HOW TO USE THE edgebased level set PROBLEMTYPE

ANTONIA LARESE DE TETTO

antoldt@cimne.upc.edu

RICCARDO ROSSI

rrossi@cimne.upc.edu

PREREQUISITES

- In order to be able to use the EDGEBASED problem type is necessary to have already installed:
  - KRATOS (click here if not)
  - GID (click here if not)

If your system is ready, CLICK HERE

PREREQUISITES: HOW TO INSTALL KRATOS

- You can download the kratos release for windows from http://kratos.cimne.upc.es/kratoswiki/index.php/Download

Windows

Current Release

File	Release	Release Date	Install Guide
Download binaries	1.1.0	October 7th, 2009	Windows Installation Guide

Older Releases

File	Release	Release Date	Install Guide
Download source	1.0.0	May 9th, 2008	Windows Installation Guide

Developers Source

http://www.rapidsvn.org/

Download: http://www.rapidsvn.org/download/release/0.9.6/

Using rapidSVN the kratos source files can be downloaded by taking the following steps:

- BACK TO PREREQUISITES

PREREQUISITES: HOW TO INSTALL GID

- You can download GiD from http://www.gidhome.com/download/

- BACK TO PREREQUISITES

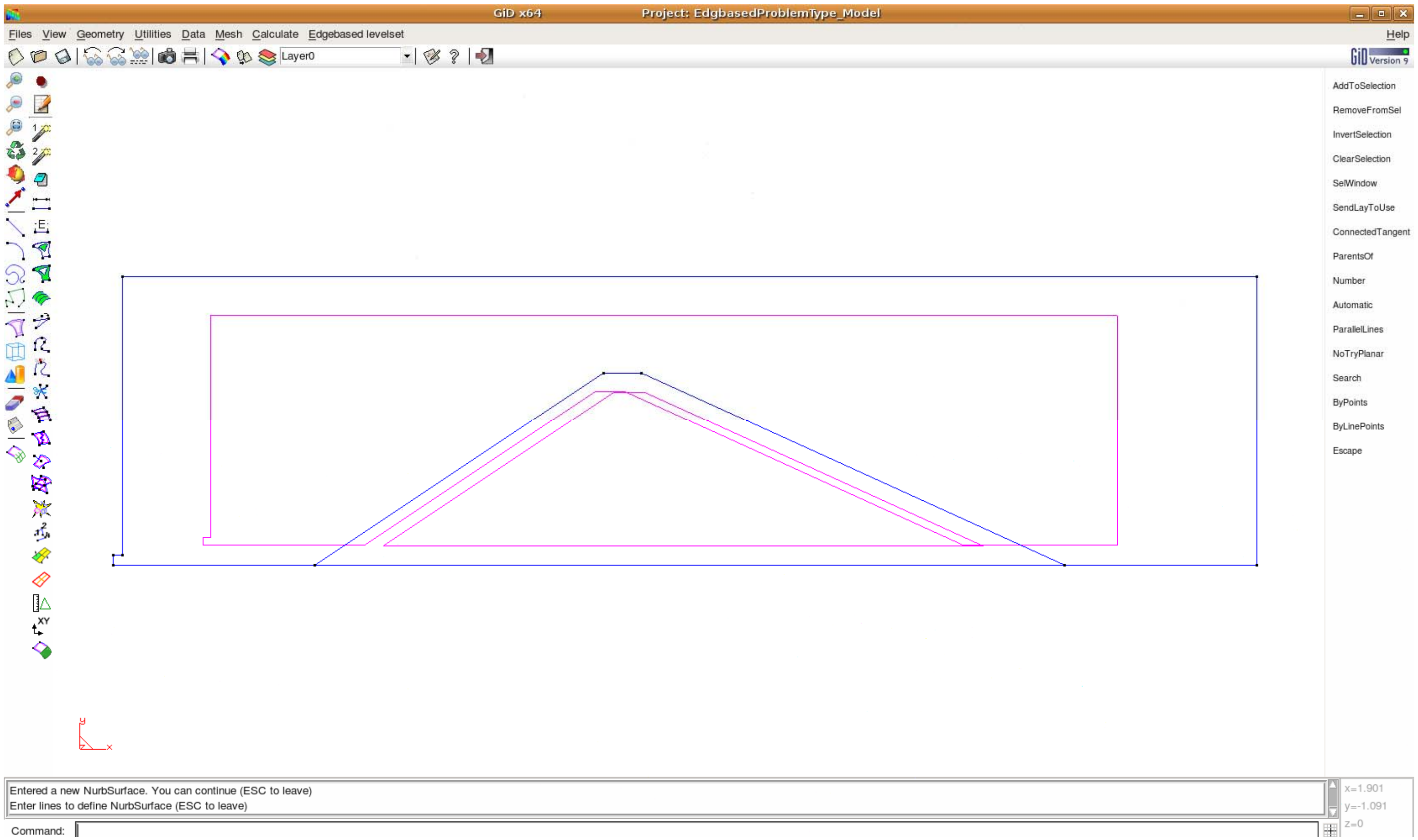
PERSONAL BASIC KNOWLEDGE

- In order to use the EDGEBASED problemtype the knowledge of the use of GiD is required.

