2-dimensional free surface flow modelling

Specialization: Hydraulics

Vincent Guinot: guinot@msem.univ-montp2.fr
Carole Delenne: delenne@msem.univ-montp2.fr
Polytech'Montpellier STE
March 2017

Contents

Chapter 1

2D free surface hydraulics

1.1. Why use 2D models ?

A Two-Dimensional (2D) model performs better than a 1D model when one or several of the following situations occur.

- a strongly contrasted geometry, with large bottom and/or friction coefficient variationsacross a given section,
- sudden stream narrowings/widenings leading to strongly converging/diverging streamlines,
- presence of lateral dead zones creating vorticity patterns,
- compound channels with drying beds that lead the main stream to divide into several loops as the water level decreases, hence changing the topology and connectivity of the stream network with the water depth (a situation that cannot be handled by a 1D model).

In any of the above four cases, using a single, averaged variable over the stream cross-section fails to provide an adequate representation of the actual flow conditions. A description of the flow variations across the channel is also needed. The consequence is that a second momentum is needed in the governing equations.

1.2. Assumptions – governing equations

The assumptions of 2D free surface hydraulics are similar to those of 1D models, except that some of the constraints are relaxed. In 2D hydrostatic models, the following assumptions are made:

- negligible vertical accelerations, hence a hydrostatic pressure distribution over the vertical,
- small bottom slope,
- the non-uniform velocity field over the vertical can be accounted for by a Boussinesq-like coefficient in the equations.

The governing equations are written in conservation form as

$$\frac{\partial \mathbf{U}}{\partial t} + \frac{\partial \mathbf{F}_x}{\partial x} + \frac{\partial \mathbf{F}_y}{\partial y} = \mathbf{S} \tag{1a}$$

$$\mathbf{U} = \begin{bmatrix} h \\ hu \\ hv \end{bmatrix}, \quad \mathbf{F}_x = \begin{bmatrix} hu \\ hu^2 + gh^2/2 \\ huv \end{bmatrix}, \quad \mathbf{F}_y = \begin{bmatrix} hv \\ huv \\ hv^2 + gh^2/2 \end{bmatrix}, \quad \mathbf{S} = \begin{bmatrix} 0 \\ (S_{0,x} - S_{f,x})gh \\ (S_{0,y} - S_{f,y})gh \end{bmatrix} \tag{1b}$$

where g lis the gravitational acceleration, h is the water depth, u and v are respectively the x- and y-flow velocity components, $S_{0,x}$ and $S_{0,y}$ are respectively the x- and y-bottom slopes, $S_{f,x}$ and $S_{f,y}$ lare respectively the x- and y-friction slopes.

The structure of the 2D free surface flow is essentially identical to those of the 1D equations. The 3×3 system (1a-b) is hyperbolic. It accounts for wave propagation processes. The difference with 1D hydraulics is that the 1D Saint Venant equations have 2 waves, each of which propagate along the x-direction only. In 2D free surface flow problems, the waves propagate in the x- and y-directions at the same time. This makes 2D simulation more difficult to interpret than 1D simulations.

1.3. Some possible problems and mistakes

The main two mistakes to be avoided with 2D free surface hydraulics are the following.

- Free surface flow modellers trained on 1D software tend to use the same reasonings for 2D models. In particular they tend to use the same arguments and reflection patterns as they use for steady state, 1D hydraulics (M1/2 curves, uniform flow or depth, etc.).
- They also tend to forget that the three flow variables h, q_x ,q_y are independent from each other in solving the flow equations. This has strong consequences on the meaning and interpretation

of parameters such as the friction coefficient, but also the influence of the geometry and boundary conditions.
- A common mistake consists in believing that 2D simulation results are essentially more reliable than 1D simulation results. This is not the case. 2D simulations are simply more realistic than 1D ones.

2D free surface flow modelling raises a number of issues compared to 1D modelling.
- Collecting and processing the geometry is significantly more expensive and time-consuming for a 2D model than for a 1D one. Meshing the domain for a 2D model often requires several days, sometimes weeks, when a 1D model would be processed within a day or less.
- A 2D simulation requires much more time than a 1D simulation (2D may be 100 to 1000 times slower than 1D).
- Prescribing meaningful boundary conditions is much less straightforward for a 2D model than it is with a 1D model.
- When a 1D model fails to give realistic answers, finding the cause for it is usually rather easy. This is not always the case with 2Dmodels.
- While the friction coefficient is a key parameter in a 1D model (it serves as a useful parameter to “tune” the model and provide correct simulation results in many circumstances), it is much less influential in a 2Dmodel.

To summarize. 2D free surface flow modelling remains an expensive technology. Mastering it requires significantly more time and work than it does for 1D modelling. The purpose of the present working sessions is not to turn you into specialists of 2D modelling. The main objective is to make you aware of the potential of 2D FSF modelling, as well as to make you cautious about it.

Chapter 2

The SW2D modelling suite

SW2D stands for Shallow Water 2 Dimensions (the package solves the 2D shallow water equations, another name for the 2D Saint Venant equations). This modelling suite has been under constant development at Hydrosciences Montpellier since 2002. The package incorporates several engineering and research versions. It is used by 4 consultancy companies (Ginger/Grontmij, Cereg Ingenierie, Citeo, Ameten).
The software suite consists of several executables, to be used in a sequence (Figure1). Each executable name is made of two parts. The first part is the application's name, the second is the version number. To give but an example, ex27d.exe is version 2.7d for executable "ex", the application that extracts the results from the result files.

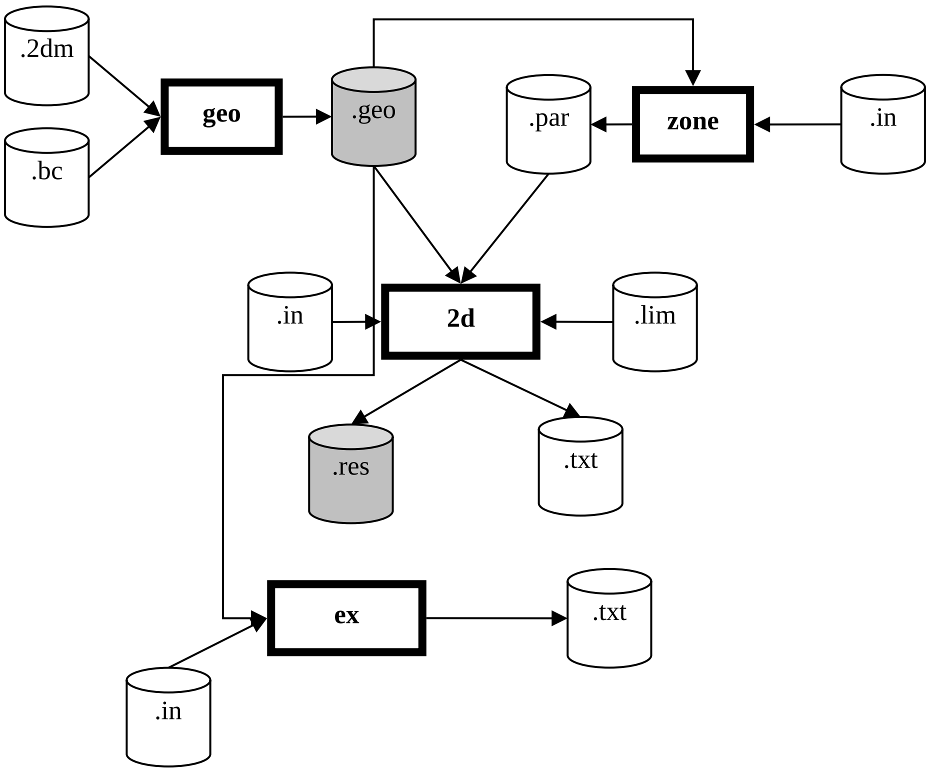

Figure 1. The SW2D modelling suite. Bold rectangles: executables. White cylinders: text (input/output) files. Grey cylinders: binary (input/output) files.

The sequence is the following

1) Use the **geo** executable to analyse and process the geometric data. **geo** reads the mesh file, the boundary conditions specification file, and processes them in order to determine e.g. which cells are neighbours what are the areas of the cells, the lengths of their interfaces, etc. **geo** creates a binary geometry file that is later used by the **2d** executable (see hereafter).
2) If needed, use the **zone** executable to create distributed parameter files. This may be the case if e.g. the Strickler's friction coefficient is spatially variable and is piecewise constant over the domain. **zone** uses the geometry file and requires the prior use of **geo**.
3) The **2d** executable is the simulation engine. It solves the 2D equations presented in Chapter 1 using a finite volume technique. It can simulate static or moving hydraulic jumps, moving bores, transcritical flow, etc. without any mass and/or momentum conservation problem.

Results are produced into two main forms: maps, that are stored at regular (user-specified) intervals into a binary file (the .res file on Figure 1), and time series (stored into text files).

4) The **ex** executable extracts the results from the .res binary file. It creates .txt files that can be displayed using the SMS mesh generation package.

The standard SW2D operation sequence is the following.

1) Create the mesh (.2dm) using the SMS package,
2) create the boundary condition code file (.bc),
3) compile the geometry using **geo**,
4) if needed, create parameter spatial distributions using **zone**,
5) edit the 2d simulation input file to specify the simulation characteristics, run the **2d** executable,
6) extract the simulation results using the **ex** module,
7) visualize simulation results using the SMS tool.

Running the various SW2D modules require that a number of input data files be provided. The structure of these files is covered in the next chapters.

The Appendix describes the structure of the main two files you will have to modify during the 2D modelling sessions: the simulation engine parameter file and the result extraction file.

Chapter 3

Modelling a schematic crossroads

3.1. Introduction

3.1.1. Model – added value of a 2D model

Crossroads are a key element to the propagation of urban floods. A number of experimental studies involving scale models and numerical simulations have indicated that empirical formulae and 1D models fail to provide an accurate description of the flow distribution between the streets at crossroads. Comparing simulation results to experimental data shows that 2D models are much more reliable than 1D ones in the field of urban hydraulics.

The present tutorial will guide you through the steps to setting up and running a two-dimensional model of idealized crossroads. For the sake of simplicity (and keeping in mind the need for small computational times), the model is simplified as follows:

– the road is assumed totally flat,
– the sidewalks are not accounted for,
– the geometry is idealized, with streets of equal width, crossing at exact straight angles.

3.1.2. Objectives

The objectives are the following:

– simulate free surface flow in a medium-sized crossroads (10m wide),
– extract and analyse the simulation results,
– asses the biases induced by the computational mesh. Infer general rules for a proper and accurate domain meshing.

3.2. Simulation specifications

The file names involved in the simulation are given in Table 3.1.

File name	Content	Format
Mod01.2dm (supplied)	Computational mesh (meshing method#1)	Text
Mod01.map (supplied)	Auxiliary file for mesh generator	Text
Mod01.materials (supplied)	Auxiliary file for mesh generator	Text
Mod01.sms (supplied)	SMS project file for Mesh#1	Text
2d_Mod01.in	Simulation file	Text
Carrefour.lim (to create)	Boundary conditions as functions of time	Text
Mod01.bc (to create)	Boundary codes for Mesh#1	Text
Mod01.geo (to generate)	Geometry file	binary
Mod01.res (to generate)	Result file	binary
Ex01.in (supplied)	Extract file	Text

Table 3.1. Crossroads model. Data files for Mod01 (Carrefour.lim will be common to simulations 1 and 2, the other files must be created for Mod02).

In this project session, you will perform steady state simulations. This achieved by simulating transient flow with constant boundary conditions and waiting for steady state as an asymptotic transient state. The present model is rather small and steady state is considered achieved after approximately 100s.

3.3. Setting up the model, launching the simulation

3.3.1. Visualizing the mesh

Double-click on file «Mod01.sms» file. The SMS grid generator starts automatically and loads the project. If this does not happen, start SMS and load "Mod01.sms". Three possible visualization types are available: the so-called "mesh mode" (*Mesh* module) alone, the "map mode" (*Map* module) alone, or both superimposed (Figure 3.1).

