Reference

- supports parallel processing

Content

1 Introduction

FLUSH is a program that allows to estimate discharge, velocity distribution and other hydraulic variables in cross-sections and river channels. It assumes non-uniform one-dimensional flow conditions. The main features are

- Manning-Strickler formula for bed shear.
- Unsteady flow calculations for sub- and supercritical flows
  - spatial discretization with Finite-Volume-Method (Roe fluxes)
  - time solution with explicit Euler scheme

- Meyer-Peter/Müller and Smart&Jäggi for uniform grain size
- Wilcock&Kenworthy for two grain sizes (sand and gravel)
- Parker (1990) for sediment mixtures

This documentation describes how to use **FLUSH** for your specific river data. Before starting you should have specified the input data on an input file (standard name is 'cin').

Important: FLUSH does not calculate on the original cross-section, but it does interpolate between cross-sections instead. The original cross-sections must be simplified to a total number of 7 vertices. There is an inbuilt routine that allows to simplify given cross-section data.

The input is given in free format using predefined keywords (see http://www.fluvial.ch/m/syntax.html for more details). Keywords starting with double arrow allow to structure the input. The default name of the input file is 'inx'. The output is written to the file 'out'.

In a command shell the program can be started by typing

```
program_path/flush inputfile
```

or simply by typing

```
program_path/flush
```

where `program_path` denotes the directory where the executable is located.

2 Flow Reference

The following keywords are used to specify the flow data.

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 '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)
`roughness_factor`	-	factor to estimate roughness diameter from mean grain-size
`>>sediment`		*parameters for mobile bed calculation*
`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: • **`mpm`** = Meyer-Peter/Müller formula (S < 0.5%). • **`smart&jaeggi`** = Smart/Jäggi formula (0.5%<S<20%) • `rickenmann90` = Rickenmann formula (0.2% <S< 20%) • `parker90` = Parker formula (published in 1990)
`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)

Input	Unit	Description
`tolerance r`	m^2	area tolerance for update of cross-section geometry (default=0.1m^2)
`stratum r`	m	thickness of the stratum layers (default=0.1m).

`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. 'name' must be a unique string.

`>>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).
`morpho_frequency i`	-	number of (hydraulic) time steps for performing one sediment calculation (default= 40)
`plot_interval`	h	interval to store sediment output on file sed.out (default= 8760)
`>>create_model`		main keyword for the specification of model-specific parameters
`name 'name'`	string	name of model (displayed on model output)
`type '1D'`		type of model: • '1D' for one-dimensional flow calculations (river branch) • '2D' for two-dimensional flow calculations
`>>section`		main keyword for the definition of cross-sections.

`cs 'csname' r1 r2`
`r11 r12`
`r21 r22`
`r31 r32`
`. . .`
	km, m	record for the cross-section definition. 'csname' is the name of the cross-section (max. 8 characters), r1 is the relative distance along the river [km], and r2 [m] is a vertical shift of the profile (default = 0). The offset (i.e. distance from the left border) is stored in the first column and the bed level is stored in the 2nd column.
		Important: The number of cross-section nodes must be 7!

`>>branch`		main keyword for the definition of a river branch.
`cs_distance r`	m	distance between cross-sections (default=10m)
`cs 'csname' r`	km	csname is the name of the cross-section as defined on the file with the cross-section data. r is the location of the section in the branch (the default value is taken from the section list).

Input	***Unit***	***Description***
`roughness r1 r2` `'xx' 'xx' 'xx'`	km	roughness between location r1[km] and r2[km]. The roughness labels in the following line are related to the left wall, the bed and the right wall, respectively. Note: For bed load calculations on alluvial beds the friction parameter of the bed section is estimated by the Strickler formula $k_{st}=\frac{21}{d_{ms}^{1/6}}$ with d_{ms} = mean diameter [m] of the surface layer (= d90 for the MPM and Smart/Jäggi formula).

`axis`
`x1 y1 km1`
`x2 y2 km2`
`. . .`
		co-ordinates of the channel axis (used for integration with 2D models) with co-ordinates in 1st and 2nd column and distance [km] in 3rd column. For distance set to -9999 the value is calculated from the distance between the co-ordinates (except the first node).

`closed km1 km2`		assumes closed sections (pressurized flow) between km1 and km2 if the water level exceeds the level of the left and right dam.
`>>boundary`		flow conditions up- and downstream of river branch, and lateral inflows (sources)
`inflow r`	m³/s	lateral discharge at location given below (** for unsteady flows).
`location r`	km	location of inflow. Note: If no location is given the inflow is assumed to be at the upstream boundary
`bed_width r`	m	local bed width for estimation of sediment inflow
`bed_slope r`	-	local energy slope used for estimation of sediment inflow
`bed_surface 'name'`		grain mixture of surface layer for estimation of sediment inflow
`bed_discharge r`	m³/s	minimum discharge for calculation to reduce calculation time (default = 0)
`closed`		no inflow at upstream section (default)
`waterlevel r`	m	water level at downstream section (** for unstead flows)
`critical`		critical flow at downstream boundary
`slope r`	-	slope of energy line at downstream boundary
`outflow r`	m³/s	discharge at downstream section (** for unsteady flows).
`connect_to 'name'`	-	connects downstream boundary to a 1d-model. The axes of both the current model and the model to connect to must be defined. Critical flow is assumed if no model is found.
`momentum r`	-	factor for momentum flux from connected branch to main channel ranging from 0 (no flux) to 1.0 (total flux) (default = 0.5).

