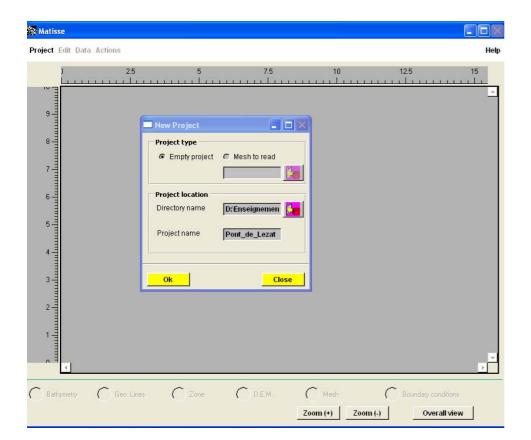# MATISSE

Step by step with Matisse

Summary

- Project Creation
- Data Importing
- Boundary limit creation
- Preparing mesh generation
- Mesh creation
- GEO file generation
- Boundary conditions : groups creation
- Boundary conditions : entities creation and association with groups
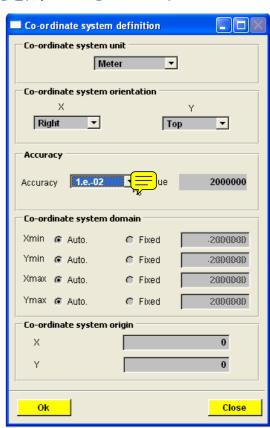- Creation of the CONLIM file

Project Creation

From one or more ASCII files including **Data** such as a polyline defining the limit of the computational area, some topographic and bathymetyric isolines and possibly (x,y,z) points, the aim is to create a geo file mesh and a conlim file for the TELEMAC 2D code .
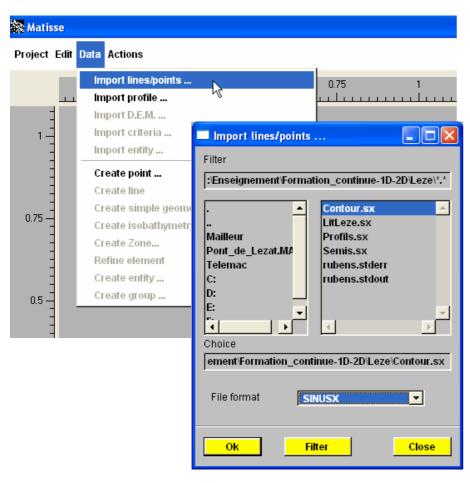
From **Project** menu, choose a new project :

Data Importing

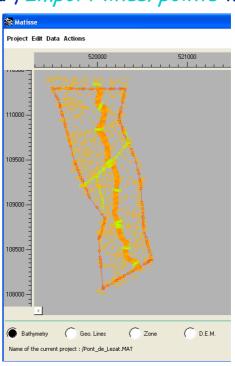In the **Bathymetry** or **Geo lines** mode , from **Data** menu , *Import lines/points* with SinusX format :
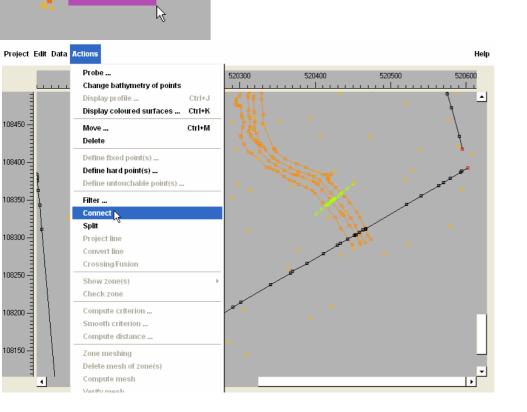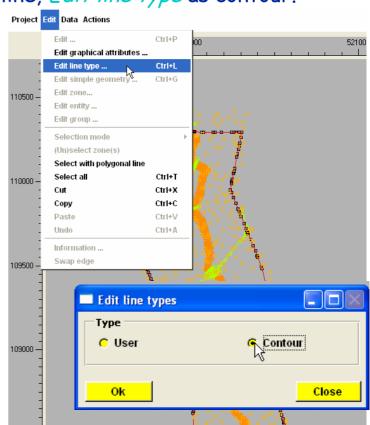
Boundary limit creation

In the *Geo lines* mode, close the line defining the domain boundary
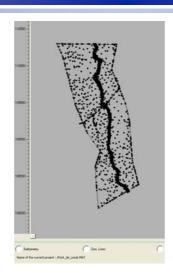with *Connect* and select line, *Edit line type* as contour.

