	Coefficient b for deep water (default value 2)
C_s	Shallow water criterion (H_s/h) (default value 0.05)
C_d	Deep water criterion (default value 0.4)
r_c	Roughness height for currents [m]

and

$$A_{ra} = \frac{-0.27 \Delta D_{50} C^2}{\mu q^2 \left(1 + 0.5 \left(\psi \frac{U_b}{q}\right)^2\right)} \quad (17.103)$$

$$\mu = \left(\frac{C}{18^{10} \log(12h/D_{90})} \right)^{1.5} \quad (17.104)$$

$$z_* = \frac{w}{\frac{\kappa q \sqrt{g}}{C} \sqrt{1 + 0.5 \left(\psi \frac{U_b}{q}\right)^2}} \quad (17.105)$$

$$U_b = \frac{\omega h_w}{2 \sinh(k_w h)} \quad (17.106)$$

$$\omega = \frac{2\pi}{T} \quad (17.107)$$

$$f_w = \exp \left(-5.977 + \frac{5.123}{a_0^{0.194}} \right) \quad (17.108)$$

$$a_0 = \max \left(2, \frac{U_b}{\omega r_c} \right) \quad (17.109)$$

$$\psi = \begin{cases} C \sqrt{\frac{f_w}{2g}} & \text{if wave effects are included } (T > 0) \\ 0 & \text{otherwise} \end{cases} \quad (17.110)$$

where

C	Chézy coefficient (as specified in input of D-Flow FM module)
h_w	wave height (H_{rms})
k_w	wave number
T	wave period computed by the waves model or specified by you as T user.
U_b	wave velocity
w	sediment fall velocity [m/s]
Δ	relative density $(\rho_s - \rho_w)/\rho_w$
κ	Von Kármán constant (0.41)

The following formula specific parameters have to be specified in the input files of the Transport module: BS , BD , C_s , C_d , dummy argument, r_c , w , ε and T user.

17.5.5.2 Transport in wave propagation direction (Bailard-approach)

If the Bijker formula is selected it is possible to include sediment transport in the wave direction due to wave asymmetry following the Bailard approach, see [Bailard \(1981\)](#) and [Stive \(1986\)](#). For a detailed description of the implementation you are referred to [Nipius \(1998\)](#).

Separate expressions for the wave asymmetry and bed slope components are included for both bedload and suspended load. Both extra bedload and suspended load transport vectors are added to the bedload transport as computed in the previous sub-section:

$$\vec{S}_b = \vec{S}_{b0} + \vec{S}_{b,asymm} + \vec{S}_{s,asymm} + \vec{S}_{b,slope} + \vec{S}_{s,slope} \quad (17.111)$$

where the asymmetry components for respectively the bedload and suspended transport in wave direction are written as:

$$S_{b;asymm}(t) = \frac{\rho c_f \varepsilon_b}{(\rho_s - \rho) g (1 - \phi) \tan \varphi} |u(t)|^2 u(t) \quad (17.112)$$

$$S_{s;asymm}(t) = \frac{\rho c_f \varepsilon_s}{(\rho_s - \rho) g (1 - \phi) w} |u(t)|^3 u(t) \quad (17.113)$$

from which the components in directions tangential and normal to flow links are obtained by multiplying with the cosine and sine of the wave angle θ_w and the bed slope components as:

$$S_{b;slope,\xi}(t) = \frac{\rho c_f \varepsilon_b}{(\rho_s - \rho) g (1 - \phi) \tan \varphi} \frac{1}{\tan \varphi} |u(t)|^3 \frac{\partial z_b}{\partial \xi} \quad (17.114)$$

$$S_{s;slope,\xi}(t) = \frac{\rho c_f \varepsilon_s}{(\rho_s - \rho) g (1 - \phi) w} \frac{\varepsilon_s}{w} |u(t)|^5 \frac{\partial z_b}{\partial \xi} \quad (17.115)$$

and similar for the η direction, where:

$u(t)$	near bed velocity signal [m/s]
ρ	density of water [kg/m^3]
ρ_s	density of the sediment [kg/m^3]
c_f	coefficient of the bottom shear stress [-] (constant value of 0.005)
ϕ	porosity [-] (constant value of 0.4)
φ	natural angle of repose [-] (constant value of $\tan \varphi = 0.63$)
w	sediment fall velocity [m/s]
ε_b	efficiency factor of bedload transport [-] (constant value of 0.10)
ε_s	efficiency factor of suspended transport [-] (constant value of 0.02, but in implemented expression for suspended bed slope transport the second ε_s is replaced by a user-defined calibration factor; see Equation 17.118).

These transports are determined by generating velocity signals of the orbital velocities near the bed by using the [Rienecker and Fenton \(1981\)](#) method, see also [Roelvink and Stive \(1989\)](#).

The (short wave) averaged sediment transport due to wave asymmetry, Equations [17.112](#) and [17.113](#), is determined by using the following averaging expressions of the near bed velocity signal (calibration coefficients included):

$$\langle u |u|^2 \rangle = FacA \langle \tilde{u} |\tilde{u}|^2 \rangle + 3FacU \bar{u} \langle |\tilde{u}|^2 \rangle \quad (17.116)$$

$$\langle u |u|^3 \rangle = FacA \langle \tilde{u} |\tilde{u}|^3 \rangle + 4FacU \bar{u} \langle |\tilde{u}|^3 \rangle \quad (17.117)$$

in which:

\tilde{u}	orbital velocity signal
\bar{u}	averaged flow velocity (due to tide, undertow, wind, etc.)
$FacA$	user-defined calibration coefficient for the wave asymmetry
$FacU$	user-defined calibration coefficient for the averaged flow

The suspended transport relation due to the bed slope according to [Equation 17.115](#) is implemented as:

$$S_{s;slope,\xi}(t) = \frac{\rho c_f \varepsilon_s}{(\rho_s - \rho) g (1 - \phi) w} \frac{\varepsilon_{sl}}{w} |u(t)|^5 \frac{\partial z_b}{\partial \xi} \quad (17.118)$$

where:

ε_{sl}	user-defined calibration coefficient EpsSL
--------------------	--

To activate this transport option, you have to create a separate file named <coef.inp> which contains on three separate lines the calibration coefficients: $FacA$, $FacU$ and $EpsSL$. The other parameters are read from the transport input file or are specified as general sediment characteristics.



Note: the user-defined $FacU$ value is currently treated as a dummy value, $FacU = 0.0$ will always be used.

A validation study ([Nipius, 1998](#)) showed that the following coefficient settings yielded the best results for the Dutch coast:

$FacA = 0.4$
 $FacU = 0.0$
 $EpsSL = 0.11$

If a relatively straight coast is considered the effect of the parameters is:

- ◊ The wave asymmetry causes onshore directed sediment transport (i.e. in the wave propagation direction). An increased F_{acA} results in an increased onshore transport and hence steepening of the cross-shore bottom profile.
- ◊ The bed slope transport is in general offshore directed. By increasing $EpsSL$ an increased flattening of the bottom profile occurs (i.e. increased offshore transports).
- ◊ The ratio between these parameters determines the balance between onshore and offshore transport and hence the shape and slope of the cross-shore bottom profile. The associated response time of the cross-shore morphology can be influenced by modifying the values of the two parameters, but maintaining a constant ratio. Increased values result in increased gross transports and consequently a reduced morphological response time (and vice versa).

17.5.6 Van Rijn (1984)

The [Van Rijn \(1984a,b,c\)](#) sediment transport relation is a transport formula commonly used for fine sediments in situations without waves. Separate expressions for bedload and suspended load are given. The bedload transport rate is given by:

$$S_b = \begin{cases} 0.053\sqrt{\Delta g D_{50}^3} D_*^{-0.3} T^{2.1} & \text{for } T < 3.0 \\ 0.1\sqrt{\Delta g D_{50}^3} D_*^{-0.3} T^{1.5} & \text{for } T \geq 3.0 \end{cases} \quad (17.119)$$

where T is a dimensionless bed shear parameter, written as:

$$T = \frac{\mu_c \tau_{bc} - \tau_{bcr}}{\tau_{bcr}} \quad (17.120)$$

It is normalised with the critical bed shear stress according to Shields (τ_{bcr}), the term $\mu_c \tau_{bc}$ is the effective shear stress. The formulas of the shear stresses are

$$\tau_{bc} = \frac{1}{8} \rho_w f_{cb} q^2 \quad (17.121)$$

$$f_{cb} = \frac{0.24}{(10 \log(12h/\xi_c))^2} \quad (17.122)$$

$$\mu_c = \left(\frac{18 \cdot 10 \log(12h/\xi_c)}{C_{g,90}} \right)^2 \quad (17.123)$$

where $C_{g,90}$ is the grain related Chézy coefficient

$$C_{g,90} = 18 \cdot 10 \log \left(\frac{12h}{3D_{90}} \right) \quad (17.124)$$

The critical shear stress is written according to Shields:

$$\tau_{bcr} = \rho_w \Delta g D_{50} \theta_{cr} \quad (17.125)$$

in which θ_{cr} is the Shields parameter which is a function of the dimensionless particle parameter D_* :

$$D_* = D_{50} \left(\frac{\Delta g}{\nu^2} \right)^{\frac{1}{3}} \quad (17.126)$$

The suspended transport formulation reads:

$$S_s = f_{cs} q h C_a \quad (17.127)$$

In which C_a is the reference concentration, q depth averaged velocity, h the water depth and f_{cs} is a shape factor of which only an approximate solution exists:

$$f_{cs} = \begin{cases} f_0(z_c) & \text{if } z_c \neq 1.2 \\ f_1(z_c) & \text{if } z_c = 1.2 \end{cases} \quad (17.128)$$

$$f_0(z_c) = \frac{(\xi_c/h)^{z_c} - (\xi_c/h)^{1.2}}{(1 - \xi_c/h)^{z_c} (1.2 - z_c)} \quad (17.129)$$

$$f_1(z_c) = \left(\frac{\xi_c/h}{1 - \xi_c/h} \right)^{1.2} \ln(\xi_c/h) \quad (17.130)$$

where ξ_c is the reference level or roughness height (can be interpreted as the bedload layer thickness) and z_c the suspension number:

$$z_c = \min \left(20, \frac{w_s}{\beta \kappa u_*} + \phi \right) \quad (17.131)$$

$$u_* = q \sqrt{\frac{f_{cb}}{8}} \quad (17.132)$$

$$\beta = \min \left(1.5, 1 + 2 \left(\frac{w_s}{u_*} \right)^2 \right) \quad (17.133)$$

$$\phi = 2.5 \left(\frac{w_s}{u_*} \right)^{0.8} \left(\frac{C_a}{0.65} \right)^{0.4} \quad (17.134)$$

The reference concentration is written as:

$$C_a = 0.015 \alpha_1 \frac{D_{50}}{\xi_c} \frac{T^{1.5}}{D_*^{0.3}} \quad (17.135)$$

The bedload transport rate is imposed as bedload transport due to currents S_{bc} , while the computed suspended load transport rate is converted into a reference concentration equal to $f_{cs} C_a$. The following formula specific parameters have to be specified in the input files of the Transport module: calibration coefficient α_1 , dummy argument, reference level (bedload layer thickness) or roughness height ξ_c [m] and settling velocity w_s [m/s].

17.5.7 Soulsby/Van Rijn

The sediment transport relation has been implemented based on the formulations provided in [Soulsby \(1997\)](#). References in the following text refer to this book.

If the wave period T_p is smaller than 10^{-6} s, the wave period T_p is set to 5 s and the root-mean-square wave height is set to 1 cm. Furthermore, the wave period is limited to values larger than 1 s. The root-mean-square wave height is limited to values smaller than 0.4 H , where H is the water depth.

The sediment transport is set to zero in case of velocities smaller than 10^{-6} m/s, water depth larger than 200 m or smaller than 1 cm.

The root-mean-square orbital velocity is computed as:

$$U_{rms} = \sqrt{2} \frac{\pi H_{rms}}{T_p \sinh(kH)} \quad (17.136)$$

Furthermore, D_* is defined as (Soulsby, 1997, p.104):

$$D_* = \left(\frac{g\Delta}{\nu^2} \right)^{1/3} D_{50} \quad (17.137)$$

Using the critical bed shear velocity according to Van Rijn (Soulsby, 1997, p.176):

$$U_{cr} = \begin{cases} 0.19 D_{50}^{0.1} 10 \log(4H/D_{90}) & \text{if } D_{50} \leq 0.5 \text{ mm} \\ 8.5 D_{50}^{0.6} 10 \log(4H/D_{90}) & \text{if } 0.5 \text{ mm} < D_{50} \leq 2 \text{ mm} \end{cases} \quad (17.138)$$

larger values of D_{50} lead to an error and to the halting of the program.

The sediment transport is split into a bedload and suspended load fraction. The direction of the bedload transport is assumed to be equal to the direction of the depth-averaged velocity in a 2D simulation and equal to the direction of the velocity at the reference height a (see section 17.3.3) in a 3D simulation (Soulsby, 1997, p.183):

$$S_{bx} = A_{cal} A_{sb} u \xi \quad (17.139)$$

$$S_{by} = A_{cal} A_{sb} v \xi \quad (17.140)$$

and the suspended transport magnitude is given by the following formula (this quantity is later converted to a reference concentration to feed the advection-diffusion equation for the suspended sediment transport as indicated in section 17.3.3):

$$S_s = A_{cal} A_{ss} \xi \sqrt{u^2 + v^2} \quad (17.141)$$

where

A_{cal}	a user-defined calibration factor
A_{sb}	bedload multiplication factor

$$A_{sb} = 0.005H \left(\frac{D_{50}/H}{\Delta g D_{50}} \right)^{1.2} \quad (17.142)$$

A_{ss}	suspended load multiplication factor
----------	--------------------------------------

$$A_{ss} = 0.012D_{50} \frac{D_*^{-0.6}}{(\Delta g D_{50})^{1.2}} \quad (17.143)$$

ξ	a general multiplication factor
-------	---------------------------------

$$\xi = \left(\sqrt{U^2 + \frac{0.018}{C_D} U_{rms}^2} - U_{cr} \right)^{2.4} \quad (17.144)$$

where U is the total depth-averaged velocity and C_D is the drag coefficient due to currents, defined by:

$$C_D = \left(\frac{\kappa}{\ln(H/z_0) - 1} \right)^2 \quad (17.145)$$

where z_0 equals 6 mm and the Von Kármán constant κ is set to 0.4.

The bedslope correction factor is not explicitly included in this formula as it is a standard correction factor available in the online morphology module. The method is intended for conditions in which the bed is rippled.

The following formula specific parameters have to be specified in the input files of the Transport module: the calibration factor A_{cal} , the ratio of the two characteristic grain sizes D_{90}/D_{50} and the z_0 roughness height.

17.5.8 Soulsby

The sediment transport relation has been implemented based on the formulations provided in [Soulsby \(1997\)](#). References in the following text refer to this book.

If the wave period T_p is smaller than 10^{-6} s, the wave period T_p is set to 5 s and the root-mean-square wave height is set to 1 cm. Furthermore, the wave period is limited to values larger than 1 s. The root-mean-square wave height is limited to values smaller than $0.4 H$, where H is the water depth.

The sediment transport is set to zero in case of velocities smaller than 10^{-6} m/s, water depth larger than 200 m or smaller than 1 cm.

The root-mean-square orbital velocity U_{rms} and the orbital velocity U_{orb} are computed as

$$U_{rms} = \sqrt{2}U_{orb} = \sqrt{2} \frac{\pi H_{rms}}{T_p \sinh(kH)} \quad (17.146)$$

For a flat, non-rippled bed of sand the z_0 roughness length is related to the grain size as ([Soulsby, 1997](#), eq.25, p.48) where χ is a user-defined constant:

$$z_0 = \frac{D_{50}}{\chi} \quad (17.147)$$

The relative roughness is characterised using a_* :

$$a_* = \frac{U_{orb} T_p}{z_0} \quad (17.148)$$

which is subsequently used to determine the friction factor of the rough bed according to [Swart \(1974\)](#):

$$f_w = \begin{cases} 0.3 & \text{if } a_* \leq 30\pi^2 \\ 0.00251 \exp(14.1a_*^{-0.19}) & \text{if } a_* > 30\pi^2 \end{cases} \quad (17.149)$$

which corresponds to formulae 60a/b of Soulsby (p.77) using $r = a_*/(60\pi)$ where r is the relative roughness used by Soulsby. The friction factor is used to compute the amplitude of the bed shear-stress due to waves as:

$$\tau_w = 0.5\rho f_w U_{orb}^2 \quad (17.150)$$

Furthermore, the shear stress due to currents is computed as:

$$\tau_c = \rho C_D U^2 \quad (17.151)$$

Table 17.2: Overview of the coefficients used in the various regression models ([Soulsby et al., 1993a](#))

Model	b1	b2	b3	b4	p1	p2	p3	p4
1 (FR84)	0.29	0.55	-0.10	-0.14	-0.77	0.10	0.27	0.14
2 (MS90)	0.65	0.29	-0.30	-0.21	-0.60	0.10	0.27	-0.06
3 (HT91)	0.27	0.51	-0.10	-0.24	-0.75	0.13	0.12	0.02
4 (GM79)	0.73	0.40	-0.23	-0.24	-0.68	0.13	0.24	-0.07
5 (DS88)	0.22	0.73	-0.05	-0.35	-0.86	0.26	0.34	-0.07
6 (BK67)	0.32	0.55	0.00	0.00	-0.63	0.05	0.00	0.00
7 (CJ85)	0.47	0.29	-0.09	-0.12	-0.70	0.13	0.28	-0.04
8 (OY88)	-0.06	0.26	0.08	-0.03	-1.00	0.31	0.25	-0.26

where

$$C_D = \left(\frac{\kappa}{1 + \ln(z_0/H)} \right)^2 \quad (17.152)$$

as defined on [Soulsby \(1997, p.53–55\)](#). The interaction of the currents and waves is taken into account using the factor Y in the following formula for mean bed shear stress during a wave cycle under combined waves and currents ([Soulsby, 1997, p.94](#)):

$$\tau_m = Y (\tau_w + \tau_c) \quad (17.153)$$

The formula for Y is given by:

$$Y = X [1 + bX^p (1 - X)^q] \quad (17.154)$$

where:

$$X = \frac{\tau_c}{\tau_c + \tau_w} \quad (17.155)$$

and b is computed using:

$$b = (b_1 + b_2 |\cos \phi|^J) + (b_3 + b_4 |\cos \phi|^J)^{10} \log(f_w/C_D) \quad (17.156)$$

and p and q are determined using similar equations. In this formula ϕ equals the angle between the wave angle and the current angle, and the coefficients are determined by the model index modind and tables 17.2 and 17.3 (related to [Soulsby \(1997, Table 9, p.91\)](#)):

Using the shear stresses given above, the following two Shields parameters are computed:

$$\theta_m = \frac{\tau_m}{\rho g \Delta D_{50}} \text{ and } \theta_w = \frac{\tau_w}{\rho g \Delta D_{50}} \quad (17.157)$$

Furthermore, D_* is defined as ([Soulsby, 1997, p.104](#)):

$$D_* = \left(\frac{g \Delta}{\nu^2} \right)^{1/3} D_{50} \quad (17.158)$$

Table 17.3: Overview of the coefficients used in the various regression models, continued
[\(Soulsby et al., 1993a\)](#)

Model	q1	q2	q3	q4	J
1 (FR84)	0.91	0.25	0.50	0.45	3.0
2 (MS90)	1.19	-0.68	0.22	-0.21	0.50
3 (HT91)	0.89	0.40	0.50	-0.28	2.7
4 (GM79)	1.04	-0.56	0.34	-0.27	0.50
5 (DS88)	-0.89	2.33	2.60	-2.50	2.7
6 (BK67)	1.14	0.18	0.00	0.00	3.0
7 (CJ85)	1.65	-1.19	-0.42	0.49	0.60
8 (OY88)	0.38	1.19	0.25	-0.66	1.50

with which a critical Shields parameter is computed ([Soulsby, 1997](#), eq.77, p.106):

$$\theta_{cr} = \frac{0.30}{1 + 1.2D_*} + 0.055 (1 - \exp(-0.02D_*)) \quad (17.159)$$

The sediment transport rates are computed using the following formulations for normalised transport in current direction and normal direction ([Soulsby, 1997](#), eq.129, p.166/167):

$$\Phi_{x1} = 12(\theta_m - \theta_{cr}) \sqrt{\theta_m + \varepsilon} \quad (17.160)$$

$$\Phi_{x2} = 12(0.95 + 0.19 \cos(2\phi)) \theta_m \sqrt{\theta_w + \varepsilon} \quad (17.161)$$

$$\Phi_x = \max(\Phi_{x1}, \Phi_{x2}) \quad (17.162)$$

$$\Phi_y = \frac{12(0.19\theta_m\theta_w^2 \sin(2\phi))}{(\theta_w + \varepsilon)^{1.5} + 1.5(\theta_m + \varepsilon)^{1.5}} \quad (17.163)$$

where ε is a small constant (10^{-4}) to prevent numerical complications. From these expression are finally the actual bedload transport rates obtained:

$$S_{b,x} = \frac{\sqrt{g\Delta D_{50}^3}}{U} (\Phi_x u - \Phi_y v) \quad (17.164)$$

$$S_{b,y} = \frac{\sqrt{g\Delta D_{50}^3}}{U} (\Phi_x v - \Phi_y u) \quad (17.165)$$

The transport vector is imposed as bedload transport due to currents. The following formula specific parameters have to be specified in the input files of the Transport module: calibration coefficient A_{cal} , the model index for the interaction of wave and current forces $modind$ (integer number 1 to 8) and the D_{50}/z_0 ratio χ (about 12).

17.5.9 Ashida-Michiue (1974)

The transport rate is given by a generalised version of the Ashida-Michiue formulation:

$$S_{bc} = \alpha \sqrt{\Delta g D_{50}^3} \theta^m \left(1 - \xi \frac{\theta_c}{\theta}\right)^p \left(1 - \sqrt{\xi \frac{\theta_c}{\theta}}\right)^q \quad (17.166)$$

where ξ is the hiding and exposure factor for the sediment fraction considered and:

$$\theta = \left(\frac{q}{C} \right)^2 \frac{1}{\Delta D_{50}} \quad (17.167)$$

in which q is the magnitude of the flow velocity. The transport rate is imposed as bedload transport due to currents S_{bc} . The following formula specific parameters have to be specified in the input files of the Transport module: α , θ_c , m , p and q .

17.5.10 Wilcock-Crowe (2003)

The Wilcock-Crowe transport model is a fractional surface based transport model for calculating bedload transport of mixed sand and gravel sediment. The equations and their development are described in [Wilcock and Crowe \(2003\)](#). The bedload transport rate of each size fraction is given by:

$$S_{bi} = \frac{W_i^* F_i U_*^3}{\Delta g} \quad (17.168)$$

$$W_i^* = \begin{cases} 0.002\phi^{7.5} & \text{for } \phi < 1.35 \\ 14 \left(1 - \frac{0.894}{\phi^{0.5}}\right)^{4.5} & \text{for } \phi \geq 1.35 \end{cases} \quad (17.169)$$

$$\phi = \frac{\tau}{\tau_{ri}} \quad (17.170)$$

$$\frac{\tau_{ri}}{\tau_{rm}} = \left(\frac{D_i}{D_m} \right)^b \quad (17.171)$$

$$\tau_{rm} = (0.021 + 0.015 \exp(-20F_s)) (\rho_s - \rho_w) g D_g \quad (17.172)$$

$$b = \frac{0.67}{1 + \exp\left(1.5 - \frac{D_i}{D_g}\right)} \quad (17.173)$$

where:

D_i	D_{50} of size fraction i
D_g	geometric mean grain size of whole grain size distribution
F_i	proportion of size fraction i on the bed surface
F_s	proportion of sand on the bed surface
S_{bi}	bedload transort rate of size fraction i
W_i^*	dimensionless bedload transport rate of size fraction i
Δ	the relative density of the sediment $(\rho_s - \rho_w) / \rho_w$
τ_{ri}	reference shear stress of grains of size D_i
τ_{rm}	reference shear stress of grains of size D_g

Remarks:

- ◊ The Wilcock-Crowe model incorporates its own hiding function so no external formulation should be applied.
- ◊ The roughness height used for the calculation of grain shear stress during the development of the Wilcock-Crowe transport model was $k_s = 2D_{65}$.
- ◊ This sediment transport formula does not have any input parameters that can be, or need to be, tuned.



17.5.11 Gaeuman et al. (2009) laboratory calibration

The Gaeuman et al. sediment transport model is a modified form of the Wilcock-Crowe model which uses the variance of grain size distribution on the phi scale (σ_ϕ^2) rather than the fraction of sand on the bed surface (F_s) as a measure of the bed surface condition for use in the calculation of reference shear stress. The 'laboratory calibration' implementation of the Gaeuman et al. transport model is calibrated to the experimental data used in the derivation of the Wilcock-Crowe transport model. The model, its derivation and calibration is described in [Gaeuman et al. \(2009\)](#).

The formulae for the calculation of S_{bi} , W_i^* , ϕ and τ_{ri} are the same as for the Wilcock-Crowe transport model (Equations 17.168, 17.169, 17.170 and 17.171) but the calculation of τ_{rm} and b differs.

$$\tau_{rm} = \left(\theta_{c0} + \frac{0.015}{1 + \exp(10.1\sigma_\phi^2 - 14.14)} \right) (\rho_s - \rho_w) g D_g \quad (17.174)$$

$$b = \frac{1 - \alpha_0}{1 + \exp\left(1.5 - \frac{D_i}{D_g}\right)} \quad (17.175)$$

$$\sigma_\phi^2 = \sum_{i=1}^n \left(2 \log\left(\frac{D_i}{D_g}\right) \right)^2 F_i \quad (17.176)$$

where θ_{c0} and α_0 are user specified parameters. If the values $\theta_{c0} = 0.021$ and $\alpha_0 = 0.33$ are specified the original relation calibrated to the Wilcock-Crowe laboratory data is recovered.



Remark:

- ◊ The Gaeuman et al. model incorporates its own hiding function so no external formulation should be applied.

17.5.12 Gaeuman et al. (2009) Trinity River calibration

The 'Trinity River calibration' implementation of the Gaeuman et al. transport model is calibrated to observed bedload transport rates in the Trinity River, USA and is described in [Gaeuman et al. \(2009\)](#). It differs from the 'laboratory calibration' implementation in the calculation of τ_{rm} and b .

$$\tau_{rm} = \left(\theta_{c0} + \frac{0.022}{1 + \exp(7.1\sigma_\phi^2 - 11.786)} \right) (\rho_s - \rho_w) g D_g \quad (17.177)$$

$$b = \frac{1 - \alpha_0}{1 + \exp\left(1.9 - \frac{D_i}{3D_g}\right)} \quad (17.178)$$

where θ_{c0} and α_0 are user specified parameters. If the values $\theta_{c0} = 0.03$ and $\alpha_0 = 0.3$ are specified the original Gaeuman et al. formulation calibrated to the Trinity River is recovered.



Remark:

- ◊ The Gaeuman et al. model incorporates its own hiding function so no external formulation should be applied.

17.6 Morphological updating

The elevation of the bed is dynamically updated at each computational time-step. This is one of the distinct advantages over an offline morphological computation as it means that the hydrodynamic flow calculations are always carried out using the correct bathymetry.

At each time-step, the change in the mass of bed material that has occurred as a result of the sediment sink and source terms and transport gradients is calculated. This change in mass is then translated into a bed level change based on the dry bed densities of the various sediment fractions. Both the bed levels at the cell centres and cell interfaces are updated.

Remark:

- ◊ The depths stored at the depth points (which are read directly from the bathymetry specified as input) are only updated for writing to the communication file and the result files.



A number of additional features have been included in the morphological updating routine in order to increase the flexibility. These are discussed below.

Morphological “switch”

You can specify whether or not to update the calculated depths to the bed by setting the MorUpd (or equivalently BedUpd) flag in the morphology input file. It may be useful to turn bottom updating off if only the initial patterns of erosion and deposition are required, or an investigation of sediment transport patterns with a constant bathymetry is required.

Remark:

- ◊ The use of MorUpd or BedUpd only affects the updating of the depth values (at ζ and velocity points); the amount of sediment available in the bed will still be updated. Use the CmpUpd flag to switch off the updating of the bed composition. If you wish to prevent any change in both the bottom sediments *and* flow depths from the initial condition then this may also be achieved by either setting the morphological delay interval MorStt to a value larger than the simulation period, or by setting the morphological factor MorFac to 0. See below for a description of these two user variables.



Morphological delay

Frequently, a hydrodynamic simulation will take some time to stabilise after transitioning from the initial conditions to the (dynamic) boundary conditions. It is likely that during this stabilisation period the patterns of erosion and accretion that take place do not accurately reflect the true morphological development and should be ignored. This is made possible by use of MorStt whereby you can specify a time interval (in minutes after the start time) after which the morphological bottom updating will begin. During the MorStt time interval all other calculations will proceed as normal (sediment will be available for suspension for example) however the effect of the sediment fluxes on the available bottom sediments will not be taken into account.

Morphological time scale factor

One of the complications inherent in carrying out morphological projections on the basis of hydrodynamic flows is that morphological developments take place on a time scale several times longer than typical flow changes (for example, tidal flows change significantly in a period of hours, whereas the morphology of a coastline will usually take weeks, months, or years to change significantly). One technique for approaching this problem is to use a “morphological time scale factor” whereby the speed of the changes in the morphology is scaled up to a rate that it begins to have a significant impact on the hydrodynamic flows. This can be achieved by specifying a non-unity value for the variable MorFac in the morphology input file.



Remark:

- ◊ The *Morphological scale factor* can also be time-varying. This feature is not yet supported by the GUI. You have to edit the <*.mor> file manually.

The implementation of the morphological time scale factor is achieved by simply multiplying the erosion and deposition fluxes from the bed to the flow and vice-versa by the MorFac-factor, at each computational time-step. This allows accelerated bed-level changes to be incorporated dynamically into the hydrodynamic flow calculations.

While the maximum morphological time scale factor that can be included in a morphodynamic model without affecting the accuracy of the model will depend on the particular situation being modelled, and will remain a matter of judgement, tests have shown that the computations remain stable in moderately morphologically active situations even with MorFac-factors in excess of 1 000. We also note that setting MorFac=0 is often a convenient method of preventing both the flow depth and the quantity of sediment available at the bottom from updating, if an investigation of a steady state solution is required.



Remarks:

- ◊ Verify that the morphological factor that you use in your simulation is appropriate by varying it (e.g. reducing it by a factor of 2) and verify that such changes do not affect the overall simulation results.
- ◊ The interpretation of the morphological factor differs for coastal and river applications. For coastal applications with tidal motion, the morphological variations during a tidal cycle are often small and the hydrodynamics is not significantly affected by the bed level changes. By increasing the morphological factor to for instance 10, the morphological changes during one simulated tidal cycle are increased by this factor. From a hydrodynamical point of view this increase in morphological development rate is allowed if the hydrodynamics is not significantly influenced. In that case the morphological development after one tidal cycle can be assumed to represent the morphological development that would in real life only have occurred after 10 tidal cycles. In this example the number of hydrodynamic time steps required to simulate a certain period is reduced by a factor of 10 compared to a full 1:1 simulation. This leads to a significant reduction in simulation time. However, one should note that by following this approach the order of events is changed, possible conflicts may arise in combination with limited sediment availability and bed stratigraphy simulations. In river applications there is no such periodicity as a tidal cycle. For such applications, the morphological factor should be interpreted as a speed-up factor for morphological development without changing the order of events. Effectively, it means that the morphological development is simulated using a, for instance 10 times, larger time step than the hydrodynamics, or phrased more correctly the hydrodynamics is simulated at a 10 times faster rate. This means that in case of time-varying boundary conditions (e.g. river hydrograph) the time-scale of these forcings should be sped up: a 20 day flood peak will be compressed in 2 days. However, one should take care that by speeding up the hydrodynamic forcings one

does not substantially change the nature of the overall hydrodynamic and morphological development: a quasi-steady flood period should not become a short, dynamic flash flood. For river applications, changing the morphological factor must be associated with changing all external time-varying forcings. For coastal applications only the overall simulation time should be adjusted. Note that the combination of a river-like flood peak and a tidal motion will cause problems when interpreting morphological factor not equal to 1.

- ◊ The effect of the morphological factor is different for bed and suspended load. At each time step bedload is picked-up from the bed and deposited on the bed: only the transports are increased by the morphological factor used for the time step considered. However, in case of suspended load there is a time-delay between the time of erosion and the time of deposition. The erosion and deposition fluxes are increased by the morphological factor, but the suspended concentrations are not (since that would influence the density effects). It is possible to vary the morphological factor during a simulation to speed up relatively quiet periods more than relatively active periods. Such changes in the morphological factor will not influence the mass balance of a bed or total load simulation since pickup and deposition are combined into one time step. However, in case of suspended load the entrainment and deposition may occur at time-steps governed by different morphological factors. In such cases the entrainment flux that generated a certain suspended sediment concentration will differ from the deposition flux that was caused by the settling of the same suspended sediment. A change in morphological factor during a period of non-zero suspended sediment concentrations, will thus lead to a mass-balance error in the order of the suspended sediment volume times the change in morphological factor. The error may kept to a minimum by appropriately choosing the transition times.

17.6.1 Bathymetry updating including bedload transport

The change in the quantity of bottom sediments caused by the bedload transport is calculated using the expression:

$$\Delta_{SED}^c = \frac{\Delta t f_{MORFAC}}{A^c} \sum_i S_u^i W_u^i \quad (17.179)$$

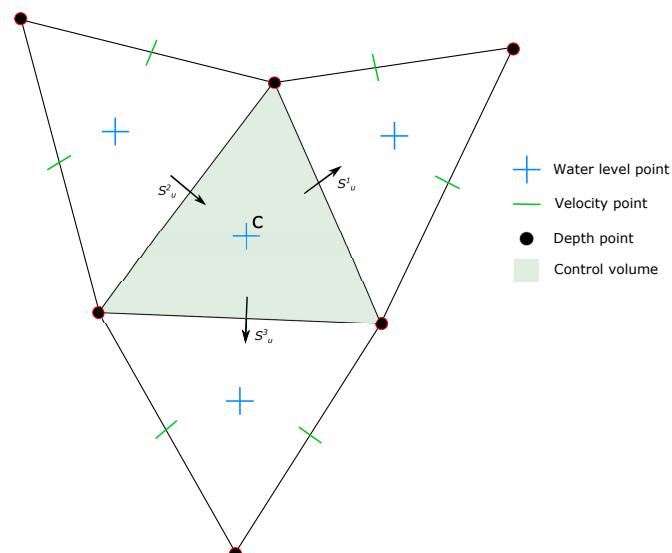


Figure 17.5: Morphological control volume and bedload transport components

where:

Δc_{SED}	change in quantity of bottom sediment at location c [kg/m ²]
Δt	computational time-step [s]
f_{MORFAC}	user-defined morphological acceleration factor, MORFAC
A^c	area of computational cell at location c [m ²]
S_u^i	computed bedload sediment transport vector in u direction (direction normal to face i), held at the velocity point [kg/(m s)]
W_u^i	width of the face i [m]

This calculation is repeated for all ‘sand’ and ‘bedload’ sediment fractions, if more than one is present, and the resulting change in the bottom sediment mass is added to the change due to the suspended sediment sources and sinks and included in the bed composition and bed level updating scheme.

17.6.2 Erosion of (temporarily) dry points

In the case of erosion near a dry beach or bank, the standard scheme will not allow erosion of the adjacent cells, even when a steep scour hole would develop right next to the beach. Therefore a scheme has been implemented that allows the (partial) redistribution of an erosion flux from a wet cell to the adjacent dry cells. The distribution is governed by a user-defined factor ThetSD , which determines the fraction of the erosion to assign (evenly) to the adjacent cells. If ThetSD equals zero the standard scheme is used, i.e. all erosion occurs at the wet cell. If ThetSD equals 1 all erosion that would occur in the wet cell is assigned to the adjacent dry cells. The ‘wet’ and ‘dry’ cells in the paragraph above are defined as cells at which the water depth is, respectively, more and less than the threshold depth SedThr for computing sediment transport.

A modification to this method may be activated by specifying a parameter HMaxTH larger than the threshold depth SedThr for computing sediment transport. In this case, the factor ThetSD is used as upper limit for the fraction of the erosion to be transferred to adjacent dry cells. The actual factor to be transferred is equal to Thet , which is computed as:

$$\text{Thet} = (h_1 - \text{SedThr}) / (\text{HMaxTH} - \text{SedThr}) \times \text{ThetSD}$$

where $\text{Thet} = \min(\text{Thet}, \text{ThetSD})$ (17.180)

Here, h_1 is the local water depth. The purpose of this formulation is to allow erosion of parts that are inactive in terms of transport but still wet, while limiting the erosion of the dry beach. If erosion of the dry beach is desired, this option is not recommended, so HMaxTH should be set less than SedThr .



Remark:

- ◊ The overall erosion flux is redistributed to the adjacent cells. Depending on the availability of individual sediment fractions at the central ‘wet’ cell and the surrounding ‘dry’ cells, the erosion from the adjacent cells will replenish the eroded cell with different sediment fractions than those that were eroded.

17.6.3 Dredging and dumping

If the bed levels are updated, you may also include some dredging and dumping activities at the end of each half time step. This feature can also be used for sand mining (only dredging, no associated dumping within the model domain) and sediment nourishment (only dumping, no associated dredging within the model domain). Dredging and dumping is performed at this stage in the following order:

- ◊ For each dredge area: if the bed level exceeds a threshold level (or the water depth drops below a certain level) then the bed level is lowered based on the dredging option and the corresponding volume of sediment is removed. If the dredging capacity is less than the volume to be dredged, the sequence of dredging (e.g. top first or uniform) determines which grid cells are dredging at the current point in time.
- ◊ The volume of dredged material is summed over all cells in a dredge area and distributed over the dump areas, using the link percentages or the link order (up to the dump capacity). In simulations with multiple sediment fractions the sediment composition is tracked.
- ◊ For each dump area: the bed level is raised and the bed composition is adjusted based on the volume and characteristics of material to be dumped. The sediment may be distributed equally or non-uniformly (e.g. deepest points first) over the grid cells in the dump area.

Remark:

- ◊ Dredging and dumping may also be performed during initialization, before the first time-step.

Warning:

- ◊ Dredging large amounts of material may harm the stability of the calculation.

The dredging and dumping feature allows you to specify dredging and dumping areas as x,y polygons. Within each dredging polygon the bed levels are lowered to a user-defined depth; by default grid cells are considered to lie within a polygon if their centre lies within the polygon. It is possible to distribute the dredged material over multiple dumping locations. You may also decide to not dump the sediment back into the model (feature referred to as sand mining); this can be implemented by defining a dump polygon outside the grid, or by not specifying any dump polygon at all. This option cannot be combined with the option to dredge only as much as dump capacity is available. For sediment nourishment one should use a [nourishment] block specifying the amount (and, if applicable, the composition) of the nourished sediment. The dredging and dumping activities should be specified in a <*.dad> file. The <mdu> file should contain a keyword `Fildad` referring to the file used. The <*.dad> file refers to the file containing the polygons.

17.6.4 Bed composition models and sediment availability

The morphology module currently implements two bed composition models:

- ◊ A uniformly mixed bed (one sediment layer). There is no bookkeeping of the order in which sediments are deposited and all sediments are available for erosion.
- ◊ A layered bed stratigraphy (multiple sediment layers). A user-defined number of bed composition bookkeeping layers may be included to keep track of sediment deposits. When sediments are deposited, they are initially added to the top-most layer. After mixing in the top layer, sediments are pushed towards the bookkeeping layers beneath it. The bookkeeping layers are filled up to a user-defined maximum thickness, if this threshold is exceeded a new layer is created. If the creation of a new layer would exceed the maximum number of layers specified by you, layers at the bottom of the stratigraphy stack will be merged. Only sediments in the top-most layer are available for erosion. After erosion, the top-most layer is replenished from below.

The default bed composition model is the uniformly mixed one. Currently only the default bed composition model is supported by the user interface.

At input you must specify the amount of sediment available at the bed as the total (dry) mass of all sediment fractions in [kg/m^2]. This may be a constant value for the entire model or,

alternatively, a space-varying initial sediment file (values to be specified at cell centres). The initial bed composition is assumed to be uniformly mixed.¹ The thickness of the total sediment layer is calculated from the sediment mass by dividing by the user-defined dry bed density CDryB. Currently, CDryB is constant in time and space for each individual sediment fraction. The top of these sediment deposits will coincide with the initial bed level. Below the bottom of these deposits the model assumes a non-erodible bed (sometimes referred to as a fixed layer).

When the model almost runs out of sediment at a particular location, the sediment flux terms will be reduced. The reduction starts when the available sediment thickness drops below a user-defined threshold Thresh. The flux terms affected are slightly different for cohesive and non-cohesive sediments, as described below.

Cohesive sediment fractions

In the case of cohesive sediment, the erosive sediment source term is reduced proportionally to the ratio of available sediment thickness over Thresh. The deposition term is never reduced. However, the cohesive sediment is still not included in D-Flow FM.

Non-cohesive sediment fractions

In the case of non-cohesive sediments all bedload transport rates out of a grid cell are reduced by the upwind ratio of available sediment thickness over Thresh. The source and sink terms of the advection-diffusion equation are not reduced unless the erosive sediment source term is predicted to be larger than the deposition (sink) term, in that case both terms are reduced by the ratio of available sediment thickness over Thresh as shown by the following equations:

$$\text{Source}_{total} = \text{Source}_{total} * f_r, \quad (17.181)$$

$$\text{Sink}_{total} = \text{Sink}_{total} * f_r, \quad (17.182)$$

where f_r is a reduction factor determined by:

$$f_r = \min \left(\frac{\Delta_{sed}}{Thresh}, 1 \right), \quad (17.183)$$

where Δ_{sed} is the thickness of sediment at the bed.

The likelihood of erosive conditions occurring is assessed by calculating the total sediment source and sink terms using the concentration from the previous time-step to evaluate the implicit sink term. If the sink term is greater than the source term, then deposition is expected, and f_r is set to 1.0 so that deposition close to the fixed layer is not hindered.

17.7 Specific implementation aspects

Negative water depth check

In rare situations (with high morphological acceleration factors) it is possible that, in one time-step, the bed accretes more than the water depth. If this occurs the water depth will become negative (water surface level is below the bed level). This situation is checked for and, if it occurs, the water surface level for the cell is set equal to the new bed level. The cell will then be set dry.

¹The uniformly mixed bed can be used as input for both bed composition models. If you have more detailed information on the bed stratigraphy, you may use the bed stratigraphy model and specify an initial layering of the bed composition by means of the IniComp keyword and associated initial bed composition file. In that case the bed composition given in the <*.sed> file will be overruled, you have to specify dummy values though.

Threshold depth for sediment calculations introduced

If the water depth in a cell is less than SedThr, specified in the morphology input file, then the sediment source and sink terms and bedload transport are not calculated for 'sand' and 'bedload' sediment fractions. This restriction has been included in order to prevent numerical problems during the computation of the reference concentration, e.g. to prevent sudden bursts of sediment from occurring when computational cells are flooded.

Remark:

- ◊ In areas with very shallow water depths and sediment sources and sinks, you must ensure that the user-defined threshold depth for drying and flooding is not set too large.



Calculation of bed shear in wave and current situations altered

The calculation of the bed shear velocity u_* has been simplified in situations with waves and currents. The bed shear is always calculated using the velocities computed in the bottom computational layer, rather than using the computational layer closest to the top of the sediment mixing layer. The reference velocity in the bottom computational layer is adjusted to the top of the sediment mixing layer using the apparent bed roughness k_a before being used to compute the bed shear velocity using the physical bed roughness k_s .

