

# Reference

## Content

1 Overview.....	2
2 Applying Hydro2de.....	2
2.1 Getting started.....	2
2.2 Model input.....	2
2.3 Hints.....	3
3 Input Reference.....	4
4 Literature.....	8

## 1 Overview

*Hydro2de* is a program that solves the depth averaged shallow water equations on a cell centred rectangular grid. It allows wet and dry domains, sub- and supercritical flow conditions and the specification of variable bed topography.

The main features of *Hydro2de* are:

- explicit time integration (1st or 2nd order)
- dissipation according to Roe (1st or 2nd order)
- bed friction with Manning-Strickler formula
- vegetation losses with Lindner formula
- zero-equation turbulence models
- mobile bed module (one grain size)
- grid independent input (allows simulation on different grids with same input data)
- culvert flow (in- and outlet controlled)

## 2 Applying Hydro2de

### 2.1 Getting started

In a shell type the following command

```
2de_Home/2de filename
```

where *2de\_Home* denotes the home directory of the program and *filename* denotes the input filename (default *dain*)

Before starting *Hydro2de* you must have specified all the information in a text file (default name *dain*). *Hydro2de* reads from this file and writes to a file called 'out'. If your input file does not have the correct syntax, *Hydro2de* will terminate and write an error message on 'out'.

### 2.2 Model input

The main information you have to prepare for *Hydro2de* is the topography of the bed and the boundary conditions. To define the calculation grid from given bed levels you can use the program [Fluviz](#). It has a function to generate rectangular grids out of unstructured data points, e.g. cross-sections or xyz- coordinates (see the [online manual](#) for details).

If the grid has been created you have to specify the (external) boundary conditions. The shallow water equations form a hyperbolic system. Therefore the number of boundary conditions depends on the wave speed. Suggestions on how to define external boundary conditions are given in the next section.

To perform a simulation run *Hydro2de* needs information about the initial conditions of the flow variables. *Hydro2de* can read from previous results (hot start) or start with dry beds (dry start). If flow depths or waterlevels are known they can be specified as closed polygons.

*Hydro2de* writes the results (water levels, flow velocities etc.) to a direct access file called 'dares'. To read from this file the program POP is needed. It has functions to plot and print to your screen and has drivers for PostScript. Summarizing, while using *Hydro2de* you will have to consider the following files

dain	model definition (input)
out	contains the input reading and error messages (ASCII)
dares	results of the simulation (direct access)
FLUVIZ	preprocessor for creating the grid (alternatively ArcInfo can be used)
POP	postprocessor for viewing the result file

## 2.3 Hints

### Create a grid

*Hydro2de* can read grid formats from Fluviz and from ArcInfo. Try different grids to control the numerical errors: The final solution should be independent of the mesh size.

### Boundary conditions

Inflow boundaries are more sensitive to handle than outflow boundaries. Use `inflow` at the inflow boundaries. Boundary types suitable for outflow are *weir, slope, waterlevel and tides*. - If an inflow is not perpendicular to the boundary you have to define the angle. - To check the amount flowing in and out of the calculation domain, see the output file 'out'. At the end of this file you find a list of the flows summed over each boundary.

### Handling instabilities

If the calculation does not converge following may help:

- check if boundary-conditions are set correctly.
- reduce the maximum CFL-number to e.g. `cfl=0.50`.
- try 1st order scheme (default) instead of 2nd order scheme.

### 3 Input Reference

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

<b>Input</b>	<b>Unit</b>	<b>Description</b>
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
<b>&gt;&gt;grid</b>		is used for the definition of the calculation domain
nx i	-	number of grids in x-direction
ny i	-	number of grids in y-direction
xllcorner r	m	the x-coordinate of the lower-left corner
yllcorner r	m	the y-coordinate of the lower-left corner
dx r	m	size of cell in x-direction
dy r	m	size of cell in y-direction
row1 ... row2 ...	m	bedlevels starting from the lower left corner (upper left corner for ArcInfo format input)
		Alternatively the grid can be given in ARC/Info format using the following keywords:  ncols instead of nx nrows instead of ny cellsize instead of dx and dy (only squares allowed).
<b>&gt;&gt;global</b>		main keyword for the definition of global values
ks r	m	default value for the equivalent sand roughness diameter
n r	SI	default value for Manning's n
kst r	SI	default value for Stricklers's k
z0 r	m	default value for boundary roughness for full log law (Smart et al. 2002)
vegetation r	1/m	default value for vegetation factor
<b>&gt;&gt;initiate</b>		reads the initial conditions of the flow variables from a previous run
from 'name'	string	reads the initial conditions from a previous run stored on the file 'name'
at r	h	denotes the time of the previous run that is used as the initial conditions (see keyword 'from' above)
		For the definition of flow depths and water levels see the >>polygon section below
<b>&gt;&gt;polygon</b>		definition of spatially variable data using closed polygons. The following items can be specified
item 'name'		
x1 y1		
x2 y2		
:		
:		