Dont'forget to save

M.M. Maubourguet - J. Chorda 5

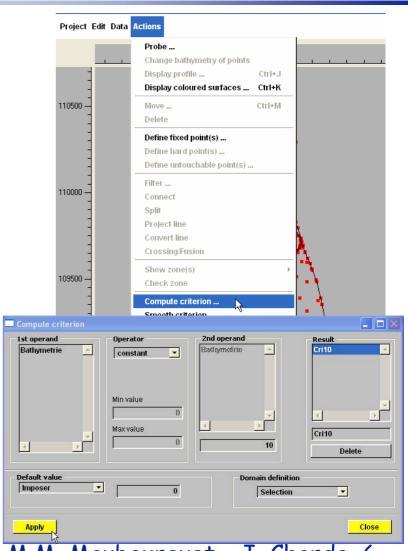# Preparing mesh generation

Go to **D.E.M.** mode,

select the menu **Edition**, then, *Select all*.

From the menu **Actions**, *Compute criterion* (a criterion corresponds to the size of a triangular mesh), Operator constant (here 10 m), fill in the result field (here Cri10m), then type **enter**. Clic on **Apply**, then **Close**.
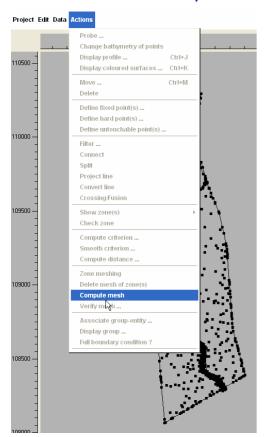
Mesh creation
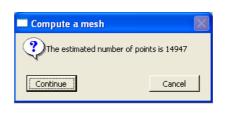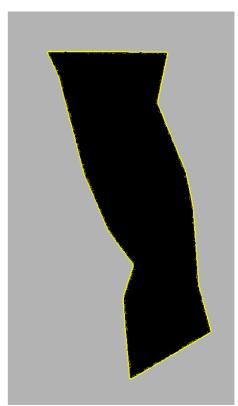
In the **Mesh** mode, select the menu **Actions**, *Compute mesh* and choose a criterion
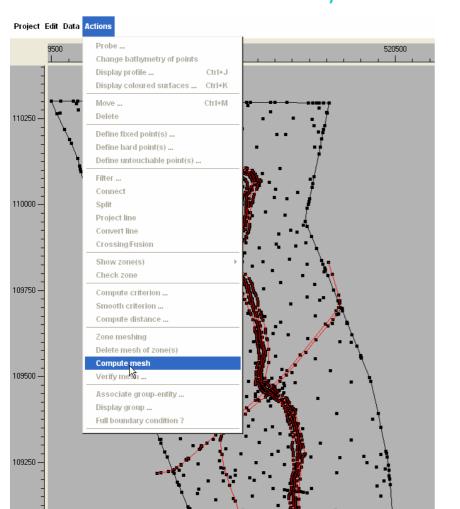
Matisse gives an estimation of the number of nodes, then calculates the mesh.
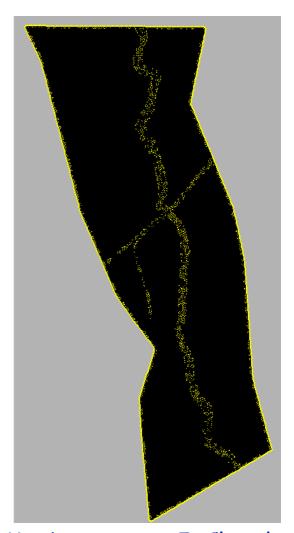
Save again

