

MIKE FLOOD

Flood Modelling Toolbox

User Guide





PLEASE NOTE

COPYRIGHT

This document refers to proprietary computer software which is protected by copyright. All rights are reserved. Copying or other reproduction of this manual or the related programs is prohibited without prior written consent of DHI A/S (hereinafter referred to as "DHI"). For details please refer to your 'DHI Software Licence Agreement'.

LIMITED LIABILITY

The liability of DHI is limited as specified in your DHI Software Licence Agreement:

In no event shall DHI or its representatives (agents and suppliers) be liable for any damages whatsoever including, without limitation, special, indirect, incidental or consequential damages or damages for loss of business profits or savings, business interruption, loss of business information or other pecuniary loss arising in connection with the Agreement, e.g. out of Licensee's use of or the inability to use the Software, even if DHI has been advised of the possibility of such damages.

This limitation shall apply to claims of personal injury to the extent permitted by law. Some jurisdictions do not allow the exclusion or limitation of liability for consequential, special, indirect, incidental damages and, accordingly, some portions of these limitations may not apply.

Notwithstanding the above, DHI's total liability (whether in contract, tort, including negligence, or otherwise) under or in connection with the Agreement shall in aggregate during the term not exceed the lesser of EUR 10.000 or the fees paid by Licensee under the Agreement during the 12 months' period previous to the event giving rise to a claim.

Licensee acknowledge that the liability limitations and exclusions set out in the Agreement reflect the allocation of risk negotiated and agreed by the parties and that DHI would not enter into the Agreement without these limitations and exclusions on its liability. These limitations and exclusions will apply notwithstanding any failure of essential purpose of any limited remedy.





CONTENTS



1	General Description	7
2	Data	9
2.1	Result	9
2.2	Topography	9
2.3	Feature Shape	9
2.4	Map View Properties	9
3	Symbology	11
4	Flood Tools	13
4.1	Flood Mapping	13
4.1.1	Results	13
4.1.2	Bathymetry	13
4.1.3	River embankments	13
4.1.4	Buildings/structures	14
4.1.5	Additional flooded areas	14
4.1.6	Output	15
4.2	Hazard Mapping	15
4.2.1	Input	15
4.2.2	Output	18
4.3	Discharge	21
4.3.1	Input	21
4.3.2	Output	23



1 General Description

This flood tools editor is used to collect results, bathymetries and shape files for post processing of results. It is possible to view the files added in the data section in the map view. The flood tools have access to the data files.

Currently included post processing tools are flood map generation-, hazard map generation- and discharge calculation tools. A tool will use one or more of the files loaded in the data section as input and from this - plus additional user input - produce relevant output.





2 Data

This section collects all input files for Result, Topography, Feature Shape and access to Map View Properties. All files in the data section will be available for the tools.

2.1 Result

River results (MIKE1D and MIKE 11), Cartesian results (M21) and Flexible Mesh results (M21FM) are supported.

2.2 Topography

Cartesian grid (.dfs2), flexible mesh (.mesh), cross sections (.xns11) are supported. When using cross sections, only cross sections having geographical coordinates defined in the xns11 file can be imported. Additionally, dfs2-files with grid codes for additional storage area are supported. The additional storage area dfs2-file contains the cross section number (defined as a number in the Cross Section ID) in all cells which represents the additional flooded area.

2.3 Feature Shape

The feature shapes are polylines or polygons representing either river embankments, buildings, discharge lines (structures) or other shape files used as input for the flood tools.

2.4 Map View Properties

The map view properties are not directly associated with the flood tool, but the tools have access to the properties and it defines the map views coordinate system, working area and background map.

The map view supports the use of background maps. Enable 'visible' and select the desired map from the list.



Note that in order to use Google map, an internet connection must be available and the map projection must be Sphere Mercator.





3 Symbology

It is possible to change and adjust the map view symbology for river results, cross sections and feature shapes.





4 Flood Tools

There are currently three tools available for result post processing: Flood Mapping, Hazard Mapping and Discharge calculation.

4.1 Flood Mapping

The flood mapping tool maps one-dimensional results and/or two-dimensional results into a two-dimensional map by spatial interpolation.

Water levels from the results files will be spatially interpolated within the defined map area coordinates. The spatial interpolation over the map area is done within different sub-areas divided by 'Embankment lines'. No interpolation takes place over an embankment line.

Before the spatial interpolation takes places, the water levels from the 1D result files are re-distributed through the length of the cross section between marker 1 and 3 (width of the river).

