
Tutorial

THE BASICS

GEOMETRY CLEANUP AND SHELL MESHING

Table of Contents

1. Introduction.....	2
1.2. Prerequisites	2
1.3. Problem description.....	2
1.4. Data files	2
2. Read the CAD file.....	3
3. Checking the geometry	9
4. Perform Cleanup.....	11
5. Defeaturing.....	42
6. Shell meshing – Uniform size mesh.....	48
7. Shell meshing – Variable size mesh.....	59
8. Conclusion	63

1. Introduction

This tutorial presents in detail all the steps taken to read a CAD file of a part, perform cleanup, as well as some geometry simplification, in order to create a good quality shell mesh.

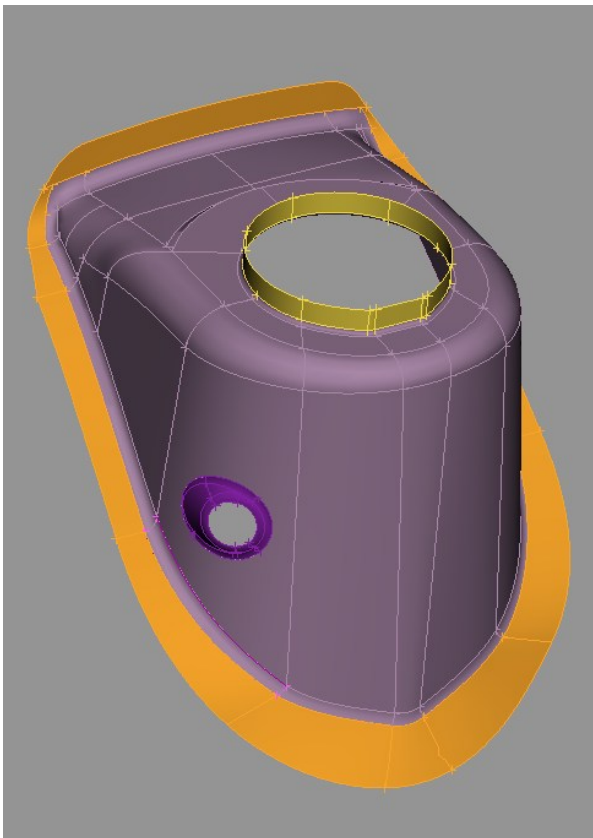
The steps described in this tutorial include:

- Read the IGES file with different tolerance settings and assess the results.
- Cleanup geometry, close gaps, modify or create new Faces etc.
- Optionally, repeat the previous step with the help of Automatic Cleanup functionality.
- De-featuring, removing details such as small holes.
- Mesh the part with a uniform mesh and an element length of 10 mm.
- Improve the mesh by manually optimizing the shape of the Macro Areas.
- Improve the mesh by using ANSA's automatic functions.
- Mesh the part with a variable size, curvature dependent mesh.

1.2. Prerequisites

Reading the first chapter of the [ANSA for CFD Quick Start Guide.pdf](#) document is recommended in order to obtain a familiarization with the ANSA interface, terminology and CFD layout.

1.3. Problem description

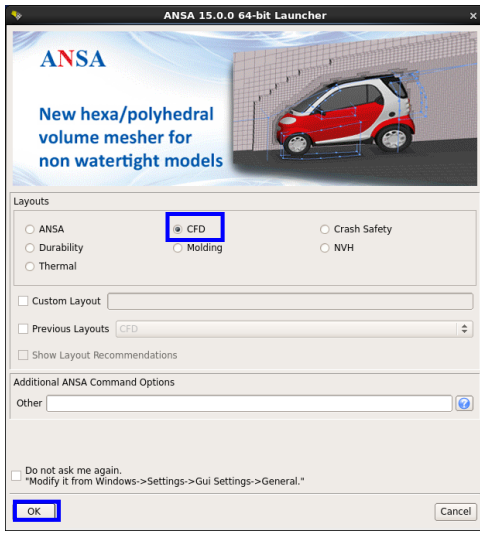


The geometry of the part is shown here in its final state. Note that the part consists of more than one Property IDs (PIDs), as shown in different colors.

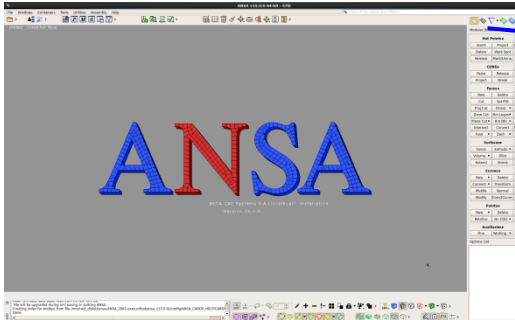
1.4. Data files

The files of this tutorial are: one IGES file named `basics.igs` containing the geometry and one ANSA file named `basics_result.ansa` containing the result for reference.

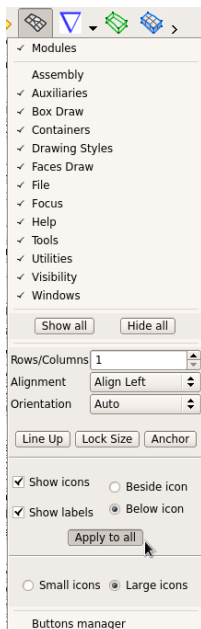
2. Read the CAD file



Start ANSA (e.g. by typing `ansa64.sh` for the 64 bit Linux version), **select the CFD option** from the pop-up launcher so that ANSA begins with the default CFD layout and click **OK**.



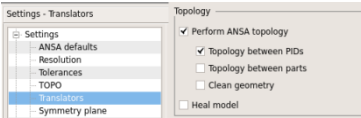
By default you are in the TOPO menu.



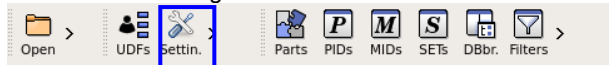
In order to get familiar with the icon buttons of the interface, you can right click on any toolbar, activate the “Show labels” flag and click “Apply to all”.

Now the label of each button appears below every icon.

Before reading any CAD data into ANSA you can specify some settings concerning the application of Topology (the connectivity of neighboring Faces), and the Resolution (the appearance of geometrical entities on the screen). These settings should correspond to the dimensions, level of detail and tolerances of the CAD file to be input.

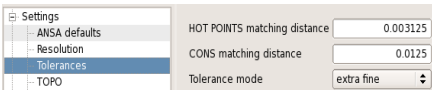


Click on the “Settings” button.



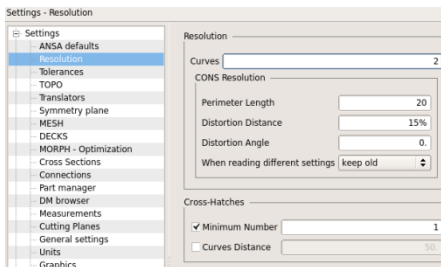
In the “Translators” section of the Settings window and the “Topology” options group, the **Perform ANSA Topology** flag ensures that the Topology is performed during CAD file opening. As a result when the IGES file is read later, the topology process will be applied.

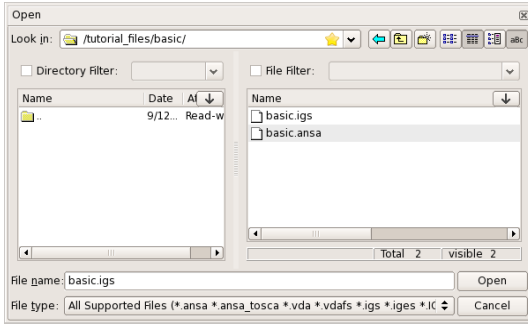
For educational purposes keep the **Clean Geometry** flag button de-activated. In this way the automatic geometry cleanup step will not be performed and the cleanup will be examined and performed manually.



Note that the automatic topology is performed according to the tolerance settings that are specified in the “Tolerances” section. Make sure to specify appropriate tolerance values before reading a CAD file, to avoid collapsed Faces (large tolerances), or gaps (very small tolerances). For demonstration select **extra fine** tolerance settings (HOT POINTS matching distance 0.003125, CONS matching distance 0.0125).

Also note that the appearance of geometrical details depends on the settings found in the “Resolution” section. Keep the default value of 20 for **Perimeter Length** in the CONS Resolution settings group. (Note that this value also corresponds to the initial discretization length or element length). Change the value in the **Curves** entry field to 2 in order to view small geometrical features in more detail. Press **OK** and then **Apply** in the warning window that opens.



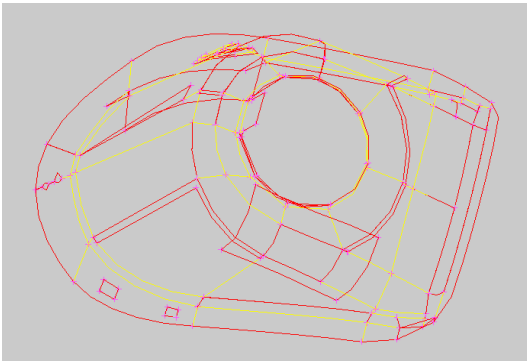


Now read in the IGES data from File>Open. The *File Manager* window appears. Ensure to switch the filter type to IGES to be able to view the IGES file!

Navigate and select the IGES file `basic.igs`. Click **Open**.

! Note that when opening a CAD file (such as IGES) and not an ANSA file, ANSA automatically opens the “Settings” window in the “Translators” section, thus prompting the user to set the “Topology” as well as the “Tolerances” settings prior to reading the file.

Close the “Settings” window.



The CAD file is read and topology is applied. You should see the part as shown. Change the view of the part to better understand the geometry.

Press:



Ctrl + left button to rotate the part



Ctrl + middle button to translate the part



Ctrl + left and middle mouse button (or mouse wheel) to zoom in and out



You can also zoom in and out with the F7 and F8 keys.



If at any time the view is lost, press F9 to fit all to the screen.



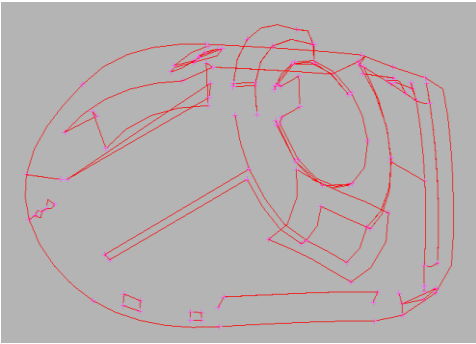
Notice that as by default the visibility mode of the model is set to ENT (Entity), you can see the red, yellow and cyan CONS (Curves ON Surfaces).

The coloring has to do with the connectivity of the Faces.

Red: a Single free edge CONS

Yellow: a Double CONS shared by two neighboring connected Faces

Cyan: a Triple CONS shared by three or more connected Faces.

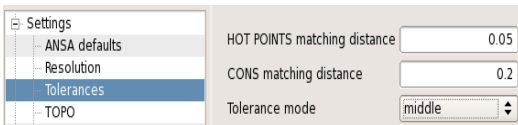


To make things clearer, you can activate and deactivate the visibility of each of the three types of CONS separately using the “Single”, “Double” and “Triple” flag buttons located in the FOCUS toolbar.



De-activating here the “Double” flag only the free red and triple cyan CONS are left visible. Apart from the free edges of the sheet metal there are other openings (gaps, untrimmed Faces etc.) that must be corrected.

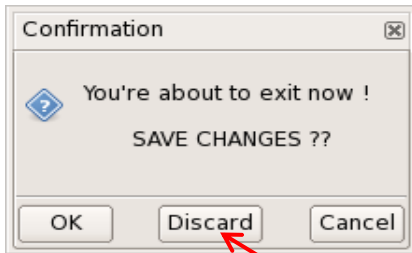
Note that the current specified Tolerances are Extra Fine and the topology process has left many gaps, as the CAD description is not very accurate. What if the specified Tolerances were larger?



Click on the “Settings” button to open the “Settings” window (alternatively you can open the IGES file and ANSA will open the “Settings” window prior to reading the file).

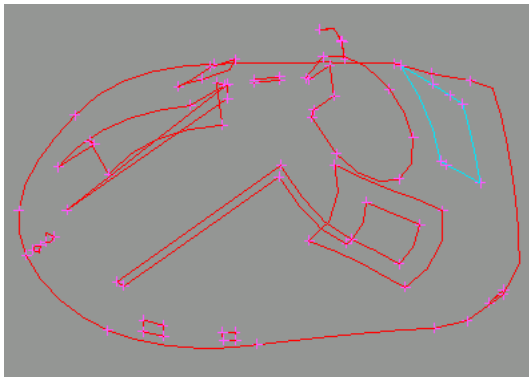
Click on **Tolerances**.

Select **middle** tolerance settings (HOT POINTS matching distance 0.05, CONS matching distance 0.2) and press **OK**.



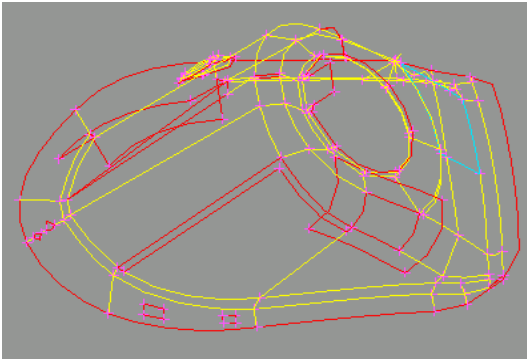
The *File Manager* opens next. Select again the `basics.igs` file and press **Open**. Be careful to select the IGES and **NOT** the ANSA file! Switch the filter type to IGES in the File Manager.

Close the “Settings” window that opens as you have already made the adjustments.

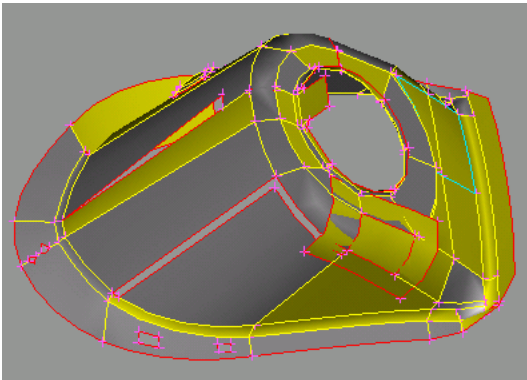


The CAD file is read and Topology is applied, this time with middle tolerance values.

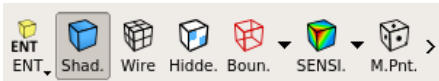
The red free edges are now less, because the topology process resulted in the connection of more Faces automatically.



Activate back the visibility of “Double” CONS to view the whole part.

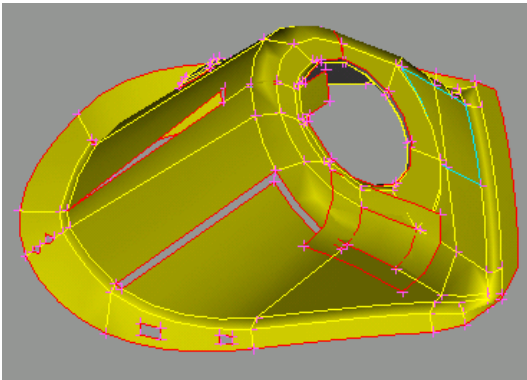


Activate the “Shadow” view mode flag, located in the Drawing styles toolbar.



You can now view the Faces in gray and yellow (as you are always in ENT view mode). Gray color represents the positive side of the Faces while Yellow the negative one.

The orientation is currently random. There is not a uniform Face orientation.

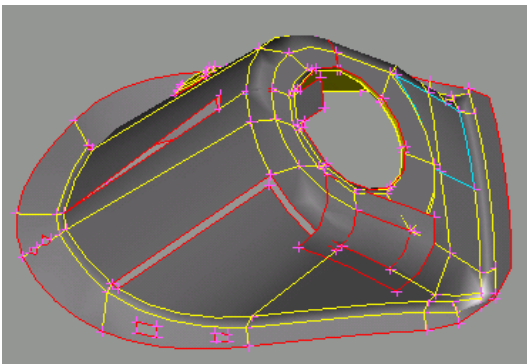


Orient

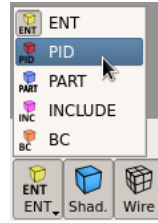
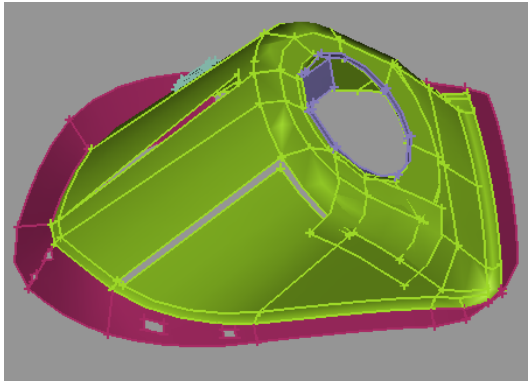
Visible

Activate the Faces>Orient> [Visible] function.

The orientation of all the visible Faces becomes uniform.

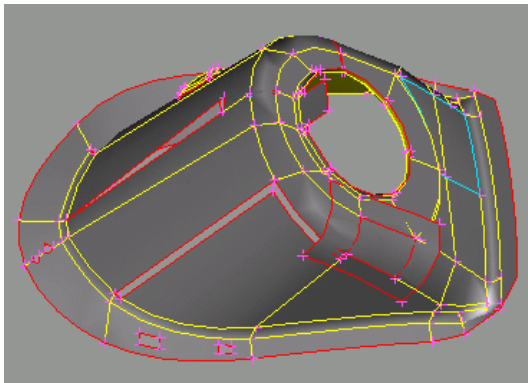


In order to flip the now uniform orientation, you can use the Faces>Orient> [Visible] function once again.



Switch to PID color view mode.

You can now view the Faces colored according to their Property ID. (Note that the assigned colors to different PIDs are random when opening a CAD file, so you may see different colors).



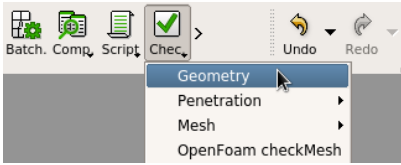
Switch back to ENT color view mode to proceed with the geometry checking and cleanup process.

Save the file to your local directory under the name `basics.ansa` by selecting “Save as” from the File menu. The *File Manager* opens in order to specify the path and the ANSA database filename. Remember when you save or output in ANSA you must always type the appropriate extension for each file type.

You will now proceed with Geometry checking.

(If at any time you want to pause your work you can exit ANSA from File>Quit, and then start ANSA again and read the ANSA database `basics.ansa` from File>Open to resume work).

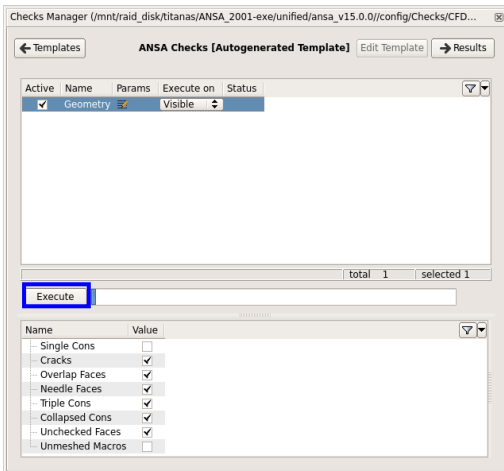
3. Checking the geometry



Before you start cleaning up the geometry of the model it is useful to perform a number of checks that are incorporated in ANSA.

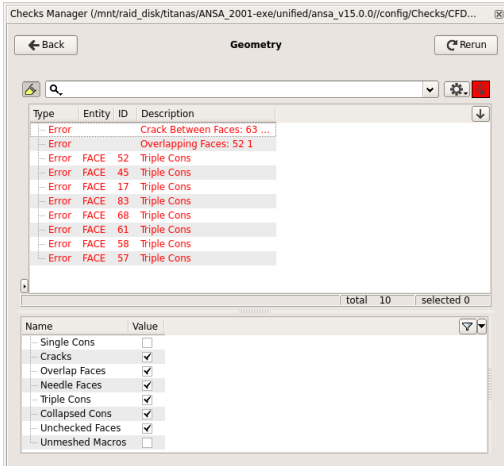
ANSA can identify and isolate on the screen the problematic areas that require the user's attention.

Click on the "Checks" button, located in the "Tools" toolbar on the top of the screen and select "Geometry".



The *Checks Manager* window opens. Here you can select the type of problems that may exist in the geometry of the model to be investigated by activating the corresponding flag in the lower part of the window.

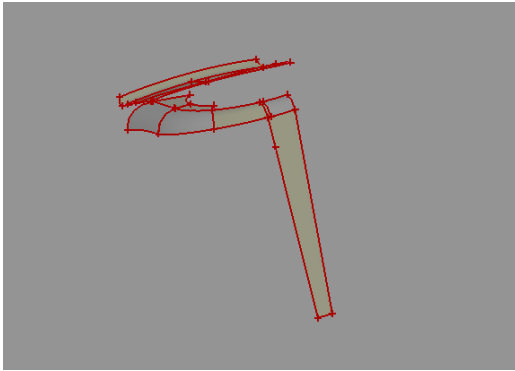
Keep the default options and click **Execute**.



ANSA identifies a total of 10 errors in the geometry of the model.

You can highlight any of the problematic areas by left-clicking on it, or isolate it on the screen by right-clicking on it and selecting "Show only".

Depending on the type of the problem ANSA also offers the option to automatically fix it.



You can visualize all the errors by pressing Ctrl+A to select them all and then right click and select the “Show only” option.

