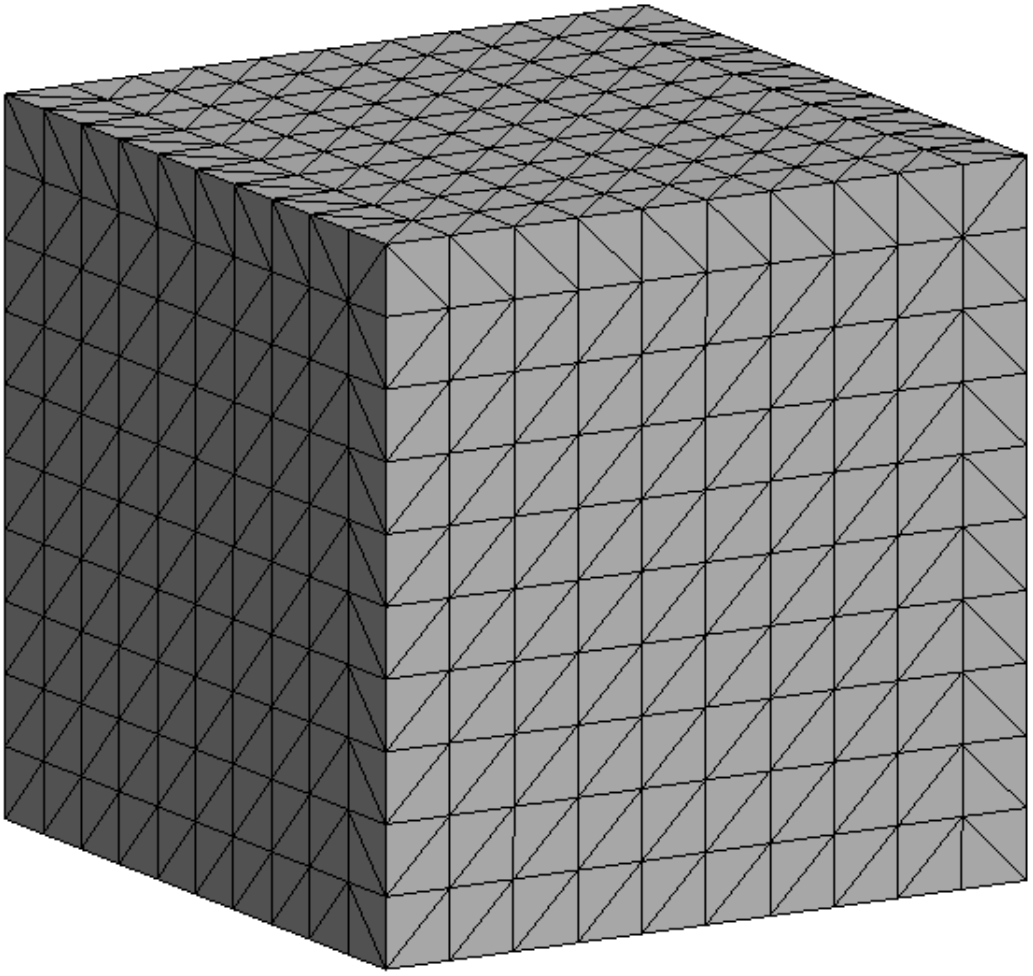
3 INITIATION TO PREPROCESSING

With this example, the user is introduced to the basic tools for the creation of geometric entities and mesh generation.



3.1 First steps

Before presenting all the possibilities that **GiD** offers, we will present a simple example that will introduce and familiarize the user with the **GiD** program.

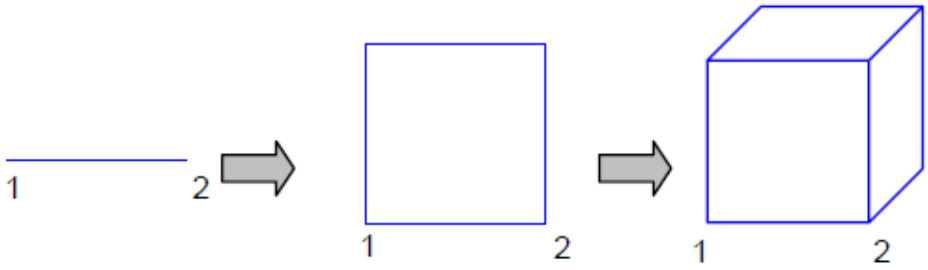
The example will develop a finite element problem in one of its principal phases, the preprocess, and will include the consequent data and parameter description of the problem. This example introduces creation, manipulation and meshing of the geometrical entities used in **GiD**.

First, we will create a line and the mesh corresponding to the line. Next, we will save the project and it will be described in the **GiD** data baseform. Starting from this line, we will create a square surface, which will be meshed to obtain a surface mesh. Finally, we will use this surface to create a cubic volume, from which a volume mesh can then be generated.

3.2 Creation and meshing of a line

We will begin the example creating a line by defining its origin and end points, points 1 and 2 in the following figure, whose coordinates are (0,0,0) and (10,0,0) respectively.

It is important to note that in creating and working with geometric entities, **GiD** follows the following hierarchical order: point, line, surface, and volume.



