

Reference

Content

1 Introduction.....	2
2 Flow Reference.....	3
3 History.....	8

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)
- Wilcock&Crowe 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](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 'mmdh'	string	real time of start of simulation (mm=month, dd=day, hh=hour)
>>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 (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)
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).

Input	Unit	Description
<pre>mixture 'name' 0.2 0. 25. 1.</pre>	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. 'name' must be a unique string.
>>compute		to define parameters for unsteady flow computation
<pre>start r</pre>	h	start time of the simulation (default=0)
<pre>end r</pre>	h	time where simulation will end (default=100h)
<pre>cfl r</pre>	-	limiting CFL number to estimate size of time step (default=0.6)
<pre>frequency i</pre>	-	refresh rate of display output (default=100).
<pre>morpho_frequency i</pre>	-	number of (hydraulic) time steps for performing one sediment calculation (default= 40)
>>create_model		main keyword for the specification of model-specific parameters
<pre>name 'name'</pre>	string	name of model (displayed on model output)
<pre>type '1D'</pre>		type of model: <ul style="list-style-type: none"> • '1D' for one-dimensional flow calculations (river branch) • '2D' for two-dimensional flow calculations
>>section		main keyword for the definition of cross-sections.
<pre>cs 'csname' r1 r2 r11 r12 r21 r22 r31 r32 . . .</pre>	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. <u>Important:</u> The number of cross-section nodes <u>must</u> be 7!
>>branch		main keyword for the definition of a river branch.
<pre>cs_distance r</pre>	m	distance between cross-sections (default=10m)
<pre>cs 'csname' r</pre>	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).
<pre>roughness r1 r2 'xx' 'xx' 'xx'</pre>	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. <u>Note:</u> 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 (= d_{90} for the MPM and Smart/Jäggi formula).

Input	Unit	Description
axis x1 y1 km1 x2 y2 km2 . . .		co-ordinates of the channel axis (used for integration with 2D models) with co-ordinates in 1 st and 2 nd column and distance [km] in 3 rd 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	m ³ /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	m ³ /s	limiting discharge where calculation is performed (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	m ³ /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.
		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:
		<pre>>>boundary inflow ** 0.0 10. 1.0 40. 2.5 75. inflow ** 0.0 3.0 1.5 7.0 location 2.5 critical</pre>
>>init		to define initial conditions
waterlevel km1 z1 q1 km2 z2 q2 . . .		initial condition where the distance [km], the waterlevel [m] and the discharge [m ³ /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 r1 r2 r3

flow over weir at location km with r1 = weir crest level [m], r2 width of weir [m] and r3 = weir (Poleni-) coefficient [-] (default = 0.58 for broad crested weirs)

gate km r1 r2 r3 r4

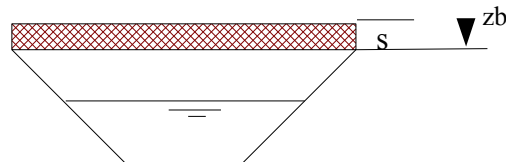
flow through sluice gate at location km with r1 = bottom of gate [m], r2 = width of gate [m], r3 = level of gate [m] and r4 = contraction coefficient [-] (default = 0.64)

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'
```

bridge km zb s cD

bridge section at distance 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 [m^{1/3} 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
```

>>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)

>>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.

deposition r m defines an initial deposition above the given bed level. For variable values replace r by ** and add a table with the distance [km] and the deposition [m] in the 1st and 2nd column.

3 History

Version 2.0 (2006-)

- 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)
- new fractional sediment formula (Parker 1990)
- outflow boundary condition
- side weir option