

HOW TO USE THE **edgebased** **level set** PROBLEMTYPE

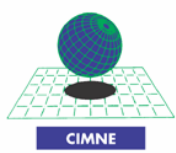
ANTONIA LARESE DE TETTO

antoldt@cimne.upc.edu

RICCARDO ROSSI

rossi@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

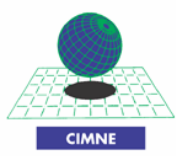
<http://www.rapidsvn.org/>

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](#)



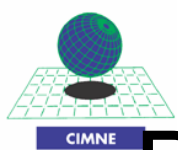


PREREQUISITES: HOW TO INSTALL GiD

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

- [BACK TO PREREQUISITES](#)



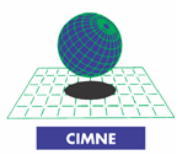


PERSONAL BASIC KNOWLEDGE

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

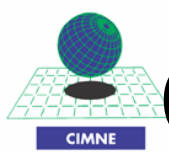
- Be sure to be able to do all the GiD tutorials that you can find at http://www.gidhome.com/support_team/su05.html





THE EDGEBASED LEVEL SET PROBLEM TYPE

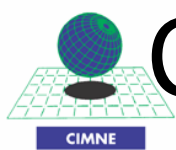




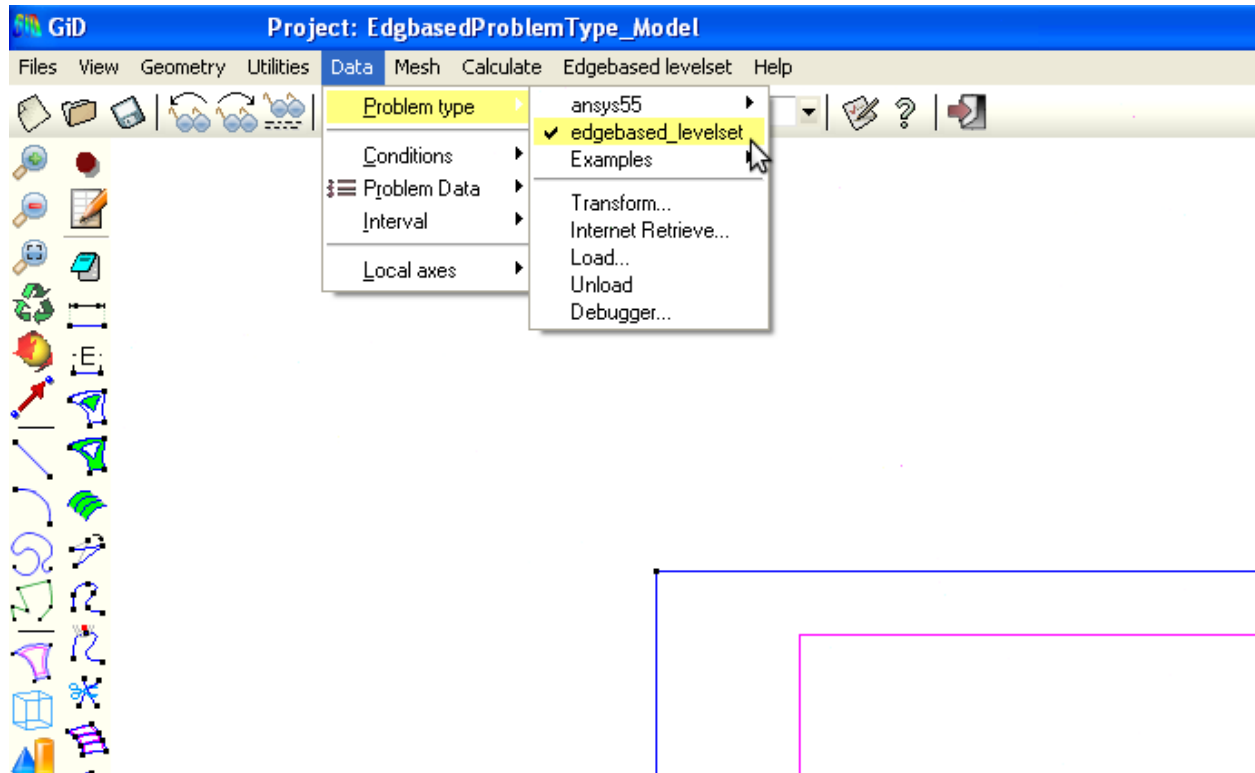
CREATION OF THE GEOMETRY OF THE MODEL

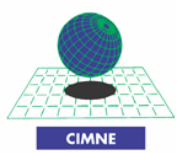
The screenshot displays the GID software interface. The title bar shows "GID x64" and "Project: EdgbasedProblemtyp Model". The menu bar includes "Files", "View", "Geometry", "Utilities", "Data", "Mesh", "Calculate", "Edgebased levelset", and "Help". The toolbar contains various icons for file operations, geometry creation, and editing. The main workspace shows a 2D geometry model of a trapezoidal structure with a gabled roof. The geometry is defined by several lines: a blue outer boundary, a pink inner boundary, and a black line for the roof. A red coordinate system is visible in the bottom left corner of the workspace. The right sidebar contains a list of actions: "AddToSelection", "RemoveFromSel", "InvertSelection", "ClearSelection", "SelWindow", "SendLayToUse", "ConnectedTangent", "ParentsOf", "Number", "Automatic", "ParallelLines", "NoTryPlanar", "Search", "ByPoints", "ByLinePoints", and "Escape". The command line at the bottom shows the following text: "Entered a new NurbSurface. You can continue (ESC to leave)", "Enter lines to define NurbSurface (ESC to leave)", and "Command:". The status bar on the right shows coordinates: "x=1.901", "y=-1.091", and "z=0".