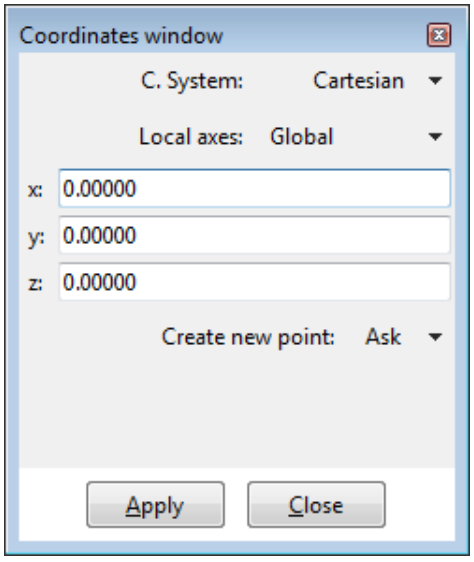
To begin working with the program, open **GiD**, and a new **GiD** project is created automatically.

From this new database, we will first generate points 1 and 2.

Next, we will create points 1 and 2. To do this, we will use an **Auxiliary Window** that will allow us to simply describe the points by entering coordinates. It is accessed by the following sequence: **Utilities->Tools->Coordinates window**

Then, from the Top Menu, select **Geometry->Create->Point**

In the coordinate window opened previously, enter the coordinates of point 1 in the "x", "y", "z" entries and click **Apply** or press **Enter** on the keyboard.



And create point 2 in the same way, introducing its coordinates in the **Coordinates window**.

The last step in the creation of the points, as well as any other command, is to press **Escape**, either via the **Escape** button on the keyboard or by pressing the central mouse button. Select **Close** to close the **Coordinates window**.

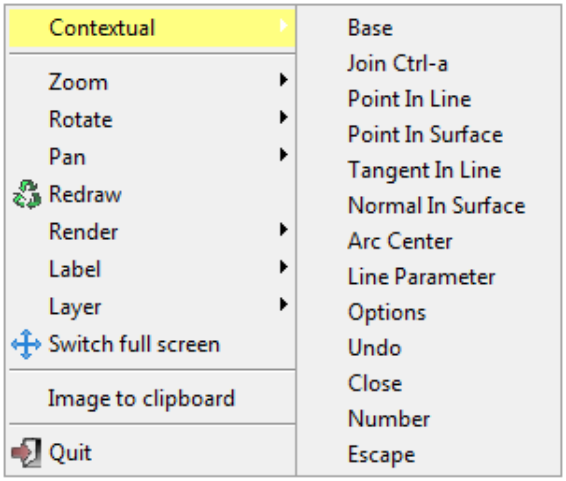
In order to view everything that has been created to this point, center the image on the

screen by choosing in the **Mouse Menu: Zoom->Frame**.

Now, we will create the line that joins the two points. Choose from the **Top Menu**: **Geometry->Create->Straight line.** Option in the **Toolbar** shown below can also be used.

Next, the origin point of the line must be defined. In the **Mouse Menu**, opened by clicking the right mouse button, select **Contextual->Join Ctrl-a.**

NOTE: It is important to note that the Contextual submenu in the Mouse Menu will always offer the options of the command that is currently being used. In this case, the corresponding submenu for line creation, has the following options:



NOTE: With option Join, a point already created can be selected on the screen. The command No Join is used to create a new point that has the coordinates of the point that is selected on the screen. We can see that the cursor changes form for the Join and No Join commands.

 Cursor during use of **Join** command

 Cursor during use of No **Join** command

Now, choose on the screen the first point, and then the second, which define the line. Finally, press **Escape** to indicate that the creation of the line is completed. Press **Escape** again to end the line creation function, if you don't press Escape you can continue creating lines.

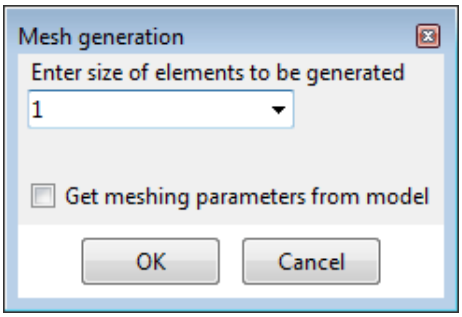
Once the geometry has been created, we can proceed to the line meshing. In this example, this operation will be presented in the simplest and most automatic way that **GiD** permits. To do this, from the **Top Menu** select: **Mesh->Generate mesh.**

And an **Auxiliary Window** appears, in which the size of the elements should be defined by

