

Reference

Content

1 Reference.....	2
2 History.....	8

1 Reference

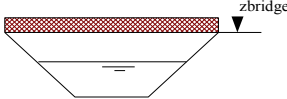
Input	Unit	Description
title 'name'	string	'name' is a string (max. 64 characters) that is stored on the result file and appears in the header of the plots
>>global		main keyword for the definition of default values holding for all models
kst r	m ^{1/3} /s	global Strickler value (default=30)
n r	SI	default Manning's n value
ks r	m	equivalent sand roughness diameter
damping_on		considers reduction of turbulent shear forces for small flow depths (Bezzola 2002) in combination with logarithmic friction law (sand roughness ks).
label 'xx' r	string	name of a label (max. 4 characters) to which the Strickler-value r is related
hdry r	m	minimum flow depth where flow equations are being solved (for 2D model, default=0.01 m)
adry r	m ²	minimum wetted area of cross-section where flow equations are being solved (for 1D model, default=0.1 m ²)
slot_width r	m	width of (Preissmann-) slot used for calculation of pressurised flows (see option >>branch/closed) (default=0.1 m)
unit 'name'	string	Defines unit of input data. Possible units are: - 'minutes' for time given in minutes (instead of hour)
date 'mmdh'	string	real time of start of simulation (mm=month, dd=day, hh=hour)
yield_stress	N/m ²	yield stress for debris flow calculations (default = 0)

parameters for integration of 1D models

seam_radius r	m	maximum distance from section-midpoints to cell-boundaries (of 2D mesh) where a seam (flow) can exist (used for connecting different flow models) (default = 10 m)
weir_coefficient	-	Poleni coefficient for calculation of fluxes between 1D and 2D models (default = 0.60)

>>sediment		<i>parameters for mobile bed calculation</i>
thcrit r	-	critical shields factor for MPM and Smart/Jäggi formula (default=0.05)
repose r	-	tangens of angle of repose of bank material (default=1.0)
density r	kg/m ³	density of the bed material (default=2650 kg/m ³)
porosity r	-	porosity of the bed material (default=0.30)
formula 'name'	-	sediment transport formula to be used: <ul style="list-style-type: none"> • mpm = Meyer-Peter/Müller formula. • smart&jaeggi = Smart/Jäggi formula • rickenmann90 = Rickenmann formula
rock_thcrit r	-	critical shields factor for transport over bedrock (default=0.01)
rock_factor r	-	factor to account for transport over bedrock (default=1.8)

Input	Unit	Description
mpm_factor r	-	factor used in transport formula of Meyer-Peter/Müller (default=8.0)
mixture 'name' 0.2 0. 25. 1.	cm	grain size distribution of sediment mixture where the the grain size [cm] and the cumulative probability (sediment finer) are given in the 1 st and 2 nd column. Note: The last value in the 1 st column must be 1.0.
>>compute		to define parameters for unsteady flow computation
start r	h	start time of the simulation (default=0)
end r	h	time where simulation will end (default=100h)
cfl r	-	limiting CFL number to estimate size of time step (default=0.6)
frequency i	-	refresh rate of display output (default=100).
batch_mode		runs the model in batch_mode, i.e. starts the computation, stores the results on the specified file, and terminates
>>create_model		main keyword for the specification of model-specific parameters
name 'name'	string	name of model (displayed on model output)
type '2D'		type of model: <ul style="list-style-type: none"> • '1D' for one-dimensional flow calculations (river branch) • '2D' for two-dimensional flow calculations
>>init		definition of the calculation domain
mesh 'name'	string	reads the mesh geometry from files. Supported formats are: .node created by program TRIANGLE (file name without suffix!) .2dm created by program SMS (splits 4-noded elements to triangles) .tin triangulated irregular network format (used in ems-i programs)
binary 'name'	string	reads mesh geometry and initial conditions (flow depths, flow and bedlevels) from a binary file created by a previous run. <i>Note: Use either mesh or binary (not both) for the definition of the mesh geometry.</i>
at r	h	time level r of initial condition to be read form the binary file.
bedlevel =+-<> 'name'		reads a mesh geometry from files created by program triangle. Depending on the operator it changes the level of the model bed. Example: project = 'new_dam.1' will read the bedlevel values from the (project-) mesh defined by the files new_dam.1.node and new_dam.1.ele. Possible operators are: = new bedlevel + lift the bedlevel - lower the bedlevel < maximum bedlevel > minimum bedlevel <i>Note: Must be used in combination with mesh (not binary).</i>