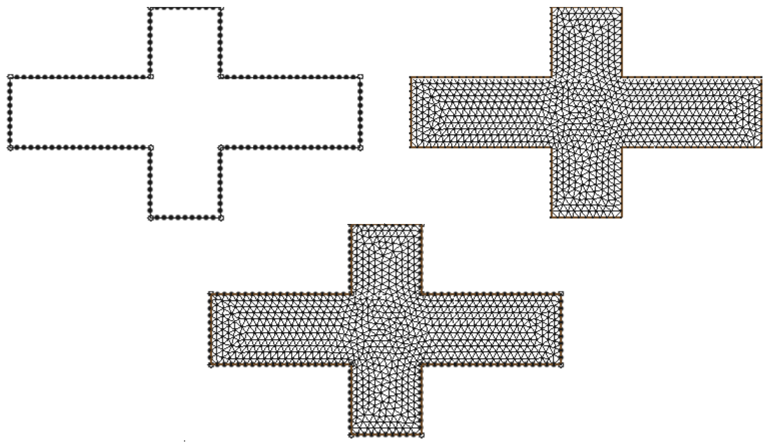

Figure 3.1. Model plan view. Upper left: *Map Data* activated. Upper right: *Mesh Data* activated. Bottom: *Map Data* and *Mesh Data* activated.

The *Map* and *Mesh* modules are activated and de-activated by clicking the corresponding boxes available in the left-hand side interface window. Figure 3.2 illustrates the effect of activating both *Map* and *Mesh*.

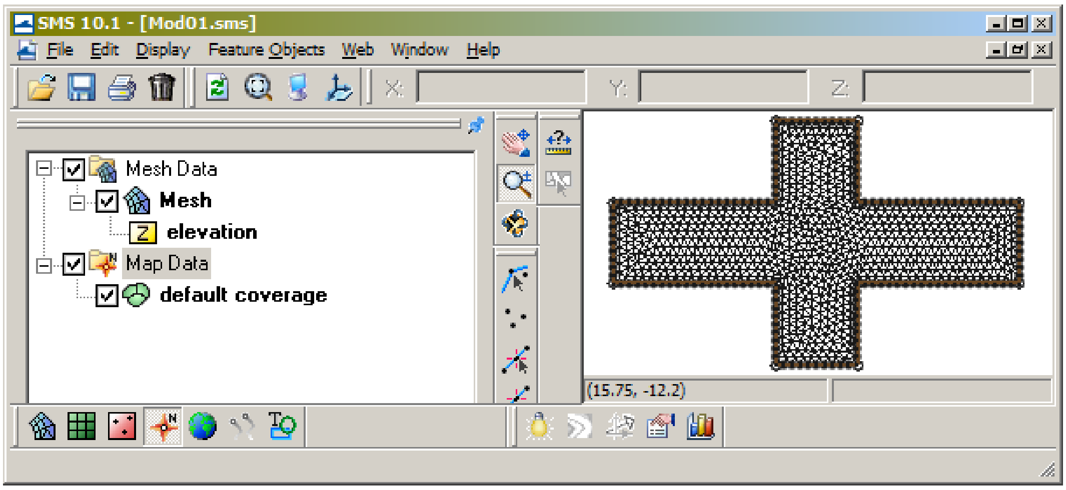

Figure 3.2. Typical model view with *Map Data* and *Mesh Data* activated.

3.3.2. Setting up the boundary condition code file, compiling the geometry

The purpose of the boundary condition file is to specify the type of boundary conditions to be applied in the various parts of the model.
The boundary code file is a text file. It must be written by the user. In the present exercise, the inflowing discharge is prescribed at the Western boundary of the model. It flows out through the Eastern model boundary. The two lateral streets are assumed impermeable due the assumption that the street network is periodic in the *y*-direction.

Three boundary condition types are needed for the present simulation.
- code 1: zero discharge;
- code 2: prescribed discharge as a function of time (Western boundary);
- code 3: prescribed free surface elevation (Eastern boundary).

The boundary code file is a text file. It usually has the extension .bc, but using such an extension is not compulsory.

1) Create a new text document in the working directory. Rename it into «Mod01.bc». Open it using Notepad or Wordpad.

2) The first line in the file indicates the number of segments for which boundary conditions are to be specified in the file. A segment is defined as an interface, that is, a computational cell edge. Any segment connects two adjacent nodes.
   - There are many segments of type 1 (impermeable boundary condition) in the model. Defining them all would be very time-consuming. To minimize the burden, code 1 will be specified as a default value when running the **geo** executable. Segments with code 1 will not be entered in the .bc file.
   - The Western and Eastern boundary are made of 10 interfaces each. Therefore there will be 20 lines in the .bc file.

   Type 20 on the first line of the file and hit the return key.

3) For each boundary interface in the .bc file, three numbers must be provided: the start node, the end node, and the boundary condition code, with a minus sign.
   3.1) Activate the *Mesh Data* module and select the mesh *Mod01* in the project manager window.
   3.2) Lateral toolbar: activate the *Select Mesh Node* tool (an arrow in the middle of three points). Once selected, the tool appears as if lowered from the rest of the toolbar.
   3.3) Click on the Southmost node on the Western boundary. Several pieces of information appear in the window (Figure 3.3).

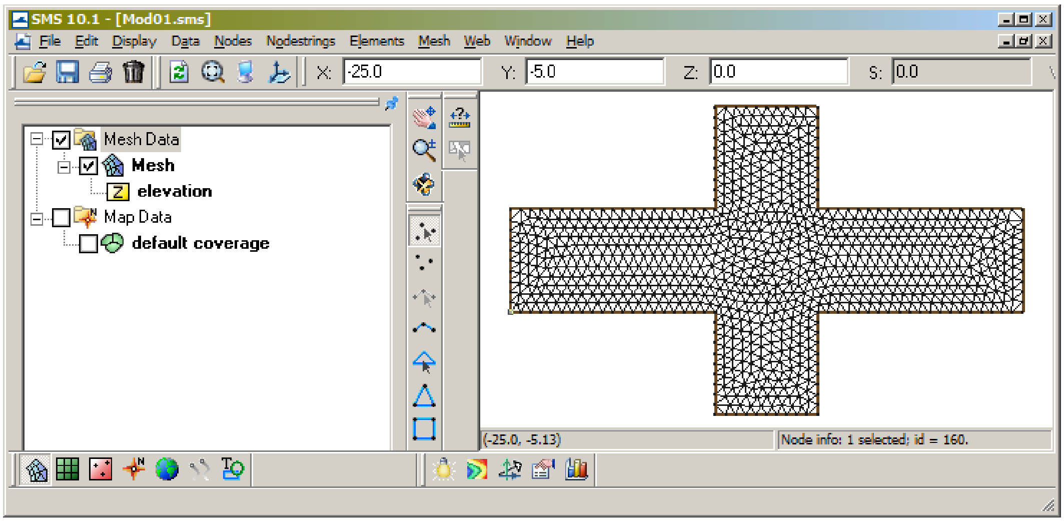

Figure 3.3. Node selection in *Mesh Data* mode.

   - The X, Y and Z fields above the visualization window now indicate the node coordinates.
   - Below the visualization window, an information line window indicates «Node info: 1 selected; id = 160.» id = identifier is the node number. Each node in the mesh is assigned a number that allows it to be identified uniquely.

   3.4) Repeat the operation for the node immediately above. The id for this node is 1.

3.5) The first Western boundary interface thus connects nodes 160 and 1. Since the code for this type of boundary condition is 2, the following line is to be entered in the .bc file:
```
160  1     -2
```
(please note that a boundary code is always negative by definition).

3.6) Repeat the operation for all Western boundary nodes. Since the last interface on this boundary connects nodes 9 and 10, the corresponding line in the .bc file must be
```
9    10    -2
```
3.7) Using the same principle, define the 10 interfaces for the Eastern boundary (code 3, so -3 must be specified in the .bc file)

3.8) Save the .bc file and quit the text editor.

4) Double-click the **geo** executable. A DOS (or command) prompt opens[1]. The user is required to provide several answers.
   4.1) «If SMS mesh file, type 1, otherwise 0»: enter 1
   4.2) «Mesh file?»: Mod01.2dm (see file nomenclature in Table 1).
   4.3) «(x, y) of unused nodes?»: 10 10 (two times 10, with a space in between). This information is needed to move artificially nodes that may happen not to be used in the mesh. Such nodes may induce graphical problems when displaying the results. Locating them outside the domain allows such problems to be eliminated.
   4.4) «Boundary conditions file ?»: Mod01.bc
   N.B.: this is the file you just created in steps 1)-3).
   4.5) «Geometry file?»: Mod01.geo.
   N.B.: this is the binary file containing all the results of geometry processing: what are the ids of all the nodes composing each cell in the mesh, their coordinates, which cells in the mesh are neighbours, etc.
   4.6) «Topography correction: type 1, 0 otherwise»: 0
   The user is given the possibility to modify artificially some of the node elevations. This is not the case here.
   4.7) The **geo** executable scans the mesh file and processes the geometry. A number of pieces of information is displayed on the screen.
   4.8) «Code for unassigned interfaces in .bc file ?»: –1
   N.B.: this is precisely the default value mentioned in Section 3.2 (here –1). After scanning the file again, **geo** indicates that 140 boundary interfaces without a code have been found and that they will be assigned the code -1 (“140 corrections made”)
   4.9) The working folder must now contain the file Mod01.geo. Another file has been created (Topoinit.txt). It contains the cell ids and gravity centre coordinates.

3.3.3. Boundary condition time series file

Boundary conditions are time-dependent. Their variations with time are described using a specific text file. The sequence of operations to create a boundary condition time series file is the following.

1) Create a new text file «Carrefour.lim», open it using the Notepad/Wordpad text editor.
2) Lines 1-2: the first two lines in the file are the file header. You might write any text you like. Note: never use empty lines. Any line in a text file must contain at least on character. Empty lines will be ignored by the program and an execution error will occur.
3) Line 3: indicate the number of boundary condition types. In the present case, there are 3 boundary condition types (1 for impermeable interface, 2 for the upstream boundary, 3 for the downstream boundary).
4) Lines 4-5: comment lines (do not leave them empty, for the same reasons as in step 2).
5) Line 6: enter successively the type of each of the 3 boundary conditions between quotes. BC type 1 is a prescribed (zero) discharge, BC type 2 is a prescribed unit discharge, BC type 3 is a prescribed water level. You must thus type the following text:
```
'q' 'q'   'z'
```
6) Lines 7-8: comment lines.

[1] if not, create a file with the extension “.bat” e.g. launch.bat, in which you will enter only two lines: geo27d.exe and pause. Save and close the file and double click on it to launch the command.

7) Lines 9 and following: time (in seconds), numerical value for each of the 3 boundary conditions. All values are in SI units. For a "q" boundary, the unit is m^2s^{-1}, "z" or "h" boundary conditions are in m, "c" boundary conditions are for prescribed Froude values and are therefore dimensionless.
   Enter the following 3 lines:
   ```
   0.        0.    0.    0.5
   3e2       0.    1.0   0.5
   1e7       0.    1.0   0.5
   ```
   With these 3 time series, BC type 1 will preserve a zero discharge from $t = 0$ to $t = 10^7$s. For BC type 2, the discharge q increases linearly from 0 to 1.0 m^2s^{-1} between $t = 0$ and $t = 3000$ s. The downstream water level (BC type 3) remains constant at $z = 0.5$ m all throughout the simulation.
8) Save the file and quit.

3.3.4. Simulation file

1) Open the file «2d_Mod01.in» using a text editor.
2) Check the following points (see description of input file in Appendix A):
   2.1) the resistance (friction) flag is set to 0,
   2.2) the simulated time Tmax is set to 600s,
   2.3) the results are saved in map format every minute (Dtmap = 60.0),
   2.4) the time series results are saved every second (DtTim = 1.0),
   2.5) the Godunov discretization is used (scheme = 'g'),
   2.6) solver 2 is used,
   2.7) the geometry data file corresponds to the .geo file you just created: Mod01.geo,
   2.8) the initial boundary conditions (z, u, v) are respectively 0.5 m, 0.0 ms^{-1} and 0.0 ms^{-1},
   2.9) the boundary condition time series file is the one you created in the steps above: Carrefour.lim,
   2.10) the storage flag for the hydraulic head is activated (Ht = 1),
   2.11) the time series will be saved at point (0.0, 0.0).

