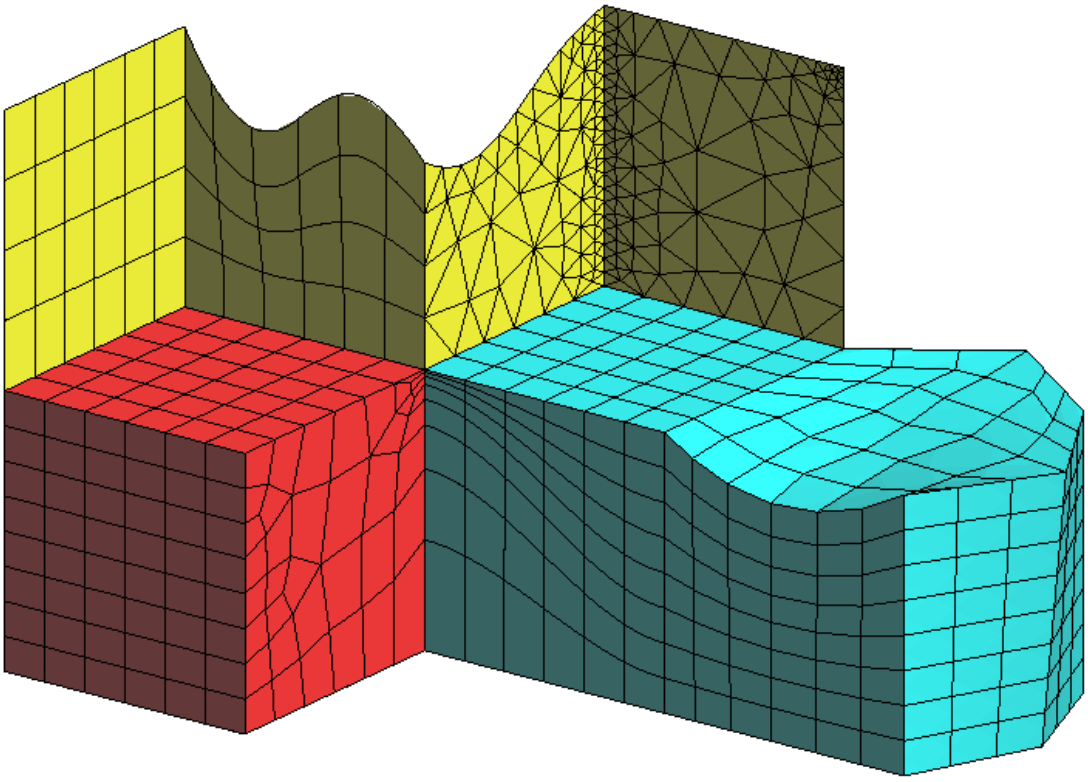
7 METHODS FOR MESH GENERATION

The objective of this example is to mesh a model using the various options available in GiD for controlling the element type in structured, semi-structured and unstructured meshes. It also presents how to concentrate elements and control the distribution of mesh sizes.

The six methods covered are:

- Generating a mesh using tetrahedral
- Generating a volume mesh using spheres
- Generate a mesh using circles
- Generating a volume mesh using points
- Generating a mesh using quadrilaterals
- Generating a structured mesh on surfaces and volumes
- Generating a semi-structured volume mesh
- Generating a mesh using quadratic elements



7.1 Introduction

In order to carry out this example, start from the project "ToMesh3.gid". This project contains a geometry that will be meshed using different types of elements.

7.1.1 Reading the initial project

1 . In the **Files** menu, select **Read**. Select the project "ToMesh3.gid" and click **Open**.

2 . The geometry appears on the screen. It is a set of surfaces and three volumes. Select **Render->Flat** from the mouse menu or from the **View** menu. In Figure 1 shows the

geometrical model loaded.

3 . Rotate and make several changes in the perspective so as to get a good idea of the geometry involved.

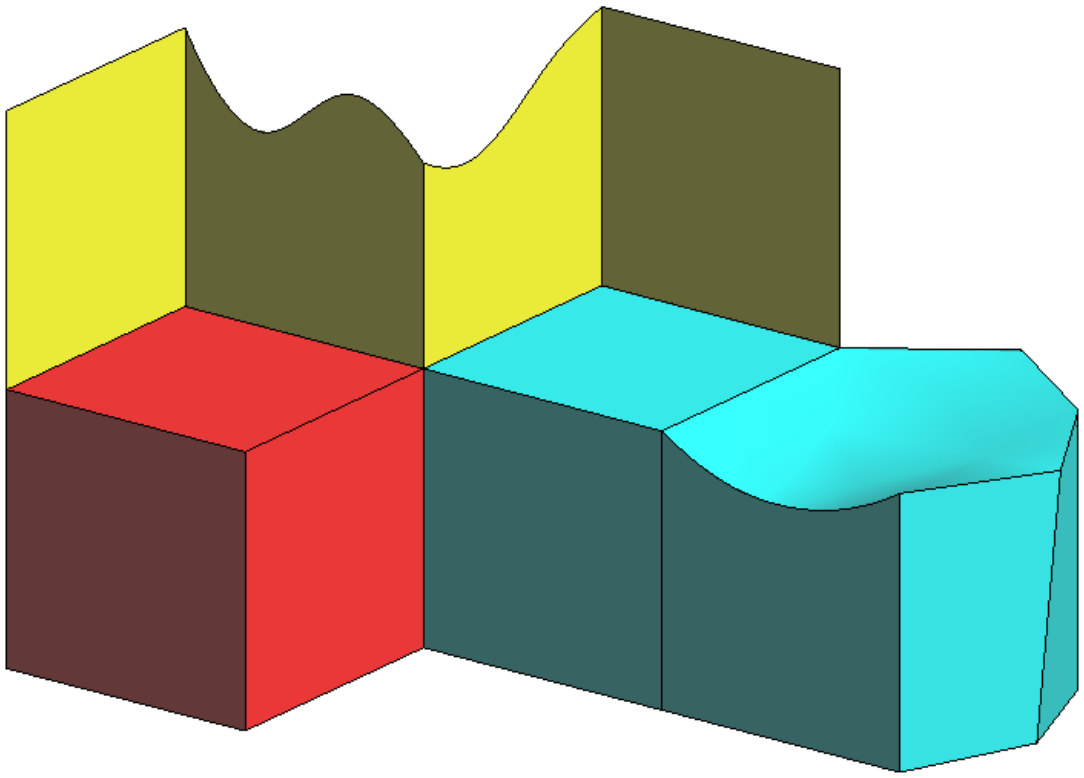


Figure 1. Contents of the project "ToMesh3.gid"

Finally, return to the normal visualization **Render->Normal**. This mode is more user-friendly.

7.2 Types of mesh

Using GiD the mesh may be generated in different ways, depending on the needs of each project. The two basic types of meshes are the structured[2] mesh and the unstructured mesh. For volumes only there is one additional type, the semi-structured[3] mesh.

For all these types of mesh a variety of elements may be used (linear ones, triangles, quadrilaterals, circles,tetrahedra, hexahedra, prisms, spheres or points). In this tutorial you will become familiarized with the mesh-generating combinations available in GiD.

---

[2] A structured mesh is one in which each node is connected to a constant number of elements.

[3] A semi-structured volume mesh is one in which you can distinguish a fixed structure in one direction, i.e. there is a fixed number of divisions. However, within each division the mesh need not be structured. This kind of mesh is only practical for topologically prismatic volumes.

7.2.1 Generating the mesh by default

In order to get the same results we will reset the mesh options.