Input	*Unit*	*Description*

		Example for an unsteady (inflow) boundary condition at the upstream boundary, a critical boundary condition at the downstream boundary and a lateral inflow at km 2.5:
		
		`>>boundary`
		`  inflow **`
		`  0.0 10.`
		`  1.0 40.`
		`  2.5 75.`
		`  inflow **`
		`  0.0 3.0`
		`  1.5 7.0`
		`  location 2.5`
		`  critical`

`>>init`		to define initial conditions

`waterlevel`
`km1  z1  q1`
`km2  z2  q2`
` .   .   .`
		initial condition where the distance [km], the waterlevel [m] and the discharge [m³/s] are given in the first, second and third column of a table

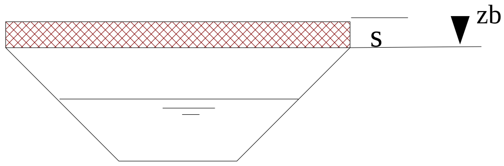
`>>structure`		to define flow over weirs, through gates and hydrodynamic loading on bridges

`weir km zw bw cw`
 	
`or`
 	
`weir km ** bw cw`
`t1  zw1`
`t2  zw2`
`.    .`
		flow over weir at location km with zw = weir crest level [m], bw width of weir [m] and cw = weir (Poleni-) coefficient [-] (default = 0.58 for broad crested weirs). The level of the weir crest can also be specified in a time table.
		
		Example: The weir crest is lowered within 5 hours from a level of 343.0 to 341.2 m s. l.
		
		`weir 0.650 ** 12. 0.62 /`
		`0.0  343.0`
		`2.5  341.5`
		`5.0  341.2`

`gate km zb bg zg cg`
 	
`or`
 	
`gate km zb bg ** cg`
`t1  zg1`
`t2  zg2`
`.    .`
		flow through sluice gate at location km with zb = bottom of gate [m], bg = width of gate [m] , zg = level of gate [m] and cg = contraction coefficient [-] (default = 0.64)
		
		Example: At km 0.650 a gate is located that is combined with a weir. The input looks like this:
		
		`>>structure`
		`gate 0.650 343.0 12. 345.4 / > 'gate1.out'`
		`weir 0.650 346.5 12. 0.62 /  > 'weir1.out'`

`throttle km qmax`		structure at location km that limits the discharge to the value qmax.

`bridge km zb s cD`	bridge section at location km with zb = level of bridge deck, s = width of bridge deck, and cD = drag coefficient of bridge (default = 2.0)



`pile km d cD`	drag force on pile (pier) with diameter d [m] at location km. Drag coefficient cD [-] depends on pile shape (default=1.0 for circular shape).
`duct km1 km2 d kst ld ce`	flow through (serial) duct connected at km1 (entrance) and km2 (exit) with d = diameter of pipe [m], kst= manning-strickler value [m1/3 s-1], ld = length of duct [m], ce = coefficient for energy losses at in- and outlet (default = 1.0)
`breach km 'side' hb tb`	Lateral breach at distance km at 'left' or 'right' side with hb = depth of breach [m] and tb=time of breaching [h]. The width of the breach is equal the distance between the cross-section (see `cs_distance` in section >>`branch`). Example for a breach with a depth of 3.5m at distance km 3.540 on the left side with immediate breaching at time 0.5h: `breach 3.540 'left'  3.5   0.5`

`sideweir cw`
`km1 z1`
`km2 z2`
`km3 z3`
	flow over side weir with cw = weir (Poleni-) coefficient [-] (default = 0.58 for broad crested weir). The dimension of the weir is specified in a table with position (kilometres) in the first column and height of the weir crest in the second column. The width of the weir is calculated from the position of the weir (=multiple of the cross-section distance).
	Hint: The following sequence stores the flow over the weir as a time table on file 'name':
	
	`sideweir 0.62  > 'name'`
	`23.300 405.30`
	`23.350 405.10`

`control_weir x b zdown zup u dt`

free parameters for level control (weir):

- x = location of weir [km]
- b = width of weir [m]
- zdown= minimum level of weir crest [m]
- zup = maximum level of weir crest [m]
- u = velocity [m/s] of weir level adjustment
- dt = time lag [Minutes] between adjustment of weir level
- fac = factor to increase weir movement (default = 1.0).