3.3.5. Launching the simulation

1) Open a command line and type the name of the simulation engine (2d26d, 2d27c, etc. depending on the version you have)
   or
   create a .bat file with two lines containing 2d26d.exe and pause and launch this file.
3) You are requested to enter the input file name: this is the file you just checked out in 3.3.4 (2d_Mod01.in).
4) The simulation starts. Simulation progress can be monitored from the simulated time that is being displayed at regular intervals. Depending on the performance of the computer, the simulation may take between 30s and 90s.

3.4. Extracting and displaying the simulation results

3.4.1. Flow variables

Each simulated variable stored in the result file has a specific code. Table 3.2 gives the available variable codes.

Code	Variable	Symbol
1	Water depth	h
2	Free surface elevation (stage)	z
3	Unit discharge (x-, y- and norm)	q_x, q_y, $\|q\|$
6	Flow velocity (x-, y- and norm)	V_x, V_y, $\|V\|$
9	Froude number	Fr
10	CFL number	Cr
11	Water depth in buildings/basements	η
12	Specific head $(h + V^2)/(2g)$	H_s
13	Hydraulic head $(z + V^2)/(2g)$	H or H_t

Table 3.2. Variable codes.

Note that a simulated flow variable will be available for display only under the condition that it is stored in the .res file during the simulation. Open the simulation file and check the *Flag for result storage* item next to the end of the file. Figure 3.4 shows the corresponding lines in the file 2d_Mod01.in.

```
===============================================================
Flag for result storage
h      z      (qx,qy,|q|)   (Vx,Vy,|V|)   Fr     Cr
1      1      1             1             1      0
--
Etha   Hs     Ht
0      0      1
===============================================================
```

Figure 3.4. Result storage specifications.

A storage flag equal to 1 means that the variable will be stored during the simulation. If the flag is 0, the variable will not be stored. In the example of Figure 3.4, all variables are stored, with the exception of the Courant number, the water depth in the buildings/basements and the specific head.

N.B. The variables are stored for all cells in the mesh with the time period *DtMap* specified at the beginning of the file. As shown by Figure 3.5, the flow variables are stored every 60 seconds.

```
--
T0     Tmax   DtMax        DtMap  DtTim
0.     600.   0.027        60.    1.0
--
```

Figure 3.5. File 2d_Mod01.in. View of the time step specification parameters.

3.4.2. Extracting and displaying scalar flow fields

Scalar variables are by definition variables that are not in vector form.

1) Open the result extraction specification file (Figure 3.6). The structure of the file is the following:
   - Lines 1-2: header (must not be empty).
   - Line 4: geometry file name (the file created in Section 3.3.2)
   - Line 5: result file name (the .res file).
   - Line 6: threshold depth. This value is used to make some maps more readable. The user may decide that all cells with a water depth smaller than this threshold value will not be extracted from the result file.

– Lines 7-8: comment lines.
– Line 9: number *N* of maps to be extracted (4 in Figure 3.6).
– *N* lines specifying the data for each map. Each of these lines must contain, in the following order:
  – the extraction time (in seconds),
  – the code of the variable to be mapped (see Table 3.2),
  – name of the output file, between quotes,
  – the chain ‘c’ (including the quotes). This chain specifies that the results will be extracted and mapped at the centre of mass of the computational cells.

The file shown on Figure 3.6 specifies that the free surface elevation *z* and the hydraulic head *H* are to be extracted for *t* = 10 minutes.

```
Result extraction file
Geometry Binary results file name, hmin
Mod01.geo
Mod01.res
0.001
--
Nb of  dates and for each: t, Var Type, Outfile (between quotes)
4
600.   2      'z_Mod01.txt''c'
600.   3      'q_Mod01.txt''c'
600.   5      'v_Mod01.txt''c'
600.   13     'H_Mod01.txt''c'

-->Max
number of max
0
-->Sub
number of sub
0
-->Sur
Number of subsurf
0
-->Tim
Number of tim
0
```

Figure 3.6. Result extraction specification file ex01.in.

Warning. The lines “→ Max” and following are not used in this tutorial. They must be kept unchanged in the file, otherwise leading to execution errors.

2) Open a command prompt and perform the following tasks.
   2.1) type the name of the extraction executable (*ex26d*, *ex27c*, etc. depending on the version you have).
   Or
   Create a .bat file in which you enter the two lines: ex26d.exe and pause. Launch it.
   2.2) Input file name: this is the file ex01.in shown on Figure 3.6.
   2.3) The **ex** application starts. A number of messages appear for the user’s information.

3) Return to the SMS mesh generator.
   3.1) *Display / Display Options:* activate the *2D Mesh* item. Disable *Contours*, *Nodes* and *Elements* in case they are activated.
   3.2) *File / Open* or Ctrl-O. The file selection window appears.
   3.3) Select the file for the variable you want to display (four possible choices in the present tutorial: z_, q_, v_ or H_Mod01.txt), click *Ouvrir*;
   3.4) In the *Open File Format* window, select *Use Import Wizard*, confirm by clicking *OK*.
   3.5) The window *File Import Wizard – Step 1 of 2* open. Click *Suivant*;

3.6) The window *File Import Wizard – Step 2 of 2* open. Click *Terminer*;
3.7) In the project window, *Scatter Data* must now have a new item. It is displayed in bold characters by default. Its name must be the same as that of the file that was just opened (without the .txt extension).
3.8) Tick *Scatter Data*, select H_Mod01.
3.9) Click *Display / Display Options*, select *Scatter* in the list. Activate *Contours*. The resulting view is shown on Figure 3.7. If the display is not the same as on Figure 3.7, see step 4.

N.B. The contour lines spreading outside the model are physically meaningless. They are only interpolation artefacts. They can be removed by applying a specific procedure (step 5).

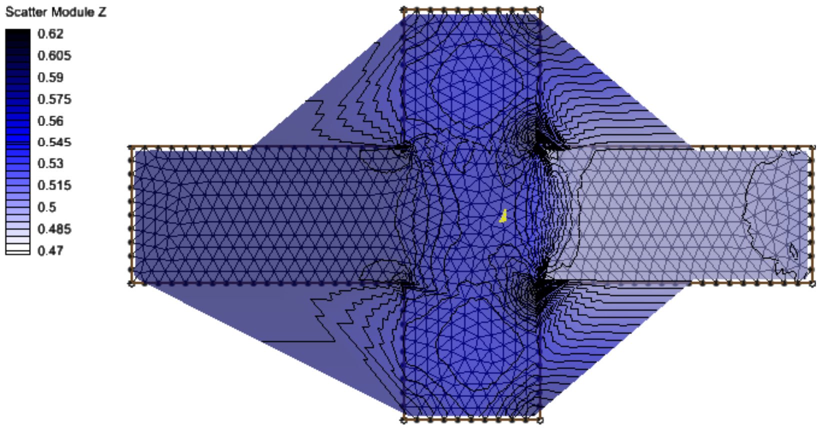

Figure 3.7. Plan view of the free surface elevation at t = 600 s.

4) The colour scale can be modified as follows.
   4.1) *Display Options / Scatter*, select *Contours*.
   4.2) The *Contour Method* field allows for 3 options:
      – Normal linear: only the contour lines are displayed.
      – Color fill: a coloured surface without contour lines.
      – Color fill and linear: combines the previous two options.
   4.3) The *Contour Interval* dialog box allows for the definition of the contour lines.
      – Specified Interval: the contour lines are equidistant; the difference between two successive levels is specified. The number of contour lines is adjusted automatically.
      – Number of contours: the number of contour line sis specified, the difference between two contour line sis computed automatically
      – Specified values: allows for arbitrary contour intervals.
   4.3) Activating *Specify a range* allows only the values between a given minimum and maximum to be plotted
      – Min: 0.5, disable «Fill below»
      – Max: 0.8, enable «Fill above»

   With this specification, water depths smaller than 0.5m will not be displayed. Water depths larger than 80 cm will be displayed, but the colour will be the same as for h = 0.8 m.

5) Making the map more readable.
   5.1) Disable the mesh display: *Display Options / 2D Mesh*, disable *Elements*, *Contours* and other items.
   5.2) The zooming tool (magnifying glass pictogram on the toolbar) may be used to visualize the details of certain zones of interest. To zoom out, use the “Frame” function on the tool bar (a rectangle included in a circle).
   5.3) To clean up the contour lines and remove the interpolation artefacts, proceed as follows.

5.3.1) Project management window: activate *Scatter Data* and select the map to be cleaned up.
5.3.2) Menu bar: click *Scatter*, then *Interpolate to Mesh*.
5.3.3) In the pop up window: *Interpolation Options* / *Interpolation*, select *Inverse Distance Weighted*. The map extrapolation options are disabled automatically.
5.3.4) I *Other Options:* a new 2D data set is created. Its default name is Z_interp. It may be renamed as on Figure 3.8 (new name: z_Mod01_G_nettoyé).

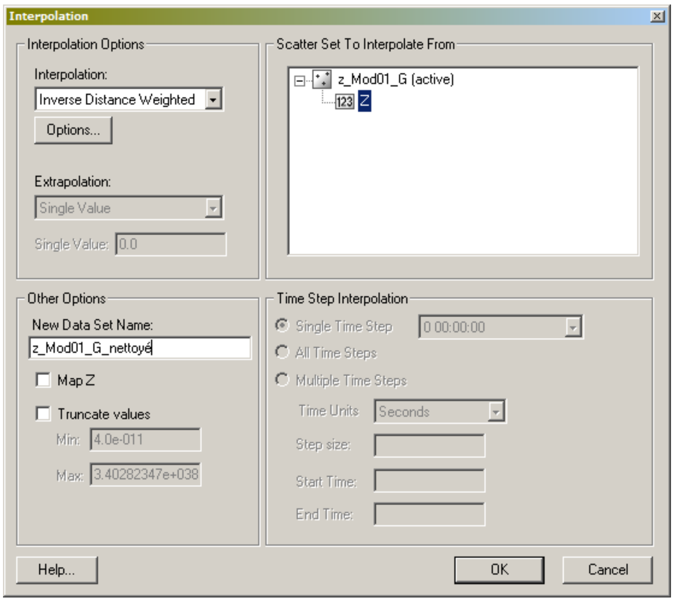

Figure 3.8. Interpolating scatter data onto the mesh. *Interpolation* dialog box.

5.3.5) Confirm with *OK*.
5.3.6) The *Interpolation* dialog bow is closed automatically. A new data set is now visible in the *Mesh Data* window. The name of this data set is that specified in Step 5.3.4).
5.3.7) Disable *Map Data* and *Scatter Data*, enable *Mesh Data*.
5.3.8) In the *Display* / *Display Options*, dialog box, select *2D Mesh*, activate *Contours*, disable *Nodes* and *Elements*.
5.3.9) In the *Contours* thumbnail, select (as on Figure 3.9):
Contour Method: Color Fill and Linear ;
Data Range: Min: 0.49, Max: 0.65 ;
Contour Interval: Specified Interval, contour line spacing 5 mm (0.005).

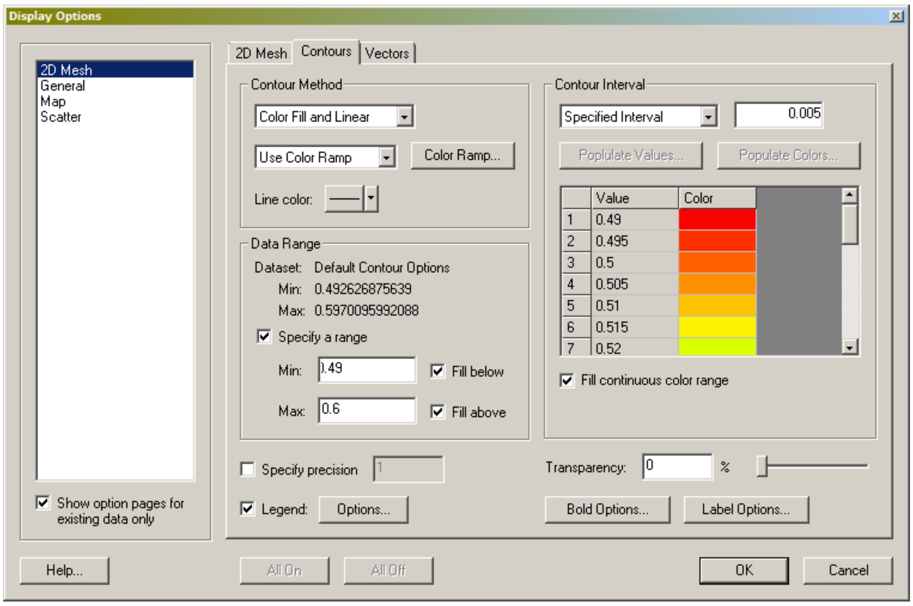

Figure 3.9. The *Contours* dialog box.