Mesh creation

In order to constrain the mesh to lie on particular lines, select these ones before *Compute mesh*
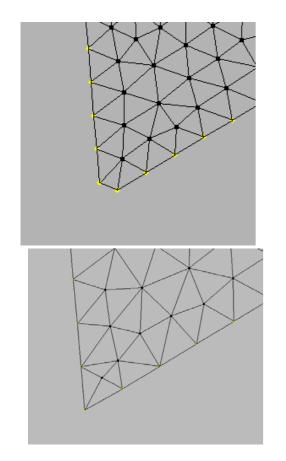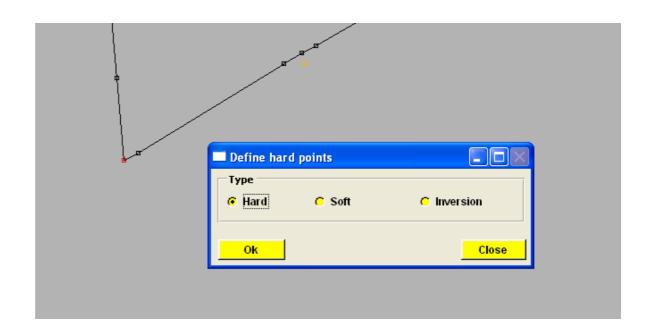
M.M. Maubourguet - J. Chorda 8

Mesh creation

In order to preserve the corner nodes location : back to *Geo lines*, select the concerned points and define them as hard ones.
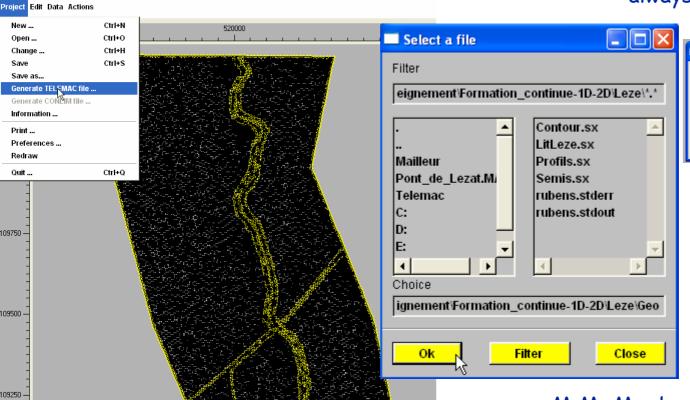
GEO file generation

One can then create the geometry file necessary to TELEMAC :

Menu **Project**, *Generate Telemac file*

Matisse suggests to renumber the mesh elements (you must always accept):

M.M. Maubourguet - J. Chorda 10
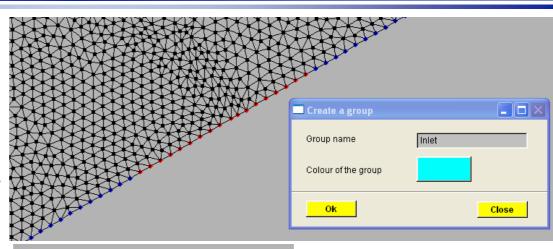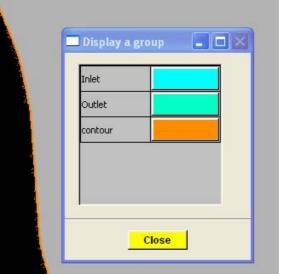
Boundary conditions : groups creation

In the menu **Edition**, it is necessary to assign a group to every boundary node in order to impose various boundary conditions types (solid and liquid). *Select Envelop*, then in the **Data** menu, *Create a Group*

Here, we create the groups « Inlet » « Outlet » and « contour ».

(SAVE AGAIN…)

Boundary conditions : entities creation and association with groups

Now, we create entities in order to define the boundary conditions : in the menu **Data**, *Create an entity*.

Here, we create « Discharge_inlet », « Zdownstream » and «solid_wall » " using the following codes :
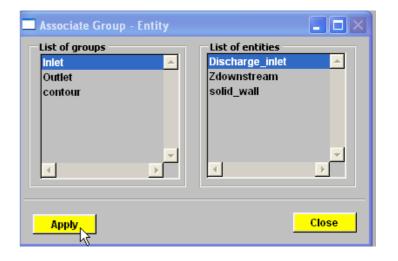
TRIGRID	STB(*)	LIHBOR	LIUBOR	LIVBOR	LITBOR	BOUNDARY CONDITION
1	11	2	2	2	2	Solid boundary.
2	4	5	4	0	4	Prescribed H, free U, zero V, free T.
3	5	1	1	1	4	Incident wave, free T.
4	7	5	0	4	4	Prescribed H , zero U, free V, free T.
5	8	4	5	5	5	Free H, prescribed Q, prescribed T.
6	9	4	6	6	5	Free H, prescribed velocities, prescribed T.
7	1	5	4	4	4	Prescribed H, free velocities, free T.
8	12	4	5	0	5	Free H, prescribed Q with zero V, prescribed T.
9	15	4	0	5	5	Free H, prescribed Q with zero U, prescribed T.
10	2	5	5	5	5	Prescribed H and Q, prescribed T.
11	3	5	6	6	5	Prescribed H and velocities, prescribed T.
12	14	2	0	2	2	Solid boundary with zero U.
13	13	2	2	0	2	Solid boundary with zero V.