<b>Input</b>	<b>Unit</b>	<b>Description</b>
bedlevel r	m	bed level
add r	m	adds a value to the given bed levels
waterlevel r	m	water level at start time
flowdepth r	m	flow depth at start time
ks r	m	roughness diameter
n r	SI	Manning's n
kst r	SI	Strickler value
z0 r	m	boundary roughness for full log law (Smart et al. 2002)
vegetation r	1/m	vegetation factor given by the formula $\text{veg} = d / a^{**2} * cw$ with d=diameter of vegetation elements [m], a=distance between elements [m] and cw=drag coefficient (range 0.8 - 1.5).
leakage r	s-1	leakage factor for infiltration into the ground. The infiltration flow rate is estimated by the formula $\text{flow} = \text{leakage} * \text{flowdepth}$
bridge r	m	to define a bridge where r is the lower level of the bridge plate, and the geometry (area) of the bridge is given by a polygon. The program accounts for the backwater effects that occur if the water level exceeds the given value r
erodable r1 r2 r3		to define erodable areas (e.g. dams). r1[N/m] is the critical bed shear (default=50), and r2[m] is a length scale that represents the distance where erosion may occur (e.g. the width of the dam; default=10). r3[m] is a minimum flow depth that has to be exceeded before erosion starts (default=0). A list of polygon points describes the erodable area. Erosion is estimated with the formula of Meyer-Peter/Mueller.
>>boundary		main keyword for the definition of the model boundaries using polygons. If no boundary condition is defined Hydro2de assumes closed boundaries.
inflow r	m <sup>3</sup> /s	discharge value (positive for an inflow and negative for an outflow boundary)
slope r	-	energy slope used to define an outflow boundary. The outflow is calculated assuming uniform flow conditions. Note: For high slopes (>0.01) the flow regime becomes supercritical and the boundary condition is ignored.
weir r	m	level of a weir crest. The flow over the weir is estimated as the critical flow (no backwater effects). The width of the weir is equal to the grid size.
waterlevel r	m	water level above datum (only outflow)
tide r	m	water level above datum. This boundary is suited to model tidal boundaries with in- and outflows to and from the sea.  Additional keywords can be used in combination with a boundary definition:
angle r	-	tangent of the flow angle (measured in counter-clockwise direction). It can be used in combination with the inflow boundary (default is normal to the boundary r=0)

<b>Input</b>	<b>Unit</b>	<b>Description</b>
width r	m	width of the weir if it is smaller than the grid size (default is the grid size)
uniform_slope r	-	bed slope that is used in combination with the inflow boundary. The flow distribution is estimated using the local bed levels, bed friction and the given slope r (default=0.001).
cross_slope r [-]	-	lateral slope of the waterlevel. It is used in combination with the inflow boundary to account for lateral slopes of the water level (default=0)
<p>If the boundary value varies in time a list of two columns has to be given defining the time (first column) and the boundary value (second column). The program interpolates linearly between the given values. The location where the boundary value holds is specified by a polygon that contains the boundary nodes. This is best illustrated with some examples.</p>		
weir 10. > 'w.out' location 95. 35. 105. 35. 105. 58. 95. 58.		<p>Example (i) Outflow over a weir with crest at 10 m.</p> <p>Note: 'w.out' denotes an ASCII table where the discharge through the boundary is stored for further usage.</p>
inflow ** 0.10 -100. 0.11 -40. 0.13 0. location 50. 135. 60. 135. 60. 140. 50. 140.		<p>Example (ii) A turbine is located at the north side of the boundary box. The discharge is 100 m<sup>3</sup>/s for the first 0.1 hour and then drops to 40 m<sup>3</sup>/s within 0.01 hours. 0.02 hours later the turbine is completely closed. (Hint: Because the flow leaves the calculation domain it has a negative value).</p> <p>Note: ** indicates that a list of two columns follows</p>
inflow 75. uniform_slope .005 angle -1. location -5. 35. 5. 35. 5. 135. -5. 135.		<p>Example (iii) A river channel enters at the west edge of the computation box at an angle of 45 degrees to the south. The flow is approximately uniform with a slope of 0.5% and a discharge of 75 m<sup>3</sup>/s.</p>
>>turbulence		specifies the turbulence model. Three different models are available that can be specified at the same time.
viscosity r	m <sup>2</sup> /s	molecular viscosity of the fluid (default = 1.01e-6 m <sup>2</sup> /s for water at 20C)
cv r	-	<p>value of the coefficient used to determine the eddy viscosity by the formula</p> $n = \text{visc} + cv * us * h$ <p>with visc=molecular viscosity, cv = the coefficient defined above, us = friction velocity and h = flow depth. A good estimate for river flow is cv = 0.07.</p>

