Reference

- supports parallel processing

Content

1 Reference

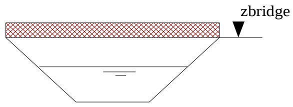
Input	Unit	Description
`title 'name'`	string	**'name'** is a string (max. 64 characters) that is stored on the result file and appears in the header of the plots
`>>global`		main keyword for the definition of default values holding for all models
`kst r`	$m^{1/3}/s$	global Strickler value (default=30)
`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
`plot_interval`	h	interval to store sediment output on file sed.out (default= 8760)
`num_threads i`	-	Maximum number of threads for computation (default = 1). Speeds up calculation on systems with multi-core (multi-thread) CPU's.
`>>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)

Input	*Unit*	*Description*
`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.
`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).

Input	***Unit***	***Description***
`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`
`x1 y1`
`x2 y2`
`.  .`
	-	closed polygon to define cells that do not connect to 1D-branches.

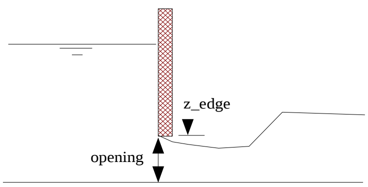
`>>boundary`		used to define (time-dependent) boundary conditions at the model boundaries that are inside a polygon. The polygon covers all the edges of the calculation mesh where the boundary condition holds. It must not match exactly with the edges. The steps are: (1) Define a boundary type (e.g. an 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

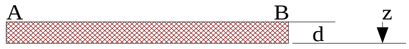
```
>>boundary
  inflow 100
  uniform_slope 0.005
  location
    100. 150.
    100. 150.
    200. 200.
    100. 200.
```

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.**
>>structure		to define internal sources and structures (culverts, weirs, controls)
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). Flow over weir can be written to a file by adding > 'filename'. Example: **`weir 433.65 0.64 > 'weirflow.out'`** **`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). Flow through gate can be written to a file by adding > 'filename'. Example: **`gate 426.45 0.80 > 'gateflow.out'`** **`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 values can be written to a file by adding > 'filename'.
`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
`control_init zw`	m	initial level of (fictitious weir) crest (default: control level) Example: **`control_gauge 29.5 94.8 433.65 >'control.out'`** **`control_param 300. 0.002`** **`control_init 433.20`** **`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.

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 that logs become 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 3.0 (2016-)

- unlimited model size (allocatable arrays)
- parallel processing (multi-thread CPU's)
- initial weir levels (for waterlevel controls)

Version 2.3 (2014-15)

- >>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 1.0 - 1.3 (1999-2005)

- 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
- 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
- time dependent boundary conditions
- distributed inflow discharge assuming uniform flow conditions
- culvert flow to connect model domains
- weir and gated flow over cell edges
- friction values (kst or n) defined with closed polygons (mesh independent)
- wetting and drying of cells