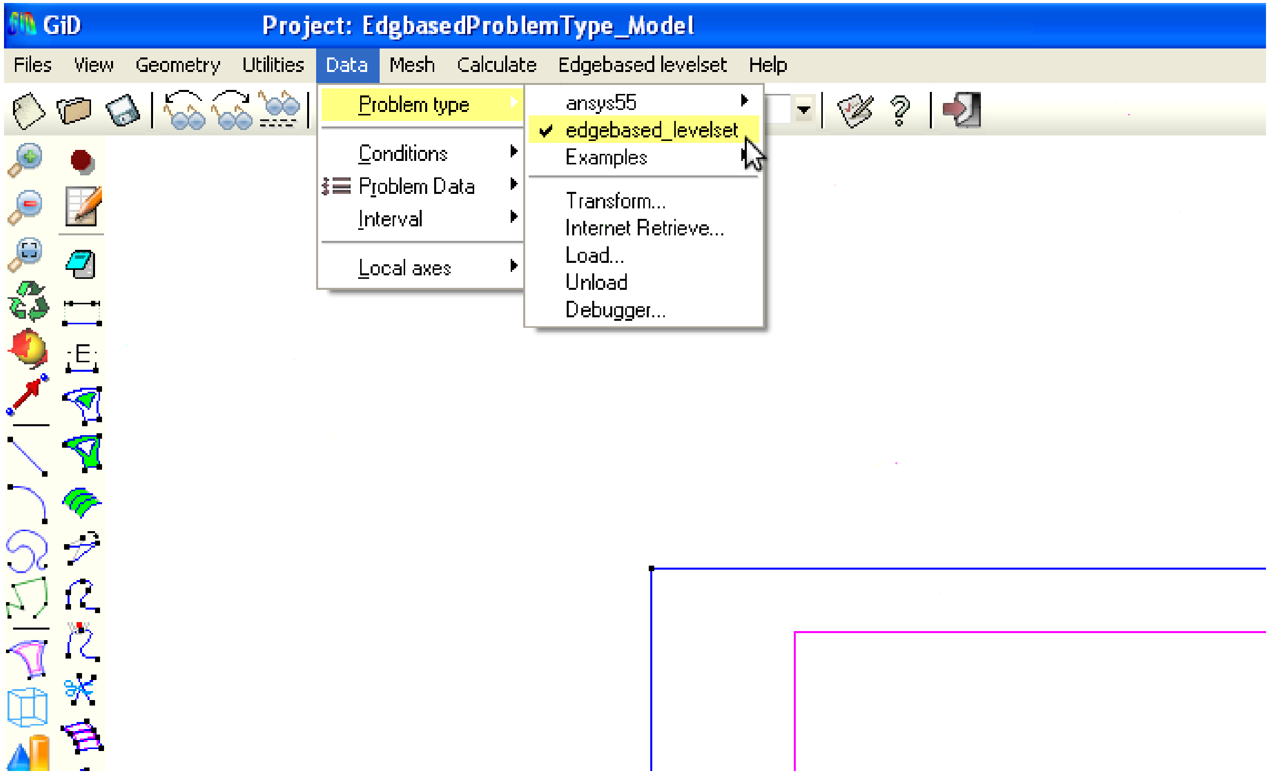
- Be sure to be able to do all the GiD tutorials that you can find at http://www.gidhome.com/support_team/su05.html

THE EDGEBASED LEVEL SET PROBLEM TYPE

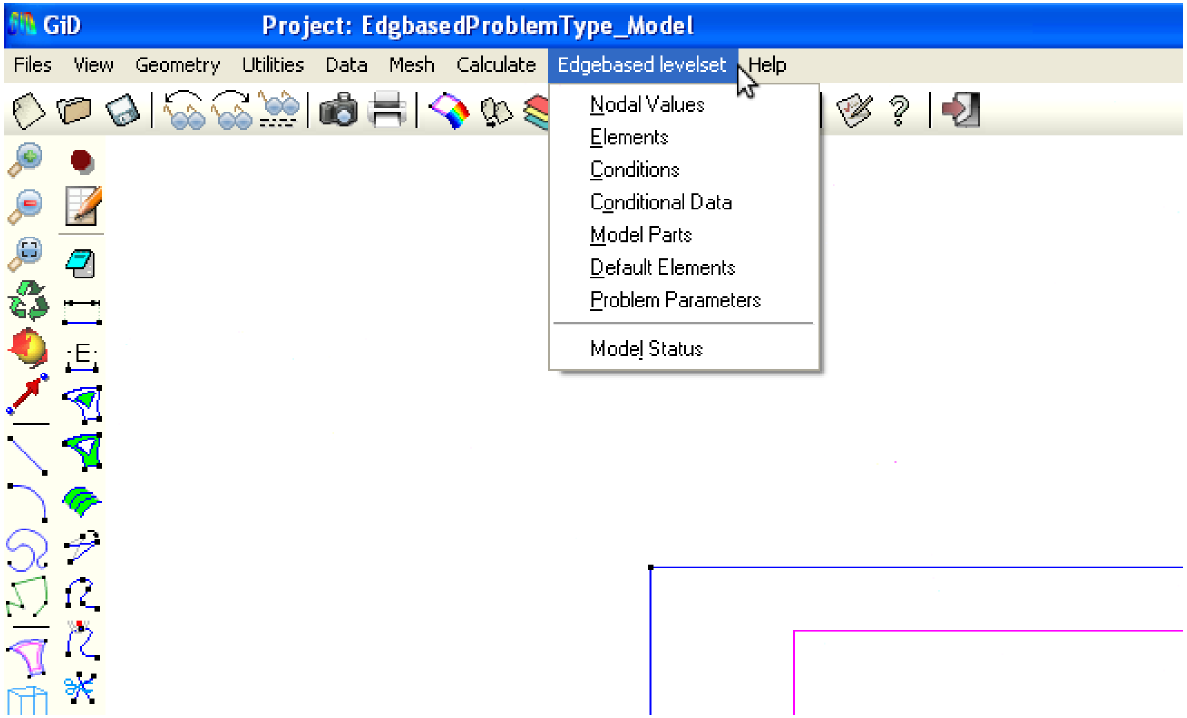
CREATION OF THE GEOMETRY OF THE MODEL



CHIOCE OF THE GID PROBLEM TYPE



EdgeBased level set toolbar menu should appear



BOUNDARY CONDITIONS

Boundaries in 2D are the control domain perimetral lines (in 3D the perimetral surfaces).

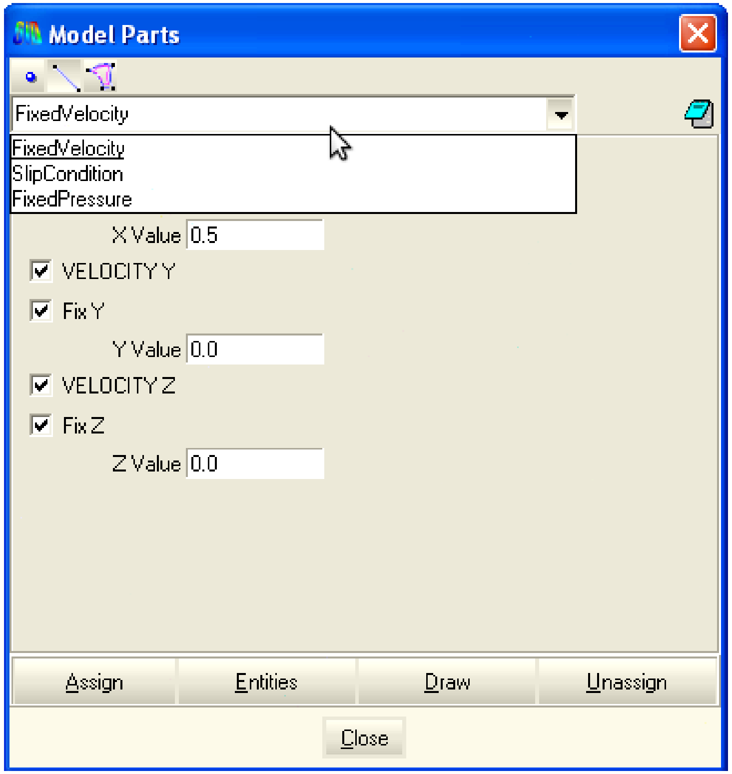
ALL of them should have a Boundary Condition assigned.