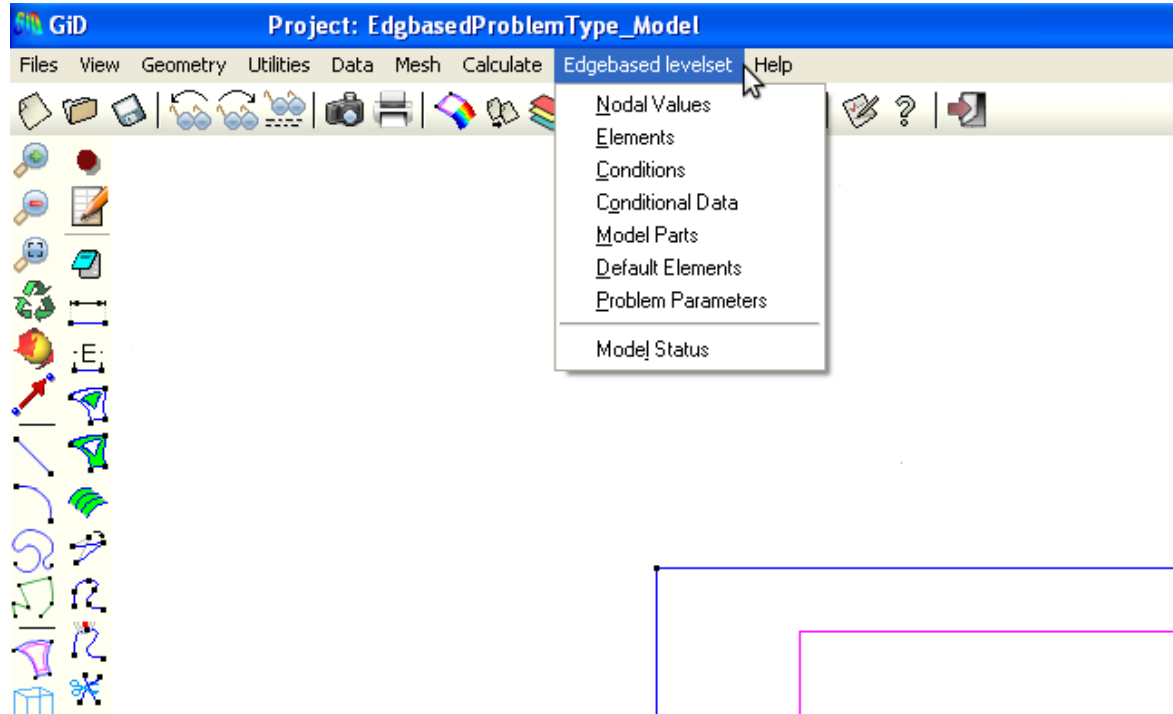


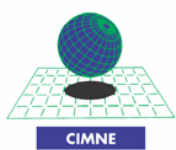
CHOICE OF THE GID PROBLEM TYPE





EdgeBased level set **toolbar menu** should appear





BOUNDARY CONDITIONS

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

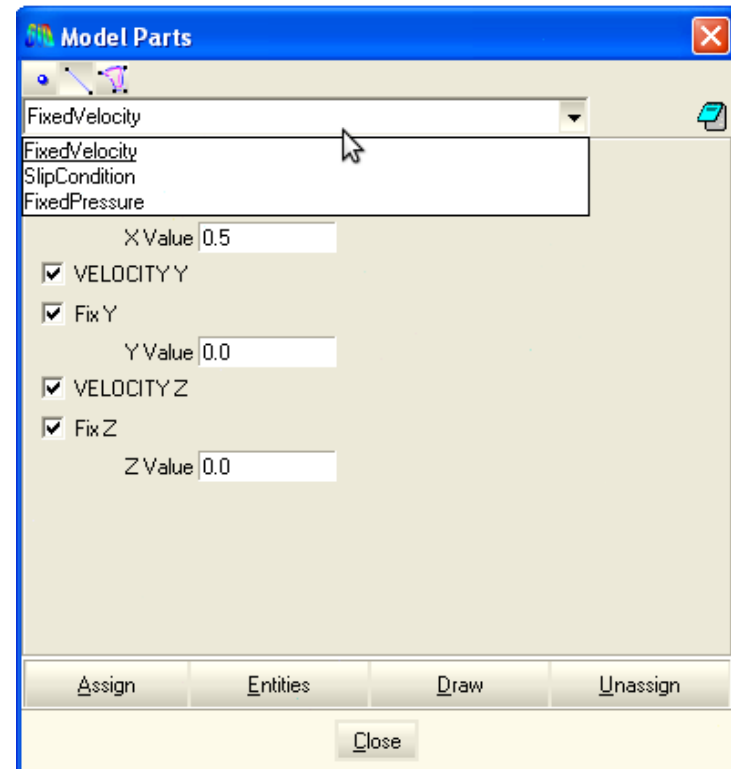
ALL of them should have a Boundary Condition assigned.

3 are the possible boundary conditions:

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

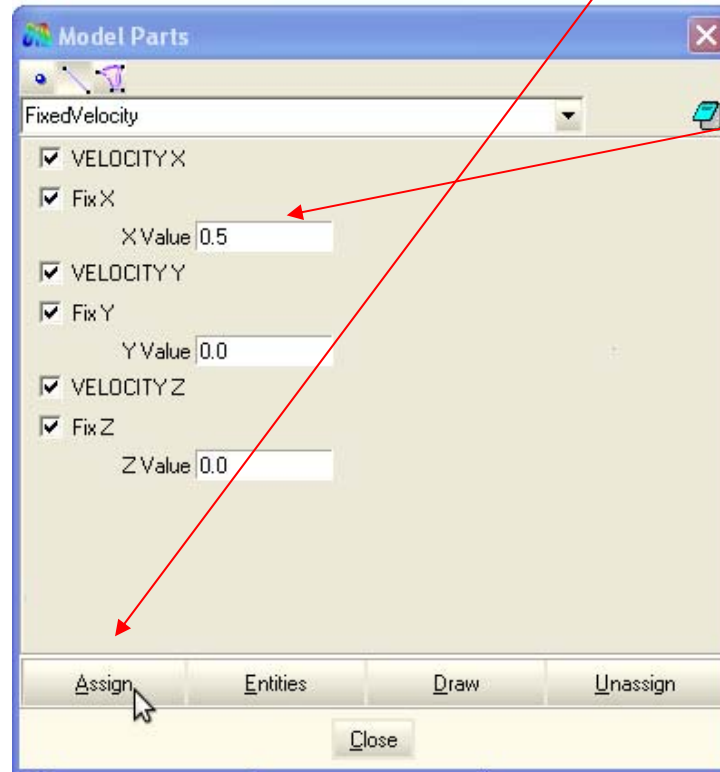
-SLIP CONDITION to be assigned to all the WALL lines in 2D (surfaces in 3D);