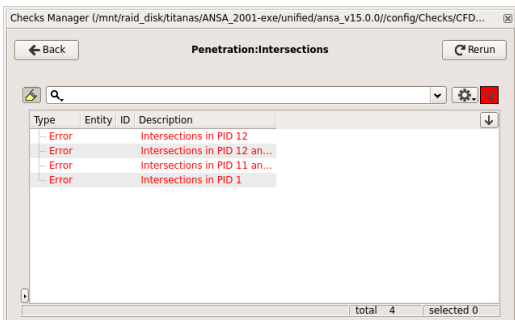
Close the *Checks Manager* window.

Click “All” from the focus toolbar to make all the faces visible again.

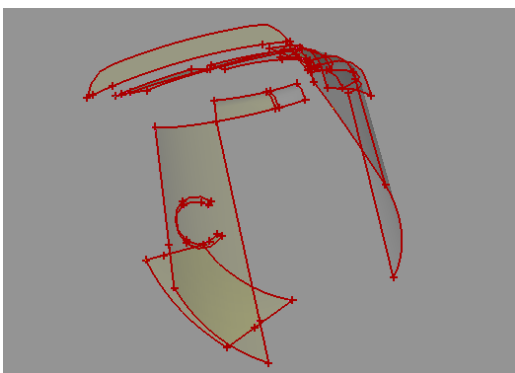


Click on the “Checks” button again and select Penetration>Intersections as shown on the left. This check will identify any intersecting geometry that needs to be corrected.

Click **Execute** in the *Checks Manager* window that opens.



ANSA identifies 4 errors.



Again you can isolate them by right clicking on any or all of them and selecting “Show only”. You can select all the errors by pressing Ctrl+A.

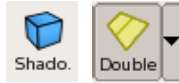
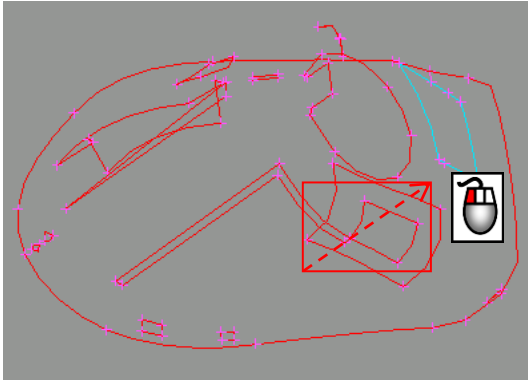
Close the *Checks Manager* window.

Click “All” from the focus toolbar to make all the faces visible again.



You will now proceed with the geometry cleanup procedure.

4. Perform Cleanup



De-activate “Shadow” mode and the visibility flag button for “Double” CONS.

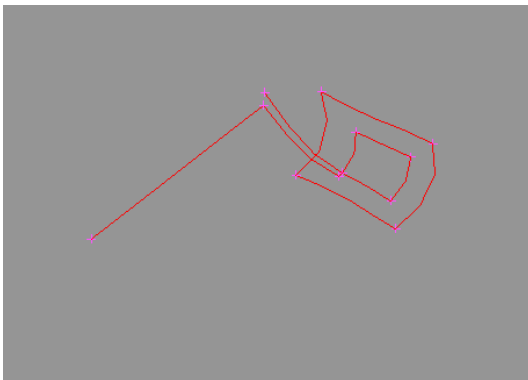
You will now isolate a region of your part to work on.



Activate the “Or” function located in the Focus Toolbar.



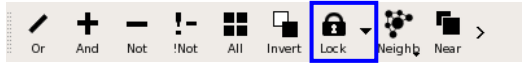
Make a box selection of the area shown on the left using the left mouse button.



The “Or” function leaves visible only the selected Faces.

(If you accidentally select wrong Faces, press the All function of the same Group to bring all the entities back to visible).

Activate the “Lock” flag button in the Focus Toolbar to lock the currently visible entities. Now when you press the “All” button, only the selected and locked entities will become visible.

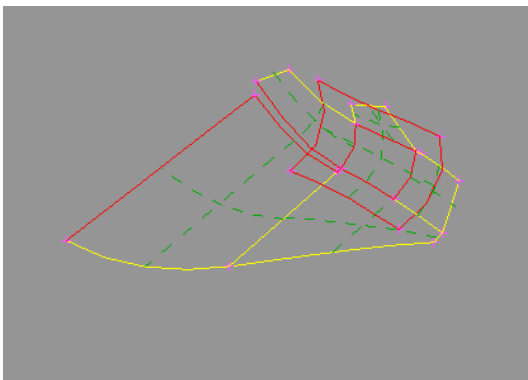


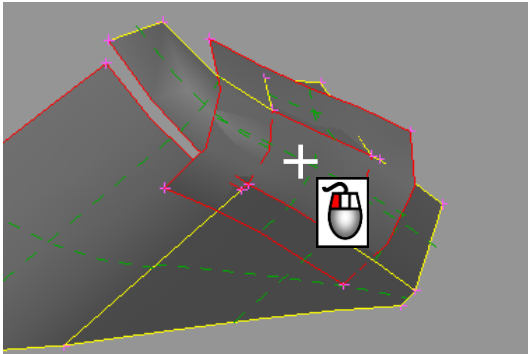
To view the complete Faces that you have left visible, activate again the “Double” CONS visibility flag.

Activate also the “Cross Hatches” visibility flag.



This allows the visualization of two green dashed lines along the center isoparametrics of the Faces. Apart from visualization purposes, the crosshatches are also used for selecting Faces.





Activate the “Shadow” mode again.

Note the problem in this area.

There seems to be a Face that is untrimmed and extends well out of the boundaries of the other Faces.



Activate the “Not” function, of the Focus group, and select with the left mouse button this Face from its crosshatch.

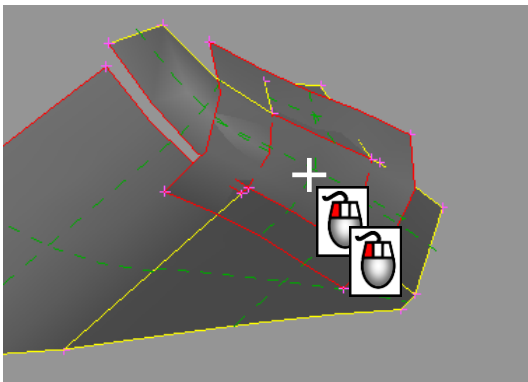
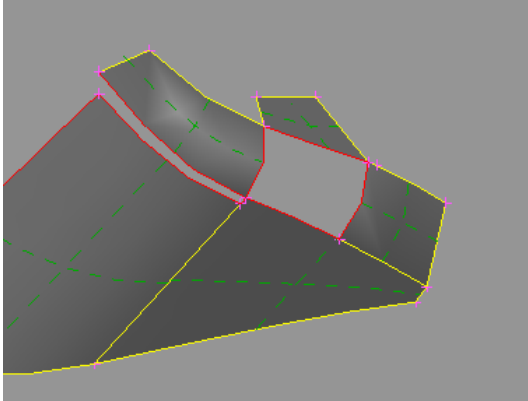
The selected Face is removed from the visible entities.

This leaves a better view of the other Faces.

You must somehow trim the big Face so that it matches exactly the boundaries of the other Faces.

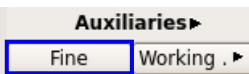


Press the “All” function of the Focus toolbar.

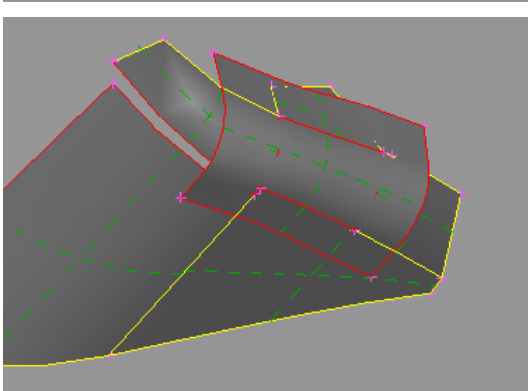


Only the locked entities (all the entities that were visible at the time that you activated the “Lock” flag) appear.

Notice that because we have specified a Resolution length of 20, some Faces appear coarser than their actual geometrical description.

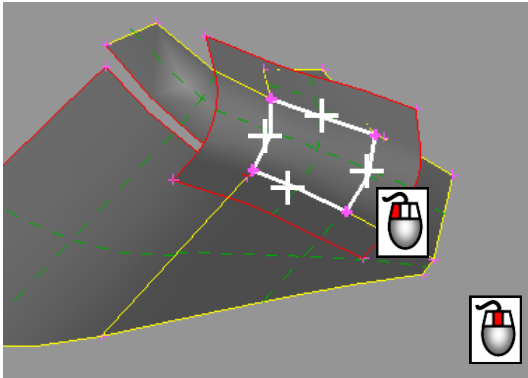


Activate the Auxiliaries>Fine function in TOPO Menu and left click on the crosshatch of the big Face twice.



Each time you press the left mouse button the Resolution length of the selected Face is halved. This makes the Face appear with more detail. The curvature of the Face is now clearly visible. (Note that if you use the Fine function with the right mouse button you double the Resolution length of selected entities, making them appear coarser).

! Note: Do not left click more than a couple of times as the resolution length will become very small and this will delay the “Shadow” operation.

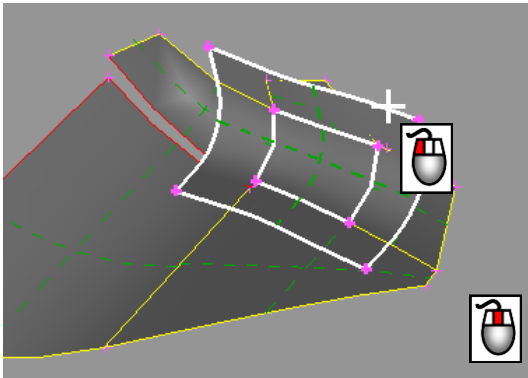


Rotating the part you can see that this Face covers well the gap of the remaining Faces. It is only a matter of trimming it correctly.

Project

Activate the CONS>Project function.

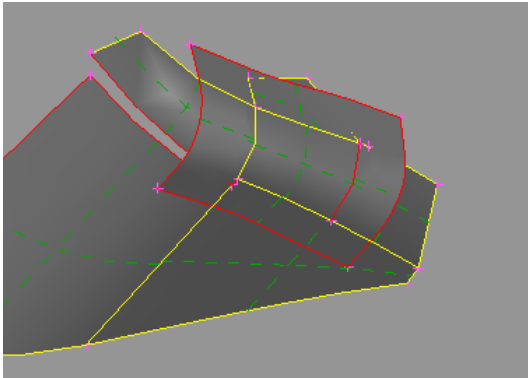
Select with the left mouse button the four red CONS of the Faces behind (if you cannot pick them, turn the view of the part around, or deactivate Shadow mode). De-select with right mouse button if required. Confirm with the middle mouse button.



Next select the Face to project and cut the selected CONS onto.

You can pick the Face from its crosshatch or one of its red CONS. De-select with right mouse button if required.

Confirm with the middle mouse button.

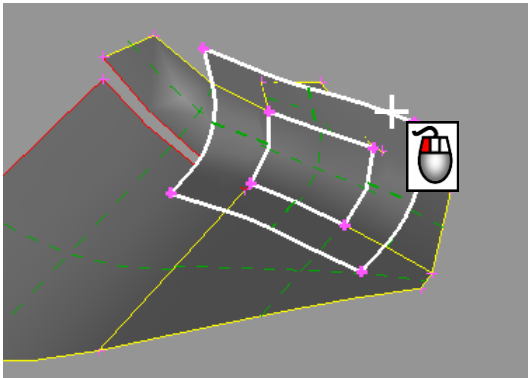


The selected red CONS are projected on the Face and cut it.

Esc

Press the ESC key to exit the Project function.

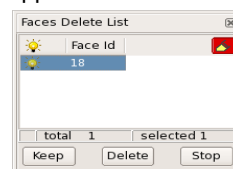
Remember you can press the ESC key to abort any function if you are unsure. Additionally, after performing an action you can always exit the corresponding function by pressing the ESC key.



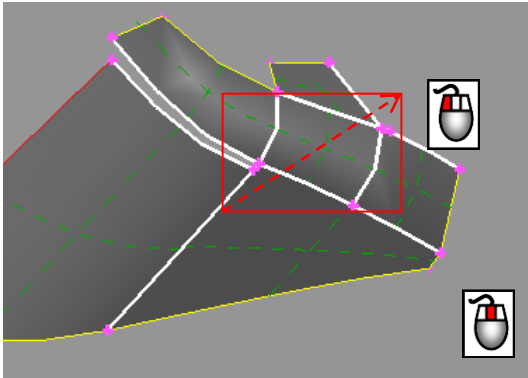
Delete

Activate the Faces>Delete function.

Select the excess Face to be deleted from its boundary. The *Faces Delete List* window appears.



Press **Delete** to delete the Face.

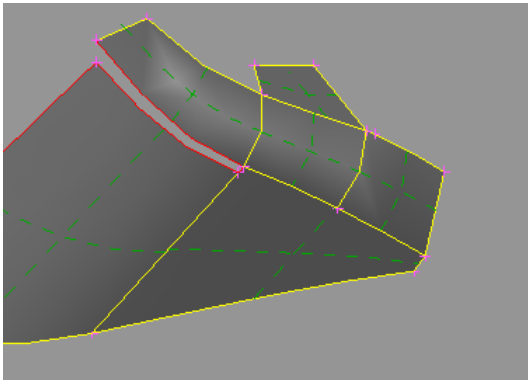


(! Note that to retrieve a Face, which was deleted unintentionally, activate the Faces>Undelete function, select it among the previewed deleted Faces and press the middle mouse button. Then use the Faces>Topo function to re-establish connectivity).

The Face is now properly trimmed, but not topologically connected with the neighboring Faces (note the red CONS).

Activate the Faces>Topo function.

Make a box selection of the CONS shown on the left, using the left mouse button. Confirm with the middle mouse button.

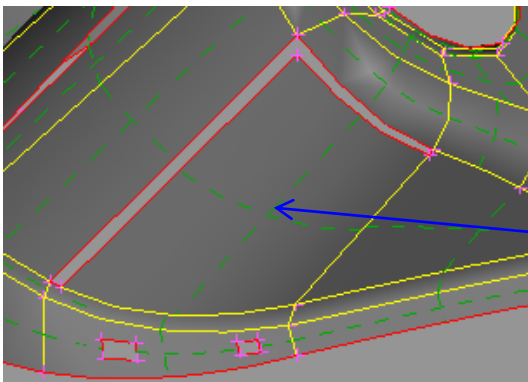


Topology is applied by ANSA according to the specified tolerances.

All CONS appear yellow, indicating proper double connectivity.



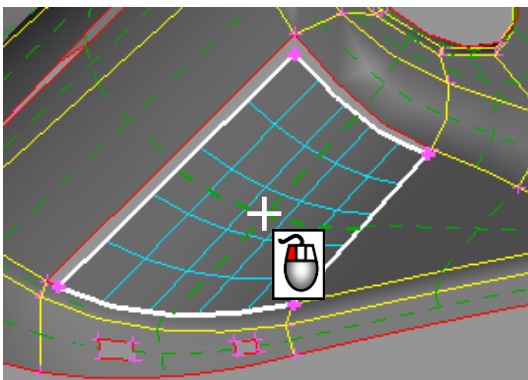
De-activate the “Lock” function from the Focus group.



Press “All” to bring all the Faces back to visible.

Moving to the next problematic region on the left.

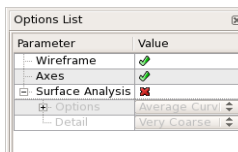
Here we have the opposite problem. A Face that is not large enough and a resulting gap.



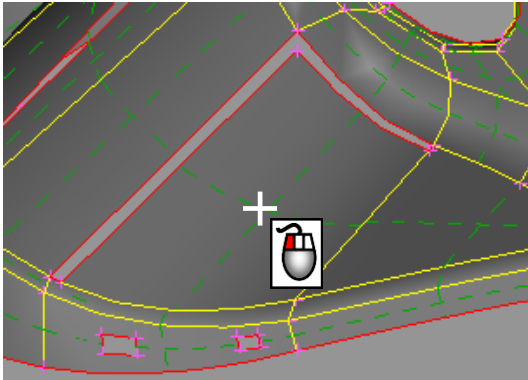
Info

Activate the function Surfaces>Info. Select this Face from its crosshatch.

Its Surface (its underlying CAD description) is also not big enough to cover the gap.



Notice that the visibility of the Surface can be controlled from the Options window.

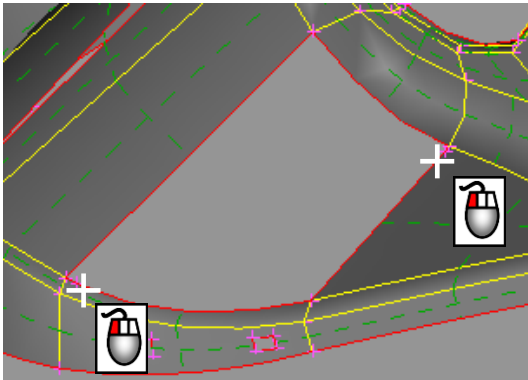


Delete

Activate the Faces>Delete function.

Select the Face with the left mouse button from its crosshatch.

! Note that when selecting a Face from its crosshatch, this is deleted without any warning. In contrast, when selecting it from its boundary CONS, the *Faces Delete List* preview window appears for confirmation.

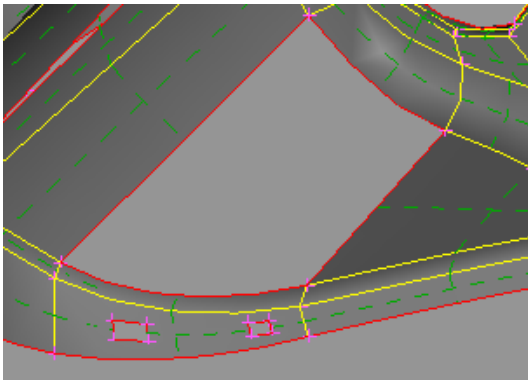


Delete

Activate the Hot Points>Delete function.

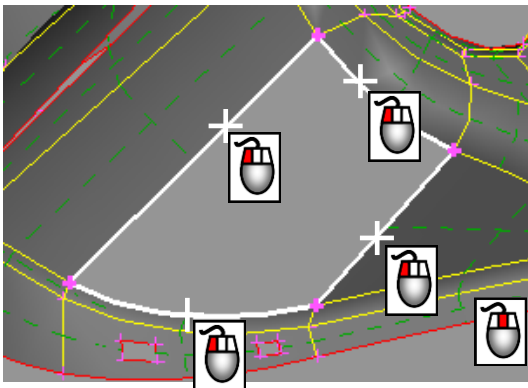
Notice two magenta cross Hot Points that were left over after the deletion of the Face.

Select with the left mouse button the two Hot Points as shown on the left.



The Hot Points are deleted. (Although not necessary, you can always delete such Hot Points. This leaves uniform longer CONS that are easier to select for Face creation, and also results in better quality mesh later).

Now you will create a new Face to cover this opening.

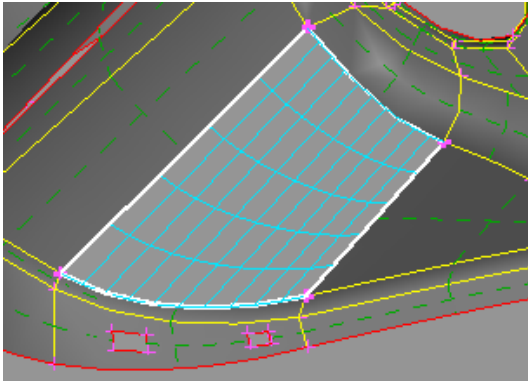


Coons

Activate the Surfaces>Coons function. (Coons stands for the name of the mathematician).

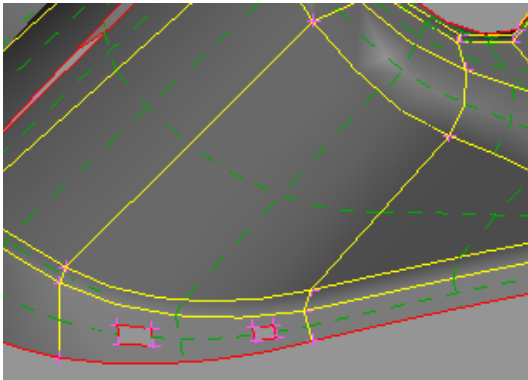
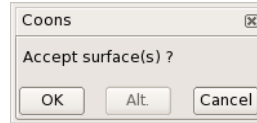
Select the four red CONS that form the boundaries of the Face to be created.

Confirm with the middle mouse button.



A preview of the Surface to be created is given as a cyan net.

Press **OK** in the *Coons* window that appears if you are satisfied with the result.



Upon accepting the Surface, the new Face is created, and is also topologically connected with the neighboring Faces.



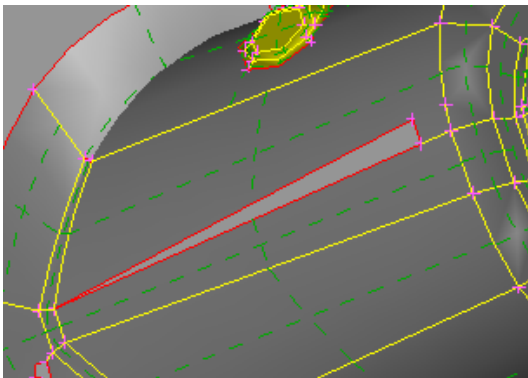
De-activate the “Shadow” mode



and the “Cross Hatches” and “Double” visibility flags

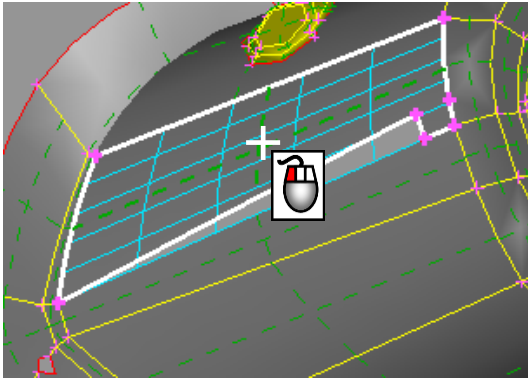


Note how the problematic regions drop in number.



Activate back the “Shadow”, “Cross Hatches” and “Double” flags.

Move to the small triangular gap at the front of the part.



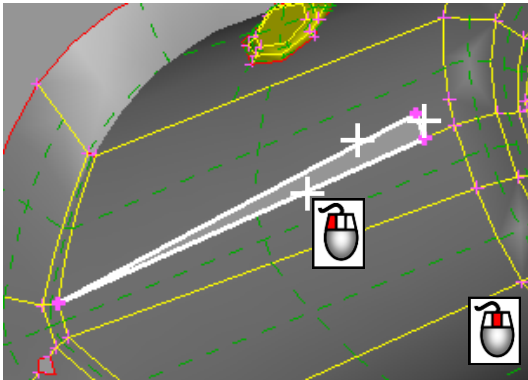
Info

Activate the Surfaces>Info function.

Select the Face from its crosshatch.

The previewed underlying Surface appears to cover the gap.

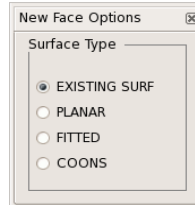
In this case you will create a new Face using the already existing Surface.



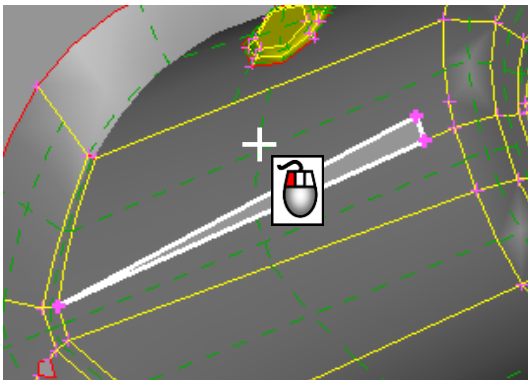
New

Activate the Faces>New function.

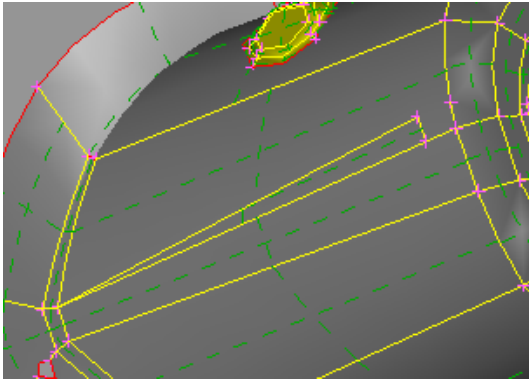
In the *New Face Options* window that appears select the *EXISTING SURF* option.



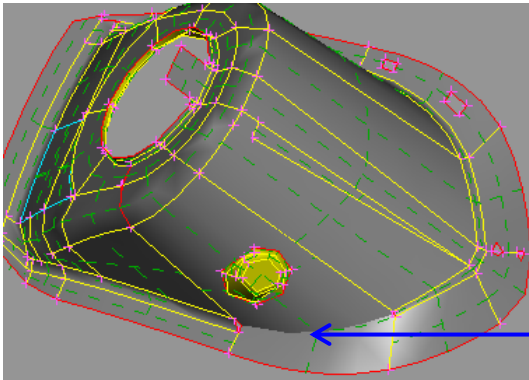
Select with the left mouse button the three red CONs as shown on the left and confirm with the middle mouse button.



Next select with the left mouse button the crosshatch of the Face whose Surface you will use for the new Face.

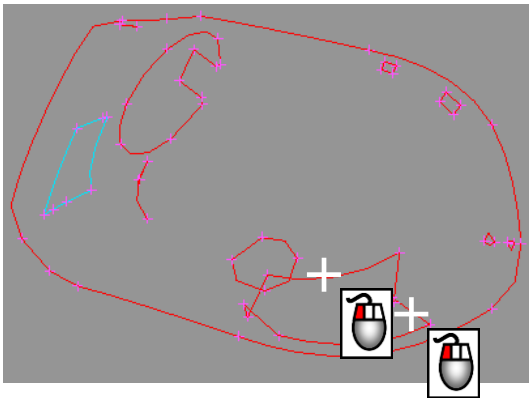


The new Face is created. If you enquire its Surface with Surfaces>Info you will see that it uses the same one as the larger neighboring Face.



Rotate the part to the other side.

You can see that in this area there are two untrimmed and intersecting Faces.



De-activate the “Shadow” flag from the Drawing styles toolbar and the “Cross Hatches” and “Double” flags from the Visibility status toolbar



to have a better view of the problematic area.



Activate the “Not” function from the Focus group.

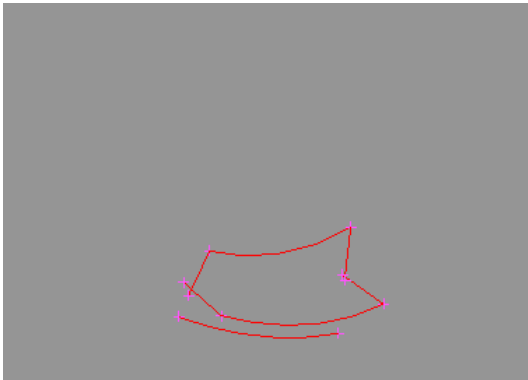
Select with left mouse button the two red CONS as shown.



Their Faces become not visible.

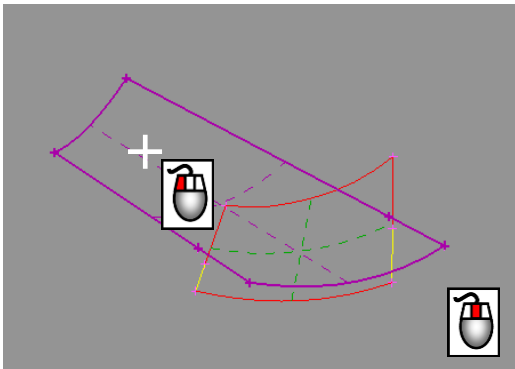


Next press the “Invert” button of the Focus group.



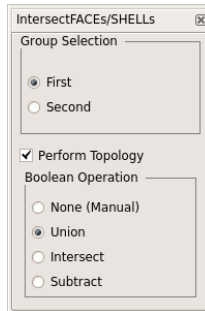
The visible / not visible status of all entities is inverted, making only the two selected Faces now visible.

Activate the “Double” and “Cross Hatches” flags to view the two intersecting Faces better.



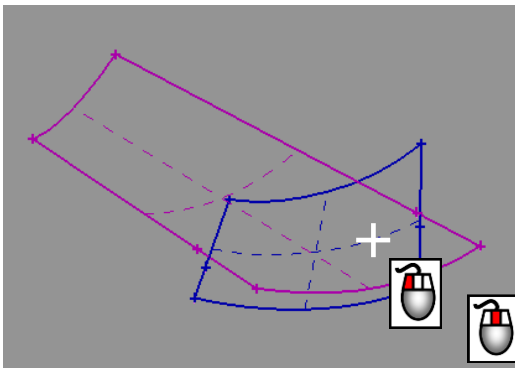
Intersect

Activate the Faces>Intersect function.

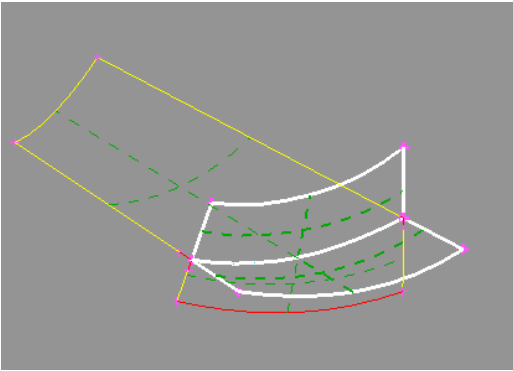


The *IntersectFACES/SHELLS* window appears. Select the “Union” option in the “Boolean Operation” group.

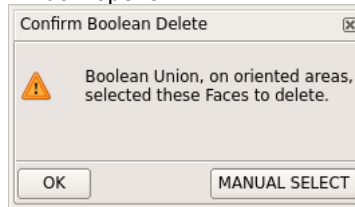
Select the first Face with the left mouse button as shown and confirm with the middle mouse button.



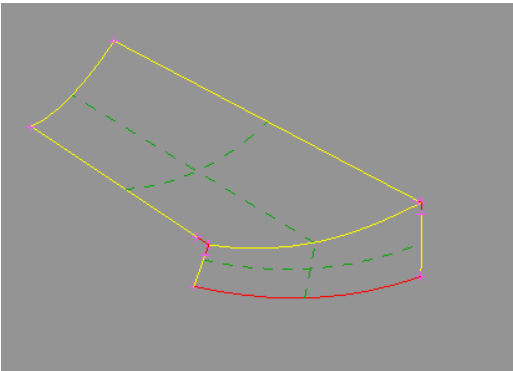
Now select the second face with the left mouse button and confirm with the middle mouse button as shown on the left.



ANSA correctly identifies the excess Faces to be deleted and the *Confirm Boolean Delete* window opens.



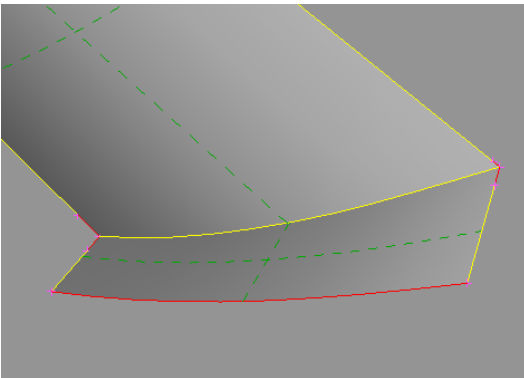
Click **OK**.



ANSA deletes the selected Faces.

Note that for the correct identification of the excess Faces to be deleted, it is important that the Faces have the correct orientation. Ensure that this is the case prior to activating the Faces>Intersect function.

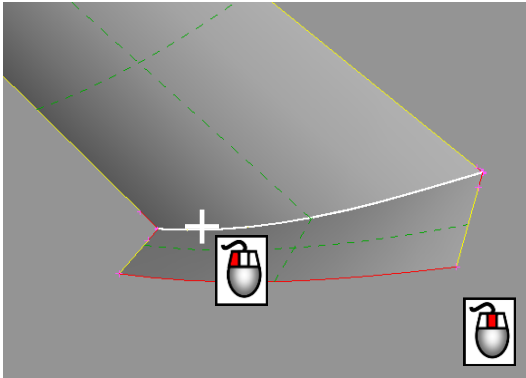
However a plain intersection would not suffice to fix this area. A fillet exists in this location between the neighboring Faces.



Activate the “Shadow” flag button to view the intersection area better.



Activate also the “Curves” visibility flag. In order to create the boundaries of the fillet you will use 3D Curves. These are auxiliary curves that are not connected to the Faces and are used for CAD construction in ANSA. 3D Curves appear in magenta color and their visibility is controlled by the Curves visibility flag.



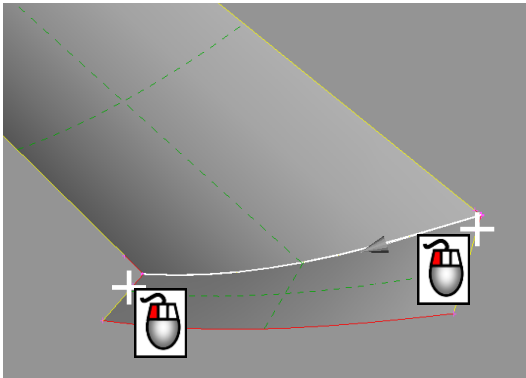
Now, you will create the boundaries of this fillet.

Transform

Activate the Curves>Transform function. This function creates a new Curve by transforming an existing one to fit new start and end point positions.

Select with the left mouse button the yellow CONS at the intersection of the two faces as shown.

Confirm with the middle mouse button.

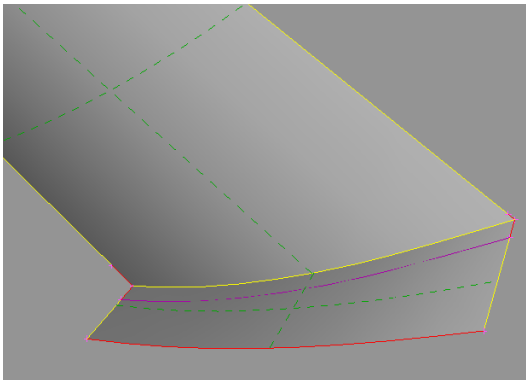


Note the arrow that appears next. This arrow indicates the start of the segment.

Select two point positions as target start and end for the new transformed 3D Curve to be created.

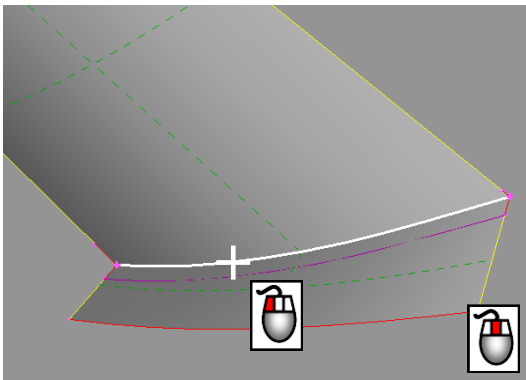
! Note that the start and end positions should correspond to the direction arrow.

Select the Hot Points as shown.



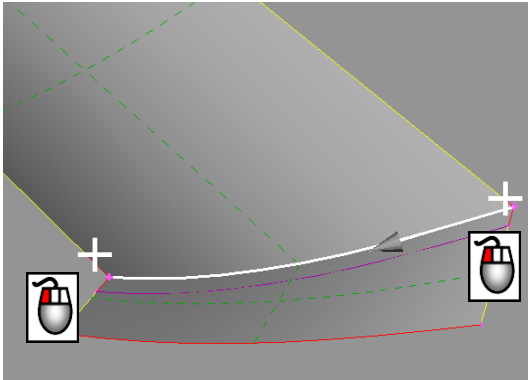
A new 3D Curve is created, from a transformation of the selected one. Notice how this Curve follows the shape of the initial one.

(if you accidentally picked wrong Hot Points, delete the unwanted new Curve, using the Curves>Delete function).

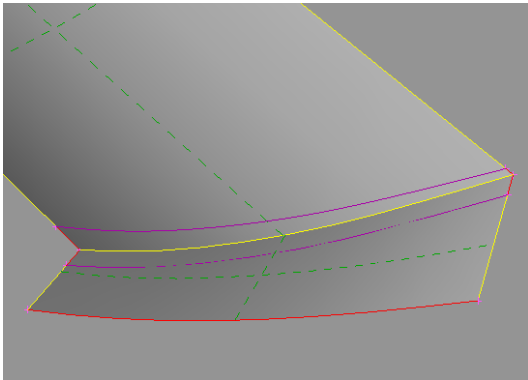


While still in the Curves>Transform function, select again the initial yellow CONS with the left mouse button.

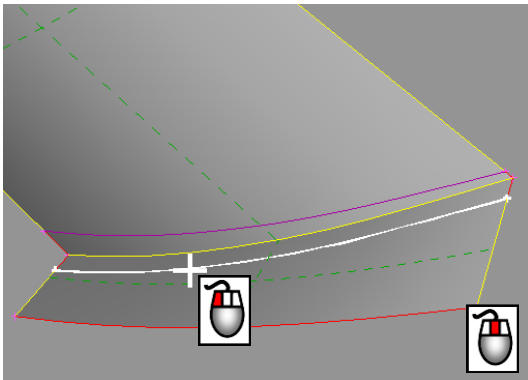
Confirm with the middle mouse button.



According to the direction of the arrow, select the two target start and end Hot Points as shown, on the other Face.



Another similar 3D Curve is created.

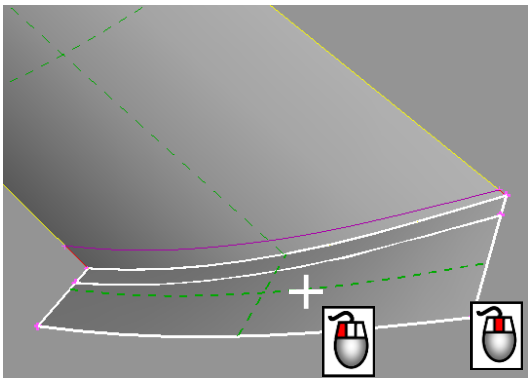


Now you will project the new 3D Curves on the Faces to cut them.

Activate the CONS>Project function.

Select the 3D Curve shown.

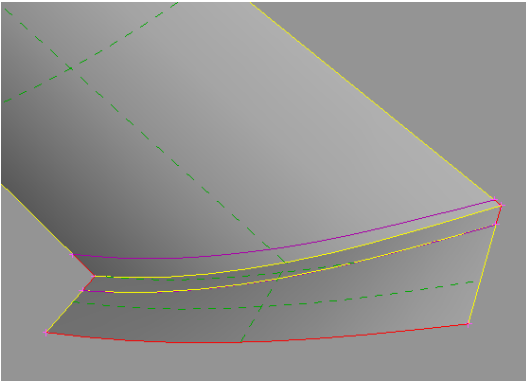
Confirm with the middle mouse button.



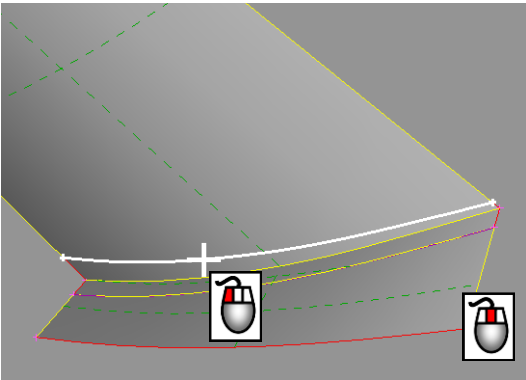
Next select the Face to project onto.

You can select the Face from its crosshatch or boundary CONS.

Confirm with middle mouse button.

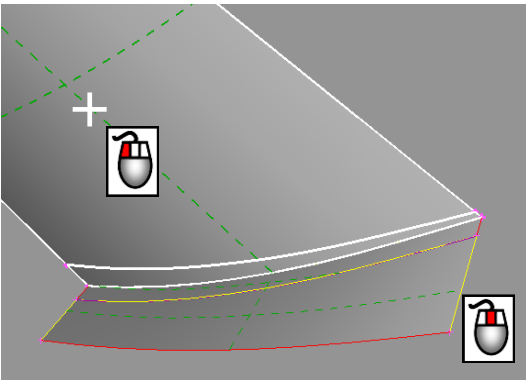


The 3D Curve is projected on the selected face cutting it. A yellow CONS appears along the cut.



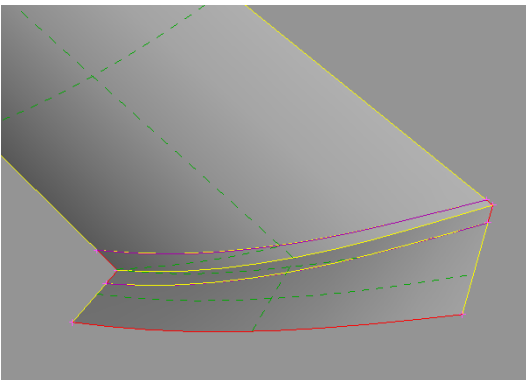
While still in the CONS>Project function select the other 3D Curve with the left mouse button.

Confirm with the middle mouse button.

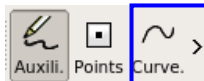


Select the Face to project onto and cut, by left clicking on its cross hatch.

Confirm with middle mouse button.

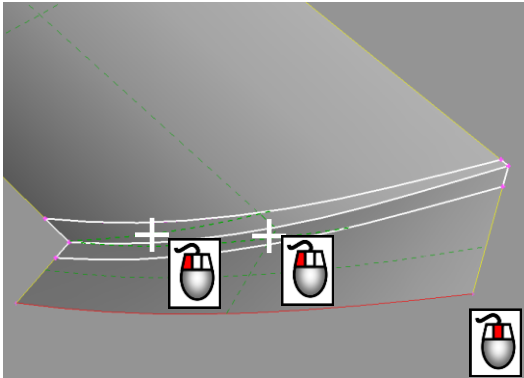


Having completed the work with the 3D Curves, de-activate their visibility flag button.



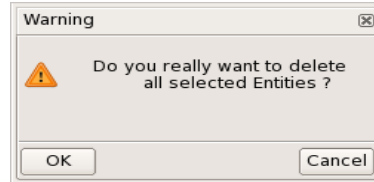
Now you will delete the small excess Faces between the two 3D curves using the Delete function from the Utilities Toolbar.