3 are the possible boundary conditions:

-FIXED VELOCITY to be assigned to the ENTRANCE OF WATER line in 2D (surface in 3D);

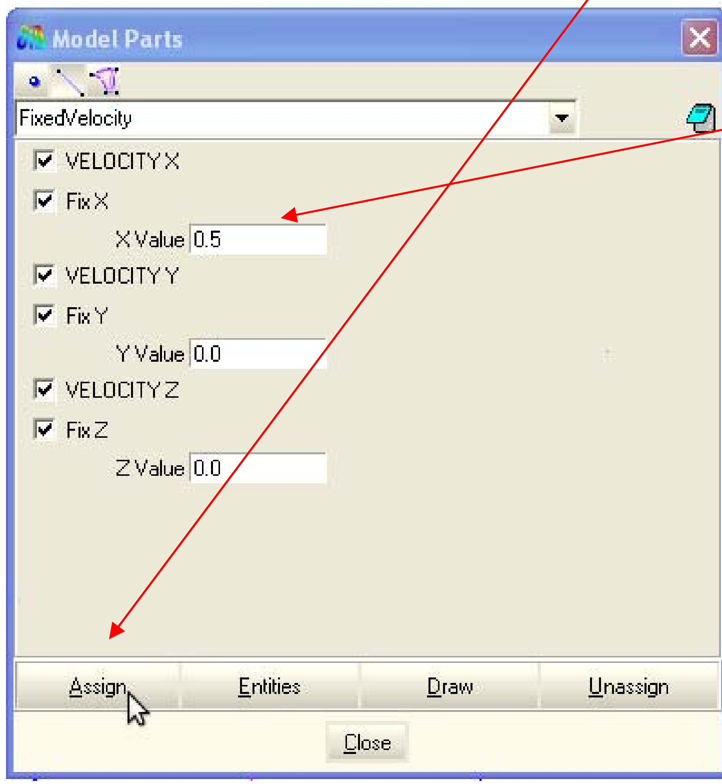
-SLIP CONDITION to be assigned to all the WALL lines in 2D (surfaces in 3D);

-FIXED PRESSURE to be assigned to the EXIT of water line in 2D (surface in 3D)



BOUNDARY CONDITIONS

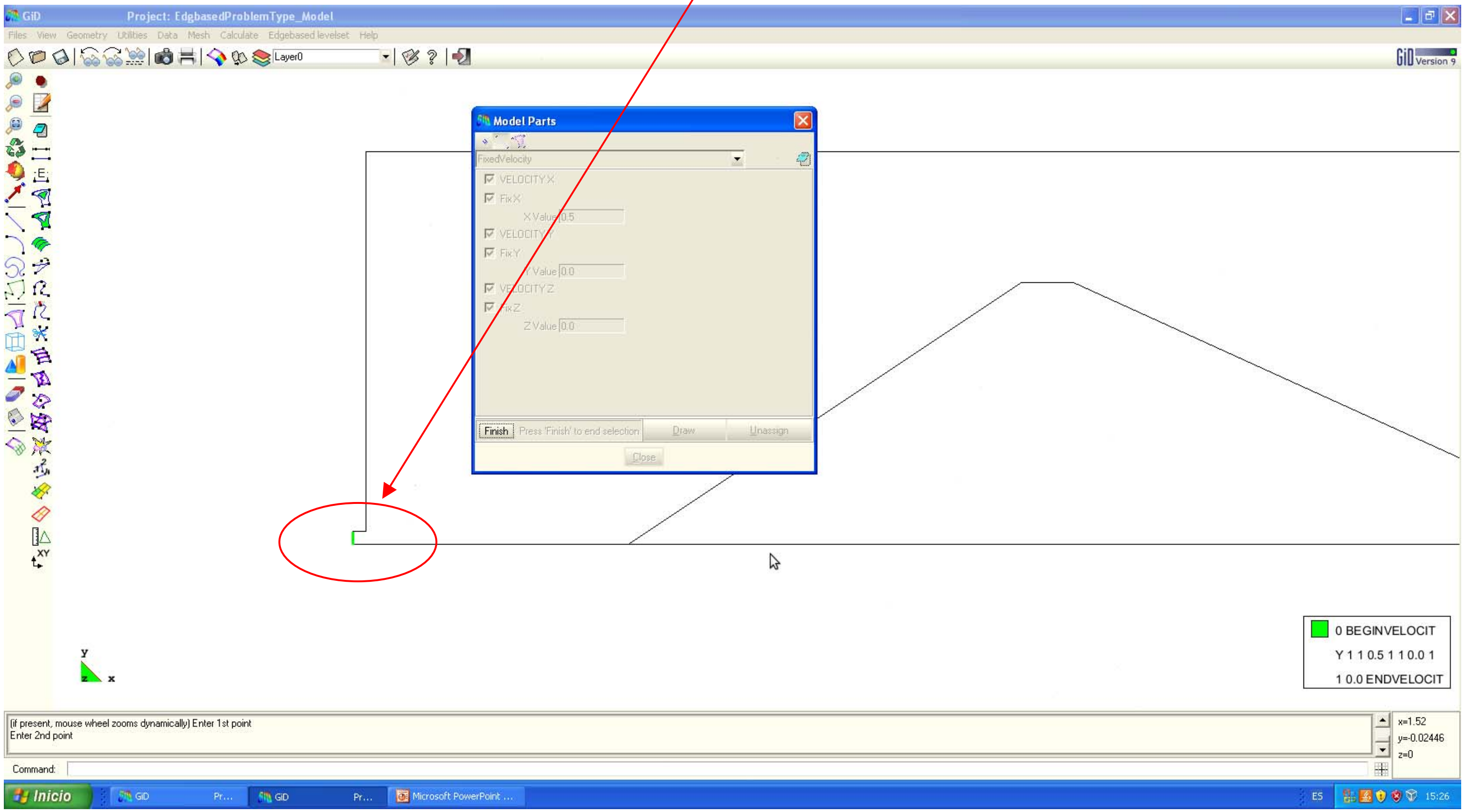
IMPOSED VELOCITY to be assigned to the ENTRANCE OF WATER line in 2D (surface in 3D);