1 . Open the preferences window selecting **Utilities->Preferences.**
2 . Select the **Meshing** card, click on **Reset** and then **Accept**.
3 . Select **Mesh->Generate mesh**.
4 . A window comes up in which to enter the maximum element size for the mesh to be generated. As default value could change from one version of GiD to another, insert **2** to get the same results as shown in images **OK**.
5 . A meshing process window comes up. Then another window appears with information about the mesh generated. Click **View mesh** to visualize the mesh.
6 . The result is the mesh in Figure 2. There are various surfaces and volumes. By default, mesh generation in GiD obtains unstructured meshes of triangles on surfaces and tetrahedra on volumes.
7 . Select **Render->Flat** in mouse menuto see the mesh in render mode. As is shown in Figure 3, volume meshes are represented a little bit differently from surface meshes, although in both cases triangles are shown. If the triangles you see are the boundary of a volume mesh, they are shown with black edges that are thicker than surface meshes triangles. If the triangles form a boundary volume mesh and, at the same time, a triangle surface mesh (this can be obtained if surfaces are selected with the option **Mesh->Mesh criteria->Mesh->Surfaces**), the wider edges are colored with the color of the surface layer. Examples of these different kinds of render are shown in Figure 3.

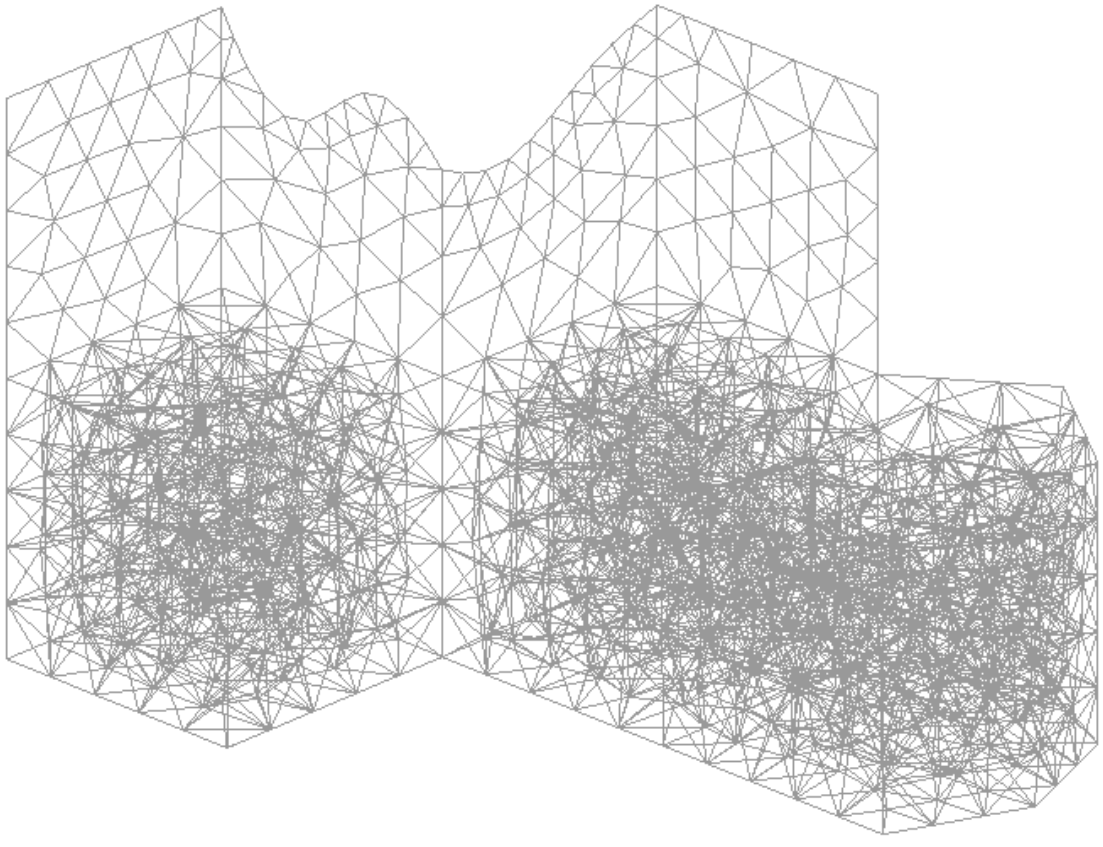


Figure 2. Generating the mesh by default.

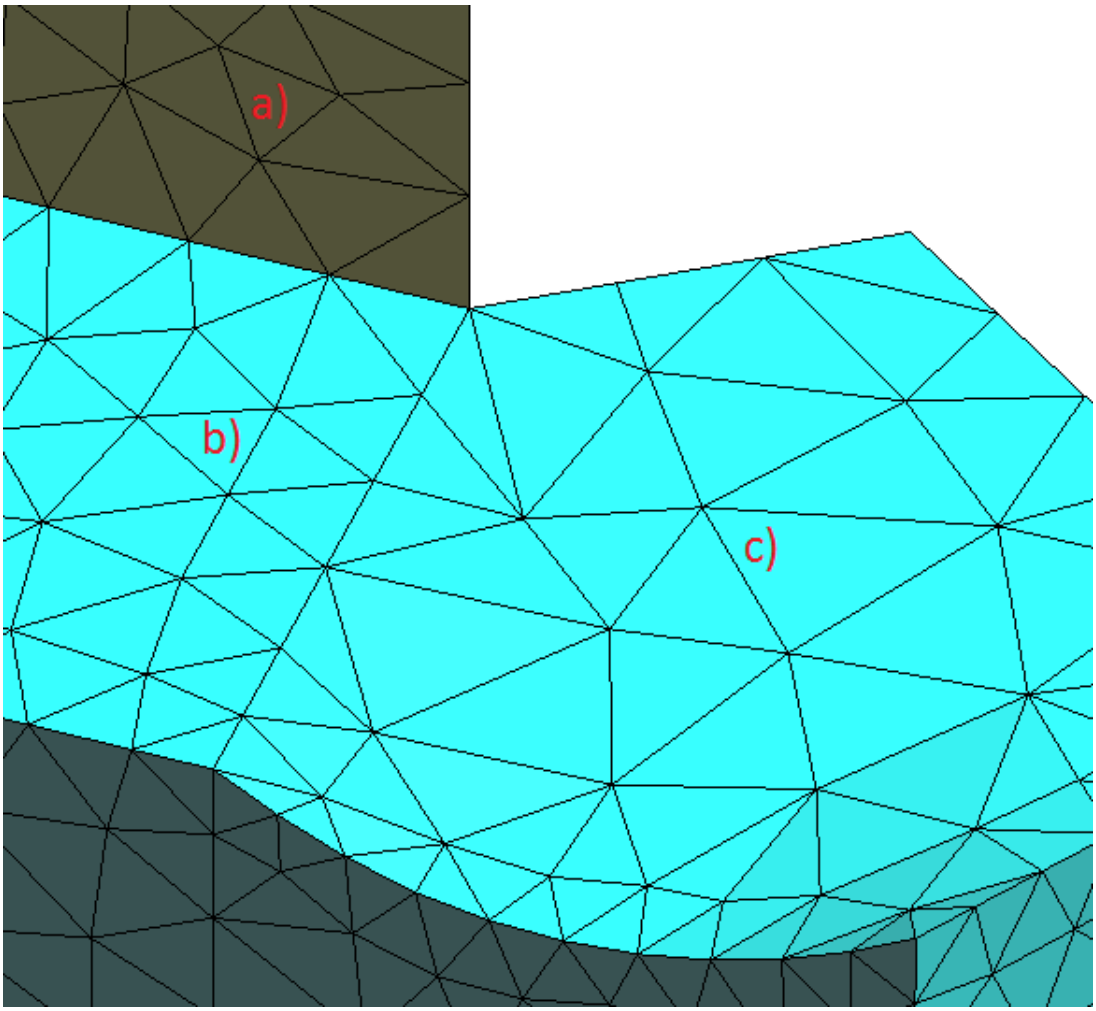


Figure 3. Different render styles: a) surface mesh, b) volume mesh, c) surface mesh and volume mesh together (surface layer is red and volume layer is blue)

7.2.2 Generating the mesh using circles and spheres

1 . Select **Mesh->Element type->Sphere**. Select volume number 1 and press **ESC.** To see entity numbers select **Label** from the mouse menu or from the **View** menu. If you wish the geometrical entity labels to be displayed, the view mode needs to be changed to Geometry using **View->Mode->Geometry** (this option may also be found in the GiD Toolbar). Select **Render->Normal** in the mouse menu to see the labels.

2 . Select **Mesh->Element type->Circle**. Select surface number 24 and press **ESC**.