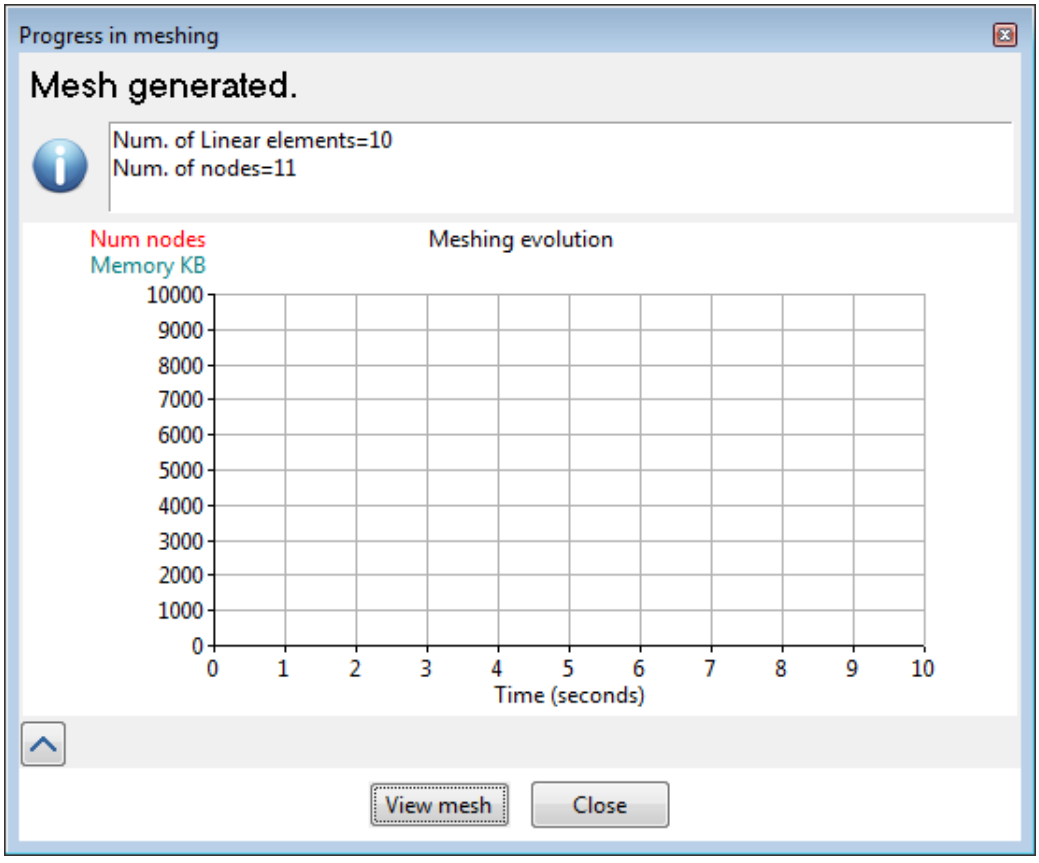
the user.

NOTE: The size of an element with two nodes is the length of the element. For, surfaces or volumes, the size is the mean length of the edge of the element.

In this example, the size of the element is defined in concordance with the length of the line, chosen for this case as size 1. Click **OK**.



Once the mesh has been generated a window with the mesh information appears. Click **View mesh**.

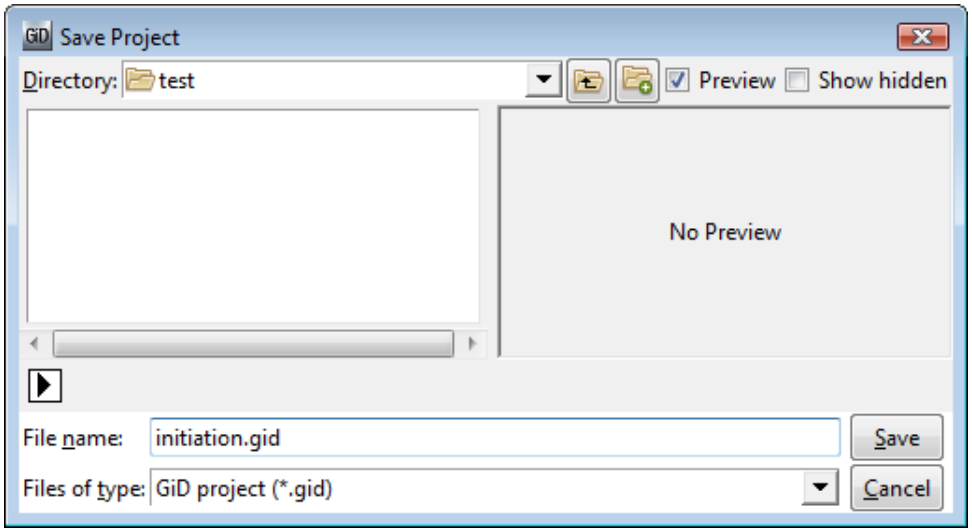


Automatically **GiD** generates a mesh for the line. The finite element mesh is presented on the screen in a grey color.

The mesh is formed by ten linear elements of two nodes. To see the numbering of the nodes and mesh elements, select from the **Mouse Menu: Label->All,** and the numbering for the 10 elements and 11 nodes will be shown, as below.

Once the mesh has been generated, the project should be saved. To save the example select from the **Top Menu: Files->Save.**

The program automatically saves the file if it already has a name. If it is the first time the file has been saved, the user is asked to assign a name. For this, an **Auxiliary Window** will appear which permits the user to browse the computer disk drive and select the location in which to save the file. Once the desired directory has been selected, the name for the current project can be entered in the space titled **File Name**. Save it as **initiation.gid**.



NOTE: Next, the manner in which **GiD** saves the information of a project will be explained. **GiD** creates a directory with a name chosen by the user, and whose file extension is **.gid**. **GiD** creates a set of files in this directory where all the information generated in the present example is saved. All the files have the same name of the directory to which they belong, but with different extensions. These files should have the name that **GiD** designates and should not be changed manually.

Each time the user selects option **save** the database will be rewritten with the new information or changes made to the project, always maintaining the same name.

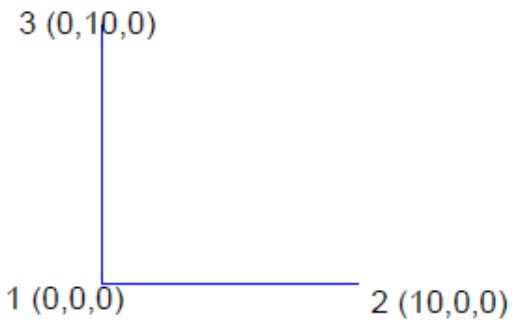
To exit **GiD**, simply choose **Files->Quit.**

To access the project that we have just created, simply open **GiD** and select from the **Top Menu: Files->Open.** An **Auxiliary Window** will appear which allows the user to access and open the directory **initiation.gid**.

3.3 Creation and meshing of a surface

We will now continue with the creation and meshing of a surface.