Choose the value of inlet velocity and the direction of the same

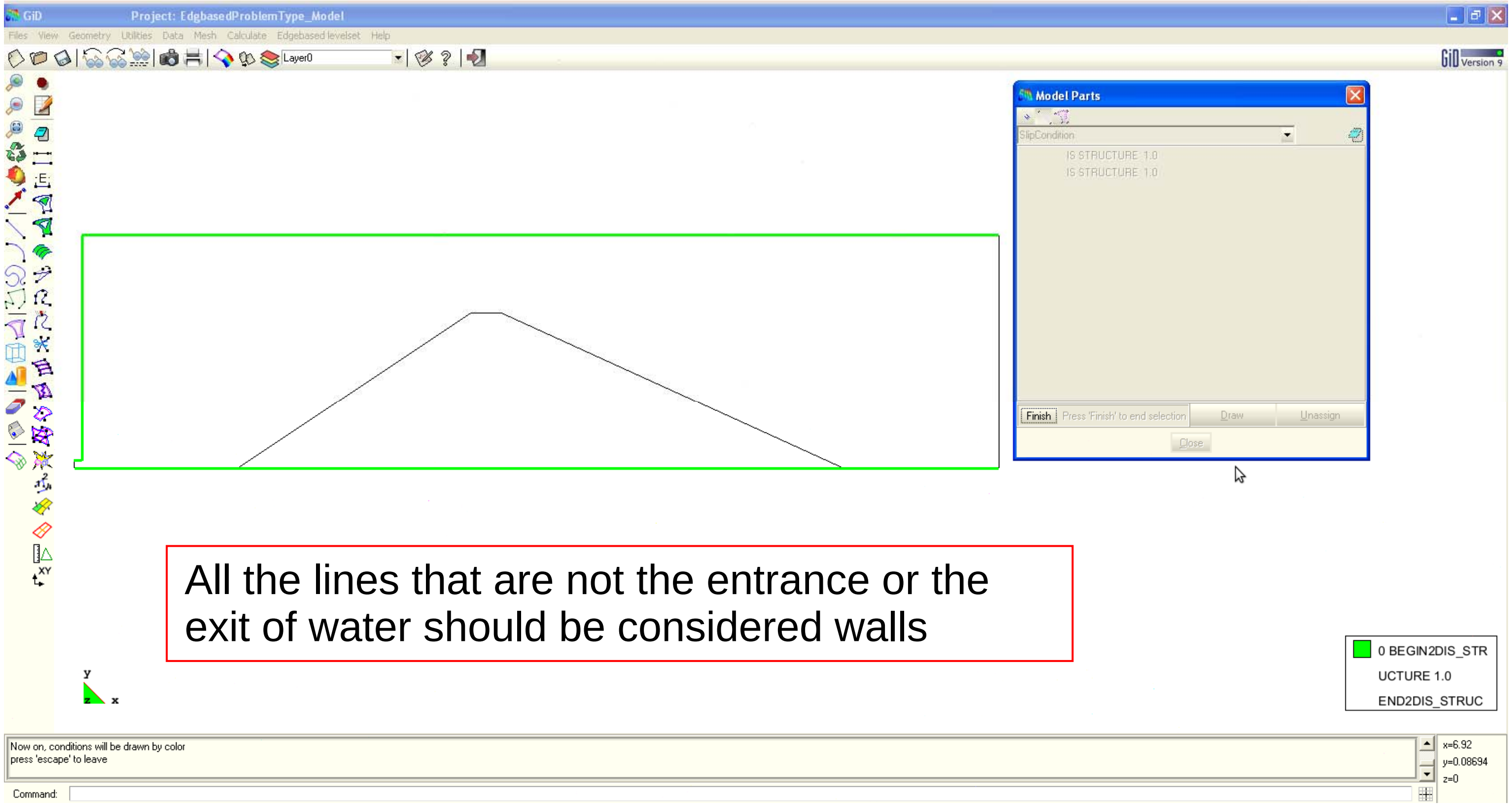
BOUNDARY CONDITIONS

IMPOSED VELOCITY to be assigned to the <u>ENTRANCE OF WATER</u> line in 2D (surface in 3D);



BOUNDARY CONDITIONS

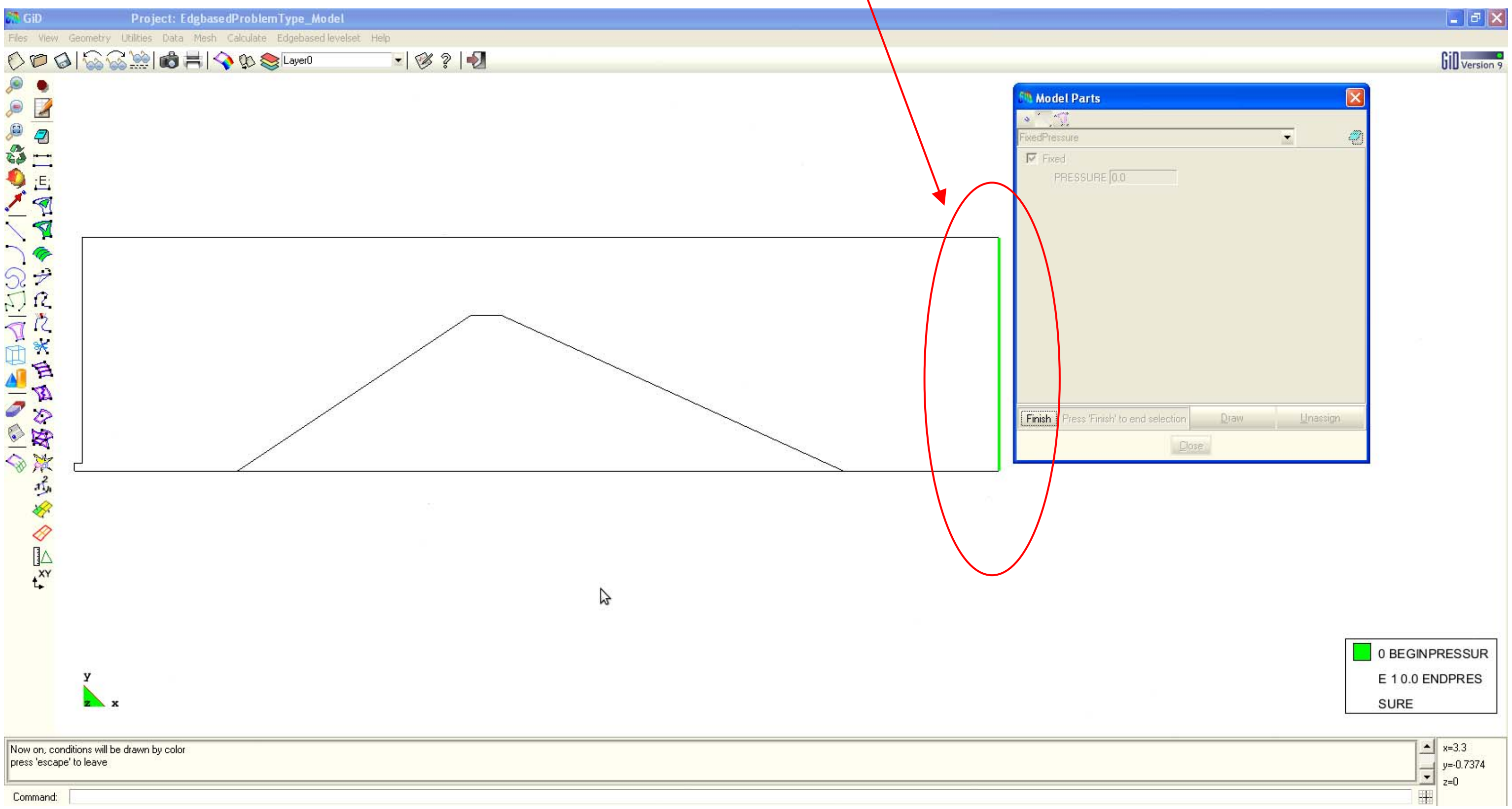
SLIP BOUNDARY CONDITION to be assigned to all the WALL lines in 2D (surfaces in 3D);