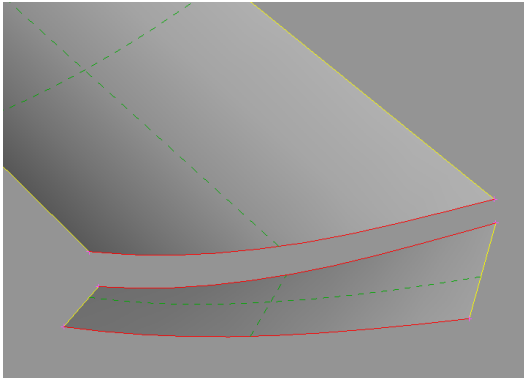


Activate the Delete function.
Select the two Faces with the left mouse button as shown on the left. Confirm with the middle mouse button.

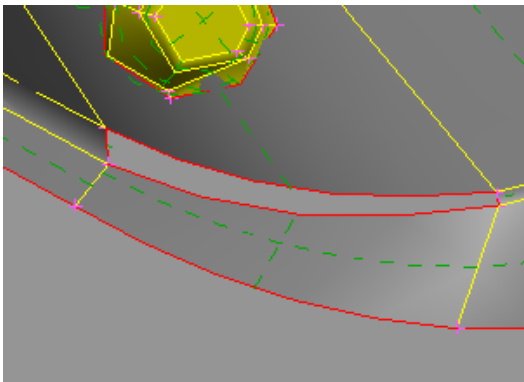
A *Warning* window appears.



Press **OK** to proceed with the deletion of the selected Faces.

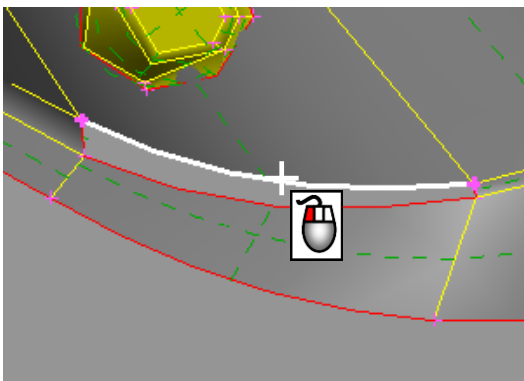


(The advantage of the Delete function of the Utilities Toolbar in comparison with the Faces>Delete function, is the fact that you can select more than one Face, without being prompted each time for confirmation).



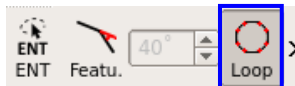
Press the “All” function in the Focus toolbar to make all the Faces visible.

Here you will create a curved face to close the fillet area.

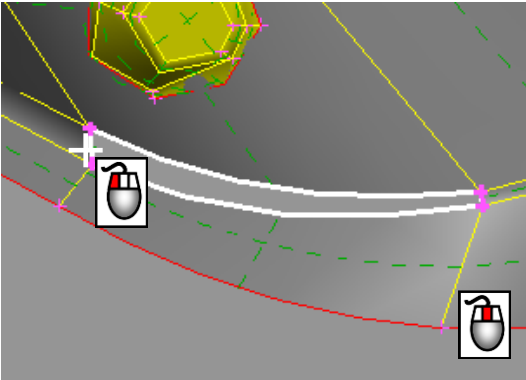


Activate the Surfaces>Coons function.

Activate the “Loop” flag button in the *Selection Assistant* toolbar.



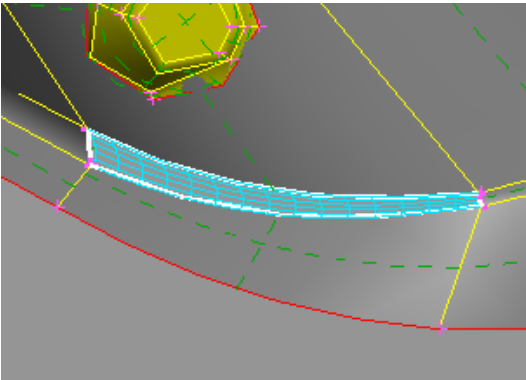
Select with the left mouse button one red CONS.



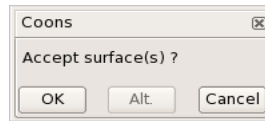
As the “Loop” flag is active, after selecting the first CONS, ANSA automatically identifies and selects the whole closed string of CONS of the same loop.

(Use right mouse button to de-select CONS if required).

Confirm with the middle mouse button.



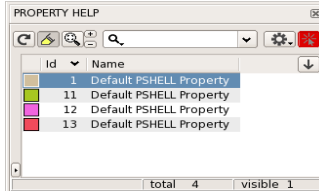
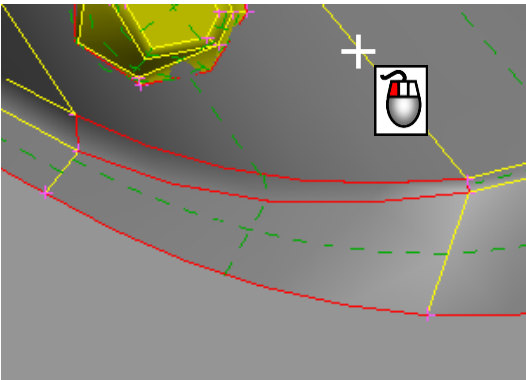
The Surface to be created is previewed as a cyan net.



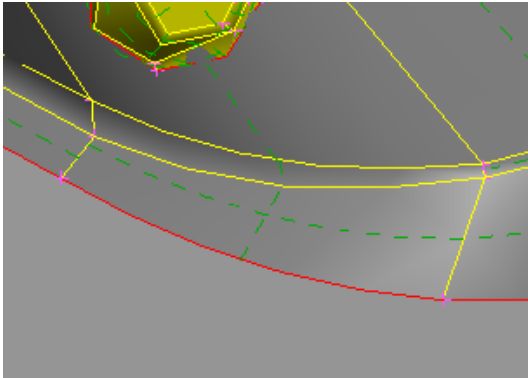
Press **OK** in the “Coons” window that appears. Note that in this case, in order to create this Face you have selected CONS that belong to Faces of different Property ID. (Remember that the flange has a different PID than the main body).

As a result ANSA prompts you to select a PID for the new Face that you have created.

The *Properties* window appears.



You can double-click on one of the available PIDs in the list. As here you may not know which one is the correct PID to select, you can left click on another Face (crosshatch or CONS) from the screen. ANSA will mark the Property of this Face on the list so that you can middle-click in order to confirm it.

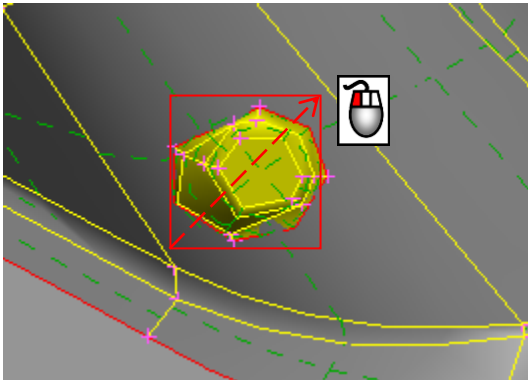


The Face is created. You can switch to PID display mode to check the different PIDs in color.



(You can always change the PID of selected Faces using the Faces>Set PID function, by selecting them and pressing the middle mouse button. The Property list window will appear; select a Property and double-click or press **OK**).

Switch back to ENT display mode.



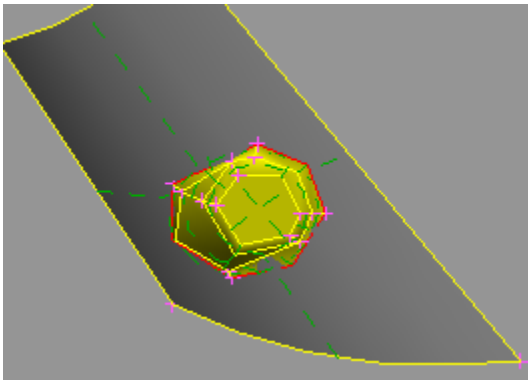
Moving now on to the next problematic area shown on the left.



Activate the “Or” function of the Focus group.

Make a box selection as shown.

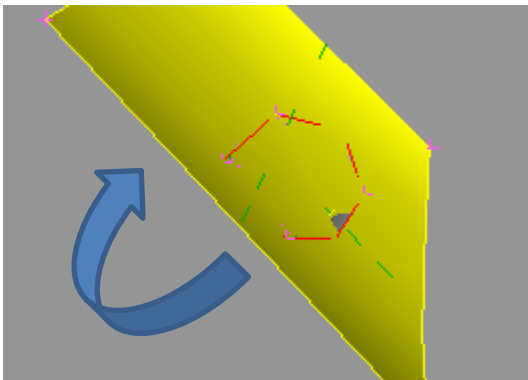
(If you select wrongly, press “All” to bring all the Faces back to visible and re-select. You may also use the “Not” and “And” functions of the Focus group).



You should leave visible only the round protrusion and the large Face shown here.

Press ESC to exit the “Or” function.

It appears that the round protrusion is not connected to the Face. See the red CONS all around.

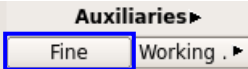


Rotate to the other side to see that the large Face has no corresponding opening for the protrusion.

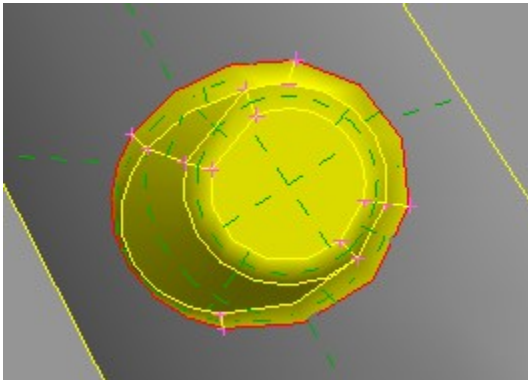


Back to the front side, notice that again due to a relatively large Resolution length the protrusion appears quite coarse and it is hard to distinguish its proper shape.

Activate the Auxiliaries>Fine function found in the TOPO Menu.

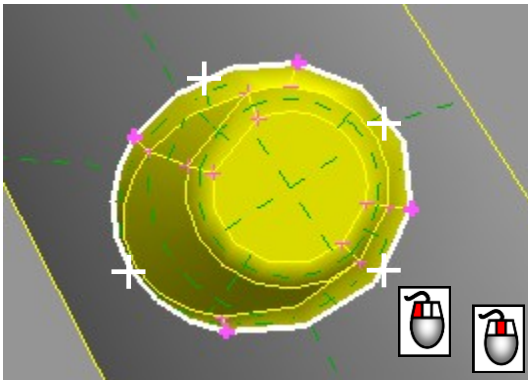


Left click a couple of times on the crosshatches of the Faces in order to reduce the Resolution length.



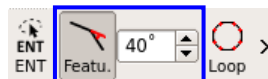
Note that if the resolution length is dropped too much, you may lose the shadow of the Face. To fix this, you can use the right mouse button to do the opposite, i.e. increase the Resolution length and make entities appear coarser.

Now the geometry appears much clearer.



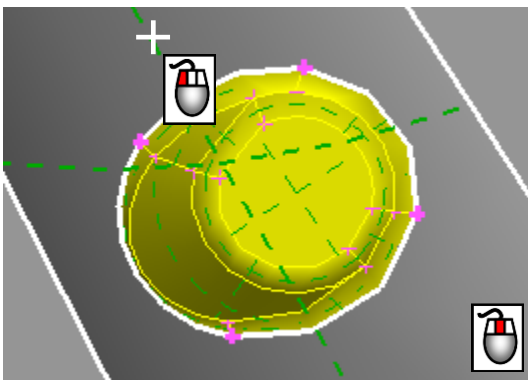
Project Activate the CONS>Project function.

Select the four red CONS around the protrusion.



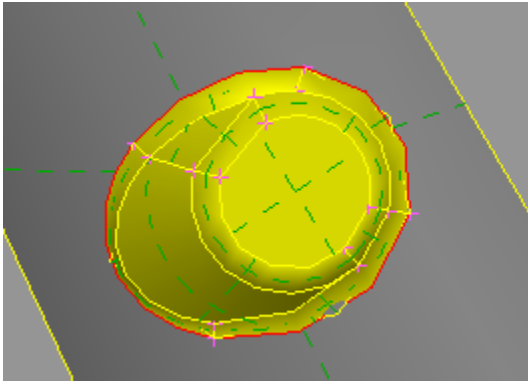
Note that if you activate the *Feature Line* flag in the Selection Assistant toolbar, you only need to select one CONS and ANSA selects the whole string provided the angle between two CONS is smaller than the specified limit.

Confirm with middle mouse button.

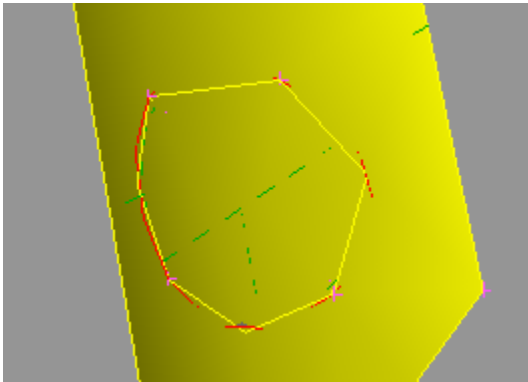


Next select the large Face to project the CONS onto.

Confirm with middle mouse button.

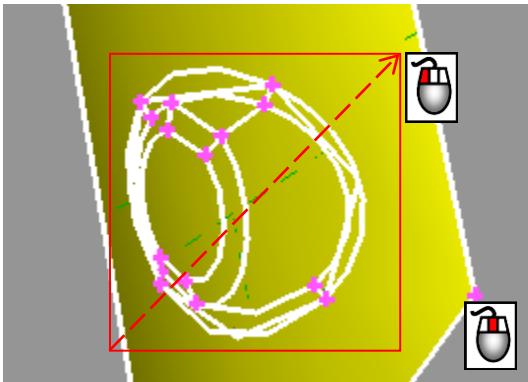


The CONS are projected and cut the Face.



Rotate the part to the rear side to view the yellow CONS along the cut that was made.

Note that although the cut was made, still the protrusion is not connected.

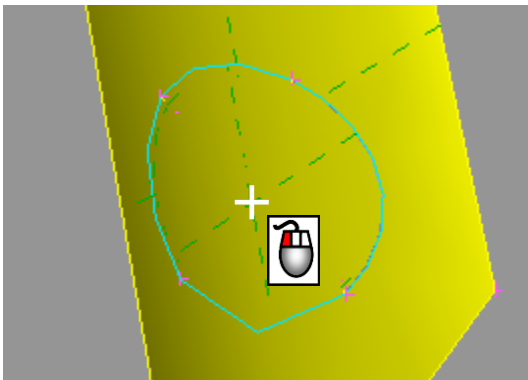


Topo

Activate the Faces>Topo function.

Make a box selection of the CONS as shown.

Confirm with the middle mouse button.



ANSA performs topology connection.

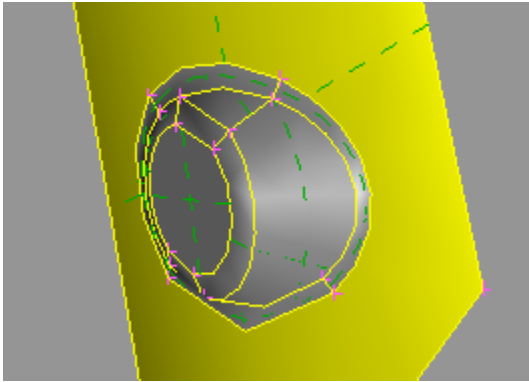
Now the boundary appears in cyan indicating that three Faces are connected at each CONS.

Now you can delete the interior excess of the large Face.

Delete

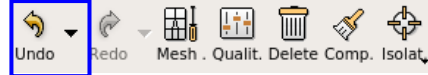
Activate the Faces>Delete function.

Select the crosshatch of the inner Face as shown.

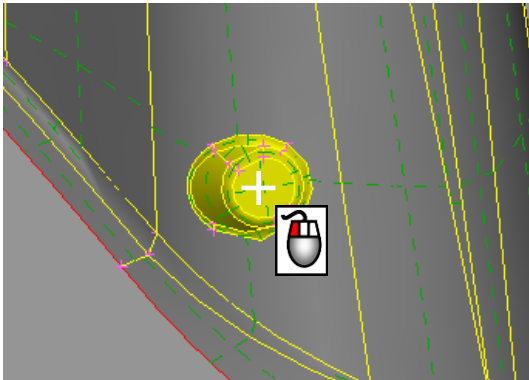



Note that as you selected from the crosshatch the Face was deleted without confirmation.


In case of accidental deletion of a Face you can always use the Undo button found in the “Utilities” toolbar to cancel the last action

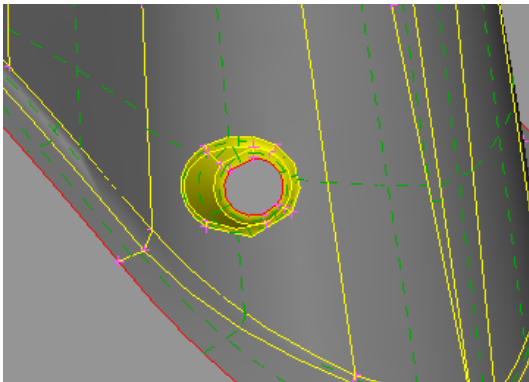


or you can use the Faces>Undelete function. ANSA previews all previously deleted Faces in white. Select one with the left mouse button to retrieve it.

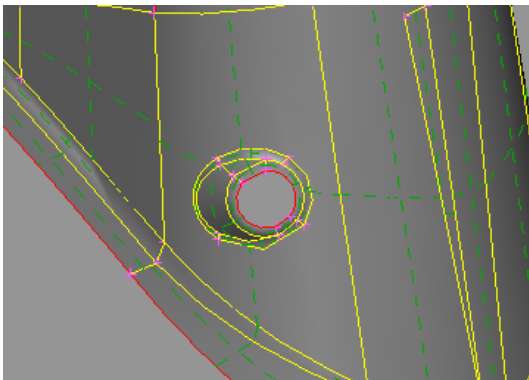



 Press “All” from the Focus group and rotate the part to view it from the outside as shown.

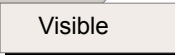
 Activate the Faces>Delete function and select with the left mouse button the crosshatch of the small round Face as shown.

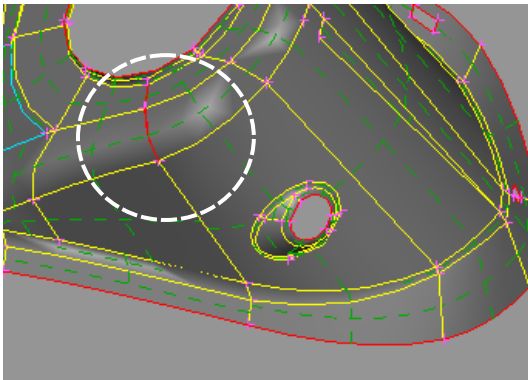


The Face is deleted without confirmation and the hole is opened.

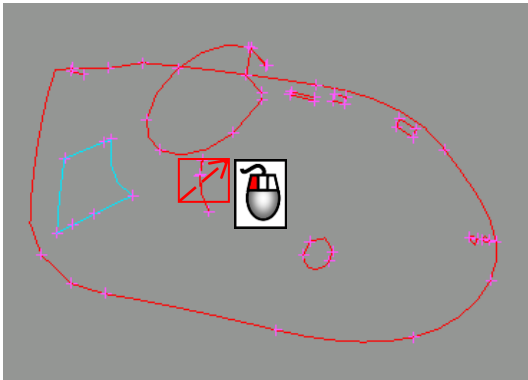


 Activate the Faces>Orient>[Visible] function to make the orientation of all the Faces uniform.





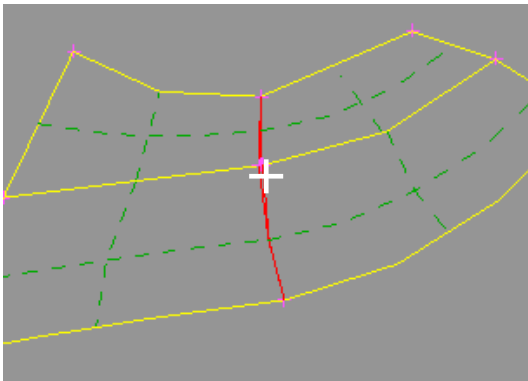
Moving now to the next area shown here in the circle.



De-activate the “Shadow” flag from the Drawing styles toolbar and the “Cross Hatches” and “Double” flags from the Visibility status toolbar, to view the remaining gaps more easily.



Activate the “Or” function from the Focus group and select with box selection the area shown.



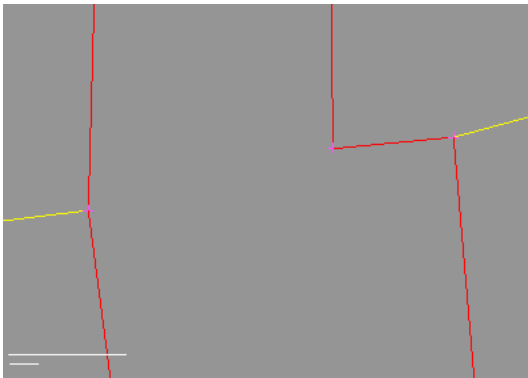
Activate back the “Cross Hatches” and “Double” flags to view the four Faces that remain visible.

Here we have a gap that was not closed by the topology operation during CAD input.



Point the cursor near the opening and zoom in real close using the F7 key. (The F8 can be used to zoom out).

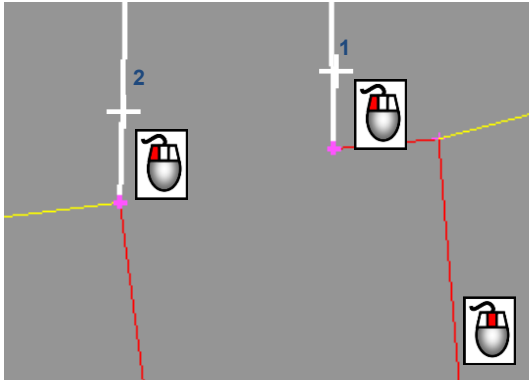
Note that as you zoom in closer and closer, two white horizontal lines appear at the bottom left corner of the display.



These lines represent graphically the tolerance values that you have specified in the “Tolerances” section of the Settings window. These are the tolerances that are used by ANSA for the automatic topology operation.

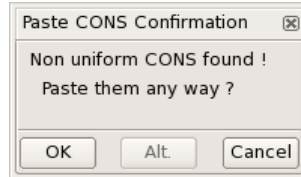
Notice that the gap here is larger than the tolerances (the CONS matching distance is set to 0.2 when the “middle” tolerance mode was selected).

As a result ANSA did not connect the Faces at this location. You will have to connect (paste) these CONS manually.



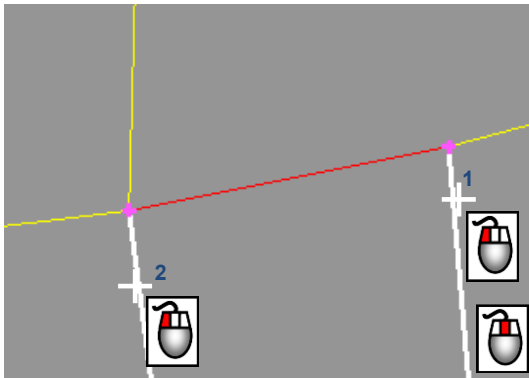
Paste

Activate the CONS>Paste function. Select with the left mouse button the CONS as shown. Confirm with the middle mouse button. As you are exceeding the tolerances a Warning window appears. Press **OK** to confirm.



Upon confirmation the first CONS is moved and pasted to the second one.

Press **OK** in all subsequent *Paste CONS Confirmation* windows that will appear.



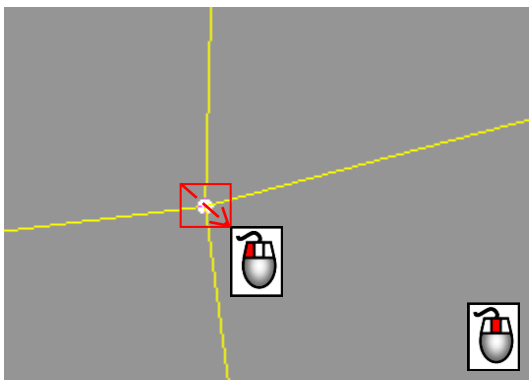
Paste

Perform the same operation for the two CONS at the bottom.

(What will happen to this small red CONS???)

Select the two CONS with the left mouse button as shown.

Confirm with the middle mouse button.



Again the first CONS is pasted on the second one.

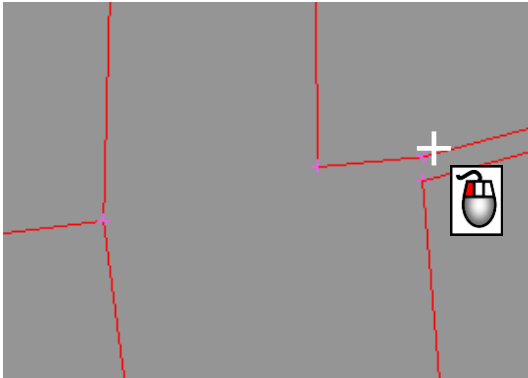
As a result a white dot appears.

This is a topological problem, as a white dot in the TOPO menu represents a collapsed CONS, that is a CONS that has its start and end points coincident.

Release

To fix such a problem you must first release all topological conditions at that location.

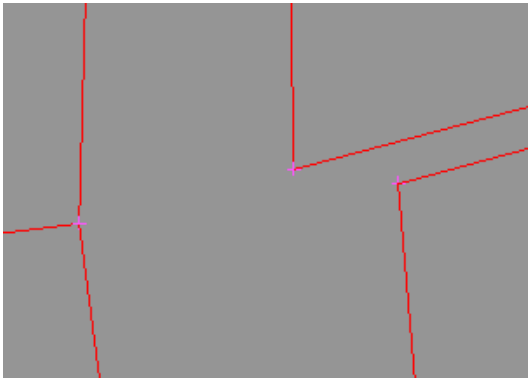
Activate the Hot Points>Release function and make a box selection of the area shown.



All topological conditions are released. The Hot Points>Release function can always be used to correct improperly pasted CONS.

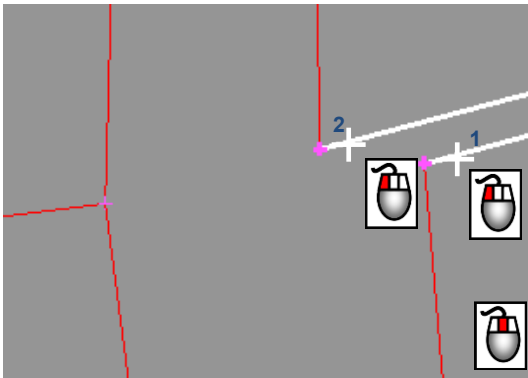
As a result all CONS now appear in red, and five distinct Hot Points are visible. To be able to paste these CONS in pairs you must ensure that they are properly segmented.

Delete Activate the Hot Points>Delete function and select with the left mouse button the Hot point shown on the left.



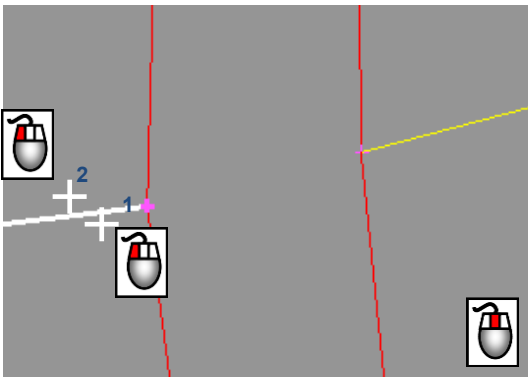
The Hot Point is deleted leaving one uniform CONS at that edge of the Faces.

Now the CONS can be pasted correctly in pairs.



Paste Activate the CONS>Paste function and select the CONS as shown on the left.

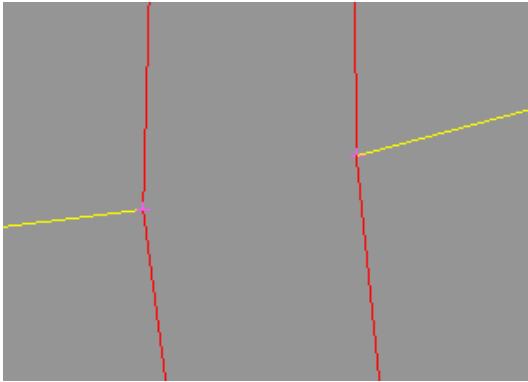
Confirm with middle mouse button.



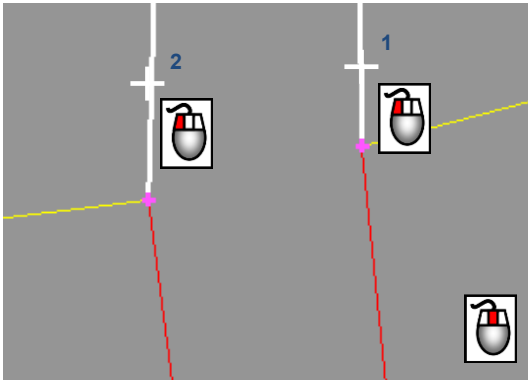
Paste Perform the same on the opposite side. (Note that if you have not exited the function you can proceed with the selection of CONS without pressing the Paste button).

Here, although you cannot distinguish them, there are two red CONS which are superimposed.

Confirm with the middle mouse button.

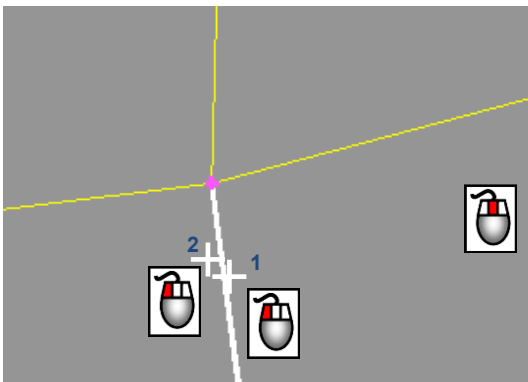


Now you have a gap that can be closed.



Still in the Paste function select the CONS with the left mouse button as shown.

Confirm with the middle mouse button.



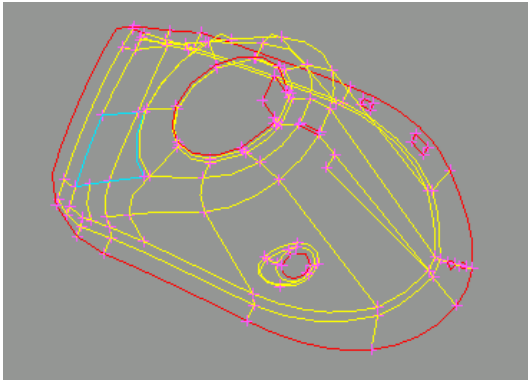
Now select the two CONS shown here.

Confirm with middle mouse button.



The problem is fixed.

Topology is correct with yellow CONS and a single Hot Point at the center.



Press “All” from the Focus menu to bring all Faces back to visible.

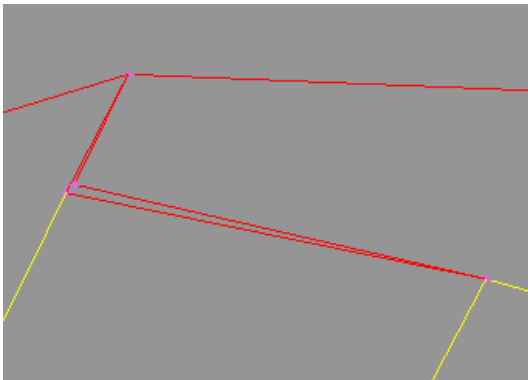


Press F9 to fit all to the screen.



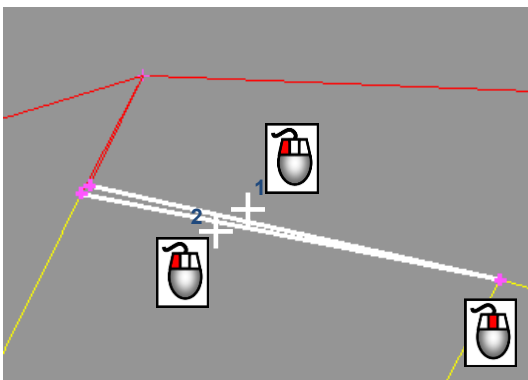
De-activate the “Cross Hatches” and “Double” visibility flags.

Some problematic red CONS that remain are indicated in the picture on the left.



Activate the “Double” visibility flag and zoom in close to that area.

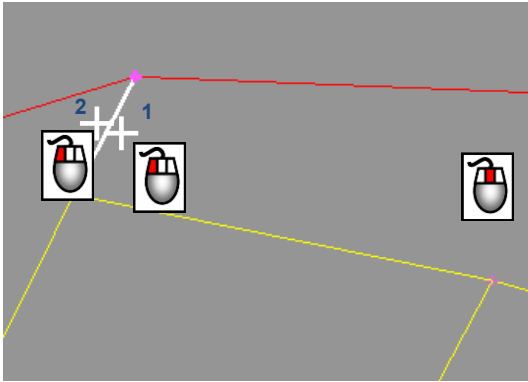
Some CONS are not connected.



Activate the CONS>Paste function and select them with the left mouse button as shown.

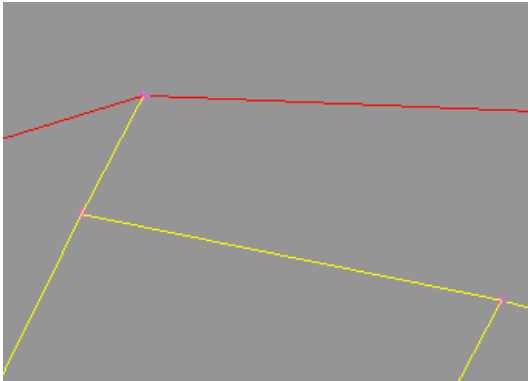
Remember, when you Paste CONS, the first CONS moves and is pasted on the second one.

Confirm with the middle mouse button.



Select also the two remaining CONS.

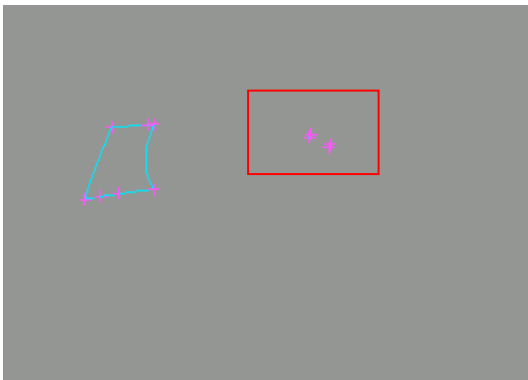
Confirm with middle mouse button.



The geometry here is now clean.



Press F9 to fit all to the screen.

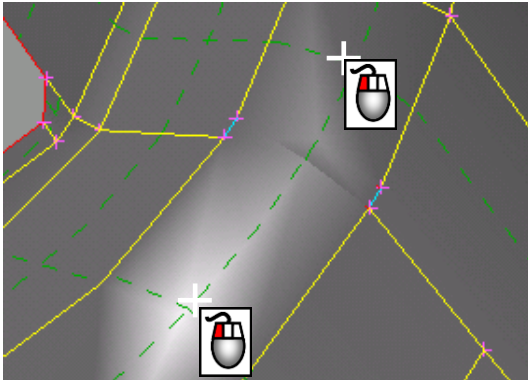


De-activate the “Single” and “Double” flag buttons from the Visibility status toolbar.

Only the cyan CONS remain visible.

Cyan CONS show connectivity of more than two Faces and are usually present at T-junctions. If there are no T-junctions present, then usually cyan CONS imply the existence of multiply defined Faces.

Zoom in to the area on the right as shown.

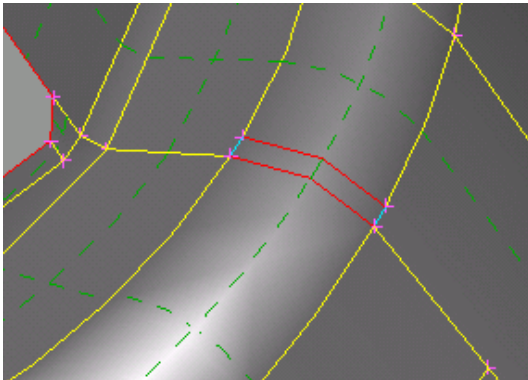


Activate back the “Double” and “Single” bounds. Activate also the “Shadow” and “Cross Hatches” buttons.

The resolution is a bit coarse to clearly identify the problem.

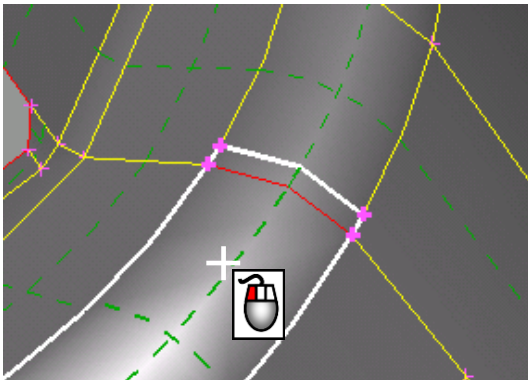
Fine

Activate the Auxiliaries>Fine function and left click once on the crosshatch of both Faces.



The resolution is better.

This is an area of overlapping Faces.

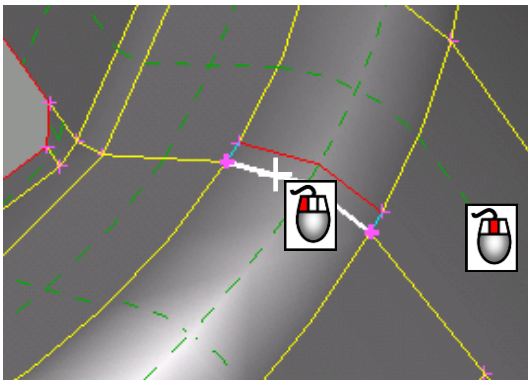


Info

To inquire, activate the Faces>Info function and click on one red CONS.

ANSA highlights the whole Face.

You can do the same for the other red CONS to see the overlap.

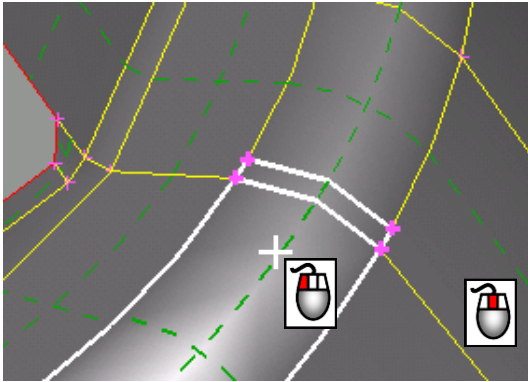


Project

Activate the CONS>Project function.

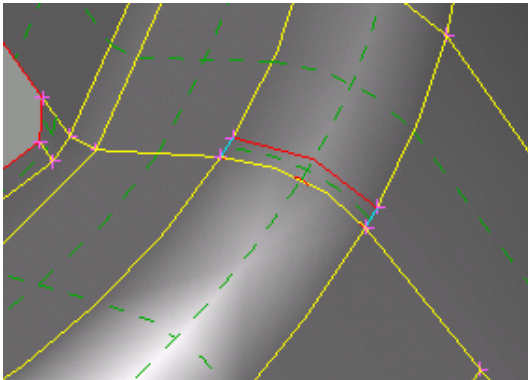
Select with the left mouse button the CONS shown.

Confirm with middle mouse button.

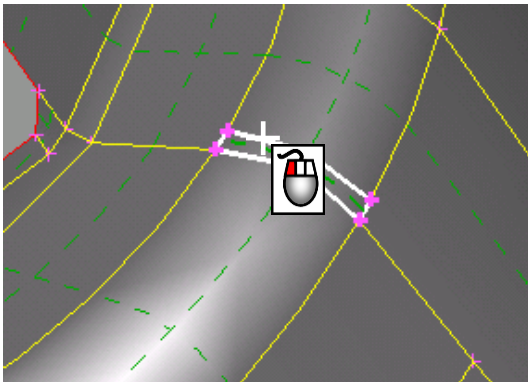


Next select the Face to be trimmed, from its crosshatch.

Confirm with middle mouse button.

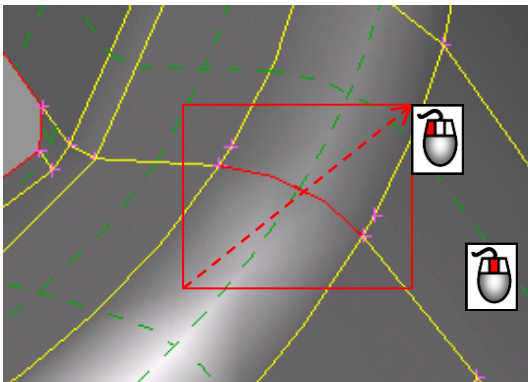


The Face is trimmed.



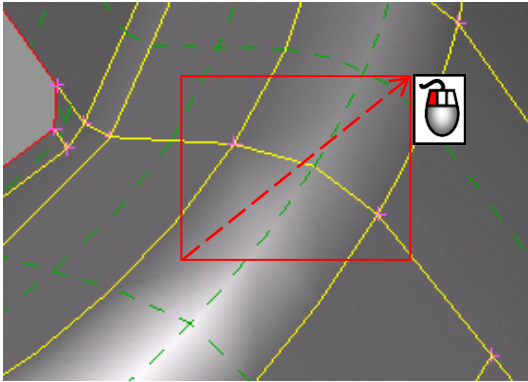
Activate the Faces>Delete function and select the excess Face.

Press **Delete** in the *Faces Delete List* window that appears.



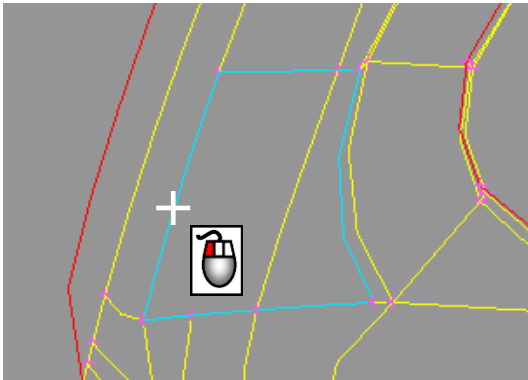
A gap remains.

Activate the Faces>Topo function, select with a box selection the area shown on the left and confirm with the middle mouse button.



Delete

Activate the Hot Points>Delete function and use box selection to delete all the remaining unnecessary Hot Points.



Finally zoom in to the area where cyan CONS are present.

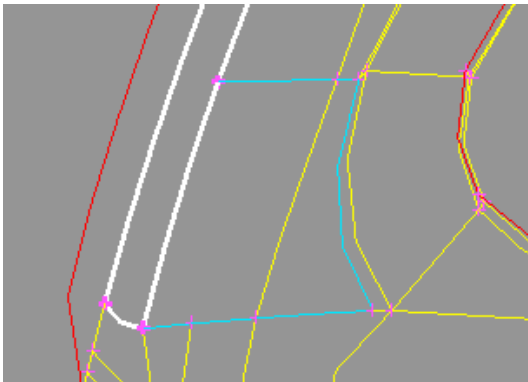
De-activate “Cross Hatches” visibility so as not to accidentally select a Face from its crosshatch. De-activate the “Shadow” mode also.

To identify and correct the problem you will use the Faces>Delete function and its preview capabilities.

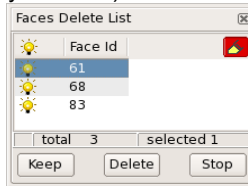
Delete

Activate the Faces>Delete function.

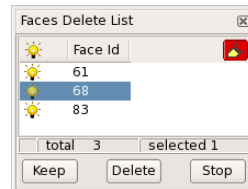
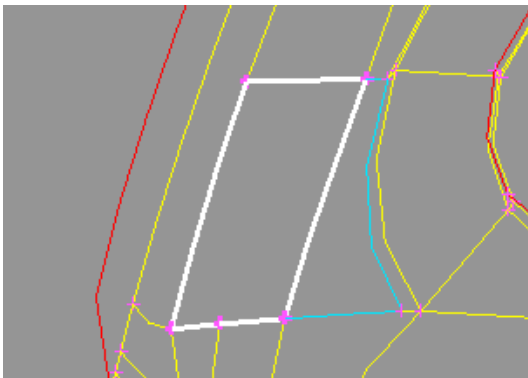
Select the Face from one cyan CONS as shown.



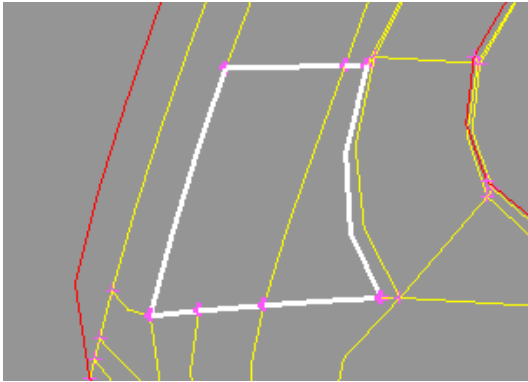
The *Faces Delete List* preview window appears, indicating that 3 Faces are connected at that CONS. The first one (1 of 3) is highlighted. (! Note the order of preview may be different in your case).



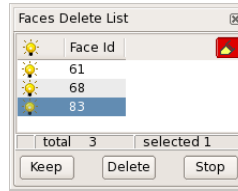
Press the **KEEP** button (or K in the keyboard) to keep this Face and proceed to the next one.



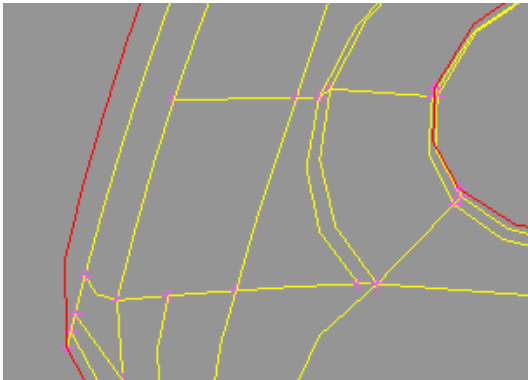
The second Face (2 of 3) is then previewed. Again press the **KEEP** button (or K in the keyboard) to keep the Face.



Finally, the third Face is previewed.
This Face must be deleted, as it is redundant.



Press the **DELETE** button (or D in the keyboard)



The Face is deleted.

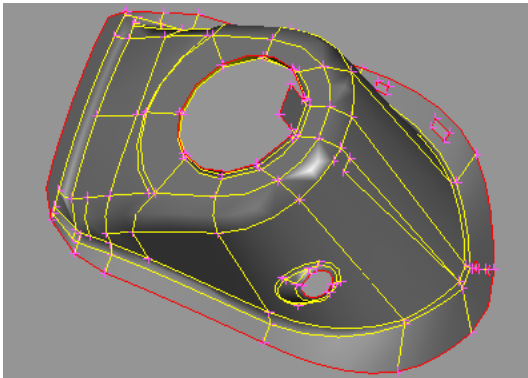
Only yellow CONS remain.

The geometry is clean.

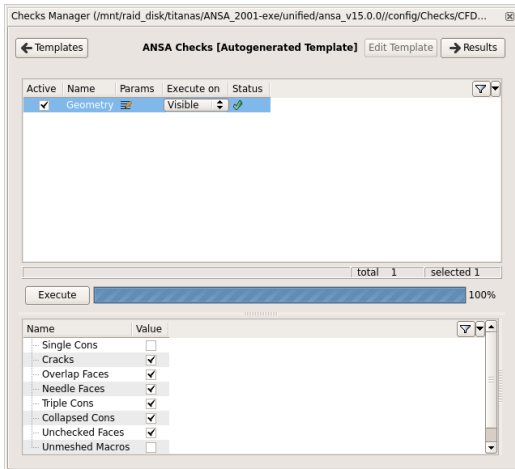
Press the ESC key to exit the function.



One last check by de-activating “Double” visibility flags leaves red CONS only at expected locations, i.e. free edges and openings.

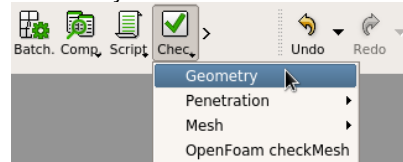


Activate the “Shadow” and “Double” flag buttons.



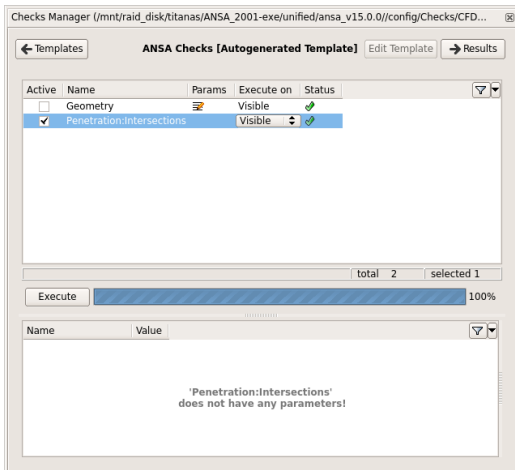
At this point you can check the geometry of the model again for any remaining problematic areas.

Click the “Checks” button and select “Geometry”.

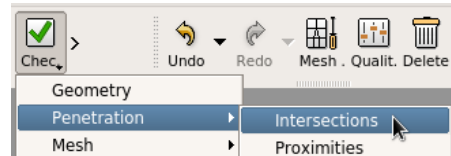


In the *Checks Manager* window that opens click **Execute**.

ANSA checks the geometry of the model and finds no problematic areas.



Next click the “Checks” button and select Penetration>Intersections.



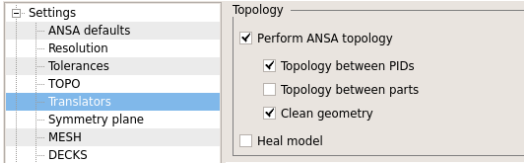
Click **Execute** in the *Checks Manager* window.

ANSA checks for intersecting geometry and finds no errors.

The geometry of the model is clean.

Close the *Checks Manager* window.

Save the file by selecting the “Save” option from the “File” menu.



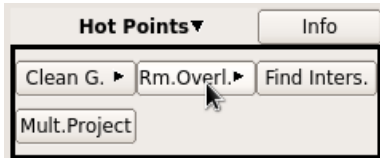
At this stage, you could optionally start again from the beginning, reopening the IGES file, but this time prior to doing so, click on the “Settings” button



and activate the **Clean Geometry** flag in the “Topology” options group found in the “Translators” section of the Settings window. Your Tolerances should be kept in Middle again. Several of the problems you previously fixed manually, can be solved automatically.

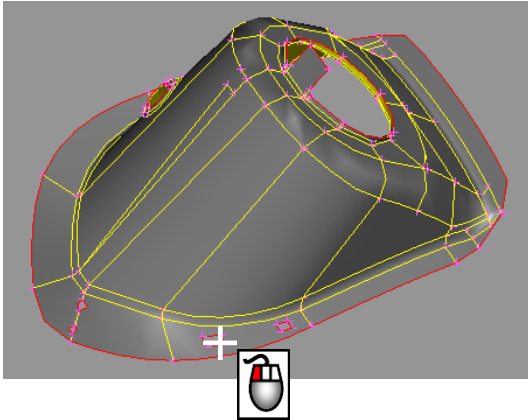
Rm.Overl.

In addition, you can use the Hot Points>Rm.Overl> [Visible] function by first clicking on “Hot Points” to bring the hidden menu forth as shown.



This function automatically trims overlapping Faces, and will fix one of the two cyan CONS problem areas. An identical example of that case is the third picture on p.37.

5. Defeaturing

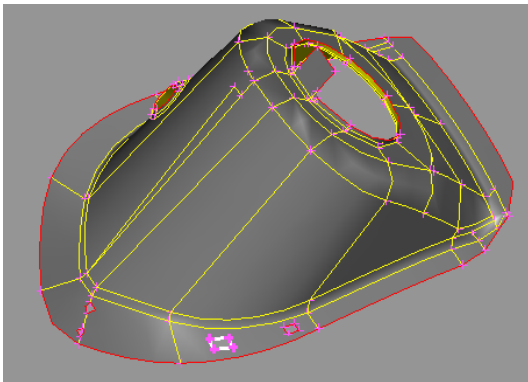
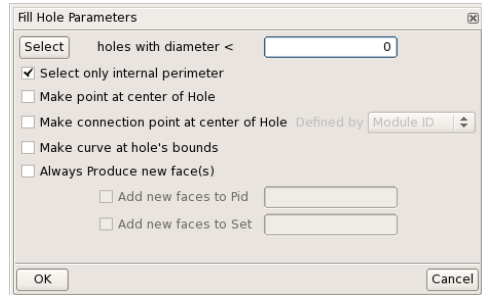


Some small openings exist at the bottom flange of the part. You will remove these, as they are too small to be included in the mesh.

Fill Hole Activate the CONS>Fill Hole function.

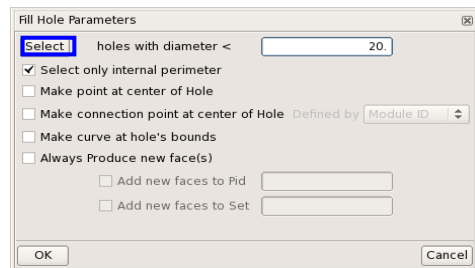
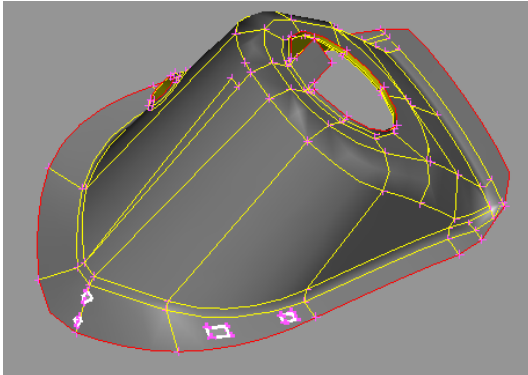
The *Fill Hole Parameters* window opens.

Select with the left mouse button the opening shown.

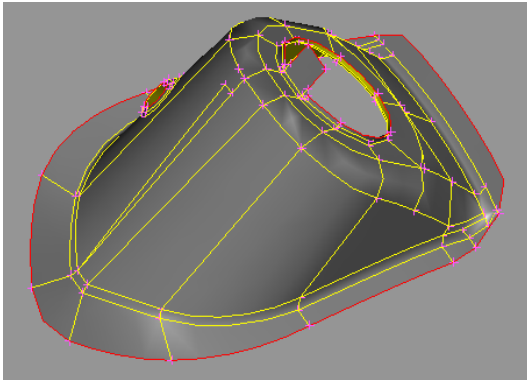


The selected opening (red CONS perimeter) is highlighted.

Alternatively you can enter a value of 20 in the “holes with diameter <” entry field and press the **Select** button.



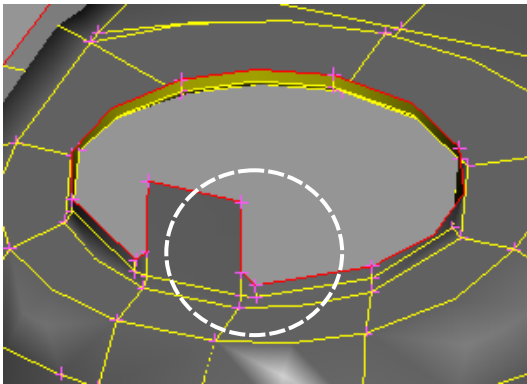
All the openings with an effective diameter smaller than the input value are selected automatically.



Press **OK**.

All the selected openings are filled.

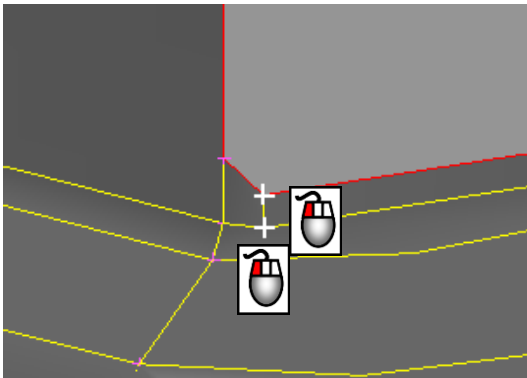
Press **ESC** to exit the Fill Hole function.



Move to the top of the part.

Here there is a small non-uniform flange.

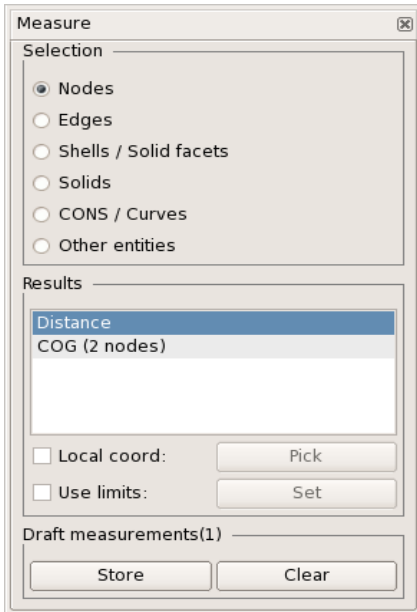
Zoom to the area in the white circle as shown on the left.



Activate the “Measure” function from the Utilities Toolbar.

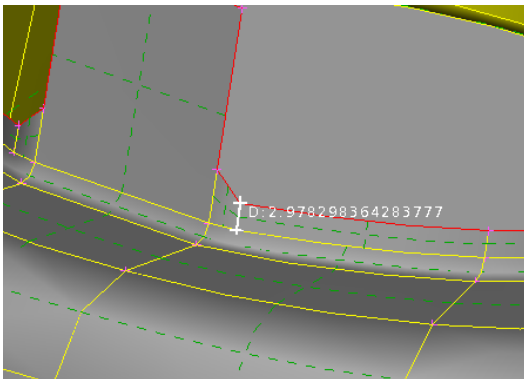
The *Measure* window opens, where by default the “Nodes” selection mode is active.

Select with the left mouse button the two Hot points shown.



As long as a selection has been made, the possible available results are listed in the Measure window.

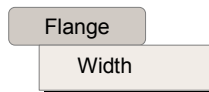
The distance as shown on the screen and reported in the Text Window is less than 3mm.



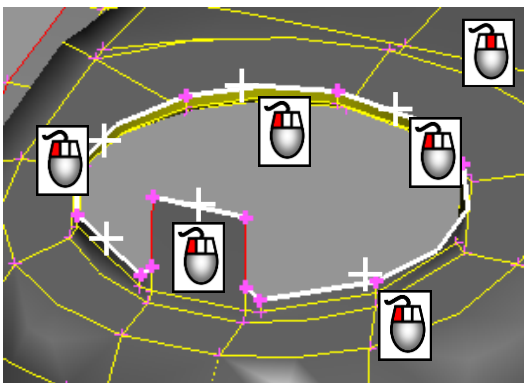
You will replace this flange all around with a constant height flange of 10mm.

Close the *Measure* window.

Click on “Faces” to bring the hidden menu forth.



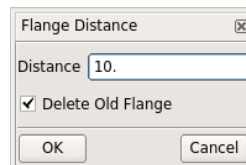
Activate the Faces>Flange>[Width] function.



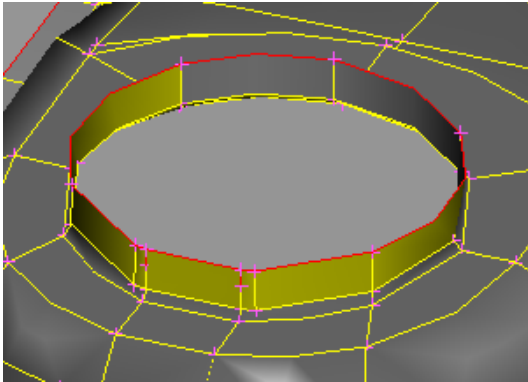
Select all the red CONS shown (in the direction of flange extension only) with the left mouse button.

Confirm with the middle mouse button.

The *Flange Distance* window appears. Type a value of 10 and activate the Delete Old Flange flag.

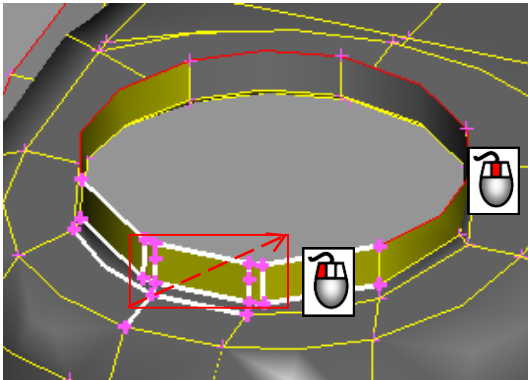


Press **OK**.



The Faces of the old flange are deleted and new Faces of the specified height are created.

Note some remaining red CONS.

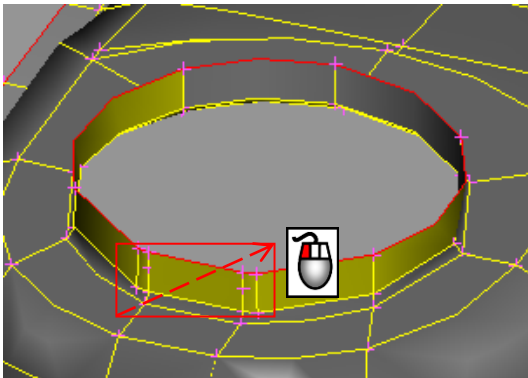


Topo

Activate the Faces>Topo function.

Make a box selection of the CONS as shown.

Confirm with the middle mouse button.



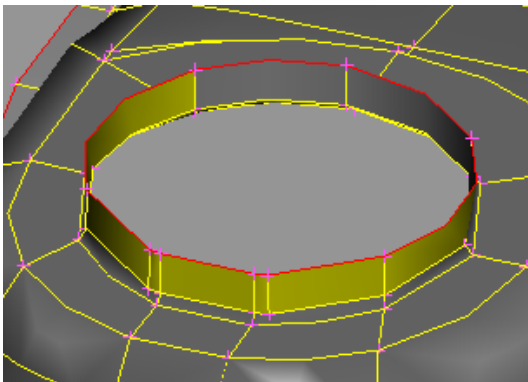
Topology is applied.

The CONS are now yellow.

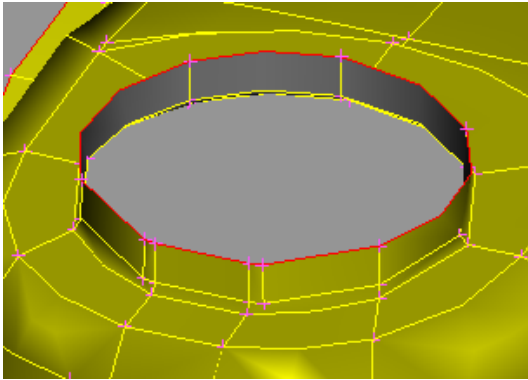
Delete

Activate the Hot Points>Delete function.

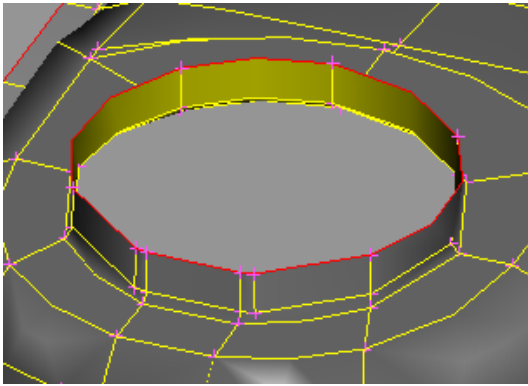
Make a box selection of the area shown.



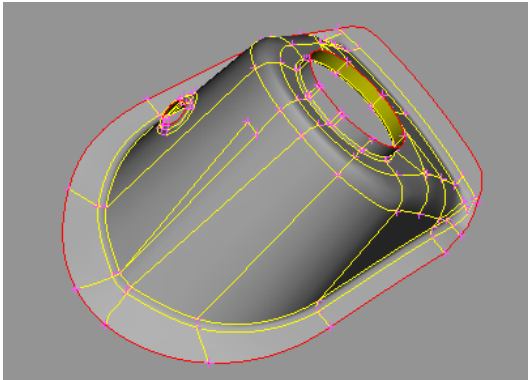
The unnecessary Hot Points are deleted.



Finally press the Faces>Orient function to assign a uniform orientation to all visible Faces.



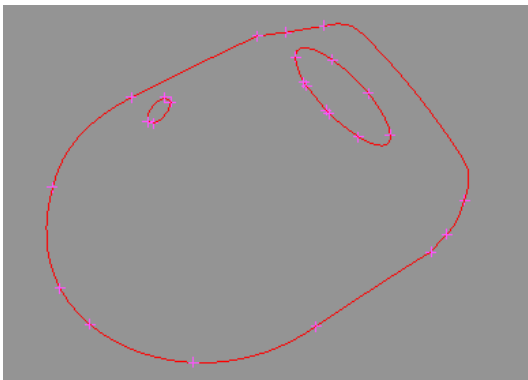
Press again the Orient button to flip the current orientation if necessary.



Press F9 to fit all to the screen.

The defeaturing process should now be complete.

You will make some final checks.



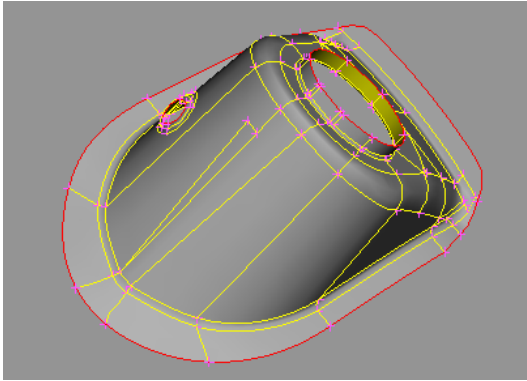
De-activate the “Shadow”, “Cross Hatches” and “Double” visibility flags.

You should only see three closed paths of red CONS corresponding to the outer perimeter and two openings of the part.

If not, correct the problems.



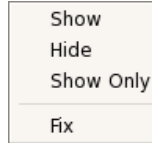
Additionally, you can perform the “Checks>Geometry” and “Checks>Penetration>Intersections” operations and see that no errors are reported.



Activate again the “Shadow” and “Double” flags. Note in case that there are Faces that fail the Shadow operation, the so called **Unchecked Faces**, then these Faces are reported on the Screen

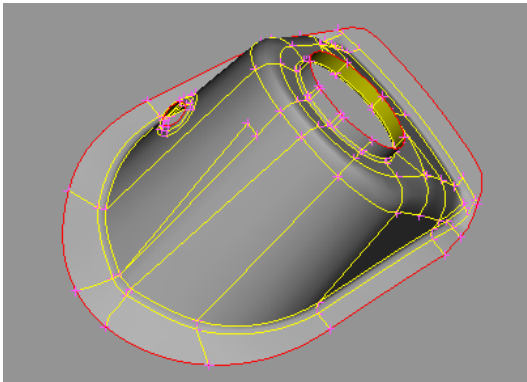
Unchecked : 2

Using the Context Menu (right click menu) of the unchecked faces, these can be isolated on the screen in order to be corrected.



Such Faces usually are thin needle shaped Faces or Faces that have bad Surface description.

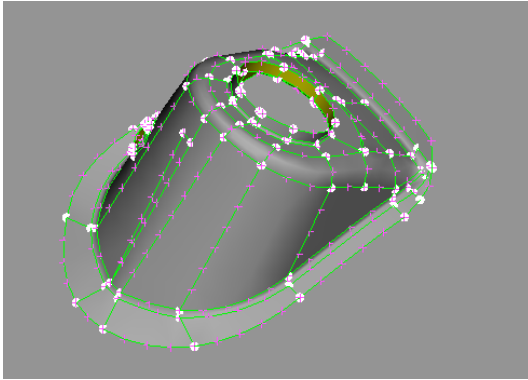
If in your case no **Unchecked Faces** are reported on the screen, then no such problems exist in your model.



Press “All” from the Focus group to make all the Faces visible.

The geometry defeaturing is completed. Save the file by selecting the “Save” option from the “File” menu.

6. Shell meshing – Uniform size mesh



Switch to MESH menu.



In MESH menu the Faces are now Macro Areas and the CONS are Perimeter Segments. The Hot Points are here displayed as white dots. Along the Perimeter Segments you can see magenta Perimeter nodes. These are placed initially according to the CONS Resolution settings. In this case the nodes are placed every 20mm, except some regions where you have used the Fine function to change the Resolution locally.

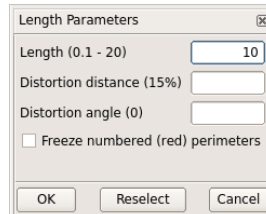
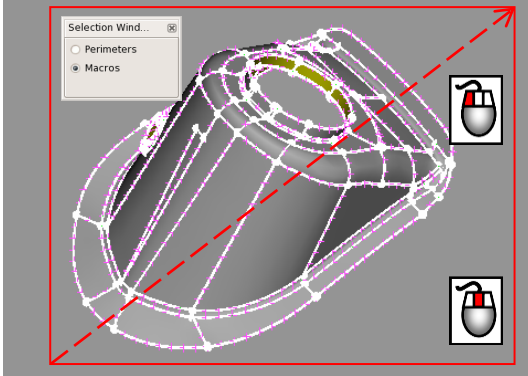
You will assign a uniform distance between Perimeter Nodes, or Element Length.

Length

Activate the Perimeters>Length function.

Select all the Macro Areas with the left mouse button making a box selection. Confirm with the middle mouse button.

The *Length Parameters* window opens.



Type in a value of 10 in the Length field, leave the other fields empty (default values will be used) and press Enter.

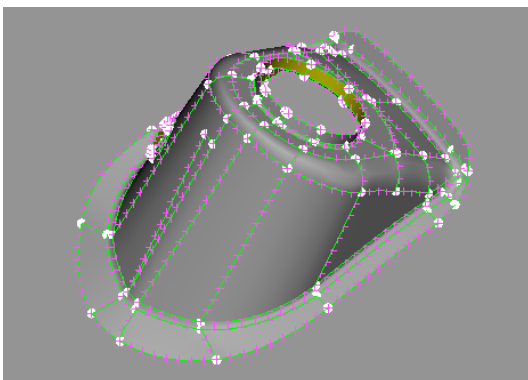
The new element length is applied to all the Perimeter Segments of the selected Macro Areas.

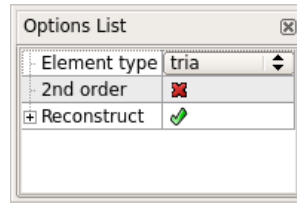
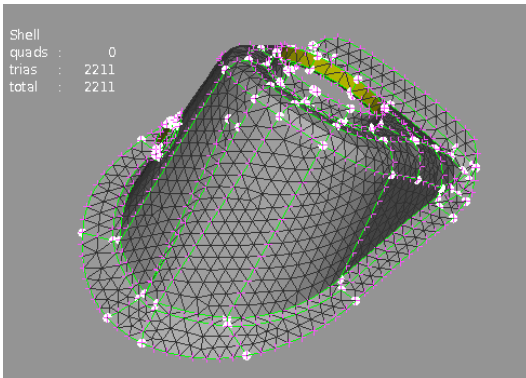
(Note that although we started with a CONS Resolution length (and correspondingly Element Length) of 20mm, we will mesh the part with an element length of 10, so as to keep more details).

Adv.Front

Activate the Mesh Generation> Adv.Front function in order to

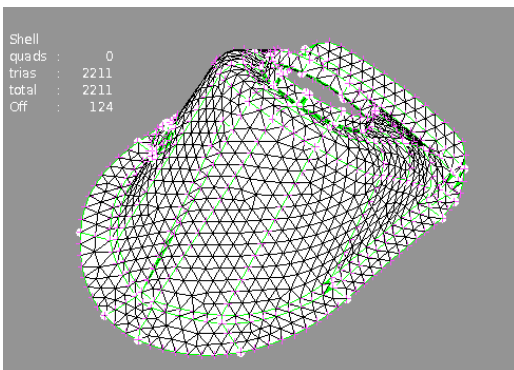
mesh the Macros with the Advancing Front algorithm, which is recommended for uniform size meshes.





In the Options window, in the lower right corner of the screen, make sure the “Reconstruct” option is enabled so it will run right after the meshing algorithms in order to get a high quality mesh. For element type select “Tria”. Make a box selection of all the Macros and confirm with the middle mouse button.

All the Macro Areas are meshed with the Advancing Front algorithm. The number of the visible elements is given in a text legend on the left (Note, the values may vary slightly).



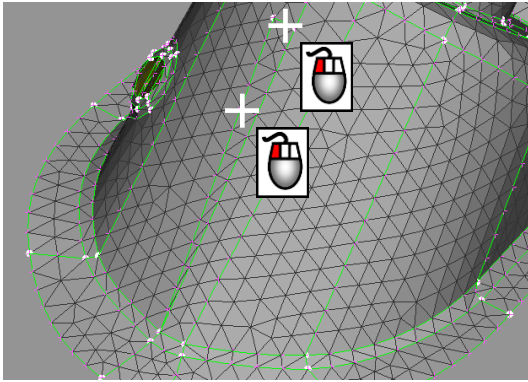
In order to check the quality of the generated Mesh, switch to “Hidden” view mode by pressing the corresponding button located in the Drawing styles toolbar at the bottom of the screen.



Hidden mode displays all the violating elements according to the current Quality Criteria settings (these settings are accessed by pressing the F11 key).

The number of the violating elements is reported in the legend as Off. They are also colored according to the type of violation (also explained in the legend).

You can isolate the violating elements on the screen by right clicking on the “Off” word in the legend and selecting “Show only”. Select “All” from the focus toolbar to make all the elements visible again.



You will use some functions to manually improve the quality of the generated mesh. As it will be shown later on, these Off elements can be handled automatically.

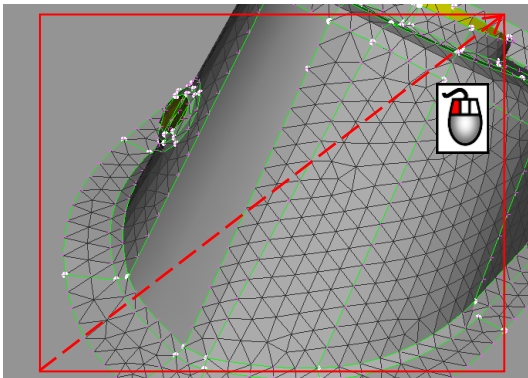
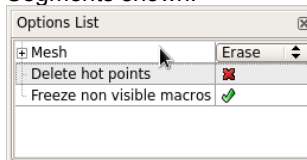


Activate the “Shadow” mode.

Zoom in to the front of the part. Notice how the elements are squeezed in the small triangular shaped Macro Area.



Activate the Macros>Join function ensuring that the Mesh option is set to “Erase” and the “Delete hot points” flag is de-activated in the options area in the lower right corner of the screen. Select with the left mouse button the two Perimeter Segments shown.



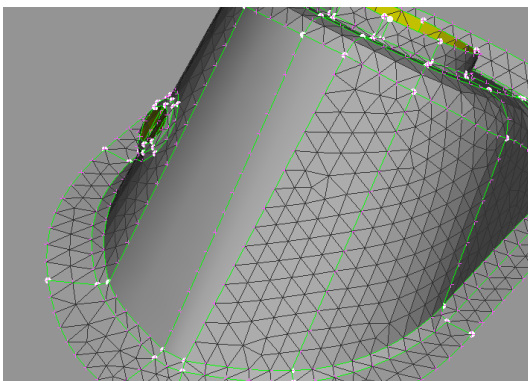
The Perimeters are removed and the two Macros are joined to form a larger one. The mesh is of course erased.

Note that if you join a Perimeter by mistake, you can retrieve it using the Macros>Release function, just picking it from the preview).

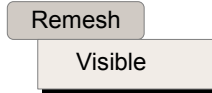
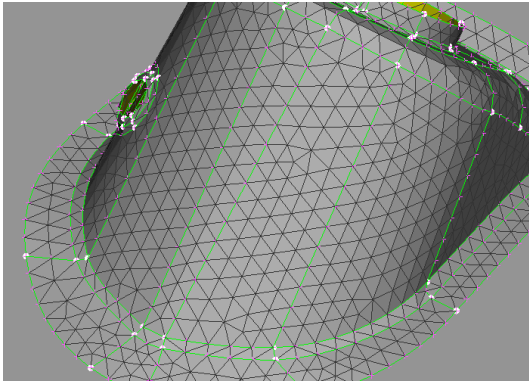
Notice that some Hot Points are left over.



Activate the Hot Points> Delete function and select them with the left mouse button making a box selection as shown here.



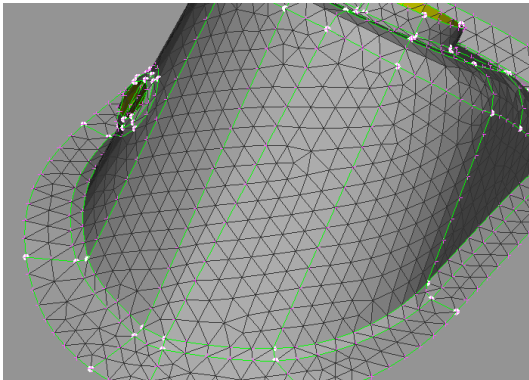
The Hot Points are removed and the Perimeter Segments are connected into longer, more uniform Segments. The mesh of the neighboring Macro is also erased.



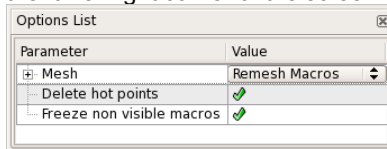
Activate the Mesh Generation> Remesh>[Visible] function.

The erased Macro Areas are remeshed with the same meshing algorithm, in this case the Advancing Front algorithm.

As you can see every action that results in the modification of a Macro, leads to the erasure of its mesh. Then you have to use the function Mesh Generation>Remesh to remesh it.

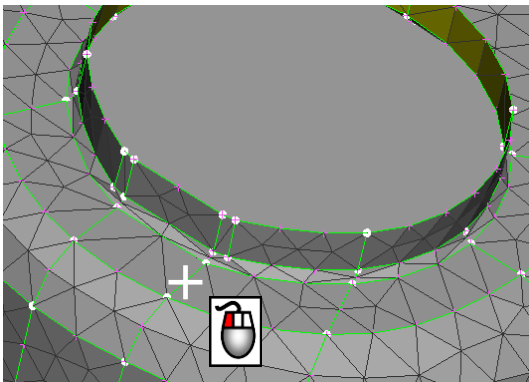


With the Join function is activated, activate the “Delete hot points” flag and set the Mesh option to “Remesh Macros” in the Options window in the lower right corner of the screen.



This option will allow the automatic deletion of any remaining Hot Points and the automatic remesh of the Macros after any Join operations.

Move to the top round flange area.

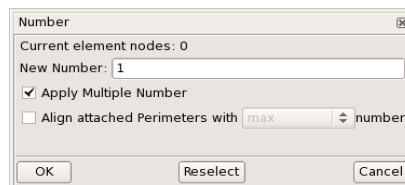


Notice here a Perimeter Segment that is too short to have even one Perimeter Node.

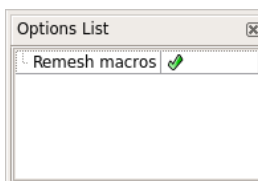


Activate the Perimeters> Number function.

Select with the left mouse button the small Perimeter Segment as shown and confirm with the middle mouse button. The Number window opens, where the current nodal number (0) is reported.

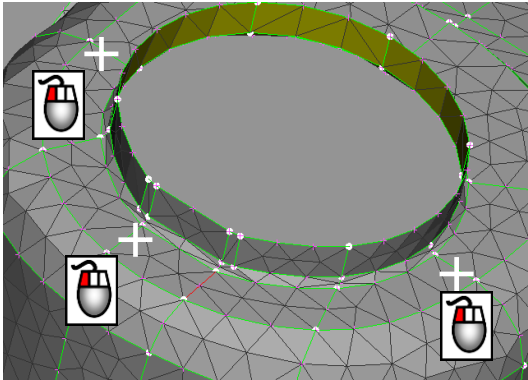


In the Options window enable the Remesh Macros option so that the Macros are remeshed after each modification is made to the mesh and causes its erasure.



Make sure that this option is from now on enabled on all those functions that support it.

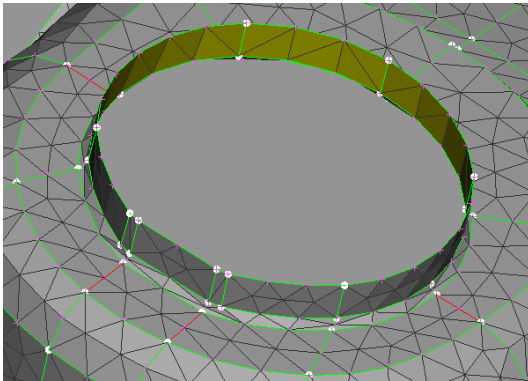
Type in the value 1 in the “New Number” entry field and press Enter.



One nodal number is assigned to this Segment, which is now colored red, to indicate the nodal number has been specified explicitly and not via the element Length.

While still in the Perimeters>Number function, right click on the three Perimeters shown on the left to assign the same nodal number of 1.

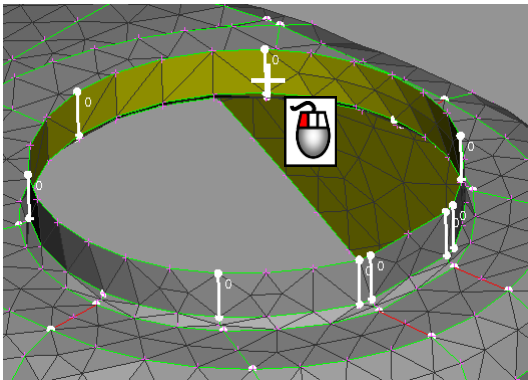
Press ESC to exit the function.



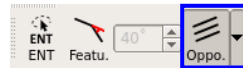
The mesh is better.

Moving now to the round flange itself.

Suppose you want to have two rows of elements along the flange.



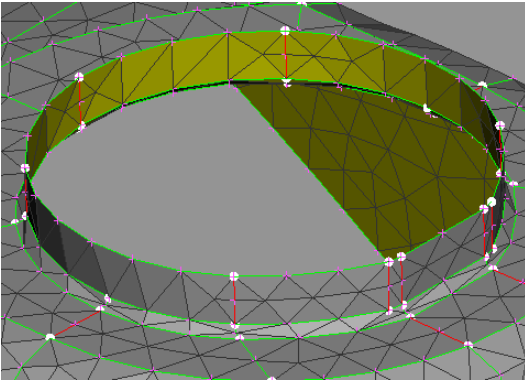
Activate again the Number function as above and activate the “Opposite” option in the Selection Assistant toolbar.



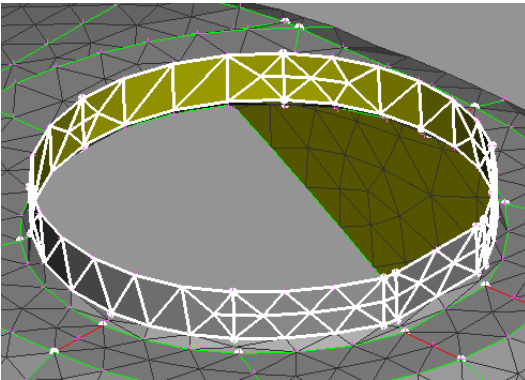
Select the perimeter shown on the left with the left mouse button. ANSA automatically selects all the Perimeter Segments that are parallel to the selected Segment in a continuous path of Macro Areas.

Confirm with the middle mouse button.

Type in a value of 1 and press Enter to assign it to all the selected Segments.



However the automatic remeshing does not give the desired result. In this case the Advancing Front algorithm did not produce two rows of elements everywhere.



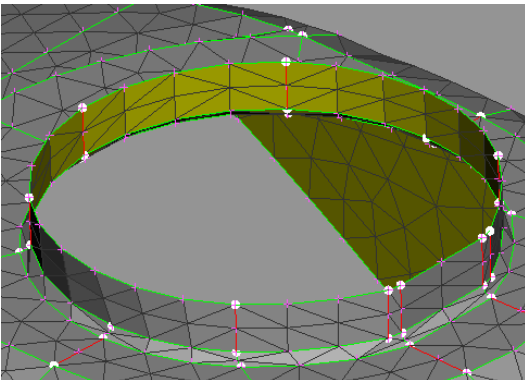
Map

Re-Generate

Activate the Mesh Generation>Map>[Re-Generate] function. The Re-

Generate option means erase and mesh from new with the current algorithm and settings).

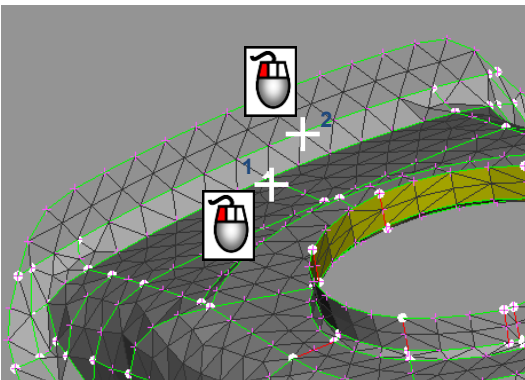
Select with the left mouse button the Macros of the flange as shown, and confirm with the middle mouse button.