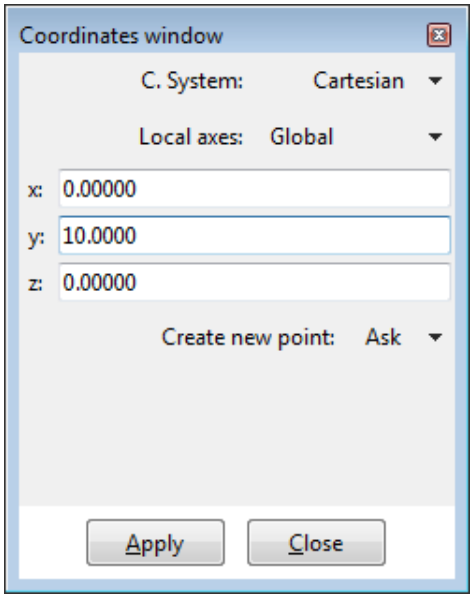
First, we will create a second line between points 1 and 3.



We will now generate the second line. We will now use again the **Coordinates Window** to enter the points. (**Utilities->Tools->Coordinates Window**)

Select the line creation tool in the toolbar.

Enter point (0,10,0) in the **Coordinates Window** and click **Apply**.

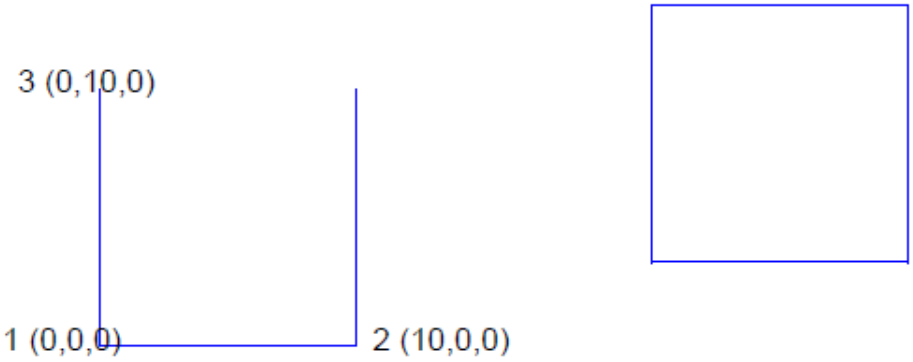


With option **Contextual->Join Ctrl-a** (mouse menu) click over point 1. A line should be created between (0,10,0) and (0,0,0). Press **Escape** twice.

With this, a right angle of the square has been defined.

Center the image in the screen with **View->Zoom->Frame**.

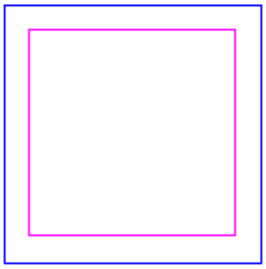
Finish the square by creating point (10,10,0) and the lines that join this point with points 2 and 3.



Now, we will create the surface that these four lines define. To do this, access the create surface command by choosing: **Geometry->Create->NURBS surface->By contour**. This option is also available in the toolbar:

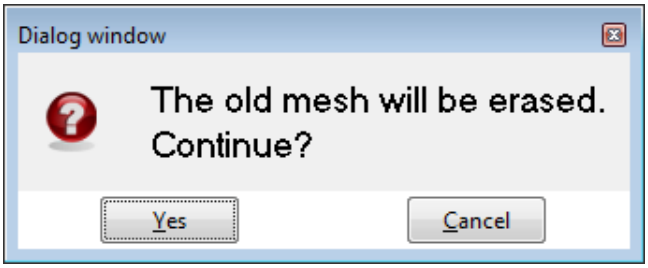
GiD then asks the user to define the 4 lines that describe the contour of the surface. Select the lines using the cursor on the screen, either by choosing them one by one or selecting them all with a window. Next, press **Escape** twice.

As can be seen below, the new surface is created and appears as a smaller, magenta-colored square drawn inside the original four lines.