All the lines that are not the entrance or the exit of water should be considered walls

BOUNDARY CONDITIONS

FIXED PRESSURE to be assigned to the EXIT of water line in 2D (surface in 3D)



MATERIAL CONDITIONS

1) We have now to indentify which portion of the control domain has porosity different to 1.

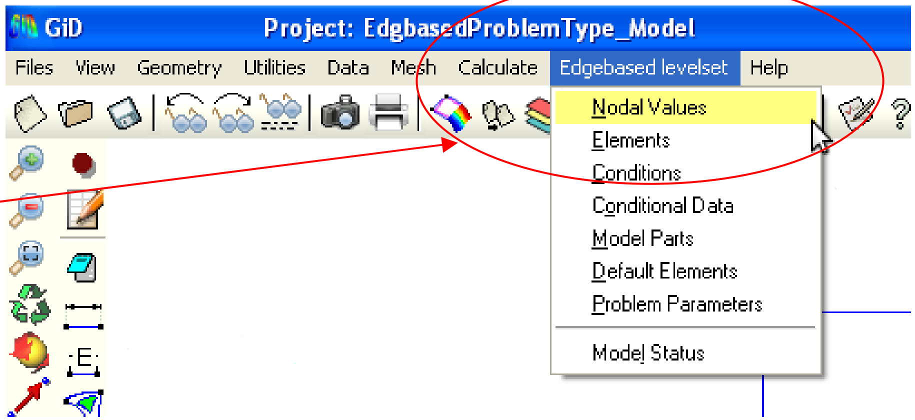
That is where the porous medium is and which is its porosity.

By default the program will assign porosity = 1 to all the nodes if no different porosity is assigned.

2) We have to identify the fluid.

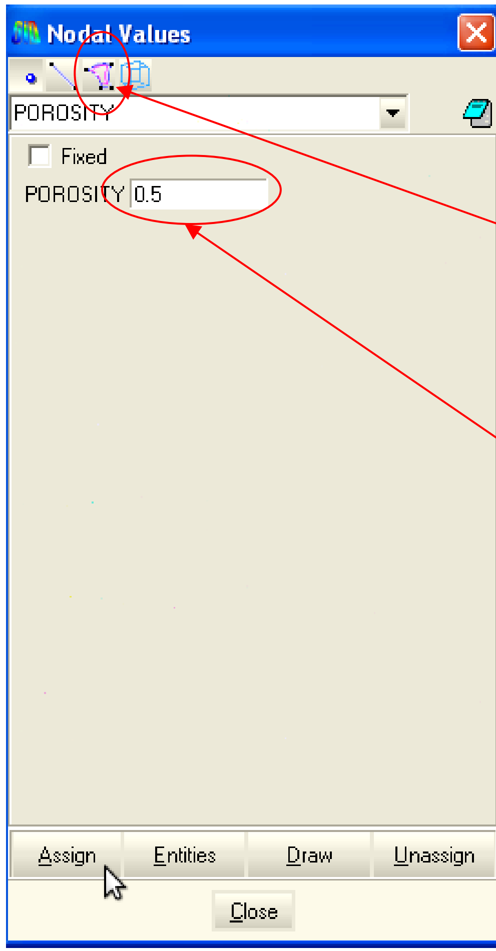
We open the menu

NODAL VALUES



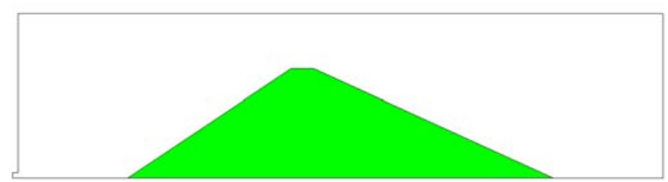
MATERIAL CONDITIONS

1) We have now to indentify which portion of the control domain has porosity different than 1.



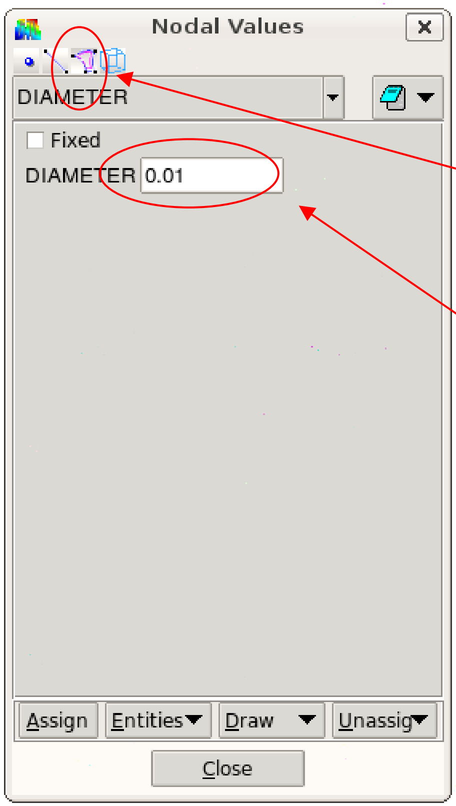
We assign this condition to SURFACES in 2D and VOLUMES in 3D

We insert the porosity value



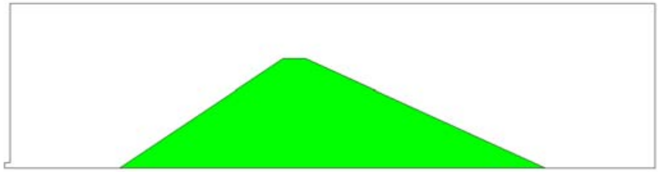
MATERIAL CONDITIONS

2) We have to set the diameter D50 to be assigned to the porous region.



We assign this condition to SURFACES in 2D and VOLUMES in 3D

We insert the diameter value



MATERIAL CONDITIONS

3) We have to identify the fluid.

Fluid region is identificated by NEGATIVE VALUES OF THE DISTANCE FUNCTION.