3 . Select **Mesh->Generate mesh**.

4 . A window comes up asking whether the previous mesh should be eliminated. Click **Yes**.

5 . Another window appears in which to enter the maximum element size. Leave the default value unaltered and click **OK**. The result is a mesh as illustrated in Figure 4.

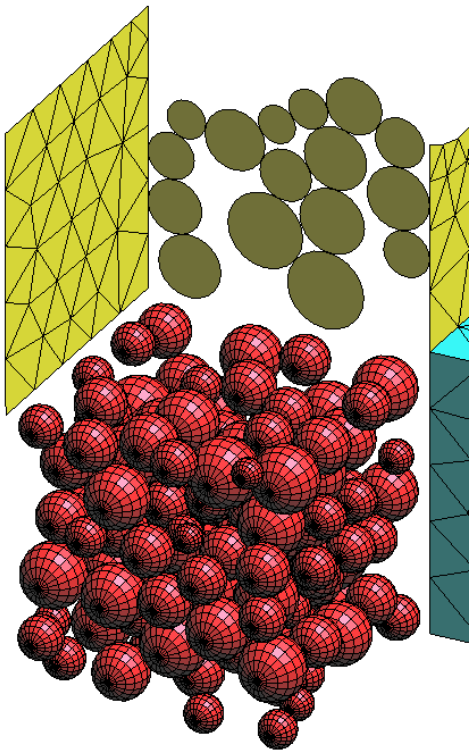


Figure 4.(Render Flat) Generating a mesh on a volume using points.

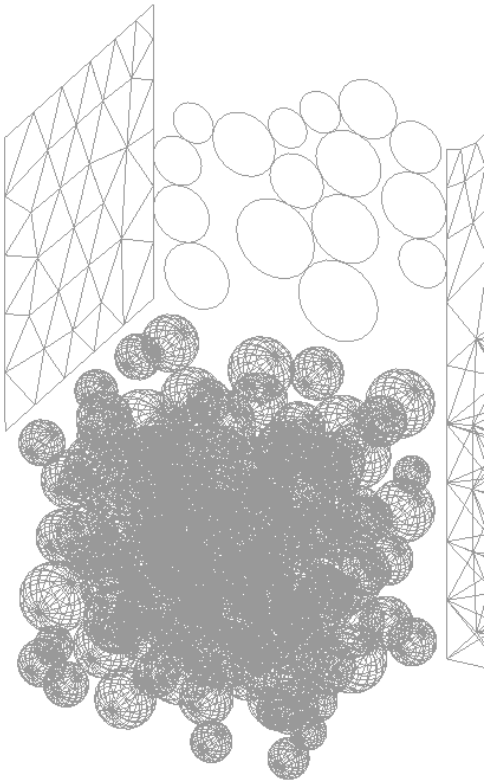


Figure 4.(Render Normal) Generating a mesh on a volume using points.

7.2.3 Generating the mesh using points

1 . Select **Mesh->Element type->Only points**. Select volume number 1 and press **ESC.**
2 . Select **Mesh->Generate mesh**.
3 . A window comes up asking whether the previous mesh should be eliminated. Click **Yes**.
4 . Another window appears in which to enter the maximum element size. Leave the default value unaltered and click **OK**. The result is a mesh as illustrated in Figure 5.

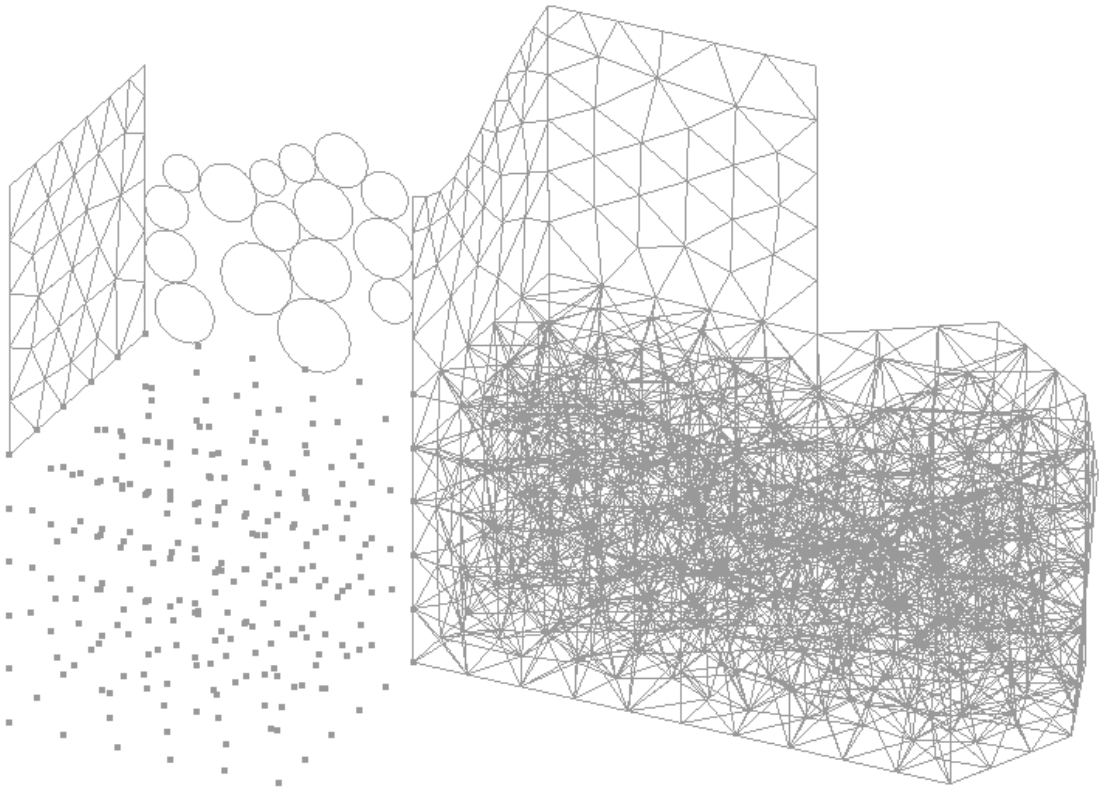


Figure 5. Generating a mesh on a volume using points.

5 . Now volume number 1 is meshed using only the generated nodes.

7.2.4 Generating the mesh using quadrilaterals

1 . Select **Mesh->Element type->Quadrilateral**. Select surfaces number 24 and 12 and press **ESC**.

2 . Select **Mesh->Generate mesh**.

3 . A window comes up asking whether the previous mesh should be eliminated. Click **Yes**.

4 . Another window appears in which the maximum element size can be entered. Leave the default value unaltered and click **OK**. The result will be the mesh illustrated in Figure 6.