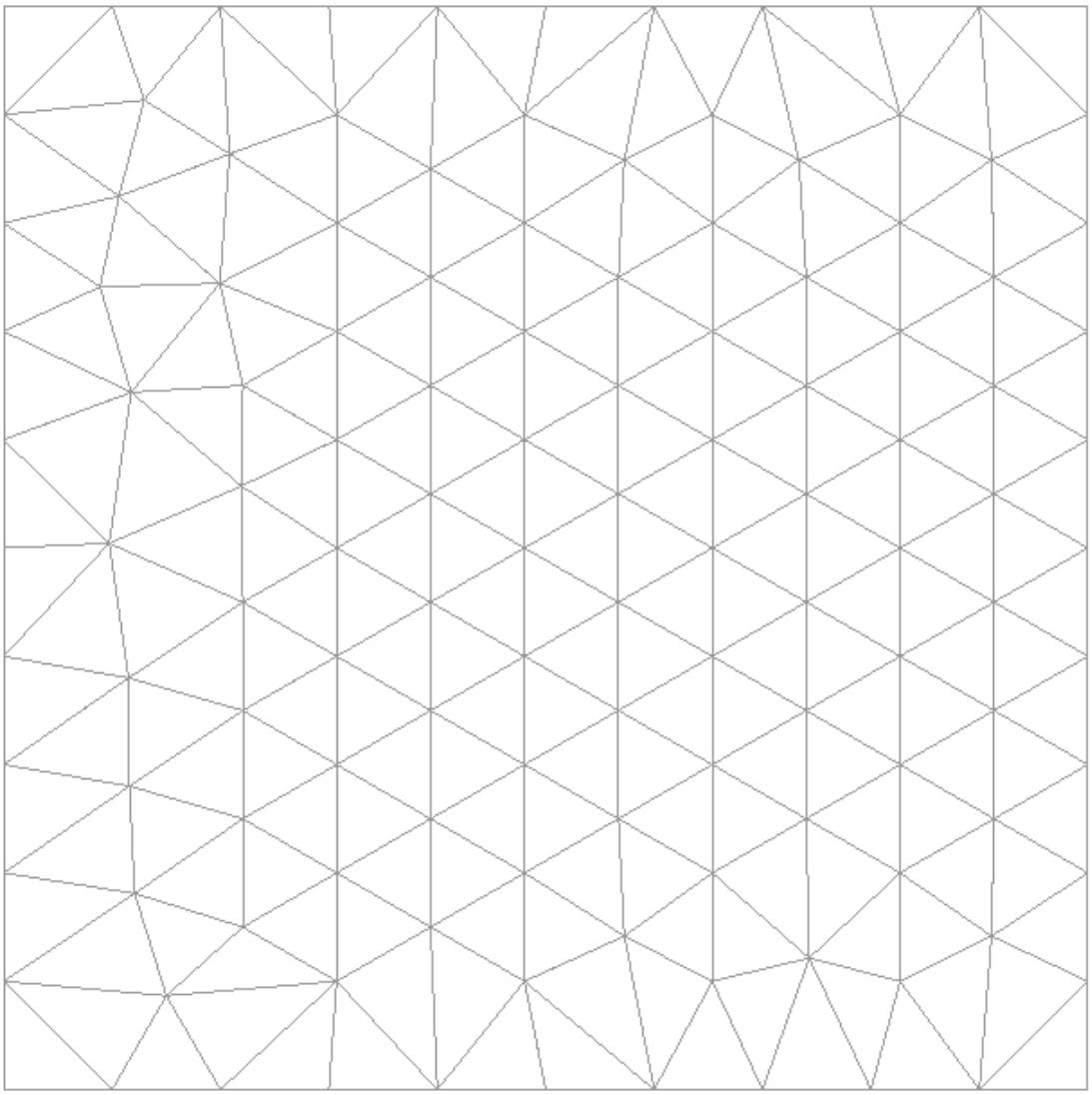


Once the surface has been created, the mesh can be created in the same way as was done for the line. From the **Top Menu** select: **Mesh->Generate mesh.**

A window appears asking if the previous mesh should be elliminated. Click **Yes**.



Another window appears which asks for the maximum size of the element, in this example defined as 1.

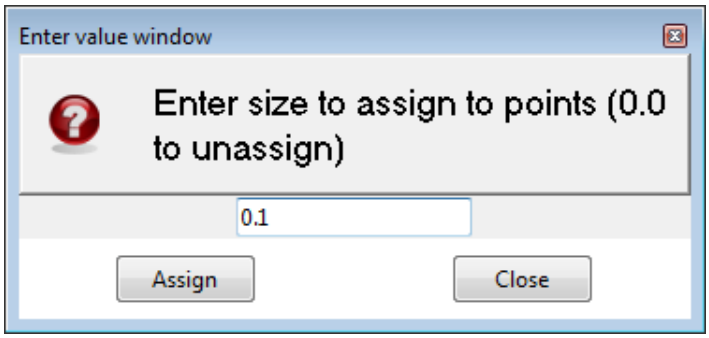


We can see that the lines containing elements of two nodes have not been meshed. Rather the mesh generated over the surface consists of planes of three-nodded, triangular elements.

NOTE: GiD meshes by default the entity of highest order with which it is working.

GiD allows the user to concentrate elements in specified geometry zones. Next, a brief example will be presented in which the elements are concentrated in the top right corner of the square.

