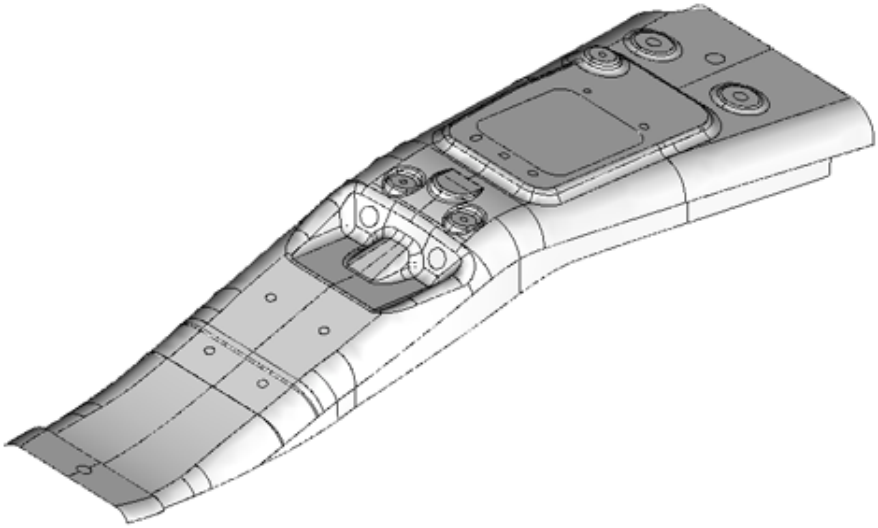
9 CAD CLEANING OPERATIONS

IMPORTING FILES

The objective of this case study is to see how GiD imports files created with other programs. The imported geometry may contain imperfections that must be corrected before generating the mesh.

For this study an IGES formatted geometry representing a stamping die is imported. These steps are followed:

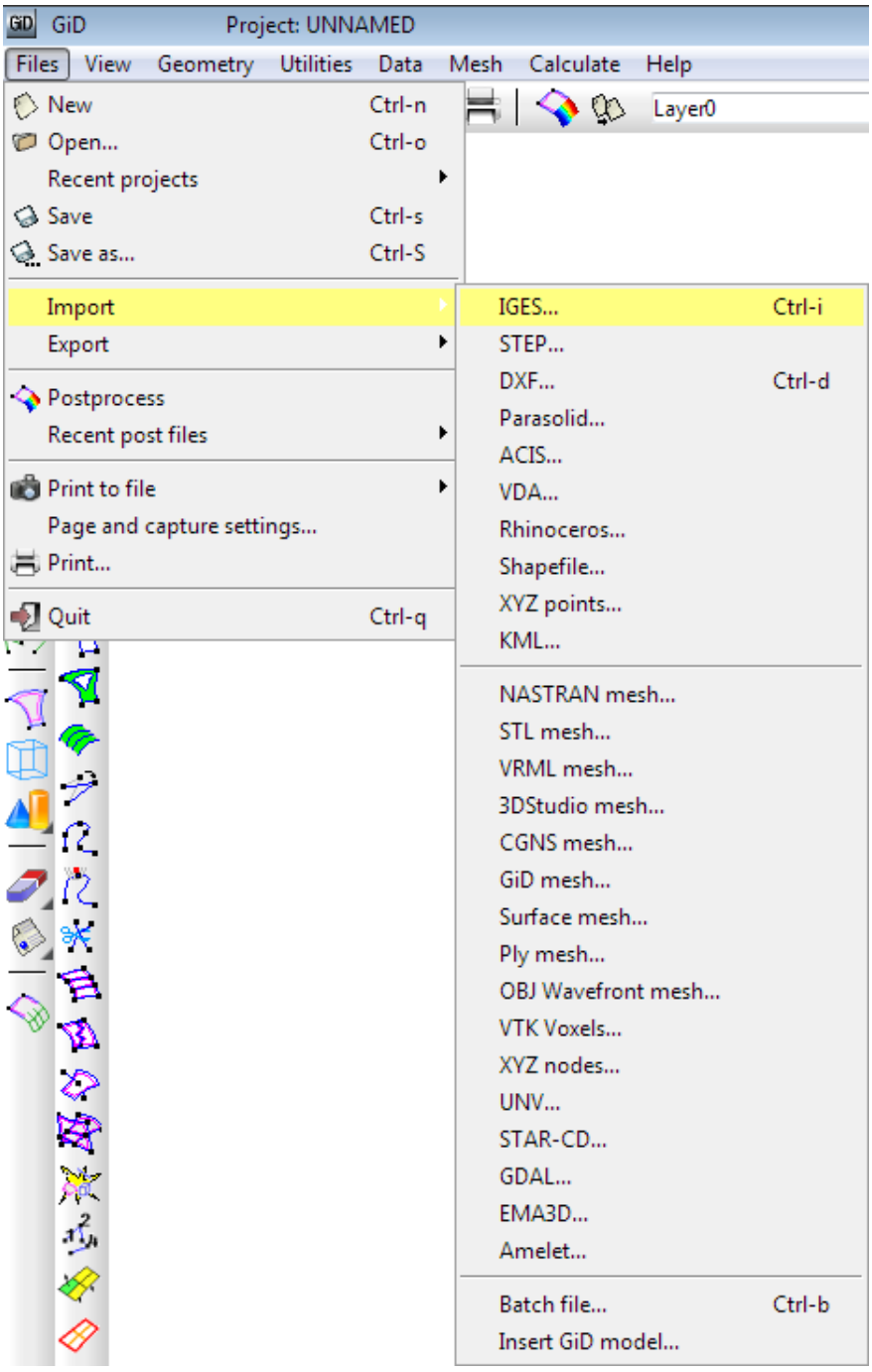
- Importing an IGES-formatted file to GiD
- Correcting errors in the imported geometry and generating the mesh
- Generating a conformal mesh and a non-conformal mesh



Pice provided by courtesy of PSA DEGAD-MAC AIE

9.1 Importing on GiD

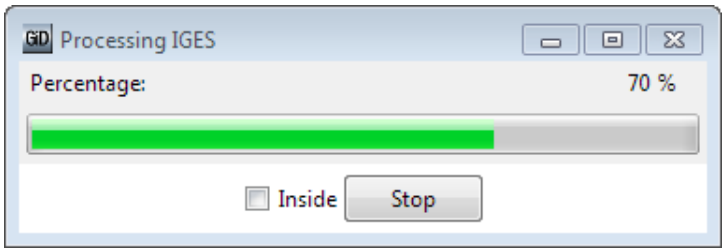
GiD is designed to import a variety of file formats. Among them are standard formats such as IGES, DXF, or VDA, which are generated by most CAD programs. GiD can also import meshes generated by other programs, e.g. in NASTRAN or STL formats.



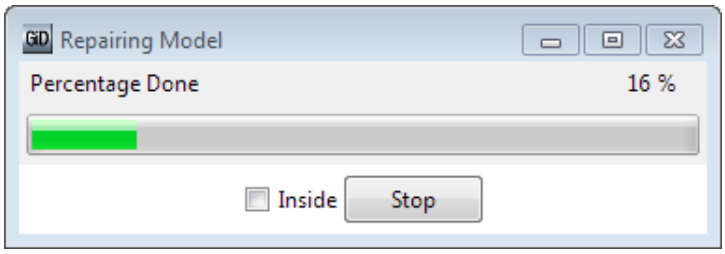
The file importing process is not always error-free. Sometimes the original file has incompatibilities with the format required by GiD. These incompatibilities must be overcome manually. This example deals with various solutions to the difficulties that may arise during the importing process.

9.1.1 Importing an IGES file

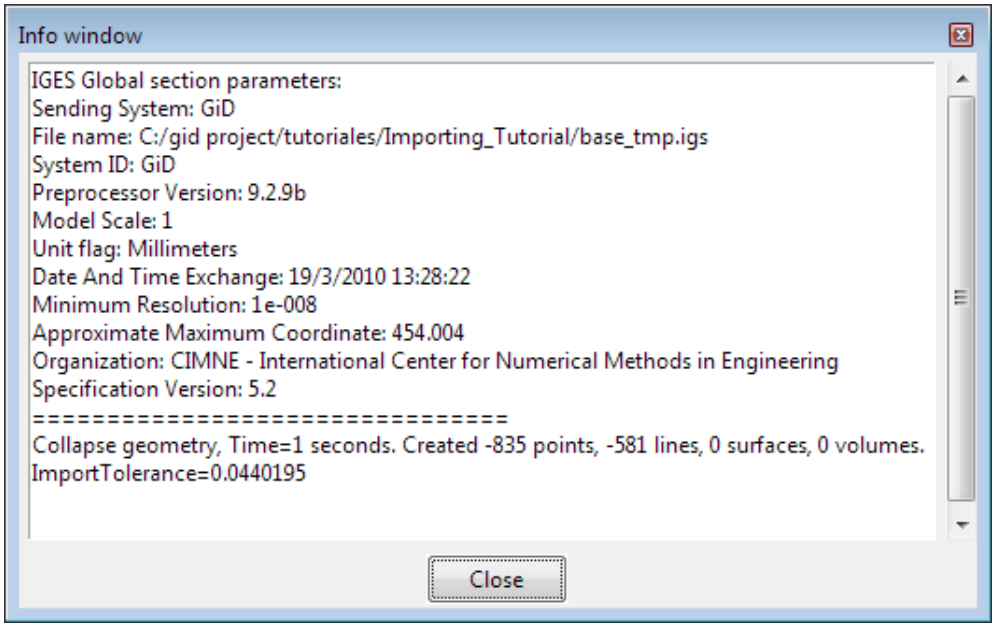
1 . Select **Files->Import->IGES ...**
2 . Select the IGES-formatted file "base.igs" and click **Open**.