<b>Input</b>	<b>Unit</b>	<b>Description</b>
mixing_length r	m	length scale for Prandtl's mixing length model, where the eddy viscosity is $\nu = l_m^2 * ( du/dy  +  dv/dx )$ with $l_m$ = mixing length, $u$ and $v$ = components of the flow velocity, $x$ and $y$ = distance  Note: If $cv$ and $l_m$ are set, the maximum value of both models is taken as the eddy viscosity factor
<b>&gt;&gt;source</b>		hydraulic sources
discharge r	m <sup>3</sup> /s	point source at the location given below
point r1 r2	m,m	x-y co-ordinates of a point source given above
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 $r < 0$ .
<b>&gt;&gt;structure</b>		definition of internal culverts
culvert x1 y1 x2 y2		circular or rectangular <a href="#">culvert</a> where $(x1,y1)$ and $(x2,y2)$ = co-ordinates of in- and outlet. The module accounts for inlet and outlet controlled flow conditions. It is assumed that the vertical level of the in- and outlet corresponds to the bed level of the adjacent grid cell.  Note: 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)
<b>&gt;&gt;compute</b>		values regarding the computation
start r	h	start time of the simulation
end r	h	time when simulation ends
steps i	-	maximum number of the time steps
cfl r	-	CFL number that is used to determine the length of the time step (default=0.7)
hdry r	m	minimum flow depth. If the actual flow depth is smaller than $r$ the cell is considered to be dry (default=0.02m)
2nd		a higher order scheme is used that is 2nd order in space and time (MUSCL, minmod limiter)
2nd_boundary		2nd order extrapolation for boundary values (default = 1st order)

<b>Input</b>	<b>Unit</b>	<b>Description</b>
>>output		defines the output from the model
to 'name'	string	results are written to file 'name' (default: dares)
interval r	h	interval of the stored results (default: no storage)
plot 'name'	string	for storage of specific items. Possible items are:
'initial'		values for initial conditions (default)
'waterlevel'		waterlevel (already stored with item initial)
'flowdepth'		depth of flow
'velocity'		velocity of flow
'flow'		specific flow
'froude'		froude number
'bedlevel'		level of river bed
'bedshear'		shear stress at river bed
'ks'		sand roughness diameter
'n'		Mannings n value
'kst'		k-Strickler value
'viscosity'		eddy viscosity
'vegetation'		friction factor to account for vegetation losses
'inundation'		duration [h] of inundation
max_values 'name'	string	stores maximum values (waterlevel, flowdepth, velocity of flow, and specific) on file 'name'
max_depth 'name'	string	stores maximum flow depth on grid file 'name'
max_flow 'name'	string	stores maximum flow rate on grid file 'name'
no_data r	-	set the water level of the dry grids to the value r (default is the bed level)
dt_out r	h	length of time interval for storing hydrographs (default = 0.01h)
>>		ends the input reading

## 4 Literature

Smart G. M., Duncan M. J. , Walsh J. M. (2002). Relatively Rough Flow Resistance Equations. J. Hydr. Engrg. ASCE, Vol. 128, No. 6.