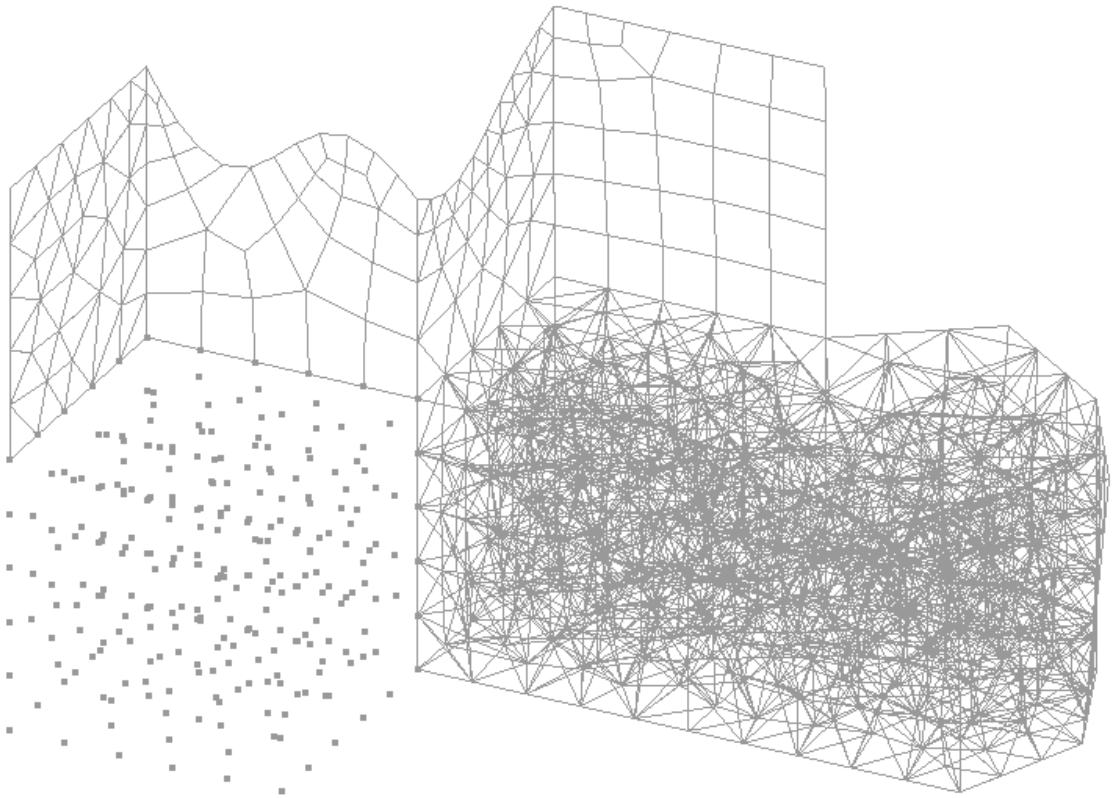


Figure 6. An unstructured mesh generated using quadrilaterals.

5 . The surface is meshed with quadrilaterals, forming an unstructured mesh, surface 12 seems structured but if you zoom in, you will see that it is not.

7.2.5 Generating a structured mesh (surfaces)

1 . To mesh surfaces with a structured mesh, select the option **Mesh->Structured->Surfaces->Assign number of cells**.
2 . Select all top surfaces 9, 24, 26 and 12 and press **ESC**.
3 . A window appears in which to enter the number of divisions that the lines to be selected will have. Enter 4.
4 . Click **Assign** and select one vertical line[5] (parallel to the **Y** axis). Press **ESC**.
5 . Another window appears in which to enter the number of divisions on the lines. Enter 6.
6 . Click **Assign** and select the 4 bottom lines[5]. Press **ESC**.
7 . Another window appears in which to enter the number of divisions on the lines. In this case, all the boundary lines have already been defined. Therefore, click **Close**.
8 . Select **Mesh->Element type->Triangle**. Select surfaces 26 and 12. Press **ESC**.

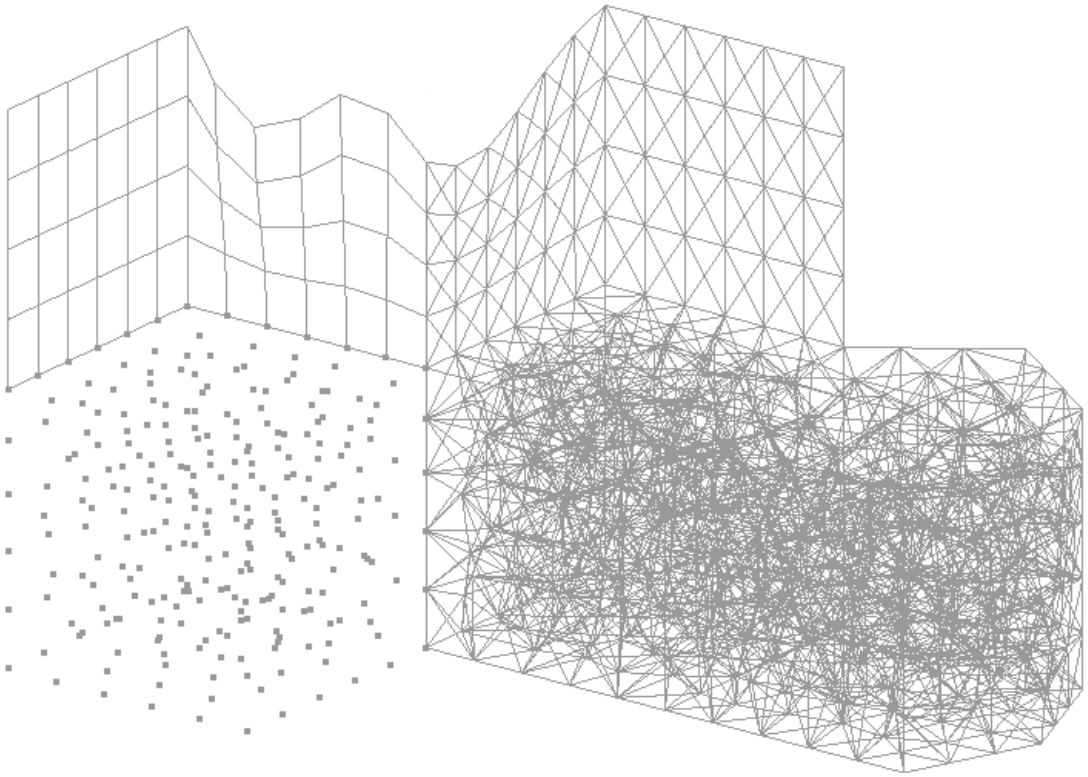


Figure 7. Structured mesh of quadrilateral and triangular elements on surfaces.

9 . Select **Mesh->Generate mesh**.

10 . A window comes up asking whether the previous mesh sould be eliminated. Click **Yes**.

11 . Another window appears in wihich to enter the maximum element size. Leave the default value unaltered an click **OK**. The result is the mesh shown in Figure 8.

12 . As seen in Figures 7 and 8, GID can obtain surface structured meshes made of quadrilaterals or triangles. There are tow kinds of structured mesh that use triangles: the one shown in Figure 7 is obtained when the **Utilities->Preferences->Meshing->Main->Structured Mesher->Symmetrical structured->triangles** option is set. If this option is not set, the mesh presented in Figure 8 is produced (with fewer nodes than if using the previous option).

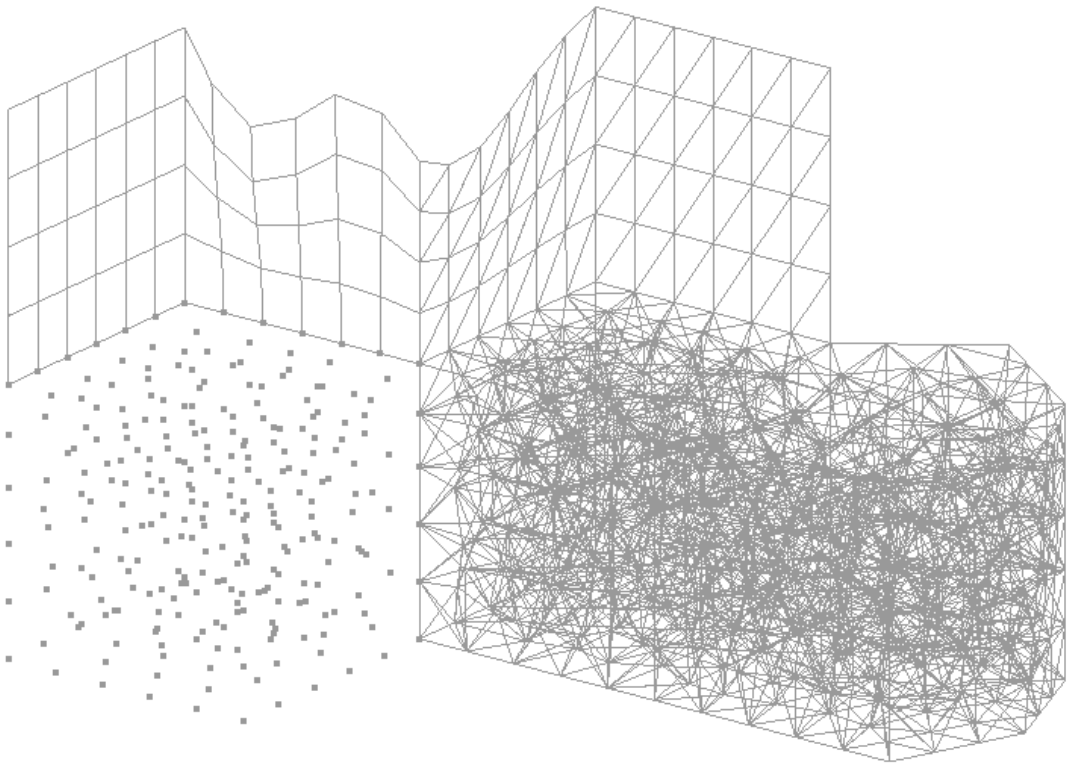


Figure 8. Structured mesh of quadrilateral and triangular elements on surfaces, with the option Symmetrical structured triangles not set.