5.3.10) Click *OK*. The resulting map is shown on Figure 3.10.

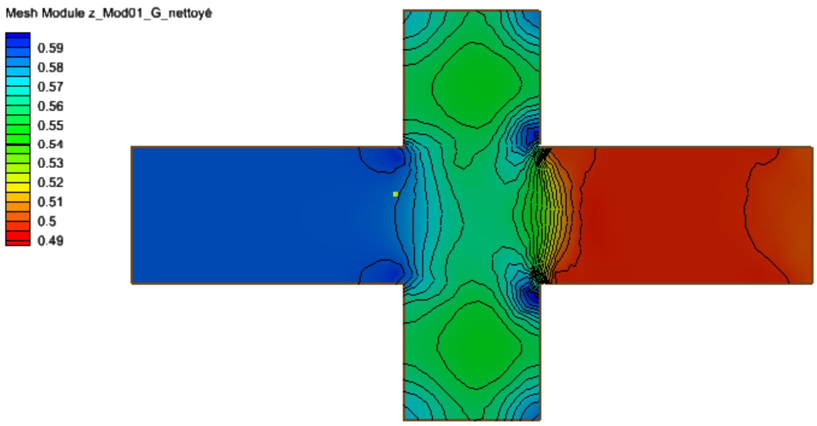

Figure 3.10. Mesh-interpolated free surface elevation map.

The hydraulic head field can be interpolated in the same way. Caution: the Min and Max in the *Data Range* field must be redefined because the range for *H* is not the same as for *z*. Figure 3.11 shows the mesh-interpolated head map.

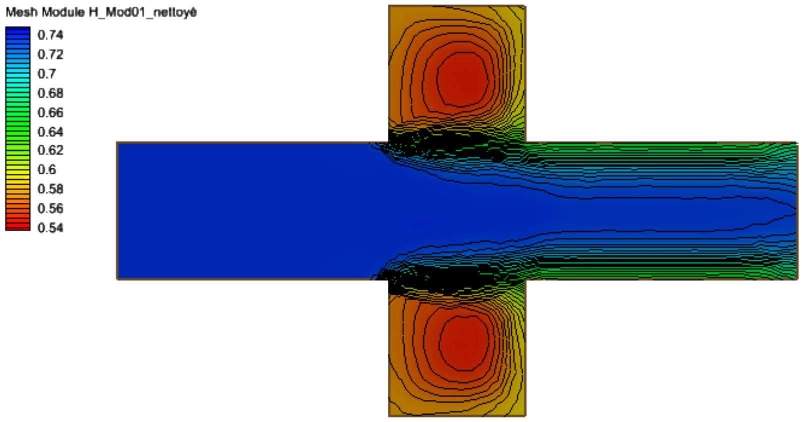

Figure 3.11. Mesh-interpolated hydraulic head.

4.3 Extracting and displaying vector maps

Vector maps are extracted and displayed as follows.

1) Extract the vector map from the .res file. Unit discharge fields are assigned code 3, velocity fields are assigned code 6. The structure of these files is the following: 1 column for the *x*-coordinate, 1 column for the *y*-coordinate, 1 column for the *x*-component of the vector, 1 column for the *y*-component, 1 column for the vector norm (Figure 3.12). Table 3.3 shows the first few lines of the *q*-file.

0.2953912064E+00	-0.4768933723E+01	0.7993282524E+00	-0.3304273878E-02	0.7993350820E+00
0.2025624430E+02	-0.2744848233E-02	0.1067718860E+01	-0.2383804426E-03	0.1067718887E+01
0.7920216803E+01	0.1156338433E+01	0.1065080175E+01	0.3476559266E-02	0.1065085849E+01
0.2258437187E+02	0.2831093483E+01	0.1019523697E+01	0.1171564419E-02	0.1019524370E+01
0.1655658317E+02	0.4710757780E+01	0.8487796796E+00	-0.4048538819E-05	0.8487796797E+00
0.2124786920E+02	0.3820000900E+01	0.9709012145E+00	0.2714782269E-02	0.9709050099E+00
0.2466666667E+02	0.4666666667E+01	0.8770681439E+00	-0.2951798687E-03	0.8770681936E+00
0.2304811553E+02	0.4417560000E+01	0.8763283584E+00	0.9508623065E-02	0.8763799436E+00

Table 3.3. The first 8 lines of the unit discharge file.

2) In the SMS interface.
   2.1) *File / Open* (or Ctrl-O).
   2.2) Select the unit discharge file.
   2.3) In the *Open File Format* dialog box, click *Use Import Wizard*.
   2.4) In *File Import Wizard – Step 1 of 2*, click *Suivant*.
   2.5) A new window pops up.
      – Row *Type*, on the third field, replace *Z* with *Vector X* ;
      – 4th column: replace *<Not Mapped>* with *Vector Y* ;
      – In the blank cells right underneath «Vector X» and «Vector Y», the same variable name must be specified, otherwise generating an error. For the screen capture on Figure 3.12, «Débit unitaire» is entered.
      – The dialog box should look as displayed on Figure 3.12. Click *Terminer* ;
      – The SMS package may issue a warning about duplicate points. This is secondary. Click OK.

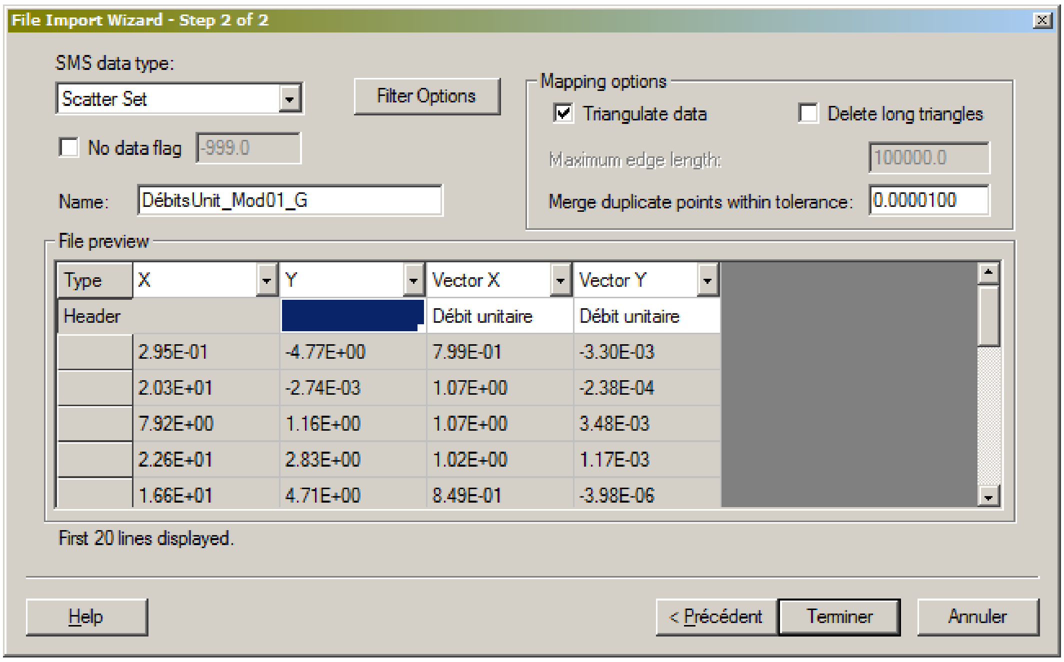

Figure 3.12. Importing vector data.

2.6) A new data set is now visible in the project management window. The vector field may be visible on the screen depending on the visualization default settings. If this is not the case, see step 3.

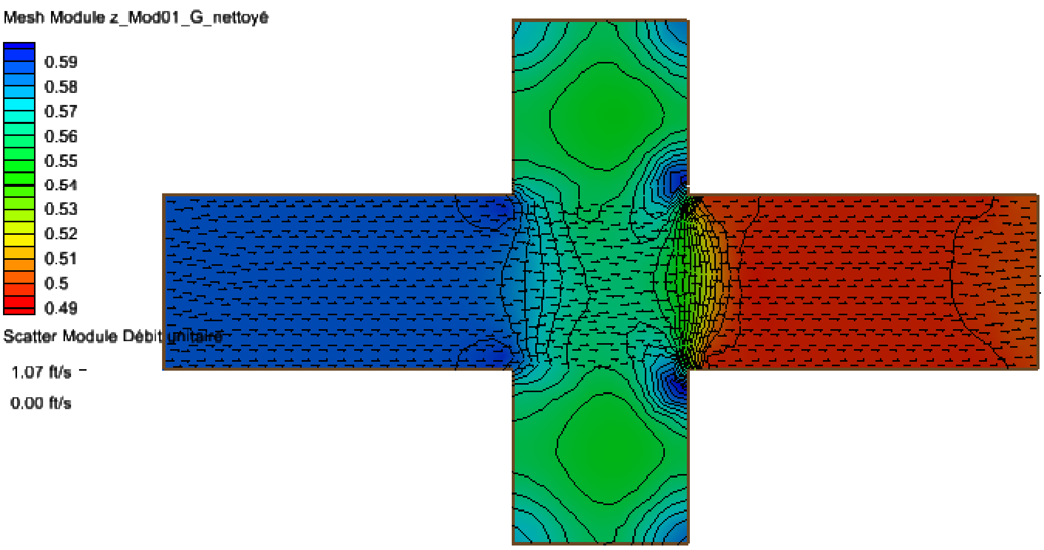

Figure 3.13. Displaying the vector field (after completing step 2.6).

3) Improving map settings. The default scale for the vector field dis in most case not ideal. The vectors may appear too small in some parts of the map, too big in other parts. It might also be that the vectors are not displayed at all. The visualization may be improved as follows.

4.1) *Display Options / Scatter*: check that the *Velocity Vectors* option is activated.

4.2) In the *Vectors* thumbnail.

– *Arrow Options / Shaft length*: select «Scale length to magnitude».

– Ratio: to be decreased if the arrows are too big, to be increased if they are too small. Figure 3.13 was obtained with Ratio = 4. Figure 3.14 was obtained with Ratio = 8.

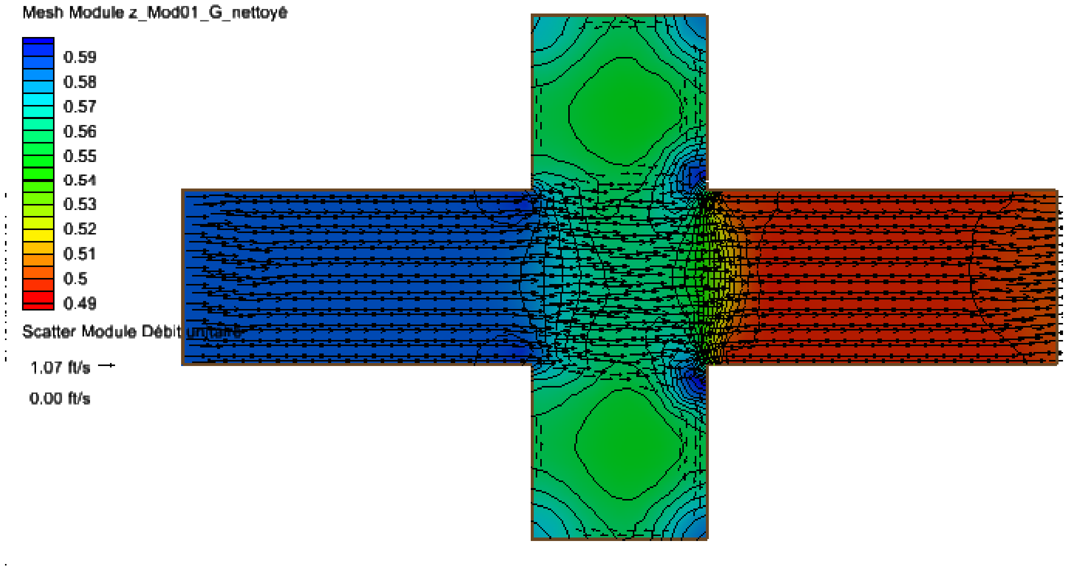

Figure 3.14. Modifying the Ratio parameter.