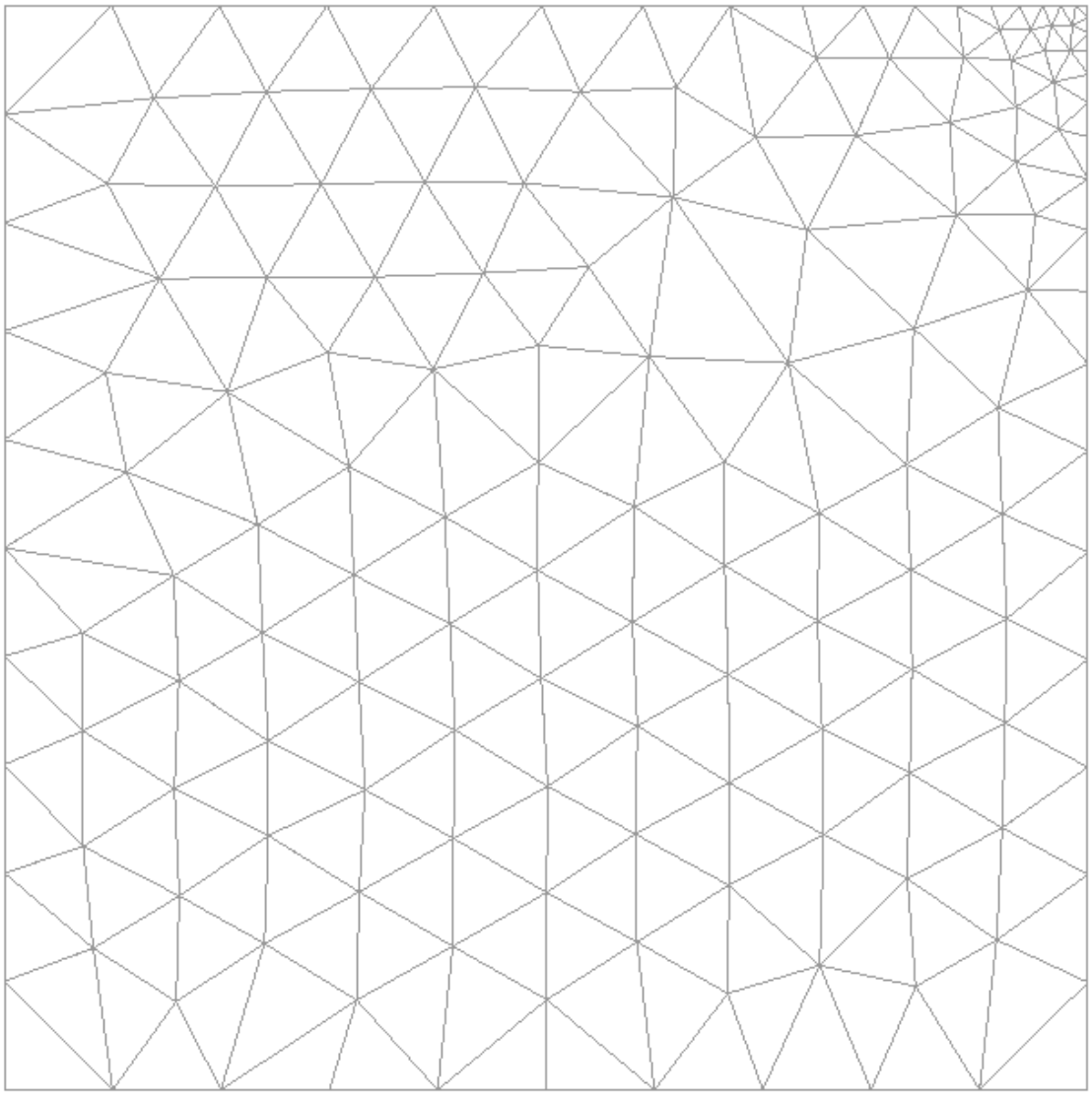
This operation is realized by assigning a smaller element size to the point in this zone than for the rest of the mesh. Select the following sequence: **Mesh->Unstructured->Assign sizes on points.** The following dialog box appears, in which the user can define the size:



Enter **0.1** and click **Assign.**

Select one of the four corners and press **Escape.** The same window comes up again, click **Close.**

We must now regenerate the mesh, erasing the mesh generated earlier, and we obtain the following:



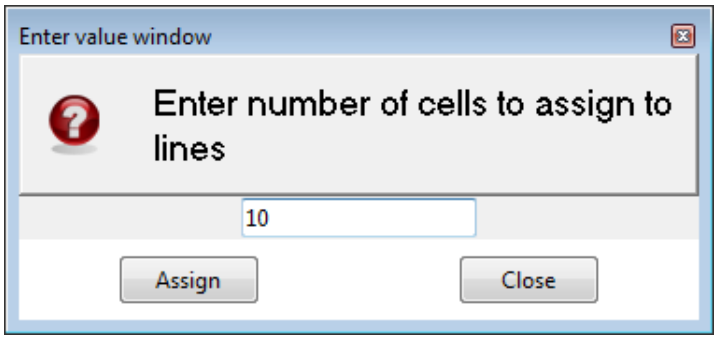
As can be seen in the figure above, the elements are concentrated around the chosen

point. Various possibilities exist for controlling the evolution of the element size, which will be presented later in the manual.

To generate a surface mesh in which the elements are presented uniformly, the user can select the option for a structured mesh. This guarantees that the same number of elements appears around a node and that the element size is as uniform as possible. To generate this type of mesh, choose: **Mesh->Structured->Surfaces->Assign number of cells.**

Using this command, the user should first select the **4-sided NURBS surface** that will be defined by the mesh and press **ESC.**

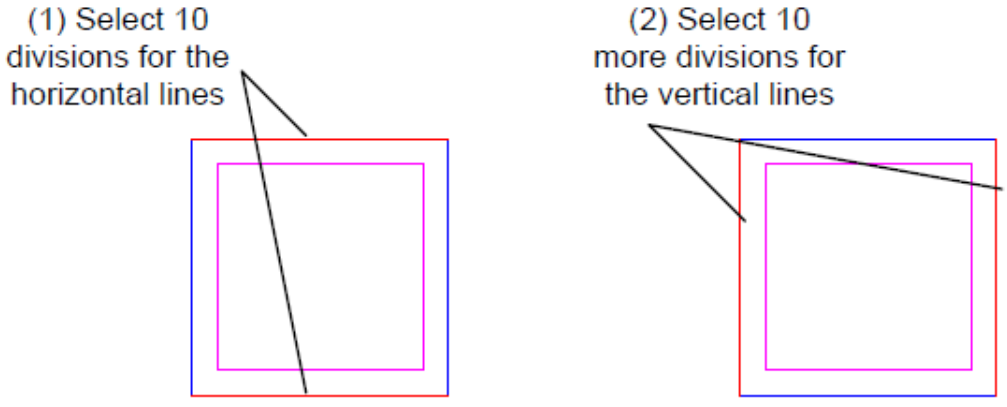
Then a window appears where the number of subdivisions for the surface limit lines should be entered.



Enter **10** and click **Assign** and select one of the **horizontal** lines, the parallel line is also selected. Press **ESC**.