Note :   Boundary conditions on the tracer are managed by SISYPHE as sedimentological boundary conditions in the following way :

Then, we associate groups with entities : menu *Actions*, *Associate Group-Entity*, select and **Apply**.

SAVE project again

Generation of Conlim file

We can now generate the Conlim file for TELEMAC : menu **Project**, *Generate Conlim file*.
Matisse verifies that boundary conditions are complete et creates an ASCII file an example of which is given hereafter :

Verification CondLim
Boundary conditions completly defined
Ok

LIHBOR	LIUBOR	LIVBOR	H	U	V		LITBOR	T				
2	0	0	.000	.000	.000	.0	2	.000	.000	.000	20	1
2	0	0	.000	.000	.000	.0	2	.000	.000	.000	21	2
2	0	0	.000	.000	.000	.0	2	.000	.000	.000	22	3
2	0	0	.000	.000	.000	.0	2	.000	.000	.000	23	4
2	0	0	.000	.000	.000	.0	2	.000	.000	.000	24	5
2	0	0	.000	.000	.000	.0	2	.000	.000	.000	25	6
2	0	0	.000	.000	.000	.0	2	.000	.000	.000	26	7
2	0	0	.000	.000	.000	.0	2	.000	.000	.000	27	8
2	0	0	.000	.000	.000	.0	2	.000	.000	.000	28	9
...	...	...	...	...	...	...	...	...	...	...	...	...
2	0	0	.000	.000	.000	.0	2	.000	.000	.000	244	225
2	0	0	.000	.000	.000	.0	2	.000	.000	.000	245	226
2	0	0	.000	.000	.000	.0	2	.000	.000	.000	246	227
2	0	0	.000	.000	.000	.0	2	.000	.000	.000	247	228
2	0	0	.000	.000	.000	.0	2	.000	.000	.000	248	229
4	5	5	.000	.000	.000	.0	5	.000	.000	.000	249	230
4	5	5	.000	.000	.000	.0	5	.000	.000	.000	2540	231
4	5	5	.000	.000	.000	.0	5	.000	.000	.000	2539	232
4	5	5	.000	.000	.000	.0	5	.000	.000	.000	2438	233
4	5	5	.000	.000	.000	.0	5	.000	.000	.000	2238	234
4	5	5	.000	.000	.000	.0	5	.000	.000	.000	2237	235
2	0	0	.000	.000	.000	.0	2	.000	.000	.000	2040	236
2	0	0	.000	.000	.000	.0	2	.000	.000	.000	2039	237
2	0	0	.000	.000	.000	.0	2	.000	.000	.000	1	238

Mesh is done, SAVE a last time !!

Other technics

- Local mesh refinement

- Creation of an embankment
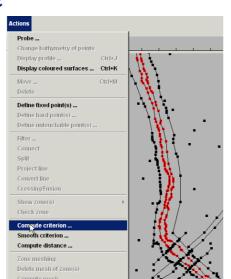
- Using Zones for river meshing
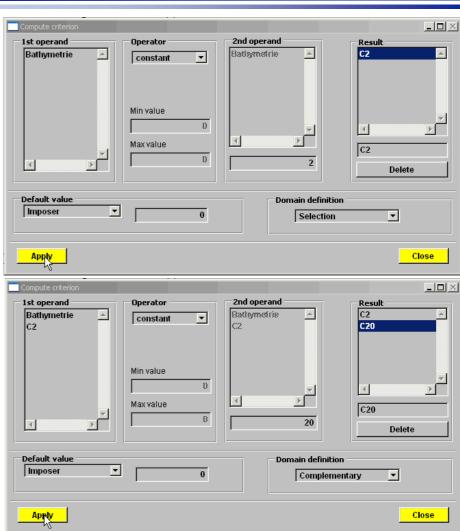
Local refinement in some areas

The first mesh is composed of constant size elements.