4.3) You may also be willing to zoom in over some parts of the model (see Figure 3.15, obtained with Ratio = 20).

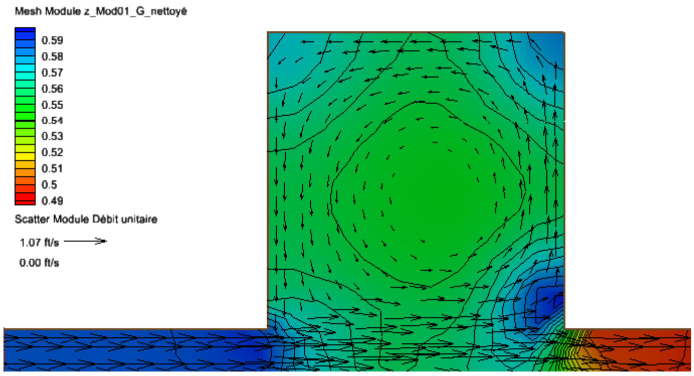

Figure 3.15. Zooming in over the Northern street.

3.5. What is expected from you

The following assignment must be completed in the form of a report (Report#3) to be sent to the teaching staff.

3.5.1. Analysing simulation Mod01_G

1) Analyse the water depth, hydraulic head and unit discharge vector field. More specifically, you are expected to comment on the following aspects.
   - 1.1) Provide an explanation for the dropping water levels at the centre of the Northern and Southern branches of the crossroads (Figure 3.15).
   - 1.2) The free surface elevation and hydraulic head fields have very different shapes (compare Figs. 3.10 and 3.11). How can this difference between the two fields be explained?
   - 1.3) In particular, why is the free surface elevation field uniform across the Eastern street (downstream street), while the hydraulic head is not? Why is the hydraulic head smaller next to the wall than in the centre of the street?

1.4) Considering that the flow is assumed ideal ("fluide parfait" in French) (because both the *res* flag for bed friction and the *dif* flag for viscosity effects are disabled), do you consider that the numerical solution is reliable? What should the analytical (exact) solution look like?

3.5.2. Comparing with another simulation

You shall now carry out the same simulation as previously, using a slightly different mesh. You shall repeat the operations described in Sections 3.2 – 3.4, bearing in mind that

– the crossroads geometry as well as the initial and boundary conditions are strictly identical to those of the above model,
– the computation mesh (Mode02.2dm) is slightly different. The cell interfaces across the intersection with the Northern and Southern streets are aligned with the main street direction (parallel to the *x*-axis).

N.B. Note that the interfaces are aligned with the *x*-direction but do not form an impermeable boundary. The mass and momentum fluxes across these interfaces are not necessarily zero.

The SMS and mesh file for the new model are named Mod02.sms and Mod02.2dm respectively.

– Construct a new .bc file for the new mesh (name it Mod02.bc not to overwrite the Mod01.bc file you created in Section 3.2).
– Generate the corresponding Mod02.geo file.
– Launch the simulation (do not forget to change the .res file name, Mod02.res so as not to overwrite the Mod01.res result file).
– Extract the water depth, hydraulic head, unit discharge and velocity field maps for t = 600 s as in the previous simulation.

Warning. Be careful not to overwrite the result files for the first model. It is advised that all input and output files be renamed from Mod01_xxx into Mod02_xxx.

2) Result analysis.
   2.1) In the light of question 1.4, do you consider the new results more accurate (closer to the exact solution) than those of the first model?
   2.2) Where does the difference with the results of Model 1 stem from? It is reminded that the same geometry, initial and boundary conditions are used.
   2.3) What kind of "good practice" for mesh generation can you infer in the presence of crossroads and bifurcations?

Chapter 4

Modelling an idealized meandering channel

4.1. Introduction

4.1.1. Model

The purpose of the present chapter is to build up and analyse the behaviour of a 2D model for a meandering channel. The channel is idealized in that its cross-section is perfectly rectangular with a constant width. The meander pattern is periodic. The upstream and downstream boundaries are the only exception, with a rectilinear section. The rectilinear reach has been introduced so as to minimize the perturbations induced by prescribing uniform boundary conditions across.

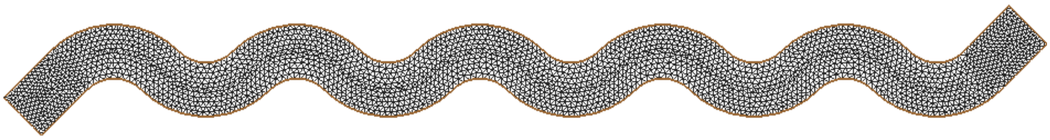

Figure 4.1. View of the model mesh.

Indeed, the meanders induce periodic flow direction changes and the free surface is not horizontal across a given section. This is due to the combined effects of the centrifugal force and hydrostatic pressure distribution. The underlying assumption of 1D models (a free surface that is the same all across a given section) thus becomes invalid. Consequence the velocity field is not uniform within a cross-section either.
For this reason, a 2D model is deemed more accurate and realistic than a 1D model. Modelling the water depth and the velocity field accurately in two dimensions is extremely important for a number of hydraulic studies, in particular those focusing on transport, erosion modelling, etc.

4.1.2. Objectives

The objectives of the present study are the following.

– Set up a 2D model (boundary conditions, initial conditions, parameterization)
– Analyse the simulation results.
– Compare the 2D results with those given by a 1D model.
– Infer a basic rule for the correspondence between the friction coefficient in a 1D model and its numerical value in a 2D model.

4.2. Simulation specifications

Table 4.1 shows the names of the various files supplied and used in the present simulation.

File name	Object	Format
ex01.in	Result extraction file	Text
Mod01.2dm	Model mesh (used by SMS)	Text
Mod01.bc	Boundary condition code file	Text
Mod01.in	2D simulation file	Text
Mod01.lim	Boundary time series file	Text
Mod01.map	Model map file (used by SMS)	Text
Mod01.materials	Auxiliary file (generated by SMS, not used hereafter)	Text
Mod01.sms	SMS project file	Text

Table 4.1. Idealized meander model. File names.

4.3. Launching the simulation, extracting the results

4.3.1. Simulation

The friction coefficient is best calibrated under steady state conditions. The present simulation is therefore carried out until steady state is achieved. Steady state is checked in the simulation by setting so-called drogue points. A drogue point is a point in the mesh where the water depth and the two components of the unit discharge are stored periodically. The storage period is labelled DtTim in the simulation data file.

1) Open the simulation parameter file
   1.1) Check that DtTim is set to 1 s.
   1.2) Go to the bottom of the file. In the original file, only one drogue point is specified (coordinates 500, 13). Figure 4.2 shows the corresponding records.

```
==============================================================
No time series points and for each: x,y
1
500.   13.
```

Figure 4.2. Drogue point file records. Meaning: 1 drogue point is specified (top line), its coordinates are (500, 13) (bottom line).

   1.3) Two points must be added: one next to the upstream boundary, the other next to the downstream boundary.
   - Number of points : change to 3.
   - Above the point (500, 13), insert a new line with the *x* and *y* of a point next to the upstream boundary (e.g. (75, 12)).
   - Below (500, 13), insert a new line with the coordinates of a point next to the downstream boundary (e.g. (1050, 0)).

   The file bottom should now look like the screen capture on Figure 4.3.

```
==============================================================
No time series points and for each: x,y
3
75.    12.
500.   13.
1050.  0.
```

Figure 4.3. Modified file bottom with 3 drogue points.

2) create the Mod01.geo file with geo27d.exe as explained in Chapter 3
3) start the simulation with 2d27d_Intel_STE.exe as explained in Chapter 3

4.3.2. Drogue point data

The drogue data generated during the simulation are stored in the output file named “Mod01_Tim.txt”. This text file may be opened using Excel.

1) Start Excel.
2) Check in the Excel *Options* that the decimal separator is « . » and not « , ». The separator must be a dot because the calculation engine is based on the Fortran language.
3) Open the drogue point file. Format type : *Largeur fixe* (Figure 4.4).

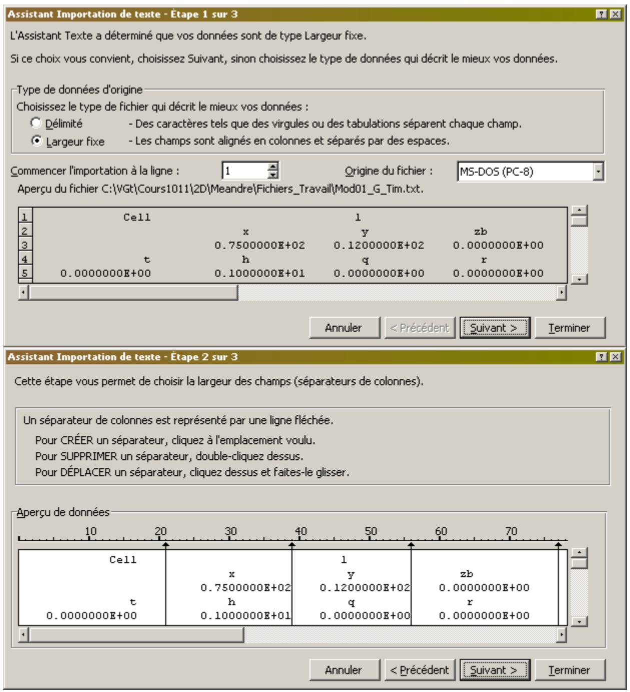

Figure 4.4. Opening the drogue point file with Excel.

4) The file should have the structure shown on Figure 4.5

Cell		1			2			3	
	X	y	zb	x	y	zb	x	y	zb
	7.50E+01	1.20E+01	0.00E+00	5.00E+02	1.30E+01	0.00E+00	1.05E+03	0.00E+00	0.00E+00
t	h	q	r	h	q	r	h	q	r
0.00E+00	1.00E+00	0.00E+00	0.00E+00	1.00E+00	0.00E+00	0.00E+00	1.00E+00	0.00E+00	0.00E+00
1.03E+00	1.00E+00	-3.12E-16	-5.55E-17	1.00E+00	3.63E-16	0.00E+00	1.00E+00	5.55E-17	-3.47E-17
2.03E+00	1.00E+00	-2.91E-16	1.32E-16	1.00E+00	5.85E-16	-4.86E-17	1.00E+00	2.08E-17	-2.29E-16
3.02E+00	1.00E+00	-2.84E-16	3.19E-16	1.00E+00	7.03E-16	-9.02E-17	1.00E+00	-4.86E-17	-3.82E-16
4.02E+00	1.00E+00	-2.84E-16	5.07E-16	1.00E+00	8.27E-16	-9.71E-17	1.00E+00	-1.25E-16	-4.93E-16
5.02E+00	1.00E+00	-2.91E-16	7.08E-16	1.00E+00	9.53E-16	-1.11E-16	1.00E+00	-2.08E-16	-6.18E-16

Figure 4.5. The first lines of the imported drogue result file.

The structure of the drogue file is the following.
Line 1: Number of the drogue point (as many as were defined in the simulation input file).
Line 2: Header specifying the name of the variable on the following line.
Line 3: For each drogue point: x, y and z_b (bottom elevation) of the cell where the drogue point is located.
Line4: Name of the variables on the following lines.
Lines 5 and following: t, and for each drogue point: h, q_x, q_y.