---

[5] When selecting a line, GID automatically selects all lines parallel to it.

7.2.6 Generating structured meshes (volumes)

1 . To mesh volumes with a structured mesh, select the option **Mesh->Structured->Volumes->Assign number of cells**.
2 . Select volumes 1 and 2 and press **ESC**.
3 . A window appears in which to enter the number of divisions that the lines to be selected will have. Enter 6 and click **Assign**.
4 . Select lines of both volumes parallel to the **X** and **Z** axes. GiD automatically selects all the lines in each volume parallel to these in order to create the structured mesh. Press **ESC**.
5 . Another window appears in which to enter the number of divisions on the lines. Divide the lines parallel to the **Y** axis into 8 segments. Enter 8 and click **Assign**.
6 . Select an edge of volume 1 or 2 parallel to the **Y** axis and press **ESC** . Again, the line-division window comes up. Since we have already finished the assignments, click **Close**.

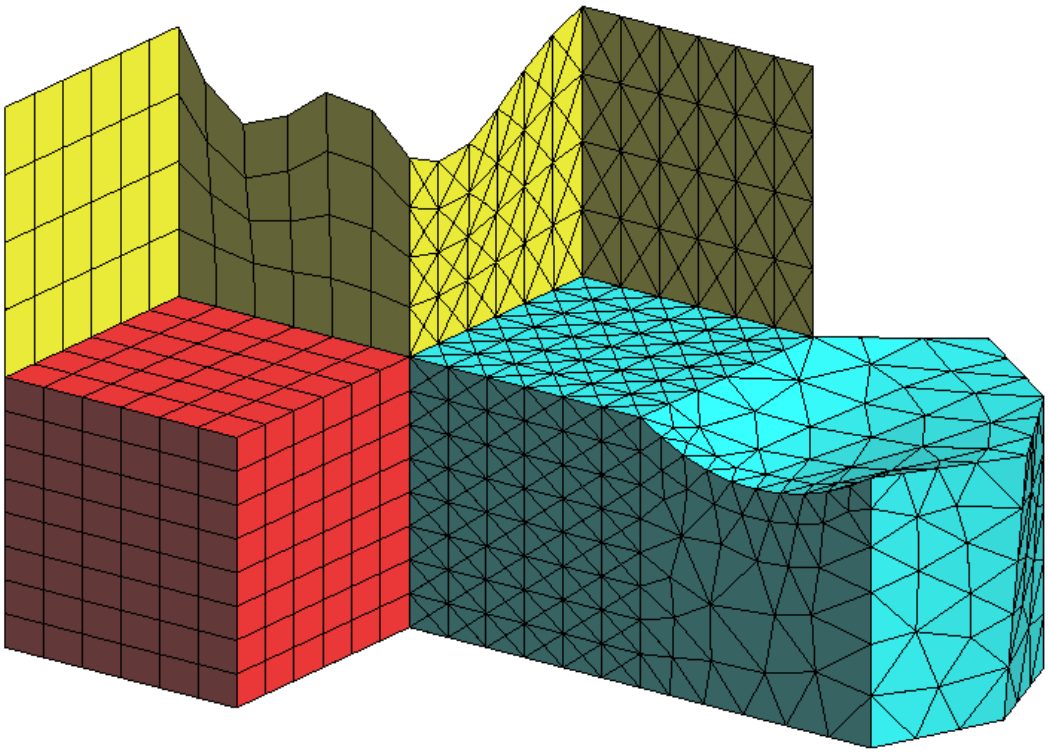


Figure 9. Structured volume mesh of hexahedra an tetrahedra.

7 . For structured volumes, GiD generates tetrahedron meshes by default, but hexahedron structured meshes can also be assigned. Let's assign the element type that we wish to volume 1 and 2. Select **Mesh->Element type->Tetrahedra**,then select volume number 2 and press **ESC**. Select **Mesh->Element type->Hexahedra**, then select volume number 1 and press **ESC**.

8 . Select **Mesh->Generate mesh**.

9 . A window appears asking whether the previous mesh should be eliminated. Click **Yes**.

10 . Another window comes up in which to enter the maximum element size. Leave the default value unaltered and click **OK** . The result is the mesh shown in Figure 10.

11 . GiD can obtain volume structured meshes made of hexahedra, tetrahedra or prisms. As can be seen in Figures 9 and 10, there are two kinds of tetrahedron structured mesh: the one shown in Figure 9 is obtained when the option **Utilities->Preferences->Meshing->Symmetrical structured->tetrahedra** is set. If this option is not set, the mesh presented in Figure 10 is produced (with fewer nodes than if using the previous option;also, it is not topologically symmetrical).

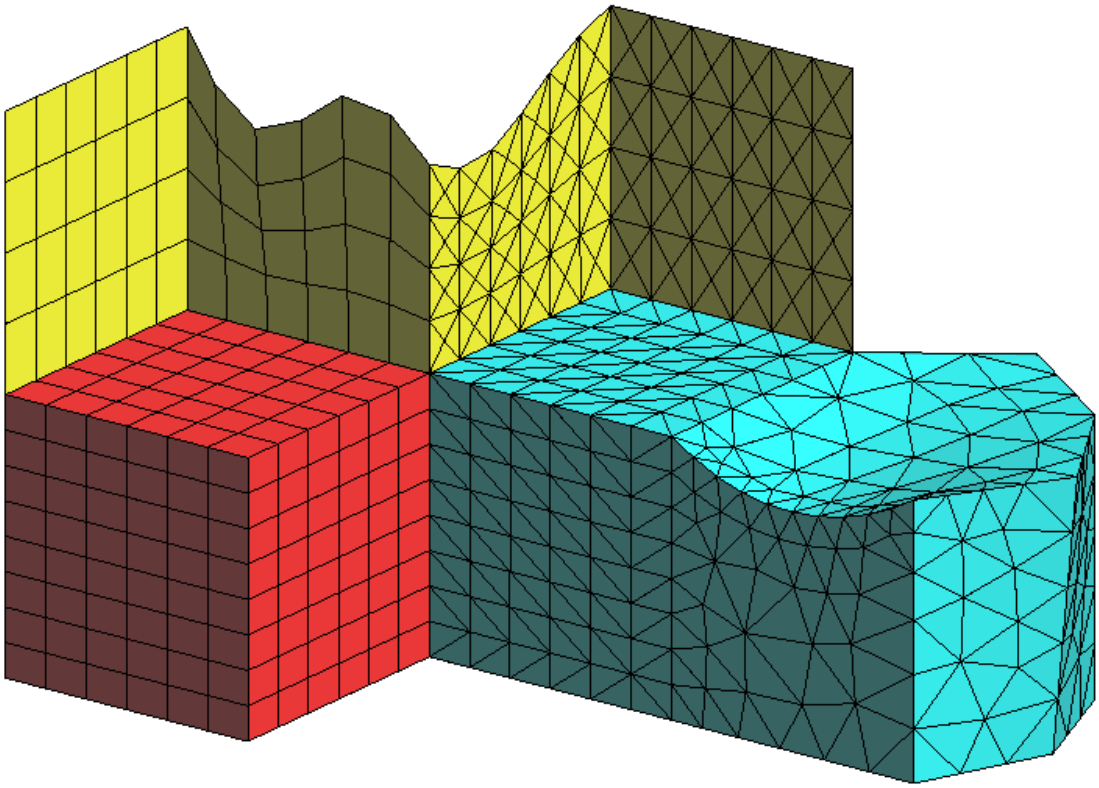


Figure 10. Structured volume mesh of tetrahedra with the option Symmetrical structured tetrahedra not set.