Input	Unit	Description
waterlevel =+-<> 'name'		Same as option bedlevel (see row above) but operates on waterlevels. <i>Note: Must be defined after definition of bedlevels.</i>
>>polygon		used to define values that hold in a domain defined by a closed polygon. Example: <pre>>>polygon !keyword n 0.02 !keyword for Manning's n value 100. 150. !list of vertices of polygon 320. 165. 240. 190. 105. 155.</pre>
bedlevel =+-<> r	m	to modify the bedlevels by a value r using an operator (see keyword 'project' for meaning of operators). Example: <pre>bedlevel > 321.0 ! minimum bed level is 321 m 100. 150. 320. 165. 240. 190. 105. 155.</pre>
flowdepth r	m	flow depth at start time (initial condition)
ks r	m	equivalent sand roughness diameter
kst r	Sl	Manning-Strickler value
n r	Sl	Manning's n value
vegetation r	1/m	vegetation factor given by the formula $vegetation = \frac{d}{a^2} c_w$ with d = diameter of vegetation elements [m], a = distance between elements [m] and c _w = drag coefficient (range 0.8 - 1.5)
waterlevel r	m	water level at start time (initial condition)
bridge r	m	level of a bridge (z _{bridge}) to account for backwater effects. It accounts for the acceleration of the flow due to the reduced flow section. It does <u>not</u> account for external forces on the bridge plate or other effects such as flow contraction (gated flows). 
no_seam	-	cells that do not connect to 1D-branches
>>boundary		used to define (time-dependent) boundary conditions at the model boundaries that are inside a polygon. The polygon covers all the edges of the calculation mesh where the boundary condition holds. It must not match exactly with the edges. The steps are: (1) Define a boundary type (e.g. an inflow) (2) Define the location where the boundary holds by a closed polygon using the keyword location.

Input	Unit	Description
inflow r	m ³ /s	defines an inflow discharge (** for time dependent inflow). The discharge is distributed among the boundary cells assuming uniform flow conditions given the slope of the energy head (uniform_slope, default = 0.001).
uniform_slope r	-	
critical		defines an outflow boundary with a critical flow regime (no backwater effect)
slope r	-	defines an outflow boundary with r = energy slope
waterlevel r	m	defines an outflow boundary with r = water level
stage-discharge		defines an stage-discharge outflow boundary with z=water level [m] and q=discharge [m ³ /s]
z1 q1		
z2 q2		
.	.	

Example (i) Given an inflow of 100 m³/s at a boundary where the mean slope is approx. 0.5%. The boundary condition reads

```
>>boundary
inflow 100
uniform_slope 0.005
location
100. 150.
100. 150.
200. 200.
100. 200.
```

Example (ii) At an outflow boundary the water level rises from 96.5 m to 98.0 m during half an hour and returns to the old value after one hour. The outflow has to be stored on the file 'waterlevel.out' for further usage. The boundary condition reads

```
>>boundary
waterlevel ** > 'waterlevel.out'
0.0 96.5
0.5 98.0
1.0 96.5
location
900. 150.
930. 155.
950. 240.
910. 260.
```

>>structure

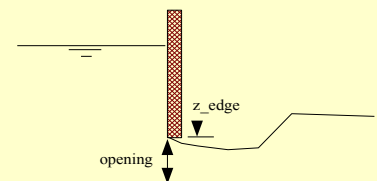
to define internal sources and structures (culverts, weirs)


Example (i) Given a sharp crested weir crest level 408.40 m and Poleni coefficient 0.75.

```
>>structure
weir_coefficient 0.75
weir_crest 408.40
location
100. 345.
150. 375.
148. 380.
98. 350.
```

point_source x y q m,m³/s defines an internal source with x,y = co-ordinates of source position and q = constant discharge. For unsteady inflows write '**' and add a timetable of the inflows.

Input	Unit	Description
culvert x1 y1 x2 y2	m	defines the flow through a circular or rectangular culvert with (x1,y1) and (x2,y2) = co-ordinates of the in- and outlet. The module accounts for in- or outlet controlled flow condition. It is assumed that the vertical level of the in- and outlet corresponds to the bed level of the adjacent grid cell. Example: culvert x1 y1 x2 y2 > 'name' writes the discharge through the culvert to file 'name'
diameter r	m	diameter of circular culvert (default= 1 m)
width r	m	width of rectangular culvert
height r	m	height of rectangular culvert.
n r	SI	Manning's n value of culvert [default= 0.02].
kst r	SI	Strickler value of culvert [default= 50].
inlet_loss r	-	inlet loss coefficient that depends on shape of culvert inlet. Values usually vary between 0.2 (rounded entrance) and 0.7 (sharp crested entrance)(default= 0.5).
maximum r	m ³ /s	maximum discharge through culvert (** for time table).
weir zw cw		Flow over weir with zw = level of weir crest [m] and cw = poleni coefficient (default = 0.58). Example: weir 433.65 0.64 location ... Time_dependent weir levels are defined in a table where the time [h] and the weir crest are given in the first and second column, respectively. Example: weir ** 0.64 0.0 433.65 0.8 434.15 1.5 433.80 location ...
gate zg cg	m,-	Flow through gate with zg = level of the lower end of the sluice gate (see figure) and cg = contraction coefficient (default = 0.62). Example: gate 426.45 0.80 location ... Time_dependent gate openings are defined in a table where the time [h] and the opening are given in the first and second column, respectively. Example: gate ** 0.80 0.0 426.45 0.1 425.00 location ...
control x y zref	m,m,m	Water level control with reference zref [m] at position defined by co-ordinates x,y.