Depth at grid cell faces (velocity points)

During a morphological simulation the depth stored at the U and V velocity points must be updated to reflect the bed level changes calculated in the water level points. This *used to be* performed by setting the new depth for the velocity point by copying the new depth held at the water level point, using a simple upwind numerical scheme. As this may introduce instabilities in the flow computation, especially near drying and flooding and in tidal simulations, this method has been replaced by setting the depth at U and V points equal to the *minimum* of the adjacent depths in water level points. This change significantly improves the smoothness of flooding dry cells.

Remarks:

- ◊ The setting of depths at velocity points as the minimum of the adjacent water level points only comes into effect if sediment is present and the user-defined flag MORUPD is `.true.` (i.e. bathymetrical changes are expected to occur at some point during the simulation period). If this condition is not met then the depths at the velocity points do not need to be updated during the course of the simulation.
- ◊ The program still requires the depth at velocity points to be set to MOR for morphological simulations. This anticipated that this restriction is lifted in a coming release.
- ◊ Since the MOR and MIN procedures for computing the depth at cell interfaces are equivalent, we advise you to use the MIN procedure during the calibration of a hydronamic model that will later on be converted into a morphological model.



18 Tutorial

18.1 Introduction

In this brief introduction, basic information is provided on either the **setup of the tutorial** and some **basic grid concepts**.



18.1.1 Setup of the tutorial

In this three-hours workshop, you will experience the basic functionality of D-Flow FM. On the one hand, you will learn how to capture a certain area with a computational grid. On the other hand, you will set up a computation yourself, including inserting a bathymetry, imposing boundary conditions, etcetera. This manual contains 9 tutorials (with indication of the estimated time):

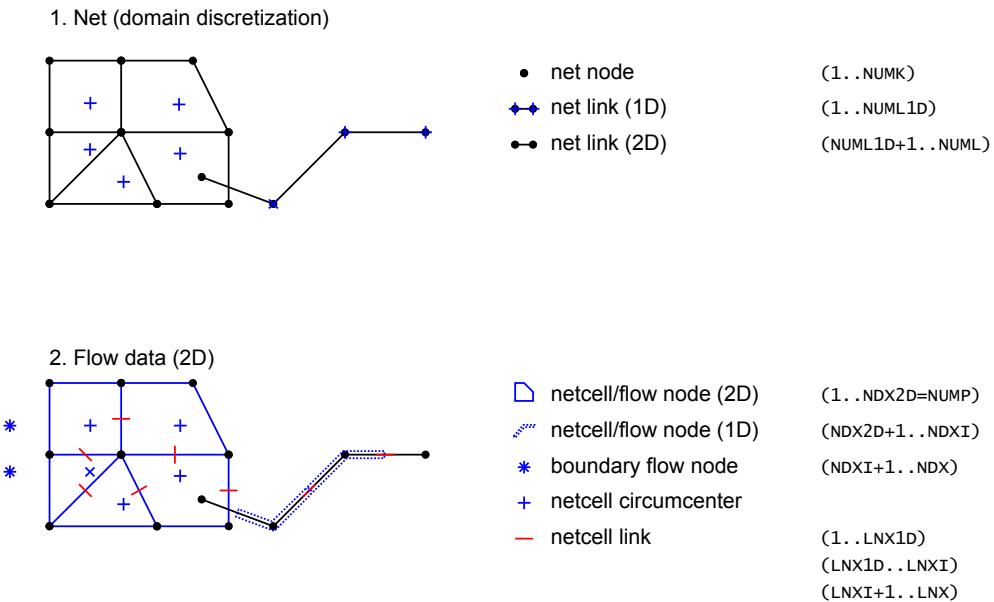
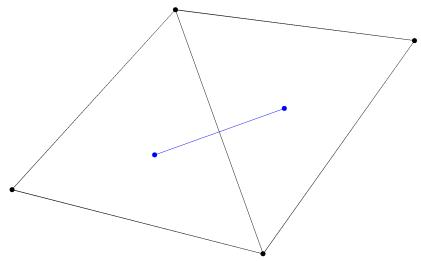
- ◊ Tutorial 1 ([section 18.2](#)): Creating a curvilinear grid (25 minutes).
- ◊ Tutorial 2 ([section 18.3](#)): Creating a triangular grid (20 minutes).
- ◊ Tutorial 3 ([section 18.4](#)): Coupling multiple distinct grids (15 minutes).
- ◊ Tutorial 4 ([section 18.5](#)): Inserting a bathymetry (15 minutes).
- ◊ Tutorial 5 ([section 18.6](#)): Imposing boundary conditions (25 minutes).
- ◊ Tutorial 6 ([section 18.7](#)): Defining output locations (15 minutes).
- ◊ Tutorial 7 ([section 18.8](#)): Defining computational parameters (10 minutes).
- ◊ Tutorial 8 ([section 18.9](#)): Running a model simulation (20 minutes).
- ◊ Tutorial 9 ([section 18.10](#)): Viewing the output of a model simulation (15 minutes).

This estimated times leave room for a 5 minutes introduction and a 15 minutes break after tutorial 4. The necessary input files for a tutorial can be found in the directory *tutorial01*, *tutorial02*, etc. You can put your output files in a separate directory, e.g. *tutorial01_your_outcome*.

18.1.2 Basic grid concepts

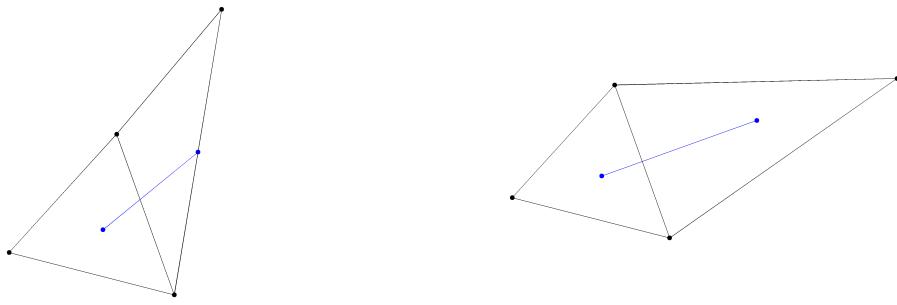
In D-Flow FM, grids (sometimes denoted as 'networks') consist of netcells and are described by **netnodes** (corners of a cell), **netlinks** (edges of a cell, connecting netnodes), **flownodes** (the cell circumcentre) and **flowlinks** (a line segment connecting two flownodes).

This grid topology is illustrated in [Figure 18.1](#). Important features of the grid are the *orthogonality* and *smoothness*. The *orthogonality* is defined as the cosine of the angle φ between a flowlink and a netlink. The *smoothness* of a grid is defined as the ratio of the areas of two adjacent cells. Ideally, both parameters equal 1, i.e. the angle $\varphi = 90^\circ$ and the areas of the cells are equal to each other. A nearly ideal setup is shown in [Figure 18.2](#).

**Figure 18.1:** Topology and definitions for a grid as used in D-Flow FM.**Figure 18.2:** Nearly perfect orthogonality and smoothness along the edge connecting two triangles. Black lines/dots are network links/nodes, blue lines/dots are flow links/nodes.

It is rather facile to generate grids that violate the orthogonality and smoothness requirements. In [Figure 18.3](#), two different setups of two gridcells are shown with different grid properties. The left picture of [Figure 18.3](#) shows how orthogonality can be deteriorated by skewing the right triangle with respect to the left triangle. While having the same area (perfect smoothness), the mutually oblique orientation results in poor orthogonality. In this particular case, the centre of the circumscribing circle is in principle located outside the right triangle. Such a triangle is denoted as an 'open' triangle, which is bad for computations.

The opposite is shown in the right picture of [Figure 18.3](#) in which the right triangle has strongly been elongated, disturbing the smoothness property. However, the orthogonality is nearly perfect. Nonetheless, both grids need to be improved to assure accurate model results.



(a) Perfect smoothness, but poor orthogonality. (b) Perfect orthogonality, but poor smoothness

Figure 18.3: Poor grid properties due to violating either the smoothness or the orthogonality at the edge connecting two triangles. Black lines/dots are network links/nodes, blue lines/dots are flow links/nodes.

18.2 Tutorial 1: Creating a curvilinear grid

In the upcoming tutorials, the Westerscheldt and the harbour of Antwerp will be used as an example case for generating a grid and setting up a computation.

To start RGFGRID from Delta Shell and set a coordinate system:

- 1 Open Delta Shell.
- 2 Choose *New Model* (top left of screen) → *D-Flow FM model* → *OK*.
- 3 Choose *Edit grid (rgfgrid)*, see [Figure 18.4](#). RGFGRID will now start.

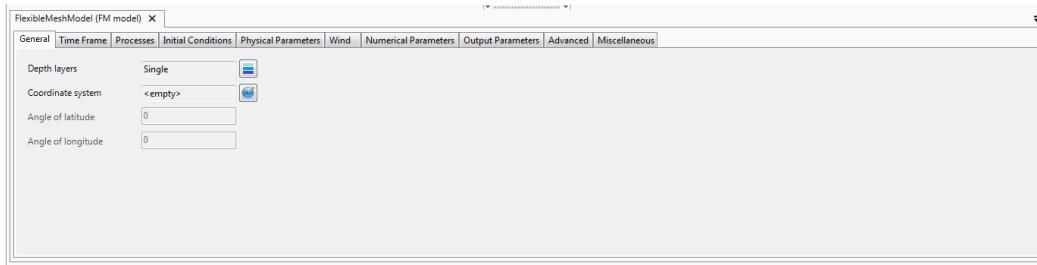


Figure 18.4: Delta Shell after opening a D-Flow FM model.

- 4 Choose *Co-ordinate System* → *Cartesian Co-ordinates*.

To generate curvilinear grids from splines, a grid is gradually developed from a base line of the Scheldt river towards its boundaries. This approach can be illustrated as follows:

- 1 Open the landboundary file <scheldriver.ldb> from the folder *tutorial01*, by choosing *File* → *Attribute files* → *Open Land Boundary*.
- 2 Select *Edit* → *Spline* → *New* and draw a spline at the left boundary of the river (use the leftmousebutton). Finalise the spline by one rightmouseclick.
- 3 Select the lefthandside spline through choosing *Edit* → *Spline* → *Select*.
- 4 Choose the option *Edit* → *Spline* → *Spline to landboundary (New)* → *Extra snapping required: Yes*. Press *Yes* several times. You will see the spline evolve towards the landboundary. Press *No* when you are satisfied with the result.
- 5 Repeat the previous three steps for the righthandside spline.
- 6 Draw 2 cross-splines: 1 spline at the North side and 1 spline at the South side of the river (also see [Figure 18.6](#) lateron). The channel is now enveloped by 4 splines!

- 7 Choose *Settings → Grow grid from splines*. You will be able to set several settings of the operator. The upper 7 entries should be adapted into the values shown in [Figure 18.5](#).

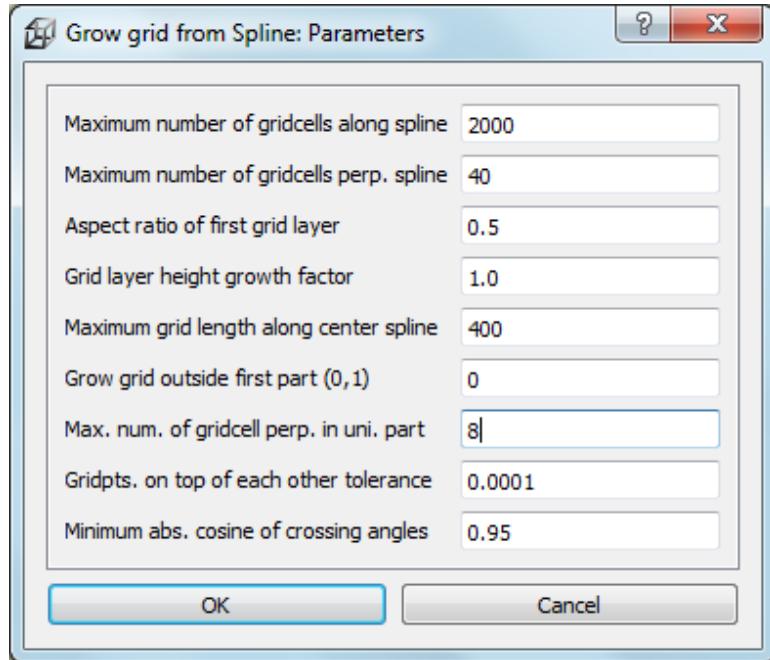


Figure 18.5: Settings for the 'grow grid from splines' procedure.

A brief explanation:

- ◊ Using the parameter *Max. num. of gridcells perp. in uni. part*, the user can give an indication of the number of cells across the width between the longitudinal splines.
- ◊ By using the parameters *Maximum grid length along center spline*, the user can give an indication of the length of the cells in longitudinal direction. Based on the value of the parameter *Aspect ratio of first grid layer*, the algorithm establishes a suitable grid, under the restrictions of the prevailing maximum numbers of gridcells (first two entries).
- ◊ The option *Grid layer height growth factor* enables the user to demand for a non-equidistant grid in cross-sectional direction. The value represents the width-ratio of two adjacent cells. Using the option *Grow grid outside first part (0/1)*, one can extend a grid outside the longitudinal splines, for instance to capture winterbed regions.

- 8 After entering the values of [Figure 18.5](#), choose *Operations → Grow grid from splines*. This will deliver the grid as shown in [Figure 18.6](#).

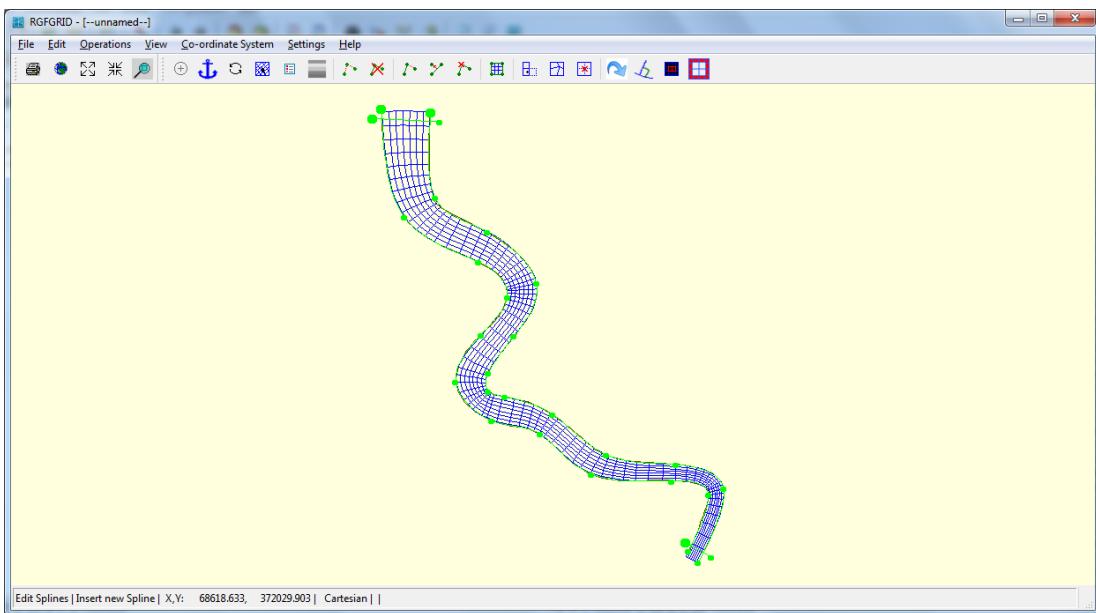


Figure 18.6: Generated curvilinear grid after the new 'grow grid from splines' procedure.

- 9 To be able to further amend the grid, choose *Operations* → *Convert grid* → *Regular to irregular*. Strictly, the grid is now not curvilinear anymore, but unstructured.
- 10 Choose *View* → *Grid property* → *Coloured edge* and then *Operations* → *Grid properties* → *Orthogonality*. The result is shown in Figure 18.7.

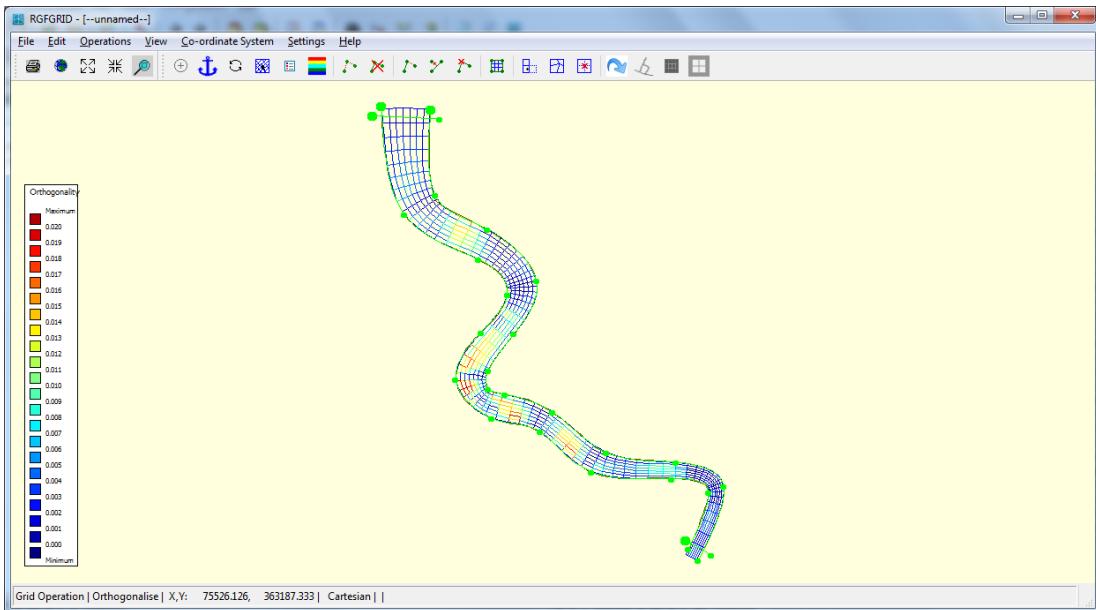


Figure 18.7: Orthogonality of the generated curvilinear grid after the new 'grow grid from splines' procedure after some orthogonalisation iterations.

To save the grid and clear the project afterwards:

- ◊ *File* → *Export* → *Grid (D-Flow FM)* → Save as <your_outcome_tut01_net.nc> in the folder *tutorial01_your_outcome*. This grid has now been saved.
- ◊ *File* → *Export* → *Splines with Intermediate Points* → Save as <your_outcome_tut01.spl>

- in folder *tutorial01_your_outcome*. The splines have now been saved.
- ◊ *File* → *New Project*. The existing grid has disappeared and a new project can start.
 - ◊ Close RGFGRID.

N.B. When closing RGFGRID, ignore the message: Failed to reload grid after RGFGrid edits: invalid empty file detected. Please save your project after editing in RGFGrid → Press OK.

18.3 Tutorial 2: Creating a triangular grid

We will continue with the grid created in the previous tutorial for the Scheldt river. The river is separated from the harbour, west of the river, by a sluice. The small area between the sluice and the Scheldt will benefit from an unstructured grid because of its irregular geometry. The unstructured grid for this irregular geometry is created first in this section.

The approach is as follows:

- 1 Start RGFGRID and set the Co-ordinate System (if necessary), as has been explained in [section 18.2](#).
- 2 Open from the folder *tutorial02* the landboundary file <scheldtharbour.lbd>, as has been explained in [section 18.2](#).
- 3 Import the grid file, by pressing: *File* → *Import* → *Grid (D-Flow FM)* and choosing the grid file <*tutorial01_net.nc*> in the folder *tutorial02*.
- 4 Click on *Edit* → *Polygon* → *New*. The intention is to mark the area of interest (i.e. the area that should be captured by the grid) through a polygon, see [Figure 18.8](#).
- 5 Start drawing a polygon at a distance of the order of a gridcell away from the curvilinear grid. Let the second point be at a relatively small distance from the first one. This distance is later used as an indication of the size of the triangular gridcells to be placed.
- 6 Mark the elementary locations of the area (follow the landboundary) and place the final point again at a distance of the order of a gridcell away from the river grid. The result is shown in [Figure 18.8](#).
- 7 Next, we choose *Edit* → *Polygon* → *Refine* and click on the first and last points of the polyline (in this case, point 1 first and then point 15), and click on the *right mouse button*. Now, the polygon is divided into a finer set of line elements (based on the distance between the first and last point of the polyline).

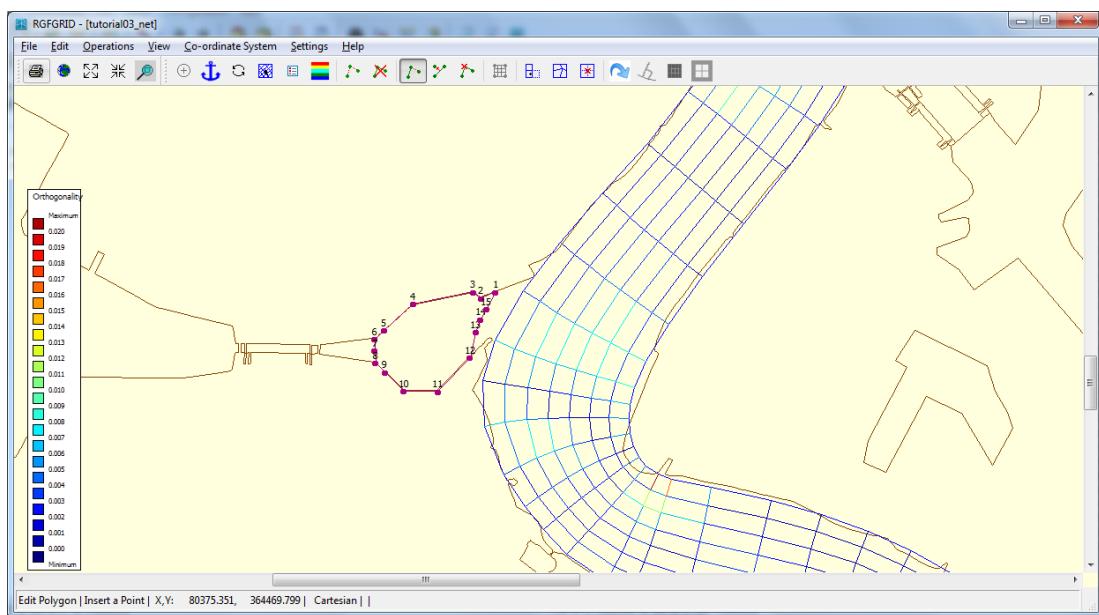


Figure 18.8: Polygon that envelopes the area in which an unstructured grid is aimed to be established.

Some remarks:

- ◊ The distance between the points of the polygon is derived from the distance of the two polyline segments at both sides of the *selected* segment. The length of the polyline segments varies linearly from the segment length at the one side of the selected segment towards the segment length at the other side of the selected segment.
 - ◊ You can play around to see how this works. If needed, you can add extra polyline points by choosing *Edit* → *Polygon* → *Insert point*. Choose *Edit* → *Polygon* → *Move point* if a point move would make sense.
 - ◊ You can snap the refined polygon to the landboundary through *Edit* → *Polygon* → *Polygon to landboundary*. Select two points for this.
- 8 Choose *Operations* → *Create grid from polygon*. The result is shown in [Figure 18.9](#).
 - 9 Select the created grid by: *Edit* → *Select Domain* → Click on the unstructured grid. The selected grid will turn blue.
 - 10 Improve the orthogonality through *Operations* → *Orthogonalise grid*.

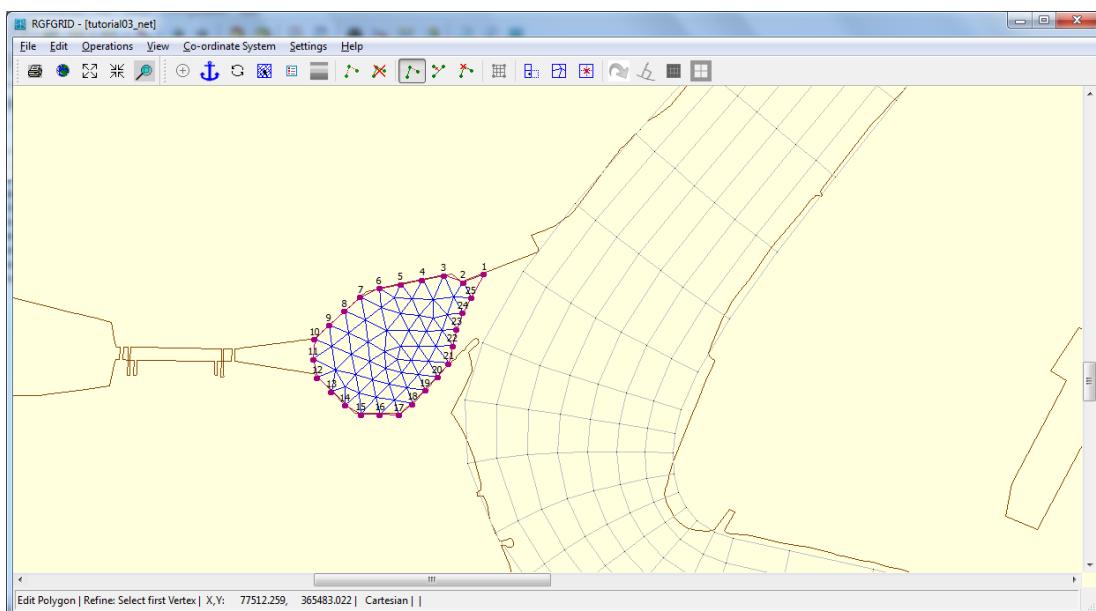


Figure 18.9: Unstructured grid, after having refined the polygon.

To save the grid and clear the project afterwards:

- ◊ *File* → *Export* → *Grid (D-Flow FM)* → Save as <your_outcome_tut02_net.nc> in the folder *tutorial02_your_outcome*. This grid has now been saved.
- ◊ *File* → *New Project*. The existing grid has disappeared and a new project can start.
- ◊ Close RGFGRID.

N.B. When closing RGFGRID, ignore the message: Failed to reload grid after RGFGrid edits: invalid empty file detected. Please save your project after editing in RGFGrid → Press OK.

18.4 Tutorial 3: Coupling multiple separate grids

From the previous tutorial, we have ended up with two separate grids. Obviously, these two grids should properly be integrated into one single grid. Before we can couple the two grids, we should first make sure that the typical gridsizes is of the same order of magnitude for both

grids at the location where the connection is to be laid. Hence, basically two operations are to be done:

- ◊ Split the gridcells in the Scheldt river grid over the full width. Hence, the gridcell size in the river will match the gridcell size of the unstructured grid.
- ◊ Merge the two grids and put connections in between of these.

The splitting can be established as follows:

- 1 Start RGFGRID and set the Co-ordinate System (if necessary), as has been explained in Tutorial 1.
- 2 Open from the folder *tutorial03* the file <scheldtharbour.ldb> (landboundaries of the harbour) and import the grid files <tutorial01_net.nc> and <tutorial02_net.nc>.
- 3 Select the river grid through *Edit* → *Select domain* and clicking the river grid.
- 4 Choose *Edit* → *Grid* → *Split row or column*.
- 5 Select the locations where the gridlines should be split by clicking on the left boundary of the Scheldt river.
- 6 Try to achieve the picture shown in [Figure 18.10](#), in particular the typical grid size in the curved area. *N.B. Ignore the red lines in this picture for now!*

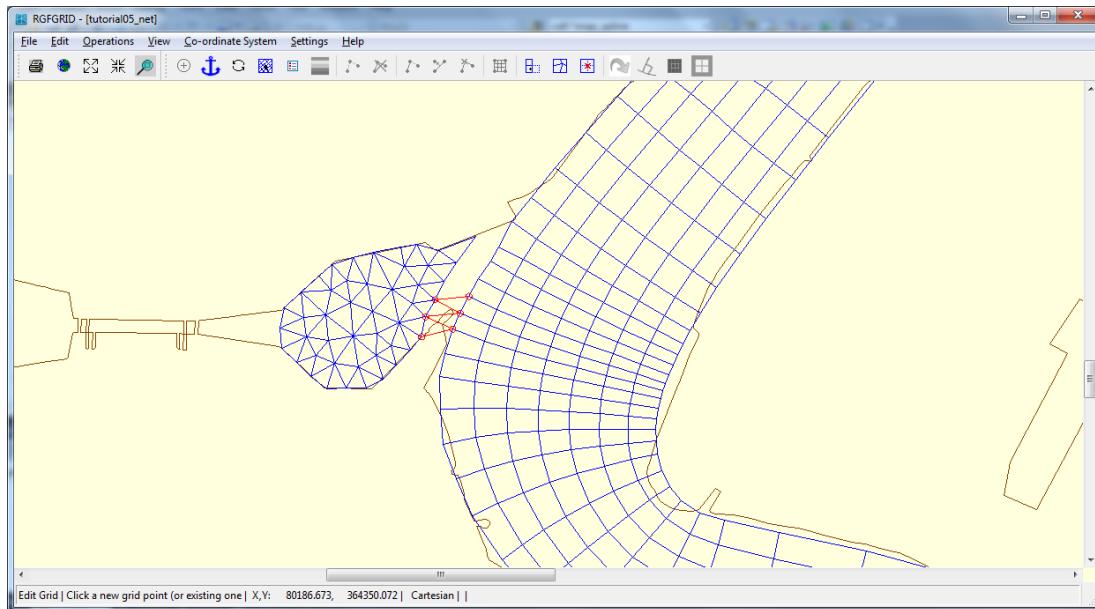


Figure 18.10: Connection of the river grid and the unstructured grid. The red lines show the inserted grid lines used to couple the two grids manually.

The merging part of the coupling schedule can be done as follows:

- 1 Choose *Edit* → *Allow Multi Select*. By now, you enable the option to select multiple grids.
- 2 Click on the triangular part of the grid. As soon as you have clicked on it, both grids are highlighted blue (both are therefore selected).
- 3 Merge the two separate grids through *Operations* → *Merge grids*. Now, the grids have been merged. Nothing will visually change, both grids will still be highlighted blue.
- 4 As soon as the grids have been merged, new connections can be laid. Hence, choose *Edit* → *Select Domain* → Click on the grid. Then *Edit* → *Grid* → *Point insert*. Insert new gridlines in a zigzag-like style: see the red grid lines in [Figure 18.10](#). Now, you will benefit from the (more or less) equal resolution in the river region as in the unstructured region.

- 5 Finally, you will end up with a picture like shown in [Figure 18.11](#). You can visualize the orthogonality through *Operations* → *Grid properties* → *Orthogonality* and *View* → *Grid property* → *Coloured edge*.

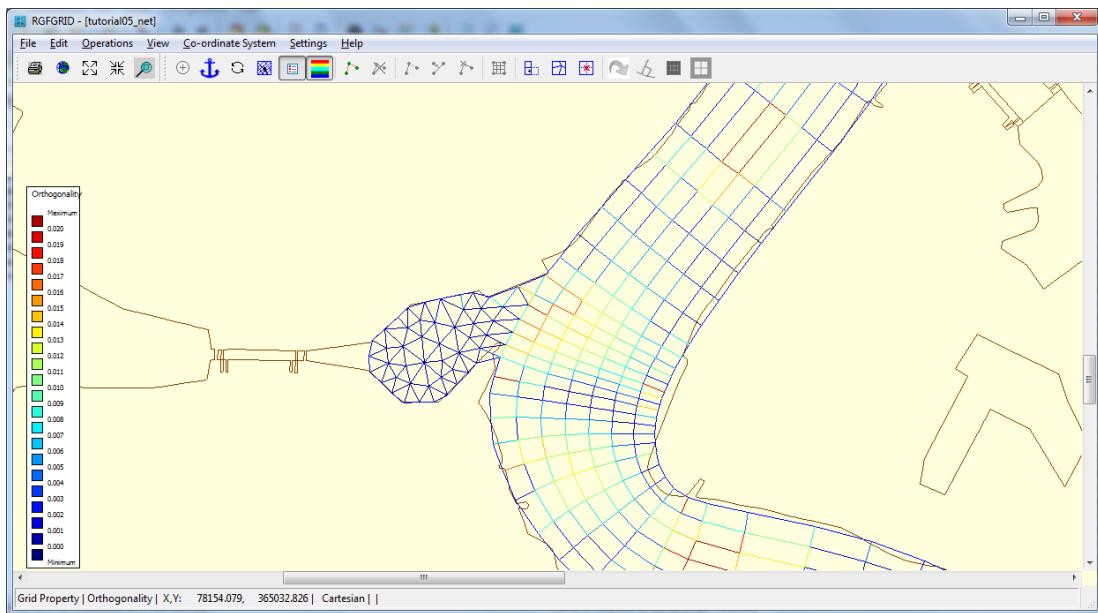


Figure 18.11: Orthogonality of the integrated grid, containing the curvilinear part, the triangular part and the coupling between the two grids.

To save the grid and clear the project afterwards:

- ◊ *File* → *Export* → *Grid (D-Flow FM)* → *Save as <your_outcome_tut03_net.nc>* in folder *tutorial03_your_outcome*. This grid has now been saved.
- ◊ *File* → *New Project*. The existing grid has disappeared and a new project can start.
- ◊ Close RGFGRID

N.B. When closing RGFGRID, ignore the message: Failed to reload grid after RGFGrid edits: invalid empty file detected. Please save your project after editing in RGFGrid → Press OK.

18.5 Tutorial 4: Inserting a bathymetry

Now, we return to Delta Shell. The grid generation has been done by now. To continue, we intend to first couple the bathymetry data to the grid.

An amended version of the grid is available from the source files: the file *<tutorial03_net.nc>* in the directory *tutorial04*. A harbour has been added to the grid, as well as the Westerscheldt part at the North side. Bottom levels are available in the file *<westerscheldt_bottom.xyz>*. The file with bottom levels should first be projected onto the unstructured grid by means of interpolation. The following actions should be done:

- 1 Start Delta Shell and start a D-Flow FM model: Choose *New Model* → *D-Flow FM model* → *OK*.
- 2 Load the following files from the folder *tutorial04*:
 - ◊ Grid file: rightmouseclick on *Domain* (see [Figure 18.12](#)) → *Import*: the available grid file *<tutorial03_net.nc>* from the directory *tutorial04*,

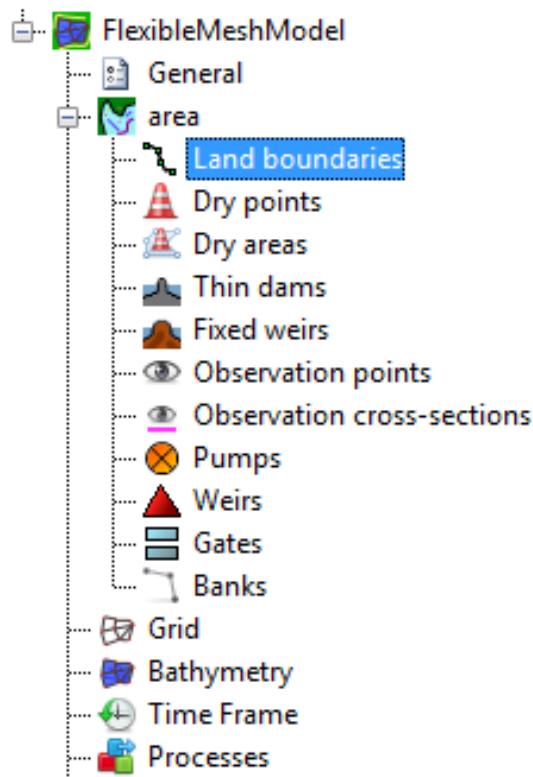


Figure 18.12: Project menu.

- ◊ Landboundary file: *Area 2D* (see [Figure 18.12](#)) → Rightmouseclick on *Land boundaries* → *Import: landboundary file <sealand.ldb>* → *Grid transformation* → *OK*,
- ◊ Sample file: in the drop down menu at the top of the screen (See [Figure 18.13](#)), choose *Bathymetry*. Then choose *Import: xyz file <westerscheldt_bottom.xyz>* which contains bottom level data for the Westerscheldt from the North Sea up to Antwerp → *Grid transformation* → *OK*.

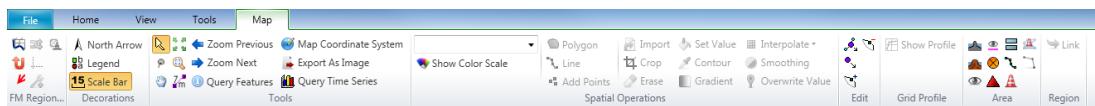


Figure 18.13: Spatial Operations menu bar (located at the upper right part of the Delta Shell).

- 3 Now we are going to project the bottom level data onto the netnodes. To that end, choose *Interpolate* (see [Figure 18.13](#)). The result will look like [Figure 18.14](#). If you do not see a colorbar, click *Show Color Scale* (see [Figure 18.13](#)).

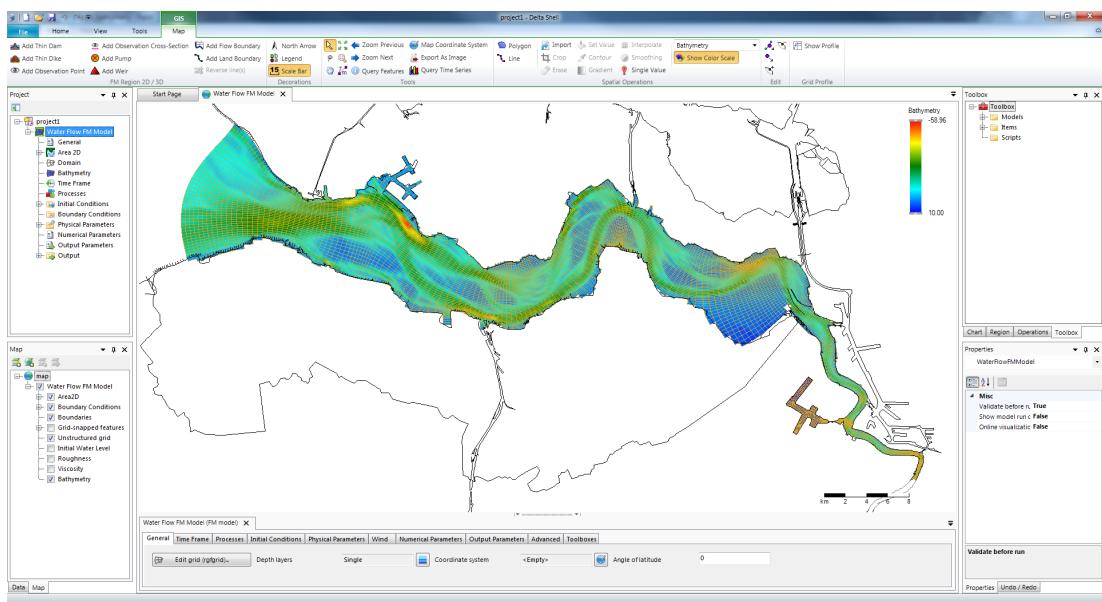


Figure 18.14: Interpolated bottom levels values at the grid.

- 4 Have a look at the sluice and the docks of the harbour. You can see that the bottom level in this area is unrealistically high. This unrealistic value has been assigned because of undesired extrapolation, since no bottom level data has been available in this area. To repair this, first draw a polygon that envelops the sluice and the docks. Do this by clicking *Polygon* (See [Figure 18.13](#)). The resulting polygon can be observed in [Figure 18.15](#).
- 5 Choose *Set value* (see [Figure 18.13](#)) → specify *uniform value* and choose an appropriate value, for instance -10 m . The result will look like the picture shown in [Figure 18.15](#).

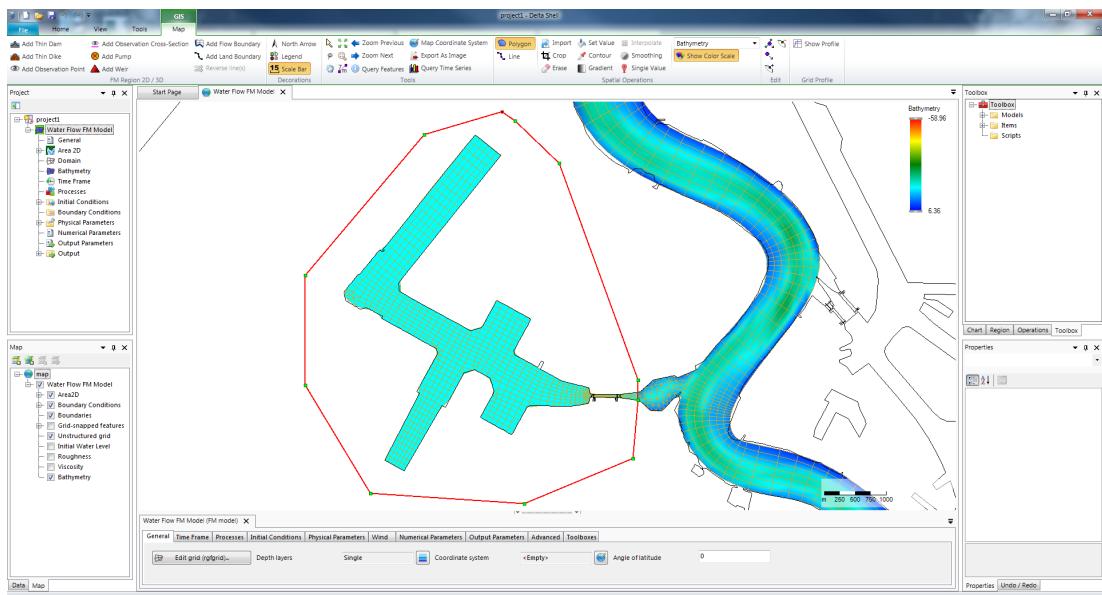


Figure 18.15: Interpolated bottom levels values at the grid.

To save the network and close the model afterwards within Delta Shell, the following should be done.

N.B. The network file will contain the grid together with the bathymetry data in the nodes.

- ◊ To save the network: In the project bar, rightmouseclick on *Domain* → *Export* → Save as <your_outcome_tut04_net.nc> in folder *tutorial04_your_outcome*. The grid has now been saved.
- ◊ To close the model: In the project bar, rightmouseclick on *Water Flow FM Model* → *Delete* → *OK*. The current D-Flow FM model has now been closed.

18.6 Tutorial 5: Imposing boundary conditions

Along both the sea boundary and the Scheldt river boundary, appropriate boundary conditions have to be imposed. The boundary conditions can be prescribed in many ways, for instance as waterlevels, velocities (also tangential and normal), discharge and Riemann. For the sake of simplicity, we are going to impose straightforward boundary conditions for the Westerscheldt:

- ◊ at the sea: a periodic waterlevel boundary, described as a harmonic signal with a mean 0.5 m+NAP, an amplitude 2.0 m and a frequency of 28,984 degrees/hour,
- ◊ at the river: a constant discharge boundary, described by a constant 2500 cubic meters per second.

First, lets start a new D-Flow FM model and import the network and landboundaries:

- 1 Choose *New Model* → *D-Flow FM model* → *OK*.
- 2 Network file: rightmouseclick on *Domain* (see [Figure 18.12](#)) → *Import*: the available network file <tutorial04_net.nc> from the directory *tutorial05*,
- 3 Landboundary file: *Area 2D* (see [Figure 18.12](#)) → Rightmouseclick on *Land boundaries* → *Import*: landboundary file <tutorial05.ldb>,

The way to define boundary conditions follows this strategy: first, draw a polyline along the boundary – second, fill a file with essential information – third, link the boundary files with the model setup. The boundary conditions can be imposed as follows:

- 1 Choose *Add Flow Boundary* (top of screen, see [Figure 18.16](#) and insert a polyline along the sea boundary. To close the polyline, doubleclick. In the Project bar to the left, under *Boundary Conditions*, this boundary has now been added. [Figure 18.16](#) shows how this will look like.