Reading the file

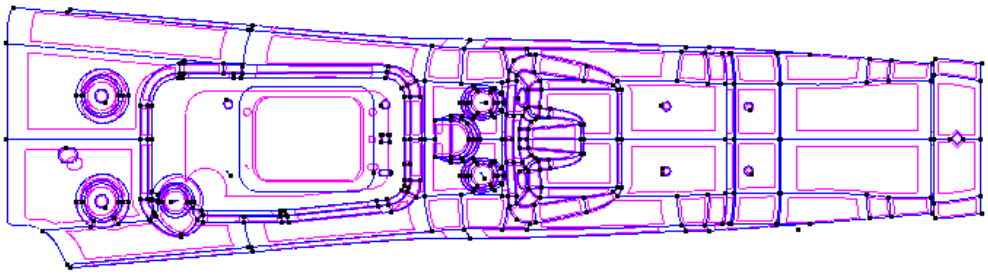


Repairing the model



Importing process information

After the importing process, the IGES file that GiD has imported appears on the screen.

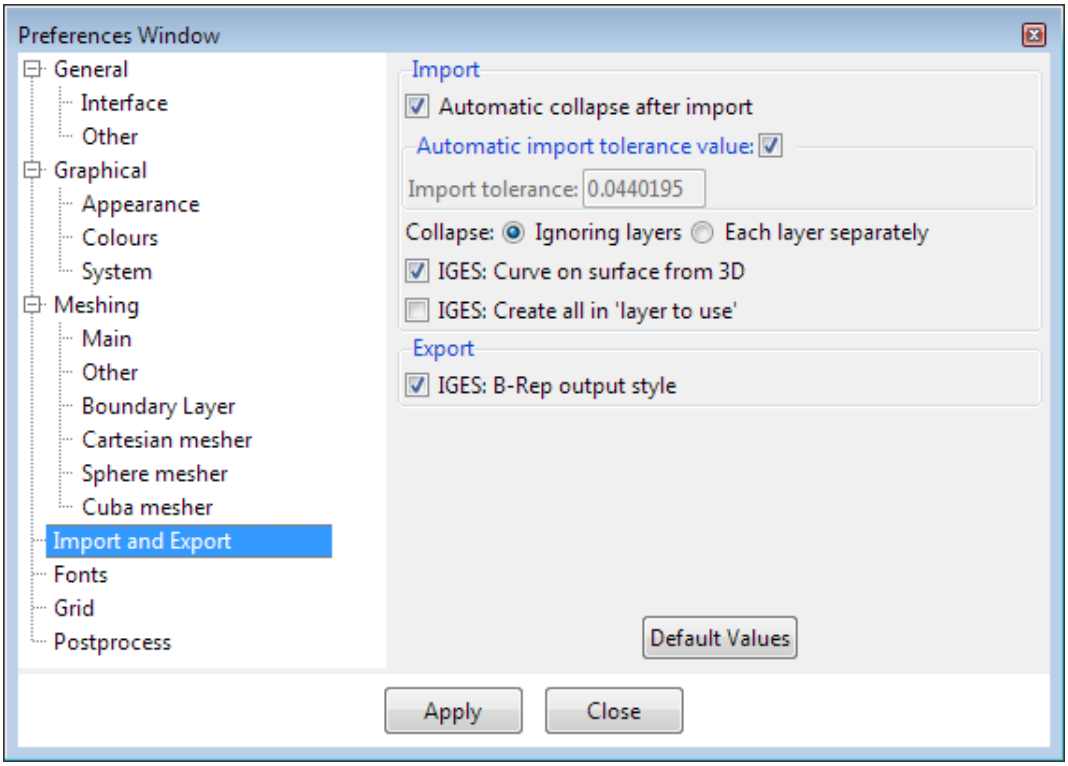


File "base.igs" imported by GiD.

NOTE: One of the operations in the importing process is repairing and collapsing the model. We say that two entities collapse when, the distance between them being less than the **Import Tolerance**, they become one.

The **Import Tolerance** value may be modified by going to the **Utilities** menu, opening **Preferences**, and selecting **Import and Export** from the tree. By default, the **Automatic import tolerance value** is selected. With this option selected, GiD computes an appropriate value for the **Import Tolerance** based on the size of the geometry.

Collapsing the model may also be done manually. This option is found in **Geometry->Edit->Collapse->Model**.



The preferences window

9.2 Correcting errors in the imported geometry

The great diversity of versions, formats, and programs frequently results in differences (errors) between the original and the imported geometry. With GiD these differences might give rise to imperfect meshes or prevent meshing altogether. In this section we will see how to detect errors in the imported geometry and how to correct them.

Importing the same file with different versions of GiD might produce slight variations in the results. For this reason from now we will use a project that contains the original IGES file translated into GiD format.

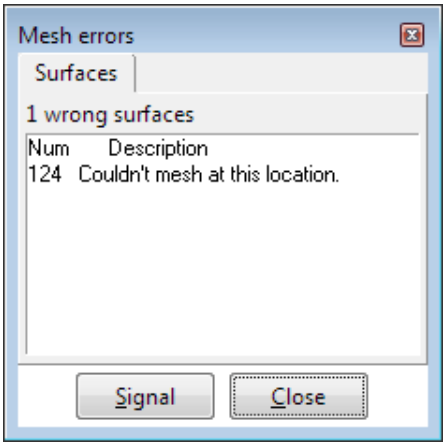
1 . Select **Files->Open...**
2 . If a dialog window appear asking to save changes to the project, click **No**.

9.2.1 Meshing by default

1 . Select **Mesh->Generate Mesh**.

A window comes up in which to enter the maximum element size for the mesh to be generated. Leave the default value provided by GiD unaltered and click **OK**.

When the **GiD** finishes the meshing process, an error message appears. This error is due to a defect in the imported geometry. As the window shows, there have been errors meshing surface number 124.



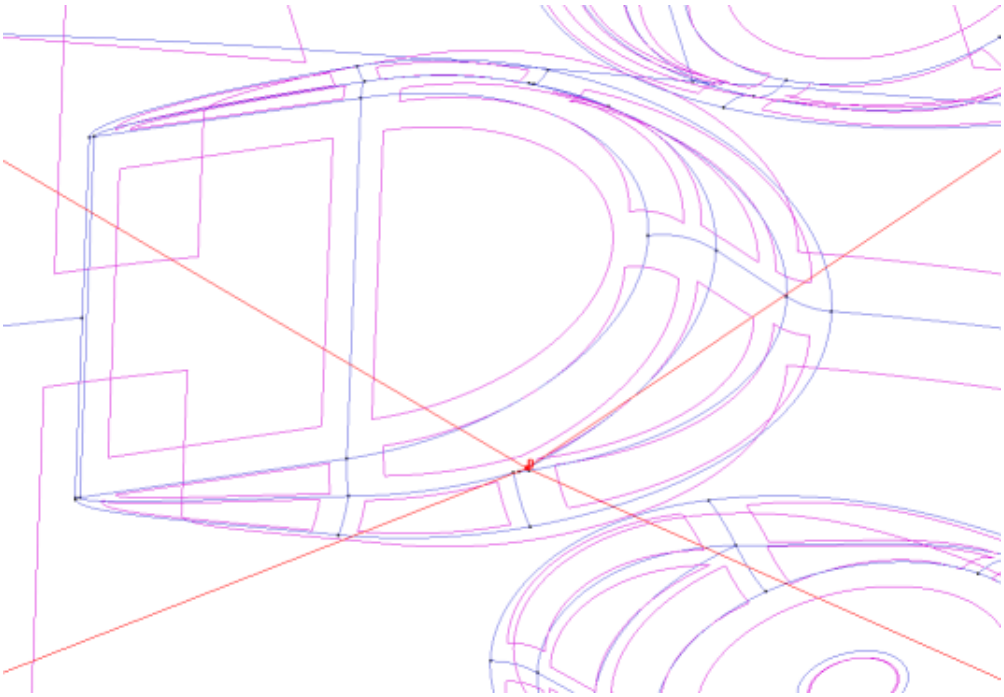
Dialog warning window of meshing errors

In this part of the tutorial we focuson repairing surface number 124.