CASE A

at t= 0 we don't have any fluid in the domain

Assign DISTANCE = -1 to the ENTRANCE OF WATER line in 2D (surface in 3D)

CASE B

at t=0 we already have a fluid region

Assign DISTANCE = -1 to the FLUID area in 2D (or volume in 3D)

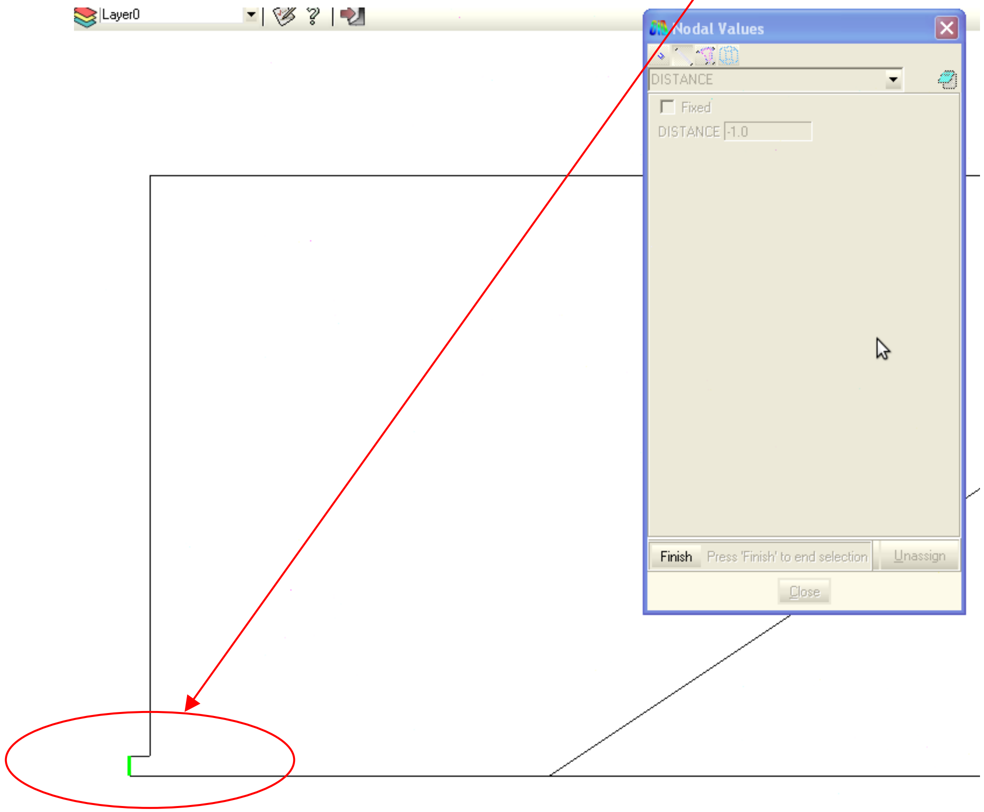
MATERIAL CONDITIONS

3) We have to identify the fluid.

<u>CASE A</u>

at t= 0 we don’t have any fluid in the domain

Assign DISTANCE = -1 to the ENTRANCE OF WATER line in 2D (surface in 3D)



MATERIAL CONDITIONS

ANY MODIFICATION

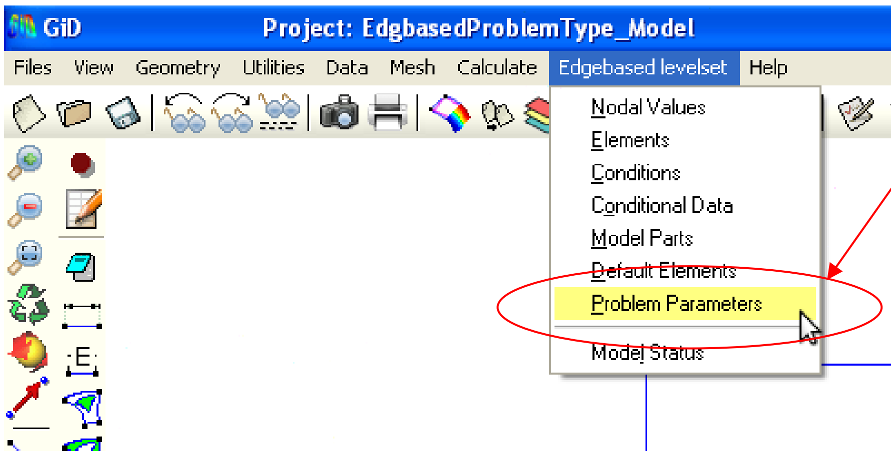
OF THE NODAL VALUES NEED TO

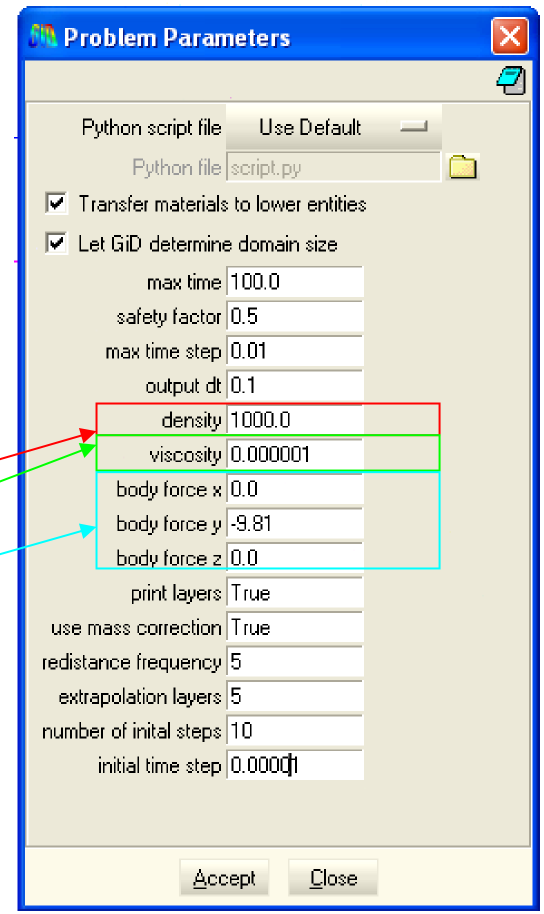
REGENERATE THE MESH

in order to assign the modified value to the nodes

MATERIAL CONDITIONS

The rest of the material properties have to be inserted as <u>problem parameters</u> (they are automatically assign to the nodes without the need to remesh when a modification is inserted)