4.1.1 Results

1D and 2D result which should be included in the flooding calculation is selected here. 1D results are spatially interpolated into a 2D flood map and 2D results are used where no 1D results are available. If no 2D results are selected these areas will consist of delete values.

A water depth item must usually be selected in the 2D result file. In case no bathymetry is provided, the output will be a map of water levels and hence the selected item must, in that case, be a water level item. This item is selected in the 'Data' tab under the 'Results' menu, in the 'Item' column.

4.1.2 Bathymetry

A dfs2-file with bathymetry of the area of interest. Water depth is calculated as water level minus ground level.

If no input bathymetry file has been loaded and if this list is left empty, then the mapped item will be the water level. In that case, the input dfs2 results must contain a water level item.

4.1.3 River embankments

Per default, the map extent for the 1D results is defined by the cross sections used in the river model, and results are only mapped between markers 1 and 3 from these cross sections.

River embankments are polyline shape features that may be used to represent the river embankments on one or both sides of a river. No interpolation will take place across a river embankment line.

If the 'Use river embankments for interpolation' is checked the cross sections are extended until intersection with an embankment line, in this case there will likely be a large water mass-balance error. A 1D-interpolation of water levels along the embankment lines and between the cross sections will take place before the 2D spatial interpolation.

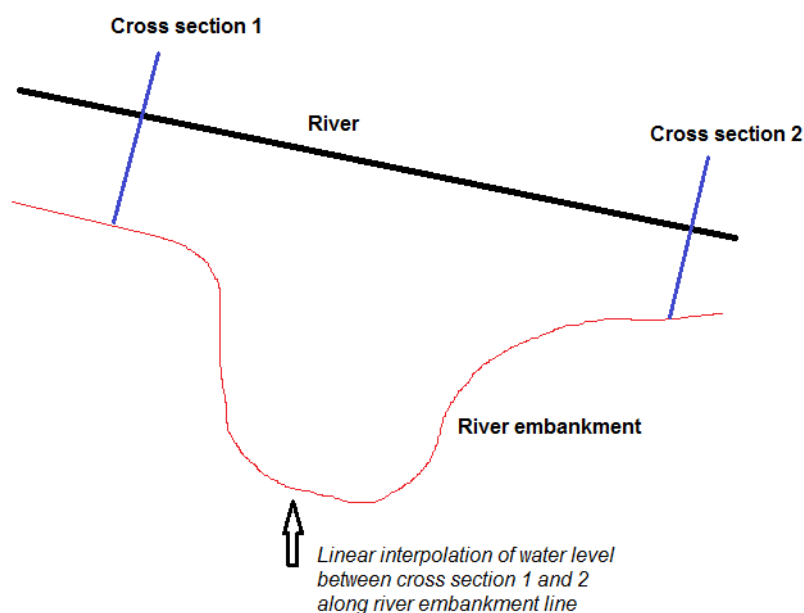


Figure 4.1 Descriptive example of river embankment

4.1.4 Buildings/structures

Polygon shape features representing buildings, roads etc. The output 2D flood map will contain delete values in the areas with buildings.

4.1.5 Additional flooded areas

Dfs2-file containing cells with grid codes in the areas with additional flooded areas connected to a river. The grid code in the dfs2 file should correspond to a cross section ID (hence the cross section ID has to be an integer number). The dfs2 file can contain many areas that will refer to the same or different cross sections. For all other cells not defined as additional storage the dfs2 file should contain delete values. Besides, the grid codes in the dfs2 file must be stored as an item type 'Grid Codes'.



An area defined in the dfs2 file will have the same water level as in the corresponding cross section and the areas do thereby not need to be directly connected with the river through the bathymetry.

4.1.6 Output

The output mesh determines the output resolution of the 2D flood map. It can be same as 2D result input, bathymetry or customized which is default. In case of customized the numbers 'Max. element area' and 'Max. number of nodes' are used.

The map area is the area of interest and output will only be calculated for this area. Output is always as dfsu, but it is possible to additionally select dfs2 which is generated based on the dfsu output. The coordinate system specified in the Map View Properties menu will be used by the output file.

The output file will cover the time frame of the 1D input file. The output will always have equidistant time steps. The output time step is defined like this.

1. Only 1D result input.

- (a) If the 1D input has equidistant time steps the output time steps will be as input.

- (b) If the 1D input has adaptive time steps:

The start time will be the same as the result file's start time.

It will have the same number of time steps as the result file.

TimeStepInSeconds is defined like

$$\frac{(\text{ResultData.EndTime} - \text{ResultData.StartTime}).\text{TotalSeconds}}{(\text{NumberOfTimeSteps} - 1)}$$

2. Combined 1D and 2D result input.

If both 1D and 2D result files were used, the output time steps will use the time steps from the file that have the shortest time steps.

4.2 Hazard Mapping

This tool post processes 2 dimensional results files (dfs2 and dfsu). It computes additional outputs such as hazard, times, etc.

4.2.1 Input

The 2-dimensional results file (dfs2 or dfsu) should contain (H, P, Q) items for dfs2 and (Surface Elevation, Total water depth, Current speed) for dfsu files.

Select the Hazard calculation method from the following options



- Hazard (v, h)
- Hazard (v * h)
- UK Methods
- Italian method

Hazard calculation method

Hazard (v, h)

For each dfs2 cell or dfsu element, the tool will read the depth and the speed values and look up the corresponding hazard value from the parameter table. In this table rows are water velocities in m/s and columns are water depths in meters.

For this method, hazard is classified into categories, corresponding to given ranges of current speed and water depths. Values of current speed provided in the table (first column) and depth (first row) represent the lower value of this range. For instance, for the example illustrated below, the hazard will be assigned the value 1 whenever the current speed will be between 0 and 1 m/s and the water depth between 0.2 and 0.4 m.

Speed/depth	0.2	0.4	10
0	1	2	3
1	2	3	3
2.3	2	3	3
3.14	2	3	3

Figure 4.2 Example of parameter table for Hazard (v, h)

Hazard (v * h)

For each dfs2 cell or dfsu element, the tool will multiply the depth and the speed values and look up the corresponding hazard value from the parameter table. In this table each column is a different set of speed*depth and its corresponding hazard value.

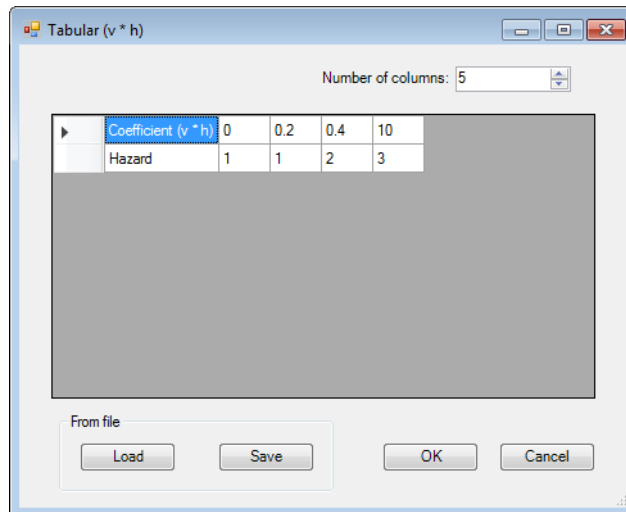


Figure 4.3 Example of parameter table for Hazard (v * h)

UK Methods

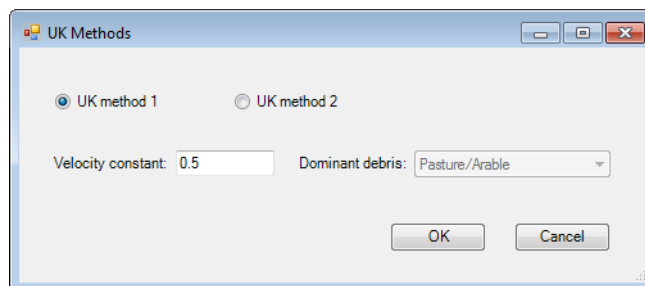


Figure 4.4 Example of input for UK Methods

Formula:

$$HR = d \cdot (v + n) + DF \tag{4.1}$$

where:

- HR* : hazard rating
- d* : depth
- v* : current speed
- n* : a constant (typically = 0.5)
- DF* : debris factor

UK-Method1:

if $d > 0.25$ then $DF = 1.0$ else $DF = 0.5$

**UK-Method2:**

DF = 0, 0.5 or 1, depending on water depth, current speed and dominant land use, with the relationship described in Table 4.1 below..

Table 4.1 Guidance on debris factors for different flood depths, velocities and dominant land uses

Depths (d)	Pasture/Arable	Woodland	Urban
0 to 0.25 m	0	0	0
0.25 to 0.75 m	0	0.5	1
d > 0.75 m and/or v > 2	0.5	1	1

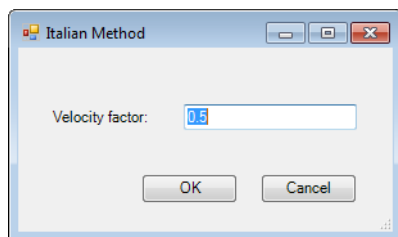
Italian method

Figure 4.5 Example of input for Italian Method

Formula:

$$HR = d + Fac \cdot \left(\frac{v^2}{2g} \right) \quad (4.2)$$

Fac would typically be either 1, 0.5 or 0.25.

4.2.2 Output

Two output files may be generated when pressing the 'Generate hazard map' button:

- One file containing the static output items: Maximum, Minimum, Time to start, Time to peak and Duration. This file will be created if at least one of these items is ticked. The file is named with the file name specified in the 'File' field underneath the table.
- One file containing the dynamic output items, i.e. instantaneous values, if the Dynamic item is ticked. This file is named with the file name specified in the 'File' field, but with the suffix '_dynamic'.



Each file may contain several variables, which are selected in the table: hazard (as defined in the Hazard group above the table), velocity, water depth and surface elevation.

Thresholds

Output values are controlled by the threshold values, specified from the Thresholds button. The threshold values are used to filter noise.

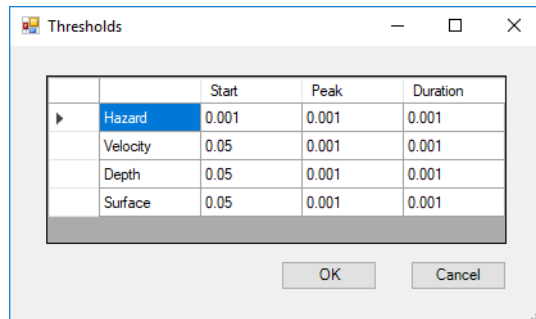


Figure 4.6 Example of threshold values

Thresholds values are only used when processing the variable they correspond to. Therefore, the calculation of hazard variable depends on hazard thresholds but not on velocity nor depth thresholds.

The Start, Peak and Duration thresholds are expressed in the same unit as the corresponding variable. The thresholds' units for velocity, water depth and surface elevation are read from the input file. The thresholds' units for hazard are expressed in hazard unit.

Maximum

The maximum value in a given cell is found by comparing the instantaneous values with the Peak threshold into account. Starting from the first value in this cell, the maximum value is updated with a new higher value from a following time step only if this new value is higher than the previous maximum value + the Peak threshold. If the values from the following time steps are higher but don't differ by more than the Peak threshold value, then the maximum value is not updated with these following values.

On the illustration below (Figure 4.7), if the difference between the value at location #1 and #3 is less than the Peak threshold, then the output maximum value is the value from location #1, which therefore slightly differs from the absolute maximum value. This may be useful in flooding applications, to catch the first value representative of the maximum, and therefore to also catch the corresponding time (see Time to peak below).

If the Peak threshold is reduced slightly, then the output maximum value will be value from location #2 and if the Peak threshold is reduced further, then



the output maximum value will be the absolute maximum value from location #3.

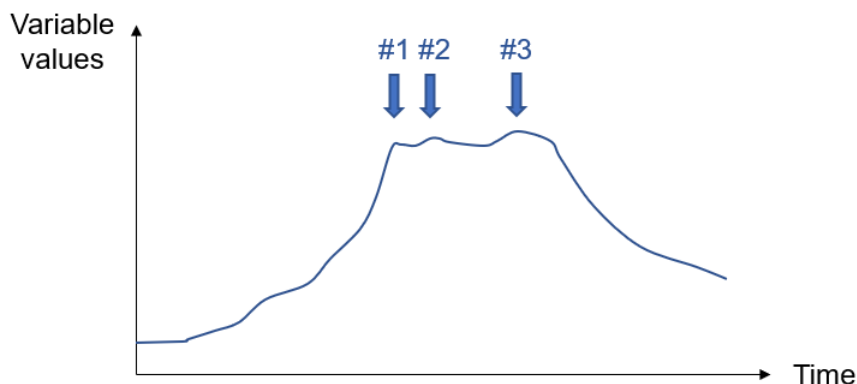


Figure 4.7 Example of captured maximum values depending on Peak threshold

Since the output maximum value may differ from the absolute maximum value, it is recommended to keep the Peak threshold relatively small. To ensure that the output maximum value is the absolute maximum value, the Peak threshold should be set to 0.