Plotting q_x as a function of time for each of the drogue points leads to the graph on Figure 4.6. The graph was obtained by selecting the first, third, sixth and ninth columns in the file. Similar graphs may be plotted for h and q_y. Since z_b is known, the free surface elevation is easily obtained as $(z_b + h)$. The x– and y–velocities may be obtained by dividing the unit discharge by h.
The graphs on Figure 4.6 indicate that steady state is almost reached at the end of the simulation.

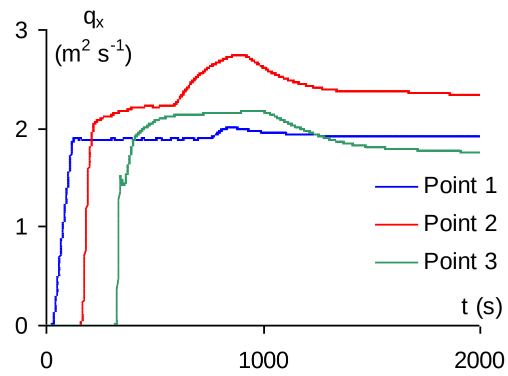

Figure 4.6. Unit discharge q_x time series at the three drogue points.

4.3.3. Map extraction

Extract the following maps at t = Tmax (end of the simulation):
- free surface elevation
- hydraulic head.

See the instructions in Chapter 3 for a refresher. If needed, clean up the maps using mesh interpolation.

4.4. Assignment

1) 2D Result analysis.
   - 1.1) Compute the average value of the hydraulic head across the downstream cross-section of the model (last row of cells). Compare it with the average value of the head in the upstream cross-section (first row of cells). Is H conserved between the upstream and downstream part of the model?
   - 1.2) Since the simulation is assumed frictionless (the “res” flag is set to zero in the simulation input file), was this result expected?
   - 1.3) How much should be the friction coefficient (be it K_{Str}, n_M or C) in a 1D model to create the same head loss?
   - 1.4) How can the difference between the behaviours of the 1D and 2D models be explained?

2) Run a new simulation with the resistance flag activated (res = 1 in the 2D simulation file) and a Strickler coefficient K_{Str} = 70 in both directions (the friction coefficient may be anisotropic, so for isotropic friction both the x- and y-values must be identical). How much should be the Strickler coefficient in a 1D model to give the same head loss?

Chapter 5

Modelling of the Gardon river near Alès

5.1. Introduction

5.1.1. Site characteristics – added value of the 2D model over a 1D model

The site under study ("Pont de Ners") is a reach of the Garon river near Ners, downstream of Alès. The site presents typical features where a two-dimensional model is needed:

– a strong bend in the main channel,
– a complex channel section geometry, where drying beds may cause the main channel to split into parallel branches, with strongly non-uniform flow velocities across a given section,
– sudden section widenings and narrowings, inducing significant transverse flow velocitites,
– the strong narrowing of the floodplain near the downstream section induced by the road embankment, that may induce orifice-like behaviours,
– a sill, making an angle of approximately 45 degrees with the flow main direction, with an elevation variation of approx. 1.80m. The steep downstream slope may induce transcritical flows, a situation that is known to be poorly handled by 1D software packages (see previous exercise sessions).

5.1.2. Objectives

The objectives of the present exercise sessions are the following :

– analyse the hydraulic behavior fo the meander,
– check the influence and accuracy of the downstream boundary condition on the simulation results,
– assess the respective influence of the geometry, friction parameters and boundary conditions on the behaviour of the model.

5.2. Simulation features

The data files provided for the study are given in Table 5.1.

Nom du fichier	Objet	Format
ex02.in	Extraction specification file	Texte
Gardon02.2dm	Mesh file	Texte
Gardon02.bc	Boundary codes file	Texte
Gardon02.geo	Compiled geometry file	Binaire
Gardon02.sms	SMS project file	Texte
Gadon02_Crue10.in	Simulation parameter file	Texte
Gardon02_Crue10.lim	Boundary condition time series file	Texte
Gardon02_Crue10.res	Simulation result file with the original parameter set	Binaire
Gardon02_Crue10_BC.txt	Boundary discharge time series file	Texte
Gardon02_Crue10_Tr.txt	Simulation report file	Texte
Gardon02_Crue10_NF.res	Simulation result file under frictionless assumption	Binaire

Table 5.1. Input and output files provided for this exercise.

The mesh and topography are illustrated by Figure 5.1

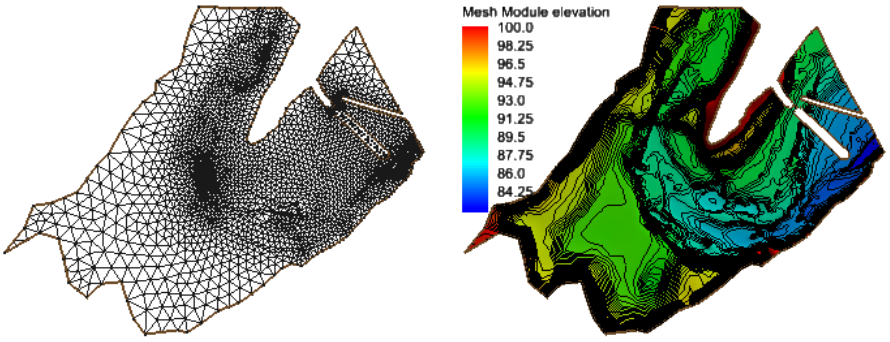

Figure 5.1. The 2D mesh (left) and model topography (right). Note the areas excluded from the mesh, that correspond to the road embankments.

In order to display the left-hand mesh on Figure 5.1, perform the following operations:

1) Start SMS.
2) open the .sms projet file.
3) Project management window :
   3.1) disable *Scatter Data*, *Map Data* and *Images*,
   3.2) enable *Mesh Data*.
4) Click *Display / Display Options*, then select *2D Mesh*.
5) Enable *Elements*, disable *Contours*
6) Disable *Elements*, enable *Contours*.

5.3. Extracting and displaying the results

5.3.1. Stored variables

The variable codes are gin in Table 5.2.

Code	Variable	Symbol
1	Water depth	h
2	Free surface elevation	z
3	Unit discharge	q_x , q_y, $\lvert \mathbf{q} \rvert$
6	Flow velocity	V_x , V_y, $\lvert \mathbf{V} \rvert$
9	Froude number	Fr
10	CFL number	Cr
11	Water depth under the ground surface	η
12	Specific head $(h + V^2)/(2g)$	H_s
13	Hydraulic head $(z + V^2)/(2g)$	H ou H_t

Table 5.2. Variable codes.

Only variables that were stored during the simulation can be extracted from the result file for display purposes. It is recalled that the codes of the variables to be stored during a given simulation can be found at the bottom of the simulation input file for the **2d** executable. Figure 5.2 shows the bottom lines of the file Gardon02_Crue10.in.

```
===============================================================
Flag for result storage
h      z      (qx,qy,|q|)  (Vx,Vy,|V|)  Fr     Cr
1      1      1            1            1      0
--
Etha   Hs     Ht
0      0      1
===============================================================
```

Figure 5.2. Simulation storage specification records in the file Gardon02_Crue10.in.

If the flag for a given variable is 1, the variable is to be stored during the simulation. A flag set to zero means that the variable is not to be stored.

The map storage time step is given by *DtMap* As shown on Figure 5.3 a storge time step of 20 minutes is specified in the file Gardon02_Crue10.in.

```
--
T0     Tmax        DtMax       DtMap DtTim
0.     57600.      5.4654      1200. 10.0
--
```

Figure 5.3. Time step specification in the Gardon02_Crue10.in file.

5.3.2. Extracting and displaying scalar maps

This step has been presented in the previous chapters. Hereafter is a brief reminder of the operation sequence.

1) Open the extraction specification file (Figure 5.4). The file structure is the following:
   - 2 comment lines for the user's convenience.
   - 3 data lines:
     - Name of the .geo file,
     - Name of the .res file,
     - Threshold value for variable extraction. IThe user may specified that the variables are not extracted in the cells where the water depth is below this threshold.
   - 2 comment lines.
   - The number *N* of maps to be extracted (15 in the present example).
   - *N* lines specifying the data for each map:
     - the extraction time (in seconds),
     - the code of the variable to be extracted (see Table 5.2),
     - the name of the map file where the results will be stored. The name shoud be between quotes,
     - The flag « c » (between quotes) that specifies that the results are to be extracted for the centroids of the computational cells.

```
Result extraction file
Geometry Binary results file name, hmin
Gardon02.geo
Gardon02_Crue10.res
0.001
--
Nb of  dates and for each: t, Var Type, Outfile (between quotes)
15
14400. 1      'G22_14400_h.txt'   'c'
18000. 1      'G22_18000_h.txt'   'c'
21600. 1      'G22_21600_h.txt'   'c'
25200. 1      'G22_25200_h.txt'   'c'
28800. 1      'G22_28800_h.txt'   'c'
32400. 1      'G22_32400_h.txt'   'c'
36000. 1      'G22_36000_h.txt'   'c'
39600. 1      'G22_39600_h.txt'   'c'
43200. 1      'G22_43200_h.txt'   'c'
46800. 1      'G22_46800_h.txt'   'c'
50400. 1      'G22_50400_h.txt'   'c'
54000. 1      'G22_54000_h.txt'   'c'

28800. 2      'G22_28800_z.txt'   'c'
28800. 3      'G22_28800_qx.txt'  'c'
28800. 4      'G22_28800_qy.txt'  'c'
```

Figure 5.4. Sample extraction file (here the file ex02.in).

Warning. The subsequent lines in the file (Max, Sub, Sur, Tim) will not be used in this exercise. However, they must not be removed from the file, otherwise leading to an execution error.

2) Start a DOS prompt (cmd in the Windows search line).
   2.1) Type the name of the **ex** executable.
   2.2) Extraction file name: enter *ex02.in*.
   2.3) **ex** is launched. During the extraction process, a number of messages are issued:
      – the file number,
      – the storage time sought in the result fils,
      – the corresponding variable code,
      – « ... found » when the storage time has been located successfully in the .res file,
      – « Opening [file] » indicates that the extracted results are being stored in [file].

3) In the SMS mesh generator.
   3.1) *Display / Display Options* : in *2D Mesh*, disable *Contours*, *Nodes* and *Elements* ;
   3.2) *File / Open* or Ctrl-O. The file selection box opens up.
   3.3) Select the extracted map file, say G22_14400_h.txt, click *Ouvrir*.
   3.4) *Open File Format* dialox box: select *Use Import Wizard*, confirm with *OK*.
   3.5) *File Import Wizard – Step 1 of 2*. Click *Suivant*.
   3.6) *File Import Wizard – Step 2 of 2*. Click *Terminer*.
   3.7) Project management window: the *Scatter Data* item now counts one more item, with the name of the map file that has been opened (without the .txt extension).
   3.8) Enable *Scatter Data*, click G22_14400_h.
   3.9) Click *Display / Display Options*, select Scatter in the item list. Enable *Contours*. The view on Figure 5.5 is obtained.

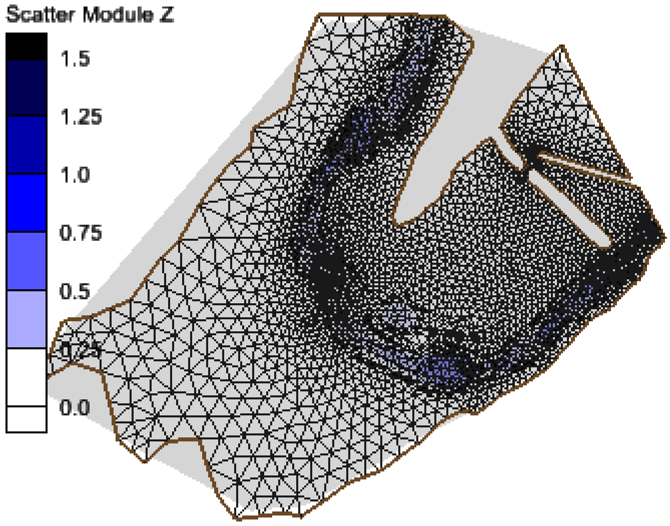

Figure 5.5. Water depths at t = 14400 s.

