Reference

Content

1 Introduction

This document describes the use of the flood model **TriPaD**. To perform a calculation you must have installed the following programs:

- The preprocessor Fluviz to format and edit the DTM data
- The Delauney triangulator Triangle to do the meshing
- The program TriPaD that does the flood modelling

You should also have a copy of the following documents:

1. The online manual for Fluviz
2. Shewchuk J.R., Triangle: A Two-Dimensional Quality Mesh Generator and Delaunay Triangulator
3. Beffa C., Two-Dimensional Modelling of Flood Hazards in Urban Areas. ICHE'98, D-Cottbus
4. Hager W. H., Del Giudice G., 1998. Generalized Culvert Design Diagram. J. Irrig. and Drainage Engrgr. Vol. 124, No. 5. ASCE. (see http://www.fluvial.ch/m/culvert.html)

TriPaD is a software that simulates the spreading of surface flows on triangulated irregular networks (TIN's). It allows for wet and dry domains, sub- and supercritical flow conditions, and the specification of variable bed topography.

The main features of TriPaD are:
- bed friction with Manning-Strickler formula
- open boundaries (without backwater effects)
- closed boundaries
- obstructions of bridges
- culvert flow

Inflows are considered as point sources defined at a location with a constant peak discharge (m3/s) and a duration of the event. Different flood sources can be overlayed. Further information on the equations and the calculation method can be found in [3].

2 How to use TriPaD

2.1 Preparing the TIN

TriPaD runs on TIN data. The first task is to format the terrain data in such a way that it can be read by TriPaD. For this reason the following steps have to be made:

1. Prepare the raw data files in FLUVIZ's unstructured data format
   - if possible separate points and breaklines in different files
   - define the maximum distance between the points of breaklines (if necessary)
   - define breaklines as boundaries (if necessary)
2. Start FLUVIZ
3. File/Load and File/Add the terrain data
   - read breaklines first and the terrain points afterwards as points that occur twice are omitted
   - use the Edit/Keep option to produce an overlay of the breaklines that makes orientation easier
4. Edit the data
   - cut points and breaklines (if necessary)
   - add new points and edges (if necessary)
   - define points as boundaries (if necessary)
5. Save the data in Node/Poly format for Triangle with the Export option.

6. Exit FLUVIZ
7. Do the triangulation of the DTM data (e.g. with the program Triangle or some other Delaunay triangulator). Using Triangle you type in the following options

   ```
   triangle -cpe test
   ```

   where 'triangle' is the name of the executable (be sure that the path is correct) and 'test' is the prefix of the .node and .poly files. If you already have defined the boundary points you choose

   ```
   triangle -pe test
   ```

   (see also [2] for more details of the different options). If Triangle is successful it produces several files that contain the information needed for TriPaD. These files all start with the prefix 'test.1' if the original files had the prefix 'test'. The results can be displayed with the program Showme that comes with the program Triangle. In FLUVIZ the nodes can be loaded and the TIN is displayed with the Map option.

2.2 Simulation

To perform a flood simulation start the program with

```
pad filename
```

where 'pad' is the name (incl. directory path) of the TriPaD executable and 'filename' is the name of the file with the data of the hydraulic model (discharge, boundary conditions etc.). For details of the file format see the Reference below). The input file must contain at least the prefix of the files where the DTM data is stored (ie. the .node, .edge, and .ele files) and the scenario definition (inflows and event branches).

TriPaD expects additional user input. The simulation is started clicking on the Compute button. The calculation mesh will be displayed. Clicking on the eXit button the run is started and the results will be displayed. For more efficient use TriPaD can also be driven in batch mode where all the scenarios defined on the input file will be simulated without additional user input.

2.3 Results

The results (depths, flow velocities etc.) of all scenarios are stored on a file named 'hazard.dat'. Clicking on the Analyse button in the main menu these different manipulations on these data can be performed e.g.

- hazard analysis showing hazard levels according to the federal office of water and geology
- intensity levels for predefined return periods
- etc.

For the data analysis load the input file again (not in batchmode!). Choose the Analyse/New option and choose the hazard.dat file where the scenarios are stored. Try out the different options you have, e.g.

- List: will list all scenarios that are stored on the file
- Alter: allows to alter the event attributes of the scenarios (e.g. event ID ,return period)
- Hazard: aggregates the scenarios and calculates the hazard values that can be displayed with the Show option in the main menu.
- Intensity: Calculates flow depths, velocities etc. for a given return period.

3 Input Reference

The following keywords are used to specify the cross-section data.

Input	*Unit*	*Describtion*
`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
`>>init`		
`mesh 'name'`	string	prefix of the files with the DTM data produced in Triangle (.node, .edge, .ele files)
`>>global`		main keyword for the definition of friction values
`kst r`	$m^{1/3}/s$	default for the Strickler value (default = 35 m1/3 s-1)
`pads i`		number of PaD elements (default=400)
`increment r`	m	adjustment of the flow depth (default=0.01 m)
`hdry r`	m	minimum height for discharge calculation (def=0.05 m)
`batch_mode`		runs TriPaD in batchmode
`search_radius`	m	search radius for internal sources and culvert in-/outflows (default=10 m)
`>>boundary`		main keyword for the definition of the model boundaries using polygons. If no boundary condition is defined TriPaD assumes closed boundaries.