7.2.7 Generating semi-structured meshes (volumes)

1 . To mesh volumes with a semi-structured mesh, select the option **Mesh->SemiStructured->Volumes**.
2 . A window appears in which to enter the number of divisions for the direction in which it is structured (the prismatic one). Enter 8 and click **Assign**.
3 . Select volume 3 and press **ESC**. As volume 3 is prismatic in one direction only (i.e. parallel to **Y** axis) GiD will automatically detect this fact and will select it to be the direction in which the semi-structured volume mesh is structured.
4 . Another window appears in which to enter the number of divisions in the direction of the structure. In this case we do not want to select any more volumes, so click **Close**.
5 . Select **Mesh->Generate mesh**.
6 . A window appears asking whether the previous mesh should be eliminated. Click **Yes**.
7 . Another window appears in which to enter the maximum element size. Leave the default value unaltered and click **OK**. The result is the mesh shown in Figure 11.

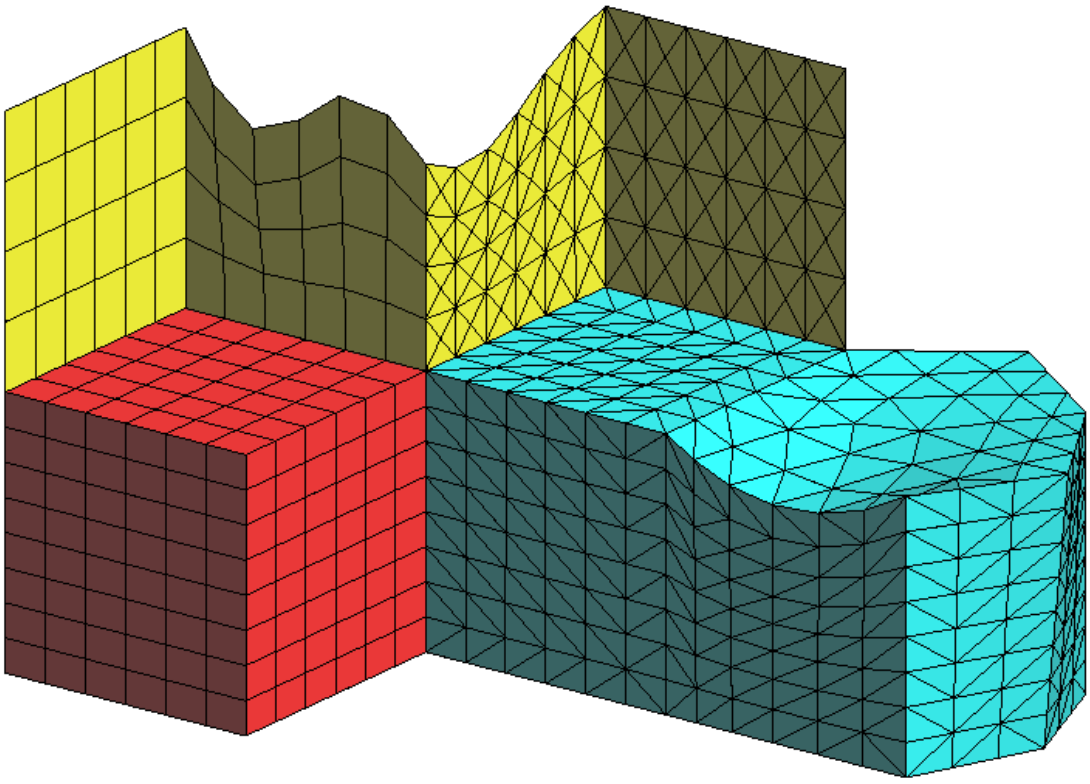


Figure 11. Semi-structured volume mesh of tetrahedra.

As can be seen, volume 3 has been meshed with tetrahedra. Semi-structured volumes are meshed with prisms, by default. However, in this case it was not possible because of volume 2, which has tetrahedra assigned and shares one surface with volume 3. In the following steps a hexahedron mesh is produced.

8 . Select **Mesh->Element type->Hexahedra.**
9 . Select volumes 2 and 3 and press **ESC**.
10 . Select **Mesh->Generate mesh**.
11 . A window opens asking whether the previous mesh should be eliminated. Click **Yes**.
12 . Another window appears in which to enter the maximum element size. Leave the default value unaltered and click **OK**. The result is the mesh shown in Figure 12.

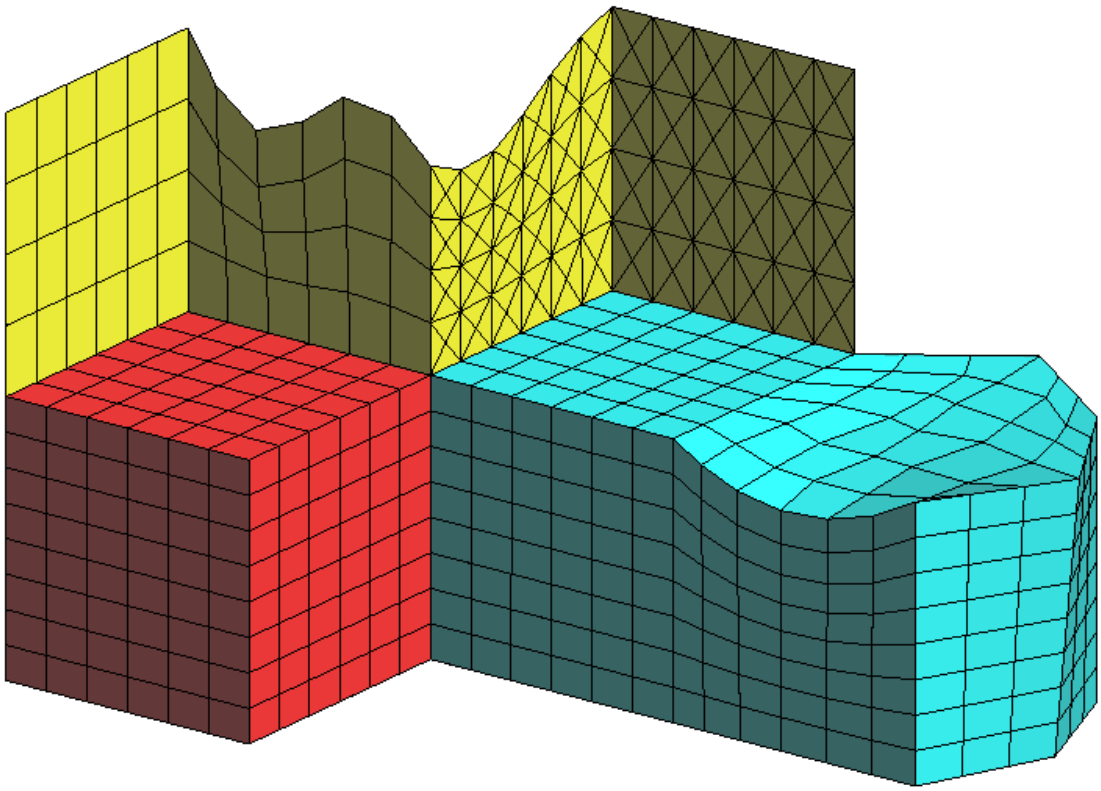


Figure 12. Semi-structured volume mesh of hexahedra.

In case of volume number 3 there is only one direction in which it can possibly be structured (i.e. in the direction of the prism). If the volume is prismatic in more than one direction, there are two ways to choose between them: selecting one top surface (**Mesh->SemiStructured->Set->Master surface**) or the direction of the structure (**Mesh->SemiStructured->Set->Structured direction**). The following example explains this procedure.