The same window appears again, click **Assign** and select one of the **vertical** lines, the parallel one is also selected. Press **ESC**.

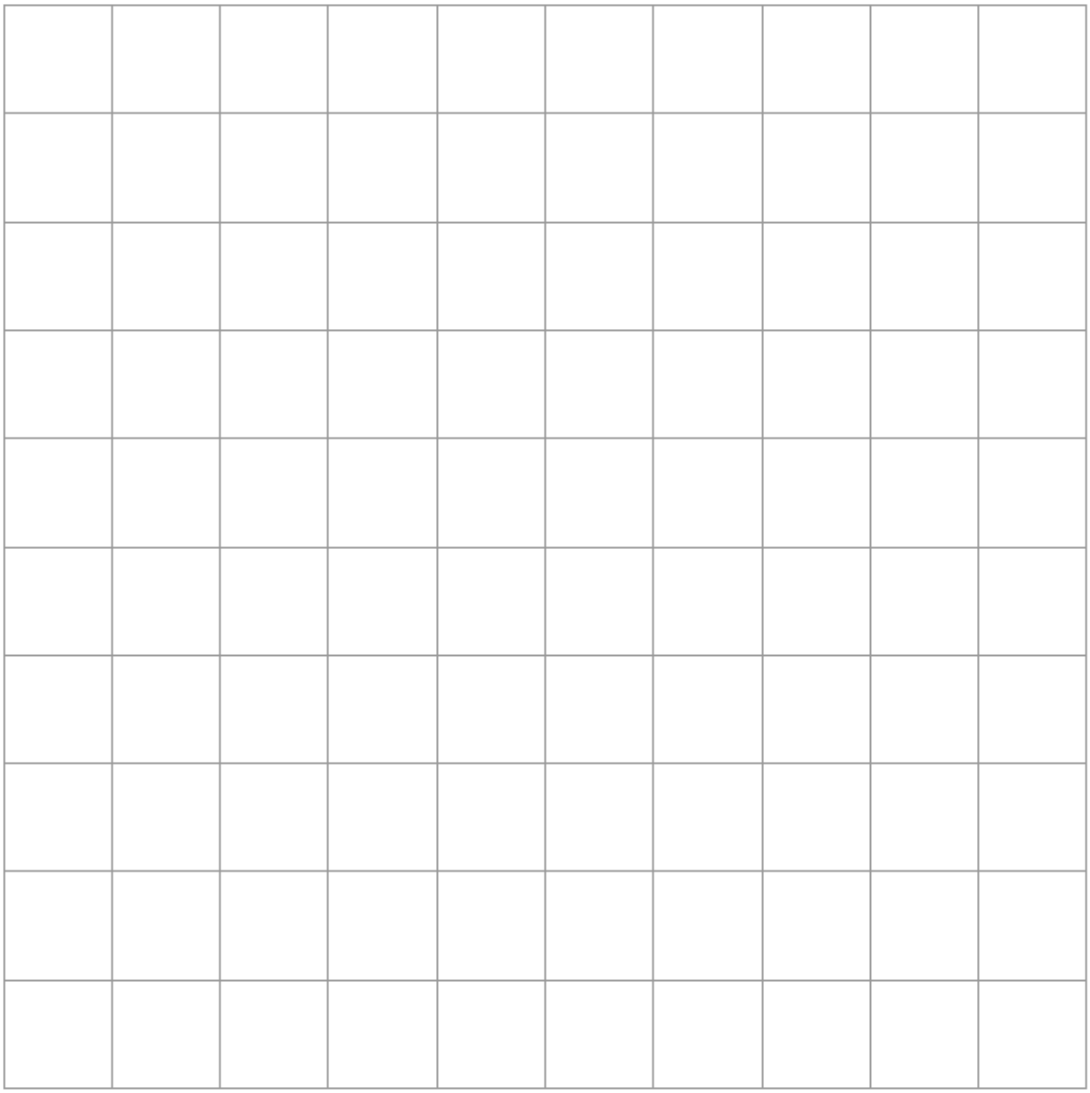
Click **Close** when the window appears again.



The number of divisions can be checked selecting **Mesh->Draw->Num of divisions**. To exit this visualization mode press **ESC.**

NOTE: GiD only generates structured meshes for surfaces of the type **4-sided surface** or **NURBS surface**.