`critical`
`x1 y1`
`x2 y2`
`. .`
		to specify an open boundary without backwater effects
		Example: To define an open boundary within a certain area the input could look like this:
		
		`critical`
		`0. 0.`
		`100. 0.`
		`100. 50.`
		`0. 50`

`waterlevel r`
`x1 y1`
`x2 y2`
`. .`
	m.s.l.	to specify an open boundary with known water level

`slope r`
`x1 y1`
`x2 y2`
`. .`
	-	to specify an open boundary with known energy slope

`>>polygon`		

Input	*Unit*	*Describtion*

`add r`
`x1 y1`
`x2 y2`
`. .`
	m	adds the value r[m] to the current bed levels
		Example: To lift the bedlevels in a certain area (eg. to simulate a dam) a polygon file could look like this:
		
		`add 3.0`
		`  0.  0.`
		` 50. 50.`
		` 50. 55.`
		`  0. 55.`

`bedlevel r`
`x1 y1`
`x2 y2`
`. .`
	m	defines a new height of the bedlevel

`waterlevel r`
`x1 y1`
`x2 y2`
`. .`
	m	defines an initial water level in the model (e.g. a lake)

`conveyance r`
`x1 y1`
`x2 y2`
`. .`
	-	conveyance factor to define local roughness values. The actual friction value is the product of the global strickler value times the conveyance factor (default = 1.0)

`>>process`		definition of inflows, culverts

`inflow 'name' T`
`x1 y1 q1 tb1 tp1`
`x2 y2 q2 tb2 tp2`
`. . . .  .`
	year	defines discharges with a given return period T (years). Multiple point sources can be specified at different location (x_ y_) with a peak discharge q_ [m³/s], a base time tb_ [h] and a peak time tp_ [h].
		
		Example for a 100 year flow with two input sources that have base and peak time of 4 and 0.5 hours, respectively:
		
		`inflow 'dorfbach100' 100`
		`687996 221647  12.  4.0 0.5`
		`687832 221832  4.0  4.0 0.5`
		
		Note: A warning is given if no mesh node is found within search_radius given in the global settings.

Input	*Unit*	*Describtion*

`culvert 'name'r1 r2 r3 r4`
`x1 y1 x2 y2`
	SI	culvert flow defined by its geometry and friction value, where r1=width [m], r2=opening [m] for rectangular culvert, or r1=diameter [m], r2=-1 for circular culvert, and r3= strickler value (reverse of mannings n), r4=discharge in case of obstruction [m^3/s] (default=0).
		The co-ordinates of in- and outlet are given in the following line.
		
		Example: Circular culvert with a diameter of 1.5 m and strickler value of 50 m1/3/s (mannings n = 0.02) and discharge of 15 m^3/s in case of obstruction:
		
		`culvert 'culvert5' 1.5  -1.  50. 15. /`
		`687855 221703 687847 221741`
		
		Note: Culvert discharge is estimated according to Hager&Del Giudice 1998.

`bridge 'name' r1 r2 r3`
`x1 y1`
`x2 y2`
`.  .`
	m,
	m,
	m^3/s
		defines a flow under a bridge as a polygon where r1 = lower level of bridge plate, r2= thickness of bridge plate and r3 = discharge when bridge section is obstructed.
		
		Example for a bridge with a capacity of 20 m^3/s in case of obstruction:
		
		`bridge 'bridge3' 745.45  1.2  20.0`
		`687847 221703`
		`687855 221704`
		`687855 221714`
		`687847 221714`

`breach 'name' r`
`x1 y1`
`x2 y2`
`.  .`
	m	defines a breach in a dam as a polygon where r = depth of the breach
		
		Example for a breach with a depth of 2 m:
		
		`breach 'breach5' 2.0`
		`687847 221703`
		`687862 221705`
		`687864 221710`
		`687848 221703`

`>>event`		
`branch i 'name' /`		defines a branch of an event tree where i is the number of the branch and 'name' denotes a process (obstruction of a culvert or bridge, breaching of a dam).The number of processes can vary from 0 to 5. If no processes are defined the culverts and bridges are assumed to be free, and breaching of dams does not occur. Example for a branch with two processes: `branch 3 'culvert1' 'bridge3' /`

Input	***Unit***	***Describtion***
`scenario 'name' i r`	-	defines a scenario that is a combination of an inflow 'name' and branch i. The last digit r defines the relative probability that this event occurs (number between 0 and 1.0). Example for a 100 year flood in the 'dorfbach' with an obstruction of culverts given above that occurs with a relative probability of 30%: `scenario 'dorfbach100'  3  0.30 /`
>>		ends the input reading.