4) The colour scales are modified as follows.
   4.1) *Display Options / Scatter*, select the *Contours* thumbnail.
   4.2) In the dialog box *Contour Interval*, selecting « Specified Interval », allows for equidistant contours. Selecting « Number of contours » allows for the specification of the number of contour levels. « Specified Values » allows for non-equidistant contour levels.
   4.3) Activating *Specify a range* allows the lower and upper limits of the colour scale to be adjusted.
   – Min : 0.01, disable« Fill below »
   – Max : 1.5, enable « Fill above »
   With this combination, water depths smaller than 1cm will not be filled with colours. Water depths larger than 1.5m will be filled with the colour that corresponds to 1.5m.
   4.4) The transparency of the map may be adjusted. This feature is useful when several maps are to be superimposed with each other, or with e.g. the mesh (see 5) hereafter).
5) For a better map readability,
– the mesh may be disabled (*Display Options / 2D Mesh* disable *Elements*, *Contours* and other items),
– activate the image background (IGN map): enable *Images* in the project management window. Figure 5.6 is obtained.

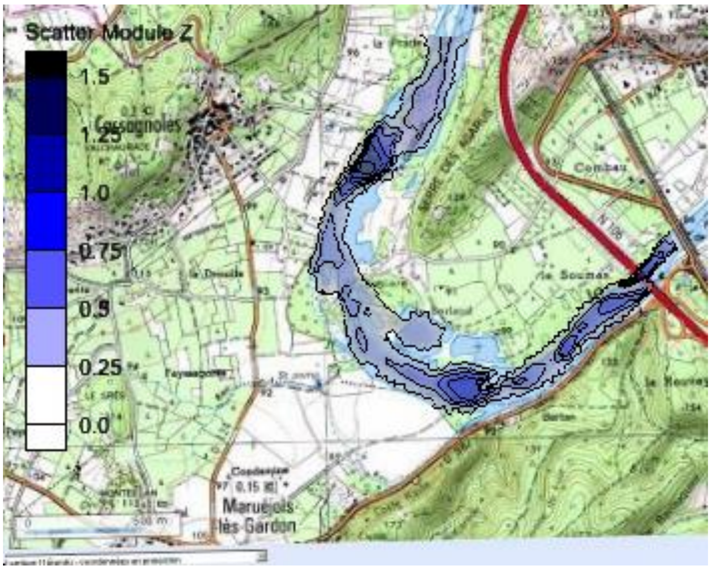

Figure 5.6. Superimposing the water depth map and the background image.

The zooming too (magnifying glass on the toolbar) allows for the inspection of zones of interest in the model (Figure 5.7).

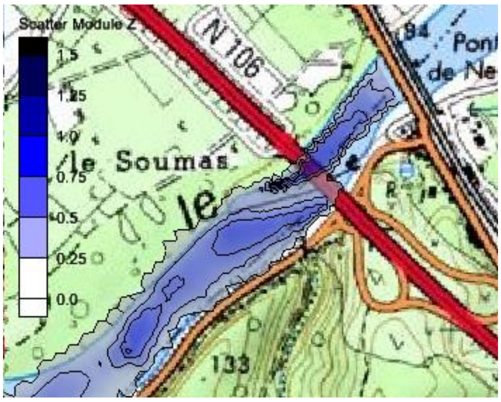

Figure 5.7. Using the zooming tool.

To go back to the global model view, click the « Frame » tool (a rectangle within a cercle) on the menu bar.

5.3.3. Extracting and visualizing vector maps

1) Start SMS,
   1.1) *File / Open* or Ctrl-O.
   1.2) Select the vecor map extracted using **ex**.
   1.3) In the *Open File Format* dialog box, *Use Import Wizard*.
   1.4) In *File Import Wizard – Step 1 of 2*, click *Suivant*.
   1.5) In the newly opened dialog box,
      – *Type*: click the menu arrow on the right-hand side of the 3rd column, change *Z* into *Vector X*.
      – Click the menu arrow on the right-hand side of the 4th column, change the blank field into *Vector Y*.
      – In the two blank cells, enter the same variable name, e.g. “unit discharge” or “Débit unitaire”.
      – The dialog box should look like Figure 5. Click *Terminer*.

– The software may warn about duplicate points in the data set. You may ignore this, click OK.

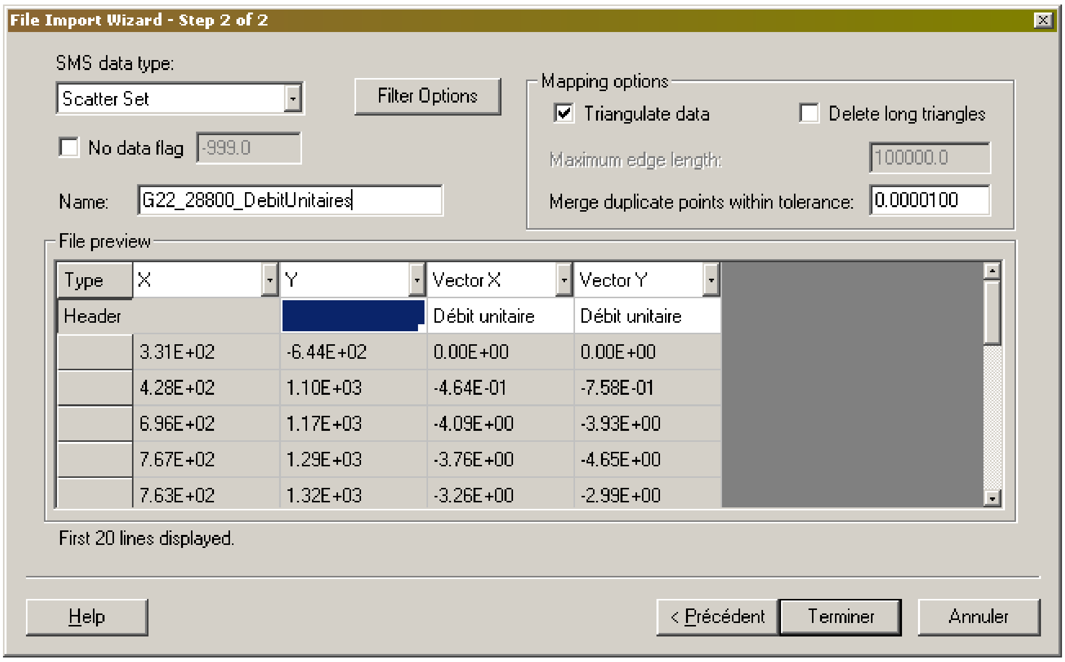

Figure 5.8. Importing vector data.

1.6) A new item appears in the project management window. The vector field is displayed (Figure 5.9). If this is not the case, see step 2).

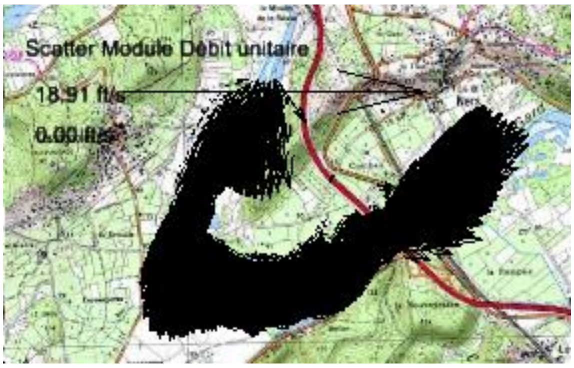

Figure 5.9. Displaying the vector field at the end of step 1.6).

2) Improving the visualization. The default vector scale is most likely not to be optimal. It may also occur that the default settings of SMS do not allow for vector visualization. These problems may be overcome as follows.
   2.1) *Display Options / Scatter*: enable *Velocity Vectors*.
   4.2) *Vectors* thumbnail.
   – *Arrow Options / Shaft length*: select « Scale length to magnitude »,
   – Ratio: decrease it if the arrows are too big, increase it if the arrows are too small. Figure 5.10 was obtained with Ratio = 2.0.
   4.3) Il est également possible de zoomer sur les parties du modèle.

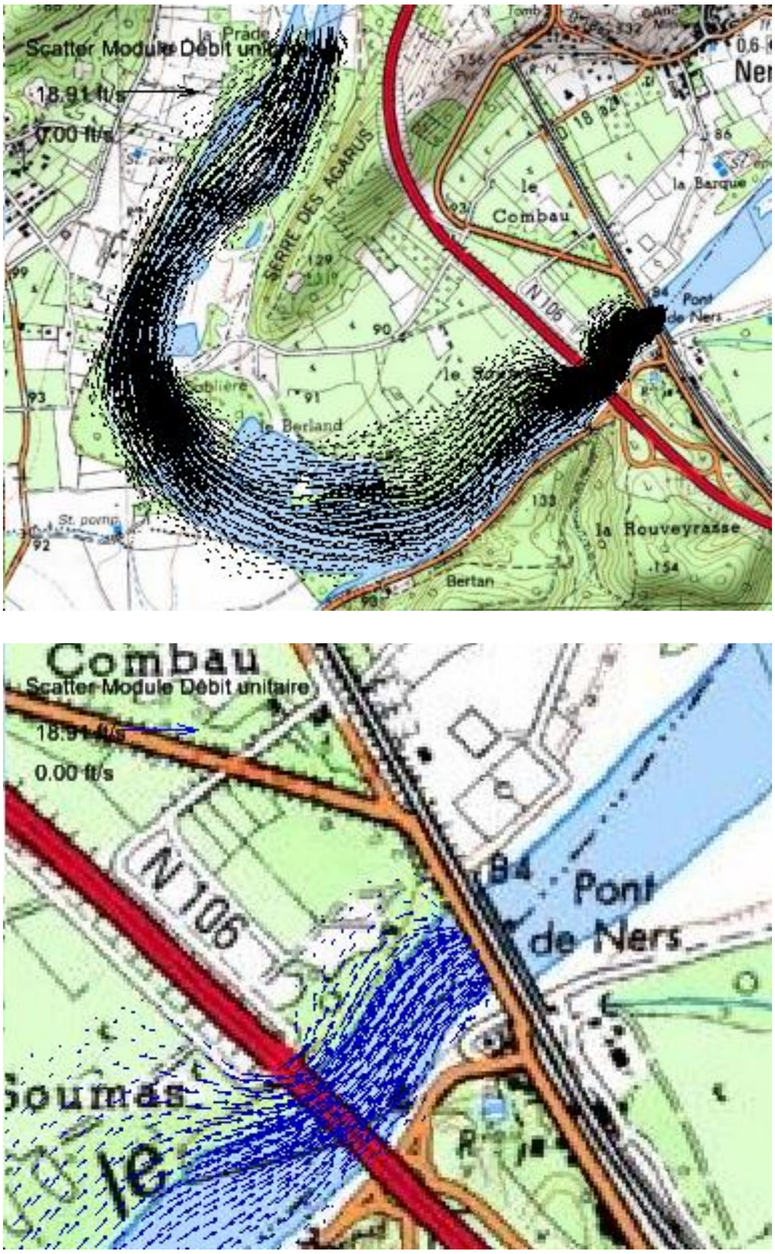

Figure 5.10. Vector field with Ratio = 2.

5.4. Assignment

5.4.1. Analysing the vector fields

Extract the velocity and unit discharge vector fields near the time of the maximum discharge.
Compare the two vector fields. Are the maximum unit discharge and flow velocity fields correlated? Which of the two fields seems the more relevant to you in the analysis of the hydraulics of the floodplain?

5.4.2. Analysing the flow regime

For the same simulated time as above, map the Froude number. Zoom-in over the downstream region of the model. Considering the Froude number values in this area, what is your assessment of the accuracy of the downstream boundary condition?

In other words: assuming that the boundary condition is wrongly defined (e.g. because the discharge coefficient prescribed at the downstream boundary condition has been wrongly evaluated), what is your assessment of the consequences of this error on the simulation results all over the model? For instance, is this error likely to remain confined downstream of the road embankment and the sill?
Your conclusions should be substantiated with maps, graphs, etc. Carrying out additional simulations is allowed (and advised).