13 . Select the option **Mesh->SemiStructured->Volumes**.
14 . A window opens in which to enter the number of divisions in the structured direction (prismatic). Enter 6.
15 . Select volume 1 and press **ESC**.
16 . Another window appears, click **Close**.
17 . Select **Mesh->SemiStructured->Set->Structured direction**.
18 . Select one line parallel to the **X** axis of volume number 1 (for example line number 11) and press **ESC**.
19 . Select **Mesh->Unstructured->Assign entities->Surfaces.**
20 . Select surfaces 1 and 6 and press **ESC**.
21 . Select **Mesh->Generate mesh**.
22 . A window opens asking whether the previous mesh should be eliminated. Click **Yes**.
23 . Another window appears in which to enter the maximum element size. Leave the default value unaltered and click **OK**.

7.2.8 Concentrating elements and assigning sizes

1 . Select **Mesh->Structured->Lines->Concentrate elements**.

2 . Select some structured lines, for example line 43. Press **ESC**.

3 . A window comes up in which to enter two values for the concentration of elements. Positive values concentrate the elements and negative values spread them. Enter 0.5 as **Start Weight** and –0.5 as **End Weight**[10]. Click **Ok** and press **ESC**.

4 . Select **Mesh->Generate mesh**.

5 . A window opens asking whether the previous mesh should be eliminated. Click **Yes**.

6 . Another window appears in which to enter the maximum element size. Leave the default value unaltered. The result is the mesh shown in Figure 13.

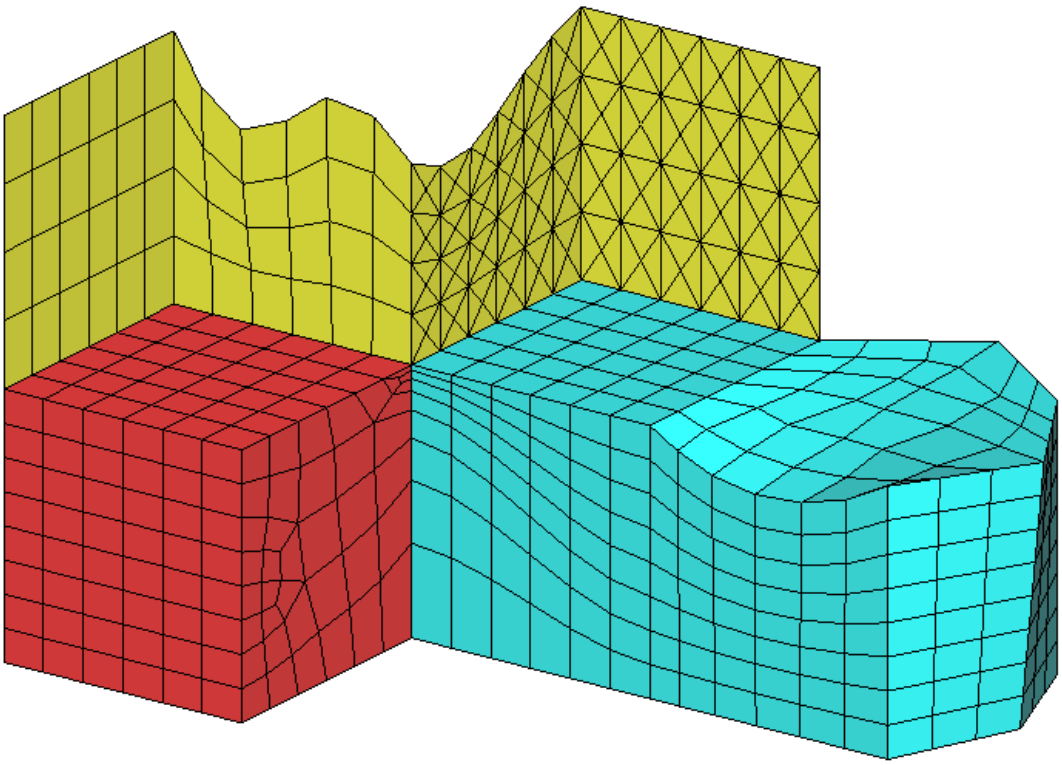


Figure 13. Concentration of elements on line 43.

It is also possible to assign sizes to geometrical entities, so that mesh elements can be concentrated in certain zones. In the following steps some examples are given.

7 . Select **Mesh->Unstructured->Assign sizes on points.**

8 . A window appears in which to enter the size to be assigned to points. Enter 0.1 and click **Assign**.

9 . Select point number 15 and press **ESC**.

10 . Another window appears in which to enter the size to be assigned to points. In this case, we donot want to assign sizes to any other points, so click **Close**.

11 . Select **Mesh->Unstructured->Assign sizes on lines.**

12 . A window appears in which to enter the size to be assigned to lines. Enter 0.5 and click **Assign**.

13 . Select line number 25 and press **ESC**.

14 . Another window appears in which to enter the size to be assigned to lines. In this case, we do not want to assign sizes to any more lines, so click **Close**.

15 . Select **Mesh->Generate mesh**.

16 . A window appears asking whether the previous mesh should be eliminated. Click **Yes**.
17 . Another window appears in which the maximum element size should be entered. Leave the default value unaltered and click **OK**. The result is not the desired, we just get the previous mesh. This is because surrounding surfaces and lines are structured, so they do not have enough freedom to achive the given sizes.
18 . Select **Mesh->Unstructured->Assign entities->Surfaces.**
19 . Select Surfaces 26 and 12. Press **ESC**.
20 . Select **Mesh->Unstructured->Assign entities->Lines.**
21 . Select lines 48, 26 and 27. Press **ESC**.
22 . Select **Mesh->Generate mesh**.
23 . A window appears asking whether the previous mesh should be eliminated. Click **Yes**.
24 . Another window appears in which the maximum element size should be entered. Leave the default value unaltered and click **OK**. The result is the mesh shown in Figure 14.

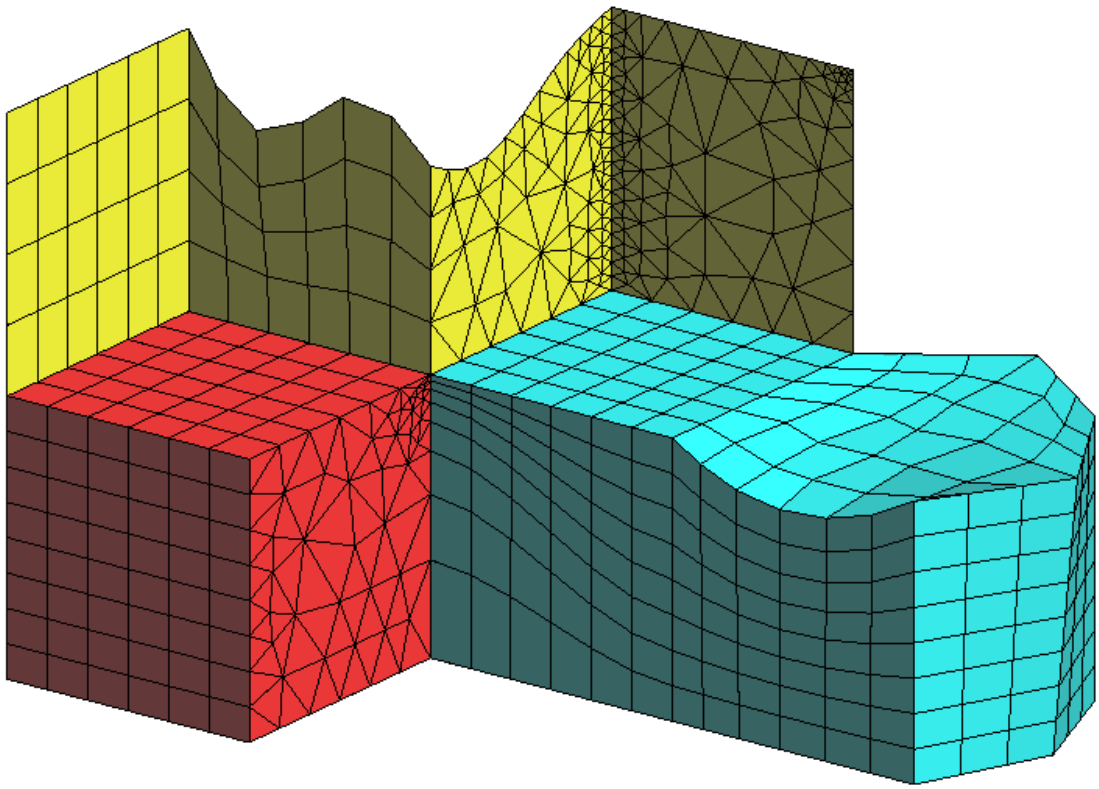


Figure 14. Unstructured size assigned in a point and a line.

