Appendices
Appendix A: The first trial to get the new mesh.

Here I want to capture on of the problems I have had during the long the way I followed to achieve the correct implementation. This is an example of the beginning when I tried to build the new mesh taking a wrong way, it cost me a month of work and I think it is also a part of the thesis, because to implement the model, the trial-error mode is what I have practiced every day. What I did is:

1.1 Define the hardlines (islands) creating a New closed line (the eleventh command on the second bar at the top which seems a closed poligon),
1.2 Then do 3 clicks separately in the 2D view window to draw a closed poligon like in the image and return to click on the same command before,
1.3 Just after, the Query window is opened and here must be put the name of the new file, in this case-> england or ireland and then do the steps again for the other island.

Three first sections of the Second step
1.4 Right click on the poligon created and cut it

1.5 Right click on the England’s profile and copy it and Paste it with the right click on the England file, then put this file on HardLines file. Do the same with the Ireland island
Now is the turn to redefine the mesh where the study has to be more accurate, and this is through the Density.

2.1 Define the new density creating a New closed line (the eleventh command on the second bar at the top which seems a closed polygon),

2.2 Then do a polygon in the 2D view window which surrounds the interesting area clicking on the map the area (Belgian Coast) and return to click on the same command before,

2.3 Just after, the Query window is opened and here must be put the name of the new file and the new edge size. It Must done as many times as necessary on the river with new density files until it has an accurate mesh size to represent all the river boundary.

2.4 Once having all the densities defined, put them on the Density file as I did with Hardlines.
4. Finally it is ready to run and to create the new mesh. Double click on the newT3Mesh file, and click on the Run command. Then press OK and in few seconds a new file is created which is the new mesh file.

Finally I chose a size of 0,1m on the Belgian coast, and 0,02m on the river. Later I did a new more little density for the upstream part of the river, it was 0,01 m. But I decided these sizes after did several trials until I found the most accurate size for each part of the boundary. This is done once create the mesh and see that it does not create an appropriate mesh, then you must delete and modify densities, then put them back on the density file and then re-run the program.
Appendix B: The parallel version

Telemac2d and Tomawac are generally run on single-processor computers of the workstation type. When simulations call for high capacity computers but you have not taken one, then is useful to run the computations on multi-processors computers. A parallel version of those programs is available for use in this type of computer architecture.

A considerable amount of information on the use of the parallel version is given in the system installation documents. Initially, It is necessary a new library documentation, in order to supply to the program tools to detect the news orders, otherwise it won’t understand what you are demanding. It comes from a new version of TelamacV7, the folder can be found in the website: www.opentelemac.org.

Moreover a new command has to be used to run the model, taking into account the new configurations. It is (in Tomawac is the same changing the steering file in the command):

1) cd c:\Coupled_2004
2) runcode.py telemac2d -c win7telmpi -s steeringFile_June_2002.txt

Secondly the user must specify the number of processors used by means of the keyword PARALLEL PROCESSORS in the steering file. The keyword may have the following values:

0: Use of the classical version of TELEMAC-2D
1: Use of the parallel version of TELEMAC-2D with one processor
N: Use of the parallel version of TELEMAC-2D with the specified number of processors

Finally there are some changes inside the fortran file, because what we do with this partition of the computation is divide the whole domain in some subdomain, depending on the processors you want to use. So what we have to do is do the implementation of some commands, in order to tell to the program that it has to translate each point in the subdomain into the general domain, getting the same results as if it was computed with one processor.

At Bord tab:

Ln 1887: IF(BOUNDARY_COLOUR%I(K).EQ.POINTNUM(NE)) THEN
At Meteo tab:

- Ln 265: DO N=1,NPOIN
  VX=VAR1X(MESH%KNOLG%I(N))
  VY=VAR1Y(MESH%KNOLG%I(N))
  VAR1X(MESH%KNOLG%I(N))=SQRT(VX*VX+VY*VY)
  VAR1Y(MESH%KNOLG%I(N))=ATAN2(VY,VX)
ENDDO