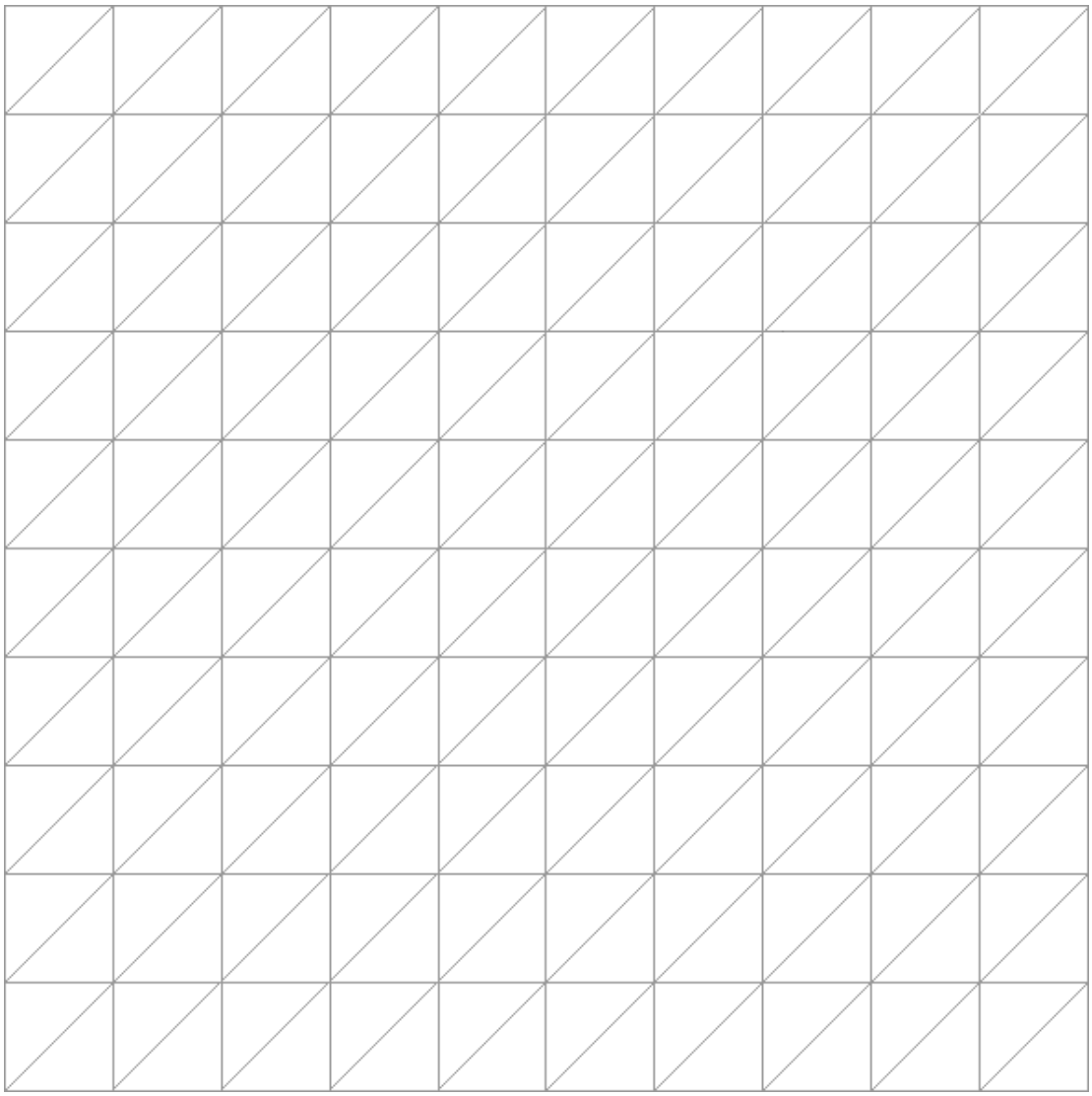
When this has been done, the mesh is generated in the same way as the unstructured mesh, by choosing **Mesh->Generate mesh.** Erase the old mesh and assign a general element size of 1, though in this case it is not necessary.



NOTE: Another way to get the same result is using the option **Mesh->Structured->Surfaces->Assign size**. With this option we set the element size. If we want to get 10 elements per line and the line measures 10 units, we should set 1 as size.

If we don't know how much measures a line we can use the option **Utilities->Distance** and select the 2 points defining the line.

We can see here that the default element type used by **GiD** to create a structured mesh is a square element of four nodes rather than a three-nodded, triangular element. To obtain triangular elements, the user can specifically define this type of element, by choosing **Mesh->Element type->Triangle,** and selecting the surface to mesh as a triangular element. Regenerate the mesh, and the following figure is obtained:

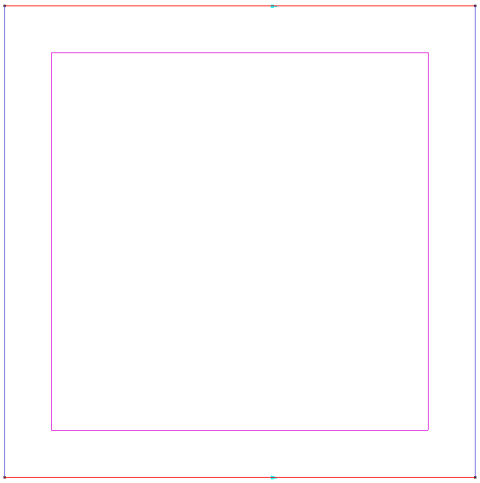


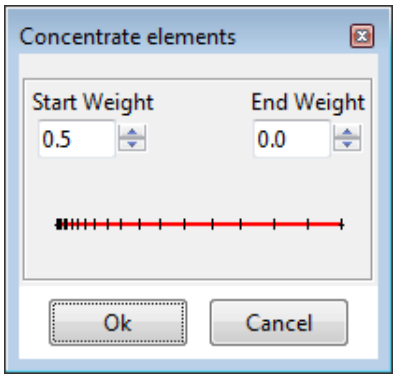
GiD also allows the user to concentrate elements in structured meshes. This can be done by selecting **Mesh->Structured->Lines->Concentrate elements.**

First, we must select the lines that need to be assigned an element concentration weight. The value of this weight can be either positive or negative, depending on whether the user wants to concentrate elements at the beginning or end of the lines. Next, a vector appears which defines the start and end of the line and which helps the user assign the weight correctly.

We want to concentrate the elements in the left zone of the square.

Select both **horizontal** lines and press **ESC**. A window appears to enter the weights values. Both lines should have the same direction so enter a weight of 0.5 to the beginning of the line and click **Ok**. Press **ESC** again to leave the function.





If lines have different direction, to obtain the same result, we should assign the weight for one line to the beginning and to the end for the other line.

From these operations, we obtain the following mesh: