

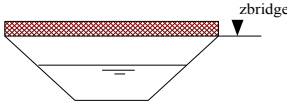
Reference

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
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)
<i>parameters for integration of 1D models</i>		
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)
>>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

Input	Unit	Description
mesh 'name'	string	reads mesh geometry from previously created files using program TRIANGLE
binary 'name'	string	reads mesh geometry and initial conditions (flow depths, flow and bedlevels) from a binary file created by a previous run. Note: Use either <code>mesh</code> or <code>binary</code> (not both) for the definition of the mesh geometry.
project =+-<> 'name'		reads a mesh geometry from files created by program triangle. Depending on the operator it changes the bedlevels of the calculation mesh. 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
>>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: bedlevel > 321.0 ! minimum bed level is 321 m 100. 150. 320. 165. 240. 190. 105. 155.
bridge r	m	level of a bridge (z_{bridge}) to account for backwater effects 
flowdepth r	m	flow depth at start time (initial condition)
ks r	m	equivalent sand roughness diameter
kst r	SI	Manning-Strickler value
n r	SI	Manning's n value

Input	Unit	Description
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)
>>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 <code>location</code> .
inflow r	m ³ /s	defines an inflow discharge. 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		
.		

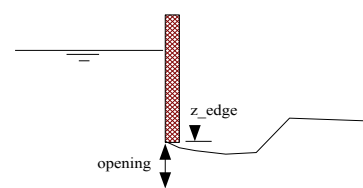
Example (i) Given an inflow of 100 m³/s at a boundary where the mean slope is approx. 0.5%. The boundary condition reads

```
>>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.
```

Input	Unit	Description
>>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. <pre>>>structure weir_coefficient 0.75 weir_crest 408.40 location 100. 345. 150. 375. 148. 380. 98. 350.</pre>
point_source x y q	m,m ³ /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.
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).
weir_coefficient r		poleni coefficient for weir flow (default=0.58). Note: The weir_coefficient applies to all weirs listed below until an other weir_coefficient is defined.
weir_crest r	m	height of weir crest. Time_dependent levels are defined in a table where the time [h] and the weir_crest are given in the first and second column, respectively (see example below). <pre>weir_crest ** 0 325. 1 320. 1.5 319.</pre>
gate_coefficient r		contraction factor for gated flow (default=0.62). Contractions starts from 1.0 if the water level equals the edge of the gate and reduces to the given coefficient. Note: The coefficient applies to all gates listed below until an other coefficient is defined.
gate_edge r	m	level of the lower end of the sluice gate (see figur above).



Input	Unit	Description										
location . .		co-ordinates of polygon vertices to define location of weir or gate flow. Must contain cell edges where weir/gate flow occurs (important: only one edge per cell !!). Weir/gate edges show up as red lines using the Show/Mesh option.										
precipitation r	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 $r < 0$.										
pile x y d cD		accounts for drag forces on a pile (pier) at location x,y with d= diameter [m], cD = drag coefficient (default = 1.0 for circular shape).										
>>breach		is used to define breaches that develop during computation										
level ** t1 z1 t2 z2 t3 z3 . .		table containing the time level in the first column and the corresponding level of the breach in the second column.										
location . .		co-ordinates of polygon vertices to define location of the breach.										
>>output		is used to define the output from the model										
to 'name'	string	results are written to file name (default: res)										
interval r	h	interval between time steps to be stored on result file (default = 1 h)										
hydrograph 'item' x y > 'filename'		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 [m ² /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 [m ² /s]											
'velocity'	flow velocity [m/s]											
write 'item' at r > 'filename'		write nodal values at time r [h] to file. See table above for accepted items.										
flow > 'filename' x1 y1 x2 y2 . .		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..										
>>		denotes the end of the input. Any further input is ignored.										

2 History

Version 2.0 (2006-)

- considering multiple models (1D and 2D)
- project files
- 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
 - Smart/Jäggi and Meyer-Peter/Müller formula
 - bed rock
 - streamwise sloping bed correction
 - bank failure approach
- bug in the drying routine removed
- 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