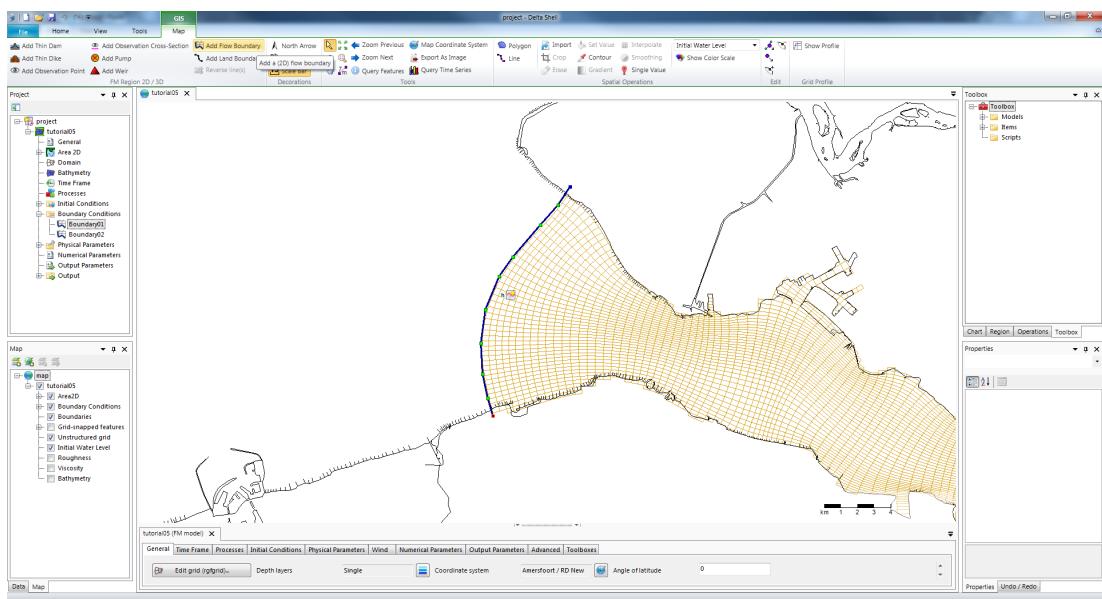


Figure 18.16: Line along the sea boundary.

Tip: to remove a wrongly placed point in the polyline, use *Remove point from geometry* (top of screen, above *Edit*, see [Figure 18.13](#)). It is also possible to replace or add a point.

- 2 Draw another polyline at the river entrance south of Antwerp.
- 3 To impose boundary conditions at the seaboundary, double click at *Boundary01* → Choose *Waterlevel* and press + → Choose *Forcing type: Astronomical*.
- 4 Choose *Select components* → (at the lefthandside of the screen) Choose *A0* and *M2* and press *OK* → Select *All support points*. All points along the polyline will now have components *A0* and *M2* defined.
- 5 Fill in the amplitudes of the components in the table, see [Figure 18.17](#). Do this for all support points.

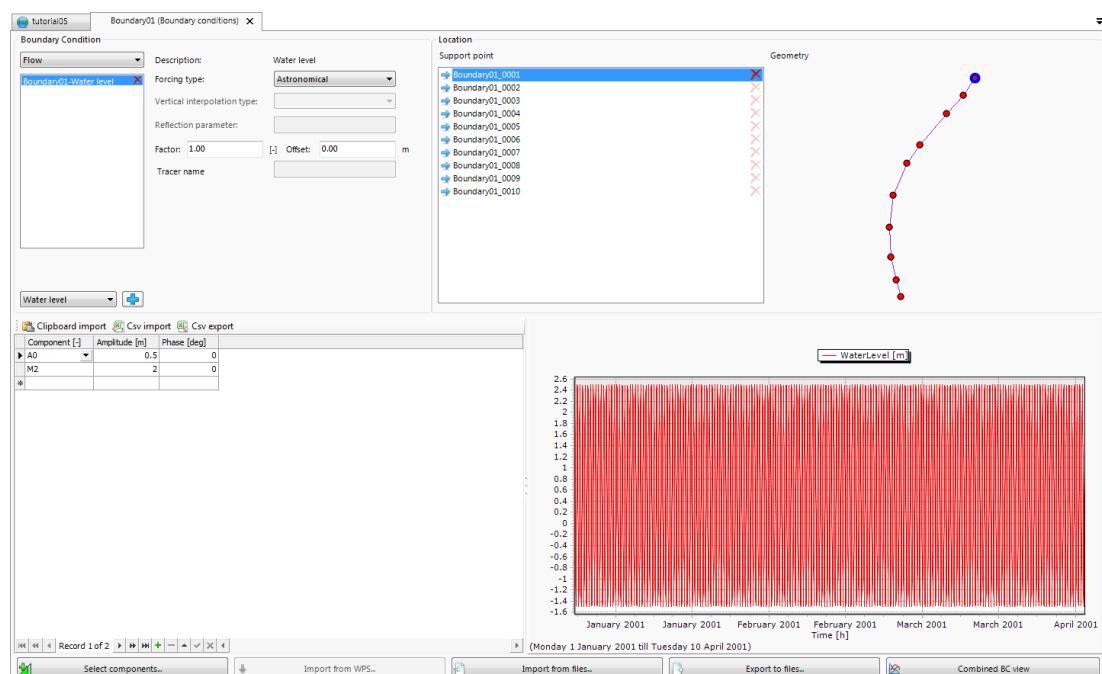


Figure 18.17: Boundary conditions seaside.

We have now prescribed an astronomical waterlevel signal, consisting of two components. The first component has an amplitude equal to 0.5 m with a frequency of 0 degrees/hour, i.e. a constant value of 0.5 m+NAP. The second component has an amplitude of 2.0 m with a frequency of 28,984 degrees/hour. Basically, the signal $h(t) = 2.0 \cdot \cos(28.984 \cdot t) + 0.5$ has been prescribed now, with h in meters w.r.t. the reference level (in case: NAP) and t in hours.

- 6 To impose boundary conditions at the riverboundary, double click at *Boundary02* → Choose *Discharge* and press + → Choose *Forcing type: Time Series*.
- 7 To impose a contant discharge of 2500 cubic meters per second, fill in the table like [Figure 18.18](#). Make sure you insert the same times!

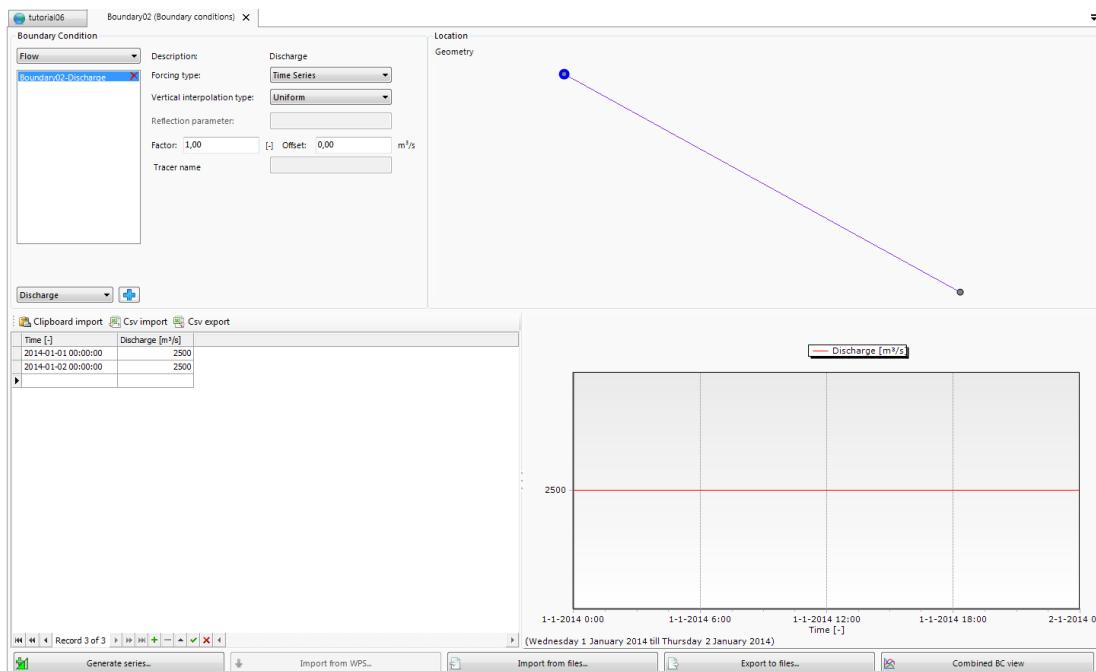


Figure 18.18: Boundary condition riverside.

To be able to load this D-Flow FM model in the future, it is necessary to save the model:

- ◊ In the project tree, rightclick on *Water Flow FM Model* → *Rename*: tutorial05.
- ◊ Save the model in *tutorial05_your_outcome*: In the project tree, rightclick on *tutorial05* → Choose *Export* → Save as <your_outcome_tutl05.mdu>.

The model can now be loaded in a next tutorial, without inserting all network files or land & flow boundaries individually again.

- ◊ Now close the D-Flow FM model (In the project bar, rightmouseclick on *tutorial05* → *Delete* → *OK*).

18.7 Tutorial 6: Defining output locations

Often, one would like to monitor computational outcomes at certain specific cross-sections or locations (i.e. 'observation points').

First:

- ◊ Import the D-Flow FM Model from the folder *tutorial06*, by choosing (in the top ribbon) *Home* → *Import* → *D-Flow FM Model* → *OK* → Choose <*tutorial06.mdu*> and *Open*.

Inserting cross-sections and observation points is straightforward. The following actions are necessary.

- 1 Now insert the cross-sections first. Choose *Add Observation Cross-Section* (see [Figure 18.19](#) on the upperleft). Draw as many cross-sections as you like. Separate each cross-section by doubleclicking the left mouse button. To view how many cross sections have been made, double click on *Area 2D* → *Observation cross-sections*, see [Figure 18.19](#).

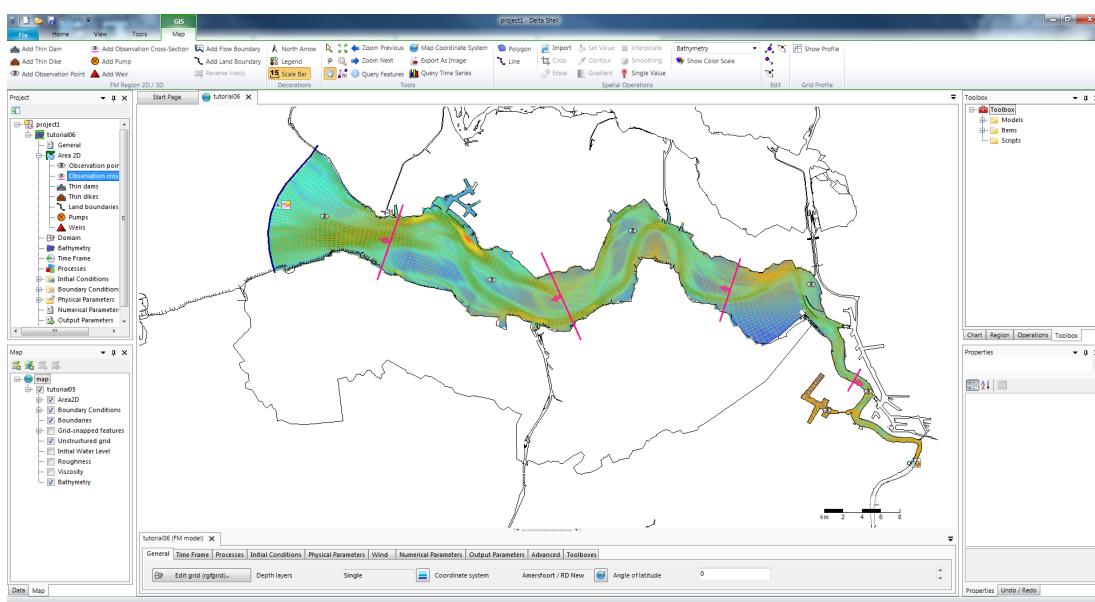


Figure 18.19: Overview cross sections and observation points.

- 2 Then the observation points. Choose *Add Observation Point*. Draw as many observation points as you like. To view how many cross sections have been made, double click on *Area 2D* → *Observation points*.
- 3 Save the current FM model in the folder *tutorial06_your_outcome* (In the project tree, right-mouseclick on *tutorial06* → Choose *Export* → Save as <*your_outcome_tut06.mdu*>). The observation points and cross sections have now been saved in the new .mdu file.
Tip: It is also possible to save only the observations points for instance to a certain location. To try this, rightclick on *Observation points* → *Export* → *Observation points to .xyn file*. Then, choose a filename and a location.
- 4 Now close the current FM model (In the project bar, rightmouseclick on *tutorial06* → *Delete* → *OK*).

18.8 Tutorial 7: Defining computational parameters

Before we can run the model, first of all the computational parameters need to be set, e.g. the Time Frame, Initial Conditions and the Output Parameters. These can be specified in the grey box at the bottom of Delta Shell. All other parameters are set by default, and will not be taken into account in this tutorial.

First:

- ◊ Import the D-Flow FM Model from the folder *tutorial07*, by choosing *Home* → *Import* → *D-Flow FM Model* → *OK* → Choose <*tutorial07.mdu*> and *Open*.

The following parameters needs to be selected: We will start with the Time Frame, see [Figure 18.20](#).

- 1 At the Time Frame section: fill in the parameters of [Figure 18.20](#).

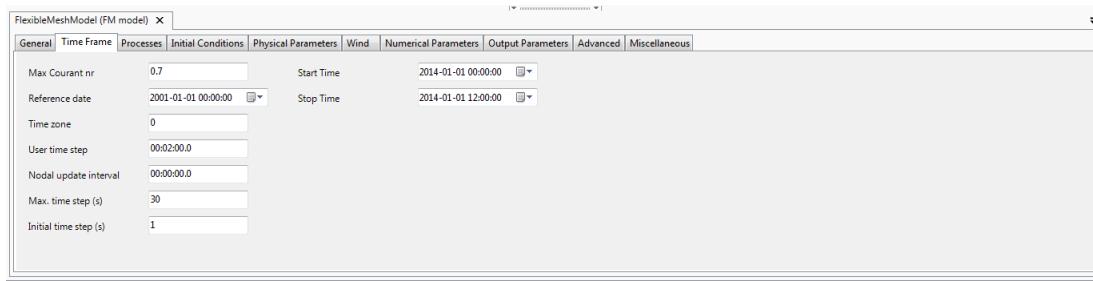


Figure 18.20: The time frame of the simulation.

2 At the Processes section: cross Tide generating forces, see [Figure 18.21](#)

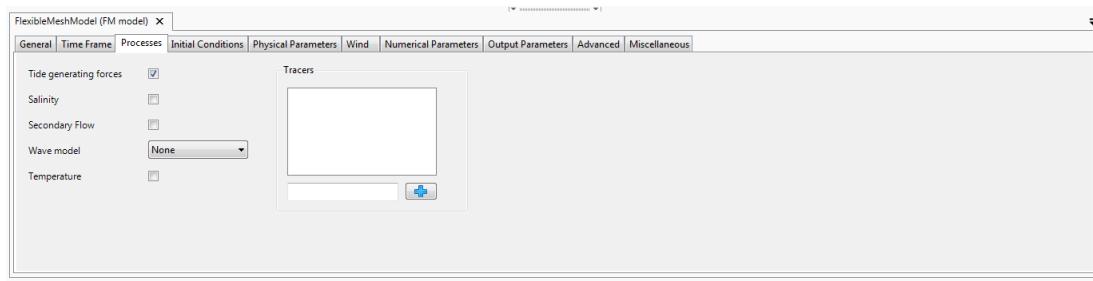


Figure 18.21: Relevant processes incorporated within the simulation.

3 At the Initial Conditions section: fill in 0,5 m for the Initial water level, see [Figure 18.22](#).

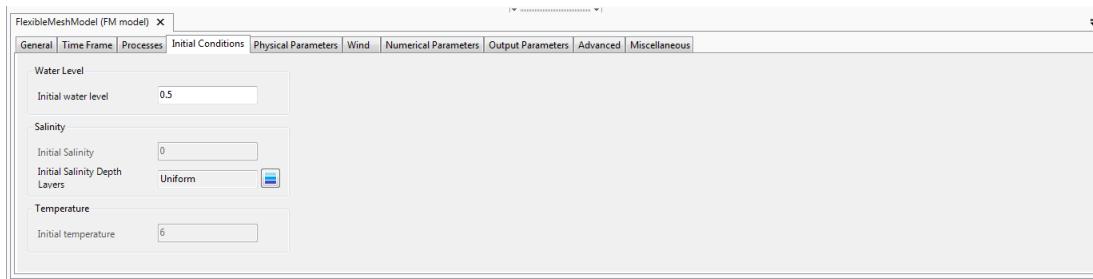


Figure 18.22: Imposed initial conditions for the simulation.

4 At the Output Parameters section: fill in the parameters of [Figure 18.23](#).

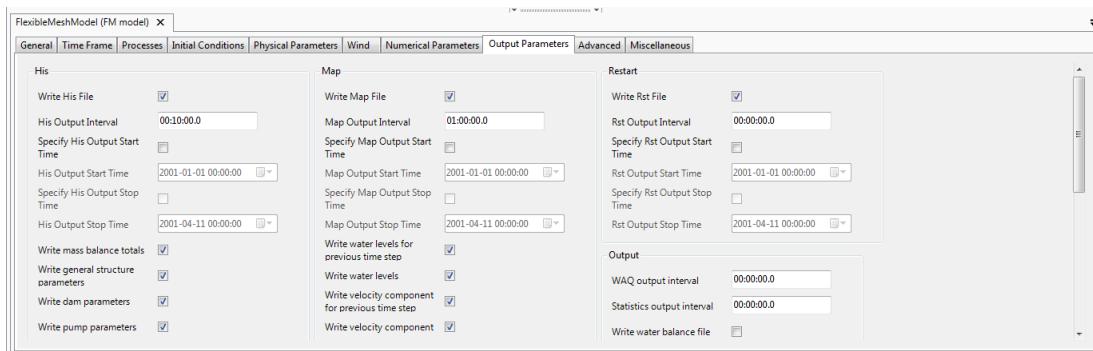


Figure 18.23: Optional output parameters for the computation.

As an explanation, the computation will deliver his-files and map-files:

- ◊ In his-files, timeseries are stored at the cross-sections and observation locations, at a frequency specified via the parameter His Output Interval.
- ◊ The map-files collect data over the entire domain, at a frequency specified via the parameter Map Output Interval. Be aware that these periods are clipped by the parameter User time step, which has been specified under Time Frame. That means, if User time step > His Output Interval, then the period with which his-files are written is given by User time step.
- ◊ Restart files could be written by crossing Write Rst File. The period with which these files are written can then be specified at Rst Output Interval. This period is not clipped by User time step. To start a computation with a restart file is not part of this tutorial though.

Save the current D-Flow FM model in the folder *tutorial07_your_outcome* (In the project tree, rightmouseclick on *tutorial07* → Choose *Export* → Save as <*your_outcome_tut07.mdu*>). The computational parameters have now been saved in the mdu file.

- ◊ Now close the current FM model (In the project bar, rightmouseclick on *tutorial07* → *Delete* → *OK*).

18.9 Tutorial 8: Running a model simulation

Up till now, we have saved the D-Flow FM model by means of a mdu file. For running computations, it is necessary to save the entire project.

First:

- ◊ Import the D-Flow FM Model from the folder *tutorial08/mdu*, by choosing *Home* → *Import* → *D-Flow FM Model* → *OK* → Choose <*tutorial08.mdu*> and *Open*.

For saving the project, the following should be done:

- ◊ Choose *File* → *Save As* *project.dsproj* in the folder *tutorial08_your_outcome* and click *Save*. The project has now been saved in a folder called *project.dsproj_data* and a project file <*project.dsproj*> has been created (See [Figure 18.24](#)). Within this folder you will find all ASCII input files of the model, output files of the model (when it will be run) and, if applicable, zip folders containing the restart files.

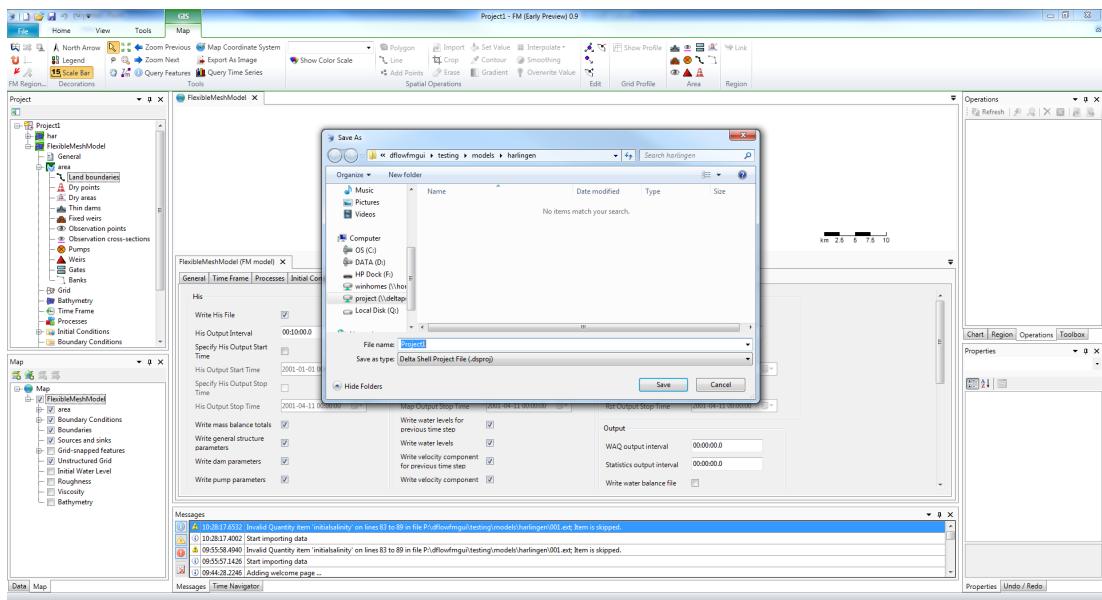


Figure 18.24: Menu that appears if one would like to save the project.

For starting a computation, the following needs to be done:

- ◊ In the project tree, select the model you want to run by means of clicking the first attribute of the desired model (which has been named *tutorial08*, see [Figure 18.25](#)) → Choose *Home* → *Run Current*. The model will start running now.

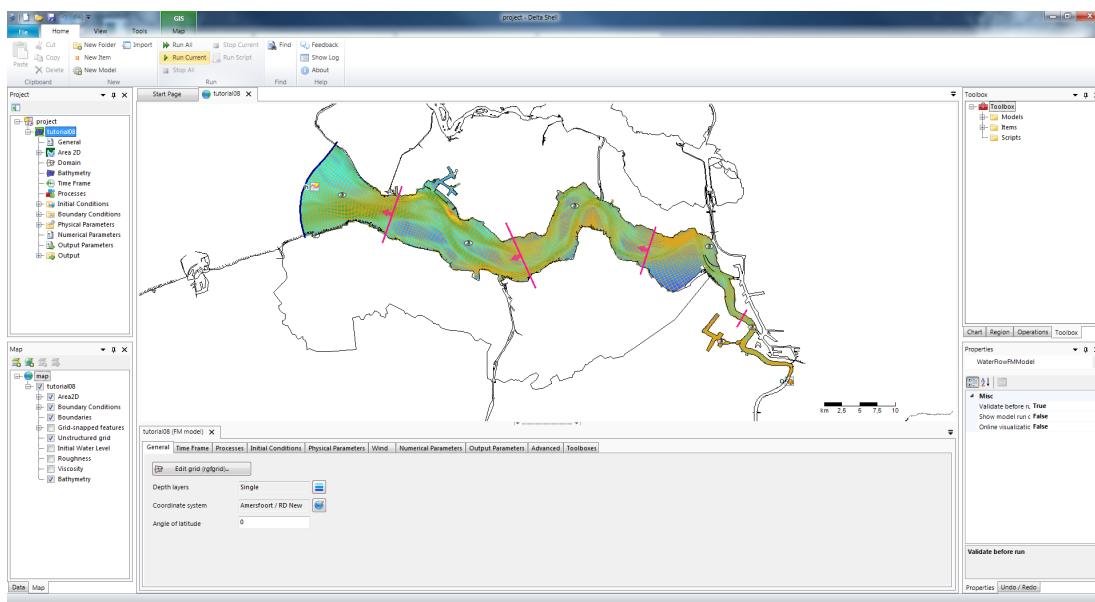


Figure 18.25: View of Delta Shell when running a model.

The output of the run is being stored in the folder *DFM_OUTPUT_tutorial08*, within the folder *tutorial08_output*. Now:

- ◊ Save the project with generated output in the folder *tutorial08_your_outcome* (*File* → *Save As project.dsproj*).
- ◊ Now close the current FM model (In the project bar, rightmouseclick on *tutorial08* → *Delete* → *OK*).

18.10 Tutorial 9: Viewing the output of a model simulation

The output of the model can be observed within Delta Shell.

First, load the project (which has the output stored within):

- ◊ Choose *File* → *Open*. Select the file <project.dsproj> from the folder *tutorial09* and choose *Open*. The project will now open.
- ◊ Doubleclick on *Domain*. The grid will now appear.

To view the output of the map files:

- 1 Open the Time Navigator, by choosing *View* → *Time Navigator*. The Time Navigator will appear at the bottom of Delta Shell.
- 2 Choose in the map tree *Add New Wms Layer* (that is the second icon just above the map tree) and choose <http://openstreetmap.org>. See [Figure 18.26](#). Now, openstreet maps can be observed in the model too (this might take some time).

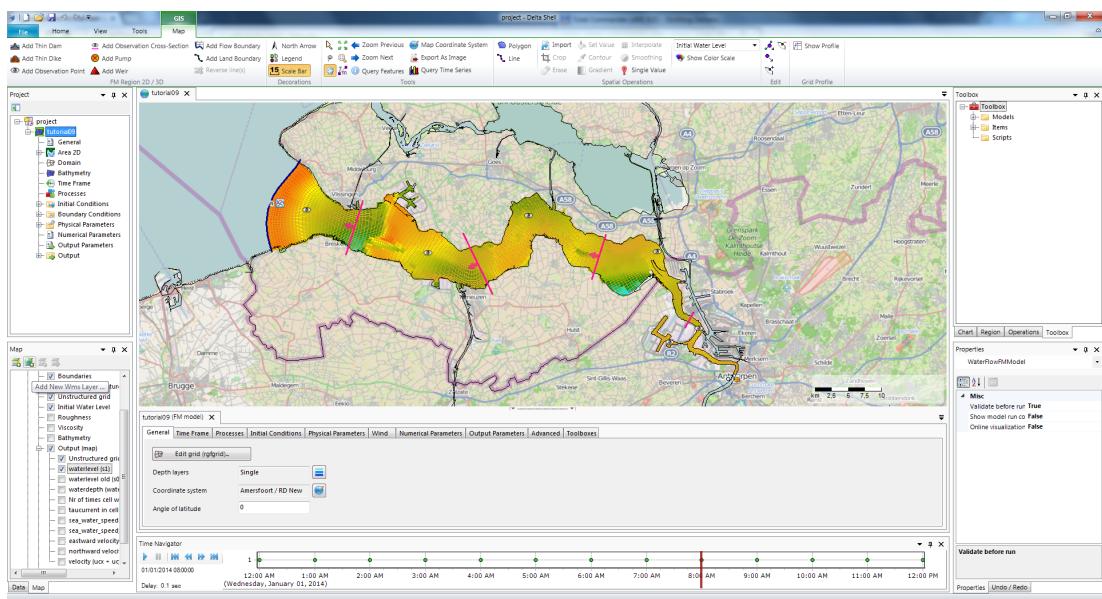


Figure 18.26: View of Delta Shell in combination with Openstreet maps.

- 3 Open *Output* in the map tree to the left. Cross *waterlevel(s1)* to observe waterlevels for instance, see [Figure 18.26](#).
- 4 Click the play button in the Time Navigator to see the waterlevels change in time. Choose another parameter, e.g. *waterdepth*, to observe afterwards too.

To view the output of the his files (for the observation points and cross-sections):

- 1 Click with leftmousebutton on an observation point (while being in *select mode!* Your mouse looks like an arrow in this mode).
- 2 Choose at the top part of the screen *Query Time Series*, see [Figure 18.27](#) (option available within the Map-ribbon).

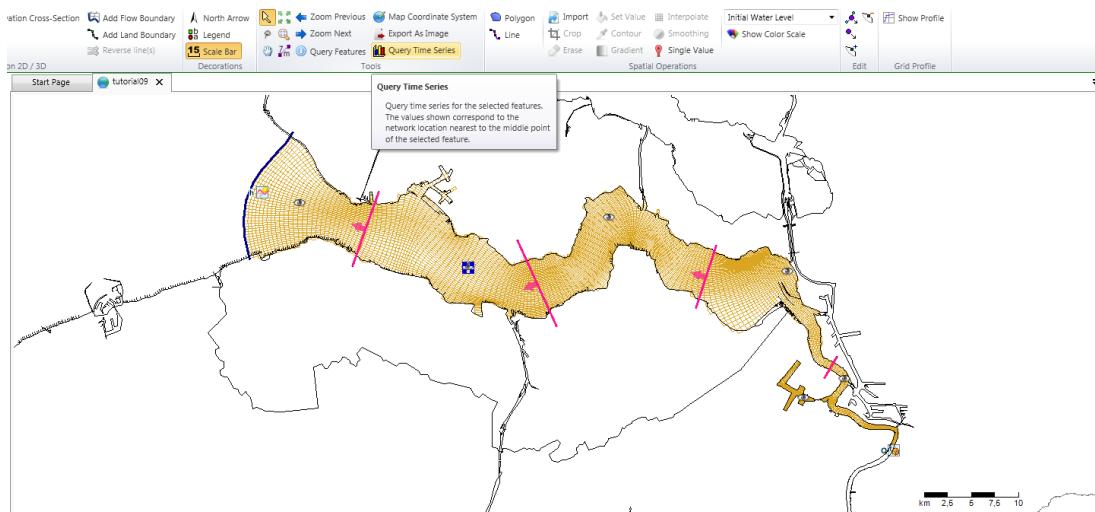


Figure 18.27: View of Delta Shell in combination with Openstreet maps, available to select a location for timeseries in.

- 3 Choose *Water level (waterlevel)* → *OK*. You can now observe the waterlevels in the observation point you selected. You can choose other parameters as well to observe. Also cross-sections can be observed this way.

19 Calibration and data assimilation



20 Calibration and data assimilation

20.1 Introduction

A flow model in D-Flow FM will generally benefit from parameter *calibration* to closer match observation data. When the model runs in an operational system *data assimilation* can be used to into the running model. For the automatic calibration and data assimilation, the open source toolbox **OpenDA** is available.

OpenDA basically provides three types of building blocks: an optimisation algorithm that performs the calibration or data assimilation, communication routines for passing information between OpenDA and D-Flow FM, and methods for handling observation data. The communication between OpenDA and D-Flow FM is realised using a Black Box approach. A number of wrapper objects (so called dataObjects) are available for reading and writing D-Flow FM input and output files.

This chapter contains a description of how the OpenDA toolbox could be deployed to apply calibration and data assimilation. The OpenDA tools can run D-Flow FM models and analyze the model results. A more extensive design description of the D-Flow FM wrappers for OpenDA can be found in the D-Flow FM Technical Reference [D-Flow FM Technical Reference Manual \(2015\)](#). More information on OpenDA can be found on the website <http://www.opendata.org>.

General information on the installation of OpenDA to get started is provided in [section 20.2](#). [section 20.4](#) gives an overview of the black box wrapper for D-Flow FM. [section 20.4](#) describes the OpenDA configuration and the related D-Flow FM files. The generation of noise and how this noise is added to forcings and boundaries is given in [section 20.5](#). Examples case for calibration and data assimilation using the ensemble Kalman filter are described in [section 20.6](#).



20.2 Getting started with OpenDA

The required D-Flow FM wrapper is enclosed in the official OpenDA release since version 2.2.2. The following three elements are needed for a calibration or data assimilation run with D-Flow FM:

- 1 The D-Flow FM Command Line Interface (CLI) installation. All OpenDA algorithms start D-Flow FM with a shell script `start_dflowfm.sh` or a batch script `start_dflowfm.bat`. Both scripts assume that D-Flow FM executable is available on the search path (i.e. it should be executable from the command line). If this is not the case change the environment variable PATH to include the installation path of the D-Flow FM executable.
- 2 An OpenDA installation including the OpenDA core, the D-Flow FM specific wrapper code and examples.
- 3 A Java Runtime Environment (JRE) version 7 (or higher) is needed to run OpenDA (2.2.2). OpenDA can use one of the system installed JREs or alternatively an JRE can be installed directly in the OpenDA directory at the same level as the `<bin>` directory.

For Linux it is required to run `source settings_local.sh` to setup OpenDA. The OpenDA GUI can be started from the `<bin>` directory using `oda_run_gui.bat` (Windows) or `oda_run.sh` with the `-gui` command line option (Linux).

20.3 The OpenDA black box model wrapper for D-Flow FM

For a D-Flow FM model to function within the OpenDA toolbox we need to establish two things: 1) the control to propagate the model over time and 2) access to the model state, physical parameters, boundary conditions and external forcings. In the black box approach the standard D-Flow FM command line executable is used to propagate the model over time. Access to the model state, physical parameters, boundary conditions and external forcings is obtained by reading from and writing to the D-Flow FM input and output files. Inside OpenDA all data is available in the form of exchange items which all have an unique identifier.

Calibration and data assimilation typically need multiple model evaluations with altered parameters, forcings, boundary conditions or the initial model state. In the black box approach this is achieved by creating multiple work directories containing altered model input files and starting the D-Flow FM executable in each work directory. D-Flow FM model results are then read from the work directories and compared to observation data.

For calibration this is an iterative process. Results from model evaluations $1 \dots n$ are required to obtain a better estimate for parameter values, which are then evaluated in run $n + 1$.

In case of an ensemble Kalman filter (EnKF) run, the D-Flow FM computations (one run for each ensemble member) is stopped each time an observation is available, the input files for each ensemble member are modified according to the ensemble Kalman filter algorithm, after which the D-Flow FM run is restarted.

Next to the D-Flow FM model configuration OpenDA has its own configuration for selecting the algorithm, observations and interfacing with the D-Flow FM input and output files.

20.4 OpenDA configuration

20.4.1 Main configuration file and the directory structure

The OpenDA main configuration file has the .oda extension. All OpenDA configuration files use the xml format. It is advised to use a schema aware xml editor, when making changes to the OpenDA configuration. These editors provide direct access to the documentation that is stored in the schema and can validate the correctness of the xml files.

All other xml config file names and directories are configurable. However, there is a commonly used directory layout and naming convention. All the examples are configured using this convention. For example, the directory structure for the simple_waal_kalman example is given in [Table 20.1](#).

All the work directories are available in the <stochModel> directory, after performing a run with OpenDA they contain the D-Flow FM results.

It is a good practice to name the main configuration file corresponding to the algorithms executed by OpenDA. The following files are used in the provided examples:

- ◊ <Simulation.oda>: performs a regular D-Flow FM simulation run, only the executable is started by OpenDA. This algorithm is useful to check the configuration.
- ◊ <Dud.oda>: Dud (Doesn't Use Derivative) is one of the optimisation algorithms available for calibration purposes.
- ◊ <SequentialSimulation.oda>: performs a D-Flow FM simulation run through which the executable is stopped and restarted by OpenDA at the moments for which observed data are available.
- ◊ <Enkf.oda>: performs an ensemble Kalman filtering.

```

<simple_waal_kalman>
  <algorithm>..... calibration method or data-assimilation algorithm
    <enkfAlgorithm.xml> ..... algorithm specific configuration
    ...
  <stochModel> ..... model and its uncertainty description
    <bin>..... bat and .sh scripts for calling D-Flow FM
    <input_dflowfm> ..... D-Flow FM template configuration
    ...
    <work0/>..... work directory for propagated mean
    <work1/> ..... work directory for first ensemble member
    ...
    <dflowfmModel.xml> ..... exchange items configuration
    <dflowfmStochModel.xml>..... predictor, state and parameter
    <dflowfmWrapper.xml> ..... actions and data objects configuration
    ...
  <stochObserver> ..... observations and uncertainty
    <noosObservations.xml>..... observations and uncertainty
    <waterlevel_Obs01.noos> ..... raw observation file
    ...
  <Enkf.oda> ..... main configuration file
  ...

```

Table 20.1: Directory structure of the OpenDA Ensemble Kalman filtering configuration for the simple Waal D-Flow FM model.

The main configuration for an OpenDA application has three mandatory components, which make up an OpenDA application: stochModelFactory, stochObserver and algorithm. Each component is configured by specifying its `className` attribute, `workingDirectory` and `configFile/configString`. There are optional components to enable writing OpenDA results, to define OpenDA restart input and output files and to enable timings. The `resultwriter` is typically useful for writing stochastic properties that are only available to OpenDA and not in D-Flow FM.

The main configuration file `<Enkf.oda>` for the `simple_waal` example:

```

<?xml version="1.0" encoding="UTF-8"?>
<openDaApplication
  xmlns="http://www.opendata.org"
  xmlns:xsi="http://www.w3.org/2001/XMLSchema-instance"
  xsi:schemaLocation
    ="http://www.opendata.org http://schemas.opendata.org/openDaApplication.xsd">
  <stochObserver className="org.opendata.observers.NoosTimeSeriesStochObserver">
    <workingDirectory>./stochObserver</workingDirectory>
    <configFile>noosObservations.xml</configFile>
  </stochObserver>
  <stochModelFactory className="org.opendata.blackbox.wrapper.BBStochModelFactory">
    <workingDirectory>./stochModel</workingDirectory>
    <configFile>dflowfmStochModel.xml</configFile>
  </stochModelFactory>
  <algorithm className="org.opendata.algorithms.kalmanFilter.EnKF">
    <workingDirectory>./algorithm</workingDirectory>
    <configString>EnkfAlgorithm.xml</configString>
  </algorithm>
  <resultWriter className="org.opendata.resultwriters.MatlabResultWriter">
    <workingDirectory>.</workingDirectory>
    <configFile>Enkf_results.m</configFile>
    <selection>
      <resultItem id="pred_f"/>
      <resultItem id="pred_f_0"/>
    
```

```

<resultItem id="pred_f_1"/>
<resultItem id="pred_f_std"/>
<resultItem id="pred_f_central"/>
<resultItem id="pred_a_linear"/>
<resultItem id="analysis_time"/>
<resultItem id="obs"/>
</selection>
</resultWriter>
</openDaApplication>

```

Note that the directory layout in this section is created by setting the `workingDirectory` elements in `stochModelFactory`, `stochObserver` and `algorithm` parts of the configuration. Each of these components have their own configuration, which are described in the following sections.

20.4.2 The algorithm configuration

All provided methods for calibration and data assimilation are configurable through an xml file. The convention is to include the algorithm name in the file name e.g `<EnkfAlgorithm.xml>`. For data assimilation algorithms the configuration typically specifies the ensemble size and the option to use stochastic parameters, forcing and initialisation. For calibration algorithms the configuration typically contains definition of the cost function and tolerances and stopping criteria. For a list of algorithms and their configuration options see the general OpenDA documentation.

20.4.3 The stochObserver configuration

The access to observations is standardized in OpenDA using a `stochObserver` object. The configuration and the observation data are placed in the `<stochObserver>` directory. OpenDA contains a number of different `stochObserver` objects that can handle different types of data files. In this manual, we discuss the `NoosTimeSeriesStochObserver` and the more generic `IoObjectStochObserver`. For a more complete list of available `stochObservers` see the OpenDA web site.

20.4.3.1 NoosTimeSeriesStochObserver

The NOOS file format is used to store time series. The format is created for use by the members of the North West European Shelf Operational Oceanographic System <http://www.noos.cc/>. An example of (a part of) a NOOS file is:

```

#-----#
#-----#
# Location : station01
# Position : (0.0,0.0)
# Source : twin experiment DFlowFM
# Unit : waterlevel
# Analyse time: null
# Timezone : null
#-----#
199101010000  1.0000
199101010100  0.8944
199101010200  0.6862
199101010300  0.5956
199101010400  0.3794
199101010500  0.1372
199101010600  -0.1300
199101010700  -0.3044
199101010800  -0.3963
199101010900  -0.3739

```

199101011000	-0.1930
...	

The file contains a header with meta data specifying among others the location name (Location) and the quantity (Unit). The data is written in two columns, the first gives the time of the observation using the 'YYYYMMDDhhmm' format and the second column gives the measured value.

The file <noosObserver.xml> defines a number of time series, each coupled to a NOOS file containing the measurements.

```
<?xml version="1.0" encoding="UTF-8"?>
<noosObserver
    xmlns="http://www.opendata.org"
    xmlns:xsi="http://www.w3.org/2001/XMLSchema-instance"
    xsi:schemaLocation
        ="http://www.opendata.org http://schemas.opendata.org/schemas/noosObservations.xsd">
    <timeSeries status="use" standardDeviation="0.05">
        den_helder_waterlevel_astro.noos
    </timeSeries>
    <timeSeries status="use" standardDeviation="0.05">
        aberdeen_waterlevel_astro.noos
    </timeSeries>
</noosObserver>
```

OpenDA creates an exchange item for each observation time series. The default exchange item id (identifier) is created using the location (Location) and the quantity (Unit). The standard deviation (measurement error) is specified with standDeviation attribute.

20.4.3.2 IoObjectStochObserver

This observer uses a dataObject for handling the file IO. All exchange items that are provided by dataObject can be used by the observer. For instance the NetcdfDataObject can read and write to NetCDF files, and has an exchange item for each variable in the NetCDF file. The lake_kalman example uses this approach to use the *.his file from a D-Flow FM run as synthetic observations for a ensemble Kalman filter run. The <dflowfmStochObsConfig.xml> file reads:

```
<?xml version="1.0" encoding="UTF-8"?>
<ioObjectStochObserver
    xmlns="http://www.opendata.org"
    xmlns:xsi="http://www.w3.org/2001/XMLSchema-instance"
    xsi:schemaLocation
        ="http://www.opendata.org http://schemas.opendata.org/openDaStochObserver.xsd">
    <uncertaintyModule
        workingDirectory=". "
        className="org.opendata.uncertainties.UncertaintyEngine">
        <arg>stochObsUncertainties.xml</arg>
    </uncertaintyModule>
    <ioObject
        workingDirectory=". "
        className="org.opendata.exchange.dataobjects.NetcdfDataObject">
        <fileName>lake2d_his.nc</fileName>
    </ioObject>
</ioObjectStochObserver>
```

In the configuration of UncertaintyEngine OpenDA object, a selection of these exchange items id's is made and a standard deviation is specified.

```
<?xml version="1.0" encoding="UTF-8"?>
<uncertainties
    xmlns="http://www.wldelft.nl"
```

```
xmlns:xsi="http://www.w3.org/2001/XMLSchema-instance"
xsi:schemaLocation
  ="http://www.wldelft.nl http://schemas.opendata.org/uncertainties.xsd"
version="1.0">
<uncertaintyType>ProbabilityDistributionFunction</uncertaintyType>
<probabilityDistributionFunction id="S1.waterlevel" isActive="true">
  <normal mean="0" stdv="0.05" stdvIsFactor="false"/>
</probabilityDistributionFunction>
<probabilityDistributionFunction id="S2.waterlevel" isActive="true">
  <normal mean="0" stdv="0.05" stdvIsFactor="false"/>
</probabilityDistributionFunction>
<probabilityDistributionFunction id="S3.waterlevel" isActive="true">
  <normal mean="0" stdv="0.05" stdvIsFactor="false"/>
</probabilityDistributionFunction>
<probabilityDistributionFunction id="S4.waterlevel" isActive="true">
  <normal mean="0" stdv="0.05" stdvIsFactor="false"/>
</probabilityDistributionFunction>
</uncertainties>
```

20.4.4 The stochModel configuration

The stochModel configuration usually consists of three <.xml> files in the <stochModel> directory. These are called:

- ◊ <dflowfmWrapper.xml>: This file specifies the actions to perform in order to run a D-Flow FM simulation and list the D-Flow FM input and output files that can be used to let OpenDA interact with the model. For each file the dataObject is specified which is used for handling the specified file.
- ◊ <dflowfmModel.xml>: This file contains a list of the exchange items which are provided by the configured dataObject. The model time information is constructed by the time-InfoExchangeItems element. It also contains a number of alias values which can be used in three stochModel configuration files.
- ◊ <dflowfmStochModel.xml>: In this file defines the predictor, the selection of observations which are compared to the model results. For calibration this file also specifies which parameters can be changed. For data assimilation this files contains the definition of the model state and the noise (uncertainty) specification for the boundaries and forcings.