-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);

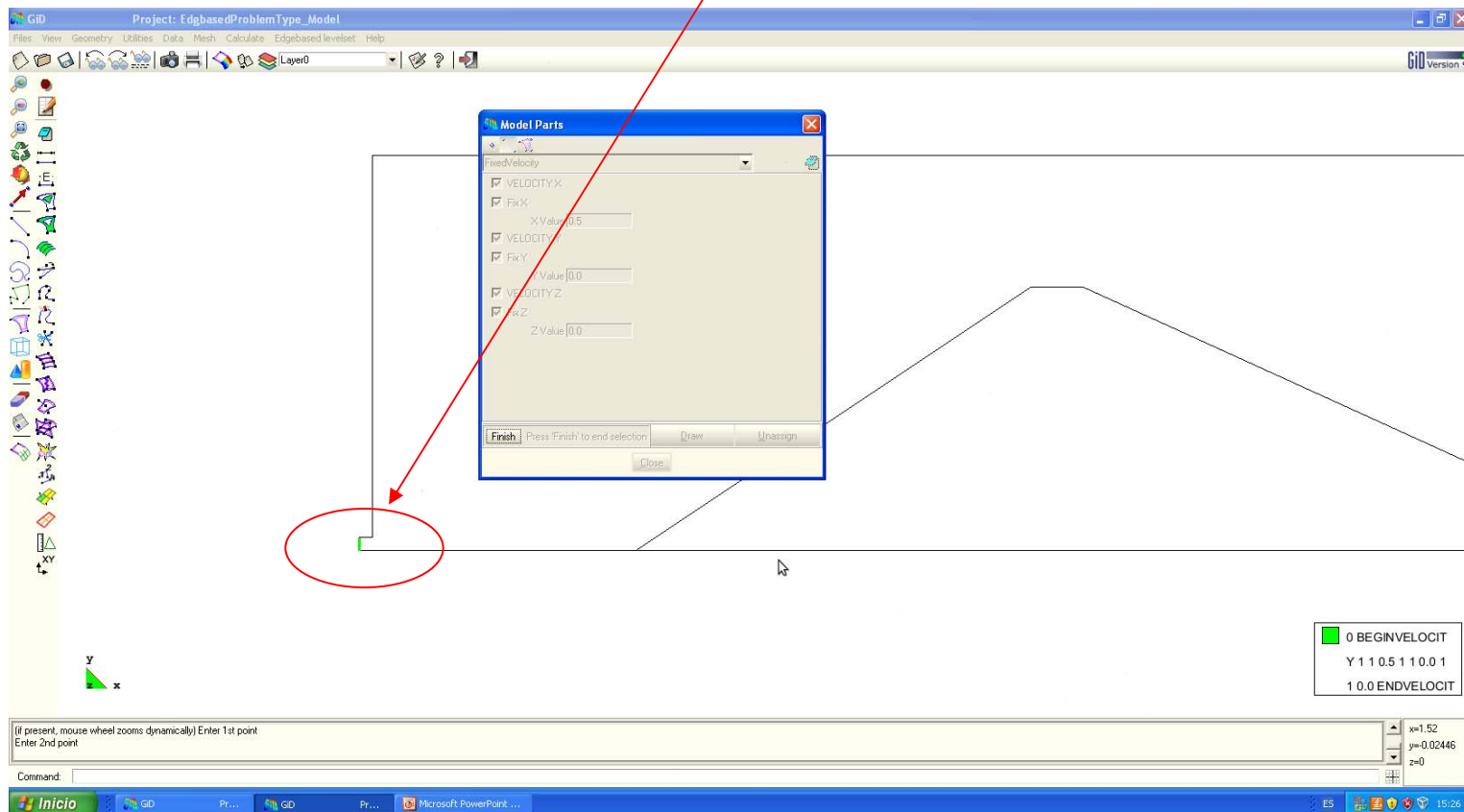


Choose the value of inlet velocity and the direction of the same



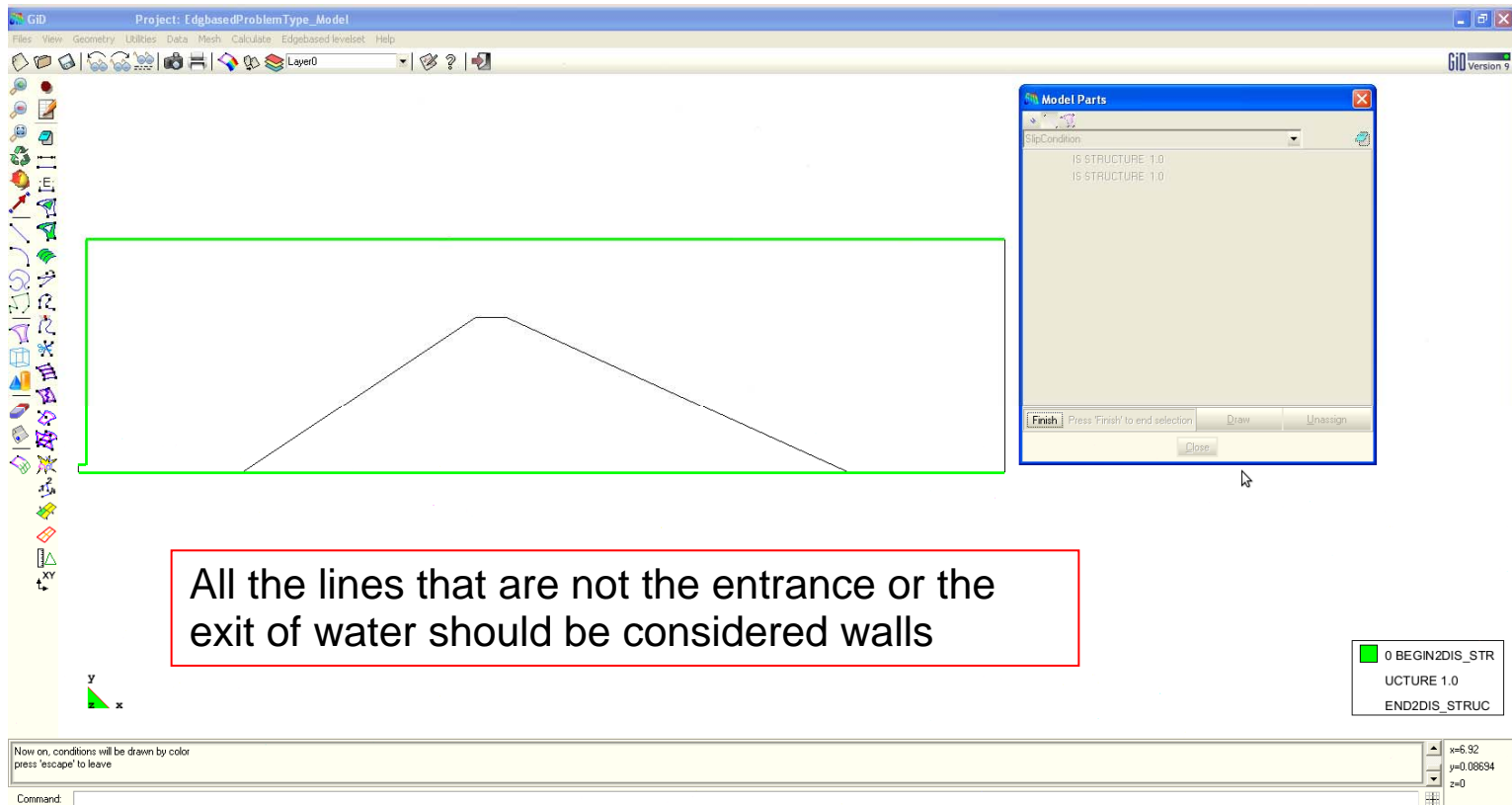
BOUNDARY CONDITIONS

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



BOUNDARY CONDITIONS

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



Project: EdgebasedProblemType_Model

Files View Geometry Utilities Data Mesh Calculate Edgebased levelset Help

Layer0

GID Version 9

Model Parts

SlipCondition

IS STRUCTURE 1.0
IS STRUCTURE 1.0

Finish Press 'Finish' to end selection Draw Unassign

Close

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

0 BEGIN2DIS_STR UCTURE 1.0 END2DIS_STRUC

Now on, conditions will be drawn by color
press 'escape' to leave

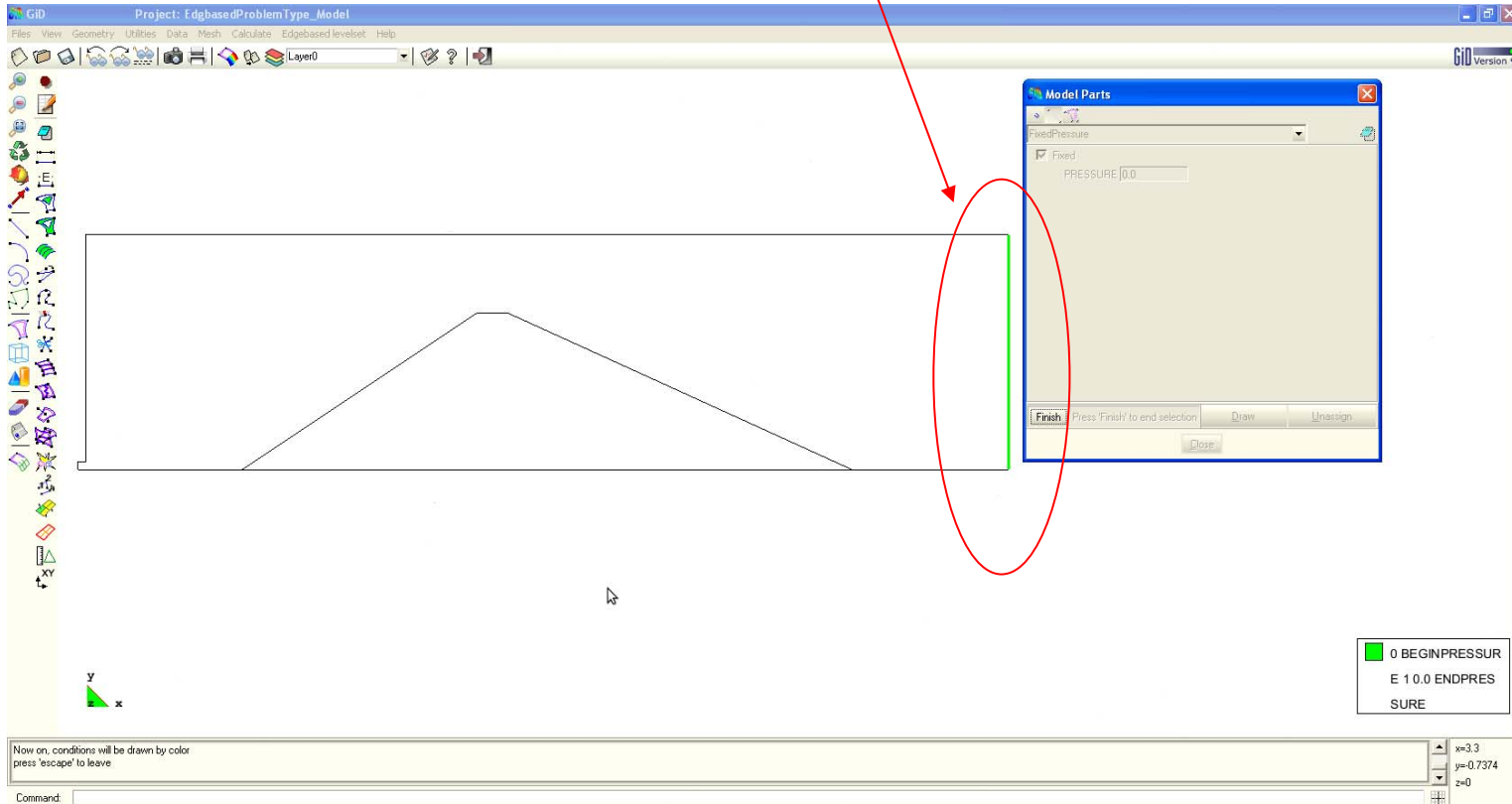
Command:

x=6.92
y=0.08634
z=0



BOUNDARY CONDITIONS

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



The screenshot shows the GiD software interface. The main window displays a 2D domain with a triangular obstacle. A red arrow points from the text above to the right boundary of the domain, which is highlighted with a green line. A 'Model Parts' dialog box is open, showing the 'FixedPressure' boundary condition applied to the selected boundary. The 'Fixed' checkbox is checked, and the 'PRESSURE' value is set to 0.0. The 'Finish' button is highlighted. A status bar at the bottom right shows the coordinates of the selected boundary: x=3.3, y=0.7374, z=0.