Now we want to refine the mesh around the bed river zone.

In *D.E.M.* mode, select the thalweg lines and *Compute creterion*. The first creterion is for the **Selection** (C2) and the second for the **Complementary** (C20)
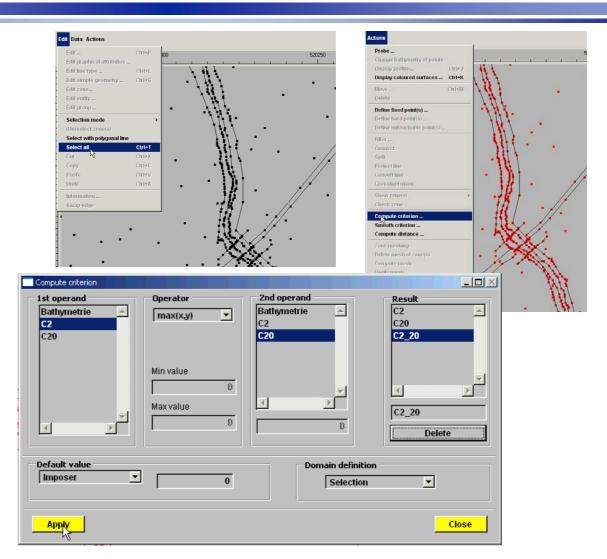
Local mesh refinement

Exit *Compute criterion* mode, unselect thalweg lines, then *select all*.

Compute creterion again with Operator max(x,y).

Select C2 for **1st operand** and C20 for **2nd operand,** choose a name for the new **Result**, clic **Enter**, **Apply**, then **Close** and **Save**.
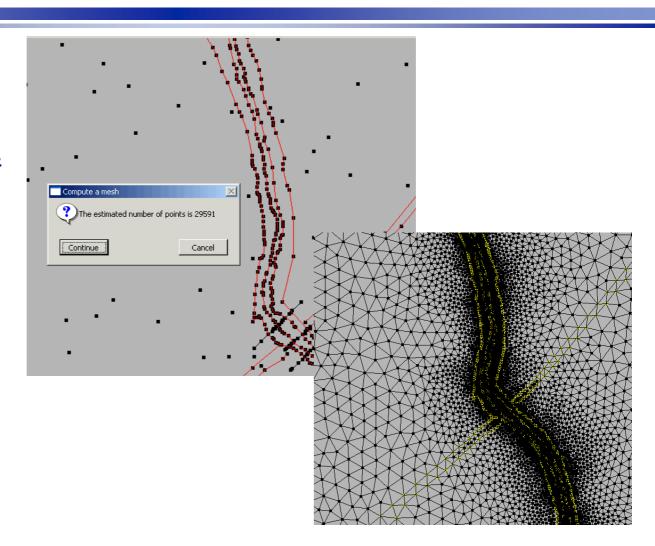
Local mesh refinement

In the **Mesh** mode, select the menu **Actions**, *Compute mesh* and choose C2_20 as criterion.
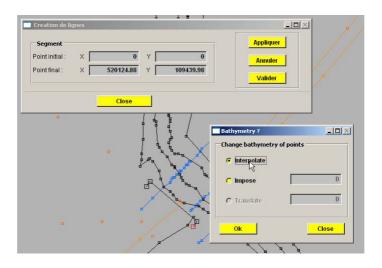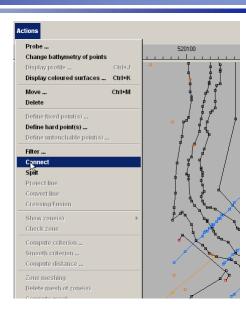
The new mesh shows refined elements near the river bed.