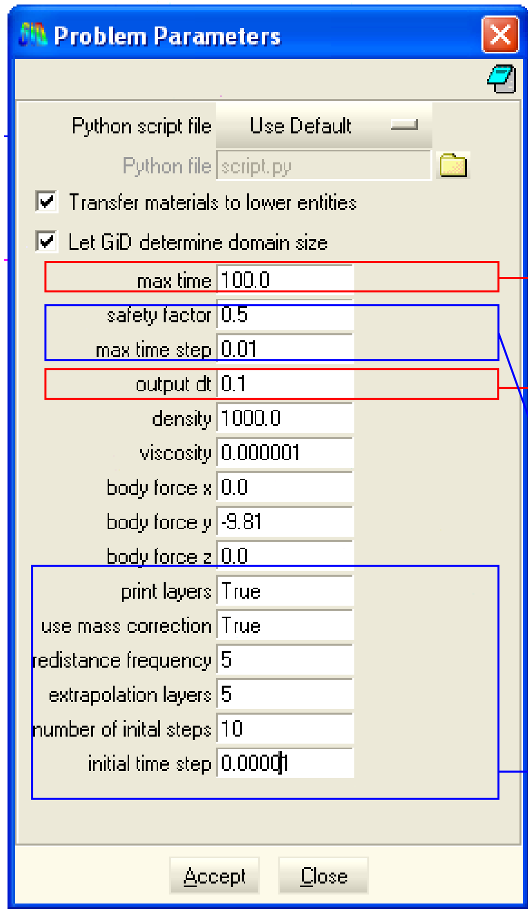
They are:

WATER DENSITY

WATER VISCOSITY

GRAVITY

PROBLEM PARAMETERS



<u>Duration of the simulation</u>

<u>Delta t to print results.</u>

Often we don't want to print all the time steps (output dt = 0.0) but for example each 0.1s of simulation (output dt = 0.1)

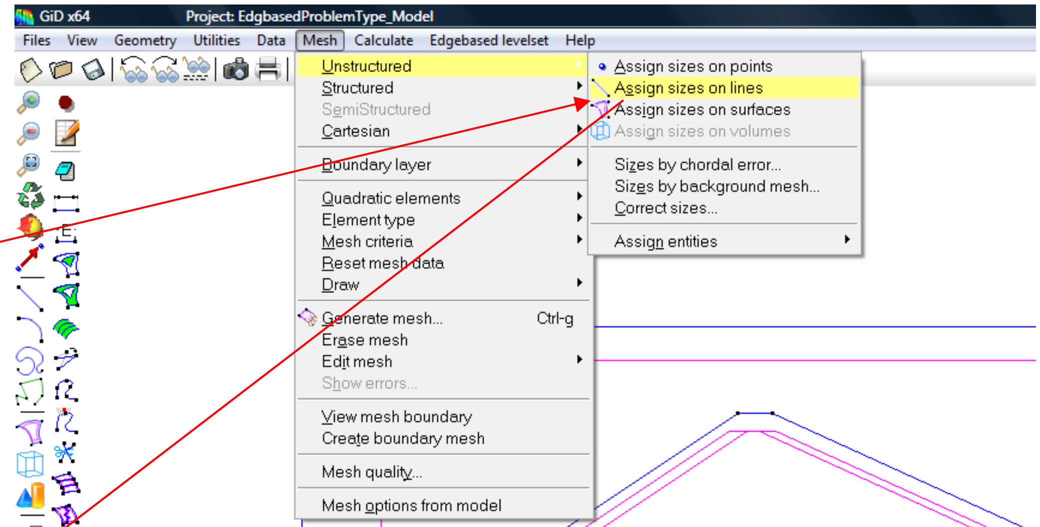
Check that the other values coincide with those that appears in this photo.

Push the ACCEPT botton at the end

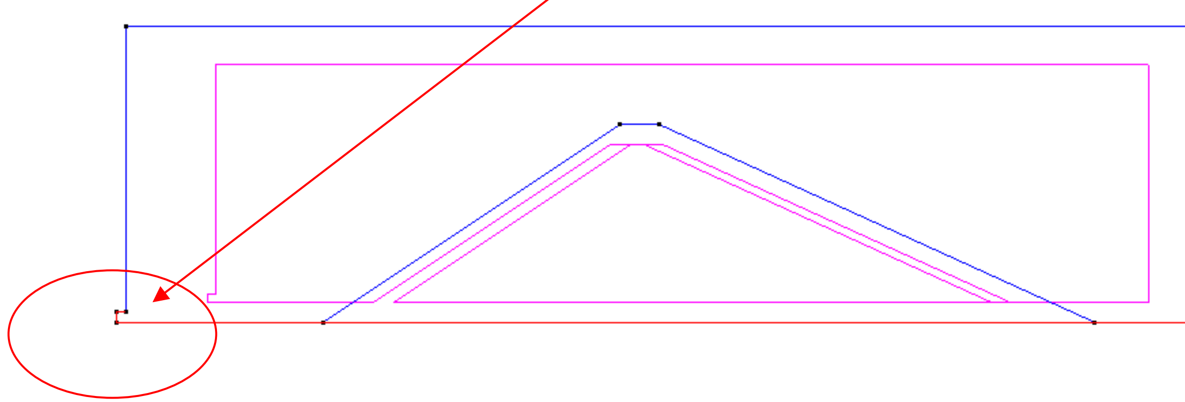
GENERATE THE MESH

Pay attention that the entrance of water line in 2D (or surface in 3D) has to be divided in more than 2 elements. Otherwise numerical problems can occurs.

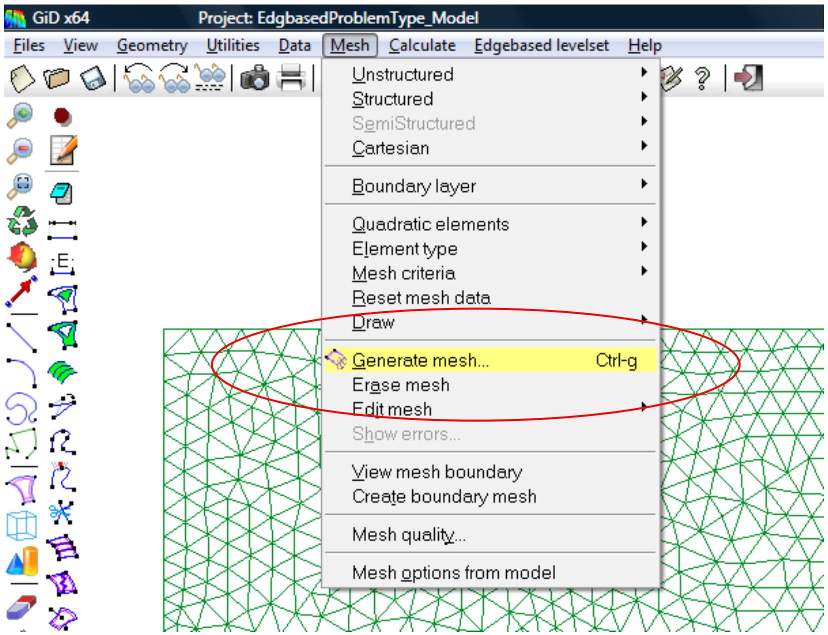
Better to refine a little more the bottom line in 2D