Input	Unit	Description
<pre>control_param dt u location x1 y1 x2 y2 . .</pre>	s,m/s	<p>free parameters for level control (fictitious weir):</p> <ul style="list-style-type: none"> • dt = time lag [s] between adjustment of weir level • u = velocity [m/s] of weir level adjustment <p>Example:</p> <pre>control 645129.5 312294.80 433.65 control_param 300. 0.002 location ... location x1 y1 x2 y2 . .</pre> <p>location of weir/gate/control section defined as a set of vertices with x- and y- coordinates in first and second column, respectively.</p>
<pre>precipitation r</pre>	mm/h	intensity of the precipitation over the whole area. For unsteady values a time table can be given. Evaporation can be modelled with values $r < 0$.
<pre>pile x y d cD</pre>		accounts for drag forces on a pile (pier) at location x,y with d = diameter [m], c_D = drag coefficient (default = 1.0 for circular shape).
<pre>cross-pile z d cD xA yA xB yB</pre>		<p>accounts for drag forces on a horizontal element that spans from A to B with z = level [m], d = diameter [m], c_D = drag coefficient (default = 1.0 for circular shape).</p>  <p>Hint: Cross-piles can be used to model drag forces due to bridge plates.</p>
<pre>>>breach level ** t1 z1 t2 z2 t3 z3 . . location . . unlimited</pre>		<p>is used to define breaches that develop during computation</p> <p>table containing the time level in the first column and the corresponding level of the breach in the second column. Level does not exceed initial bed level except with keyword <code>unlimited</code> (see below).</p> <p>co-ordinates of polygon vertices to define location of the breach.</p> <p>moves bedlevels independent of initial value.</p>
<pre>>>output to 'name' interval r hydrograph 'item' x y > 'filename'</pre>	string h	<p>is used to define the output from the model</p> <p>results are written to file <code>name</code> (default: res)</p> <p>interval between time steps to be stored on result file (default = 1 h)</p>

Input	Unit	Description										
		writes nodal value at location (x,y) as a hydrograph table to file. Accepted items are:										
		<table border="1"> <tbody> <tr> <td>'bedlevel'</td> <td>bed level [m s. l.]</td> </tr> <tr> <td>'waterlevel'</td> <td>water level [m s. l.]</td> </tr> <tr> <td>'depth'</td> <td>flow depth [m]</td> </tr> <tr> <td>'flow'</td> <td>Specific flow [m2/s]</td> </tr> <tr> <td>'velocity'</td> <td>flow velocity [m/s]</td> </tr> </tbody> </table>	'bedlevel'	bed level [m s. l.]	'waterlevel'	water level [m s. l.]	'depth'	flow depth [m]	'flow'	Specific flow [m2/s]	'velocity'	flow velocity [m/s]
'bedlevel'	bed level [m s. l.]											
'waterlevel'	water level [m s. l.]											
'depth'	flow depth [m]											
'flow'	Specific flow [m2/s]											
'velocity'	flow velocity [m/s]											
<code>write 'item' at r > 'filename'</code>		write nodal values at time r [h] to file. See table above for accepted items.										
<code>flow > 'filename'</code> <code>x1 y1</code> <code>x2 y2</code> <code>. .</code>		writes total flow across a section to file 'filename'. The section must be defined as a polygon with vertices x,y. The location of edges can be displayed with Show/Mesh option. Note that the direction of the flow is <u>not</u> considered..										
<code>>></code>		denotes the end of the input. Any further input is ignored.										

2 History

Version 2.1 (2009-)

- accepts mesh geometries in .2dm format created by SMS (Surface water Modeling System) and .tin format (see [description](#))
- serial linking of 2d and 1d models
- cross-piles to account for hydraulic resistance of horizontal structures such as bridges
- export of results in (ESRI) shape format
- simplified input for flow over weirs and gated flows
- modeling of water level controls (e.g. hydropower stations)
- modeling debris flow with two-parameter approach (turbulent & yield)

Version 2.0 (2006-2009)

- considering multiple models (1D and 2D)
- accepts project files to change river bed topography
- new flow option

Version 1.3 (2004-2005)

- stage-discharge outflow boundary
- development of breaches during simulation time
- export of hydrograph tables
- variable bed_evolution values and boundary conditions
- bed armouring (2-grain-size model)
- automatic generation of animated output (movies)
- perspective views with shading
- modelling of precipitation/evaporation
- energy losses due to vegetation

Version 1.2 (2002-2003)

- multiple (independent) meshes
- mobile bed module
- improved interpolation of bedlevels for narrow dams
- improved graphic routines
- output of multiple time steps on same result file
- logarithmic friction law
- backwater effects due to bridges

Version 1.1 (2001-2002)

- time dependent boundary conditions
- distributed inflow discharge assuming uniform flow conditions
- culvert flow to connect model domains
- weir and gated flow over cell edges

Version 1.0 (1999-2000)

- shallow water solver on unstructured meshes
- steady boundary conditions (inflow, outflow, waterlevel, energy slope)
- friction values (kst or n) defined with closed polygons (mesh independent)
- wetting and drying of cells