Project: EdgbasedProblemType_Model
GiD Version 9

Model Parts

FixedPressure

Fixed

PRESSURE 0.0

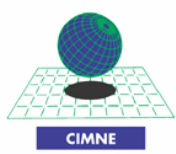
Finish Press 'Finish' to end selection Draw Unassign Close

0 BEGINPRESSUR
E 1 0.0 ENDPRES
SURE

Now on, conditions will be drawn by color
press 'escape' to leave

Command:

x=3.3
y=0.7374
z=0



MATERIAL CONDITIONS

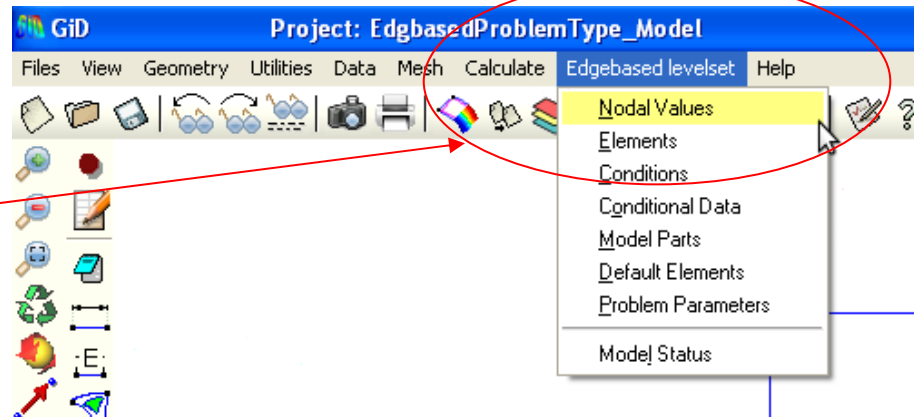
1) We have now to identify which portion of the control domain has porosity different to 1.

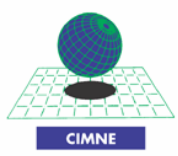
That is where the **porous medium** is and which is its porosity.

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**.

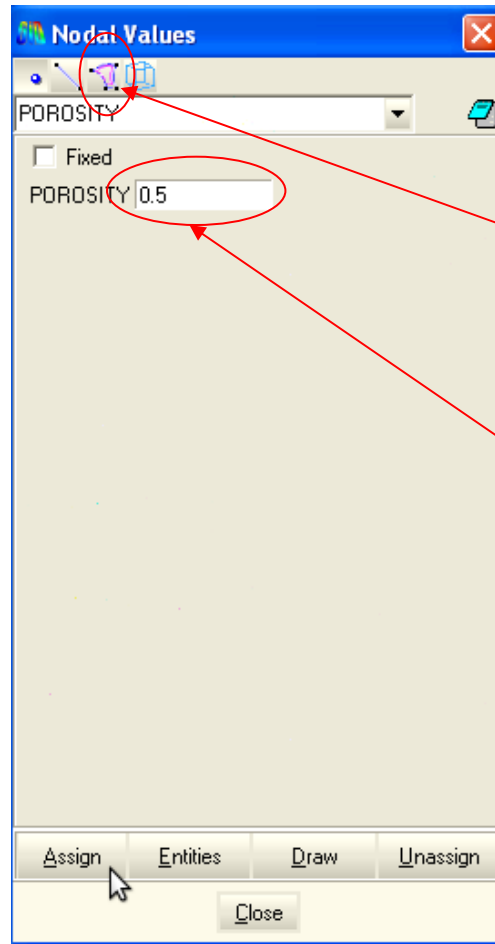
We open the menu
NODAL VALUES





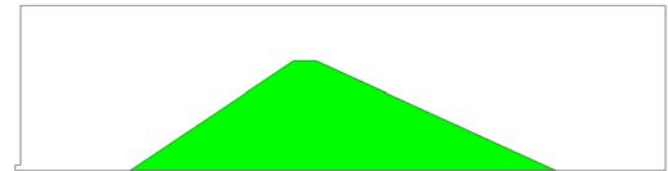
MATERIAL CONDITIONS

1) We have now to identify 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



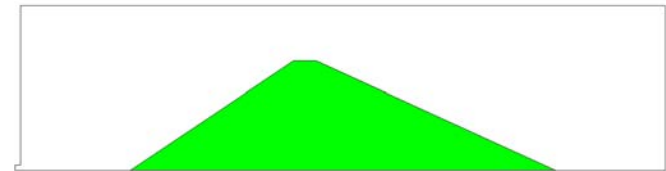
MATERIAL CONDITIONS

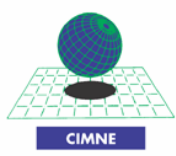
2) We have to set the **diameter D50** to be assigned to the porous region.



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 identified 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

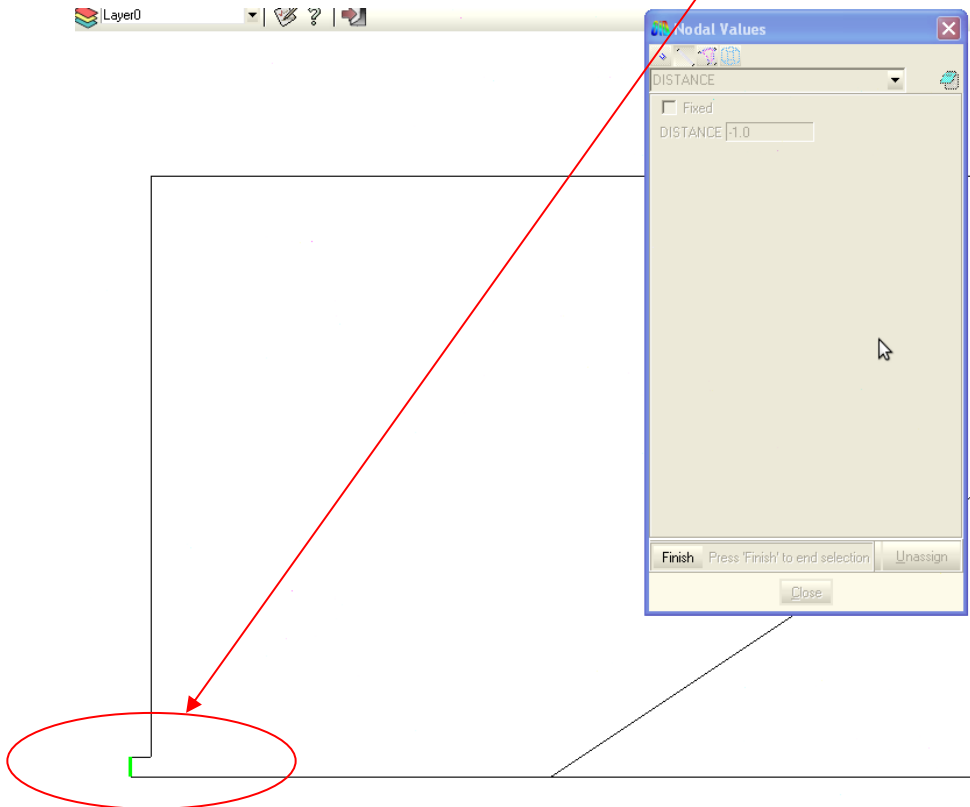
3) We have to identify the **fluid**.

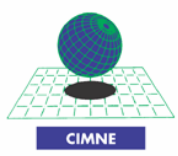
CASE A

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)





MATERIAL CONDITIONS

ANY MODIFICATION

OF THE NODAL VALUES NEED TO

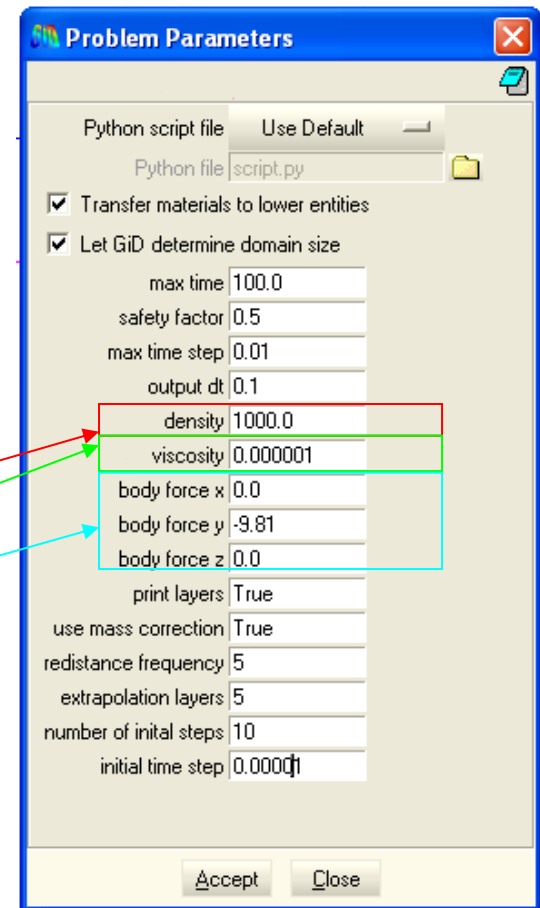
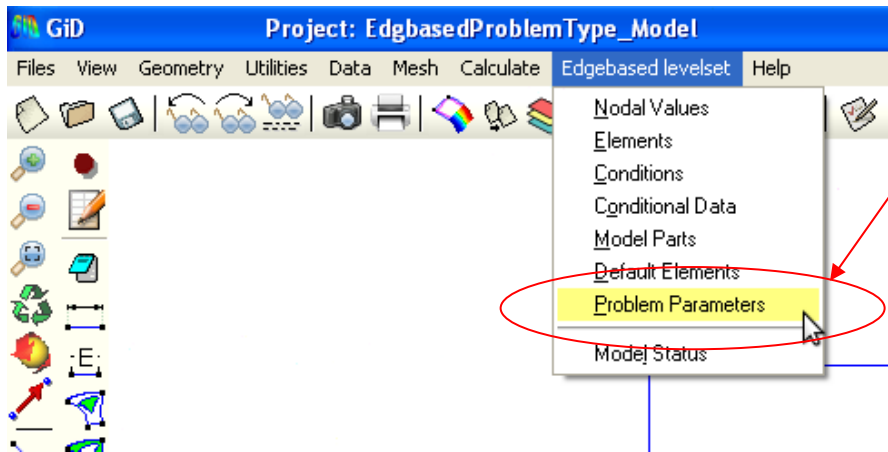
REGENERATE THE MESH

in order to assign the modified value to the
nodes



MATERIAL CONDITIONS

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



They are:

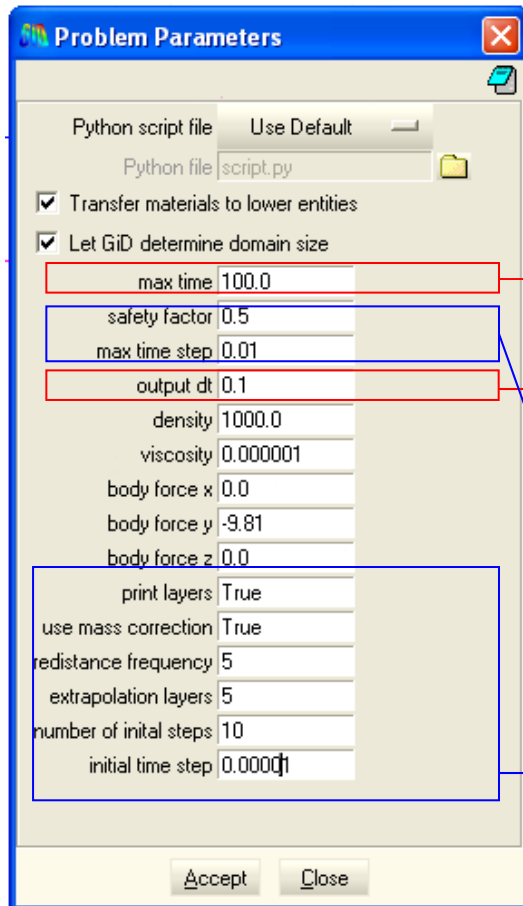
WATER DENSITY

WATER VISCOSITY

GRAVITY



PROBLEM PARAMETERS



Duration of the simulation

Delta t to print results.

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 button at the end

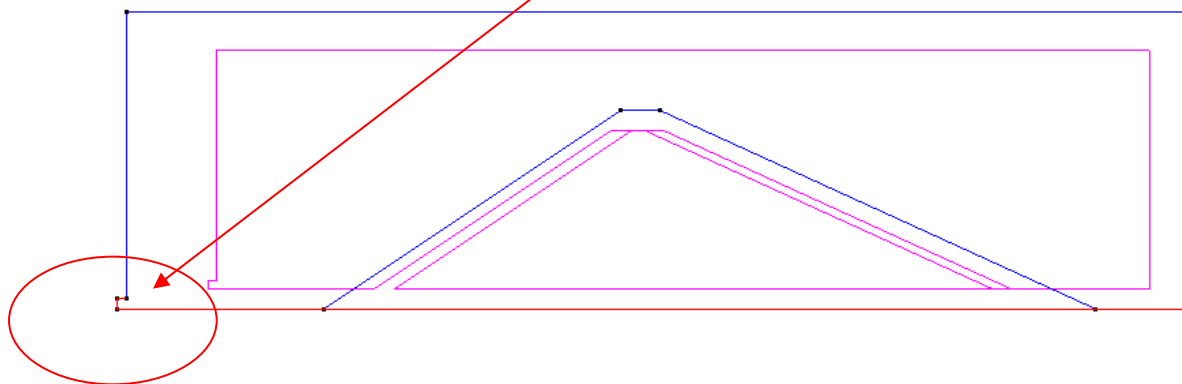
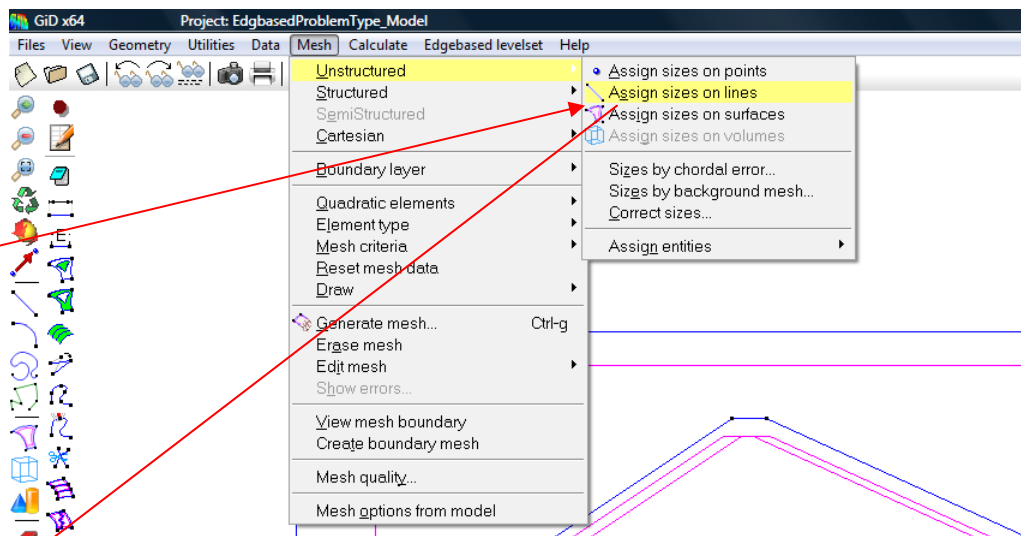