20.4.5 D-Flow FM files and the OpenDA dataObjects configuration

The dataObjects for reading and writing provide so-called exchange items that allow OpenDA to manipulate specific parts of the files of D-Flow FM. The D-Flow FM files that can be manipulated by OpenDA and the corresponding OpenDA class names are given in [Table 20.2](#).

Table 20.2: D-Flow FM files that can be manipulated and the corresponding OpenDA class names to be used in the `dflowfmWrapper.xml` file.

D-Flow FM filetype	OpenDA dataObject & exchange items
<.mdu>	org.opendata.model_dflowfm.DFlowFMTIMEINFO IDs: start_time, end_time
<*.amu>	org.opendata.model_dflowfm.DFlowFMMeteoFile ID: x_wind
<*.amv>	org.opendata.model_dflowfm.DFlowFMMeteoFile IDs: y_wind
<*.amp>	org.opendata.model_dflowfm.DFlowFMMeteoFile ID: air_pressure
<*.tim>	org.opendata.model_dflowfm.DFlowFMTIMESERIESDataObject IDs: BOUNDARY_ID.#:QUANTITY
<*.xyz>	org.opendata.model_dflowfm.DFlowFMXYZFile IDs: FILENAME_#
<*_his.nc>	org.opendata.exchange.dataobjects.NetcdfDataObject IDs: STATION_ID.VARIABLE_NAME
<*_map.nc>	org.opendata.model_dflowfm.DFlowFMRSTARTFILEWrapper IDs: VARIABLE_NAME

All the dataObjects and their configuration are described in the following sections.

20.4.5.1 Start and end time in the model definition file (.mdu)

The start and end time are set in `<.mdu>`-file by OpenDA using the `start_time` and `end_time` exchange items. These are provided by the `DFlowFMTIMEINFO` data object.

D-Flow FM reference	Exchange Item Id	Remark
TStart	start_time	RefDate and Tunit needed for interpretation
TStop	end_time	RefDate and Tunit needed for interpretation

20.4.5.2 External forcings (.xyz)

All D-Flow FM external forcings are specified via the `<.ext>` forcings file. Here a spatial forcing can be defined by using a `.xyz`-file (e.g. the bed friction coefficients). For instance the file `<nikuradse.xyz>` contains:

```
x1 y1 0.9994357525438934
x2 y2 0.9994357525438934
x3 y3 2.0021214673505600
x4 y4 2.0021214673505600
x5 y5 2.0021214673505600
x6 y6 2.0021214673505600
```

When performing calibration of a spatial field it is often required to group points in a select number of regions. The calibration then does not change the individual values but applies a factor to all values in the group. The best approach is to create a file with multipliers (e.g. `<friction_multiplier.xyz>`) which are initially all equal to one.

```
x1 y1 1.0
x2 y2 1.0
x3 y3 1.0
x4 y4 1.0
x5 y5 1.0
x6 y6 1.0
```

The multiplication (or addition) with the values in `nikuradse.xyz` should be configured in the `*.ext` file.

Group from keywords in file

One option to construct groups is to use keywords directly in the `<friction_multiplier.xyz>` file:

```
x1 y1 1.0 #friction_3
x2 y2 1.0 #friction_3
x3 y3 1.0 #friction_1
x4 y4 1.0 #friction_1
x5 y5 1.0 #friction_4
x6 y6 1.0 #friction_4
```

In the OpenDA wrapper config the dataObject is than configured as:

```
<ioObject className="org.opendata.model_dflowfm.DFlowFMXyzFile">
  <file>friction_multiplier.xyz</file>
  <id>frictionCoeffFile</id>
  <arg>idsFromKeywordsInFile</arg>
</ioObject>
```

This will create exchange items with identifier `friction_3`, `friction_1` and `friction_4`.

Group from template file

An other options is to use a template file (`<friction_multiplier_template.xyz>`) with exactly the same (x,y) coordinates as in (`<friction_multiplier.xyz>`). The third column is used to define groups by using these values as group numbers:

```
x1 y1 3.0
x2 y2 3.0
x3 y3 4.0
x4 y4 4.0
x5 y5 1.0
x6 y6 1.0
```

In the OpenDA wrapper config the dataObject must be configured as:

```
<ioObject className="org.opendata.model_dflowfm.DFlowFMXyzFile">
  <file>friction_multiplier.xyz</file>
  <id>frictionCoeffFile</id>
  <arg>idsFromTemplateFile=friction_multiplier_template.xyz</arg>
</ioObject>
```

This will create an exchange item for each group with identifier ‘FILE_BASENAME + _ + number from template file’, e.g. friction_multiplier_3, friction_multiplier_4 and friction_multiplier_1.

20.4.5.3 Boundary time series (.tim)

Boundary conditions are specified as a combination of a <.pli> file and one or more .tim files. The DFlowFMTimeseriesDataObject dataObject creates exchange items for all boundary conditions. It starts with reading the name of the external forcing file name from the <.mdu>-file (key ExtForceFile). The <.ext>-file contains formatted blocks, one for each forcing. Forcings are defined along polylines, given in .pli-files. A <.pli>-file is accompanied by a <.cmp>- or a (number of) <.tim>-file(s).

Noise can be added by means of an extra block in the .ext-file. As an example, noise is added to a boundary with a discharge imposed as:

```
QUANTITY =dischargebnd
FILENAME =sw_east_dis.pli
FILETYPE =9
METHOD =3
OPERAND =0

QUANTITY =dischargebnd
FILENAME =sw_east_dis_noise.pli
FILETYPE =9
METHOD =3
OPERAND =+
```

The discharge is set by the first block (operand=0), the information in the <.pli>-files is identical and noise is added as a time series: the <_noise.pli> file is always accompanied by a (number of) <.tim> file(s). The location-information on the first line of the .pli-file combined with the quantity is used to construct the exchange item identifier: location.1.-dischargebnd. The numbering is used to discern between multiple <.tim>-files possibly linked to a single <.pli>-file.

20.4.5.4 Meteorological boundary conditions (.amu, .amv, .amp)

OpenDA can read and write to the D-Flow FM <.amu>, <.amv> and <.amp> files using the org.opendata.model_dflowfm.DFlowFMMeteoFile dataObject. These contain the *x*- and *y* components of the wind and the air pressure at the free surface on an equidistant grid. In a typical data assimilation use noise fields are added to the wind. For the this purpose OpenDA can generate a spatial noise field on an equidistant grid (see section 20.5). D-Flow FM can combine fields defined in files on different to single field on the computational grid.

20.4.5.5 Result time series (_his.n)

The <_his.nc>-file contains time series with D-Flow FM model results for a number of stations. The generic org.opendata.exchange.dataobjects.NetcdfDataObject is used for handling this type of files. The NetcdfDataObject expects a NetCDF-file that contains dimensions time and stations plus a variable station_id of type string and dimension stations. For each variable in this NetCDF-file with dimensions (time,stations) an exchange item is created, that can be referred to as station_id(i).variablename. For instance:

```
dimensions:
```

```

time = UNLIMITED ; // (2882 currently)
stations = 3 ;
station_name_len = 40 ;
variables:
char station_id(stations, station_name_len) ;
(containing strings Obs1, Obs2, Obs3)
double waterlevel(time, stations) ;
(containing the computed values of the waterlevel)

```

results in three exchange items (a 1D vector in this case) with identifiers: Obs1.waterlevel, Obs2.waterlevel and Obs3.waterlevel.

20.4.5.6 Restart file (_map.nc)

OpenDA provides the `org.opendata.model_dflowfm.DFlowFMRestartFileWrapper` for reading and writing the `<_map.nc>`-file. This file contains all information needed to restart a D-Flow FM computation. Not all variables are relevant for manipulation by OpenDA: variable names that start with 'time', 'NetLink', 'BndLink', 'FlowLink', 'NetElem', 'FlowElem', 'NetNode', 'wgs84' and 'projected_coordinate_system' are ignored. Variables of other types than float or double are also ignored. For all other variables an exchange item is created, where the name of the variable in the NetCDF is used as the exchange item id.

Note: for Kalman filtering it is essential that the model starts from the `<_map.nc>` file. It is not possible to specify an initial field by setting the `initialwaterlevel` or `initialsalinity` quantities in the `<*.ext>` file. If you want to set an initial field using these settings, make a custom D-Flow FM run where `TStop` is equal to `TStart` and use the created `<_map.nc>` file for the starting point of the Kalman filtering.

20.5 Generating noise

To add uncertainty to the external forcings and the boundary conditions, OpenDA has functionality for generating noise time series and spatial fields. The noise generation can be specified within the state definition in `<dflowfmStochModel.xml>`. For instance in the example `simple_waal_kalman`, a noise time series is created (ID `dischargenoise`), which is added to the inflow discharge (ID `eastboundary.1:dischargebnd`).

```

<?xml version="1.0" encoding="UTF-8"?>
<blackBoxStochModel
    xmlns:xsi="http://www.w3.org/2001/XMLSchema-instance"
    xsi:schemaLocation
        ="http://www.opendata.org http://schemas.opendata.org/blackBoxStochModelConfig.xsd"
    xmlns="http://www.opendata.org">
<modelConfig>
    <file>./dflowfmModel.xml</file>
</modelConfig>
<vectorSpecification>
    <state>
        <noiseModel
            id="boundaryNoiseModel"
            className="org.opendata.noiseModels.TimeSeriesNoiseModelFactory"
            workingDirectory=".">
            <configFile>BoundaryNoise.xml</configFile>
            <exchangeItems>
                <exchangeItem
                    id="dischargenoise"
                    operation="add"
                    modelExchangeItemId="eastboundary.1:dischargebnd"/>
            </exchangeItems>
        
```

```

</noiseModel>
<vector id="s1"/>
<vector id="unorm"/>
</state>
</vectorSpecification>
</blackBoxStochModel>
```

The config file <BoundaryNoise.xml> contains the details of the noise:

```

<?xml version="1.0" encoding="UTF-8"?>
<mapsNoiseModelConfig>
    <simulationTimespan timeFormat="dateTimeString">
        199208311200,199209010000,...,199212090000
    </simulationTimespan>
    <timeSeries
        id="dischargenoise"
        location="eastboundary"
        quantity="discharge"
        standardDeviation="0.2"
        timeCorrelationScale="6.0"
        timeCorrelationScaleUnit="hours"/>
</mapsNoiseModelConfig>
```

A noise time series is created (ID dischargenoise) with a correlation time of 6 hours and a standard deviation of 0.2. Note that currently OpenDA does not support the creation of D-Flow FM files, all files should exist in the <input_dflowfm>. In this case the boundary is defined in <sw_east_dis.pli>, but the noise is in a separate file <sw_east_dis_noise.pli>. Initially it contains zeros, but is filled with the noise values at run time. The time points in the file need to match the simulationTimespan in the OpenDA config. The total discharge at the boundary is constructed by D-Flow FM where the noise is added to original boundary values (see <simple_waal.ext>).

To generate a spatial correlation field the same approach can be used. In the lake_kalman example a noise field is added the wind. Instead of timeSeries a noiseItem is specified which contains the grid definition.

```

<?xml version="1.0" encoding="UTF-8"?>
<mapsNoiseModelConfig>
    <simulationTimespan timeFormat="dateTimeString">
        200106240000,200106240100,...,200106270000
    </simulationTimespan>
    <noiseItem id="2DNoise" quantity="wind-x" unit="m/s" height="10.0"
        standardDeviation="20.0"
        timeCorrelationScale="12.0" timeCorrelationScaleUnit="hours"
        initialValue="0.0"
        horizontalCorrelationScale="10" horizontalCorrelationScaleUnit="km">
        <grid type="cartesian" coordinates="XY">
            <x>0,3000,...,63000</x>
            <y>0,3000,...,63000</y>
        </grid>
    </noiseItem>
</mapsNoiseModelConfig>
```

The calculation of spatially correlated noise on a cartesian grid is quite fast as the x and y -direction are independent. The interpolation from an equidistant grid to the computational grid is performed within D-Flow FM. Again, note that an zero valued wind files should be present <input_dflowfm> folder, where the time stamps match with the ones given in simulationTimespan in the OpenDA configuration.

20.6 Examples of the application of OpenDA for D-Flow FM

In this section, some examples are elaborated for both the calibration of a model and the ensemble Kalman filtering (abbreviated as ‘EnKF’) of a model. All the examples can be found in the directory <examples/model_dflowfm_blackbox>.

20.6.1 Example 1: Calibration of the roughness parameter

The automatic calibration of a model needs two main choices from the user:

- 1 Which model parameters may be modified during the calibration process?
- 2 Which model results need to be compared to observations, to judge the model quality?

The remainder of this section will be in the form of a tutorial, to directly illustrate all steps in an example model. In this example you will use a small river model ‘simple_waal’ and use the bed roughness to calibrate this model for its three water level observation stations.

Step 1: Inspect the model

All the required model files can be found in the directory <simple_waal_calibration_roughness>. Consider the following steps:

- 1 Start D-Flow FM (standalone) in directory <input_dflowfm/>.
- 2 Select *Files* → *Load MDU-file*.
- 3 Load the model: select <simple_waal.mdu>.
- 4 You can run the model if you like (right mouse button).

The basic model is built to simulate a simple two-dimensional river with a spatially varying bed friction coefficient. It is driven by two boundary conditions: an upstream discharge inflow at the eastern boundary and a downstream water level at the western boundary. Inspect the model forcing in the following way:

- 1 Open the external forcings file <simple_waal.ext> in a text editor.
- 2 Notice how, in addition to the two boundaries, there are two blocks for the friction coefficient. The first one is a spatially varying roughness field in the <sw_nikuradse.xyz> file. The second refers to the <sw_frcfact_all.xyz> file that contains multipliers for the original friction coefficients. A third file <sw_frcfact_template.xyz> is present, which is used to define a number of subdomains.
- 3 In D-Flow FM, select *Files* → *Load sample file* and select <sw_frcfact_template.xyz>.
- 4 Notice how the loaded samples have three distinct values 1, 2 and 3, which act as identifiers: they approximately define the corner points of three subdomains of the entire river stretch. For each subdomain, a different roughness can be calibrated.

Step 2: Select the model parameters

Currently, the only calibratable parameters are the time-independent parameters in the external forcings file that use the .xyz sample file format. The most obvious parameter is the bed friction coefficient.

Step 3: Select the model results

The example directory <simple_waal> contains all the necessary configuration files for the so-called Black Box model wrapper for D-Flow FM to run a ‘twin experiment’. In a twin experiment a model setup with given solution (the synthetic observations) is perturbed after which OpenDA is applied to re-estimate the original settings. The effects of the parameter variations may be judged by comparing time series output in the _his.nc history file to observed data in NOOS time series format. The model output and observation data are compared by calculating a cost function. Which cost function is used is configured in the <dudAlgorithm.xml> in the <algorithm> directory. See the OpenDA documentation for the available options for the cost function.

The D-Flow FM model simulates a 1D river flow. The input files for D-Flow FM are located in the directory <simple_waal/stochModel/input>. D-Flow FM allows the user to specify regions with a different bed friction coefficient (constant for each region) and is able to handle the interpolation of the coefficients between these regions. In the experiment we try to re-estimate the values of the bed friction coefficients of an earlier run. As observations, the waterlevel at three locations (stations) along the river is used. These results are written to the main output file (<*_his.nc>) as time series.

In the directory <simple_waal>, there are two main configuration files of OpenDA present:

- ◊ Simulation.oda: runs a single run of the model, this configuration is mainly used to test the black box configuration files.
- ◊ Dud.oda runs a calibration experiment with algorithm DUD (Doesn’t Use Derivative).

These files configure the main ingredients of an OpenDA run:

- 1 the stochObserver (`org.opendata.observers.NoosTimeSeriesStochObserver`). Observations for this experiment were created by extracting the timeseries for all stations from the netcdf-file and convert them to NOOS format (script `nchis2noos.sh`)
- 2 the stochObserver (`org.opendata.observers.NoosTimeSeriesStochObserver`). Observations for this experiment were created by extracting the time series for all stations from the netcdf-file and convert them to NOOS format (script `nchis2noos.sh`)
- 3 the stochModelFactory (`org.opendata.blackbox.wrapper.BBStochModelFactory`). A black box model configuration consist of three configuration files that are described in more detail below.
- 4 the algorithm (`org.opendata.algorithms.Dud`). Dud is a well known algorithm in calibration experiments, more information about it can be found on the OpenDA website or in the literature.

The configuration files for these 3 components are located in different sub directories to reflect the Object Oriented architecture of OpenDA. The fourth block in the configuration file specifies the result writer (`org.opendata.resultwriters.MatlabResultWriter`). The resulting m-file may be loaded in Matlab to visualize results of the OpenDA run.

In this example, the data exchange between OpenDA and D-Flow FM is limited to the bed friction coefficients and the computed waterlevel at observation locations. The waterlevel at a observation location is expected to be written to NetCDF-file with the following features

- 1 dimension ‘time’ and ‘stations’ are defined
- 2 there exists a variable ‘station_id(stations)’ defined that contains strings with the station_id

For NetCDF-files that satisfy these two conditions OpenDA creates an ‘exchange item’ for

each variable that has the dimensions (time,stations). The exchange item is referred to as 'station id(nr)'.name of variable'.

20.6.2 Example 2: EnKF with uncertainty in the tidal components

The geometry for this test case is the same as used in the Delft3D model example for calibration that was presented in a Deltares webinar (recording, slides and all configuration files for this example are available at the OpenDA website).

Regularly, D-Flow FM uses 1 component file to specify all tidal component (one component at a line). In order to add different noise models to different components, you must split the component file and add one .tim-files for noise for each component.

Again, all configuration files are available, but not much effort has been put into the exact configuration of the EnKF algorithm or the noise model specifications. The results of the SequentialSimulation show that this test case suffers much less from the inexact restart. The whole workflow is highlighted in more detail below.

A few remarks are made:

- ◊ the directories <bin> and <jre> should be on the same level,
- ◊ the computation is started through running the file oda_run_gui.bat,
- ◊ the bare D-Flow FM model is located in the directory <input_dflowfm>,
- ◊ the observations are in .noos-format, and are located in the directory <stochObserver>.

Step 2: Start the EnKF computation

The OpenDA run is launched through the core oda_run_gui.bat file. Once having opened this file, a user interface appears. Within the user interface, an .oda-file can be opened from a certain case directory (in this case, we have <estuary_kalman>). One can choose Enkf.oda, SequentialSimulation.oda or Simulation.oda. In this case, we choose for Enkf.oda.

Step 3: Examine the applied noise

The basic necessary component of an EnKF computation comprises the noise applied to some variable. In this case, the noise is applied to the waterlevel boundary representing the tidal motion. Within the directory <input_dflowfm>, this noise is explicitly declared through a separate polyline and a separate data file for the boundary. The configuration of the noise is accomplished through two .xml-files in the directory <stoch_model>.

Step 4: Run the EnKF computation

By means of the user interface, the EnKF computation can be launched. After having opened the file EnKF.oda, the 'Run'-button can be pressed. The computation is being performed. Along the computation's duration, multiple 'work' directories are generated in the directory <stochModel>, i.e. <work0>, <work1>, <work2>, etc.

Step 5: Evaluate the outcomes

After having run the computation, output files have been generated in Matlab-format. The relevant files are placed in the directory <estuary_kalman>. The data are stored in the Matlab file Enkf_results.m. Visualisation could be accomplished like shown in [Figure 20.1](#).

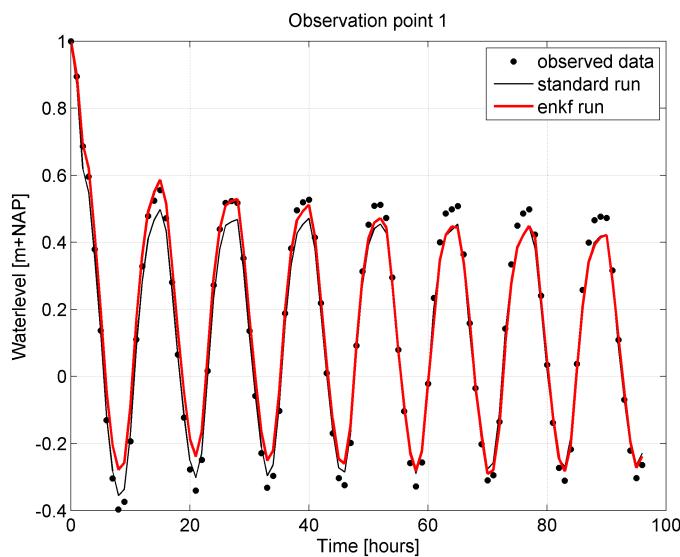


Figure 20.1: Visualisation of the EnKF computation results from OpenDA for a certain observation point. The dots represent the observed data, the black line represents the original computation with D-Flow FM (without Kalman filtering) and the red line represents the D-Flow FM computation with Kalman filtering.

20.6.3 Example 3: EnKF with uncertainty in the inflow velocity

The geometry in this example is the same as for the calibration example: a two-dimensional river model, the initial waterlevel is zero, the river bed is filled gradually due to the boundary conditions. At the inflow boundary, a constant discharge is prescribed (along the line `sw_east_dis_0001.pli`), whereas at the outflow boundary, a constant waterheight is prescribed (along the line `sw_west_wlev_0001.cmp`).

There are 3 observation locations along the river (Obs01, Obs02 and Obs03). The simple matlab script `plot_results_histfile.m` is available to plot the waterlevel as a function of time for these 3 stations. Simulation time span is 100 days (Start: 199208310000, End: 199212090000). The noise contribution is found in file `sw_east_dis_noise.pli`. Initially, no noise is present, so the file contains zeroes in directory `<input_dflowfm>`.

20.6.4 Example 4: EnKF with uncertainty in the inflow condition for salt

The geometry in this example and the boundary conditions for waterlevel and velocity are exactly the same as in `simple_waal_kalman`. The transport of salt is added to the computation by a discharge boundary condition `sw_east_dis_sal_001.pli` and a noise component added to this boundary.

20.6.5 Example 5: EnKF with uncertainty on the wind direction

This example is also converted from a Delft3D test case (d3d_lake_2d): a lake forced by an uniform wind field. This example is a twin experiment with spatially correlated 2D-noise added to the wind field. The noise is created on cartesian (equidistant grid), the noise realisations are written by OpenDA to the <lake2d_windx_noise.amu>. The 2D noise field is interpolated at computational grid by D-Flow FM and added to the uniform wind field. The <SequentialSimulationNoise.oda> can be used to perform a single D-Flow FM run where wind with noise is used as forcing. The resulting <lake2d_his.nc> file can be used as synthetic observations by the <stochObserver>.

20.6.6 Example 6: EnKF with the DCSM v5 model and uncertainty on the wind direction

This example is converted from the SIMONA DCSM v5 model with spatially correlated 2D-noise added to the wind field.

References

- Bagnold, R. A., 1966. *An approach to the sediment transport problem from general physics*. US government Print Office.
- Bailard, J. A., 1981. "An Energetics Total Load Sediment Transport Model for Plane Sloping Beaches." *Journal of Geophysical Research* 86 (C11): 10938-10954.
- Baptist, M. J., 2005. *Modelling floodplain biogeomorphology*. Ph.D. thesis, Delft University of Technology.
- Barenblatt, G. I., M. Bertsch, R. Dal Passo, V. M. Prostokishen and M. Ughi, 1993. "A mathematical model of turbulent heat and mass transfer in stably stratified shear flow." *Journal of Fluid Mechanics* 253: 341-358.
- Baumert, H. and G. Radach, 1992. "Hysteresis of Turbulent Kinetic Energy in Non-rotational Tidal Flows: A Model Study." *Journal of Geophysical Research* 97 (C3): 3669-3677.
- Beckers, J. M., H. Burchard, J. M. Campin, E. Deleersnijder and P. P. Mathieu, 1998. "Another reason why simple discretizations of rotated diffusion operators cause problems in ocean models: comments on "isoneutral diffusion in a z co-ordinate ocean model"." *American Meteorological Society* 28: 1552-1559.
- Bijker, E. W., 1967. *Some considerations about scales for coastal models with moveable bed*. Tech. Rep. 50, WL | Delft Hydraulics, Delft, The Netherlands.
- Blumberg, A. F. and G. L. Mellor, 1985. "Modelling vertical and horizontal diffusivities with the sigma co-ordinate system." *Monthly Weather Review* 113 (8): 1379.
- Burchard, H. and H. Baumert, 1995. "On the performance of a mixed-large model based on the k-epsilon turbulence closure." *Journal of Geophysical Research* 100 (C5): 8523-8540.
- Charnock, H., 1955. "Wind-stress on a water surface." *Q. J. Royal Meteorol. Soc.* 81: 639–640.
- Christoffersen, J. B. and I. G. Jonsson, 1985. "Bed friction and dissipation in a combined current and wave motion." *Ocean Engineering* 12 (5): 387-423.
- D-Flow FM Technical Reference Manual, 2015. *D-Flow FM Technical Reference Manual*. Deltares, 1.1.124 ed.
- Davies, A. G., R. L. Soulsby and H. L. King, 1988. "A numerical model of the combined wave and current bottom boundary layer." *Journal of Geophysical Research* 93 (C1): 491-508.
- Davies, A. M. and H. Gerritsen, 1994. "An intercomparison of three-dimensional tidal hydrodynamic models of the Irish Sea." *Tellus* 46A: 200-221.
- Dingemans, M. W., 1997. *Water Wave Propagation over Uneven Bottoms, Vol. 1 and 2*. Advanced Series on Ocean Engineering, Vol. 13. World Scientific, London.
- Dingemans, M. W., A. C. Radder and H. J. de Vriend, 1987. "Computation of the driving forces of wave-induced currents." *Coastal Engineering* 11: 539-563.
- Eckart, C., 1958. "Properties of water, Part II. The equation of state of water and sea water at low temperatures and pressures." *American Journal of Science* 256: 225-240.
- Fredsøe, J., 1984. "Turbulent boundary layer in wave-current interaction." *Journal of Hydraulic Engineering* 110: 1103-1120.

- Gaeuman, D., E. Andrews, A. Krause and W. Smith, 2009. "Predicting fractional bed load transport rates: Application of the Wilcock-Crowe equations to a regulated gravel bed river." *Water Resources Research* 45.
- Gill, A. E., 1982. *Atmosphere-Ocean dynamics*, vol. 30 of *International Geophysics Series*. Academic Press.
- Grant, W. D. and O. S. Madsen, 1979. "Combined wave and current interaction with a rough bottom." *Journal of Geophysical Research* 84 (C1): 1797-1808.
- Grasmeijer, B. and L. Van Rijn, 1998. "Breaker bar formation and migration." *Coastal Engineering* pages 2750-2758. Virginia, USA.
- Groeneweg, J., 1999. *Wave-current interactions in a generalized Lagrangian mean formulation*. Delft University of Technology, Delft, The Netherlands. Ph.D. thesis.
- Haney, R. L., 1991. "On the pressure gradient force over steep topography in sigma coordinate models." *Journal of Physical Oceanography* 21: 610-619.
- Hirsch, C., 1990. *Numerical computation of internal and external flows*. John Wiley & Sons, New York.
- Huang, W. and M. Spaulding, 1996. "Modelling horizontal diffusion with sigma coordinate system." *Journal of Hydraulic Engineering* 122 (6): 349-352.
- Huynh-Thanh, S. and A. Temperville, 1991. "A numerical model of the rough turbulent boundary layer in combined wave and current interaction." In R. L. Soulsby and R. Bettes, eds., *Sand transport in rivers, estuaries and the sea*, pages 93-100. Balkema Rotterdam.
- Hwang, P., 2005a. "Comparison of the ocean surface wind stress computed with different parameterization functions of the drag coefficient." *J. Oceanogr.* 61: 91–107.
- Hwang, P., 2005b. "Drag coefficient, dynamic roughness and reference wind speed." *J. Oceanogr.* 61: 399–413.
- Ikeda, S., 1982. "Incipient Motion of Sand Particles on Side Slopes." *Journal of the Hydraulics Division, ASCE* 108 (1): 95-114.
- Ikeda, S., 1988. *Not yet known*.
- Isobe, M. and K. Horikawa, 1982. "Study on water particle velocities of shoaling and breaking waves." *Coastal Engineering in Japan* 25: 109-123.
- Kalkwijk, J. P. T. and R. Booij, 1986. "Adaptation of secondary flow in nearly horizontal flow." *Journal of Hydraulic Research* 24 (1): 19-37.
- Klopstra, D., H. J. Barneveld and J. M. Van Noortwijk, 1996. *Analytisch model hydraulische ruwheid van overstroomde moerasvegetatie*. Tech. Rep. PR051, HKV consultants, Lelystad, The Netherlands. Commissioned by Rijkswaterstaat/RIZA, The Netherlands.
- Klopstra, D., H. J. Barneveld, J. M. Van Noortwijk and E. H. Van Velzen, 1997. "Analytical model for hydraulic roughness of submerged vegetation." In *The 27th IAHR Congress, San Francisco, 1997; Proceedings of Theme A, Managing Water: Coping with Scarcity and Abundance*, pages 775-780. American Society of Civil Engineers (ASCE), New York.
- Koch, F. G. and C. Flokstra, 1980. "Bed level computations for curved alluvial channels." In *Proceedings of the XIXth congress of the International Association for Hydraulic Research, 2-7 Feb. 1981, New Delhi, India*, vol. 2, pages 357-364.

- Kolmogorov, A. N., 1942. "Equations of turbulent motion in incompressible fluid." *Izv. Akad. Nauk. SSR, Seria fizicheska Vi* No.1 2 (1-2): 56-58. English translation: 1968 Imperial College, Mech. Eng. Dept. Rept. ON/6.
- Lane, A., 1989. *The heat balance of the North Sea*. Tech. Rep. 8, Proudman Oceanographic Laboratory.
- Leendertse, J. J., 1990. "Turbulence modelling of surface water flow and transport: part IVa." *Journal of Hydraulic Engineering* 114 (4): 603-606.
- Lesser, G., J. van Kester and J. A. Roelvink, 2000. *On-line sediment transport within Delft3D-FLOW*. Tech. Rep. Z2899, wl.
- Myrhaug, D. and O. H. Slaattelid, 1990. "A rational approach to wave-current friction coefficients for rough, smooth and transitional turbulent flow." *Coastal Engineering* 14: 265-293.
- Nipius, K. G., 1998. *Transverse transport modelling using Bailard applied to Grevelingen-mouth delta*. Delft University of Technology, Delft, The Netherlands. M.Sc. thesis, in Dutch (Dwarstransportmodellering m.b.v. Bailard toegepast op de Voordelta Grevelingen-monding).
- O'Connor, B. A. and D. Yoo, 1988. "Mean bed friction of combined wave-current flow." *Coastal Engineering* 12: 1-21.
- Octavio, K. A. H., G. H. Jirka and D. R. F. Harleman, 1977. *Vertical Heat Transport Mechanisms in Lakes and Reservoirs*. Tech. Rep. 22, Massachusetts Institute of Technology.
- Parker, G. and E. D. Andrews, 1985. "Sorting of bed load sediment by flow in meander bends." *Water Resources Research* 21: 1361-1373.
- Partheniades, E., 1965. "Erosion and Deposition of Cohesive Soils." *Journal of the Hydraulics Division, ASCE* 91 (HY 1): 105-139.
- Phillips, N. A., 1957. "A co-ordinate system having some special advantages for numerical forecasting." *Journal of Meteorology* 14: 184-185.
- Postma, L., G. S. Stelling and J. Boon, 1999. "Three-dimensional water quality and hydrodynamic modelling in Hong Kong. Stratification and water quality." In *Proceedings of the 2nd International Symp. on Environmental Hydraulics, Hong Kong, December 1998*, pages 43-49. Balkema, Rotterdam.
- Prandtl, L., 1945. "Über ein neues Formelsystem für die ausgebildete Turbulenz." *Nachrichten von der Akademie der Wissenschaften in Göttingen. Mathematisch-Physikalische Klasse* pages 6-19.
- Richardson, J. F. and W. N. Zaki, 1954. "edimentation and fluidization: Part I." *I. Trans. Institution of Chemical Engineers* 32: 35-53.
- Rienecker, M. M. and J. D. Fenton, 1981. "A Fourier approximation method for steady water waves." *Journal of Fluid Mechanics* 104: 119-137.
- Rijn, L. C. van, 1984a. "Sediment transport, Part I: bed load transport." *Journal of Hydraulic Engineering* 110 (10): 1431-1456.
- Rijn, L. C. van, 1984b. "Sediment transport, Part II: suspended load transport." *Journal of Hydraulic Engineering* 110 (11): 1613-1640.
- Rijn, L. C. van, 1984c. "Sediment transport, Part III: bed form and alluvial roughness." *Journal of Hydraulic Engineering* 110 (12): 1733-1754.

- Rijn, L. C. van, 1993. *Principles of Sediment Transport in Rivers, Estuaries and Coastal Seas*. Aqua Publications, The Netherlands.
- Rijn, L. C. van, 2001. *General view on sand transport by currents and waves : data analysis and engineering modelling for uniform and graded sand (TRANSPOR 2000 and CROS-MOR 2000 models)*. Z2899.20 / Z2099.30 / Z2824.30. WL | Delft Hydraulics, Delft, The Netherlands.
- Rijn, L. C. van, 2003. "Sediment transport by currents and waves; general approximation formulae Coastal Sediments." In *Corpus Christi, USA*.
- Rijn, L. C. van, 2007. "Unified View of Sediment Transport by Currents and Waves. I: Initiation of Motion, Bed Roughness, and Bed-Load Transport." *Journal of Hydraulic Engineering* 133 (6): 649-667.
- Rijn, L. C. van, J. A. Roelvink and W. T. Horst, 2000. *Approximation formulae for sand transport by currents and waves and implementation in DELFT-MOR*. Tech. Rep. Z3054.40, WL | Delft Hydraulics, Delft, The Netherlands.
- Rijn, L. C. van, D. R. Walstra and M. v. Ormondt, 2004. *Description of TRANSPOR2004 and implementation in Delft3D-ONLINE*. Tech. Rep. Z3748.10, WL | Delft Hydraulics, Delft, The Netherlands.
- Rijn, L. van, D. Walstra, B. Grasmeijer, J. Sutherland, S. Pan and J. Sierra, 2003. "The predictability of cross-shore bed evolution of sandy beaches at the time scale of storms and seasons using process-based profile models." *Coastal Engineering* 47: 295-327.
- Rodi, W., 1984. "Turbulence models and their application in Hydraulics, State-of-the-art paper article sur l'etat de connaissance." *IAHR* Paper presented by the IAHR-Section on Fundamentals of Division II: Experimental and Mathematical Fluid Dynamics, The Netherlands.
- Roelvink, J. A. and M. J. F. Stive, 1989. "Bar-generating cross-shore flow mechanisms on a beach." *Journal of Geophysical Research* 94 (C4): 4785-4800.
- Ryan, P. J., D. R. F. Harleman and K. D. Stolzenbach, 1974. "Surface Heat Loss From Cooling Ponds." *Water Resources Research* 10 (5): 930-938.
- Schrama, E., 2007. "Tides. Lecture Notes AE4-876."
- Slørdal, L. H., 1997. "The pressure gradient force in sigma-co-ordinate ocean models." *International Journal Numerical Methods In Fluids* 24: 987-1017.
- Smith, S. D. and E. G. Banke, 1975. "Variation of the sea surface drag coefficient with wind speed." *Quarterly Joournal of the Royal Meteorological Society* 101: 665-673.
- Soulsby, R., 1997. *Dynamics of marine sands, a manual for practical applications*. Thomas Telford, London.
- Soulsby, R. L., A. G. Davies, J. Fredsøe, D. A. Huntley, I. G. Jonnson, D. Myrhaug, R. R. Simons, A. Temperville and T. J. Zitman, 1993a. "Bed shear stresses due to combined waves and currents." In *Abstracts-in-depth of the Marine Science and Technology G8-M overall workshop, Grenoble.*, pages 2.1-1/2.1-4.
- Soulsby, R. L., L. Hamm, G. Klopman, D. Myrhaug, R. R. Simons and G. P. Thomas, 1993b. "Wave-current interaction within and outside the bottom boundary layer." *Coastal Engineering* 21: 41-69.
- Stelling, G. S. and J. A. T. M. van Kester, 1994. "On the approximation of horizontal gradients in sigma co-ordinates for bathymetry with steep bottom slopes." *International Journal Numerical Methods In Fluids* 18: 915-955.

- Stive, M. J. F., 1986. "A model for cross-shore sediment transport." In *Proceedings 20th International Coastal Engineering Conference*, pages 1550-1564. American Society of Civil Engineers, New York.
- Swart, 1974. *Offshore sediment transport and equilibrium beach profiles*. Ph.D. thesis, Delft University of Technology, Delft, The Netherlands. Delft Hydraulics Publ. 131.
- Sweers, H. E., 1976. "A nomogram to estimate the heat exchange coefficient at the air-water interface as a function of windspeed and temperature; a critical survey of some literature." *Journal of Hydrology* 30: –.
- Talmon, A. M., N. Struiksma and M. C. L. M. van Mierlo, 1995. "Laboratory measurements of the direction of sediment transport on transverse alluvial-bed slopes." *Journal of Hydraulic Research* 33 (4): 495-517.
- Uittenbogaard, R. E., J. A. T. M. van Kester and G. S. Stelling, 1992. *Implementation of three turbulence models in 3D-TRISULA for rectangular grids*. Tech. Rep. Z81, WL | Delft Hydraulics, Delft, The Netherlands.
- Walstra, D. J. R., J. A. Roelvink and J. Groeneweg, 2000. "Calculation of Wave-Driven Currents in a 3D Mean Flow Model." In *Proceedings 27th International Conference on Coastal Engineering, Sydney, Australia, July 16-21, 2000*.
- Wilcock, P. and J. Crowe, 2003. "Surface-based transport model for mixed-size sediment." *Journal of Hydraulic Engineering* 129 (2): 120-128.
- Winterwerp, J. C. and R. E. Uittenbogaard, 1997. *Sediment transport and fluid mud flow*. Tech. Rep. Z2005, WL | Delft Hydraulics, Delft, The Netherlands.

A The master definition file

The MDU-file contains the key information of the flow model. Besides the names of the relevant user specified files, such as the grid file and the external forcings file, the values of various model parameters should be specified in the MDU-file. The entire list of available model parameter settings is given in [Table A.1](#) as well as the associated default setting for these parameters.

The basename of the MDU-file, without `.mdu`, is also used as the model identification string, and is often denoted as *mdident* throughout this User Manual. It is equivalent to Delft3D-FLOW's *runid* concept.

Table A.1: Standard MDU-file with default settings.

Keyword	Default setting	Description
[model]		
Program	D-Flow FM	Program
Version	1.1.134.38754	Version
MDUFormatVersion	1.02	File format version. Do not edit this.
AutoStart	0	Autostart simulation after loading MDU or not (0=no, 1=autostart, 2=autostartstop).
[geometry]		
NetFile	*_net.nc	
BathymetryFile	*.xyz	
DryPointsFile	Dry points file *.xyz, third column dummy z values, or polygon file *.pol.	
WaterLevIniFile	Initial water levels sample file *.xyz	
LandBoundaryFile	Only for plotting	
ThinDamFile	*_thd.pliz, Polyline(s) for tracing thin dams.	
FixedWeirFile	*_fxw.pliz, Polyline(s) x,y,z, z = fixed weir top levels (formerly fixed weir)	
VertplizFile	*_vlay.pliz), = pliz with x,y, Z, first Z =nr of layers, second Z = laytyp	
ProflocFile	*_proflocation.xyz) x,y,z, z = profile refnumber	
ProfdefFile	*_profdefinition.def) definition for all profile nrs	
ProfdefxyzFile	*_profdefinition.def) definition for all profile nrs	
ManholeFile	File containing manholes (e.g. *.dat)	
PartitionFile	*_part.pol, polyline(s) x,y	
Uniformwidth1D	2.	Uniform width for 1D profiles not specified bij profloc
WaterLevIni	0.	Initial water level
Bedlevuni	-5.	Uniform bottom level, (only if bedlev-type>=3, used at missing z values in netfile)
Bedslope	0.	bedslopeinclination, sets zk = bedlevuni + x*bedslope and sets zbnz = xbndz*bedslope
BedlevType	3	1: at cell center (tiles xz,yz,bl,bob=max(bl)), 2: at face (tiles xu,yu,blu,bob=blu), 3: at face (using mean node values), 4: at face (using min node values), 5: at face (using max node values), 6: with bl based on node values

(continued on next page)

Keyword	Default setting	Description
<i>(continued from previous page)</i>		
Blmeanbelow	-999.	if not -999d0, below this level the cell centre bedlevel is the mean of surrounding netnodes
Blminabove	-999.	if not -999d0, above this level the cell centre bedlevel is the min of surrounding netnodes
AngLat	0.	Angle of latitude S-N (deg), 0=no Coriolis
AngLon	0.	Angle of longitude E-W (deg), 0=Greenwich
Conveyance2D	-1	-1:R=HU,0:R=H, 1:R=A/P, 2:K=analytic-1D conv, 3:K=analytic-2D conv
Nonlin2D	0	Non-linear 2D volumes, only icm ibedlev-type = 3 and Conveyance2D>=1
Sillheightmin	0.5	Weir treatment only if both sills larger than this value (m)
Makeorthocenters	0	1=yes, 0=no switch from circumcentres to orthocentres in geominit
Dcenterinside	1.	limit cell center; 1.0:in cell <-> 0.0:on c/g
Bamin	1.d-6	Minimum gridcell area , icm cutcells
OpenBoundaryTolerance	3.	Search tolerance factor between boundary polyline and grid cells. Unit: in cell size units (i.e., not metres).
Kmx	0	Max nr of vertical layers
Layertype	1	1= all sigma, 2 = all z, 3 = use VertplizFile
Numtopsig	0	Nr of sigmalayers in top of Zlayer model
SigmaGrowthFactor	1.	layer thickness growth factor from bed up
 [numerics]		
CFLMax	0.7	Max. Courant nr.
CFLWaveFrac	0.1	Wave velocity fraction, total courant vel = u + cflw*wavevelocity
AdvecType	33	Adv type, 0=no, 1= Wenneker, qu-udzt, 2=1, q(ui0-u), 3=Perot q(ui0-u), 4=Perot q(ui-u), 5=Perot q(ui-u) without itself
Lincontin	0	Default 0; Set to 1 for linearizing d(Hu)/dx; link to AdvecType
TimeStepType	2	0=only transport, 1=transport + velocity update, 2=full implicit step_reduce, 3=step_jacobi, 4=explicit
Limtyphu	0	Limiter type for waterdepth in continuity eq., 0=no, 1=min-mod,2=vanLeer,3=Kooren,4=Monotone Central
Limtypmom	4	Limiter type for cell center advection velocity, 0=no, 1=min-mod,2=vanLeer,3=Kooren,4=Monotone Central
Limtypsa	4	Limiter type for salinity transport, 0=no, 1=min-mod,2=vanLeer,3=Kooren,4=Monotone Central
Maxdegree	6	Maximum degree in Gauss elimination
Vertadvtypsal	5	Vertical advection type for salinity, 0=No, 1=Upwexpl, 2=Centralepl, 3=UpwimpL, 4=CentralImpl, 5=4 but 3 for neg. stratif., 6=higher order expl, no forester

(continued on next page)

Keyword	Default setting	Description
<i>(continued from previous page)</i>		
Icgsolver	4	Solver type , 1 = sobekGS_OMP, 2 = sobekGS_OMPthreadsafe, 3 = sobekGS, 4 = sobekGS + Saadilud, 5 = parallel/global Saad, 6 = parallel/Petsc, 7 = parallel/GS
FixedWeirScheme	0	0 = no, 1 = compact stencil, 2 = whole tile lifted, full subgrid weir + factor
FixedWeirContraction	1.	flow width = flow width*FixedWeirContraction
Jbasqbnddownwindhs	0	0 : original hu on qbnd, 1 = downwind hs on qbnd
Izbndpos	0	Position of z boundary, 0=D3Dflow, 1=on net boundary, 2 = on specifiend polyline
Tlfsmo	0.	Fourier smoothing time on waterlevel boundaries (s)
Slopedrop2D	0.	Apply droplosses only if local bottom slope > Slopedrop2D, <=0 =no droplosses
Chkadvd	0.1	Check advection terms if depth < chkad-vdp, => less setbacks
Teta0	0.55	Teta of time integration, 0.5 < Teta < 1d0
Qhrelax	1.d-2	Relaxation qhbnd ()
cstbnd	0	Delft-3D type velocity treatment near boundaries for small coastal models (1) or not (0)
Maxitverticalforestersal	100	0 : no vertical filter, > 0 = Max nr of iterations
Maxitverticalforesterem	0	0 : no vertical filter for temp, > 0 = Max nr of iterations
Jaorgsethu	1	0 : setumod, sethu, setau sequence, 1 : sethu, setau, setumod sequence (standard)
ilutype	0	0: parms-default
nlevel	0	0: parms-default
dtol	0.	0d0: parms-default
Epshu	1.d-4	Input for threshold water depth for wet and dry cells
Maxwaterleveldiff	0.	upper bound (in m) on water level changes, <= 0: no bounds
Maxvelocitydiff	0.	upper bound (in m/s) on velocity changes, <= 0: no bounds
TransportMethod	1	Transport method
Turbulencemode	3	0=no, 1 = constant, 2 = algebraic, 3 = k-eps, 4 = k-tau
Turbulenceadvection	3	0=no, 3 = hor. expl., vert. impl.
<hr/>		
[physics]		
UnifFrictCoef	2.3d-2	Uniform friction coefficient, 0=no friction
UnifFrictType	1	0=Chezy, 1=Manning, 2=White Colebrook, 3=idem, WAQUA style
UnifFrictCoef1D	2.3d-2	Uniform friction coefficient in 1D links, 0=no friction
UnifFrictCoefLin	0.	Uniform linear friction coefficient for ocean models (m/s), 0=no
Umodlin	0	linear friction umod , ifrctyp 4,5,6
Vicouv	1.	Uniform horizontal eddy viscosity (m2/s)
Dicouv	1.	Uniform horizontal eddy diffusivity (m2/s)
Vicoww	5.d-5	Uniform vertical eddy viscosity (m2/s)
Dicoww	5.d-5	Uniform vertical eddy diffusivity (m2/s)

(continued on next page)

Keyword	Default setting	Description
<i>(continued from previous page)</i>		
Vicwminb	0.	Minimum visc in prod and buoyancy term (m ² /s)
Smagorinsky	0.	Add Smagorinsky horizontal turbulence : vicu = vicu + ((Smagorinsky*dx)**2)*S, e.g. 0.1
Elder	0.	Add Elder contribution : vicu = vicu + Elder*kappa*ustar*H/6), e.g. 1.0
irov	0	0=free slip, 1 = partial slip using wall_ks
wall_ks	0.	Nikuradse roughness for side walls, wall_z0=wall_ks/30
Rhomean	1000.	Average water density (kg/m ³)
Idensform	0	1=Eckard, 2=Unesco, 3=baroclin case
Ag	9.81	Gravitational acceleration
TidalForcing	0	Tidal forcing (0=no, 1=yes) (only for jsferic == 1)
Doodsonstart	55.565	TRIWAQ = 55.565D0 , D3D = 57.555D0
Doodsonstop	375.575	TRIWAQ = 375.575D0 , D3D = 275.555D0
Doodsoneps	0.0	TRIWAQ = 0.0 400 cmps , D3D = 0.03 60 cmgs
Salinity	0	Include salinity, (0=no, 1=yes)
InitialSalinity	0.	Initial salinity concentration (ppt)
Sal0abovezlev	-999.	Salinity 0 above level (m)
DeltaSalinity	-999.	for testcases
Temperature	0	Include temperature, (0=no, 1=only transport, 5=heat flux model (5) of D3D), 3=excess model of D3D
InitialTemperature	6.	Initial temperature (degC)
Secchidepth	1.	Water clarity parameter (m)
Stanton	-1.	Coefficient for convective heat flux (), if negative, use Cd wind
Dalton	-1.	Coefficient for evaporative heat flux (), if negative, use Cd wind
SecondaryFlow	0	Secondary flow (0=no, 1=yes)
<hr/>		
[wind]		
ICdtyp	2	(), 1=const, 2=S&B 2 breakpoints, 3= S&B 3 breakpoints, 4=Charnock constant (), e.g. 0.00063 0.00723
Cdbreakpoints	6.3d-4 7.23d-3	(m/s), e.g. 0.0 100.0
Windspeedbreakpoints	0. 100.	Air density (kg/m ³)
Rhoair	1.200	Average air Pressure on open boundaries, (N/m ²), only applied if value > 0
PavBnd	0.	Only relevant for Spiderweb: Global Atmospheric Pressure, (N/m ²)
Gapres	101325.	
<hr/>		
[time]		
RefDate	20010101	Reference date (yyyymmdd)
Tzone	0.	Data Sources in GMT are interrogated with time in minutes since refdat-Tzone*60
Tunit	S	Time units in MDU (H, M or S)
DtUser	300.	User timestep in seconds (interval for external forcing update & his/map output)
DtMax	30.	Max timestep in seconds
DtInit	1.	Initial timestep in seconds
TStart	0.	Start time w.r.t. RefDate (in TUnit)
TStop	8640000.	Stop time w.r.t. RefDate (in TUnit)
<hr/>		
[restart]		
<i>(continued on next page)</i>		

Keyword	Default setting	Description
<i>(continued from previous page)</i>		
RestartFile		Restart file, only from netcdf-file, hence: either *_rst.nc or *_map.nc
RestartDateTime		Restart time (YYYYMMDDHHMMSS), only relevant in case of restart from *_map.nc
[externalforcing]		
ExtForceFile		Old format for external forcings file *.ext, link with tim/cmp-format boundary conditions specification
ExtForceFileNew		New format for external forcings file *.ext, link with bc -format boundary conditions specification
[output]		
OutputDir		Output directory of map-, his-, rst-, dat- and timings-files, default: DFM_OUTPUT_<modelname>. Set to . for no dir/current dir.
ObsFile		*.xyn Coords+name of observation stations.
CrsFile		*_crs.pli Polyline(s) defining cross section(s).
HisFile		*_his.nc History file in NetCDF format.
HisInterval	120.	History output, given as 'interval' 'start period' 'end period' (s)
XLSInterval	0.	Interval (s) between XLS history
FlowGeomFile		*_flowgeom.nc Flow geometry file in NetCDF format.
MapFile		*_map.nc Map file in NetCDF format.
MapInterval	1200.	Map file output, given as 'interval' 'start period' 'end period' (s)
MapFormat	1	Map file format, 1: netCDF, 2: Tecplot, 3: netCFD and Tecplot
Heatfluxesonoutput	0	1=yes,0=no
Richardsononoutput	0	1=yes,0=no
RstInterval	86400.	Restart file output, given as 'interval' 'start period' 'end period' (s)
S1incinterval	0.	Interval (m) in incremental file for water-levels S1
WaqFileBase		Basename (without extension) for all Delwaq files to be written.
WaqInterval	0.	Interval (in s) between Delwaq file outputs
StatsInterval	0.	Interval (in s) between simulation statistics output.
Writebalancefile	0	Write Balancefile, 1=yes, 0=no
TimingsInterval	0.	Timings output interval
TimeSplitInterval	0X	Time splitting interval, after which a new output file is started. value+unit, e.g. '1 M', valid units: Y,M,D,h,m,s.
MapOutputTimeVector		File (.mpt) containing fixed map output times (s) w.r.t. RefDate
FullGridOutput	0	0:compact, 1:full time-varying grid data
SnapshotDir	figures	Directory where snapshots/screenumps are saved.

B Attribute files

B.1 Introduction

In the following sections we describe the attribute files used in the input file (MDU-file) of D-Flow FM. Most of these files contain the quantities that describe one specific item, such as the location of open boundaries, or time dependent data of fluxes discharged in the model area by discharge stations. Most of the attribute files can be generated by the Delta Shell GUI after defining an input scenario. Some files can almost only be generated by utility programs such as the unstructured grid generated by RGFGRID. Still, we describe both type of files as it might be useful to know how the input data is structured to be able to generate (large) files, such as astronomic boundary conditions, or time-series for wind speed and direction by client specific tools.

B.2 Polyline/polygon file

D-Flow FM uses the same format for polylines and (closed) polygons. When used as a polygon file, there is *not* the requirement that the first and last point should be identical. It is good modelling practice to name files containing polygons with the extension .pol, and files containing polylines with the extension .pli. When the polylines have a third column with z -coordinates, the extension .pliz is advised.

File contents	The coordinates of one or more polylines. Each polyline (piecewise linear) is written in a single block of data.
Filetype	ASCII
File format	Free formatted
Filename	<name.pol>
Generated	RGFGRID, QUICKIN, Delta Shell, etc

Record description:

Record	Record description
	Preceding description records, starting with an asterisk (*), and will be ignored.
1	A non blank character string, starting in column one.
2	Two integers N_r, N_c representing the numbers of rows and number of columns for this block of data.
	Two reals representing the x, y or λ, ϕ -coordinate, followed by remaining data values at that location (if $N_c > 2$).

Example:

```
*  
* Polyline L007  
*  
L007  
6  2      132400.0  549045.0  
        132345.0  549030.0  
        132165.0  549285.0  
        131940.0  549550.0  
        131820.0  549670.0  
        131585.0  549520.0  
*  
* Polyline L008  
*
```

```
L008
4 2
    131595.0    549685.0
    131750.0    549865.0
    131595.0    550025.0
    131415.0    550175.0
*
* Polyline L009
*
L009
6 2
    131595.0    549655.0
    148975.0    564595.0
    150000.0    564935.0
    152105.0    565500.0
    153150.0    566375.0
    154565.0    567735.0
```

B.3 Sample file

Sample file are used for several of D-Flow FM's input files.

File contents	The location and value of samples.
Filetype	ASCII
File format	Free formatted
Filename	<name.xyz>
Generated	Manually or Offline with QUICKIN or Delta Shell and data from digitised charts or GIS-database.

Record description:

Filetype	Record description
Free formatted	Location and sample value per row Two reals representing the x, y or λ, ϕ -coordinate and one real representing the sample value

Example:

Sample file with 12 sample values with their location (free formatted file).

```
213813.2    603732.1    -4.053000
214686.0    607226.1    -4.522000
214891.7    610751.2    -5.000000
210330.8    601424.1    -2.169000
211798.0    604444.8    -2.499000
212460.0    607475.7    -2.760000
212436.9    610362.5    -2.865000
185535.4    606607.9    1.360000
186353.0    603789.4    1.122000
187959.2    601197.6    0.9050000
190193.0    599101.5    0.7050000
208578.7    602513.7    -0.7990000
```

B.4 Time series file (ASCII)

Time series files are used for several of D-Flow FM's input files. There is no header, except for optional comment lines. There should be two or more columns: the first column contains time in minutes since the model's reference date, all remaining columns contain the data values. Each line contains a single time with its (space-uniform) values.

B.5 The external forcings file

Two definition files for the external forcings are supported, each with their own format.

B.5.1 Old style external forcings

The name of the file is specified in the MDU-file as "ExtForceFile". External forcings are specified in blocks key-value pairs, e.g.

```
* comments
* comments

QUANTITY =waterlevelbnd
FILENAME =tfl_01.pli
FILETYPE =9
METHOD =3
OPERAND =0

QUANTITY = ...
FILENAME = ...
...
```

The format is not case-sensitive, but key-value pairs are expected in the *particular order* shown below.
Accepted keywords are:

quantity	character	Name of the quantity, see the list below
filename	character	File associated with this forcing
sourcemask*	character	File containing a mask
filetype	integer	Indication of the filetype: 1. Time series (D.1.6) 2. Time series magnitude and direction 3. Spatially varying weather 4. ArclInfo 5. Spiderweb data (cyclones) (B.10.3) 6. Curvilinear data (B.10.2) 7. Samples (B.3) 8. Triangulation magnitude and direction 9. Polyline (<.pli>-file, B.2) 11. NetCDF grid data (e.g. meteo fields) 14. NetCDF wave data
method	integer	Method of interpolation: 1. Pass through (no interpolation) 2. Interpolate time and space 3. Interpolate time and space, save weights 4. Interpolate space 5. Interpolate time 7. Interpolate/Extrapolate time
operand	integer	
value*	float	custom coefficients for transformation
factor*	float	
ifrctyp*	float	
averagingtype*	float	
relativesearchcellsize*	float	
extrapoltol*	float	
area*	float	Area for source/sink

note: Keywords marked with * are optional

B.5.2 New style external forcing (boundary conditions only)

The name of the file is specified in the MDU-file as "ExtForceFileNew". External forcings are specified in the ini-file format in blocks headed by [boundary], followed by a list of key-value pairs in random order, e.g.

```
* comments
[boundary]
quantity=waterlevelbnd
locationfile=tfl_01.pli
forcingfile=tfl_01.bc

[boundary]
quantity= ...
...
```

Accepted keywords are:

quantity	character	Name of the quantity, see the list below (boundaries only)
location	character	Boundary polyline <.pli>
forcing	character	BC-file with boundary data <.bc>
	character	NetCDF-file with boundary time series <.nc>

B.5.3 Accepted quantity names

Most members on the list of accepted quantity names can be sorted in either of five categories:

- ◊ Boundary conditions
- ◊ Meteorological fields
- ◊ Structure parameters
- ◊ Initial fields
- ◊ Spatial physical properties

Table B.1: List of accepted external forcing quantity names.

Quantity	pg.	Description
Boundary conditions:		
waterlevelbnd	113	Water level
neumannbnd	115	Water level gradient
riemannbnd	116	Riemann invariant
outflowbnd		
velocitybnd	115	Velocity
dischargebnd	114	Discharge
riemann_velocitybnd		Riemann invariant velocity
salinitybnd	143	Salinity
temperaturebnd	143	Temperature
sedimentbnd	209	Suspended sediment
uxuyadvectionvelocitybnd		
normalvelocitybnd,		
tangentialvelocitybnd		
qhbnd	117	Discharge-water level dependency
tracerbnd<tracername>	143	User-defined tracer
Meteorological fields:		
windx, windy, windxy	168	Wind components, wind vector
airpressure_windx_windy	168	Atmospheric pressure and wind components
atmosphericpressure	168	Atmospheric pressure
rainfall		Precipitation
humidity_airtemperature...		
..._cloudiness		

Quantity	pg.	Description
humidity_airtemperature... ..._cloudiness_solarradiation discharge_salinity... ..._temperature_sorsin	142	Discharge, salinity and heat sources
Structure parameters: pump damlevel gatelowerededgelevel generalstructure	46	Pump capacity
Initial fields: initialwaterlevel initialsalinity initialsalinitytop initialtemperature initialverticaltemperatureprofile initialverticalsalinityprofile initialtracer< <i>tracername</i> >	331	
Spatial physical properties: frictioncoefficient horizontaleddyviscositycoefficient horizontaleddydiffusivitycoefficient advectiontype ibotlevtype	332	
Miscellaneous: shiptxy movingstationxy	342	Moving observation point for output (time,x,y)

B.6 Trachytopes

The trachytope functionality allows for the usage of different types of roughness formulations at different locations within the computational domain. Multiple formulation may be active in the same grid cell. Several keywords in the MDU file influence the functioning. All keywords below should be placed underneath the [trachytopes] section in the MDU file.

Keyword	Value	Description	Default
TrtRou	Y or N	Trachytope option activated	N
TrtDef	<name.ttd>	Definition file trachytopes	
Trtl	<name.arl>	Area Roughness on Link file trachytopes	
DtTrt	pos. real	Time step in seconds for updating roughness and resistance coefficients based on trachytopes. Must be a multiple of DtMax.	1 DtMax

B.6.1 ARL-file (Area Roughness on Links)

File contents	The Area Roughness on Links file (ARL-file) is the input file for the spatial distribution of the alluvial and vegetation roughness which are handled by the trachytopes module.
Filetype	ASCII
File format	Space separated file format
Filename	$\langle name.mdu \rangle$
Generated	At present: Conversion possible from structured Delft3D-FLOW input files with matlab conversion tool from Open Earth Tools (http://www.openearth.info/), see example below (Section B.6.1.2)).

The ARL file has the following properties:

- ◊ A single line comment, starts with a hash tag "#" or an asterisk "*".
- ◊ Each line has the format
"xu yu zu TrachytopesNr Fraction",
 - where xu, yu, zu is the coordinate of the midpoint of the netlink,
 - TrachytopesNr is an integer corresponding to the number as described in the TrachyTopes Definition File (cf. Section B.6.2),
 - and Fraction is a Fraction between 0.0 and 1.0
- ◊ A line with a midpoint-netlink-coordinate which is the same as the preceding line (comments not considered) allows multiple types of trachytopes roughness definitions, provided the sum of the Fraction keywords does not add to a value greater than 1.0.
- ◊ If the sum is less than 1.0 the background roughness prescribed in the [physics] chapter in the MDU file is prescribed for the remaining Fraction, cf. Appendix A. Only a single roughness type is allowed in combination with the Trachytopes module.
- ◊ If a midpoint-netlink-coordinate is repeated in the .arl file with the midpoint-netlink-coordinate xu, yu, zu in the preceding line being different, all previous instances of the specific xu, yu, zu are ignored.

B.6.1.1 Example

An example of the .arl file is given below based on the .ttd file given in Section B.6.2. In this case the netlink with u point located at $x_u = 10.542$ and $y_u = 11.6$, which is covered for 30%, 30% 20% and 20% for TrachytopesNr = 1, 2, 4 and 3 respectively.

(continued)
...
10.542 11.6 0 1 0.3
10.542 11.6 0 2 0.3
10.542 11.6 0 4 0.2
10.542 11.6 0 3 0.2
...
(continued)

B.6.1.2 Conversion from Delft3D input files

Open Earth Tools has different matlab tools available for the conversion of Delft3D models to D-Flow FM models (e.g. [dflowfmConverter.m](#) and [d3d2dfowfm.m](#)). An extra Matlab conversion script has been added to convert Delft3D's .aru and .arv files to .arl files for D-Flow FM: [d3d2dfowfm_friction_trachytopes.m](#).

It can be called as follows:

```
oetsettings
d3d2dfowfm_friction_trachytopes(filgrd,filaru,filarv,filnet,filarl)
```

where the variables are defined as follows:

filgrd	delft3d grid filename (*.grd)
filaru	delft3d area definition file in u-direction (*.aru)
filarv	delft3d area definition file in v-direction (*.arv)
filnet	name of the dflowfm network file (*.net.nc)
filarl	name of the dflowfm trachytope file (*.arl)

B.6.2 TTD-file

The trachytope definition file contains lines of the following format defining the different types of trachytopes. The types which are supported are general, discharge dependent and water-level dependent formats.

B.6.2.1 General format

The general format is formatted as follows:

```
TrachytopeNr    FormulaNr    ...Parameters...
```

where ...Parameters... indicates a space separated list of formula specific parameters; the parameters required and their order are specified in [section B.6.2.5](#). The user must specify for each trachytope (combination of formula number and parameters) a unique positive trachytope number. This trachytope number is used in the area files (see [section B.6.1](#)) to indicate the roughness types on a net link.

B.6.2.2 Example

An example of such a file where the first three codes 1-3 with a Nikuradse roughness height are defined, and the fourth (code 4) is Chezy roughness height.

```
1      51      0.1
2      51      0.2
3      51      0.3
4      52      35
```

B.6.2.3 Discharge dependent format

The discharge dependent format is formatted as follows:

```
TrachytopeNr    DISCHARGE    "Cross-section name"
TrachytopeNr    Q1          FormulaNr    ...Parameters...
TrachytopeNr    Q2          FormulaNr    ...Parameters...
TrachytopeNr    Q3          FormulaNr    ...Parameters...
TrachytopeNr    Q4          FormulaNr    ...Parameters...
...
...
```

which again expects a user defined TrachytopeNr. The first row contains the keyword “DISCHARGE” and subsequently a cross-section name occurring in CrsFile in the mdu-file. The cross-section name can be enclosed by quotation marks, for instance when the cross-section name is made up of more than one word. The subsequent rows all have the same TrachytopeNr as the first row and furthermore every FormulaNr should be the same. The list of discharges Q1, Q2, Q3, Q4 should be monotonically increasing. The ...Parameters... for each formula type are specified in Section [section B.6.2.5](#).

B.6.2.4 Water level dependent format

The water-level dependent format is similar to the discharge dependent format and is formatted as follows:

```
TrachytopeNr    WATERLEVEL    "Observation-station name"  
TrachytopeNr    ZS1        FormulaNr     ...Parameters...  
TrachytopeNr    ZS2        FormulaNr     ...Parameters...  
TrachytopeNr    ZS3        FormulaNr     ...Parameters...  
TrachytopeNr    ZS4        FormulaNr     ...Parameters...  
...
```

which again expects a user defined TrachytopeNr. The first row contains the keyword “WATERLEVEL” and subsequently an observation-station name occurring in ObsFile in the mdu-file. The observation-station name can be enclosed by quotation marks, for instance when the observation-station name is made up of more than one word. The subsequent rows all have the same TrachytopeNr as the first row and furthermore every FormulaNr should be the same. The list of waterlevels ZS1, ZS2, ZS3, ZS4 should be monotonically increasing. The ...Parameters... for each formula type are specified in [section B.6.2.5](#).

B.6.2.5 Supported roughness formulations

The roughness formulations specified in the table below are supported by D-Flow FM. These formulations are specified in the .ttd by `FormulaNr` and `...Parameters...` in the order in which they appear in the table below. The related formulae are presented in the Technical Reference Manual.

FormulaNr	Description	Parameters
Special classes (1–50)		
1	flood protected area, reduces the effective area of the grid cell. It has no influence on the continuity equation (i.e. it does not decrease the surface area of the grid cell).	–
2	composite trachytope: fraction α of type T_1 and fraction β (generally $\beta = 1 - \alpha$) of type T_2	T_1, T_2, α, β
Area trachytope classes: simple type (51–100)		
51	constant White-Colebrook/Nikuradse value	k [m]
52	constant Chézy value	C [$m^{1/2}/s$]
53	constant Manning value	n [$s/m^{1/3}$]
54	constant z_0 value	z_0 [m]
61	constant White-Colebrook/Nikuradse values for ebb and flood	k_{ebb} [m], k_{flood} [m]
62	constant Chézy values for ebb and flood	C_{ebb} [$m^{1/2}/s$], C_{flood} [$m^{1/2}/s$]
63	constant Manning values for ebb and flood	n_{ebb} [$s/m^{1/3}$], n_{flood} [$s/m^{1/3}$]
64	constant z_0 values for ebb and flood	$z_{0,ebb}$ [m], $z_{0,flood}$ [m]
Area trachytope classes: alluvial type (101–150)		
101	simplified Van Rijn	A [$m^{0.3}$], B [$m^{0.3}$]
102	power relation	A [$m^{1/2}/s$], B [-]
103 ¹	Van Rijn predictor	-
104 ¹	Struiksma predictor	A_1 [$m^{1/2}/s$], A_2 [-], θ_c [-], θ_m [-], C_{min} [$m^{1/2}/s$]
105 ¹	bedforms quadratic	-
106 ¹	bedforms linear	-
Area trachytope classes: vegetation type (151–200)		
151	Barneveld 1	h_v [m], n [1/m]
152	Barneveld 2	h_v [m], n [1/m], C_D [-], k_b [m]
153	Baptist 1	h_v [m], n [1/m], C_D [-], C_b [$m^{1/2}/s$]
154	Baptist 2	h_v [m], n [1/m], C_D [-], C_b [$m^{1/2}/s$]
Linear trachytope classes: various (201–250)		
201	hedges 1	h_v [m], n [1/m]
202	hedges 2	h_v [m], n [1/m]
Linear trachytope classes: various (251–300)		
251	trees	h_v [-], C_D [-]

¹ The alluvial roughness predictors 103, 104, 105 and 106 are not yet supported, because the coupling with the morphology module is not yet available.

B.7 Sources and sinks

Sources and sinks ([section 7.8](#)) are defined as part of the <*.ext> file, as follows:

```
QUANTITY=discharge_salinity_temperature_sorsin
FILENAME=chan1_westeast.pli # A file 'chan1_westeast.tim', with same basename
                           # as the polyline should be present
FILETYPE=9
METHOD =1
OPERAND =0
AREA   =1.5
```

The <*.pli> file is a polyline file ([section B.2](#)) with two or three columns (for 2D or 3D models, respectively). Along with the <*.pli>-file, there has to be a time series file ([section D.1.6](#)) with the same basename as the <*.pli>-file, and extension <.tim>. The <*.tim> file should always have four columns: time in minutes, discharge Q in [m^3/s], $\Delta\text{salinity}$ in [ppt] and ΔT in [$^\circ\text{C}$].

B.8 Dry points and areas

Dry points can either be defined by a basic sample file (see [section B.3](#)), or a polygon file (see [section B.2](#)). Special attention must be given to the optional third column of a polygon file: when the first point has a z-value of -1 , the polygon mask is inverted, i.e., all points outside the polygon are set as dry points. More details can be found in [section 4.4.2.7](#).

B.9 Structure INI file

Hydraulic structures are defined in a <structures.ini> file. Each structures is defined in its own [structure] section, followed by a set of key-value parameters that depend on the type of structure:

```
[structure]
type          = weir           # Type of structure
id            = weir01         # Name of the structure
polylinefile  = weir01.pli    # *.pli; Polyline location for structure
crest_level   = weir01_crest.tim # Crest height in [m]

[structure]
type          = gate           # Type of structure
id            = gate01         # Name of the structure
polylinefile  = gate01.pli    # *.pli; Polyline location for structure
sill_level    = gate01_crest.tim # Sill (crest) height in [m]
sill_width    = 39.5          # (Optional) sill width in [m], between
                             # the fixed side walls.
door_height   = 8.54           # Gate door height in [m], used for
                             # detecting flow across door.
lower_edge_level = gate01_ledge.tim # Position of gate's lower edge in [m],
                                    # used for flow underneath door.
opening_width  = gate01_opening.tim # (Optional) opening width between two
                                    # sideways closing gate doors.
horizontal_opening_direction = from_left/from_right/symmetric # (Optional) Opening
                                                               # direction of the gate door(s).

[structure]
type          = pump           # Type of structure
id            = pump01         # Name of the structure
polylinefile  = pump01.pli    # *.pli; Polyline location for structure
```

capacity = pump01_cap.tim # Pumping capacity in [m ³ /s]

B.10 Space varying wind and pressure

In many cases the space varying wind data is provided by a meteorological station. This data is often defined on a different grid than the computational grid used in D-Flow FM. Translating these files into files defined on the grid of the computational engine is often a lengthy process and can result in huge files. This feature facilitates the reading of the meteorological data on its own grid and interpolates the data internally to the grid of D-Flow FM. D-Flow FM can handle three types of meteorological input:

- 1 Time- and space-varying wind on an equistant grid
- 2 Time- and space-varying wind on a curvilinear grid
- 3 Time- and space-varying wind on a Spiderweb grid

B.10.1 Meteo on equidistant grids

File contents	Time-series of a space varying wind and atmospheric pressure defined on an equidistant (Cartesian or spherical) grid.
File format	Free formatted or unformatted, keyword based.
Generated	Some offline program.

Remark:

- ◊ The keywords are case insensitive.



Header description for the wind velocity files:

Keywords	Value	Description
FileVersion	1.03	version of file format
Filetype	meteo_on_equidistant_grid	meteo input on equidistant grid
NODATA_value	free	value used for input that is to be neglected
n_cols	free	number of columns used for wind datafield
n_rows	free	number of rows used for wind datafield
grid_unit	m or degree	unit of distances on the grid in both x- and y-direction
x_llcorner	free	x-coordinate of lower left corner of lower left grid cell (in units specified in grid_unit)
y_llcorner	free	y-coordinate of lower left corner of lower left grid cell (in units specified in grid_unit)
x_llcenter	free	x-coordinate of centre of lower left grid cell (in units specified in grid_unit)
y_llcenter	free	y-coordinate of centre of lower left grid cell (in units specified in grid_unit)

Keywords	Value	Description
dx	<i>free</i>	gridsize in <i>x</i> -direction in units specified in <i>grid_unit</i>
dy	<i>free</i>	gridsize in <i>y</i> -direction in units specified in <i>grid_unit</i>
n_quantity	1	number of quantities specified in the file
quantity1	x_wind <i>or</i> y_wind	the velocity component given in unit <i>unit1</i>
unit1	m s-1	unit of quantity1: metre/second

The user must specify the location of the equidistant grid on which the meteorological data is specified. If one has the location of the lower left corner of the lower left grid cell, one can specify the starting point of the grid using keywords *x_llcorner* and *y_llcorner*. If one has the location of the cell centre of the lower left grid cell, one should use the keywords *x_llcenter* and *y_llcenter*. Using the first option, the first data value is placed at (*x_llcorner* + $\frac{1}{2}dx$, *y_llcorner* + $\frac{1}{2}dy$), which is the cell centre of cell (1,1). Using the latter option, the first data value is placed at (*x_llcenter*, *y_llcenter*), which is again the cell centre of cell (1,1), i.e. the data values are always placed at the cell centres of the meteorological grid. Note that the lower left grid cell is defined to be the grid cell with index (1,1). When using the option of meteorological data on a separate curvilinear grid, the origin and orientation of the data set can be chosen freely with respect to the grid on which it is specified, see [section B.10.2](#) for details.

Time definition and data block description for the wind velocity files

Keywords	Value	Description
Time	<i>fixed format described below</i>	time definition string

The time definition string has a fixed format, used to completely determine the time at which a dataset is valid. The time definition string has the following format:

TIME *minutes/hours since YYYY-MM-DD HH:MM:SS TIME ZONE*, e.g.

360 minutes since 2008-07-28 10:55:00 +01:00

The format of the string is completely fixed. No extra spaces or tabs can be added between the different parts of the definition. The time definition is followed by the datablock of input values corresponding to the specified time. The data block contains values for the wind velocity in *x*- or *y*-direction for *n_cols* by *n_rows* points, starting at the top left point. The time definition and the data block are repeated for each time instance of the time-series.

The atmospheric pressure file

The header for the atmospheric pressure is similar to that of the wind velocity files, except for the following differences.

Keywords	Value	Description

quantity1	air_pressure	air pressure
unit1	Pa or mbar	unit of quantity1: Pascal or millibar

The specification of the time definition and the data block is fully conform the wind velocity files.

File version and conversion

The current description holds for FileVersion 1.03. The table below shows the latest modifications in the file format (and version number).

FileVersion	Modifications
1.03	Use of keyword Value_pos to indicate the position of the lower left corner of the grid replaced by use of the combination of keywords: x_llcorner and y_llcorner or x_llcenter and y_llcenter
1.02	No changes for this meteo input type, but for the meteo type <i>meteo_on_spiderweb_grid</i>
1.01	Changed keyword MeteoType to FileType Changed fixed value of input type (Keyword Filetype) from <i>ArclInfo</i> to <i>meteo_on_equidistant_grid</i>

Restrictions:

- ◊ The contents of the file will not be checked on its domain.
- ◊ Keywords are followed by an equal sign '=' and the value of the keyword.
- ◊ When a keyword has value *free*, the value of this keyword is free to choose by the user. When only one value is given for a keyword, this keyword has a fixed value and when 2 or more options are shown, the user can choose between those values.
- ◊ Times must be specified exactly according to the time definition. See the examples shown in this section.
- ◊ The atmospheric pressure file must use the same grid definition and time frame as the files for the wind velocity components.
- ◊ The unit of the meteo grid must be the same as the computational grid, i.e. both with *grid_unit* = [m] or both with *grid_unit* = [degree].
- ◊ Input items in a data block are separated by one or more blanks.
- ◊ The wind components are specified at the cell centres (water level points) of the numerical grid.
- ◊ The wind components are specified in the west-east (*x_wind*) and south-north directions (*y_wind*).



Remarks:

- ◊ The time definition in the meteo files contains the number of minutes or hours since a reference date and time in a certain time zone. The reference time and time zone may differ from those of the simulation. During a simulation the computational engine will search in the meteo file for the current simulation time and interpolate between neighbouring times if necessary. Possible differences in time zone will be accounted for by shifting the meteo input data. The reference times within the time definition string may vary in a meteo file, i.e. it is possible to attach new input with a different reference time, behind the last data block. Consecutive times must always be increasing in the input file.
- ◊ Comments can be added after pound signs (#). These are not read.



Example of a file containing wind in x-direction (west-east)

The data blocks in this example are the result of the following FORTRAN statements:

```

do j = nrows,1,-1
    write(out,*) (xwind(i,j),i=1,ncols)
enddo

```

The *x*-wind velocity file for a 3 (*n_cols*) by 4 (*n_rows*) grid has the following layout:

```

FileVersion      =      1.03
filetype        =      meteo_on_equidistant_grid
NODATA_value   =      -999.000
n_cols          =      3
n_rows          =      4
grid_unit       =      degree
x_llcenter     =      -12.000
y_llcenter     =      48.000
dx              =      0.12500
dy              =      0.083333333
n_quantity      =      1
quantity1       =      x_wind
unit1           =      m s-1
TIME =      0.0 hours since 2008-01-15 04:35:00 +00:00
2   3.0  3.6
3   4.5  2
2.2 1   2.3
1.2 0.7 -0.4
TIME =      6.0 hours since 2008-01-15 04:35:00 +00:00
-1.1 -2.3 -3.6
-3.2  0.8  1.1
2.2 -1   -1.6
1.2 -0.7 -0.4

```

This results in an *x*-component of wind velocity given - in [m/s] - on a spherical, 3 by 4, equidistant grid, with grid sizes given by *dx* and *dy* (in degrees) and where the centre point of the lower left cell of the grid lies in (longitude, latitude) (-12.0, 48.0) on the globe. Data is given at two times: 0 and 6 hours since January 15th, 2008, 4:35 AM, in UTC+0.



Remark:

- ◊ The layout of the data blocks is from north to south (whereas most of the other files in Delft3D-FLOW, such as the curvilinear grid file, are ordered from south to north).

B.10.2 Curvilinear data

File contents	Time-series of a space varying wind and atmospheric pressure defined on a curvilinear (Cartesian or spherical) grid.
File format Generated	Free formatted or unformatted, keyword based. Some offline program.



Remark:

- ◊ The keywords are case insensitive.

Header description for the wind velocity files:

Keywords	Value	Description
FileVersion	1.03	version of file format
Filetype	meteo_on_curvilinear_grid	meteo input on curvilinear grid
NODATA_value	free	value used for input that is to be neglected

Keywords	Value	Description
grid_file	free.grd	name of the curvilinear grid file on which the data is specified
first_data_value	grid_llcorner or grid_ulcorner or grid_lrcorner or grid_urcorner	<i>see example below</i>
data_row	grid_row or grid_column	<i>see example below</i>
n_quantity	1	number of quantities specified in the file
quantity1	x_wind or y_wind	the velocity component given in unit unit1
unit1	m s-1	unit of quantity1: metres/second

Time definition and data block description for the wind velocity files

For a description of the time definition and data block see [section B.10.1](#).

The atmospheric pressure file

For a description of the atmospheric file see [section B.10.1](#).

File version and conversion

The current description holds for FileVersion 1.03. The table below shows the latest modifications in the file format (and version number).

FileVersion	Modifications
1.03	Fixed bug in interpolation of data from meteo grid to computational grid: Conversion script mirrored data set erroneously. This was treated correctly by meteo module. Fixed both the conversion script and the meteo module together: Required modification in meteo input file: Change first_data_value = grid_llcorner into grid_ulcorner or vice versa or Change first_data_value = grid_lrcorner into grid_urcorner or vice versa
1.02	No changes for this meteo input type, but for the meteo type <i>meteo_on_spiderweb_grid</i>
1.01	Changed keyword MeteoType to FileType Changed keyword Curvi_grid_file to Grid_file Changed fixed value of input type (Keyword Filetype) from Curvi to <i>meteo_on_curvilinear_grid</i>

Restrictions:

- ◊ The restrictions for space varying wind and pressure on a separate curvilinear grid are the same as for space varying wind and pressure on an equidistant grid, described in [section B.10.1](#). A difference is that the data values on the curvilinear grid are not specified in the cell centres,



but in the grid points (cell corners).

- ◊ The unit of the meteo grid must be the same as the computational grid, i.e. both with `grid_unit` = [m] or both with `grid_unit` = [degree].



Remark:

- ◊ The remarks for space varying wind and pressure on a separate curvilinear grid are the same as for space varying wind and pressure on an equidistant grid, described in [section B.10.1](#).

Example:

A file for input of *x*-velocity (in west-east direction) on a 4 by 5 curvilinear grid, where the meteorological data is mirrored vertically with respect to the grid, has the following layout:

```
FileVersion      = 1.03
filetype        = meteo_on_curvilinear_grid
NODATA_value   = -999.000
grid_file       = curviwind.grd
first_data_value = grid_llcorner
data_row        = grid_row
n_quantity     = 1
quantity1      = x_wind
unit1          = m s-1
TIME = 0.0 minutes since 1993-06-28 14:50:00 -02:00
 1 2 3 4 5
 6 7 8 9 10
11 12 13 14 15
16 17 18 19 20
TIME = 600.0 minutes since 1993-06-28 14:50:00 -02:00
 1 2 3 4 5
 6 7 8 9 10
11 12 13 14 15
16 17 18 19 20
```

This results in an *x*-component of velocity given - in [m/s] - on the curvilinear grid specified in file <curviwind.grd>. The data set will be mirrored such that the first value of the data (upper left corner, in the example '1') corresponds to the lower left corner of the grid (point (1,1)) and a row of data corresponds to a row on the grid, see [Figure B.1](#). Data is given at two times: 0 and 600 minutes since June 28th, 1993, 14:50 PM, in UTC-2.

B.10.3 Spiderweb data

Cyclone winds are governed by a circular motion, combined with a cyclone track. This type of wind is generally very difficult to implement on a curvilinear grid. This feature facilitates the reading of the so-called Spiderweb files and interpolates the wind and pressure data internally to the computational grid. A special feature of the space varying wind and pressure on the Spiderweb grid is that it can be combined with one of the other meteorological input options described in this manual, i.e. to either uniform wind and pressure, or to one of the space varying wind and pressure options, see [section B.10](#).

File contents	Time-series of a space varying wind and atmospheric pressure defined on a Spiderweb grid. This grid may be specified in Cartesian or spherical coordinates.
File format	Free formatted or unformatted, keyword based.
Generated	Some offline program.



Remarks:

- ◊ The keywords are case insensitive.
- ◊ Space varying wind and pressure on a Spiderweb grid is added to other wind input and the wind fields are interpolated and combined in and around the cyclone.

Header description of the Spiderweb wind and pressure file:

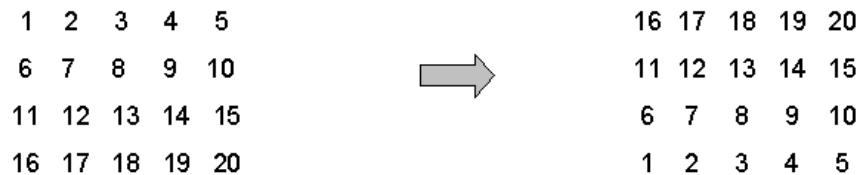
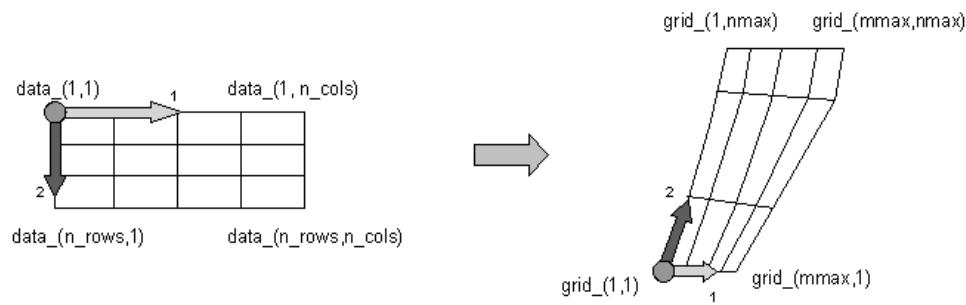


Figure B.1: Illustration of the data to grid conversion for meteo input on a separate curvilinear grid

Keywords	Value	Description
FileVersion	1.03	version of file format
Filetype	meteo_on_spiderweb_grid	meteo input on Spiderweb grid
NODATA_value	<i>free</i>	value used for input that is to be neglected
n_cols	<i>free</i>	number of gridpoints in angular direction
n_rows	<i>free</i>	number of gridpoints in radial direction
grid_unit	m or degree	unit of the Spiderweb grid
spw_radius	<i>free</i>	radius of the spiderweb given in units given by spw_rad_unit
spw_rad_unit	m	unit of the Spiderweb radius
spw_merge_frac	[0.0,1.0]	fraction of the Spiderweb radius where merging starts of the background wind with the Spiderweb wind. Default is 0.5
air_pressure_reference	air_pressure_default_from_computational_engine or <i>free</i>	Both keyword and value are too long to fit on one line. Reference value related to p_drop is the default air pressure of the computational engine or the value specified. If missing, p_drop is extracted from the actual atmospheric pressure.

Keywords	Value	Description
n_quantity	3	number of quantities specified in the file
quantity1	wind_speed	wind speed given in unit unit1
quantity2	wind_from_direction	direction where the wind is coming from given in unit unit2
quantity3	p_drop	drop in atmospheric pressure given in unit unit3
unit1	m s-1	unit of quantity1: metres/second
unit2	degree	unit of quantity2: degrees
unit3	Pa or mbar	unit of quantity3: Pascal or millibar

Time definition and data block description

For a description of the time definition see [section B.10.1](#).

Cyclone track information:

For each time in the time series of space varying wind and pressure on a Spiderweb grid, the position of the cyclone eye (and thus also the spiderweb grid) must be given, as well as the drop of atmospheric pressure in the cyclone eye.

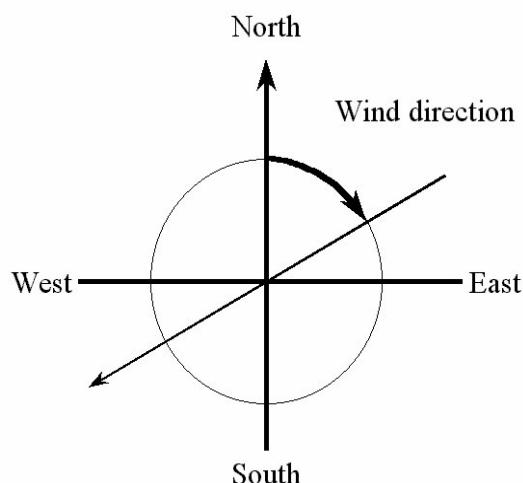


Figure B.2: Wind definition according to Nautical convention

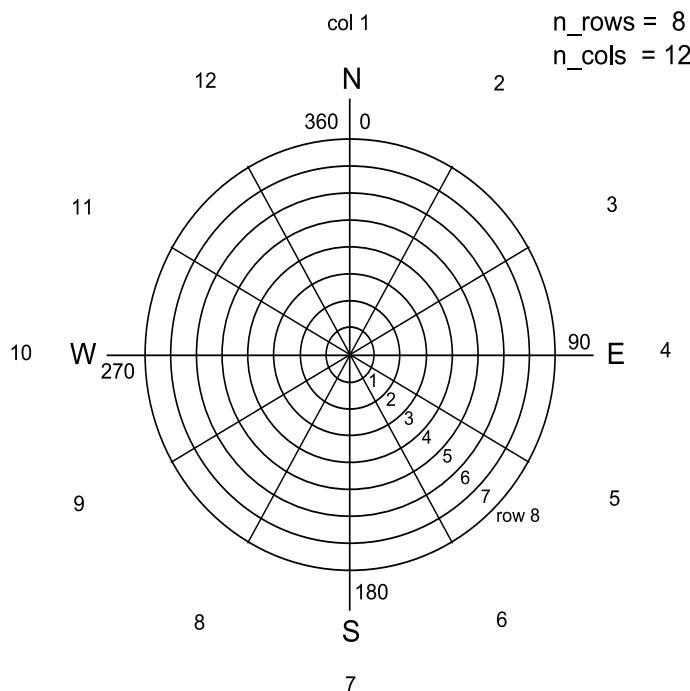


Figure B.3: Spiderweb grid definition

File version and conversion

The current description holds for FileVersion 1.03. The table below shows the latest modifications in the file format (and version number).

FileVersion	Modifications
1.03	No changes for this meteo input type
1.02	Changed the use of keyword <code>n_rows</code> and <code>n_cols</code> . The radius of the cyclone is divided in <code>n_rows</code> rings of width: spw_radius/n_rows [m] and the circle is divided in <code>n_cols</code> parts of $2\pi/n_cols$ [rad].
1.01	Changed keyword MeteoType to FileType Changed fixed value of input type (Keyword filetype) from <i>Spiderweb</i> to <i>meteo_on_spiderweb_grid</i>

Restriction:

- ◊ The restrictions for space varying wind and pressure on a Spiderweb grid are the same as for space varying wind and pressure on an equidistant grid, described in [section B.10.1](#).



Remarks:

- ◊ The remarks for space varying wind and pressure on a separate curvilinear grid are the same as for space varying wind and pressure on an equidistant grid, described in [section B.10.1](#).
- ◊ The Spiderweb grid is circular and the definitions of the number of rows `n_rows` and the number of columns `n_cols` is therefore different than for the other meteo input formats. For the Spiderweb grid, the number of rows determines the grid size in radial direction. The number of columns defines the grid size in angular direction. See [Figure B.3](#).
- ◊ The wind is specified according to the nautical convention, i.e. wind from the true North has direction zero and the wind turns clockwise with an increasing angle. See [Figure B.2](#).



Example:

A file for input of space varying wind and pressure on a 5x3 Spiderweb grid, has the following layout:

```
FileVersion      = 1.03
filetype        = meteo_on_spiderweb_grid
NODATA_value   = -999.000
n_cols          = 3
n_rows          = 5
grid_unit       = degree
spw_radius     = 600000.0
spw_rad_unit   = m
air_pressure_reference = air_pressure_default_from_computational_engine
n_quantity      = 3
quantity1       = wind_speed
quantity2       = wind_from_direction
quantity3       = p_drop
unit1           = m s-1
unit2           = degree
unit3           = Pa
TIME            = 0.0 hours since 1997-07-14 03:00:00 -06:00
x_spw_eye       = 115.1
y_spw_eye       = 18.9
pdrop_spw_eye  = 5300.0
1.38999 1.38261 1.38315
1.28251 1.34931 1.22571
1.27215 1.31214 1.32451
1.38999 1.86592 2.87732
1.43899 1.24912 2.21519
60.0000 180.0000 270.0000
28.7500 20.0000 31.2500
42.5000 53.7500 65.0000
49.3400 60.2400 81.5200
51.4100 62.0000 43.1200
5301.280 5294.490 5156.240
5043.460 5112.040 5264.020
5140.020 5202.520 5411.210
5294.730 5285.760 5235.250
5242.530 5156.190 5124.240
TIME            = 1.0 hours since 1997-07-14 03:00:00 -06:00
x_spw_eye       = 114.8
y_spw_eye       = 18.8
pdrop_spw_eye  = 5250.0
1.35763 1.35763 1.35763
1.35763 1.87273 2.24784
1.92214 2.47836 2.17266
1.87662 2.72116 2.82375
1.26585 2.24146 2.38722
159.0000 346.5200 290.6400
342.3200 282.1400 20.2400
10.7500 25.5300 36.4500
61.8400 81.6200 45.5100
49.5250 56.7500 75.1300
5314.520 5104.490 5287.240
5124.240 5285.760 5252.420
5152.460 5247.040 5222.020
5242.020 5223.520 5475.210
5244.270 5211.210 4998.110
```

This results in the following set of meteo data. Velocities given in [m/s] and pressure drops in [Pa] on a Spiderweb grid which is given in spherical coordinates (grid_unit = degree). The cyclone and spiderweb grid have a radius of 600 km. The grid is 5x3, which means the radius is divided in five parts of 120 km and the 360 degrees are divided in 3 parts of 120 degrees each. Wind speeds, wind directions and pressure drops are given at two times: 0 and 1.0 hour since July 14th, 1997, 03:00 AM,

in UTC-6. Between these two times the cyclone eye moves from (longitude, latitude) (115.1, 18.9) to (114.8, 18.8) on the globe and the pressure drop in the cyclone eye decreases from 5300.0 [Pa] to 5250.0 [Pa].

C Initial conditions and spatially varying input

C.1 Introduction

Spatially varying input is input data that varies in space, not in time, such as initial conditions (water levels, etc.), or spatially varying model parameters (friction coefficients, etc.). D-Flow FM uses input data in *model-independent coordinates*, that is, the input files should contain their own spatial coordinates, and do not correspond with the D-Flow FM grid cell numbering. As a result, the model grid can always be changed, without the need to change all other spatial input. The spatial input will be interpolated onto the active model grid. Spatial input has to be specified in the ext-file ([section B.5](#)). All supported quantities are listed in [Table B.1](#) under “Initial fields” and “Spatial physical properties”.

Initial conditions can also be set using a restart-file, which assigns the exact values in the file to the current model grid. No interpolation is performed, and the restart file should correspond exactly with the current model and its grid.

C.2 Supported quantities

C.2.1 Water levels

Water levels can come from both all file types listed below. The restart file sets the water level both at the old and new time step. Non-restart files should be specified using QUANTITY=initialwaterlevel.



C.2.2 Initial velocities

Initial velocities should come from restart files, such that they exactly correspond with the model's waterdepths. They can also come from non-restart files using QUANTITY=initialvelocityx/y, but this is not advised.

C.2.3 Salinity

Salinity concentrations from non-restart files are possible as spatially varying, depth-averaged values, or (for 3D models) as linearly interpolated values between a top layer concentration and a bottom layer concentration. The latter is achieved by specifying two QUANTITY blocks: initialsalinity and initialsalinitytop. For 3D models, alternatively, spatially constant, but depth-varying salinities can be set using the QUANTITY=initialverticalsalinityprofile, and a vertical profile file (see [section C.3.3](#)).

C.2.4 Temperature

Temperature values from non-restart files are possible as depth-averaged values, using QUANTITY=initialtemperature.

For 3D models, alternatively, spatially constant, but depth-varying temperatures can be set using the QUANTITY=initialverticaltemperatureprofile, and a vertical profile file (see [section C.3.3](#)).

C.2.5 Tracers

Initial tracer concentrations from non-restart files are similar to temperature values, now specified using QUANTITY=initialtracer. The set of tracer name(s) is not listed in the MDU-file explicitly, they are inferred from all supplied initial tracer definitions *and* the tracer boundary conditions (see also [section D.1.7](#)). The <tracername> postfix can uniquely associate an initialtracer quantity with a tracerbnd quantity.

C.2.6 Sediment

Initial sediment concentrations for the Delft3D-SED-based sediment transport module (see [section 8.2](#) for an explanation) is via the <.sed> file. See the Delft3D-SED User Manual for further details.

C.2.7 Physical coefficients

A second group of spatially varying input are the physical and numerical coefficients listed below. They allow changing model parameters in space or in certain sub-regions of the domain, and as such these values override the uniform values from the MDU file. The input data can come from the in-polygon file type ([section C.3.1](#) or the sample file type ([section C.3.2](#)).

quantity name	MDU uniform equivalent	description
frictioncoefficient	UnifFrictCoef	Friction coefficient (use IFRCTYP and UnifFrictType, resp. to set the friction type).
horizontaleddyy... ...viscositycoefficient	Vicouv	Horizontal eddy viscosity coefficient (m ² /s).
horizontaleddyy... ...diffusivitycoefficient	Dicouv	Horizontal eddy diffusivity coefficient (m ² /s).
advectiontype	AdvecType	Type of advection scheme.
ibotlevtype	BedlevType	Type of bed-level handling.

C.3 Supported file formats

All file formats below, except for restart files, are in *model-independent coordinates* and will be cropped and/or interpolated onto the model grid.

C.3.1 Inside-polygon option

A spatially constant value in a restricted part of the domain can be set by specifying a polygon file ([section B.2](#)) in the ext-file as follows:

```
QUANTITY=frictioncoefficient
FILENAME=winterbed.pol
FILETYPE=10
METHOD =4
VALUE   =55
IFRCTYP =1 # Optional, override uniform [physics] UnifFrictType=..
```

The interpolation option METHOD=4 simply specifies the no-interpolation is to be performed, only inside-polygon cropping. For QUANTITY=frictioncoefficient the default friction type may be overridden with the keyword IFRCTYP.

C.3.2 Sample file

Spatial data from sample files (<*.xyz>, see [section B.3](#)) is interpolated by triangle interpolation or sample averaging. The following options control the type of interpolation:

interpolation method	options	description
METHOD=5	EXTRAPOLTOL=20	Triangle interpolation Allow extrapolation from convex hull of samples up to ... meters (default: 0).
METHOD=6	AVERAGINGTYPE=1 AVERAGINGTYPE=2 AVERAGINGTYPE=3 AVERAGINGTYPE=4 AVERAGINGTYPE=5 AVERAGINGTYPE=6 RELATIVESEARCHCELLSIZE =...	Sample averaging inside control volumes: Normal (unweighted) averaging Nearest neighbour Maximal value Minimal value Inverse weighted distance averaging Minimal absolute value Control size of search volume: 1=actual cell, 1.5=50% larger in all directions.

C.3.3 Vertical profile file

Initial values that are constant in 2D space, but vary in depth can be specified using vertical profile files, which are basic polyline files ([section B.2](#)). For example for a linearly varying salinity from 30 to 20 ppt from -10 m to 0 m :

```
L1
2 2
-10 30
0 20
```

The METHOD keyword must be specified, but is further ignored: linear interpolation is always used.

C.3.4 Map file

Initial field data from a map file may be used as non-restart input. Values are read in from the map files as if they were samples, and further treated and interpolated as such. All interpolation options from [section C.3.2](#) can be used here. This option may be used as a non-exact restart state when the model grid has been changed (e.g., locally refined or cropped to a subdomain).

C.3.5 Restart file

Restart files are discussed in [section E.3.3](#). They should be included in your model in the mdu-file as follows:

```
[restart]
RestartFile      = mdident_yyyymmdd_HHMMSS_RST.nc
RestartDateTime  = # Restart time (YYYYMMDDHHMMSS), only relevant
                  # in case of restart from *_map.nc
```


D Boundary conditions specification

Boundary conditions are part of the external forcing of a model and as such declared in the external-forcings file ([section B.5](#)).

D.1 Supported boundary types

D.1.1 Open boundaries

[yet empty: Riemann boundaries?]

D.1.2 Astronomic boundary conditions

Boundary values can be specified for any location in terms of astronomical components in attribute files with extension <cmp> (the BC-format, discussed furtheron, is supported as an alternative). Tidal motion are described as a series of simple harmonic constituent motions, each with its own characteristic period:

$$H(t) = A_0 + \sum_{i=1}^k A_i F(c_i) \cos \left(\frac{2\pi}{T(c_i)} t + (V_0 + u)(c_i) - G_i \right) \quad (\text{D.1})$$

in which:

c_i	i -th component, specified by label (name)
A_i	amplitude of the i -th component
G_i	phase of the i -th component
$T(c_i)$	period
$H(t)$	boundary value at time t
A_0	constant offset value
k	number of relevant components
$F(c_i)$	nodal amplitude factor
$(V_0 + u)(c_i)$	astronomical argument

The component (by label) c_i , amplitude A_i and phase G_i for each component i are required in the <cmp>-file. In addition to the primary constituents, compound and higher harmonic constituents may have to be used. This is the case in shallow water areas for example, where advection, large amplitude to depth ratio, and bottom friction give rise to non-linear interactions.

F , V_0 and u are also *time-dependent*. For a given period, their values are easily calculated or obtained from various tidal year books. V_0 is a frequency dependent phase correction factor relating the local time frame of the observations to an internationally agreed celestial time frame. F and u are slowly varying amplitude and phase corrections. The variations depend on the frequency, often with a cyclic period of 18.6 years. By default, the nodal amplitude factors and astronomical arguments are re-calculated every 6 hours.

D.1.3 Astronomic correction factors

For individual astronomic components, correcting factor α_i for amplitude and offset δ_i may be specified by the user so that the effective amplitude for that component becomes $\alpha_i A_i$ and the effective phase becomes $G_i + \delta_i$.

D.1.4 Harmonic flow boundary conditions

Harmonic boundary conditions are specified in a manner similar to their astronomical counterpart, except that the periods (first column of a .cmp file) are declared explicitly, rather than using component names. Equation (D.1) applies with $T(c_i) = T_i$ and furthermore $F = 1$ and $(V_0 + u) = 0$.

D.1.5 QH-relation flow boundary conditions

In this type of boundary condition, a waterlevel is prescribed as a function of the integrated discharge through a crossection defined by the polyline of the boundary. The relation between waterlevel and discharge is specified in a lookup-table. Within this table linear interpolation is applied. Note that only one waterlevel is set for the entire QH-boundary, and that its behaviour is specified by a *single* QH-table for the entire boundary.

D.1.6 Time-series flow boundary conditions

Time-series in the general time-series file format () can be used to specify values on boundaries. Boundary values are provided at discrete points in time, which are then used for interpolation at times in between. Extrapolation beyond the range of specified times is not supported, Time-series are also supported at multiple vertical levels for quantities that are defined on layers (tracers, salinity, temperature, velocity). These need to be specified in the BC-format ()�.

D.1.7 Time-series transport boundary conditions

The boundary conditions of tracers, salinity and temperature are specified similar to any other quantity already mentioned. The quantity name should be in the form tracerbnd<name>, where name is the user-defined tracer name.

D.1.8 Time-series for the heat model parameters

In order to conduct a heat flux simulation (if in the MDU-file “Temperature” under [physics] was set to 5), then four time-varying meteorological fields become relevant:

- ◊ humidity
- ◊ airtemperature
- ◊ cloudiness
- ◊ solar radiation

These are read as a three- or four-column vector (the first three variables of this list, either with or without solar radiation). The corresponding quantity names in the <ext>-file are:

- ◊ humidity_airtemperature_cloudiness
- ◊ humidity_airtemperature_cloudiness_solarradiation

As for filetype, currently only the curvilinear format ([B.10.2](#)) and the multicolmnn time series ([D.1.6](#)) are supported.

D.2 Boundary signal file formats

D.2.1 The <cmp> format

```
* comments
* ...
c[0]      amplitude[0]    phase[0]
c[1]      amplitude[1]    phase[1]
...
```

where c represents a period in minutes or the name of a valid astronomic component. Amplitudes are assumed to be in the unit of the quantity.

D.2.2 The <qh> format

```
* comments
* ...
discharge[0]      waterlevel[0]
discharge[1]      waterlevel[1]
...
```

D.2.3 The <bc> format

The BC-format for boundary signal data was introduced to combine multiple quantities for multiple locations in a single (ASCII) file. Each location will be defined in a separate 'BC block', which consists of a column-wise table, (the data), and a header specifying how this table should be interpreted.

The BC-format is pseudo-formally defined below, using EBNF¹-notation. Though all BC keywords in this description are uppercase, nothing in the BC-format is case-sensitive, not even user-defined names. Whitespace is optional, except for the new lines as in the definitions below.

```
BC-file = { BC-block }

BC-block = "[FORCING]" ,
           "FUNCTION = " , function ,
           "NAME = ",location ,
           ["OFFSET = " , offset ] ,
           ["FACTOR = " , factor ] ,
           ["VERTICAL POSITION SPECIFICATION = " , {vertical-position} ,
            "VERTICAL INTERPOLATION = " , vertical-interpolation ,
            "VERTICAL POSITION TYPE = " , vertical-position-type] ,
           "MISSING VALUE DEFINITION = " , float ,
           "TIME INTERPOLATION = " , time-interpolation,
           { quantity-block | vector-block },
           { data-block }
```

```
function = "TIMESERIES" | "HARMONIC" | "ASTRONOMIC" | "HARMONIC-CORRECTION" |
           "ASTRONOMIC-CORRECTION" | "T3D" | "QHTABLE"
```

The FUNCTION attribute specifies how the BC-block should be interpreted. The options are:

"TIMESERIES"	Value(s) as a function of time
"HARMONIC"	Period, amplitude, phase
"ASTRONOMIC"	Component-name, amplitude, phase
"HARMONIC-CORRECTION"	Period, amplitude-factor, phase-offset
"ASTRONOMIC-CORRECTION"	Component-name, amplitude-factor, phase-offset
"T3D"	Values on multiple vertical positions versus time
"QHTABLE"	Discharge as a function of waterlevel

A single BC-block is uniquely tied to a single boundary location by the location value:

```
location = pli-name , [ "_" , pointnumber]
```

The pli-name pointnumber combination identifies the BC-block (when searched). The pointnumber should be omitted for blocks providing values for an entire polyline. Currently this is *only* the case for blocks with the function "QHTABLE", which apply to *all* points on a polyline,

¹https://en.wikipedia.org/wiki/Extended_Backus%20%93Naur_Form

```
vertical-interpolation = "LINEAR" | "LOG" | "BLOCK"
time-interpolation = "LINEAR" | "BLOCK-TO" | "BLOCK-FROM"
```

"LINEAR"	linear interpolation between times or vertical positions.
"LOG"	logarithmic interpolation between vertical positions (e.g. vertical velocity profiles).
"BLOCK"	equal to the value at the directly lower specified vertical position.
"BLOCK-FROM"	equal to that at the start of the time interval (latest specified time value).
"BLOCK-TO"	equal to that at the end of the time interval (upcoming specified time value) .

```
vertical-position-type = "PERCBED" | "ZBED"
```

- "PERCBED" Percentage wrt. waterdepth from the bed upward (sigma layers).
 "ZBED" Absolute distance from the bed upward (z-layers).

vertical-position

The vertical position is specified as a space- or comma-separated list of floats, the interpretation of which varies according to the vertical position type. The order of the positions in the list becomes relevant when referring to them from the quantity blocks. However, no specific ordering of these positions (ascending or descending) is assumed.

```
quantity-block = "QUANTITY = ", quantity-name
               {quantity-property}

quantity-property = | "VERTICAL POSITION = ", vertical-position-index |
                     "UNIT = " unit |

vector block = vectorname, ":" , [{quantity-name}], "]", quantity-name
data-block = {column-data}
```

quantity-name (character string)

The quantity name, in conjunction with the BC-block name (representing location) serves as a searching key in the BC-file to find the BC-block and column within that block. These names can be chosen freely, except that some names and combinations have a special meaning.

1. a quantity *bndname* in a block with function "HARMONIC" or "HARMONIC-CORRECTION" should have the quantities
 - HARMONIC COMPONENT
 - bndname* AMPLITUDE
 - bndname* PHASE
2. a quantity *bndname* in a block with function "ASTRONOMIC" or "ASTRONOMIC-CORRECTION" should have the quantities
 - ASTRONOMIC COMPONENT
 - bndname* AMPLITUDE
 - bndname* PHASE
3. Blocks with function "TIMESERIES" or "T3D" require a quantity
 - TIME

This also implies that the component-quantity in harmonic and astronomic BC-blocks applies to any quantity in that block. The same holds for the time-quantity in timeseries and T3D blocks.

vertical-position-index (integer)

This is a (one-based) index into the vertical-position-specification, assigning a vertical position to the quantity (T3D-blocks only)

offset, factor (floating points)

These are set for the complete BC-block. In the case of timeseries (function is "TIMESERIES" or "T3D"), all values in the table are multiplied with the factor and increased by the offset. In the case of astronomic or harmonic components (function is "HARMONIC" or "ASTRONOMIC"), only the amplitude column is multiplied by the given factor.

datablock

This block holds the actual data in columns. The interpretation of the columns is fully determined by the order of the quantity-blocks in the header, i.e. the n -th quantity-block in the header refers to the n -th column. Therefore, The columns should contain valid floating point numbers (scientific notation is also allowed). The only exception is the column associated with astronomic component names, which is expected to contain strings.

D.2.4 The NetCDF-format for boundary condition time-series

Time series data for 2D boundaries can also be ingested from NetCDF files. These files should meet the following structure:

- ◊ To look up boundary signal locations with a certain label, a character variable should be defined with attribute `cf_role = 'timeseries_id'`. This variable should have two dimensions. The first dimension corresponds with the different boundary signal locations in the file. The contents are interpreted as location identifiers of support points of the boundaries.
- ◊ The variables containing the timeseries for the support points should have the attribute `standard_name = <quantity-name>`, so as to associate the variable with a requested quantity. The first dimension corresponds with the different boundary signal locations in the file. The second dimension corresponds with time and is the only dimension that can be 'UNLIMITED'.

So, similar to the (ASCII) BC-format, location name and quantity name together determine the time-series to be read, only now

- ◊ the quantity name is sought among the variables standard-name attributes.
- ◊ the location name is sought in a separate list of timeseries IDs, of which
 - the first dimension matches the first dimension of the 'data'-variables.
 - the role as 'timeseries_id' is signified through the `cf_role` attribute.

E Output files

D-Flow FM can produce several types of output into several different files. This chapter summarizes the types of output and how to configure them in them model definition. We distinguish four types of output here:

- 1 The diagnostics file, a log file with live details of a single model run.
- 2 NetCDF output files for history, map and restart data.
- 3 Tecplot
- 4 Timings file with performance statistics

E.1 Diagnostics file

The diagnostics (dia) file is the log file of a single model simulation run. It is an important file to check for warning or error messages, when suspecting a faulty model run. The filename is automatically chosen as <mdident.dia>. For parallel model runs, each process writes its own file <mdident_000X.dia>. The dia file has the following global structure:

```
Various INFO/DEBUG messages:  
** INFO : Opened file : unstruc.hlp  
[..]  
The version of D-Flow FM used for this calculation:  
* Deltares, D-Flow FM Version 1.1.145.41271M, Aug 11 2015, 18:05:31  
Date and time of model run start:  
File creation date: 18:27:44, 06-09-2015  
[..]  
  
Check for possible model initialization errors:  
** WARNING: readMDUFile: [numerics] cflwavefrac=0.1 was in file, but not used.  
[..]  
  
Upon successful initialization, a full printout of the effective model settings is given:  
** INFO : ** Model initialization was successful **  
** INFO : * Active Model definition:  
# Generated on 18:27:45, 06-09-2015  
# Deltares, D-Flow FM Version 1.1.145.41271M, Aug 11 2015, 18:05:31  
[model]  
[..]  
  
Next, the time loop is started:  
** INFO : **  
** INFO : Writing initial output to file(s)...  
** DEBUG : Opened NetCDF file 'DFM_OUTPUT_035hammen/035hammen_his.nc' as #3.  
** DEBUG : Opened NetCDF file 'DFM_OUTPUT_035hammen/035hammen_map.nc' as #4.  
** INFO : Done writing initial output to file(s).  
[..]  
  
Finishing summary of model run:  
** INFO : simulation period (s) : 6400.0000000000  
** INFO : nr of timesteps ( ) : 771.0000000000  
** INFO : average timestep (s) : 8.3009079118  
[..]  
Some basic run time statistics  
** INFO : time steps (s) : 76.2840000003  
** INFO : Computation started at: 18:27:45, 06-09-2015  
** INFO : Computation finished at: 18:29:04, 06-09-2015  
** INFO :  
Which parallelization options have been active in this run:  
** INFO : MPI : no.  
** INFO : OpenMP : yes. #threads max : 8
```



E.2 Demanding output

The configuration of what output should be produced during a model run is via the MDU file, and several attribute files for history files.

E.2.1 The MDU-file

The MDU-file has an aptly named [output]-section, with a range of key-value-pair options. See [Table A.1](#) on page [307](#) for all available output options.

E.2.2 Observation points

Observation points in a single observation point file <*.xyn> define the locations for time series output for single grid cells. The observation point file is basically like a sample file ([section B.3](#)), but now with station id strings in the third column:

```
-2.06250000e+00 5.71583333e+01 ABDN
-2.73750000e+00 5.15083334e+01 AVMH
-2.11250000e+00 4.91750000e+01 'St Helier Jersey'
```

! **Remarks:**

- ◊ The observation points are stationary by default.
- ◊ The x,y -locations are snapped to the grid cell in which they lie. When an observation point lies on a grid cell edge, snapping results are unpredictable.
- ◊ Time series are reported for the cell-centered solution data, that is: *no* interpolation to the exact x,y -location takes place.
- ◊ Observation points may have non-unique names (also together with moving observation points), but the results can only be distinguished by the column number in the output file, and this practice is not advised.
- ◊ The station id/name may contain spaces, in which case it should be surrounded by single quotes.

E.2.3 Moving observation points

Moving observation points are specified via the <.ext>-file:

```
QUANTITY=movingstationtxy
FILENAME=movingstation1.tim
FILETYPE=1
METHOD=1
OPERAND=0
```

The <*.tim> file (one for each moving observation point) is a standard time series file ([section D.1.6](#)) with three columns, containing the time in minutes since the reference data, the x - and y -position of the moving station at that time.

! **Remarks:**

- ◊ Stationary and moving observation points form one large set of observation points in the output his file.
- ◊ No space, nor time interpolation is done: the reported values are instantaneous values for the grid cell in which the moving observation point lies at that each output time.

E.2.4 Cross-sections

Cross-sections are polylines in a single <_crs.pli> polyline file ([section B.2](#)). Cross section polylines are snapped to the grid cell edges, and cumulative, space-integrated discharges and constituent transport are reported as time series.

Remarks:

- ◊ The first header line of each polyline block contains the name of each cross section.
- ◊ The cross section polyline is snapped to all grid cell edges (i.e., velocity points), whose flow link is intersected by the polyline.
- ◊ The flow data on velocity points is integrated along the polyline and across all vertical layers, resulting in a single time series per cross section and flow quantity.



E.3 NetCDF output files

All flow data output produced by D-Flow FM is written in the NetCDF format. This cross-platform storage format is widely used in the world. The contents of a file can be documented using variable attributes and standardized meta data. For this purpose the D-Flow FM output files aim to satisfy the CF-conventions.

E.3.1 Timeseries as NetCDF his-file

The history file <mdident_his.nc> contains the dimensions and variable definitions in its header, followed by the actual data. For parallel runs, all data is gathered by domain #0 into <mdident_0000_his.nc> ([section 5.2.5](#)).

Dimensions

The following dimensions are defined in a his file header:

```
netcdf weirfree_his {
dimensions:
    time = UNLIMITED ; // (64 currently)
    name_len = 64 ;
    stations = 18 ;
    cross_section = 4 ;
    cross_section_name_len = 64 ;
    cross_section_pts = 3 ;
    npumps = 2 ;
    // and possibly nweirgens, nweirs, ngategens, ngates
```

Location variables

The original location and ids for stations and cross sections is also included in the his file:

```
variables:
    double station_x_coordinate(stations) ;
    station_x_coordinate:units = "m" ;
    station_x_coordinate:standard_name = "projection_x_coordinate" ;
    station_x_coordinate:long_name = "x" ;
    double station_y_coordinate(stations) ;
    station_y_coordinate:units = "m" ;
    station_y_coordinate:standard_name = "projection_y_coordinate" ;
    station_y_coordinate:long_name = "y" ;
    char station_name(stations, name_len) ;
    station_name:cf_role = "timeseries_id" ;
    station_name:long_name = "Observation station name" ;
```

```

double cross_section_x_coordinate(cross_section, cross_section_pts) ;
cross_section_x_coordinate:units = "m" ;
cross_section_x_coordinate:standard_name = "projection_x_coordinate" ;
cross_section_x_coordinate:long_name = "x" ;
double cross_section_y_coordinate(cross_section, cross_section_pts) ;
cross_section_y_coordinate:units = "m" ;
cross_section_y_coordinate:standard_name = "projection_y_coordinate" ;
cross_section_y_coordinate:long_name = "y" ;
char cross_section_name(cross_section, cross_section_name_len) ;

```

Variables on stations

Variables on observations station are typically defined as follows:

```

double waterlevel(time, stations) ;
waterlevel:standard_name = "sea_surface_height" ;
waterlevel:long_name = "Water level" ;
waterlevel:units = "m" ;
waterlevel:coordinates = "station_x_coordinate station_y_coordinate station_name" ;
waterlevel:_FillValue = -999. ;

```

Other quantities that can be written are:

- ◊ Water depth
- ◊ Flow velocity (x and y)
- ◊ Salinity, temperature and other transported quantities, such as tracers

Variables on cross sections

Variables on cross sections are typically defined as follows:

```

double cross_section_discharge(time, cross_section) ;
cross_section_discharge:units = "m^3/s" ;
cross_section_discharge:coordinates = "cross_section_name" ;

```

Other quantities that can be written are:

- ◊ Time-integrated discharge since Tstart;
- ◊ Time-averaged discharge since Tstart;
- ◊ Cross section flow area at current time;
- ◊ Time-averaged cross section flow area since Tstart;
- ◊ Cross section average velocity at current time;
- ◊ Time-averaged cross section average velocity since Tstart;
- ◊ Time-integrated constituent flux since Tstart (for salinity, and other transported quantities, such as tracers);

Mass balance output

The history file may also contain model-global, time-integrated mass balance output, for example:

```
double WaterBalance_total_volume(time) ;
    WaterBalance_total_volume:units = "m3" ;
```

Other quantities that can be written are:

- ◊ Net storage since Tstart;
- ◊ Volume error since Tstart;
- ◊ Cumulative inflow through boundaries since Tstart;
- ◊ Cumulative outflow through boundaries since Tstart;
- ◊ Net inflow through boundaries since Tstart;
- ◊ Cumulative inflow via SOBEK-D-Flow FM 1D–2D boundaries since Tstart;
- ◊ Cumulative outflow via SOBEK-D-Flow FM 1D–2D boundaries since Tstart;
- ◊ Net inflow via SOBEK-D-Flow FM 1D–2D boundaries since Tstart;
- ◊ Total precipitation volume since Tstart;
- ◊ Net inflow via source–sink elements since Tstart;

Hydraulic structure output

A history file can also contain automatic output for hydraulic structures, without the need to add an explicit cross section at the same location. An example variable is:

```
double weirgen_discharge(time, weirgens) ;
    weirgen_discharge:long_name = "Weir accumulated discharge (via general structure)" ;
```

Other quantities that can be written are:

- ◊ weir crest level at current time;
- ◊ Total discharge through gate at current time;
- ◊ gate sill level at current time;
- ◊ gate sill width at current time;
- ◊ gate door lower edge level at current time;
- ◊ gate door opening width at current time;
- ◊ Total pump discharge at current time;
- ◊ Max. pump capacity at current time;

E.3.2 Spatial fields as NetCDF map-file

The map file <mdident_map.nc> contains data on the entire model grid in 1D, 2D and 3D. Three formats are currently supported, selected by the MapFormat keyword in the MDU:

```
[output]
MapFormat = 1 # Map file format, 1: NetCDF, 2: Tecplot, 3: netCFD and Tecplot, 4: NetCDF-UGRID
```

D-Flow FM is migrating towards the more standardized UGRID¹ format. Delta Shell does not support this format yet, but Delft3D-QUICKPLOT does. In this section only the conventional NetCDF format (1) is further discussed.

¹<https://github.com/ugrid-conventions/ugrid-conventions>

For parallel runs, each process writes a separate file for each partition as <mdident_000X_map.nc>.

The map file contains the dimensions and variable definitions in its header, followed by the actual data.

Dimensions

The following dimensions are defined in a map file header:

```
netcdf weirfree_map {
dimensions:
    nNetNode = 310 ;
    nNetLink = 486 ;
    nNetLinkPts = 2 ;
    nBndLink = 252 ;
    nNetElem = 180 ;
    nNetElemMaxNode = 4 ;
    nNetLinkContourPts = 4 ;
    nFlowElem = 180 ;           // Nr. of grid cells
    nFlowElemMaxNode = 4 ;
    nFlowElemContourPts = 4 ;
    nFlowLink = 246 ;          // Nr. of flow links (open cell edges)
    nFlowLinkPts = 2 ;
    time = UNLIMITED ; // (64 currently)
```

For plotting solution data, the grid cells (flow nodes) are important, and the flow links (open cell edges). All Net* dimensions relate the flow grid tot the original unstructured network. More explanation of these topological naming conventions can be found in [section 7.3.1](#).

Location variables

The location and shape of grid cells is stored in several x,y variables:

```
double FlowElem_xcc(nFlowElem) ;
FlowElem_xcc:units = "m" ;
FlowElem_xcc:standard_name = "projection_x_coordinate" ;
FlowElem_xcc:long_name = "Flow element circumcenter x" ;
FlowElem_xcc:bounds = "FlowElemContour_x" ;
double FlowElem_ycc(nFlowElem) ;
FlowElem_ycc:units = "m" ;
FlowElem_ycc:standard_name = "projection_y_coordinate" ;
FlowElem_ycc:long_name = "Flow element circumcenter y" ;
FlowElem_ycc:bounds = "FlowElemContour_y" ;
// [...]
double FlowElemContour_x(nFlowElem, nFlowElemContourPts) ;
FlowElemContour_x:units = "m" ;
FlowElemContour_x:standard_name = "projection_x_coordinate" ;
FlowElemContour_x:long_name = "List of x-points forming flow element" ;
FlowElemContour_x:_FillValue = -999. ;
// [...]
double FlowElem_bl(nFlowElem) ;
FlowElem_bl:units = "m" ;
FlowElem_bl:positive = "up" ;
FlowElem_bl:standard_name = "sea_floor_depth" ;
FlowElem_bl:long_name = "Bottom level at flow element\`s circumcenter." ;
```

Variables on grid cells/pressure points

Variables on grid cells (represented by the pressure point/cell circumcenter) are typically defined as follows:

```
double s1(time, nFlowElem) ;
s1:coordinates = "FlowElem_xcc FlowElem_ycc" ;
s1:standard_name = "sea_surface_height" ;
s1:long_name = "waterlevel" ;
s1:units = "m" ;
s1:grid_mapping = "projected_coordinate_system" ;
```

Other quantities that can be written are:

- ◊ Water level at beginning of time step;
- ◊ Water depth;
- ◊ Numlimdt: number of times each cell was limiting the time step;
- ◊ Bed shear stress;
- ◊ Cell-centered velocity components;
- ◊ Effective Chézy roughness;
- ◊ Salinity;
- ◊ Temperature;
- ◊ Tracers and other constituents;
- ◊ Heat flux quantities, air temperature, relative humidity, cloudiness, and more detailed fluxes ([chapter 10](#));
- ◊ Streamline curvature and spiral flow intensity ([section 7.6](#));
- ◊ Suspended sediment concentrations and erodible layer thicknesses ([chapter 17](#));
- ◊ Cell-centered wind speed components and atmospheric pressure;

Variables on flow links/velocity points

Variables on grid cell edges (represented by the velocity point) are typically defined as follows:

```
double q1(time, nFlowLink) ;
q1:coordinates = "FlowLink_xu FlowLink_yu" ;
q1:long_name = "Flow flux" ;
q1:units = "m3/s" ;
```

Other quantities that can be written are:

- ◊ Normal velocities at the old and new time level;
- ◊ Horizontal viscosity and diffusivity;
- ◊ Edge-centered wind speed components;
- ◊ Effective roughness values from trachytopes/vegetation ([section 13.2](#))

E.3.3 Restart files as NetCDF rst-file

Restart files <mdident_yyyymmdd_HHMMSS_rst.nc> are almost identical to map files, except that they contain all data for just a single time. Also the flow geometry arrays with cell coordinates and more are not present. For this reason, restart files are less suitable for plotting, but efficient for restarting a computation.

F Spatial editor

F.1 Introduction

The spatial editor is a generic feature of the Delta Shell framework for editing spatial data (e.g. bathymetry, roughness, viscosity, initial conditions, sediment availability). The spatial editor supports both point clouds and coverages (e.g. data on a grid or network). Therefore, you can use the spatial editor both to edit spatial data in general and to prepare model input. This Chapter describes the general features of the spatial editor ([section F.2](#)), (spatial) quantity selection ([section F.3](#)), geometry operations ([section F.4](#)), spatial operations ([section F.5](#)) and the operation stack ([section F.6](#)). The examples given below are typically focusing on use of the spatial editor in the D-Flow FM plugin, but are in principal applicable to any Delta Shell plugin.

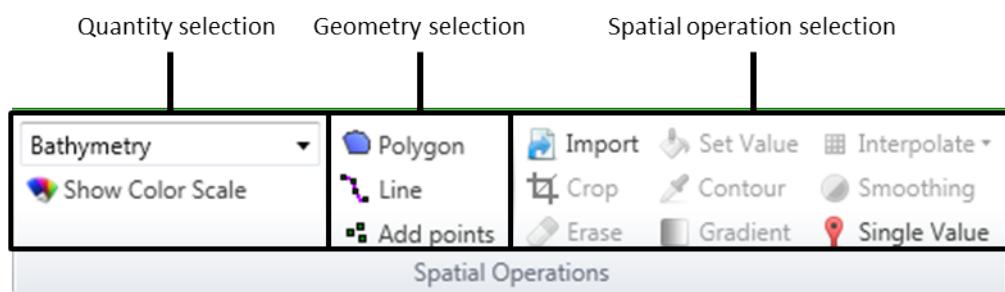


Figure F.1: Overview of spatial editor functionality in Map ribbon

F.2 General

F.2.1 Overview of spatial editor

The spatial editor functionality can be accessed from the “Spatial Operations” section of the “Map Ribbon” ([Figure F.1](#)). Typically, you first select which dataset or quantity (either a point cloud or a coverage) to work on (e.g. bathymetry, roughness, viscosity, initial conditions, sediment availability), then a geometry (e.g. point, line, polygon) and finally which spatial operation to perform (e.g. crop, erase, set value, contour, interpolate, smoothing). Typical workflows are as follows:

Working on a point cloud dataset:

- 1 Import the dataset as point cloud ([section F.2.2](#))
- 2 Activate/select the dataset (quantity) in the spatial editor ([section F.3](#))
- 3 Select a geometry to perform an operation on ([section F.4](#))
- 4 Select the spatial operation for this geometry ([section F.5](#))
- 5 Repeat steps 3 and 4 until you are satisfied with the data
- 6 Export the dataset ([section F.6.7](#))

Working on a coverage (e.g. model input):

- 1 Activate/select the spatial quantity to work on in the spatial editor ([section F.3](#))
- 2 Optional: import a dataset as point cloud ([section F.5.1](#))
- 3 Select a geometry to perform an operation on ([section F.4](#))
- 4 Select the spatial operation for this geometry ([section F.5](#))
- 5 Repeat steps 3 and 4 until you are satisfied with the data
- 6 Interpolate the point cloud to the grid or network (([section F.5.7](#)))
- 7 Optional: export the dataset ([section F.6.7](#))

Upon performing a spatial operation, the ‘Operations’ panel will open (see [Figure F.42](#)) with the operations stack. This stack keeps track of the workflow of spatial operations that you performed. This helps you to make transparent how you arrived at your ‘final’ dataset without having to save all the intermediate datasets (steps) separately. Moreover, the stack is reproducible and easily editable without having

to start all over again. When working on a coverage (e.g. the second workflow), point clouds can be used to construct the coverage. In this case the coverage (for example ‘Bathymetry’) is the ‘trunk’ of the workflow and the point clouds (appearing as sets in the stack) are ‘branches’ or subsets of this trunk (see [Figure F.42](#)). By selecting the set or coverage in the ‘Operations’ panel you determine on which dataset you are working. The interpolate operation ([section F.5.7](#)) allows you to bring data from the point cloud (branch) to the coverage (trunk). For more information on the stack you are referred to section [section F.6](#).

F.2.2 Import/export dataset

To import a (point cloud) dataset use the context menu on “project” in the “Project Tree”, select “import” from the context menu and select the point cloud importer ([Figure F.2](#)). After the import the point cloud will be added to the project tree. To activate the point cloud in the spatial editor either double click the dataset in the project tree ([Figure F.3](#)) or select it from the drop down box in the spatial editor ribbon ([Figure F.4](#)).



Note: Exporter still to be implemented

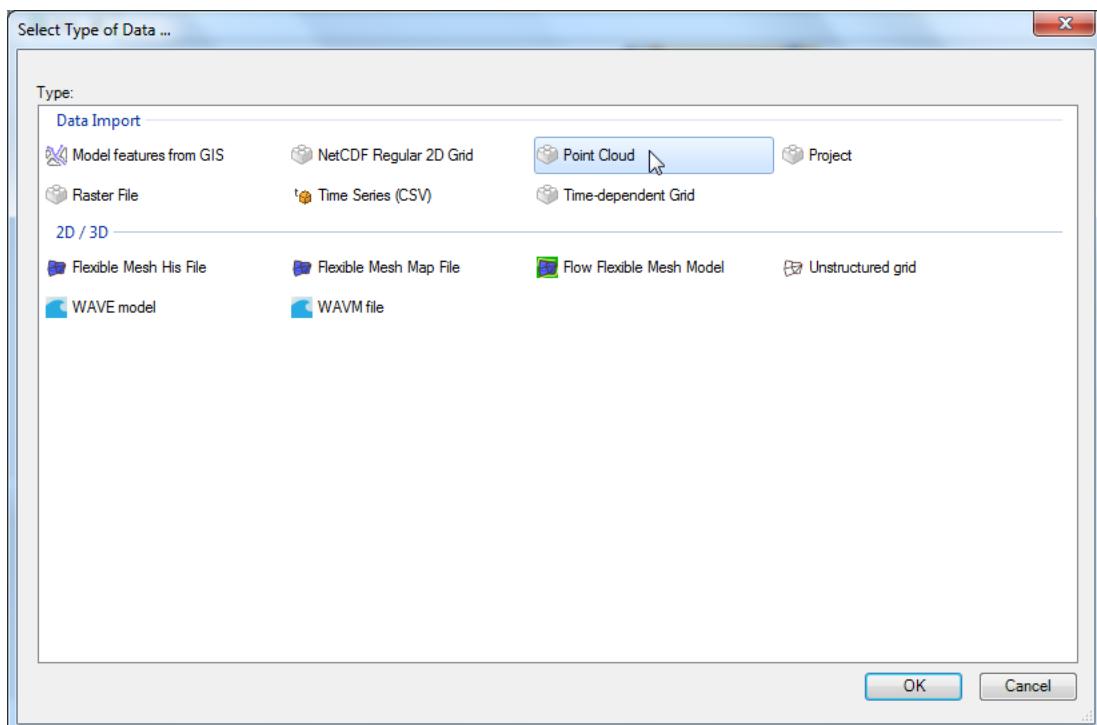


Figure F.2: Importing a point cloud into the project using the context menu on “project” in the project tree

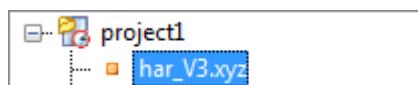


Figure F.3: Activate the imported point cloud in the spatial editor by double clicking it in the project tree

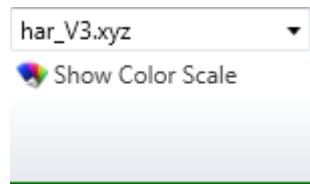


Figure F.4: Activate the imported point cloud in the spatial editor by selecting it from the dropdown box in the Map ribbon

F.2.3 Activate (spatial) model quantity

Similar to activating an imported dataset in the spatial editor, you can also activate a (spatial) model quantity (e.g. bathymetry, initial conditions, roughness, viscosity) in the spatial editor by double clicking the quantity in the project tree or selecting it from the dropdown box in the “Map” ribbon.

F.2.4 Colorscale

Upon activating a spatial quantity in the spatial editor it becomes visible in the map window with a corresponding colorscale (Figure F.5 top). You can (de-)activate the colorscale by clicking on the “Show Color Scale” button in the “Map” ribbon (Figure F.5 left panel). By default the colorscale is ranging from the minimum value of the active dataset. You can adjust/decrease this range using the sliders at the top and bottom of the colorbar (Figure F.5 right panel).

You can also adjust the colormap and classes of the colorscale using the context menu on the spatial quantity in the “Map tree” and selecting “Properties” (Figure F.6 left panel). A layer properties editor will open in which you can set the colormap to your own preferences (Figure F.6 right panel). **Note:** Please note that currently you can only edit the properties of the colorscale before you perform spatial operations. Once you have performed a spatial operation this functionality will be disabled.

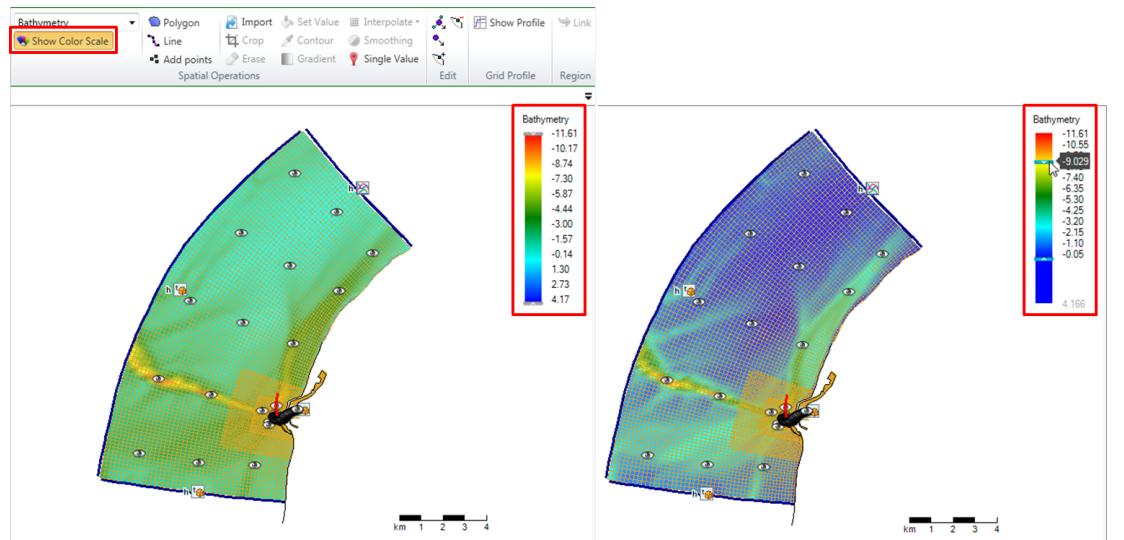


Figure F.5: Activate the colorscale using the button in the map ribbon (top) and the colorscale will become visible in the map window (left). You can adjust/decrease the range of the colorscale using the sliders at the top and bottom of the colorbar (right).

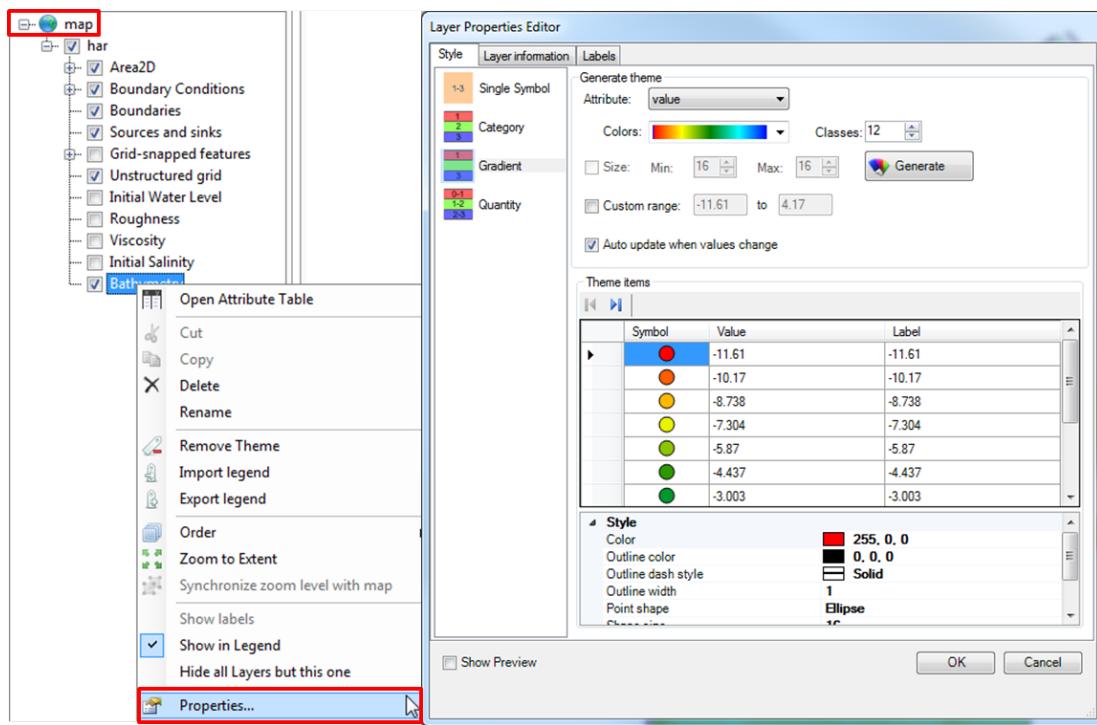


Figure F.6: Edit the colorscale properties using the context menu on the active layer in the Map Tree

F.2.5 Render mode

By default point clouds are rendered as (colored) points and coverages as shades (e.g. 'FillSmooth'). The render mode can be edited using the properties of the active layer [Figure F.7](#). Delta Shell offers the following render modes:

- ◊ Points
- ◊ Lines (only for coverages)
- ◊ Shades or 'FillSmooth' (only for coverages)
- ◊ Colored numbers
- ◊ Mono colored numbers

An example of a coverage rendered as colored numbers is given in [Figure F.8](#).

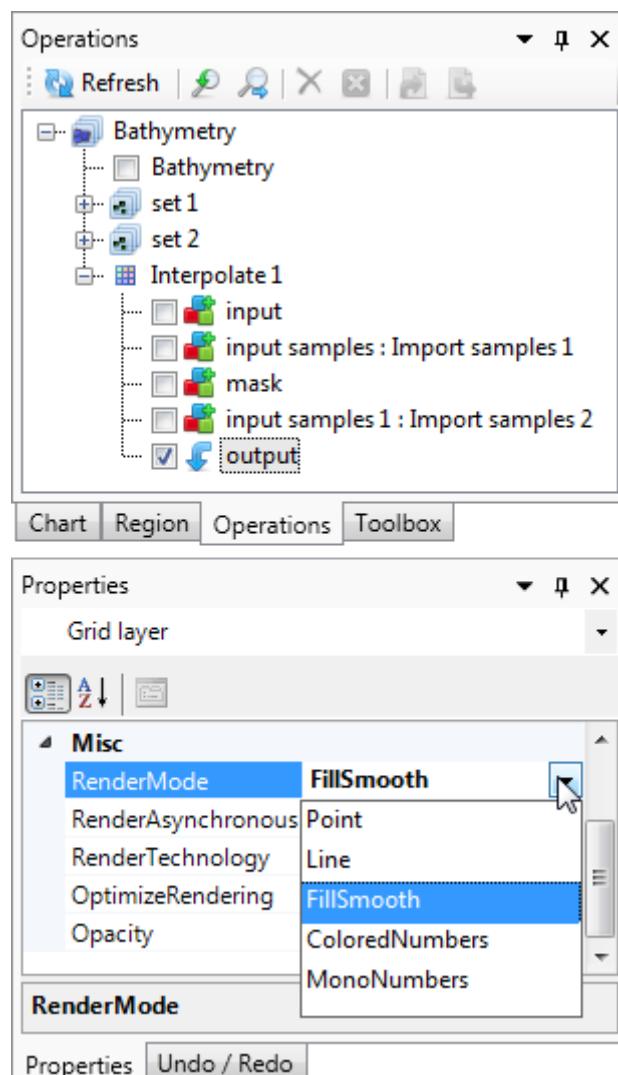
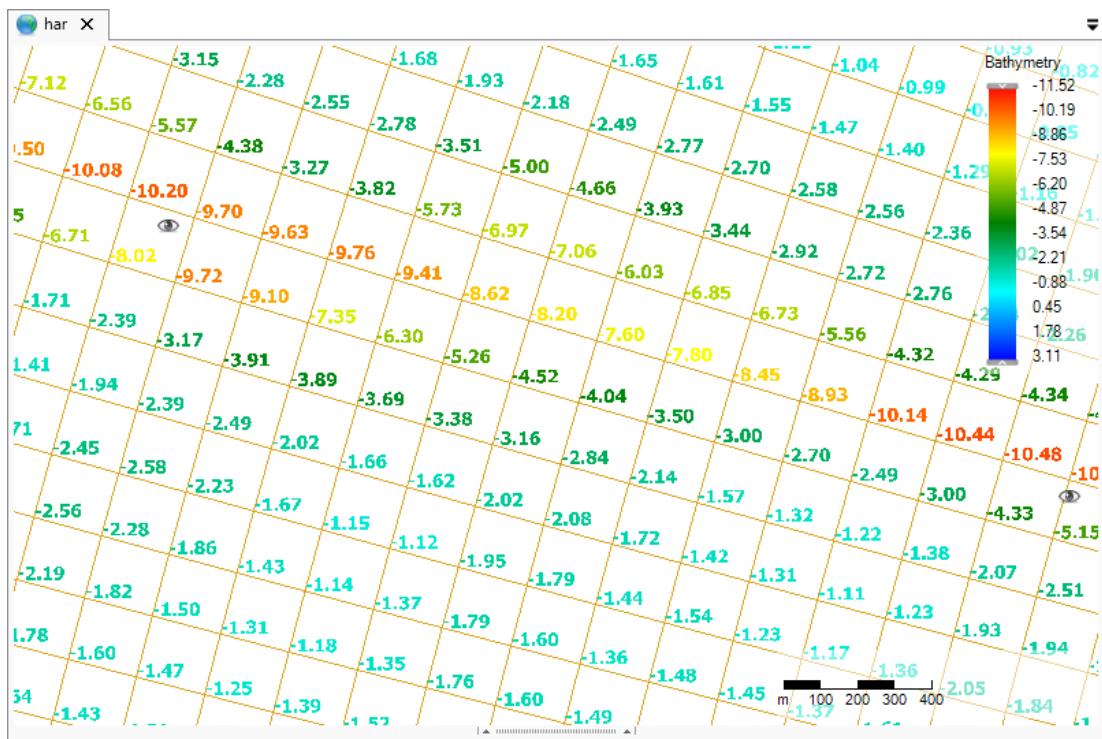


Figure F.7: Select the rendermode for the active layer in the property grid.



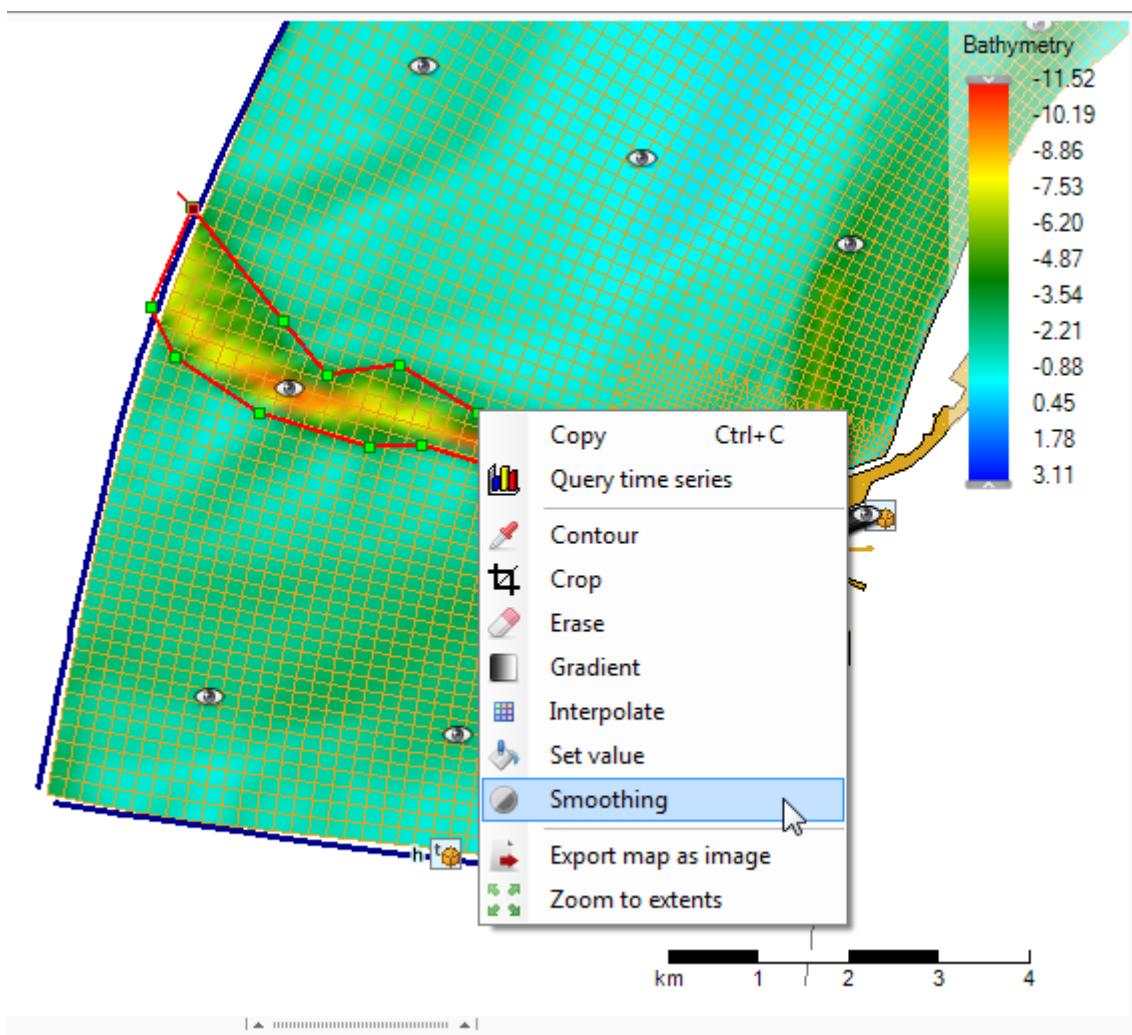


Figure F.9: Selecting a smoothing operation for a polygon geometry from the context menu (using context menu)

F.3 Quantity selection

A spatial quantity can be activated/selected either by double clicking it in the project tree (Figure F.10) or by selecting it from the dropdown box in the “Map” ribbon (Figure F.11). Upon selecting the spatial quantity it will be shown as a point cloud (for a dataset) or coverage (for model input) on the central map. Typically, you start from a point cloud (either obtained from import or by generating samples yourself) which will eventually be interpolated to a grid or network (e.g. coverage). The spatial editor will keep track of both the changes made to the point cloud(s) and coverage of the selected spatial quantity. The information will be saved in the Delta Shell project and available the next time you open the project. **Note:** The operations are not yet saved in a human-readable/editable file



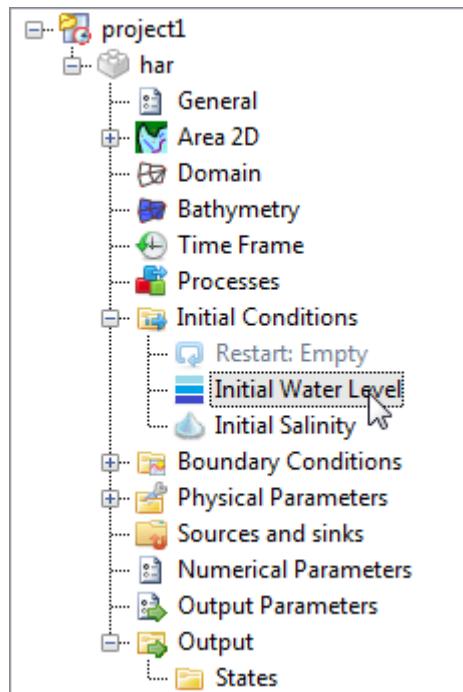


Figure F.10: Activating a spatial quantity by double clicking it in the project tree (in this example 'Initial Water Level')

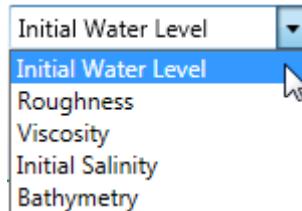


Figure F.11: Activating a spatial quantity by selecting it from the dropdown box in the 'Map' ribbon

F.4 Geometry operations

The spatial editor supports three types of geometry operations: (1) polygons, (2) lines and (3) points (see also [Figure F.12](#)). The following sub-sections subsequently describe how these geometries can be selected and edited. If you do not select any of these three geometry operations, the spatial operation automatically applies to all the data.



Note: Please note that the drawn geometries are not yet persistent, implying that the geometries once drawn cannot be edited yet. Upon pressing the “Esc” button while in editing mode all drawn geometries will disappear.

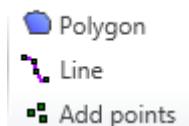


Figure F.12: Overview of the available geometry operations in the 'Map' ribbon

F.4.1 Polygons

Upon selecting “Polygon” from the “Map” ribbon you can draw one or multiple polygons (Figure F.13). Each polygon point is defined by a single click with the LMB. The polygon is closed by double clicking the LMB. After drawing the (first) polygon, the available spatial operations for polygons are enabled in the “Map” ribbon. The following spatial operations are available for polygons:

- ◊ Crop ([section F.5.2](#))
- ◊ Erase ([section F.5.3](#))
- ◊ Set Value ([section F.5.4](#))
- ◊ Contour ([section F.5.5](#))
- ◊ Gradient ([section F.5.6](#))
- ◊ Smoothing ([section F.5.8](#))
- ◊ Interpolate - only in case samples and a grid/network are available ([section F.5.7](#))

The selected spatial operation applies to all the drawn polygons.

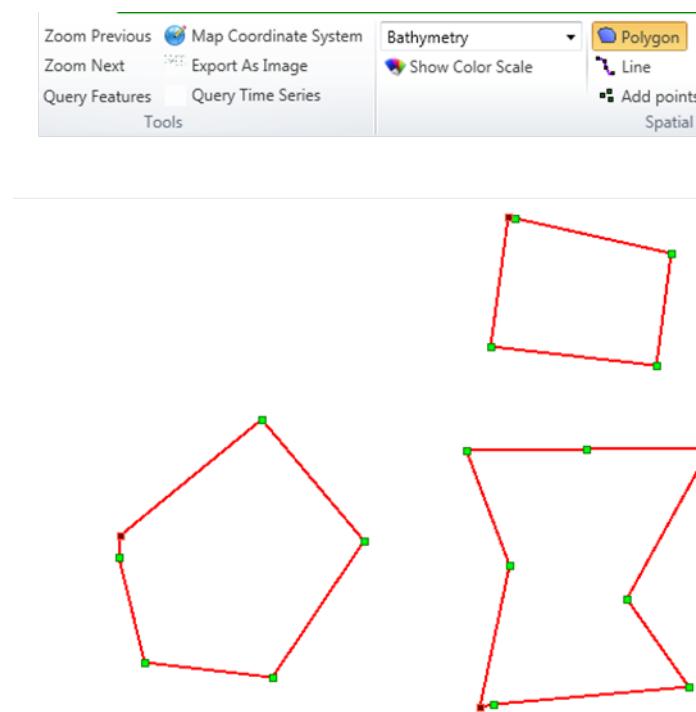


Figure F.13: Activating the polygon operation and drawing polygons in the central map.

F.4.2 Lines

Upon selecting “Line” from the “Map” ribbon you can draw one or multiple lines (Figure F.14). Each line point is defined by a single click with the LMB. The line is completed by double clicking the LMB. After drawing the (first) line, the available spatial operations for lines are enabled in the “Map” ribbon. The following spatial operations are available for lines:

- ◊ Contour ([section F.5.5](#))

The selected spatial operation applies to all the drawn lines.

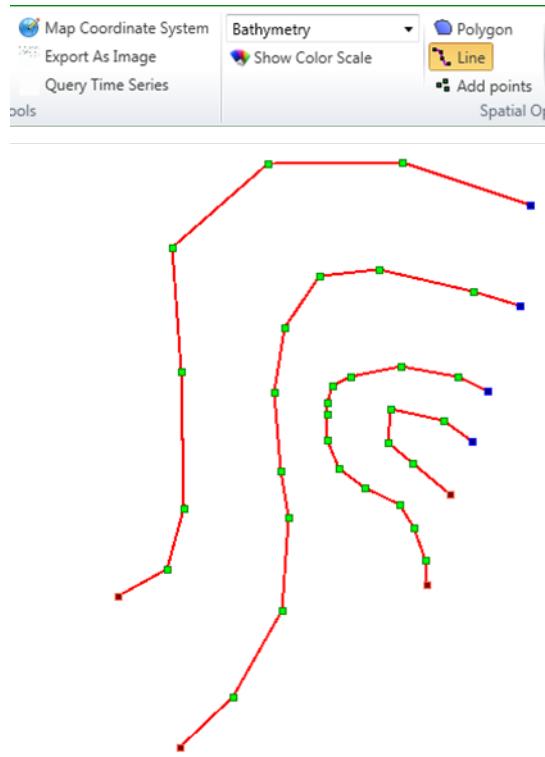


Figure F.14: Activating the line operation and drawing lines in the central map.

F.4.3 Points

Upon selecting “Add points” from the “Map” ribbon you can draw one or multiple points and assign a uniform value to them ([Figure F.15](#)). Each point is defined by a single click with the LMB. The group of points is completed by double clicking the LMB. Upon double clicking a popup appears in which you can assign a value to the points.

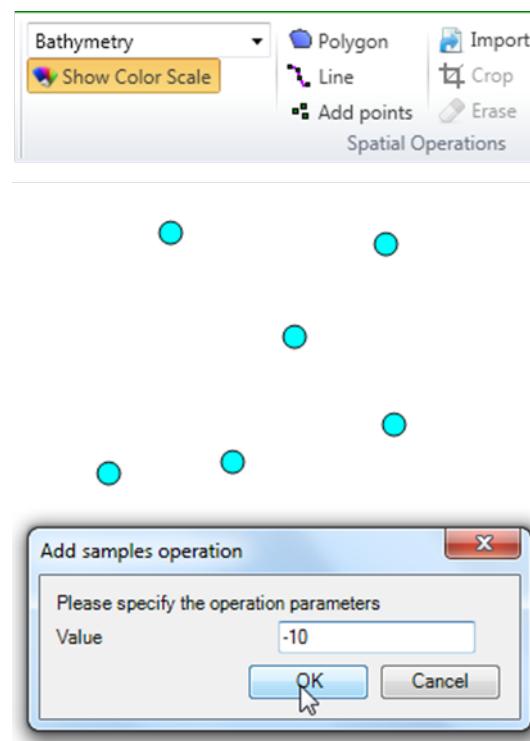


Figure F.15: Activating the 'Add points' operation, drawing them in the central map and assigning a value to them.

F.5 Spatial operations

The spatial editor supports the following spatial operations (see also [Figure F.16](#)):

- ◊ Import ([section F.5.1](#)) - only for point clouds
- ◊ Crop ([section F.5.2](#))
- ◊ Erase ([section F.5.3](#))
- ◊ Set Value ([section F.5.4](#))
- ◊ Contour ([section F.5.5](#)) - only for point clouds
- ◊ Gradient ([section F.5.6](#))
- ◊ Interpolate ([section F.5.7](#)) - only for point clouds
- ◊ Smoothing ([section F.5.8](#))
- ◊ Change single value ([section F.5.9](#)) - only for coverages



Figure F.16: Overview of the available spatial operations in the 'Map' ribbon

The sections below provide a description of each operation.

F.5.1 Import point cloud

With the import operation you can import a point cloud for the selected spatial quantity (**Note:** To import a coverage select 'Import' from the context menu of the spatial quantity in the project tree). For this operation no geometry is required. The import operation is activated from the 'Map' ribbon



(Figure F.17). Upon importing a point cloud you are asked whether a coordinate transformation should be applied to the imported dataset (Figure F.18). By default it will be assumed that the imported data is in the same coordinate system as the model. If not, you can indicate from which to which coordinate system the data should be transformed. After import the point cloud is added to the operations stack (Figure F.19). The difference between this importer and importing a point cloud on the project or model level in the project tree (section F.2.2) is that for this importer the point cloud is directly assigned to the selected spatial quantity (e.g. model input) instead of being treated as a separate dataset.

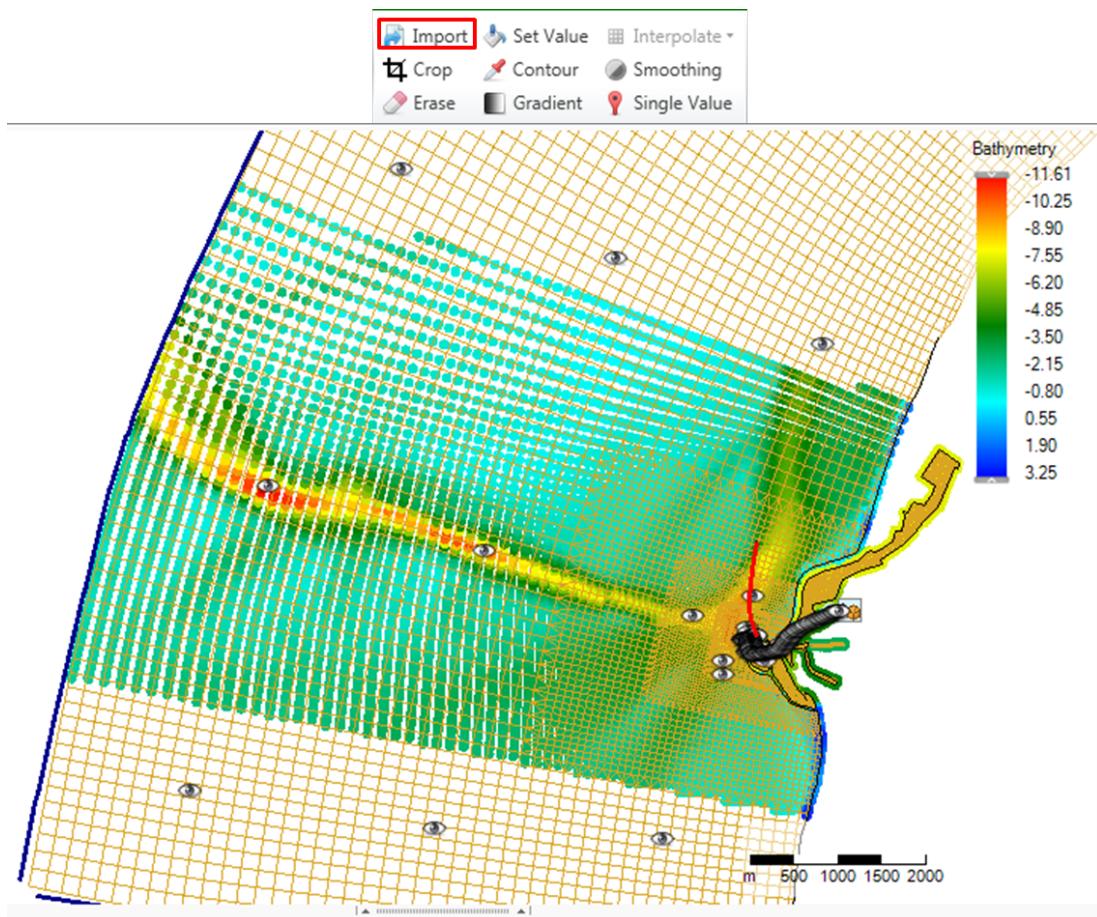


Figure F.17: Importing a point cloud using the ‘Import’ operation from the ‘Map’ ribbon

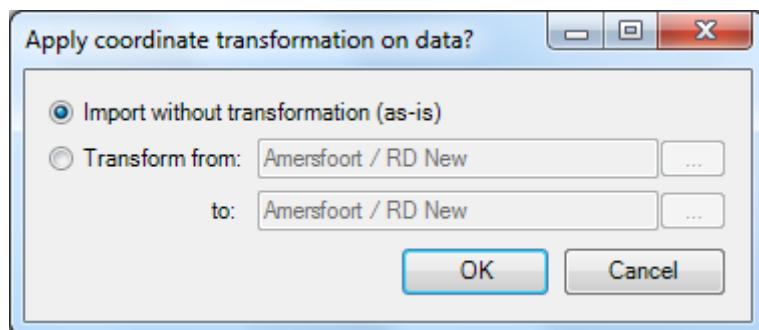


Figure F.18: Option to perform a coordinate transformation on the imported point cloud

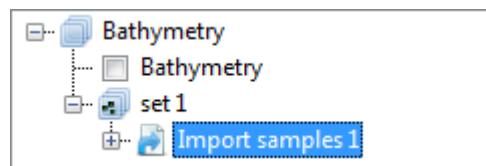


Figure F.19: Appearance of import point cloud operation in the operations stack

F.5.2 Crop

The crop operation crops a point cloud or coverage (depending on which one is active). The crop operation is activated from the 'Map' ribbon, and only available for polygon geometries. You can control which part of the data should be erased by using polygons. If you provide a polygon outside the point cloud or coverage, all data will be erased. For an example see [Figure F.20](#). After cropping (part of) the point cloud or coverage the operation is added to the operations stack ([Figure F.21](#)).

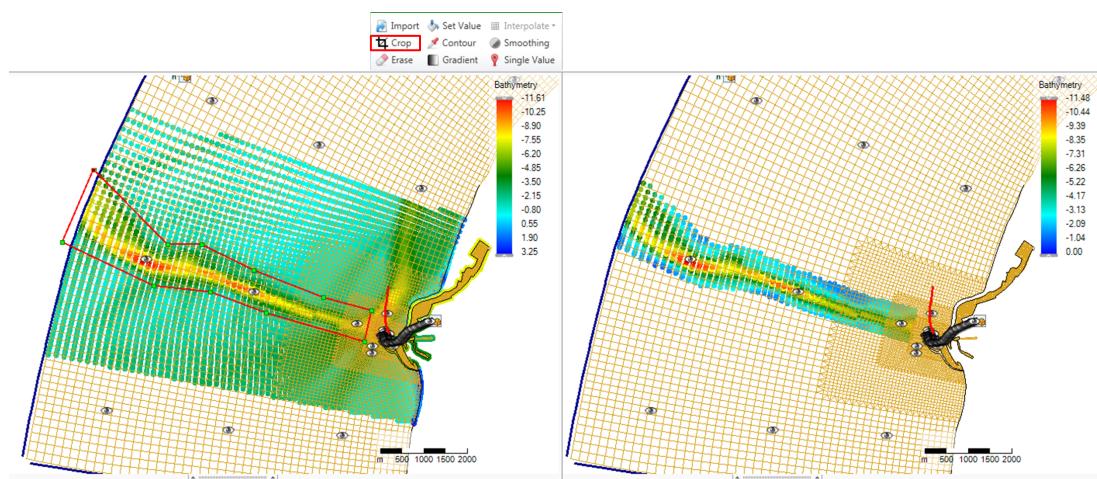


Figure F.20: Performing a crop operation on a point cloud with a polygon using 'Crop' from the 'Map' ribbon

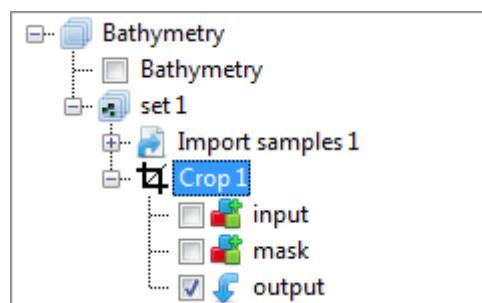


Figure F.21: Appearance of crop operation in the operations stack

F.5.3 Erase

The erase operation erases (part of) a point cloud or coverage (depending on which one is active). The erase operation is activated from the 'Map' ribbon. You can control which part of the data should be erased by using polygons. If no polygons are provided, the total dataset will be erased. For an example see [Figure F.22](#). After erasing (part of) the point cloud or coverage the operation is added to the operations stack ([Figure F.23](#)).

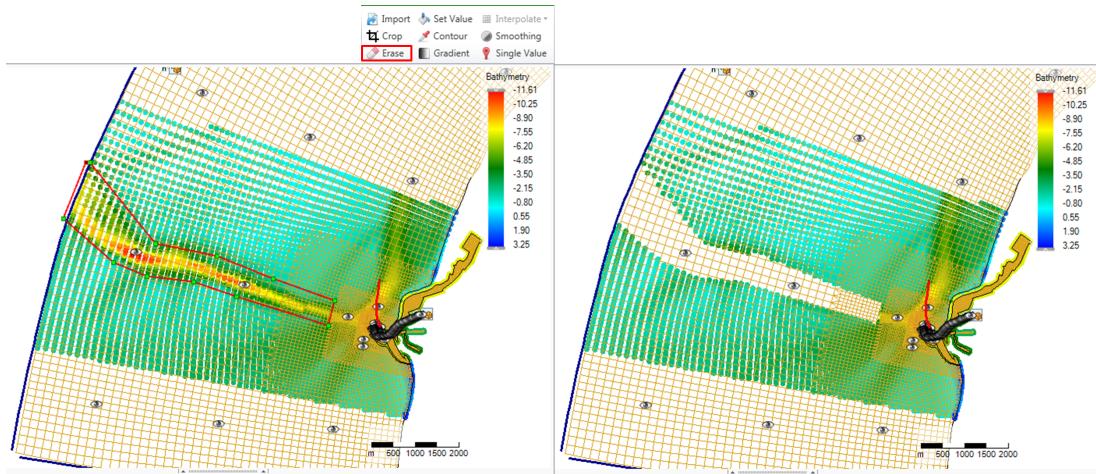


Figure F.22: Performing an erase operation on a point cloud with a polygon using 'Erase' from the 'Map' ribbon

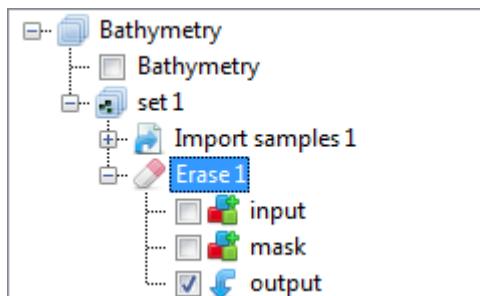


Figure F.23: Appearance of erase operation in the operations stack

F.5.4 Set value

The set value operation assigns a value to a point cloud or coverage (depending on which one is active). The set value operation is activated from the 'Map' ribbon and only available for polygon geometries or for the total data set if no polygon is provided. By assigning a value can choose from the following operations:

- ◊ **Overwrite** : overwrites all existing points within the polygon (excluding no data values) with the uniform value
- ◊ **Overwrite where missing (only for coverages)** : overwrites all missing values within the polygon with the uniform value
- ◊ **Add** : Adds the uniform value to all existing points within the polygon (excluding no data values)
- ◊ **Subtract** : Subtracts the uniform value from all existing points within the polygon (excluding no data values)
- ◊ **Multiply** : Multiplies all existing points within the polygon (excluding no data values) with the uniform value
- ◊ **Divide** : Divides all existing points within the polygon (excluding no data values) by the uniform value
- ◊ **Maximum** : Sets all existing points within the polygon (excluding no data values) to the maximum of its current value and the uniform value

- ◇ **Minimum** : Sets all existing points within the polygon (excluding no data values) to the minimum of its current value and the uniform value

For an example see [Figure F.24](#). After performing a set value operation to (part of) the point cloud or coverage the operation is added to the operations stack ([Figure F.25](#)).

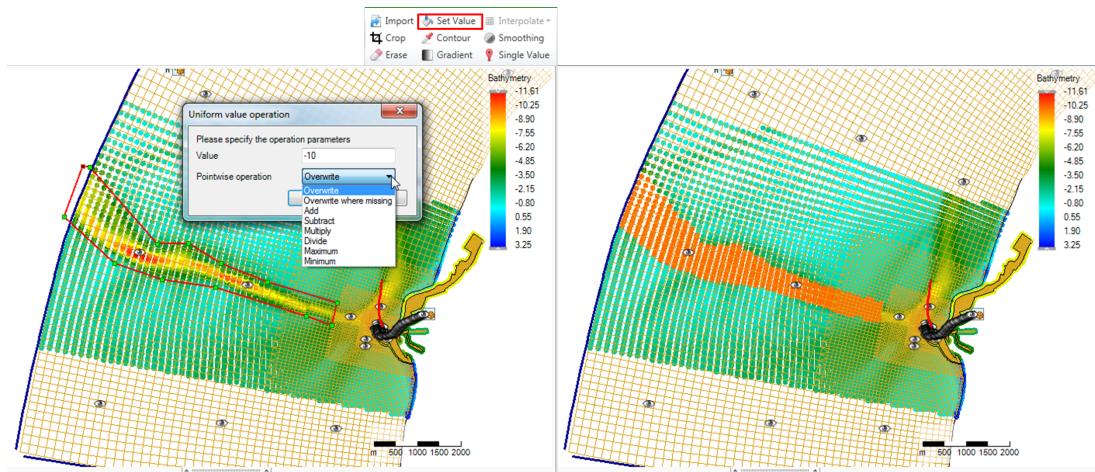


Figure F.24: Performing a set value operation (e.g. overwrite) on a point cloud with a polygon using 'Set Value' from the 'Map' ribbon

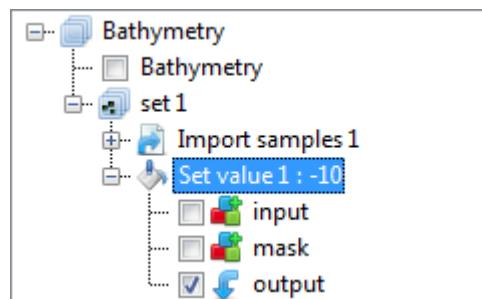


Figure F.25: Appearance of set value operation in the operations stack

F.5.5 Contour

The contour operation creates a point cloud with a uniform value along a line or polygon (depending on which one is active). The contour operation is activated from the 'Map' ribbon. After drawing the lines or polygons you have to assign the uniform value (argument) and the sampling interval in m. This spatial operation can be useful to digitalize information from nautical charts for example. In this case you first have to import the nautical chart as a geotiff ([Figure F.26](#)), set the right map coordinate system ([Figure F.27](#)) and then use the contour operation [Figure F.28](#). Sometimes the samples are created behind the geotiff. Then you can use the context menu on the samples layer in the Map tree to bring the samples to the front ([Figure F.29](#)). After applying the contour operation it is added to the stack ([Figure F.30](#)).

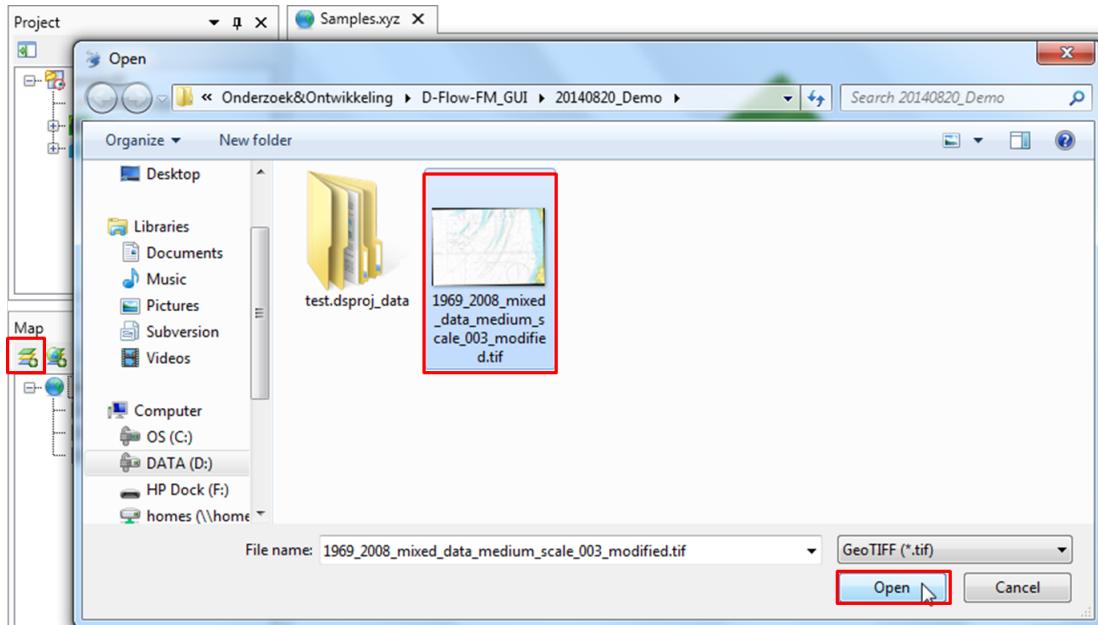


Figure F.26: Import a nautical chart as a georeferenced tiff file

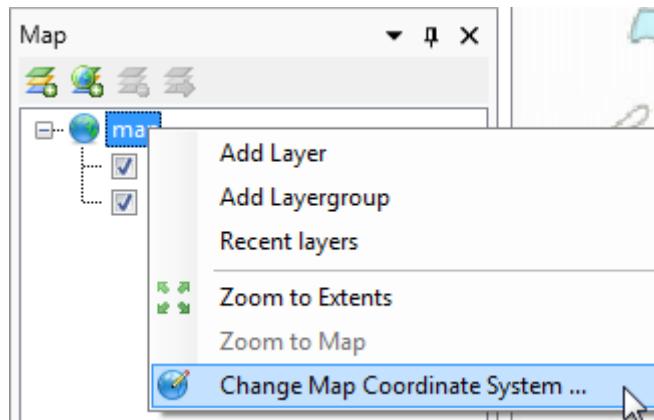


Figure F.27: Set the right map coordinate system for the geotiff

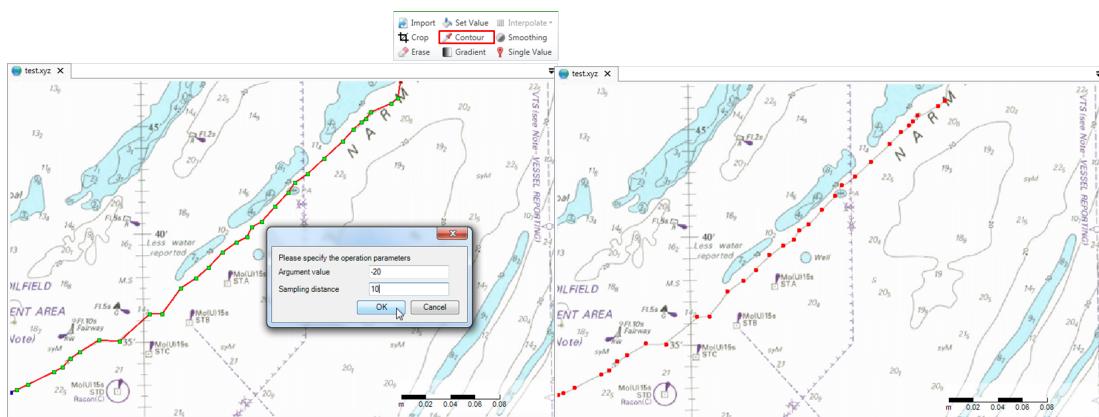


Figure F.28: Performing a contour operation on a nautical chart using lines to define the contours and 'Contour' from the 'Map' ribbon

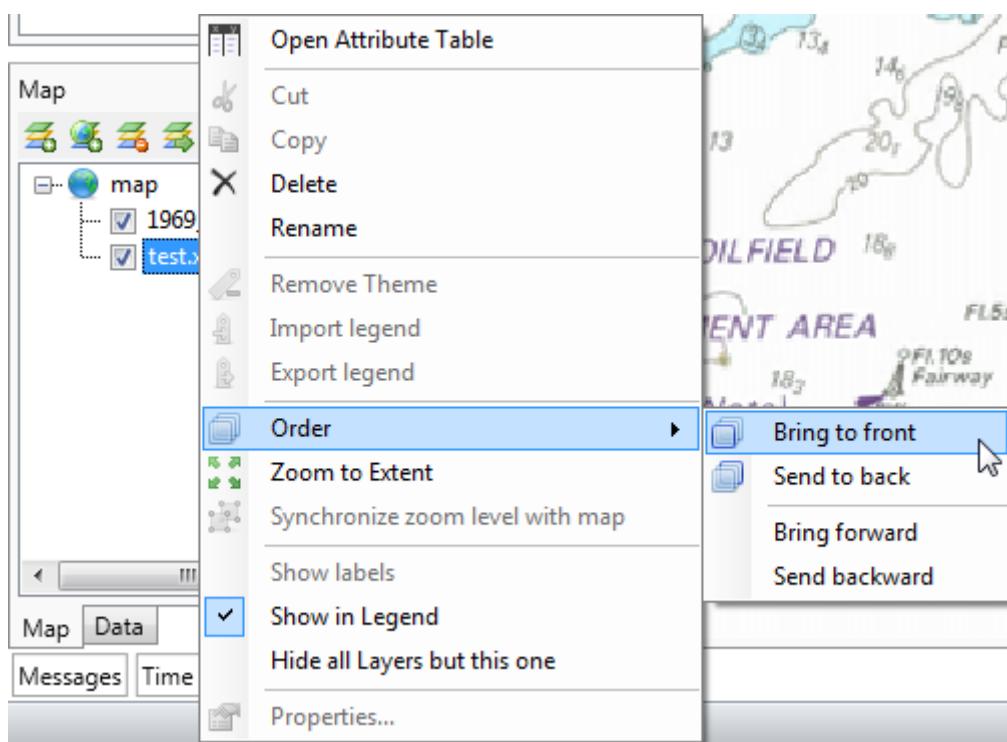


Figure F.29: Bring the sample set to the front if it appears behind the nautical chart

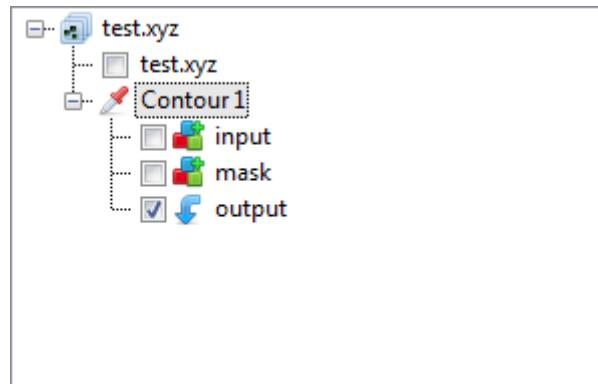


Figure F.30: Appearance of contour operation in the operations stack

F.5.6 Gradient

The gradient operation applies a gradient to a point cloud or coverage (depending on which one is active). The gradient operation is activated from the 'Map' ribbon and only available for polygon geometries or for the total data set if no polygon is provided. You have to assign the initial (start) value, the final (end) value and the going to angle (according to the Cartesian convention with 0 degrees is East, 90 degrees is North, etc). **Note:** This is not working properly yet. For an example see [Figure F.31](#). After applying a gradient to (part of) the point cloud or coverage the operation is added to the operations stack ([Figure F.32](#)).



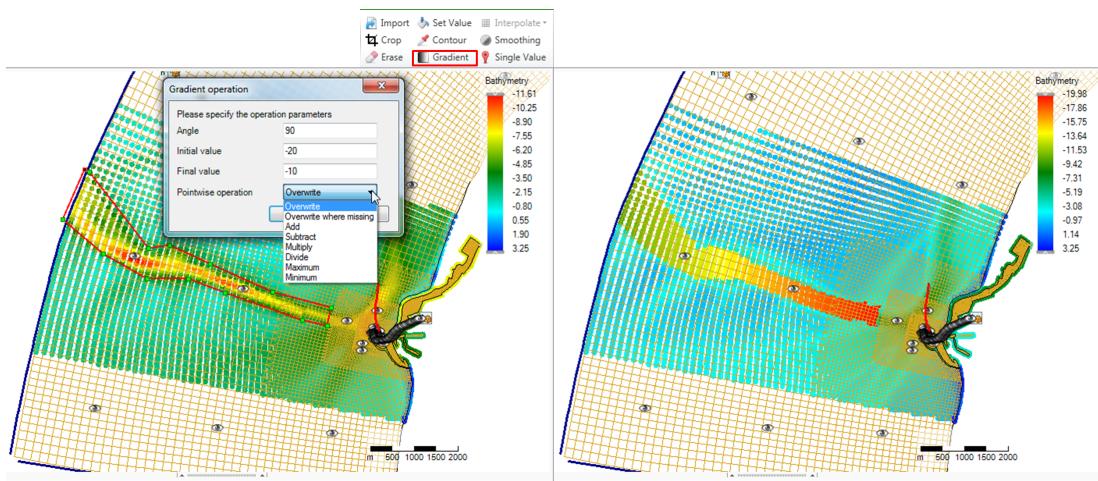


Figure F.31: Performing a gradient operation on a point cloud with a polygon using ‘Gradient’ from the ‘Map’ ribbon

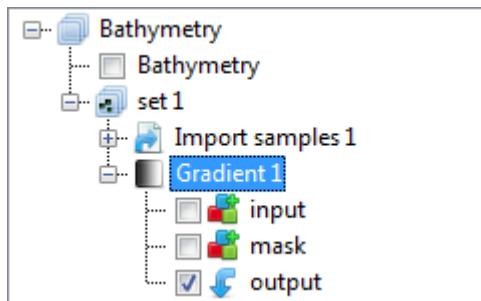


Figure F.32: Appearance of gradient operation in the operations stack

F.5.7 Interpolate

The interpolate operation is the way to get sample set(s) to a grid or network (e.g. coverage). In the operation stack this means that we are actively switching from working on a point cloud (helping to construct the coverage) to working on the selected coverage (or spatial quantity). Currently, there is one interpolation method available, which is triangulation. The triangulation is performed on the data within a polygon or polygons (if provided) or all the data (if no polygons are provided). The interpolate operation can be performed on a single (selected) sample set or on multiple sample sets. Both methods are discussed below.

Interpolate single (selected) set

To perform interpolation on a single sample set, select the sample set (i.e. ‘set1’) in the operation stack and press ‘Interpolate’ in the ‘Map’ ribbon (Figure F.33). Since no polygon is provided in this example, all the samples will be interpolated to the grid. Use polygons if you would like to have more control over the interpolation. After the interpolation the operation is added to the operations stack (Figure F.34). Please note that after performing the interpolation the workflow in the stack is shifting from the sample set (i.e. ‘set1’ - which was a side step to construct the coverage) to the coverage (i.e. ‘bathymetry’).

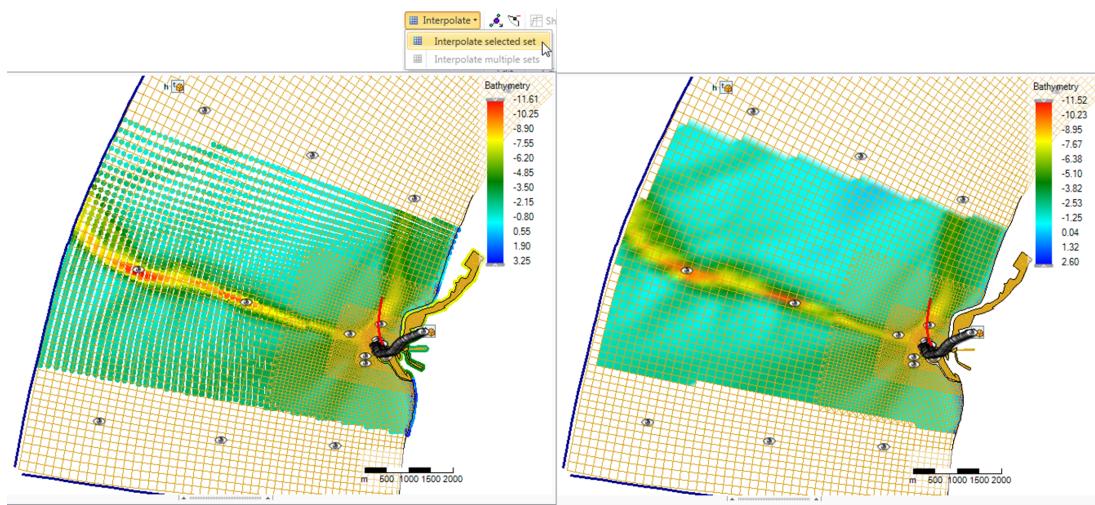


Figure F.33: Performing an interpolation operation on a single sample set (without using a polygon) using 'Interpolate' from the 'Map' ribbon

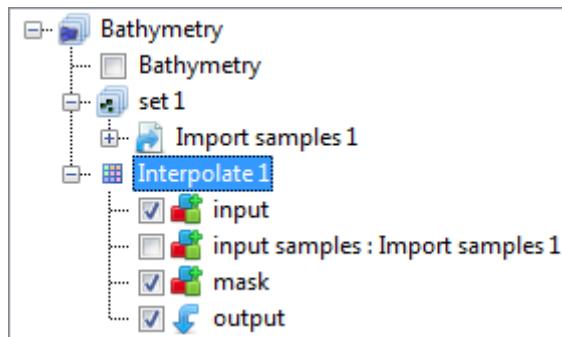


Figure F.34: Appearance of interpolation of 'set1' to the coverage 'Bathymetry' in the operations stack

Interpolate multiple sets

To perform interpolation on multiple sample sets, select the active coverage (i.e. 'Bathymetry') in the operation stack and press 'Interpolate' in the 'Map' ribbon (Figure F.35). In the popup you can select which sample sets to include in the interpolation (in this example both). Since no polygon is provided, all the samples (from the two sets) will be interpolated to the grid. Use polygons if you would like to have more control over the interpolation. After the interpolation the operation is added to the operations stack (Figure F.36). Again note that after performing the interpolation the workflow in the stack is shifting from the sample set (which was a side path to construct the coverage) to the coverage (i.e. bathymetry).

Note: Please note that interpolation of multiple sample sets can also be achieved by importing/combing different sample sets into the same set in the stack instead of using two separate sets. In this case you can just interpolate the single (selected) set.



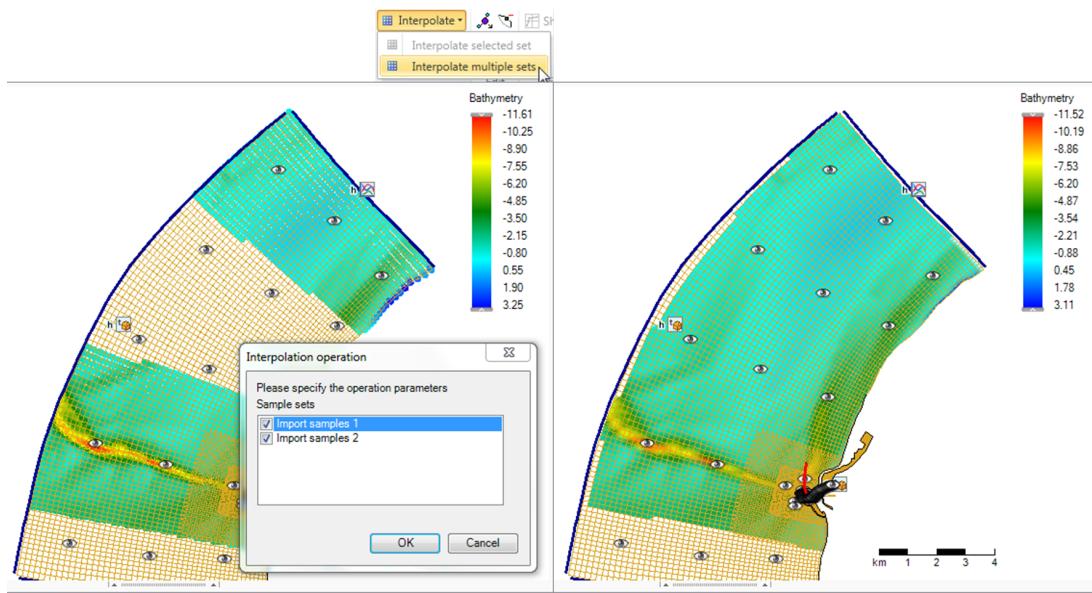


Figure F.35: Performing an interpolation operation on multiple sample sets (without using a polygon) using 'Interpolate' from the 'Map' ribbon

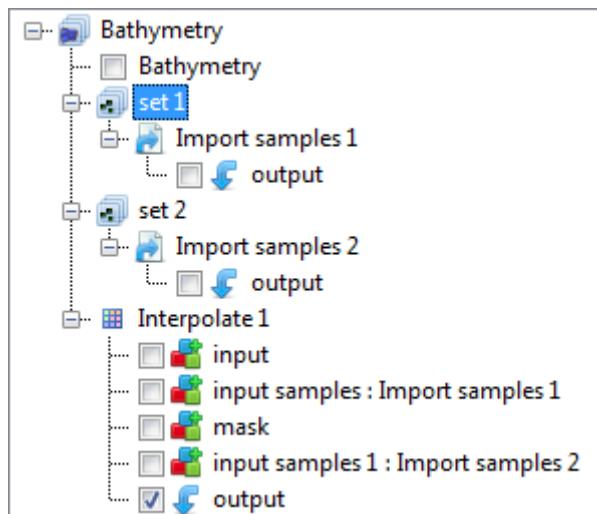


Figure F.36: Appearance of interpolation of 'set1' and 'set2' to the coverage 'Bathymetry' in the operations stack

F.5.8 Smoothing

The smoothing operation smooths out (steep) gradients in a point cloud or coverage (depending on which one is active). The smoothing operation is activated from the 'Map' ribbon and only available for polygon geometries or for the total data set if no polygon is provided. You have to assign the smoothing exponent and number of smoothing steps. The higher the exponent and the number of smoothing steps, the heavier the smoothing. For an example see [Figure F.37](#). After applying smoothing to (part of) the point cloud or coverage the operation is added to the operations stack ([Figure F.38](#)).

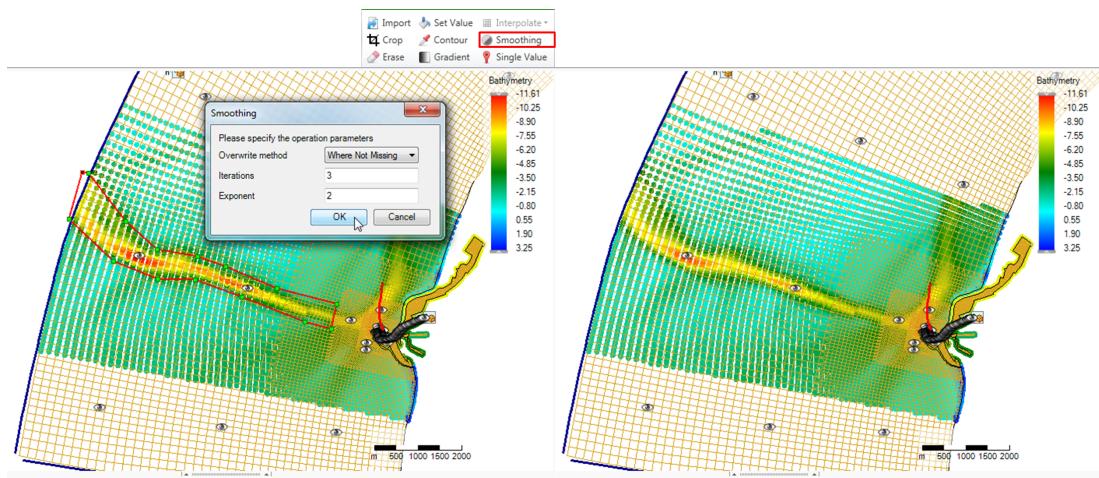


Figure F.37: Performing a smoothing operation on a point cloud with a polygon using 'Smoothing' from the 'Map' ribbon

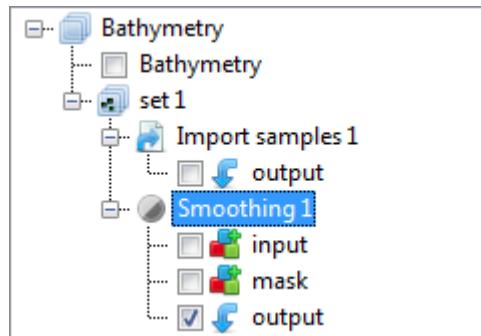


Figure F.38: Appearance of smoothing operation in the operations stack

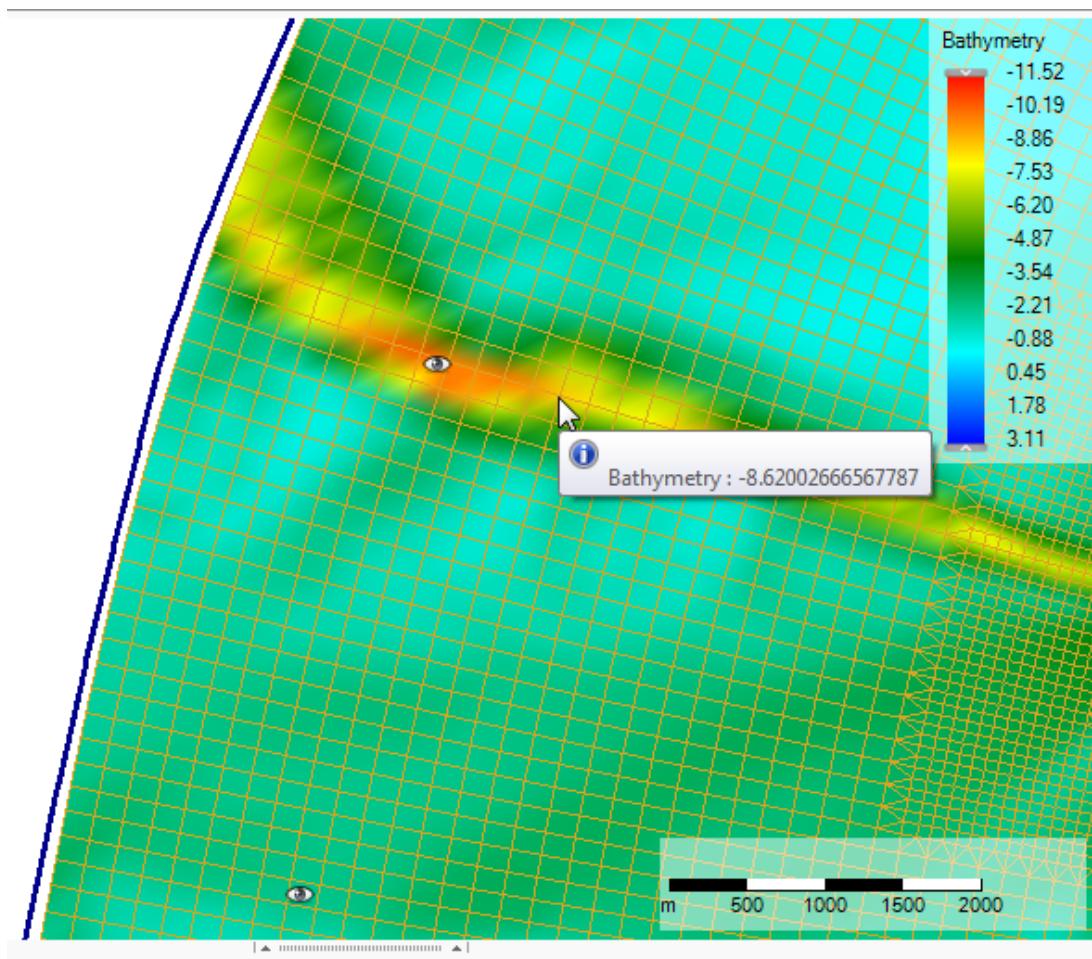


Figure F.39: The cursor for the overwrite operation showing the value of the closest coverage point

F.5.9 Overwrite (single) value

The 'overwrite (single) value' operation allows you to edit single values on the active coverage after the interpolation. The 'overwrite (single) value' operation is activated from the 'Map' ribbon. There is no geometry required for this operation. Upon selecting the operation from the ribbon a cursor will become active showing the coverage value closest to the cursor in a tooltipstring (Figure F.39). Upon clicking LMB a popup appears in which you can overwrite the value of this coverage point (Figure F.40). After applying the overwrite operation it is added to the operations stack (Figure F.41).

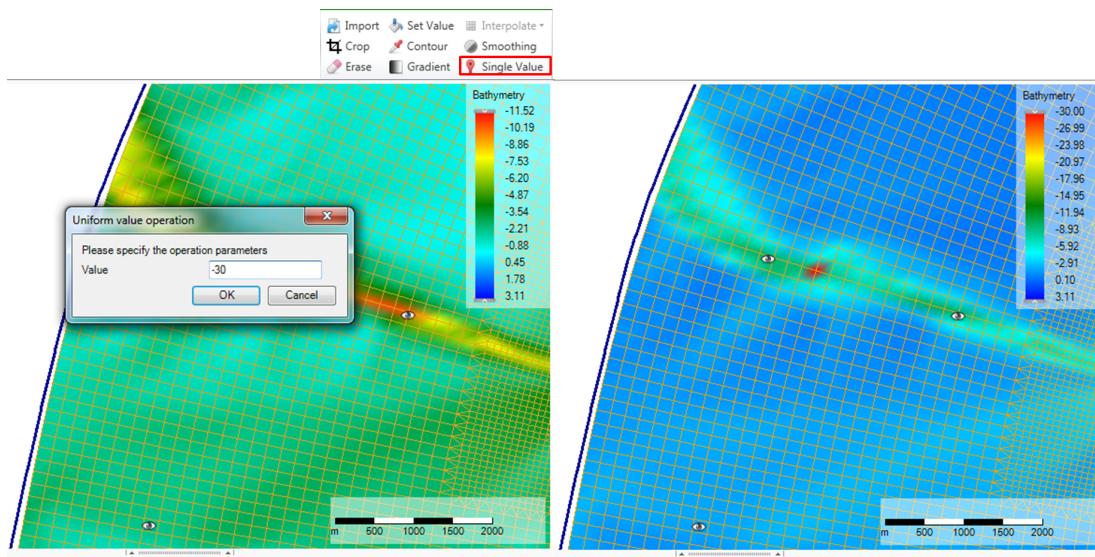


Figure F.40: Performing an overwrite operation on a coverage point using ‘Single Value’ from the ‘Map’ ribbon

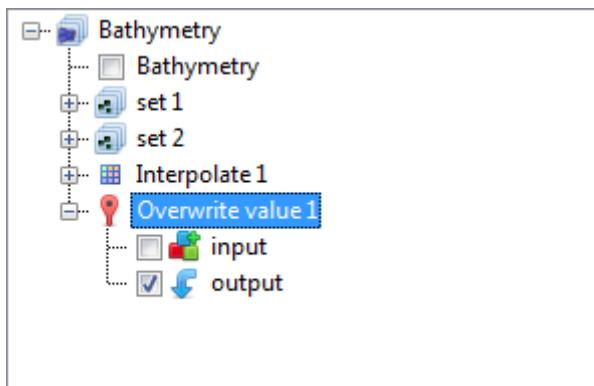


Figure F.41: Appearance of overwrite operation in the operations stack

F.6 Operation stack

The operation stack keeps track of the workflow of spatial operations that you performed. This helps you to make transparent how you arrived at your ‘final’ dataset without having to save all the intermediate datasets (steps) separately. Moreover, the stack is reproducible and easily editable without having to start all over again. This section describes the stack workflow ([section F.6.1](#)), how to edit operation properties ([section F.6.2](#)), how to enable/disable ([section F.6.3](#)), delete ([section F.6.4](#)), refresh([section F.6.5](#)) operations, quick links ([section F.6.6](#)) and import/export functionality ([section F.6.7](#)).

Note: Currently, the stack is saved in the Delta Shell project upon saving the project. The next time you open the project, the stack will reappear. The stack is not (yet) saved in a human readable/editable file.



F.6.1 Stack workflow

Upon performing a spatial operation, the ‘Operations’ panel will open (see [Figure F.42](#)) with the operations stack (tree). The stack first shows on which point cloud or coverage you are working (in this example ‘Bathymetry’). Subsequently, all the operations on this dataset are listed. For each operation you can inspect what the input, mask (e.g. the geometry used for the operation) and output are for the operation ([Figure F.43](#)). By default the stack continues from the last operation that you performed. If you wish to work on a different dataset or operation within a dataset, you have to select that dataset or

operation in the ‘Operations’ panel with the LMB.

When working on a coverage, point clouds (or sets) can be used to construct the coverage. In that case the stack jumps from the ‘trunk’ to a ‘branch’ and the subsequent operations are performed on the point cloud (see [Figure F.42](#)). By selecting the set or coverage in the ‘Operations’ panel you determine on which dataset you are working. The interpolate operation ([section F.5.7](#)) allows you to bring data from the point cloud (branch) to the coverage (trunk). See also [Figure F.42](#).

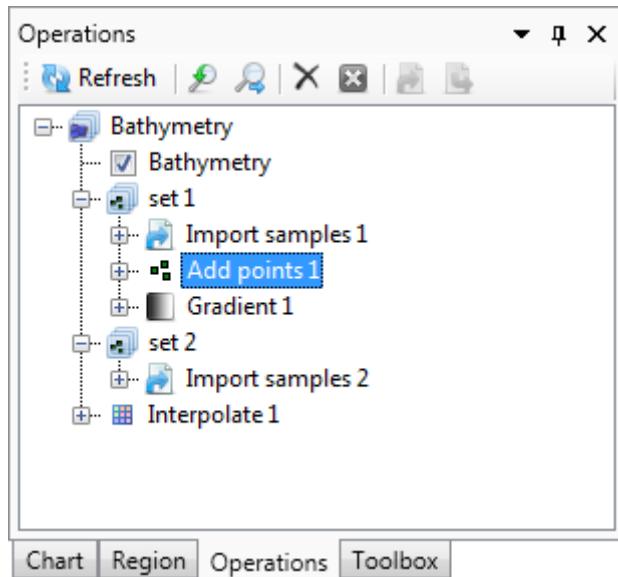


Figure F.42: The ‘Operations’ panel with the operations stack. In this example ‘Bathymetry’ is the coverage (e.g. trunk) that is edited. The point clouds ‘set 1’ and ‘set 2’ (e.g. branches) are used to construct the ‘Bathymetry’ coverage.

F.6.2 Edit operation properties

For each operation that you performed the properties (such as the value or ‘Pointwise operation’ of a ‘Set Value’ operation) can be edited using the ‘Properties’ window ([Figure F.44](#)). **Note:** Please note that the mask of an operation cannot (yet) be edited. By editing the operation properties the operation stack becomes ‘out of sync’ and has to be refreshed for the changes to become active ([see section F.6.5](#)).

F.6.3 Enable/disable operations

You can (temporarily) enable/disable operations by selecting the operation and pressing boxed cross icon in the stack menu ([Figure F.45](#)). Upon disabling an operation the operation will be made grey in the stack and the operation is not taken into account anymore upon evaluation of the overall result. The result of disabling an operation is not directly activated. This is indicated in the stack with the ‘out of sync’ exclamation mark ([Figure F.45](#)). You need to refresh the stack ([see section F.6.5](#)) for the changes to become active.

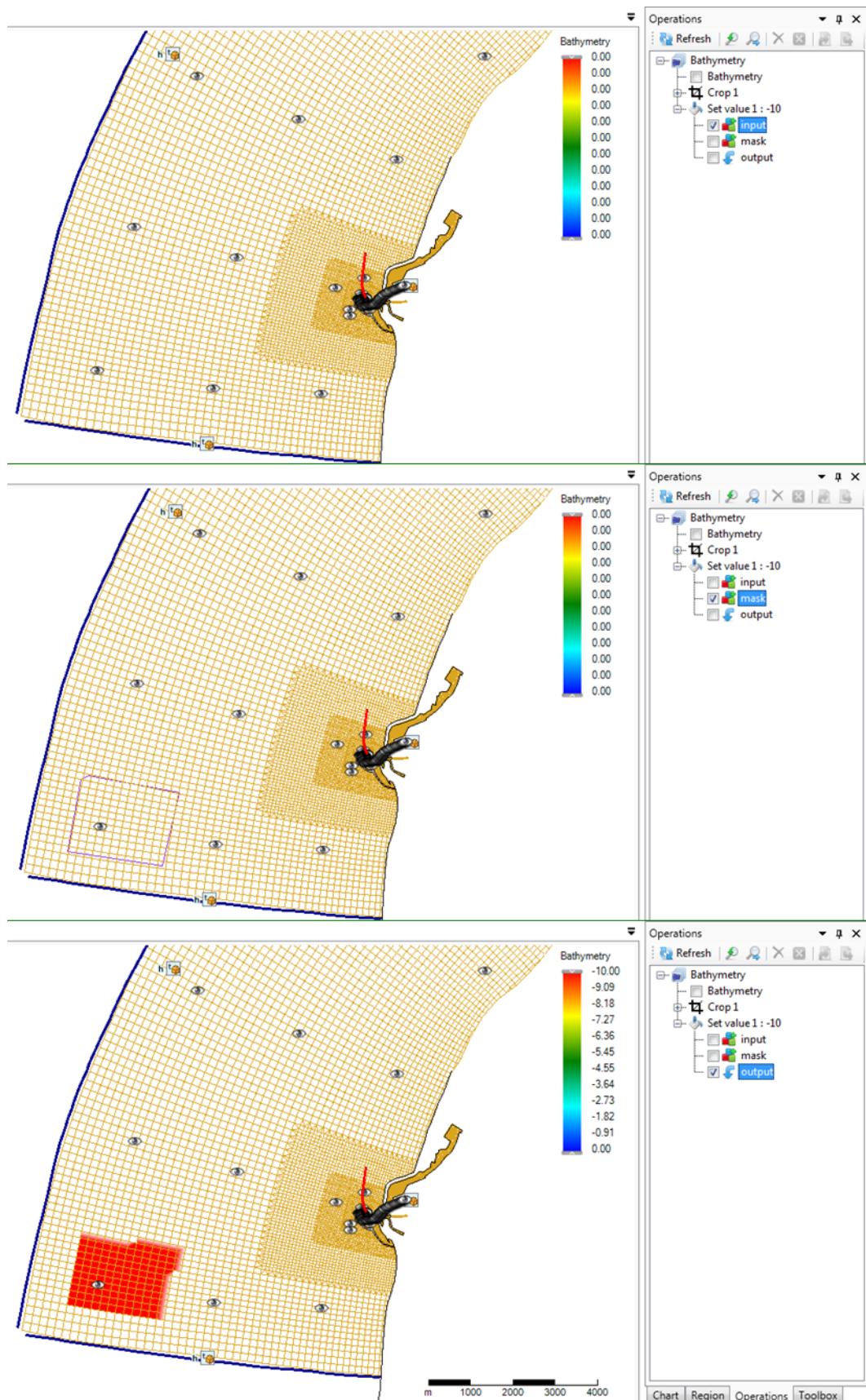


Figure F.43: Input for the operation (top panel), mask for the operation (middle panel) and output of the operation (bottom panel)

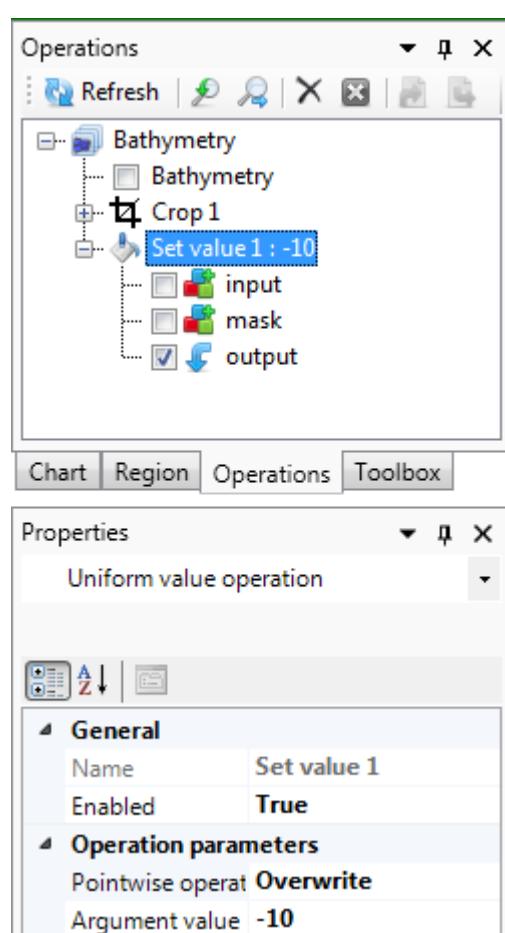


Figure F.44: Editing the value or 'Pointwise operation' of a 'Set Value' operation using the properties panel

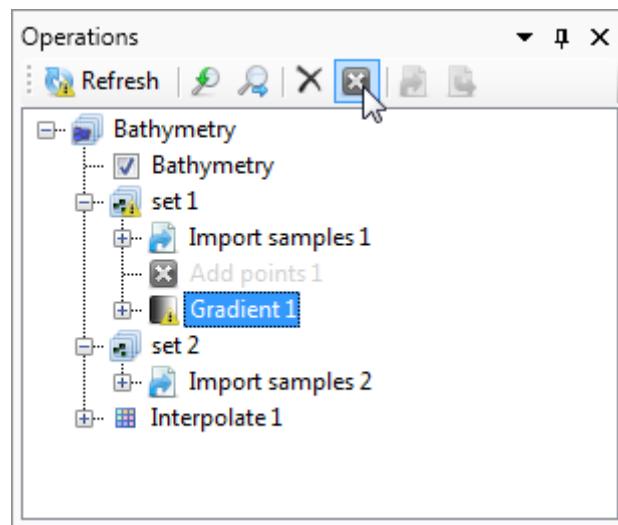


Figure F.45: Disabling an operation using the boxed cross icon in the stack menu. The operation will become grey. Note the exclamation marks marking the stack 'out of sync'.

F.6.4 Delete operations

To delete an operation permanently you have to select the operation and either press the cross icon (**Figure F.46**) or use the context menu and select delete (**Figure F.47**). The operation will be removed from the stack. The result of deleting an operation is not directly activated. This is indicated in the stack with the 'out of sync' exclamation mark. You need to refresh the stack (see [section F.6.5](#)) for the changes to become active.

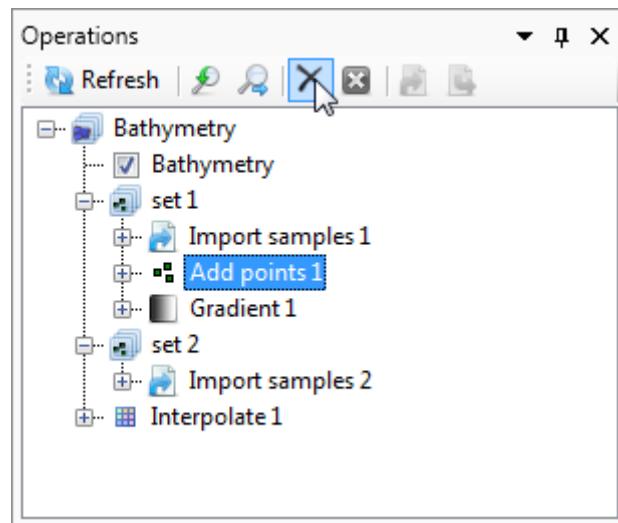


Figure F.46: Removing an operation from the stack using the cross icon in the stack menu

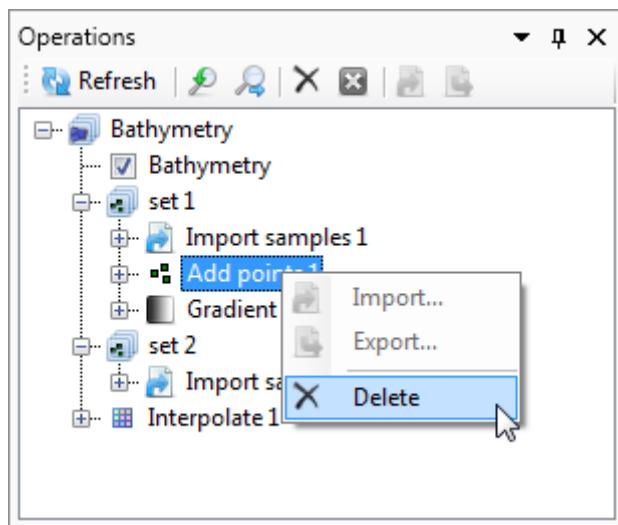


Figure F.47: Removing an operation from the stack using the context menu on the selected operation

F.6.5 Refresh stack

When the stack is marked 'out of sync' by exclamation marks, you can refresh the stack by pressing the 'Refresh' button for the changes to become active (Figure F.48). Upon refreshing the stack all the (enabled) operations in the stack will be (re-)evaluated. **Note:** Please note that refreshing the stack can take some time when large datasets are (re-) evaluated!

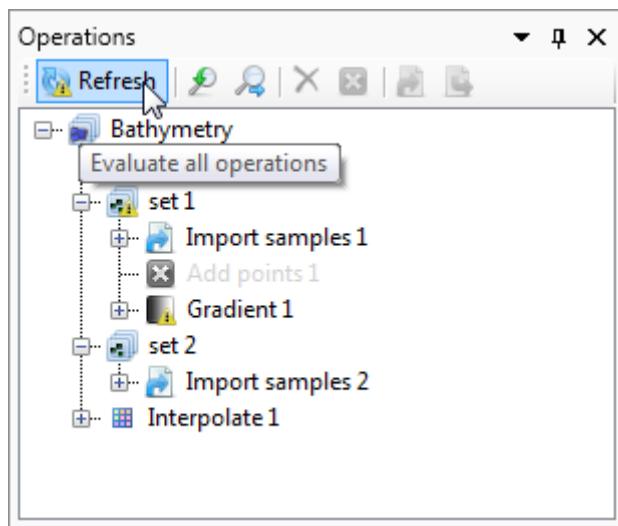


Figure F.48: Refresh the stack using the 'Refresh' button so that all operation are (re-)evaluated

F.6.6 Quick links

The stack menu contains two quick links to quickly show the original dataset (e.g. where you started from, [Figure F.49](#)) and the end result of the spatial operations ([Figure F.50](#)).

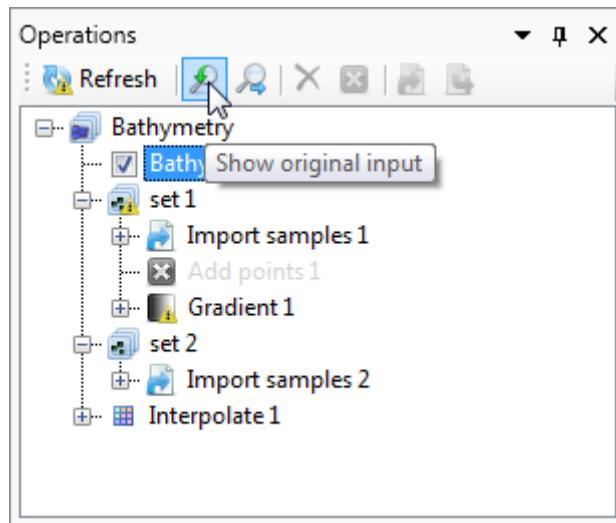


Figure F.49: Quick link to the original dataset before performing any spatial operations

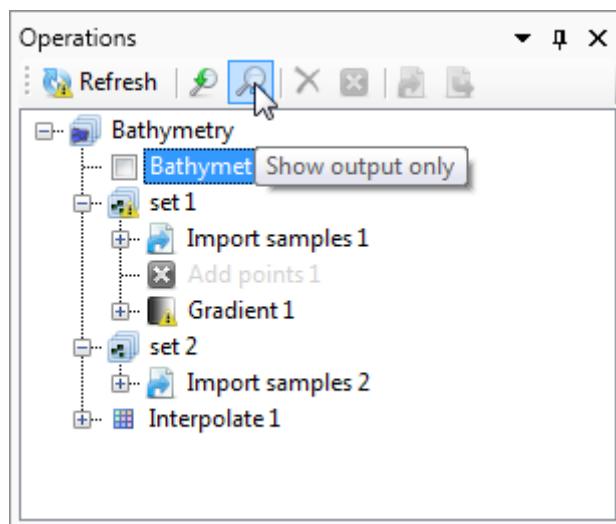


Figure F.50: Quick link to the output after performing all (enabled) operations

F.6.7 Import/export

Note: Importing and exporting data into or from the stack is still under construction





<https://beeldbank.rws.nl>, Rijkswaterstaat / Joop van Houdt



Rijkswaterstaat
Ministry of Infrastructure and the
Environment

PO Box 177
2600 MH Delft
Boussinesqweg 1
2629 HV Delft
The Netherlands

+31 (0)88 335 81 88

Deltares
Enabling Delta Life