To locate surface 124, select the line "**124 Couldn't mesh at this location**" in the dialog box and press the **Signal** button (the same effect is obtained by double-clicking over the message with the left mouse button).

NOTE : If user clicks the right button over a message in the Mesh Errors window, three options are displayed: "Signal problematic point", "More help…" or "List…" The first option is the same as the Signal button, while the "List…" option presents a list of the problematic geometrical entities to make selection easier when performing some common procedures (like sending the entities to a separate layer, erasing the entities, etc…). The "More help…" option gives advice about to correct the geometrical model so the mesh can be generated.

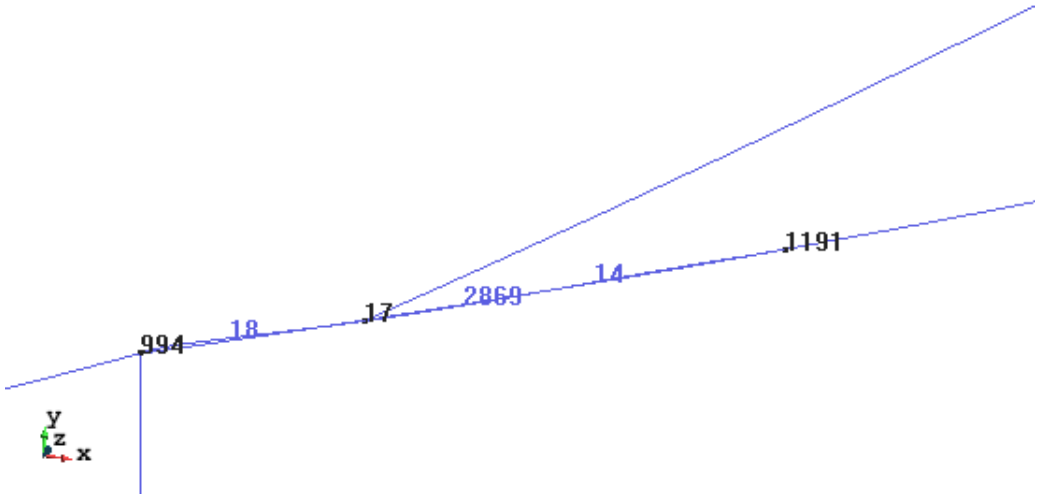
NOTE : *The* **Mesh Errors** *window can be recovered while dealing with the model by selecting the "***Show errors...***" option in the* **Mesh** *menu*.



Signaling surface 124

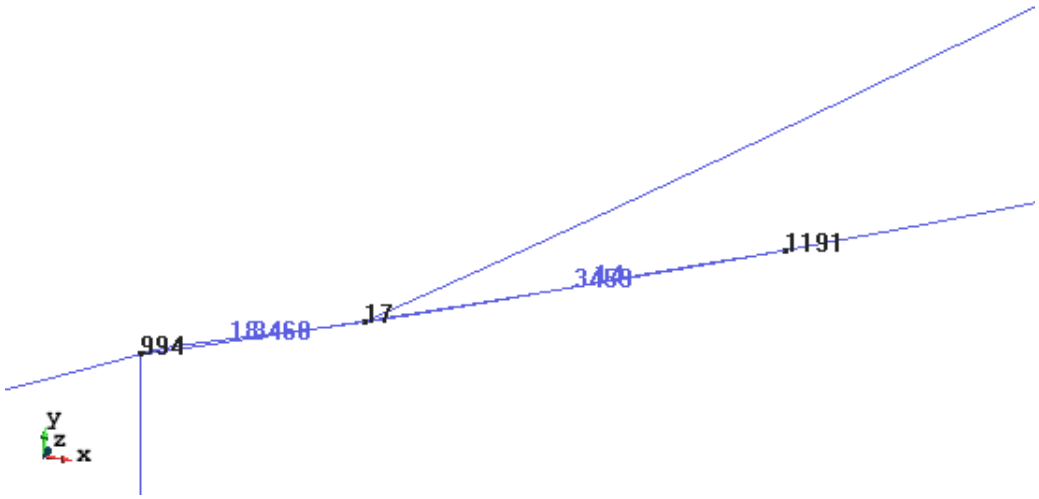
9.2.2 Correcting surfaces

1 . With the **View->Zoom->In** option in the menu or **Zoom->In** on the mouse menu, magnify the zone around surface 124.



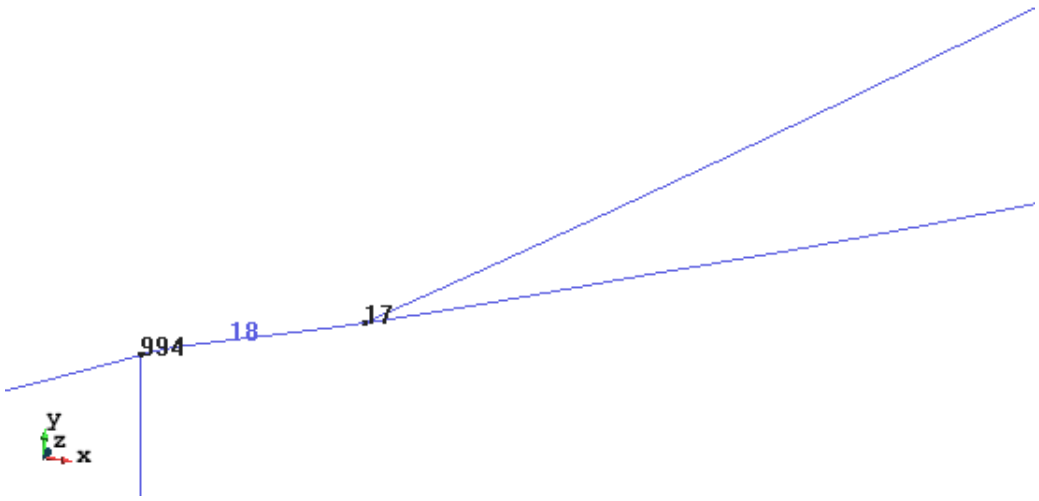
An enlargement of the zone around surface 124

2 . Several line segments are superimposed over each other, thus creating an incorrect surface boundary. Select **Geometry->Edit->Divide->Lines->Near point** and then select point 17 (to select it, go to **Contextual** in the mouse menu, then select the option **Join Ctrl-a**). Point 17 is the point at which to make the cut.



The zone after cutting line 2869 at point 17

3 . Now that the lines are precisely connected, a local collapse may be executed. Select **Geometry->Edit->Collapse->Lines**. Then select the lines that appear on the screen and press **ESC**.



The situation after collapsing the lines

4 . After the collapse, the surface boundary is correct and the surface may be drawn with the new boundary. The labels are no longer needed, so click **Label->Off** in the mouse menu.

5 . Select **Geometry->Create->NURBS surface->Trimmed**. Select surface 124. Then select the lines defining the recently repaired boundary. Press **ESC** twice.

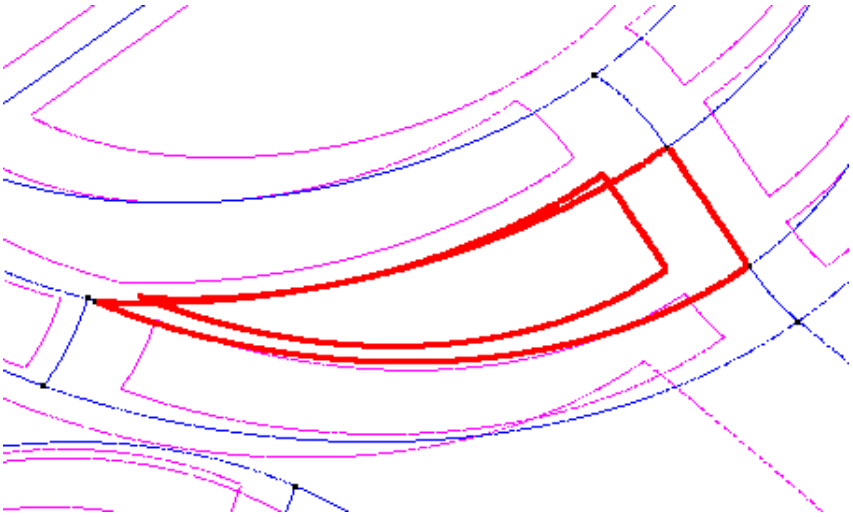


Figure 11. Surface 123 with its new boundary.

6 . Select **Geometry->Delete->Surfaces**. Select surface 124 and press **ESC**.

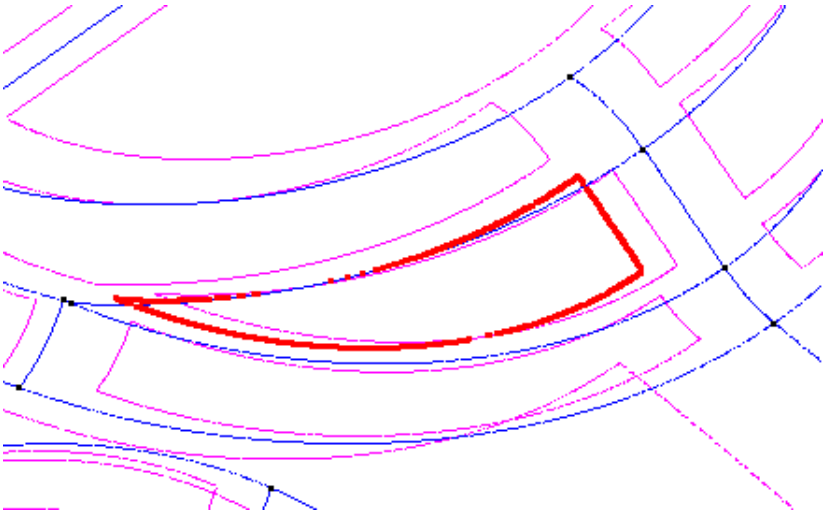
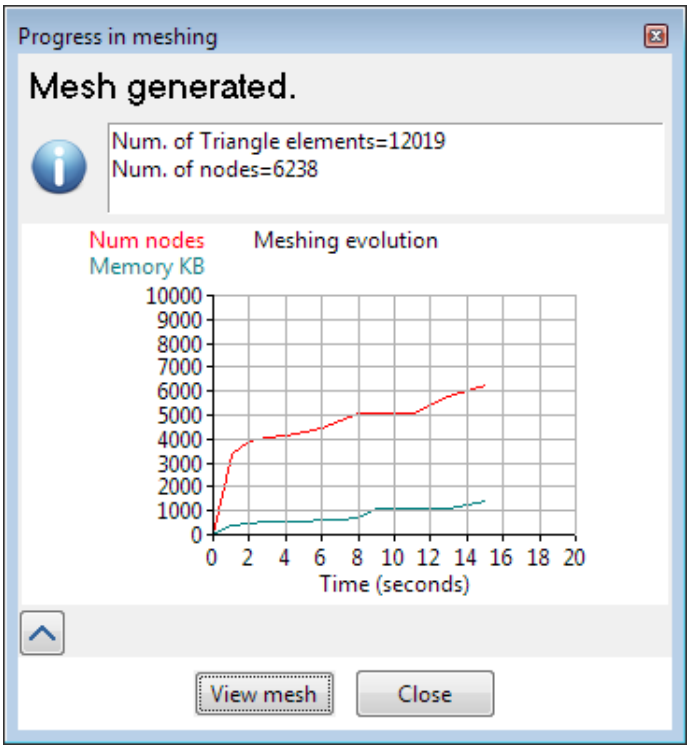


Figure 12. The surface to be eliminated.

7 . To begin the second example in this section, mesh the geometry again with **Mesh->Generate Mesh**.

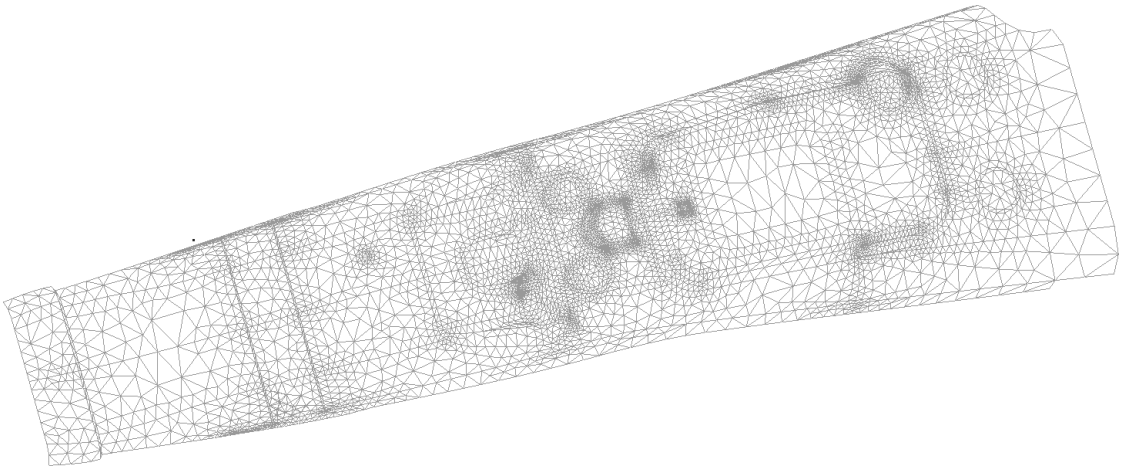
8 . A window comes up in which to enter the maximum element size for the mesh to be generated. Leave the default value provided by GiD and click **OK**.

The mesh generating process may be carried out with no further errors found.



indow with information about the generated mesh

9 . The imported piece is now meshed.



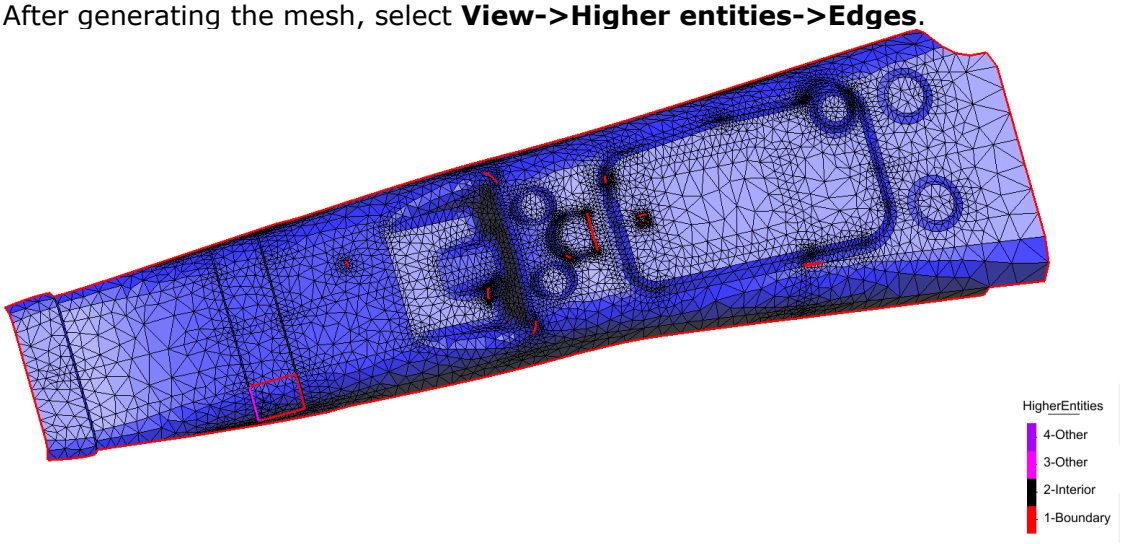
The mesh of the imported geometry

9.3 The conformal mesh and the non-conformal mesh

In the previous section, after correcting some errors, we were able to mesh the imported geometry, thus obtaining a non-conformal mesh. A conformal mesh is one in which the elements share nodes and sides. To achieve this condition, contiguous surfaces (of the piece) must share lines and points of the mesh. Most calculating modules require conformal meshes; however, some modules accept non-conformal meshes. A non-conformal mesh normally requires less computation time since it generates fewer elements.

9.3.1 Global collapse of the model

1 . After generating the mesh, select **View->Higher entities->Edges**.



The higher edges visualization

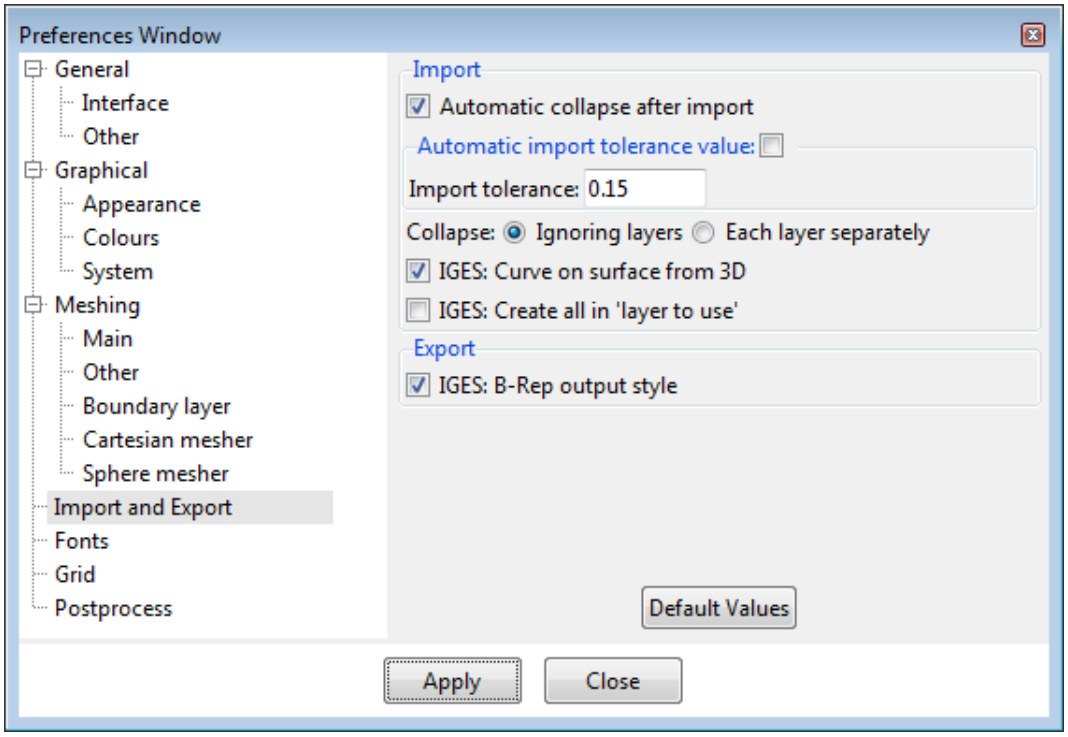
2 . Visualization of higer entities of edges shows that in the interior of the piece some surfaces are isolated.

3 . Press **ESC** to finish higer entities visualization.

4 . To generate a conformal mesh, first execute a global collapse of the model.

5 . The GiD collapse depends upon the **Import tolerance**. Two entities are collapsed (converted into one) when they are separated by a distance less than the **Import tolerance** parameter. To test this, enter a new value for the **Import tolerance** parameter.

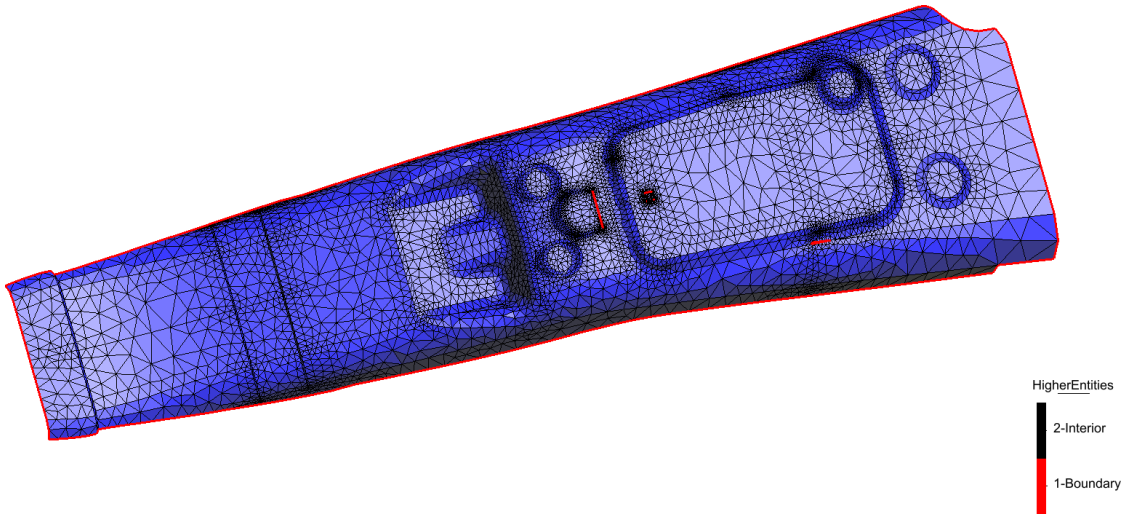
6 . Go to **Utilities->Preferences**, and select **Import and Export** branch. Uncheck the **Automatic import tolerance value** and enter 0.15 for the **Import tolerance value**. Click **Apply**.



The preferences window

7 . Select **Geometry->Edit->Collapse->Model**.
8 . A dialog window appears to confirm the seletion. Click **Ok**.
9 . Select **Mesh->Generate**. Erase the old mesh and use the default element size.
10 . Visualize the results with **View->Higher entities->Edges**.

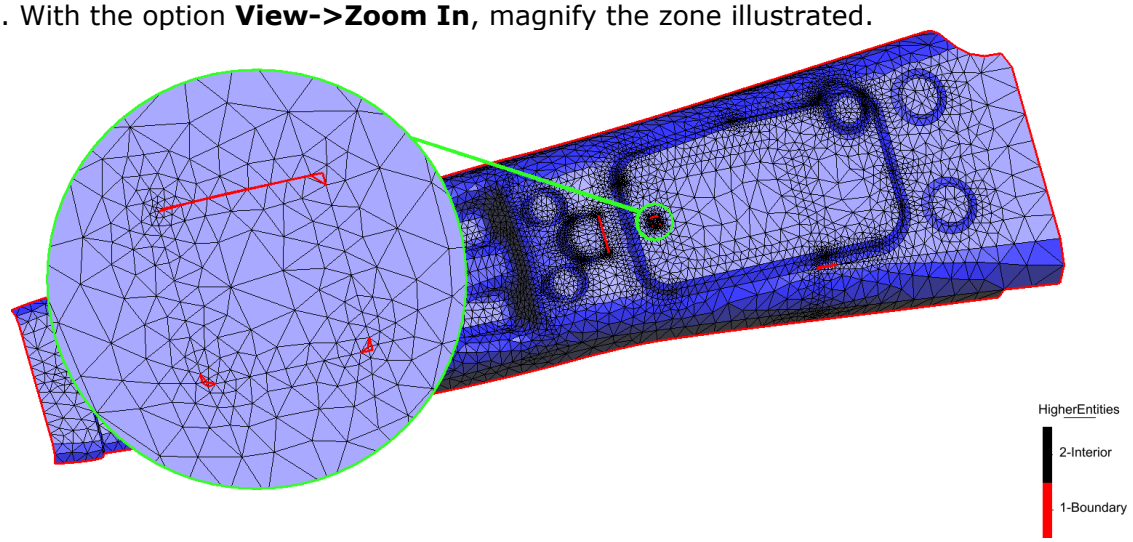
Some of the contiguous surfaces in the interior of the model have now being joined. However, there are still some surfaces that prevent the mesh from being completely conformal. These surfaces must be modified manually.



Higher edges after collapsing

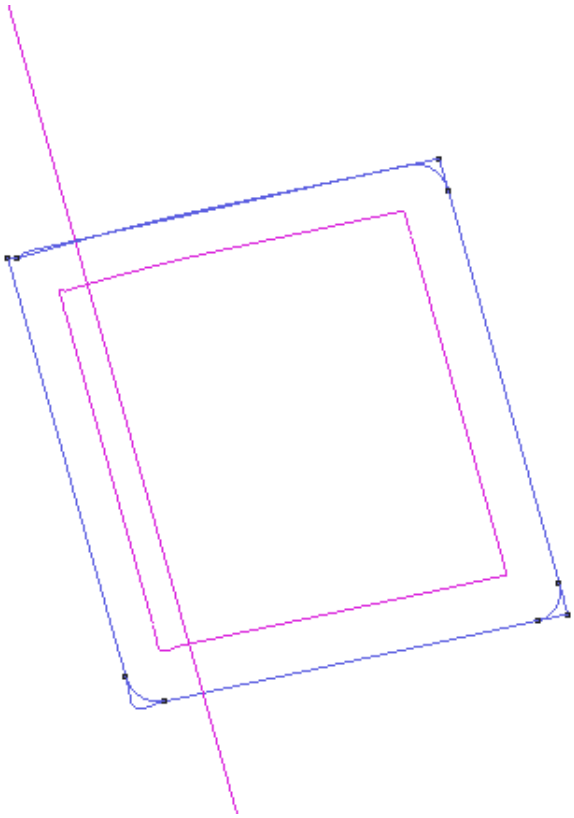
9.3.2 Correcting surfaces and creating a conformal mesh

1 . With the option **View->Zoom In**, magnify the zone illustrated.



Zone in the mesh to zoom in.

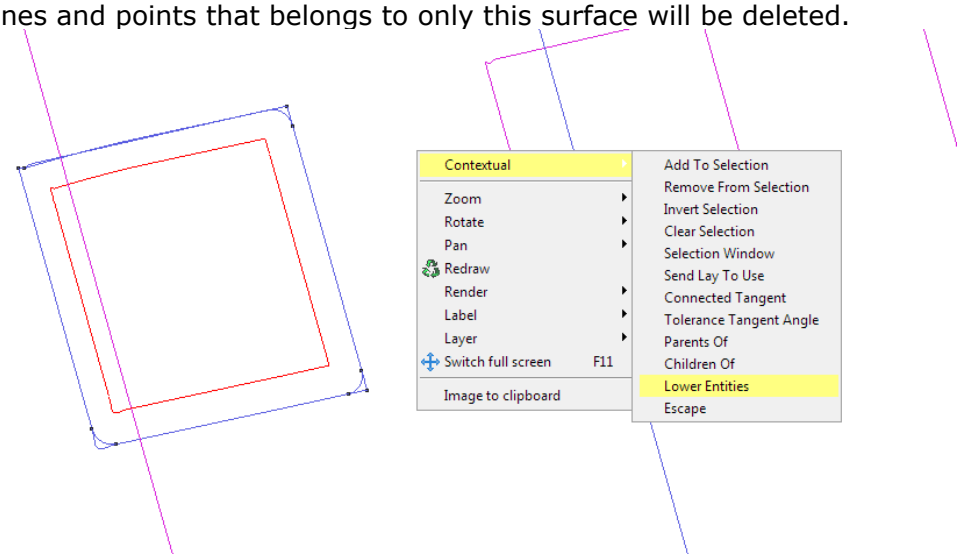
2 . Select **View->Mode->Geometry** to visualize the geometry of the piece.



The zone in geometry mode

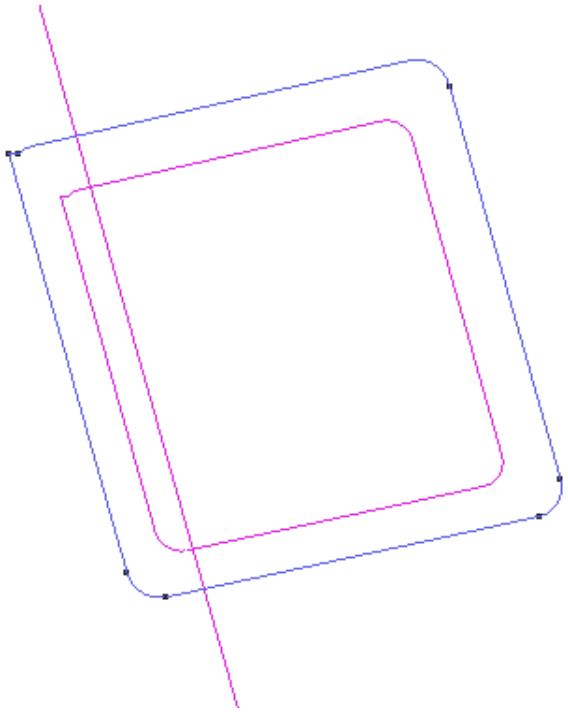
There is a rectangular surface that does not fit within the boundaries of a rounded-corner surface (a hole, in this case). We will suppose that the problematic surface is planar. This way, it can be erased and recreated in order to fit the rounded-corner boundary.

3 . Select **Geometry->Delete->Surfaces**. Select the problematic surface, but before pressing **ESC**. Go to **Contextual** menu and select **Lower Entities.** With this option, the surface and lines and points that belongs to only this surface will be deleted.



Deleting surface and its lines and points

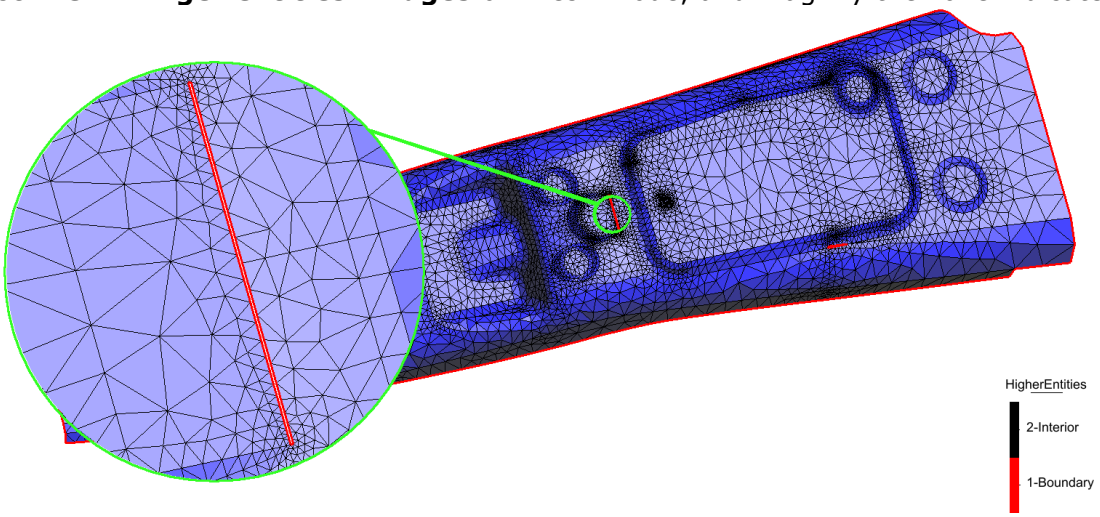
4 . With **Geometry->Create->NURBS surface->By contour** create a new surface. Select the lines defining the contourand press ESC.



result surface

5 . Visualize the mesh again **View->Mode->Mesh** You will see the previous mesh, the mesh its not recomputed. If you want to see the results of the first correction, the mesh must be regenerated with **Mesh->Generate mesh**.

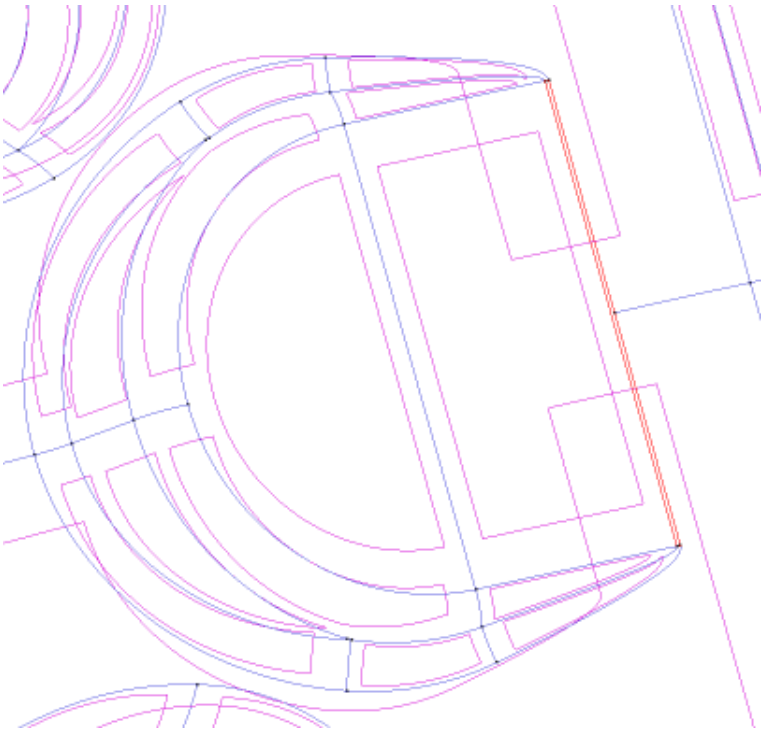
6 . Use **View->Higer entities->Edges** on mesh mode, and magnify the zone indicated.



Second part to repair

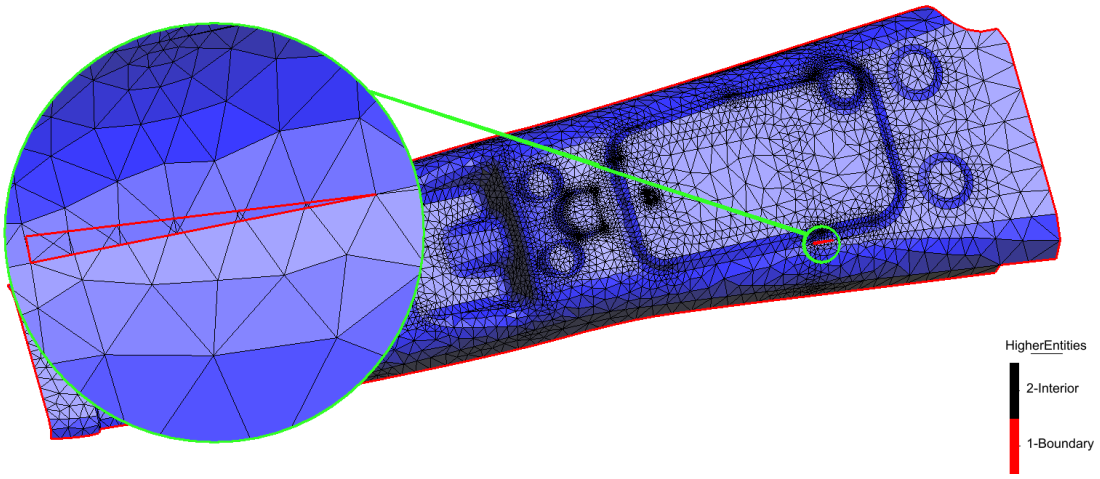
7 . Select **View->Mode->Geometry**.
In this example, the situation involves a contour of four lines that does not correspond to any real surface (of the piece). These lines were too far apart to be collapsed.

8 . Select **Geometry->Create->NURBS surface->By contour**. Select the lines. Press **ESC** twice.



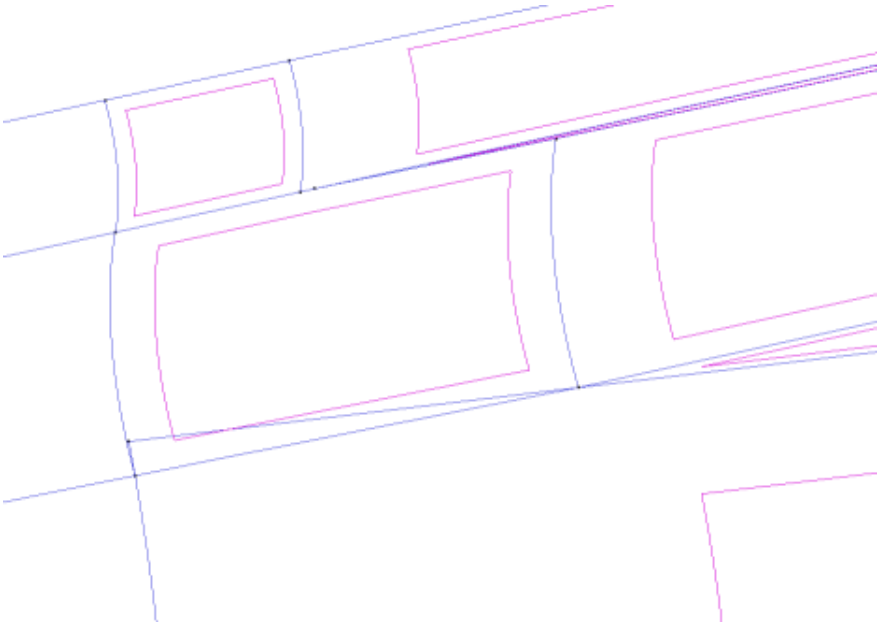
Contour lines that define the surface

9 . Visualize again higer entities **View->Higer entities->Edges** and magnify the zone indicated.



10 . Select **View->Mode->Geometry**.

There are two surfaces that overlap each other at one end.

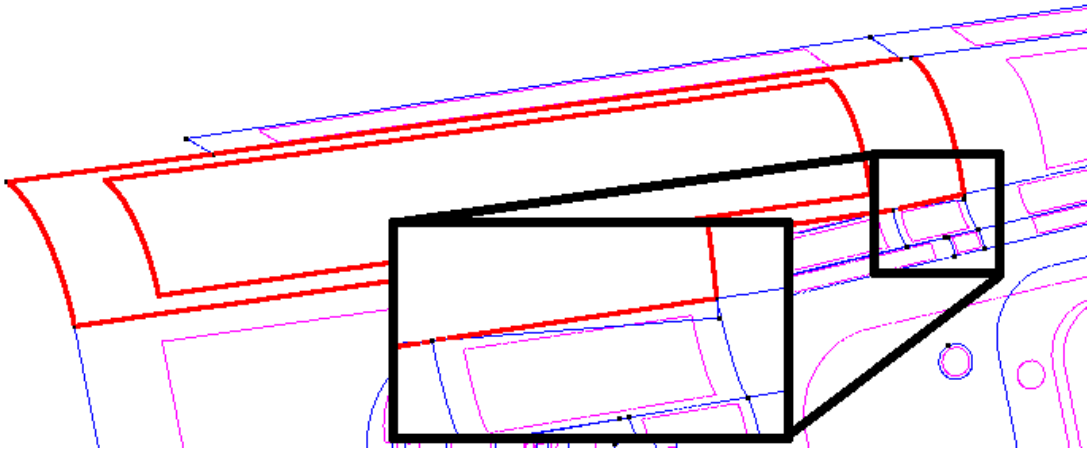


The magnified zone with two overlapping surfaces.

In this case the best solution for correcting the boundary is to trim the overlap.

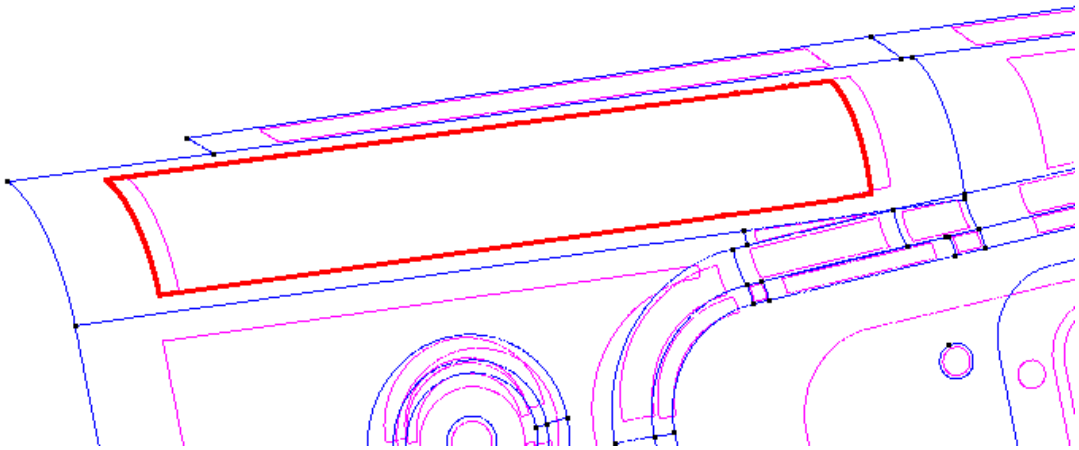
11 . Select **Geometry->Create->NURBS surface->Trimmed**.

12 . Select the surface to be trimmed. Then select the new boundary.



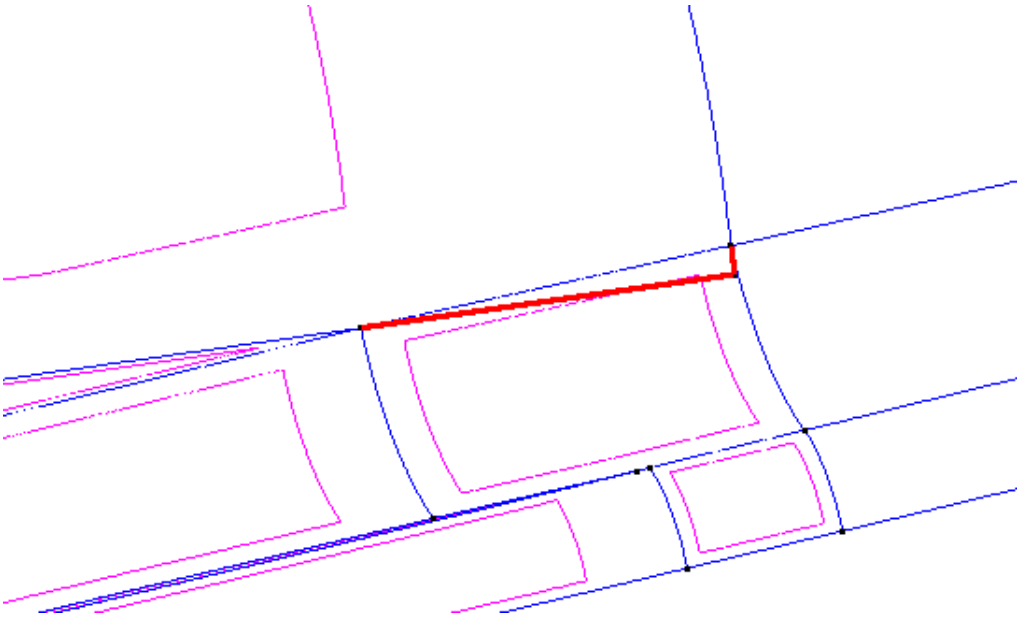
The surface to be trimmed and the new boundary.