`control_pump q h`

additional pump for level control (weir defined above):

- q = max. discharge [m3/s]
- h = max. pump level [m]

`control_gauge x y z`

free parameters of control (weir defined above):

- x = location of control gauge [km]
- y = location of discharge measurement [km]
- z = control level [m] (** if specified in a table)

The control level can also be specified in a 'discharge-level' table.

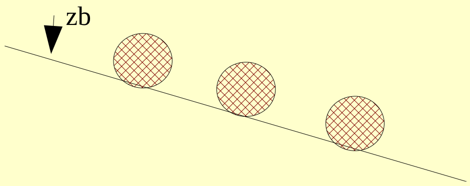
Example:

```
control_weir   2.65   24.   432.0 444.5 0.05 3.
control_gauge   2.75 2.80 442.5
```

```
blocks d  n
km1 zb1
km2 zb2
km3 zb3
```

placed blocks with d = diameter of blocks [m], and n = number of blocks placed per square metre of bed [m^{-2}]. The position of the blocks is defined by a list with distance in the 1st column and level of the blocks (foot) in the 2nd column.

Note: Level of blocks can not be higher than current bedlevel.



Flow resistance is estimated according to Whittaker et al. (1988), formulae (12)-(15).

>>output

for the specification of the output format. Valid item names are:

'q' = discharge
'h' = flow depth
'z' = water level
'zb' = mean bed level
'v' = flow velocity
'qb'= sediment load

`hydrograph r 'item' > 'n'`

writes hydrograph table at location r [km] of item to file 'n'.

Example: **hydrograph 4.5 'q' > 'q_hydro.out'**

`profile r 'item' > 'n'`

writes longitudinal section at time r [h] of item to file 'n'. If r < 0 the output is stored every |r| hour.

Example: profile **90.0 'z' > 'z_prof.out'**

`smooth` -

reduces wiggles in water levels and bed levels (applies to display output and data export)

`to 'name'`

stores results on binary file that can be viewed by FLUSH (load with command: File/Open). Preferred file suffix: .res

`interval r` h

interval between time steps to be stored on result file.

`sediment_to 'name'`

stores results of sediment transport calculation to file 'name' (default name = sed.out).

>>sediment

the following data is related to bed load calculations

`substrate 'name'`

initial mixture of bulk material

`surface 'name'`

initial mixture of active layer material

`inflow r`	kg/s	sediment inflow (r=** for unsteady inflows).
`mixture 'name'`		name of mixture of inflow material.
`location r`	km	location of (lateral) sediment inflow. If no location is given the inflow is assumed to be at the upstream boundary.
`outflow_dzb r`	m/h	velocity of bedlevel changes at the outflow boundary (default=0). Example: **`outflow_dzb 0.01 > 'qb.out'`** will store a hydrograph of sediment transport rate and total load observed at the outflow boundary in file 'qb.out'
`rock_depth r (**)`	m	defines a rock level below the initial bed level. For variable values replace `r` by **`**`** and add a table with the distance [km] and the rock_depth [m] in the 1st and 2nd column.
`rock_level r (**)`	m.s.l.	defines a rock level below the initial bed level. For variable values replace `r` by **`**`** and add a table with the distance [km] and the rock_level [m s.l.] in the 1st and 2nd column.
`deposition r (**)`	m^2	defines an initial deposition above the given bed level (unit m^3 per m).
`widening r (**)`	m	defines widening of original cross-sections.
`dmo r (**)`	cm	Diameter of mean grain-size (to be used for uniform grain calculations).
`reduce r (**)`		factor to reduce bedlevel changes to account for additional sediment storage (default = 1.0)

3 History

Version 3.0 (2017-)
- unlimited model size (allocatable arrays)
- parallel processing (multi-thread CPU's)
- control-gauge structure including pump
- rock_level input
- two-fraction sediment model (sand - gravel)

Version 2.3 (2014-2016)
- deposition unit in m^2 (instead of m)
- improved connection with 2D-models

Version 2.2 (2011-2013)
- momentum flux from lateral branch to main channel
- factor to account for sediment balance in sections with reduced width

Version 2.1 (2009-2011)
- modelling placed blocks with approach of Whittaker et al. (1988)
- modelling debris flow with two-parameter approach (turbulent & yield)

Version 2.0 (2006-2009)
- keyword '>>create_model' for definition of multiple models
- combination with 2D-models (program FLUMEN)
- revised treatment of bridge sections
- new slope boundary condition
- connect_to boundary condition
- changes in input format (e.g. >>global values)
- formula of Parker (1990) for sediment mixtures
- outflow boundary condition
- side weir option

4 Reference

Whittaker J. G., Hickman W. E., Croad R. N., 1988. Riverbed Stabilisation with Placed Blocks. Central Laboratories Report 3-88/3. Lower Hutt, New Zealand.