Save and proceed as before
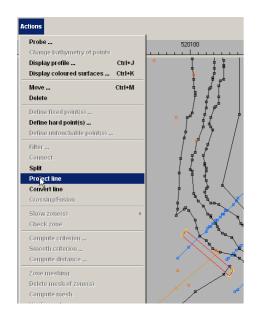

M.M. Maubourguet - J. Chorda 18

Creation of an embankment

In the *Geo lines* mode, from the menu **Data**, *Create Line*, draw the lower part of the dyke, interpolate the bathymetry of the points and connect the first and last points.
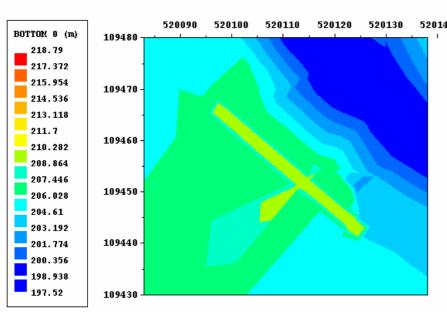

Select the four corners and define hard points.
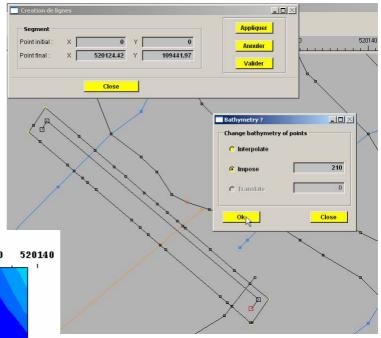
Now, select the line and project on the bathymetry

Creation of an embankment

In the **Geo lines** mode, from the menu **Data**, *Create Line*, draw the upper part of the dyke, impose the bathymetry of the points and connect the first and last points.

Then visualize bathymetry with Rubens.

Now you can save and proceed.

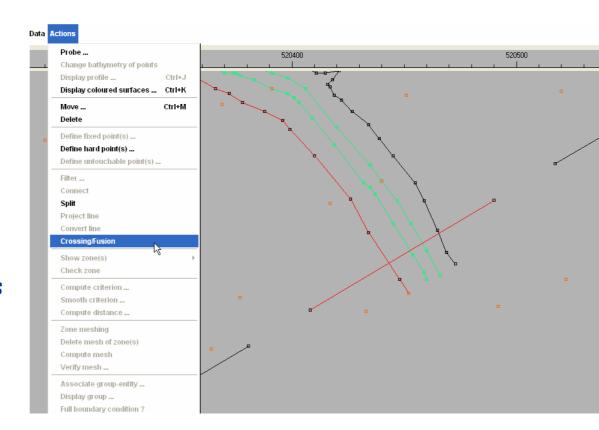M.M. Maubourguet - J. Chorda 20

Using Zones for river meshing

The aim is here to obtain a structured regular mesh along the river bed

A preliminary task is to create a pseudo polygon defining the future contour. In the *Geo lines* mode, select 2 polylines and in the menu *Action*, choose *Crossing/fusion*.

You must notice that a scission was made in the contour line. A point will be created at the intersect of the 2 lines.
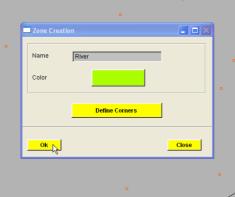
You repeat the procedure for the outlet.

M.M. Maubourguet - J. Chorda 21

Creating river zone

In the **Zone** mode, select the menu **Data**, *Create zone* then start by a left clic on the first corner point, then clic on the successive segments in order to mark the contour of the zone river.
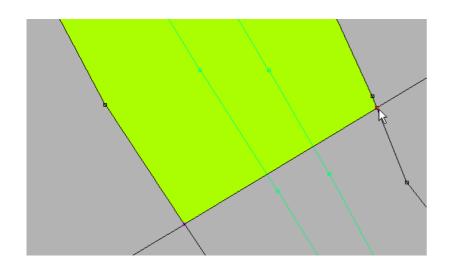
When done, clic on the middle button in order to validate the contour definition. You still have to define the corners
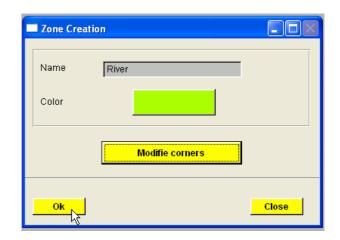
Creating river zone