---

[10] Start Weight and End Weight refer to the start point and end point of the line, oriented as it is drawn when you select it.

7.2.9 Generating the mesh using quadratic elements

Enlarge one area of the mesh with the zoom.

1 . Select **Label->All in->Points** . The result is shown in Figure 15.

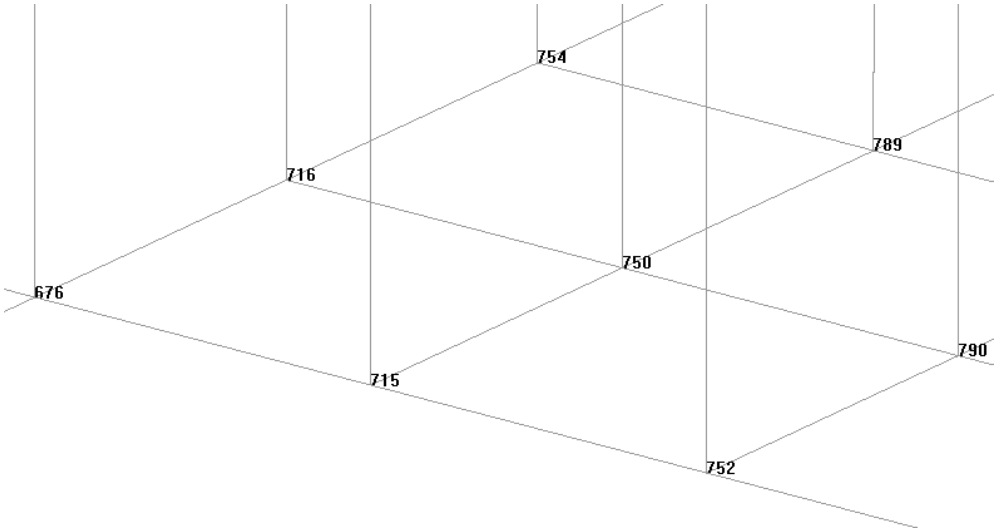


Figure 15. Each number identifies a node. There is a node for each element vertex.

2 . The node identifiers created by generating the mesh appear on the screen. There is one identifier for each vertex of each element.

3 . Select **Mesh->Quadratic type->Quadratic.**

NOTE : By default GiD meshes with first degree (linear) elements. To find out which mode GiD is working in, go to **Mesh->Quadratic type**.

4 . Select **Mesh->Generate mesh**.

5 . A window opens asking whether the previous mesh should be eliminated. Click **Yes**.

6 . Another window appears in which the maximum element size should be entered. Leave the default value unaltered and click **OK**.

7 . Once the mesh has been generated, select **Label->All in->Points** . The result is shown in Figure 16. Now, there are not only nodes at the vertices, but also at the midpoints of the edges of the elements.

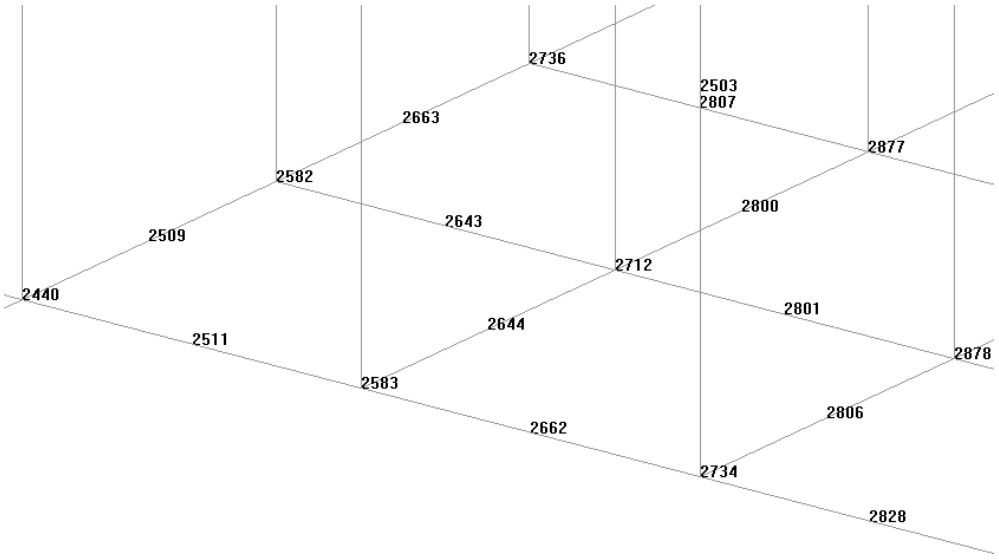


Figure 16. Each number identifies a node. There is a node at each element vertex and at the midpoint of each edge.

8 . Select **Mesh->Quadratic type->Quadratic9**.

9 . Select **Mesh->Generate mesh**.

10 . A window opens asking whether the previous mesh should be eliminated. Click **Yes**.

11 . Another window appears in which the maximum element size should be entered. Leave the default value unaltered and click **OK**.

12 . Select **Label->All in->Points** (see Figure 17).

13 . Notice that the four-sided elements (quadrilaterals) also have a node in the center, in addition to the nodes at the vertices and midpoints of the edges. Similarly, hexahedra also have a node at their center point.

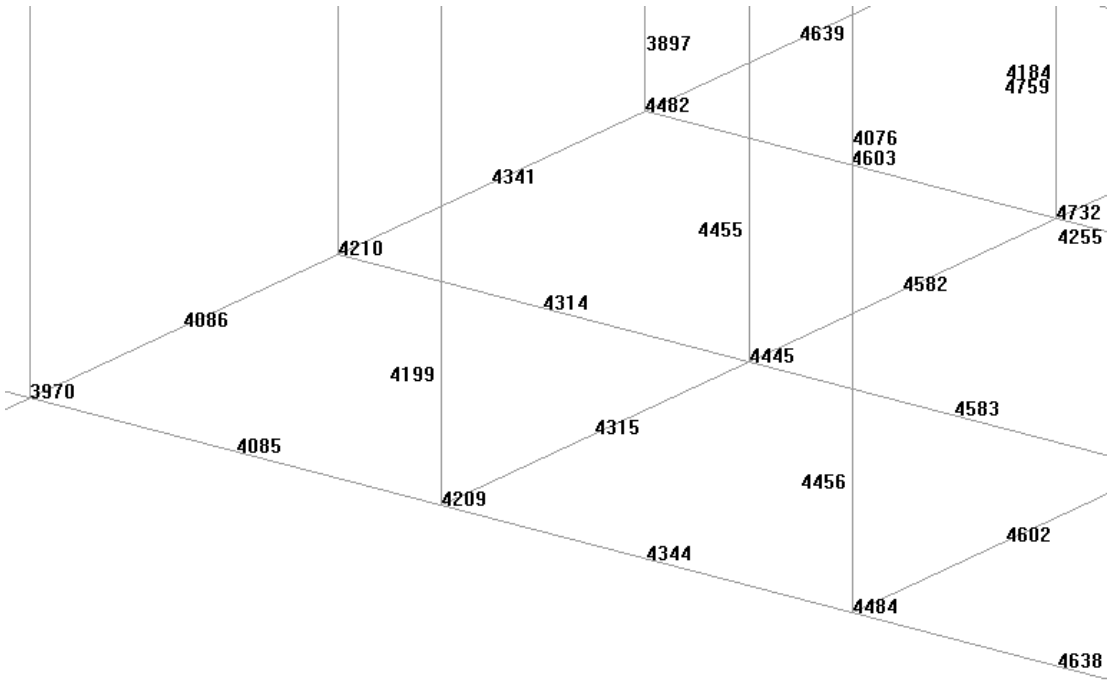


Figure 17. Each number identifies a node. There is a node at each vertex, at the midpoint of each edge and in the center of quadrilaterals and hexahedra.