Minimum

The minimum value in a given cell is found with the same approach as for the maximum, but using instead the Start threshold. That is, the minimum value from previous time steps is updated with a new smaller value from a following time step only if this new value is smaller than the previous minimum value - the Start threshold.

Since the output minimum value may differ from the absolute minimum value, it is recommended to keep the Start threshold relatively small. To ensure that the output minimum value is the absolute minimum value, the Start threshold should be set to 0.

Time to start

If the cell is empty at the first time step, then the Time to start is the accumulated time spent before the cell is no longer empty. If the cell is not empty at the first time step, then the Time to start is the accumulated time until the variable's value differs from the value in the first time step by more than the Start threshold.

The reported Time to start is expressed in hours.

Time to peak

The Time to peak is the accumulated time until the maximum value (as defined above, considering the Peak threshold) is reached.



The reported Time to peak is expressed in hours.

Duration

The Duration is the accumulated time while the variable's value is higher than the Duration threshold value.

The Duration thresholds are expressed in the same unit as the corresponding variable (for example, the Duration threshold for Depth is per default expressed in meters, etc.). The reported duration is expressed in hours.

4.3 Discharge

This tool calculates the discharge through cross sections (polyline shape features), from both 1 dimensional and 2 dimensional results files.

The assumed positive flow is from upstream to downstream, based on the 1D network. When no 1D result is provided, the positive flow is from left to right, along the cross section line.

The discharge from the 1D network is the exact discharge obtained from the closest calculation point. The discharge from the 2D domain is the accumulated discharge accumulated from the various calculation points along the cross section.

4.3.1 Input

The input parameters must be specified by the following:

- 'From' and 'To' dates that must be within the period covered by all results files.
- The shape file containing the cross sections through which the discharge will be computed must be selected, from the list of shape files loaded in the Buildings/structures menu. Line feature class of cross sections requires a field used for naming output time series. The cross section must be within the 2D area and cannot cross the same 1D branch more than once. Flow across different branches is supported but in that case the line must cross both 1D branches in the same direction (e.g. from the left bank to the right bank).
- Output time series name that will be used as base name for naming output time series
- Results file (in same coordinate system)
 - 2 dimensional .dfs2 or .dfsu file
 - And/or 1 dimensional .res11 or .res1D file



dfsu

The tool can operate with dfsu input files with the following items:

1. p, q
2. h, u, v
3. s, d, u, v
4. s, u, v

where the items are described in Table 4.2.

Table 4.2 Items in dfsu file

Name	Name in MIKE 21 Flow Model FM GUI	EUM Item description	EUM Item type
p	P flux	Flow Flux	eumIFlowFlux
q	Q flux	Flow Flux	eumIFlowFlux
h	Total water depth	Water depth	eumIWaterDepth
d	Still water depth	Still water depth	eumIStillWaterDepth
s	Surface elevation	Surface elevation	eumISurfaceElevation
u	U-velocity	u-velocity component	eumIuVelocity
v	V-velocity	v-velocity component	eumIvVelocity

If more items are available the tool will use the first available combination of parameters above, with the indicated priorities.

The following has to be noted:

- Flux on open boundaries will not be included.
- It is not possible to get correct calculations for partially wet elements since the result file has insufficient information for this. Flux on faces with neighbor cell that has delete values will be set to zero.
- The flux on a face in the input result file is an approximation to the 'real' flux used in the engine that produced the result file.



dfs2

dfs2 input files has to be data file type MIKE21 HD simulation output (H, P, Q) where the items are described in Table 4.3.

Table 4.3 Items in dfs2 file

Name	Name in MIKE 21 Flow Model GUI	EUM Item description	EUM Item type
P	P flux	Flow Flux	eumIFlowFlux
Q	Q flux	Flow Flux	eumIFlowFlux
H	Water depth	Water depth	eumIWaterDepth

4.3.2 Output

The output data is specified as follows:

- One time series per valid cross section line with 3 items (1D flow, 2D flow and Total flow). The output timestep is taken from the result file with the highest temporal resolution. The other time series is interpolated.

When working with a combined flow (1D+2D), the tool checks the river network (saved in .res11 or .res1D) and ensures the extraction of the 2D results follows the same direction, changing the sign of the 2D time series if needed.

In that way, the result does not depend on how the user digitalised the cross section but is based on the river flow direction (upstream to downstream being positive flow).

When no 1D result is provided, the tool does not know the expected flow direction and the default approach is used (positive flow is from left to right, when looking from the first digitized point towards the next points along the cross section).





INDEX



Numerics

2D flood map 15

P

Polygon 9

Polyline 9, 14