Successively clic on the corners of the inlet and the outlet

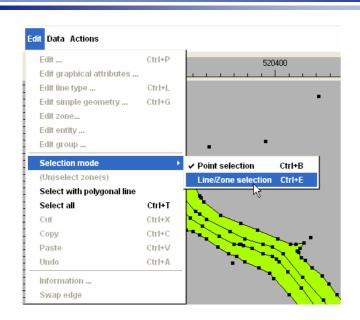When done, clic on the middle button in order to validate the contour definition. Clic OK
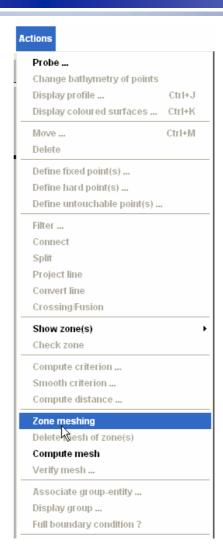
(and don't forget to save ...)
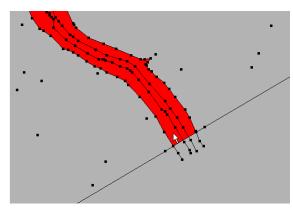
Meshing the river zone

In order to select zone, in the **Zone** mode, select the menu **Edit**, *Selection mode : Line/zone selection.*
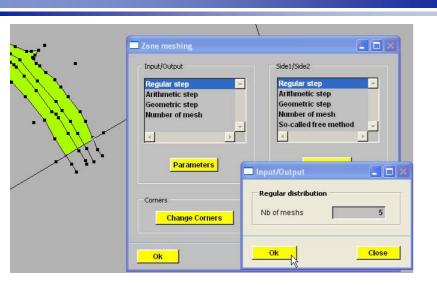
You can now select the whole zone and in the **Mesh** mode, menu **Actions** select *Zone Meshing* .
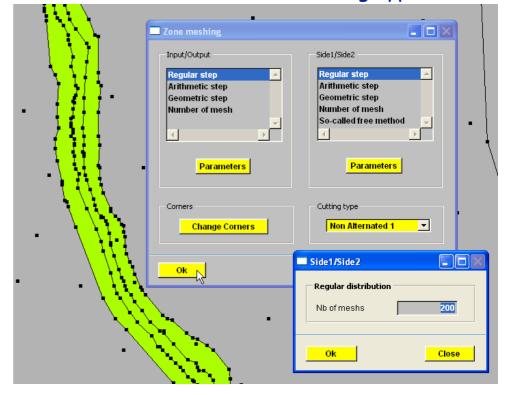
Meshing the river zone

For the Input/Output, we choose a default Regular step with the following parameters : 5 meshes across the river bed.
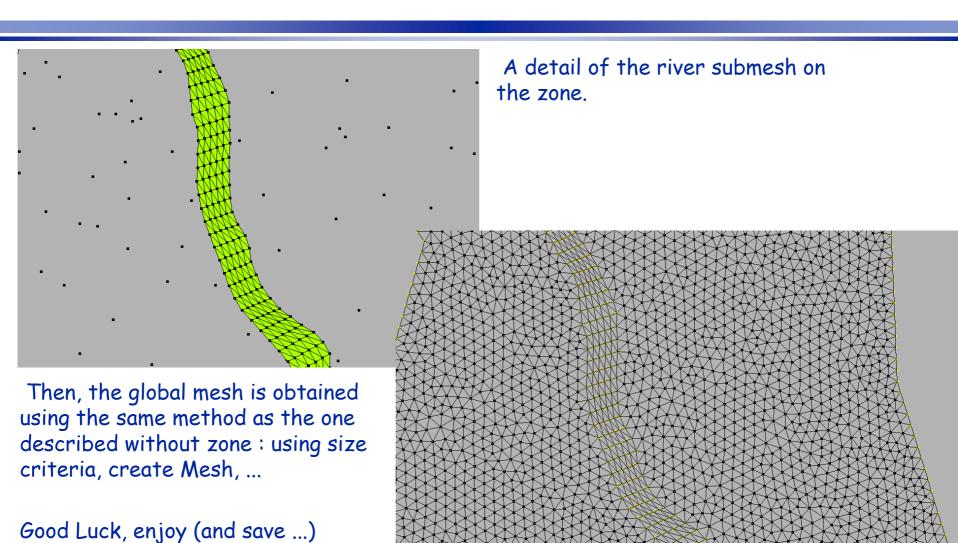
For the Side1 /Side2, we choose also a default Regular step with the following parameters : 200 meshes along the river bed with *Non Alternated 1* cutting type

River zone mesh

A detail of the river submesh on the zone.

Then, the global mesh is obtained using the same method as the one described without zone : using size criteria, create Mesh, ...

Good Luck, enjoy (and save ...)