Reference

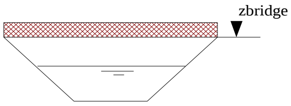
Content

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 logaritmic 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 'mmddhh'	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)
friction_slope	-	Tangens of internal friction angle $\tan\delta$ for debris flow calculations (default = 0) with viscous stress $\tau_y = \rho\, g\, h \tan\delta$. Note: If both yield stress and friction slope are defined, the sum of the two values is taken.
bingham_viscosity	Pa s	Bingham viscosity μ_B for debris flow calculations (default = 0) with Bingham shear stress $\tau_B = \frac{3\mu_B q}{h^2}$. Note: If both turbulent friction (n, kst, or ks) and bingham viscosity is defined, the maximum of the two resulting stress values is taken.
roughness_factor	-	factor to estimate roughness diameter from mean grain-size
		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		*parameters for mobile bed calculation*
thcrit r	-	critical shields factor for MPM and Smart/Jäggi formula (default=0.05)

Input	*Unit*	*Description*
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: • **mpm** = Meyer-Peter/Müller formula. • **smart&jaeggi** = Smart/Jäggi formula • parker78 = Parker formula for uniform grain size
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)
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 1st and 2nd column. Note: The last value in the 1st 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: • '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. *Note: Use either **mesh** or **binary** (not both) for the definition of the mesh geometry.*
at r	h	time level r of initial condition to be read form the binary file.

Input	***Unit***	***Description***
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: bedlevel = 'new_dam.1' will read the bedlevel values from the 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 *Note: Must be used in combination with mesh (not binary).*
waterlevel =+-<> 'name'		Same as option bedlevel (see row above) but operates on waterlevels. *Note: Must be defined after definition of bedlevels.*
>>polygon		used to define values that hold in a domain defined by a closed polygon. Example: **>>polygon !keyword** **n 0.02 !keyword for Manning's n value** **100. 150. !list of vertices of polygon** **320. 165.** **240. 190.** **105. 155.**
bedlevel =+-<> r	m	to modify the bedlevels by a value r using an operator (see keyword `>>init/bedlevel` for meaning of operators). Example: **bedlevel > 321.0 ! minimum bed level is 321 m** **100. 150.** **320. 165.** **240. 190.** **105. 155.**
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
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). Can be used to account for drift wood effects (see >>drift).
waterlevel r	m	water level at start time (initial condition)

Input	Unit	Description
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 not account for external forces on the bridge plate or other effects such as flow contraction (gated flows). 
no_seam **x1 y1** **x2 y2** **. .**	-	closed polygon to define 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**.
inflow r **uniform_slope 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).
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 **z1 q1** **z2 q2** **. .**		defines an stage-discharge outflow boundary with z=water level [m] and q=discharge [m^3/s] Example (i) Given an inflow of 100 m^3/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.
```

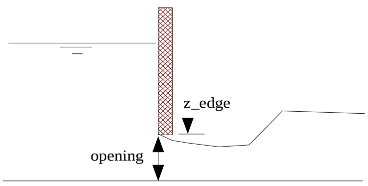
Input	*Unit*	*Description*
		Example (ii) At an outflow boundary the water level rises from 96.5 m to 98.0 m during half an hour and returns to the old value after one hour. The outflow has to be stored on the file 'waterlevel.out' for further usage. The boundary condition reads

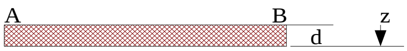
```
>>boundary
  waterlevel ** > 'waterlevel.out'
    0.0 96.5
    0.5 98.0
    1.0 96.5
  location
    900. 150.
    930. 155.
    950. 240.
    910. 260.
```

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.

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

Input	*Unit*	*Description*
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).
maximum r	m^3/s	maximum discharge through culvert (** for time table).

Input	*Unit*	*Description*
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_gauge x y z	m,m,m	water level control with reference level z [m] at position defined by co-ordinates x,y.
control_param dt u	s,m/s	free parameters for level control (fictitious weir): ● dt = time lag [s] between adjustment of weir level ● u = velocity [m/s] of weir level adjustment Example: **control_gauge 645129.5 312294.80 433.65** **control_param 300. 0.002** **location** ...
location **x1 y1** **x2 y2** . .		location of weir/gate/control section defined as a set of vertices with x- and y- coordinates in first and second column, respectively.
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.

Input	***Unit***	***Description***
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).
cross-pile z d cD **xA yA** **xB yB**		accounts for drag forces on a horizontal element that spans from A to B with z = level [m], d = diameter [m], cD = drag coefficient (default = 1.0 for circular shape).  Hint: Cross-piles can be used to model drag forces due to bridge plates.
>>variation		is used to define variation of model parameters during simulation
bedlevel =+-<> ** **t1 r1** **t2 r2** **. .**	m	definition of time-dependent values of bed level using an operator (see keyword `>>init/bedlevel` for meaning of operators).
rock_level = ** **t1 r1** **t2 r2** **. .**	m	definition of time-dependent values of rock level (for bed load calculation).
location **. .**		co-ordinates of polygon vertices to define location of variation.
deposit r1 r2 r3 r4 r5		adds depth of flow to the bed level (to account for deposition of debris flow) at time `r1` [h]. Simultaneous the global (rheological) parameters can be changed: `kst (r2)`, `yield_stress (r3)`, `friction_slope (r4)`, and `bingham_viscosity (r5)`
>>drift		is used to define effects of coarse woody debris (CWD). The shape of the CWD is a cylindrical log of which the diameter is D and the length is L = 10*D.
float r	-	relative flow depth h/D where logs becomes mobile (default = 0.5)
logdiameter r1 r2	m	minimum and maximum diameter of logs.
load r	m^3/h	volume of CWD per hour that is given into the model. For variable values replace `r` by `**` and add a table with the time [h] and the corresponding load [m^3/h] in the 1^{st} and 2^{nd} column.
location **x1 y1** **x2 y2** **. .**		location of CWD source (feeder) defined as a polygon
>>sediment		the following data is related to bed load calculations
dmo r	cm	reads mean diameter of grains (valid for whole domain)
dmo 'name'	string	reads variable grain size as a mesh created by program triangle ("name.node" and "name.ele"). To be used for uniform grain size calculation (unit [cm]).

Input	*Unit*	*Description*
inflow r	kg/s	sediment inflow (r=** for unsteady inflows).
location **x1 y1** **x2 y2** **. .**		location of sediment inflow (feeder) given as a closed polygon
rock_depth r **x1 y1** **x2 y2** **. .**	m	defines a rock level below the initial bed level in a closed polygon defined by the co-ordinates x,y

>>output		is used to define the output from the model
prefix 'name'	string	results are written to file **name.res** (binary for review) and `name.out` (input reading)
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:

`'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]

write 'item' at r > 'filenname'

write nodal values at time r [h] to file. See table above for accepted items.

flow > 'fname' **x1 y1** **x2 y2** **. .**	writes total flow across a section to file 'fname'. 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..
sediment > 'fname' **x1 y1** **x2 y2** **. .**	writes sediment flow across a section to file 'fname'. 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 2.3 (2014-)

- >>variation for time-dependent rocklevels
- adjustment of stress terms for debris flow calculations

Version 2.2 (2011-2013)

- module to account for coarse woody debris
- bingham like friction law for debris flow calculations.
- >>variation for time-dependent values (replaces >>breach)

Version 2.1 (2009-2011)

- 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. hydro power 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 bed level 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