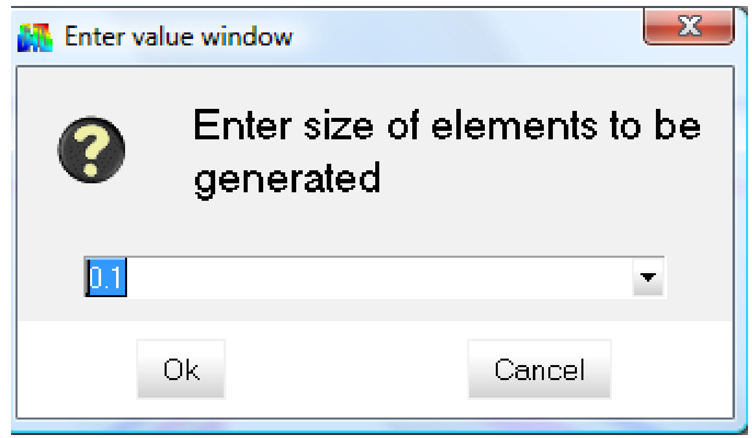


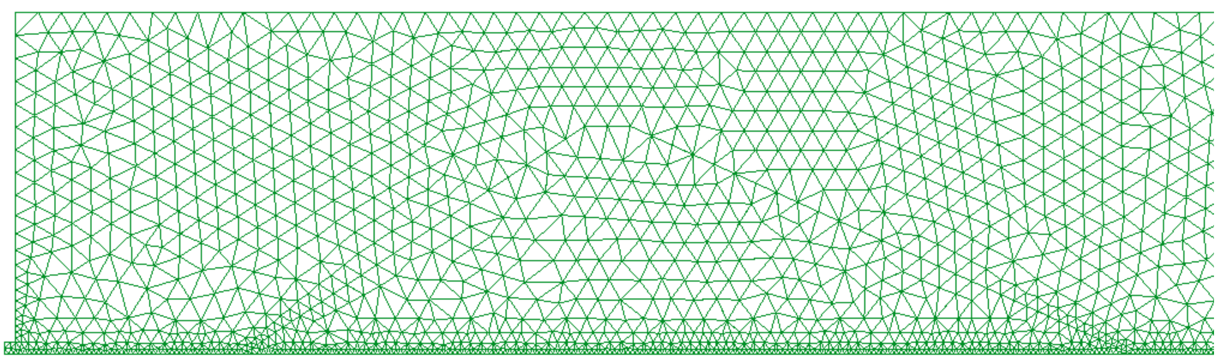
Use the unstructured mesh (assign mesh size to line or surface)



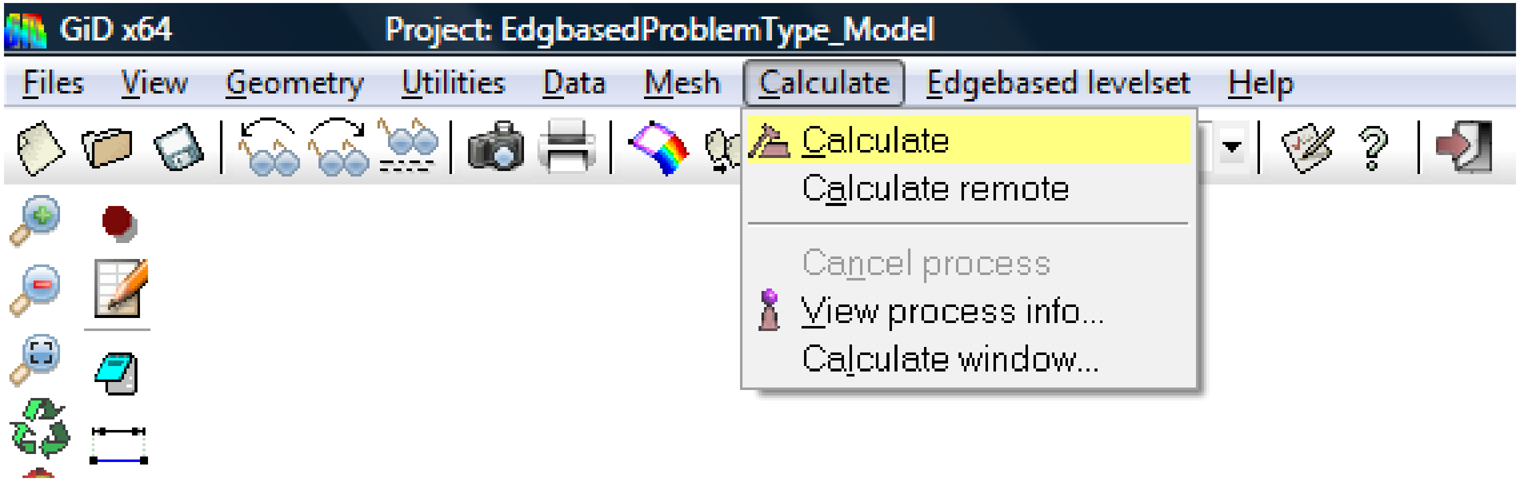
GENERATE THE MESH







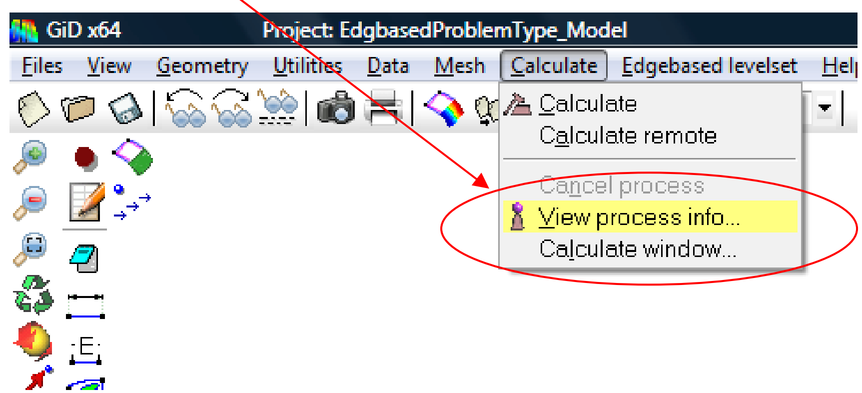
Run calculation



Control the process

To controle the output script file of your calculation you can eigther

Go to the VIEW PROCESS INFO window



Or open the file <ProblemName>.info that you can find inside your <ProblemName>.gid

For any problem please contact with

antoldt@cimne.upc.edu