5.4.3. Influence of roughness and boundary conditions

1) Assess the influence of the roughness coefficient on the simulation results using the same method as above.
2) Carry out the same analysis for the peak discharge in the upstream hydrograph.
3) Draw out conclusions: what is the most important source of error in a 2D model? A wrong assessment of the friction coefficient, or a wrong assessment of the boundary conditions?
4) Compare the respective influence of the friction coefficient and the geometry on the total head loss over the model.

Appendix. File formats

A.1. Simulation input file

A.1.1. File structure

Line no	Variables
1, 2, 3	Comment lines (for user)
4	Simtyp, hot, hyp, dif, res, ori, prec, inf, exch, wind, pmp, gat, net
5, 6	Comment lines
7	g, D
8, 9	Comment lines
10	T0, Tmax, DtMax, DtMap, DtTim
11, 12	Comment lines
13	Schem, solv, NitMax, Eps, hmin, Crmax, Vmax, FrMax, div, NitDivMax, hstop
14, 15	Comment lines
16	Geometry file name (.geo file)
17, 18, 19	Comment lines
20	**u** or **U** if Boussinesq coefficient is uniform; **v** or **V** is Boussinesq coefficient is nonuniform
21	Boussinesq coefficient (taken into account only if **u** or **U** in line 20)
22	Boussinesq coefficient file name (taken into account only if **v** or **V** in line 20)
23 – 25	Comment lines
26	**u** or **U** if porosity and Strickler are uniform; **v** or **V** is porosity and Strickler are non uniform
27	Phi, K1, K2, ThetaK (taken into account only if **u** or **U** in line 20)
28	Hydraulic parameter file name (taken into account only if **v** or **V** in line 26)
29 – 31	Comment lines
32	**u** or **U** if singular head loss parameters are uniform; **v** or **V** if nonuniform
33	s1, s2, Alpha
34	Head loss parameter file name (taken into account only if **v** or **V** in line 32)
35 – 36	Comment lines
37	Name of orifice data file
38	Comment lines
41	**u** or **U** if rainfall station code is uniform; **v** or **V** otherwise
42	Rainfall code (taken into account only if **u** or **U** in line 41)
43	Name of the file for the rainfall station map
44 – 46	Comment lines
47	**u** or **U** if infiltration coefficient is uniform; **v** or **V** otherwise
48	Infiltration coefficient (taken into account only if **u** or **U** in line 47)
49	Name of the file for the infiltration coefficient map
50 – 52	Comment lines
53	**u** or **U** if building exchange parameters are uniform; **v** or **V** otherwise
54	Dzb, Kexch (taken into account only if **u** or **U** in line 53)
55	Name of the file for the building exchange parameters map
56 – 58	Comment lines
59	**u** or **U** if wind station code is uniform; **v** or **V** otherwise
60	Wind station code (taken into account only if **u** or **U** in line 59)
61	Name of the file for the wind station map
62 – 64	Comment lines
65	**u** or **U** if initial flow field (z, u, v) is uniform; **v** or **V** otherwise
66	z_{init}, u_{init}, v_{init} (taken into account only if **u** or **U** in line 65)
67	Name of the file for the initial conditions map
68 – 70	Comment lines
71	**u** or **U** if initial water depth in buildings is uniform; **v** or **V** otherwise
72	Etha init (taken into account only if **u** or **U** in line 71)
73	Name of the file for the initial conditions in buildings map
74 – 76	Comment lines
77	Name of the file for boundary conditions time series

78	Comment lines
79	Name of the file for the precipitation time series (if prec = 0, juste write a dummy string)
80, 81	Comment lines
82	Name of the file for the wind time series (if wind = 0, juste write a dummy string)
83, 84	Comment lines
85	Name of the pump data file
86, 87	Comment lines
88	Name of the gate data file
89, 90	Comment lines
91	Name of the network data file
92 – 94	Comment lines
95	Name of the binary result file (map variables, storage period DtMap)
96	Name of the time series file (storage period DtTim)
97	Name of the boundary flux file (storage period DtTim)
98	Name of orifice output file (storage period DtTim)
99	Name of pump output file (storage period DtTim)
100	Name of gate output file (storage period DtTim)
101	Name of drainage network outpu file (storage period DtTim)
102	Name of trace file
103 – 105	Comment lines
106	Flags for storage of h, z, unit discharges, velocities, Fr, Cr (storage period DtMap)
107, 108	Comment lines
109	Flags for map storage of Etha, Hs, Ht (storage period DtMap)
110 – 112	Comment lines
113	Flag for calculation of discharge through pre-defined sections (0 or 1)
114	Name of section definition file
115	Name of section discharge output file
116, 117	Comment lines
118	Number of time series storage points (storage period DtTim)
119 – end	For each storage point: *x* and *y*

A.1.2. Meaning of the variables

Variable	Type	Unit	Meaning
Alpha	R*8	rad	Angle of principal direction 1 with the *x* axis
CrMax	R*8	–	Maximum allowed Courant number for stability. Only values smaller than 1 are taken into account
D	R*8 (double)	$m^2\ s^{-1}$	Diffusion coefficient. Active only if dif = 1
dif	I	–	0 : no diffusion ; 1: diffusion is activated
div	I	–	0 : no divergence correction 1 : divergence correction activated If div = 1 please set hstop to 0
DtMap	R*8	s	Time step for the storage of 2D maps
DtMax	R*8	s	Maximum time step fixed by the user
DtTim	R*8	s	Time step for the storage of time series at fixed points
Dzb	R*8	m	Elevation of building basement relative to ground (positive is building basement is above ground level, negative if building basement is below ground level)
Eps	R*8	–	Relative convergence criterion for conjugate gradients (diffusion step, only if dif = 1)
Etha Init	R*8	m	Initial water depth in the basement of the buildings
FrMax	R*8	–	Maximum Froude number allowed by the user (if Fr is larger, the velocities will be reduced so that Fr = FrMax)
g	R*8	$m\ s^{-2}$	Gravitational acceleration
gat	I	–	0: no controlled gates 1: gates handled via 1st-order time splitting 2: gates handled via 2nd-order time splitting
hmin	R*8	m	Water depth under which the fluxes with the neighbour cells are assumed zero
hstop	I	–	0 : computations continue in case of negative depths 1: computations are stopped in case of negative depths

K1	R*8	$m^{1/3}\ s^{-1}$	Strickler coefficient in principal direction 1
K2	R*8	$m^{1/3}\ s^{-1}$	Strickler coefficient in principal direction 2
kexch	R*8	–	Exchange coefficient between buildings and overland flow
net	I	–	Flag for drainage network model
NitMax	I	–	Maximum number of iterations for conjugate gradients (only if dif = 1)
NitDivMax	I	–	Maximum number of iterations in the Divergence Correction routine
ori	I	–	1: Orifices handled via 1st-order time splitting 2: orifices handled via 2nd-order time splitting
Phi	R*8	–	Porosity
res	I	–	0: no friction 1: time splitting + analytical solution over the time step 2: time splitting + explicit Euler 3: no time splitting 5: head loss coefficient for porosity +10: second-order time splitting
s1	R*8	–	Singular head loss coefficient in principal direction 1
s2	R*8	–	Singlura head loss coefficient in principal direction 2
scheme	Char*2 (between quotes)	–	‘g’ : Godunov ‘m1’ MUSCL-EVR ‘m2’ MUSCL-EVR with 2 points/interface
solv	I	–	1 : HLL with porosity 2: HLLC with porosity (Guinot & Soares-Frazao 2006) 22: HLLM (Guinot 2010) 3 : ApStat (Lhomme & Guinot 2007) 4 : PorAS (Finaud-Guyot et al 2009) 61 : hydrostatic reconstruction + HLL 62 : hydrostatic reconstruction + HLLC 63 : hydrostatic reconstruction + ApStat
T0	R*8	s	Simulation begin time
ThetaK	R*8	rad	Angle of Strickler principal direction 1 with the *x* axis
Tmax	R*8	s	Simulation end time
uinit	R*8	$m\ s^{-1}$	Initial flow velocity in the *x* direction
vinit	R*8	$m\ s^{-1}$	Initial flow velocity in the *y* direction
Vmax	R*8	$m\ s^{-1}$	Maximum flow velocity allowed by the user (velocities larger than Vmax are “clipped” to Vmax)
zinit	R*8	m	Initial water level

A.2. Extract file format

Line	Content
1, 2	2 comment lines
3	Name of geometry file
4	Name of simulation result file
5	Depth threshold value for extraction
6, 7	2 comment lines
8	Number *N* of maps to be extracted
9 to *N* + 8	*t*, var, file name, code
End of file	--> Max, etc. **Do not remove these lines**

t : Time for which the variables are to be extracted
var : Variable code (see Table 3.2)
file name : name of the text file where the map is to be extracted. **Between quotes (')**
code : interpolation code, between quotes. **Use 'c'**

A.3. Sample files

A.3.1. Simulation file

```
Input file for SW2D 2.6d
=====================================================================
simtyp hot   hyp   dif       res   ori prec      inf       exch      wind      pmp       gat       net
 1     0     2     0         1     0   0         0         0         0         0         0         0
--
   g    D
9.81    0.
--
T0        Tmax      DtMax               DtMap DtTim
0.        57600.    5.4654    1200.     10.0
--
schem     solv      Nitmx     Eps       hmin      CrMax     Vmax      FrMax     div       Ndvmx     hstop
'g'       61        1         0.        1d-3      0.9       20.       300.      0         10        0
==============================================================
Geometry file
Gardon02.geo
==============================================================
Boussinesq coefficient
Distribution type, File name
u
1.0
No distributed Boussinesq file
==============================================================
Porosity and Strickler
Distribution type, Values (Phi, K1, K2, Alpha), File name
u
1.0       70.       70.       0.
No distributed porosity/Strickler file
==============================================================
Head loss parameters
Distribution type, Values (s1, s2, Alpha), File name
u
0.        0.        0.
No singular head loss file
==============================================================
Orifice parameters
No orifice file
==============================================================
Precipitation codes
Distribution type, Value (code), File name
u
1
No precipitation file
==============================================================
Infiltration coefficient
Distribution type, Value (infiltr. coeff.), File name
u
0.
No infiltration file
==============================================================
Surface/buildings exchange parameters
Distribution type, Values (Dzb, kexch), File name
u
0.        0.
No exchange parameter file
==============================================================
Wind codes
Distribution type, Value (code), File name
u
1
No wind file
==============================================================
Initial conditions surface
Distribution type, Values (z, u, v), File name
u
84.       0.        0.
No initial file
==============================================================
Initial conditions buildings
Distribution type, Values (h), File name
u
0.
No initial conditions file
==============================================================
Boundary conditions time series
Gardon02_Crue10.lim
==============================================================
Precipitation time series
No precipitation file used
==============================================================
Wind time series
No time series file
==============================================================
Pump data file
No pump file used
==============================================================
Gate data file
No Gate File
==============================================================
Network data file
```

```
No network File
============================================================
Output file names
Res (binary), Tseries, Boundary fluxes, Orifice fluxes, PumpQ, Trace file
Gardon02_Crue10.res
Gardon02_Crue10_Tim.txt
Gardon02_Crue10_BC.txt
No orifice file
No pump file
No gate file
No network file
Gardon02_Crue10_Tr.txt
============================================================
Flag for result storage
h         z         (qx,qy,|q|)         (Vx,Vy,|V|)         Fr        Cr
1         1         1                   1                   1         0
--
Etha      Hs        Ht
0         0         1
============================================================
Interface storage flag (0/1)
and definition file for interfaces, result file
0
No definition file
No result file
============================================================
No time series points and for each: x,y
2
620. 930.
1030      -210.
```

A.3.2. Extraction file

```
Result extraction file
Geometry Binary results file name, hmin
Mod01.geo
Mod01_G.res
0.001
--
Nb of  dates and for each: t, Var Type, Outfile (between quotes)
4
2000.   2       'z_Mod01_G.txt'       'c'
2000.   3       'q_Mod01_G.txt'       'c'
2000.   4       'r_Mod01_G.txt'       'c'
2000.   13      'H_Mod01_G.txt'       'c'

-->Max
number of max
0
-->Sub
number of sub
0
-->Sur
Number of subsurf
0
-->Tim
Number of tim
0
```