GENERATE THE MESH

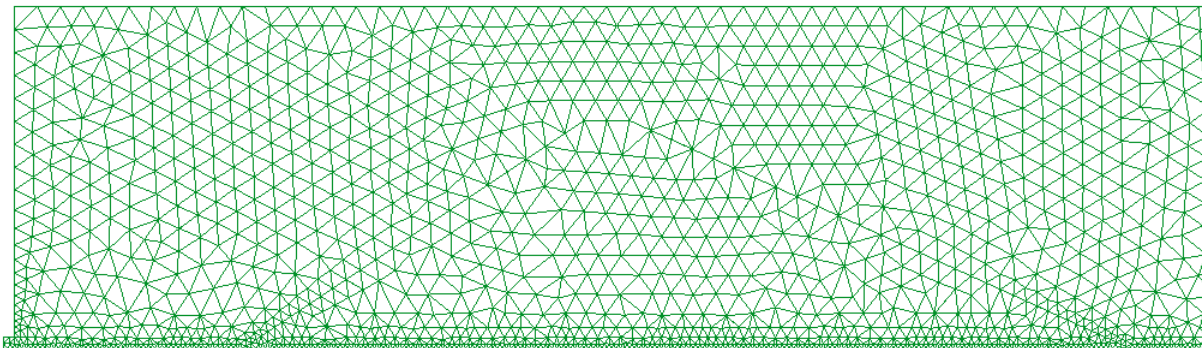
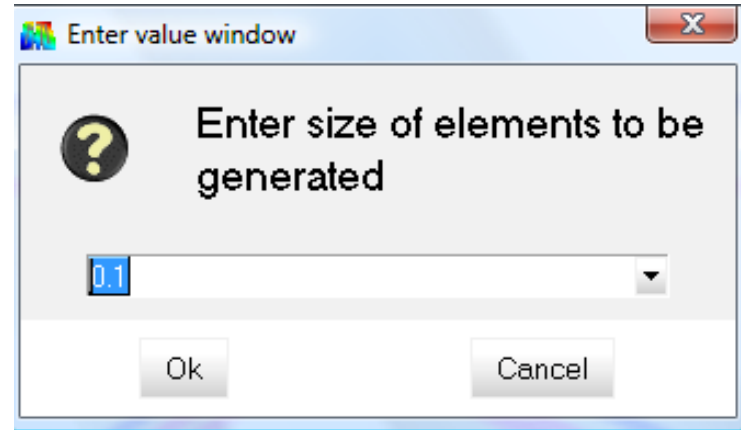
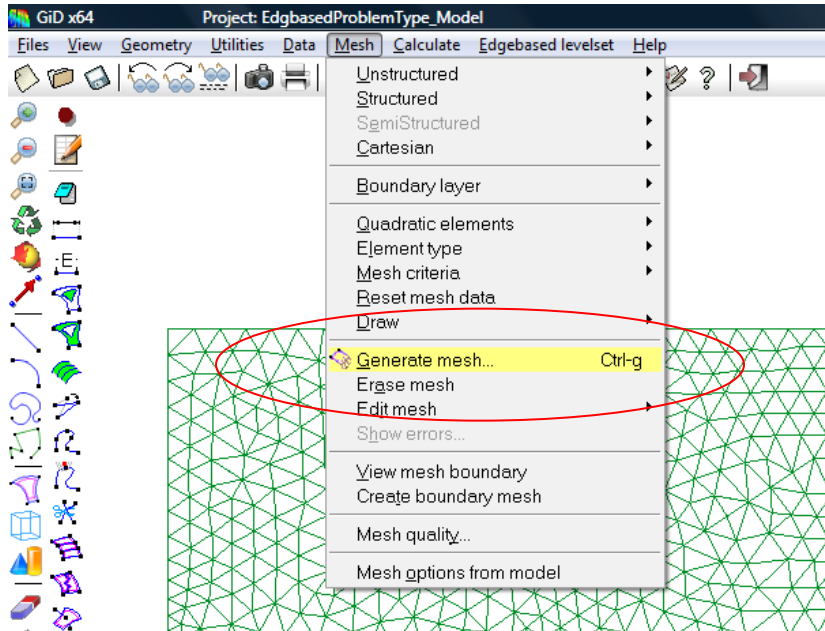
Pay attention that the **entrance of water** line in 2D (or surface in 3D) has to be divided in **more than 2 elements**. Otherwise numerical problems can occurs.

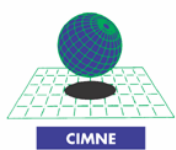
Better to refine a little more the bottom line in 2D

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

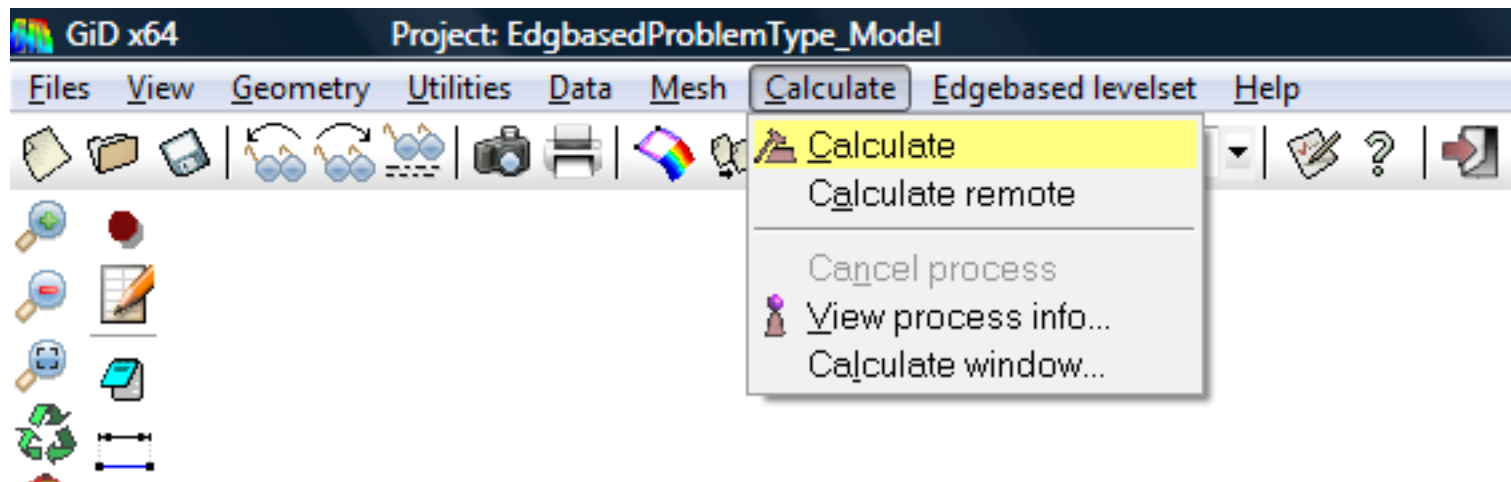


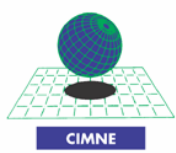
GENERATE THE MESH





Run calculation

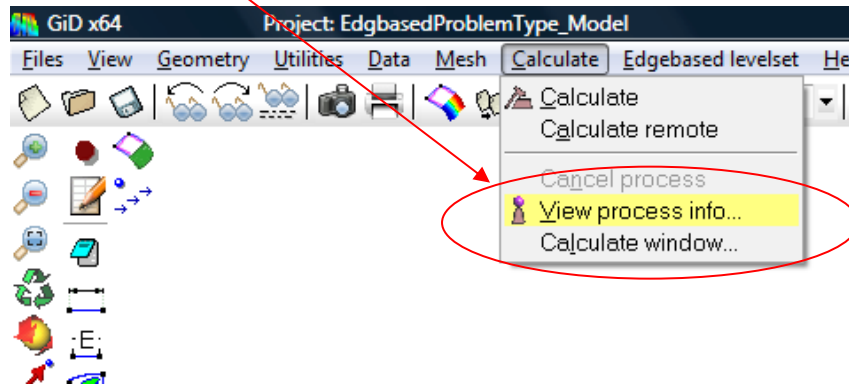




Control the process

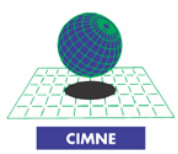
To control the output script file of your calculation you can either

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