13 . Select **Geometry->Delete->Surfaces**. Select the original surface. Press **ESC** twice.



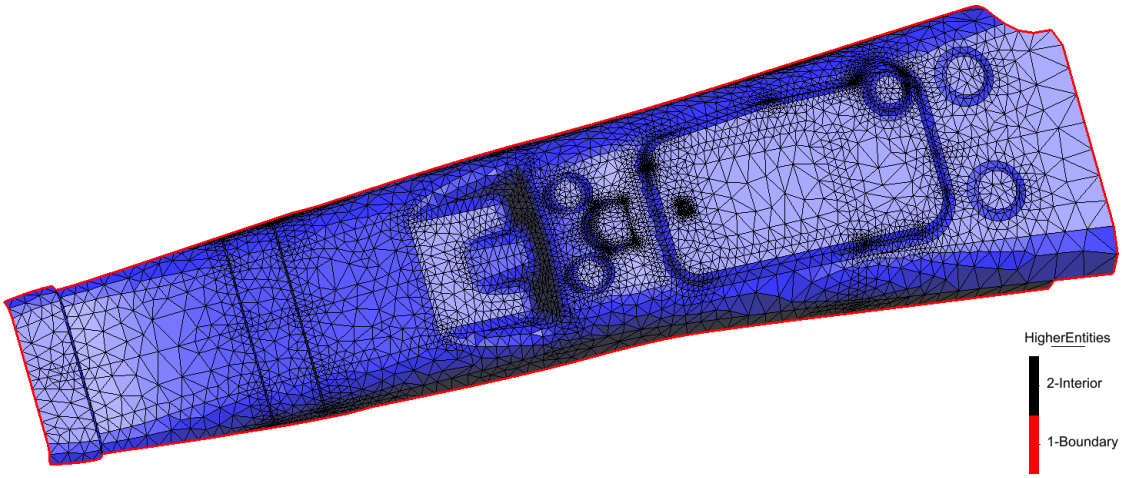
The original surface to be deleted.

14 . Use **Geometry->Delete->Lines** , and after delete the points with **Geometry->Delete->Points** to select the lines and points that belong to the surface that has been trimmed and which no longer belong to any surface. In this case, all the visible lines and ponints may be selected since the program will only eliminate those which do not have entities covering them.



Lines and point that no longer belong to any surface.

15 . Select **Mesh->Generate mesh**. Then visualize the result using the option **View->Higer entities->Edges**.



Higher entities of the result model

A conformal mesh has been achieved, all edges are interior, higer entitie 2, except the ones on the boundary with higer entitie 1.

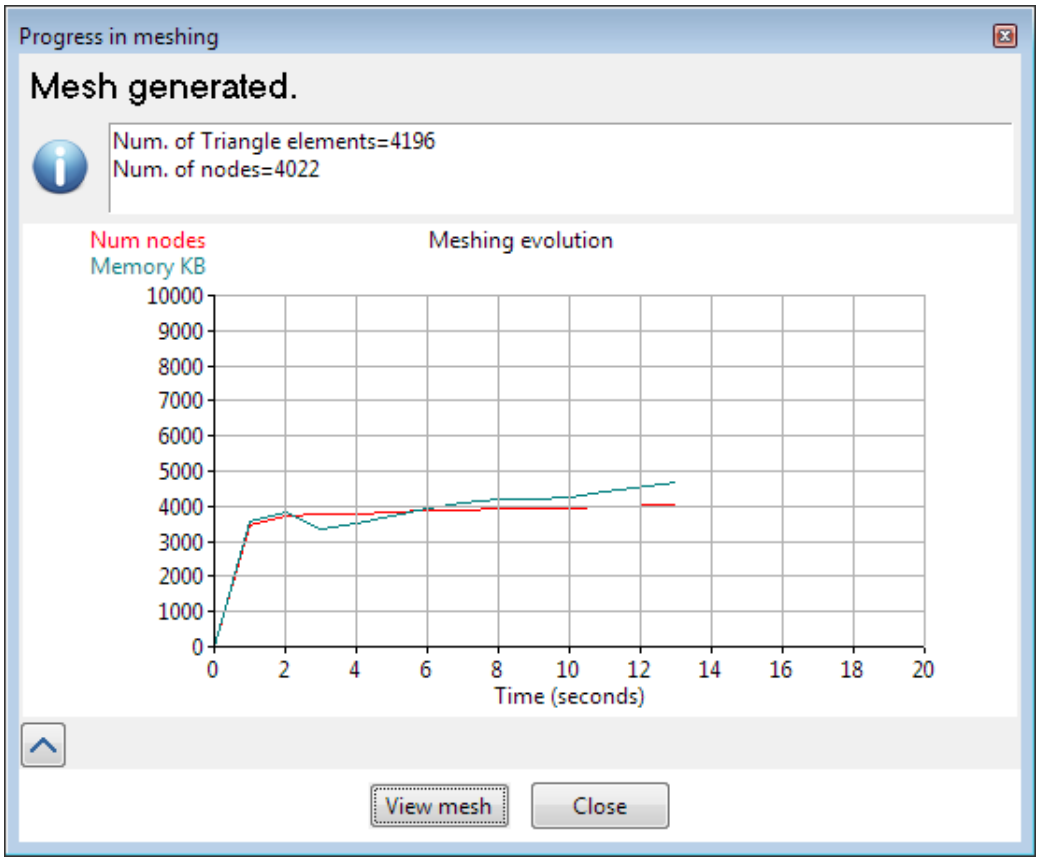
9.3.3 Creating a non-conformal mesh

NOTE: Non-conformal meshes may be used with some calculating modules, i.e. stamping a plate. Using non-conformal meshes significantly reduces the number of elements in the mesh. This cuts down on computation time.

1 . Select **View->Mode->Geometry**.

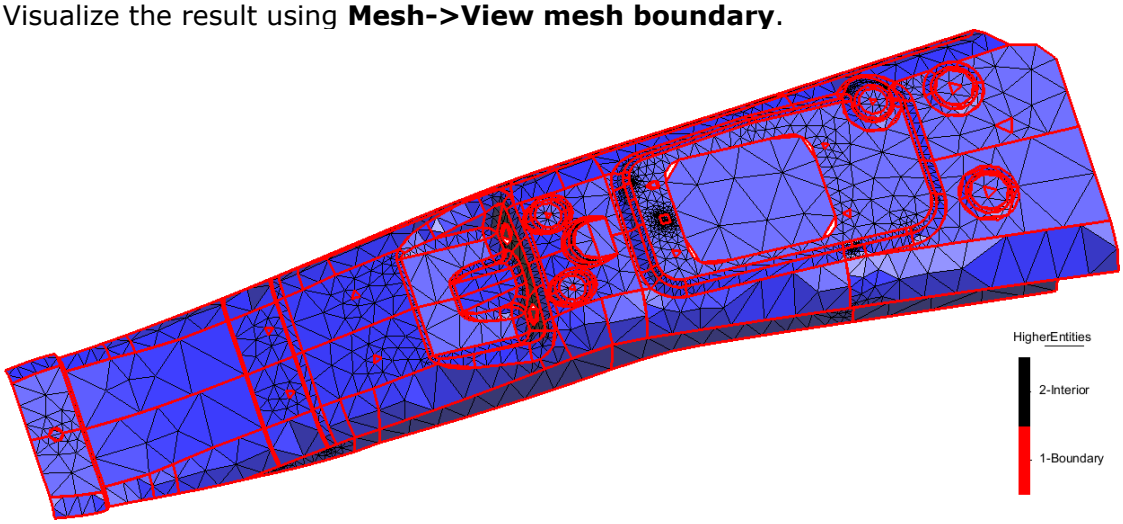
2 . Select **Geometry->Edit->Uncollapse->Surfaces**. Select all the surfaces in the model. Press **ESC**. A sufficient number of lines is created so that no surface (of the object) shares lines with any contiguous surface.

3 . Select **Mesh->Generate Mesh**. When the mesh has been generated, a window appears with information about the mesh (Figure 29). The result is a non-conformal mesh composed of far fewer elements than the meshes generated in the previous section: about 4000 elements instead of the 10.000 needed to generate the conformal mesh.



Information about the generated mesh after uncollapse

4 . Visualize the result using **Mesh->View mesh boundary**.



edges higer entities after uncollapse