ANSA erases the mesh and meshes with the MAP algorithm to produce the desired result.

(Actually you will see in the more advanced tutorial that mesh treatment of flanges, holes, fillets etc. can be prescribed in ANSA in advance and applied automatically).

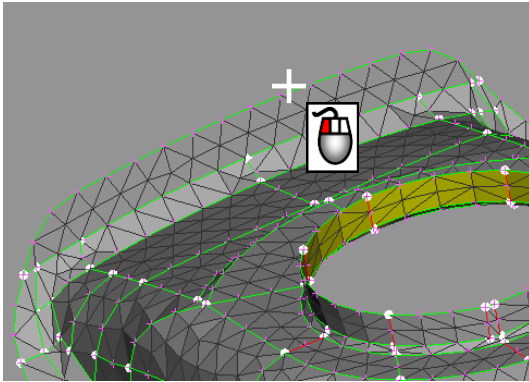
Press ESC to exit the function.



Move on to the top flange of the part. Notice how the elements are misaligned. Making a Cut along the symmetry plane of the part would improve the mesh.

Proj.Cut

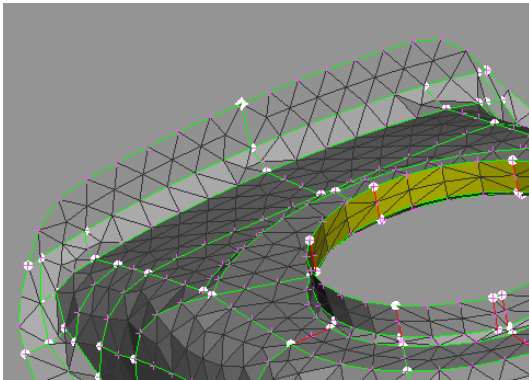
Activate the Macros>Proj.Cut function. Select the bottom Hot Point and the opposite top Perimeter as shown on the left.



A new Hot Point is created at the projection location and a Cut between the two is made.

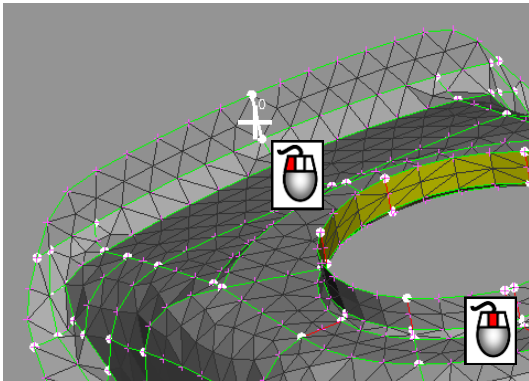
ANSA remeshes automatically.

While still in the Proj.Cut function select the top Perimeter, as shown, in order to continue from the last position with the cut operation further upwards.



ANSA projects, cuts and remeshes.

The last Hot Point remains highlighted for further cut operations, but in this case press ESC to exit the function.

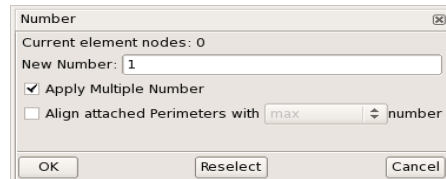


Number

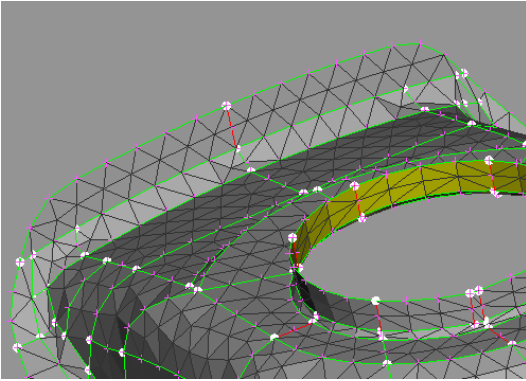
Finally use the Perimeters> Number function to place one node in the short vertical Perimeter as shown. Make sure to activate the entities selection mode in the Selection Assistant toolbar.



Confirm with the middle mouse button.

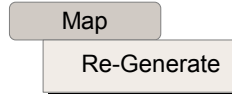
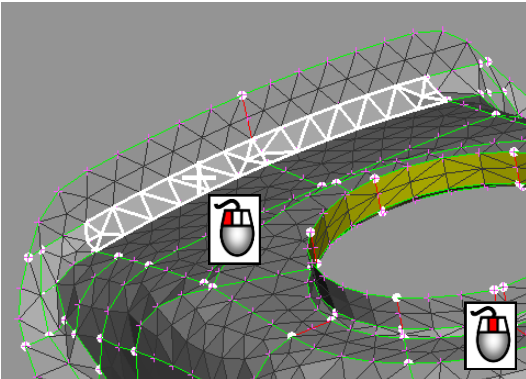


Enter the value 1 and press Enter.



ANSA applies the new nodal number and remeshes.

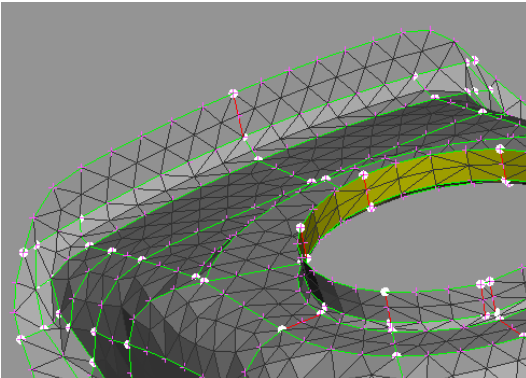
Still you can improve the mesh of the fillet region, by meshing it with a different algorithm.



Activate the function
Mesh Generation>
Map>[Re-Generate].

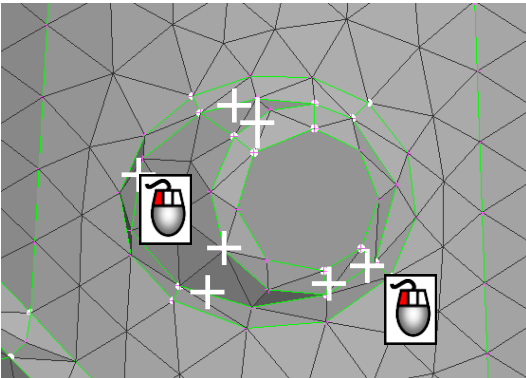
Select the Macro Areas with the left mouse button as shown. You can de-select with the right mouse button if required.

Confirm with the middle mouse button.



ANSA erases the current mesh and meshes from new with the MAP algorithm.

The resulting mesh better represents the curved fillet.

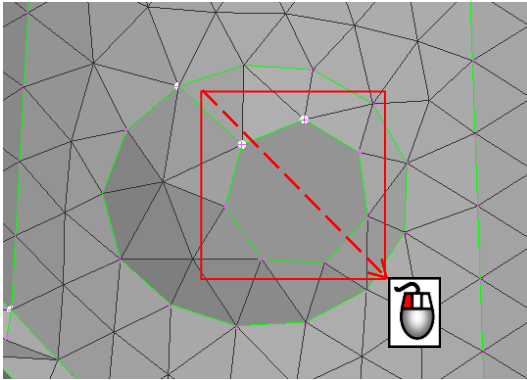


Move to the area of the round protrusion.

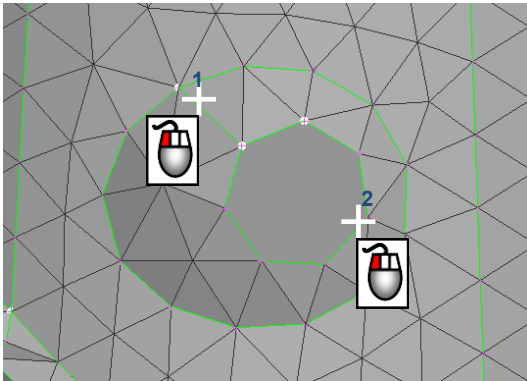
Notice how many small elements are confined in narrow Macro Areas.



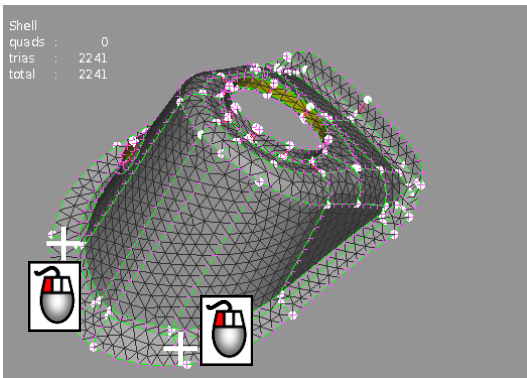
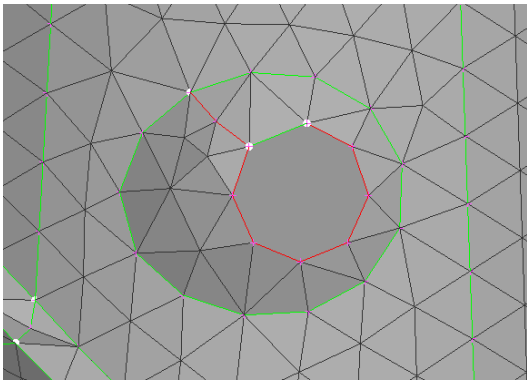
Use the Macros>Join function to join the Segments, and create larger Macros.



Delete Then activate the Hot Points> Delete function and select with box selection the area shown.

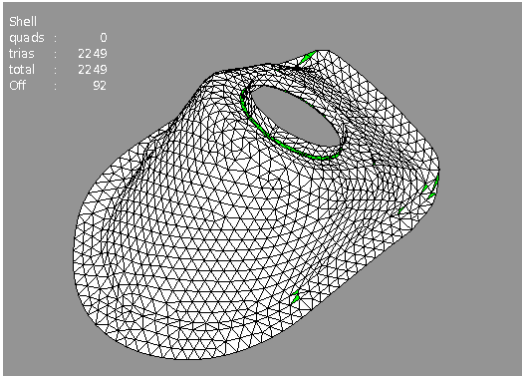


Number Optionally you can use the Perimeters>Number function and assign nodal numbers, 1 and 6 nodes respectively to the two remaining Segments as shown.



F9 Press F9 to fit all in the screen.

Join Join these two Segments that interrupt the mesh of the flange.



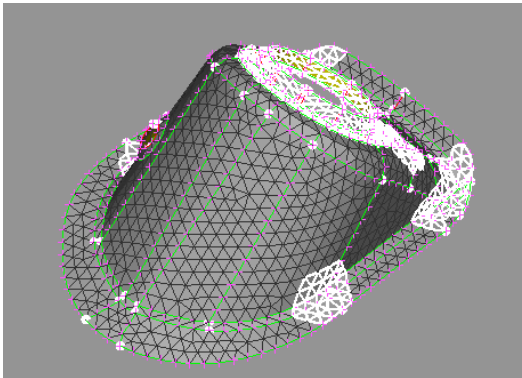
Switch to the “Hidden” mode to check the quality of the mesh.



Note that the number of the Off elements has dropped.

Up to this point, for training reasons, you have used various functions in order to manually improve the quality of the mesh. ANSA is able to improve the mesh quality automatically in an efficient and easy way.

This is achieved via the Shell Mesh>Reshape>[Advanced] function which automatically performs local Perimeter Join and mesh reconstruction.



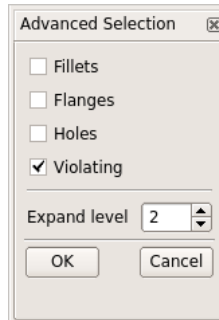
Activate the “Shadow” mode again.

Reshape.

Advanced

Activate the Shell Mesh>Reshape>[Advanced] function.

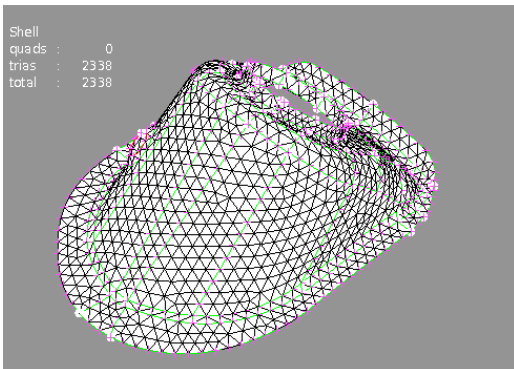
In the *Advanced Selection* window that opens keep the “Violating” option and “Expand level” value 2.

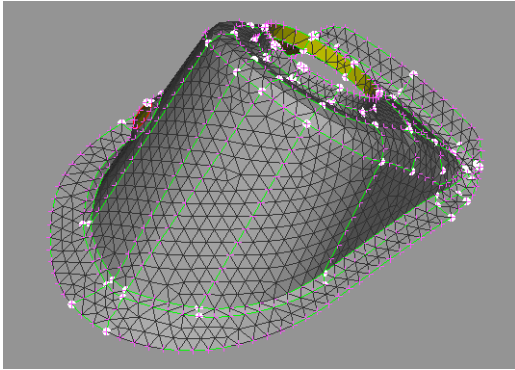


Press **OK** and then confirm with the middle mouse button.

ANSA performs local Perimeter Join operations and Reconstruction to fix the problems.

Switch to “Hidden” mode again and see that no Off elements are reported in the legend.



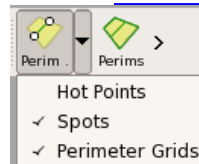
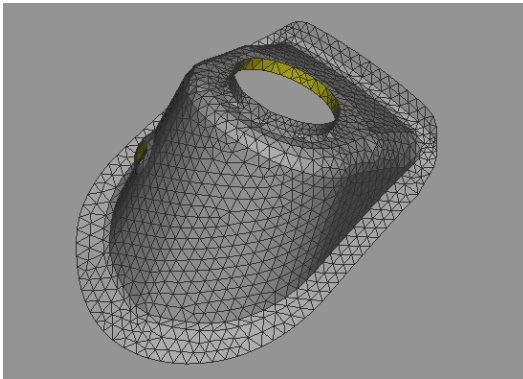


Activate again the “Shadow” mode.

The uniform mesh is complete.

Note that the number of elements displayed in the legend may differ depending on the actions that were taken during Macro manipulation.

(You can output the shell mesh in any of the available output formats, through File>Output).



You can de-activate the visibility flag buttons for “Perims” and “Hot Points”

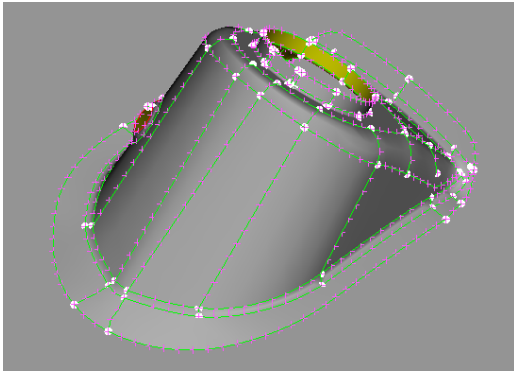
(under the Perimeters Points flag) in order to view the mesh better.

Notice that in case the “Perimeter” flag is deactivated the “Perimeters Points” get also not visible.

Activate the visibility flags for “Perims” and “Hot Points” again.

Save your work by selecting the File>Save option.

7. Shell meshing – Variable size mesh



Now you will create a variable size, curvature dependent mesh.

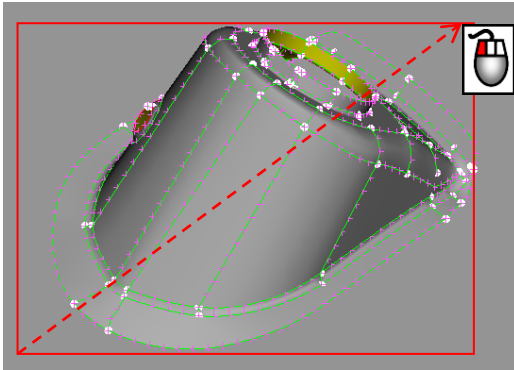
Erase

Visible

Activate the Mesh Generation>Erase>[Visible] function to erase the uniform mesh.

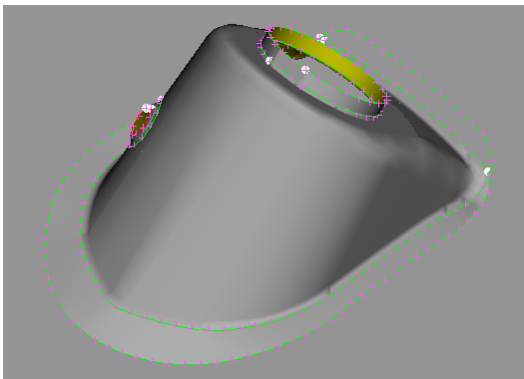
Before starting a new mesh, it is important to restore the model to its original state before any joining or perimeter modification operations.

ANSA can automatically restore the model to its initial state before the manual join actions were performed, following the steps described below:

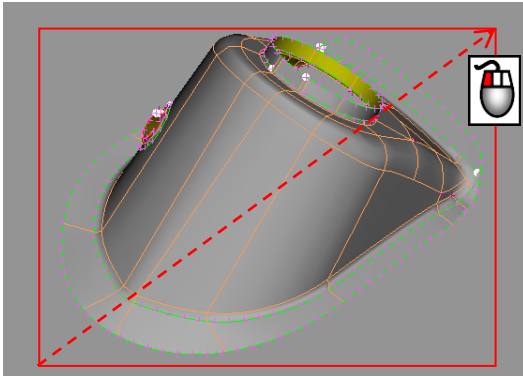


Join

Activate the Macros>Join function and make a box selection of all the Macros as shown on the left.



All the Macros belonging to the same PID are joined.

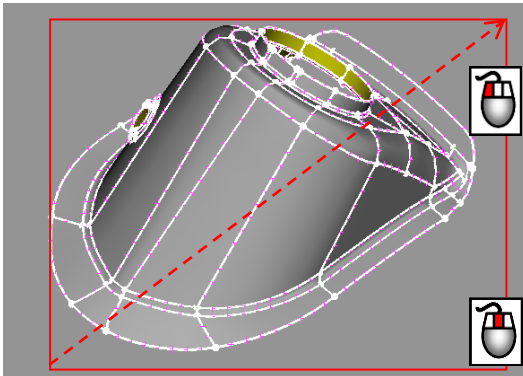


Release

Activate the Macros>Release function and make a box selection again of all the Macros as shown.

! Note that the Perimeters between Macro areas that have the same Surface (underlying CAD description) and have been previously joined, cannot be retrieved with the Release function.

This is the case for the two Perimeters that were manually joined at the beginning of the Uniform mesh chapter on page 50.

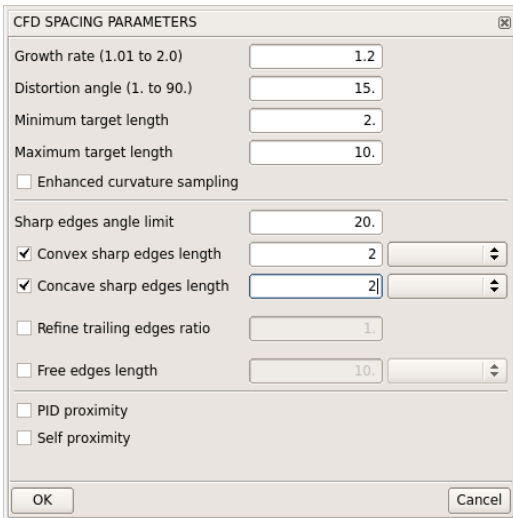


Init

Finally, activate the Perimeters>Init function to reset the Number of nodal points and the nodal distribution of the selected Perimeters and make a box selection as shown.

The selected perimeters are highlighted.

Confirm with the middle mouse button.



Spacing

Auto CFD

Activate the Perimeters>Spacing>[Auto CFD] function.

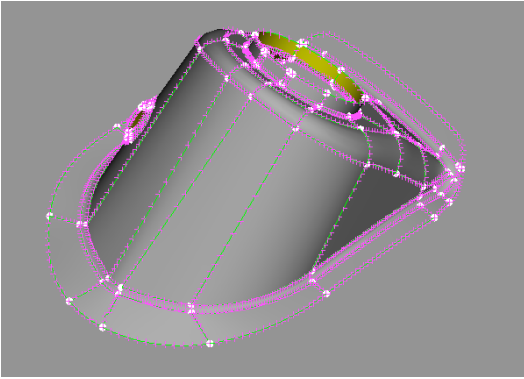
In the *CFD Spacing Parameters* window that opens enter the values as shown on the left.

Note that the resulting mesh will consist of elements with a length ranging from 2mm to 10mm.

No sharp edges are present in this model. However you can activate the convex and concave sharp edges length flags. These values are used to set the element length on Perimeters between Macros that form a sharp angle. It is advisable in general to refine such edges because of their importance to the flow.

The Perimeters>Spacing>[Auto CFD] function will automatically apply a curvature dependent refinement to all visible Perimeters.

Press **OK**.

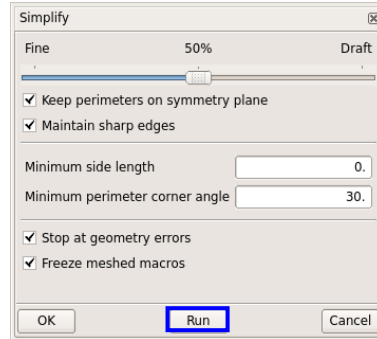


Prior to meshing, ANSA can perform an automatic Perimeter Joining and Macro simplification procedure, resulting eventually in a better mesh quality.

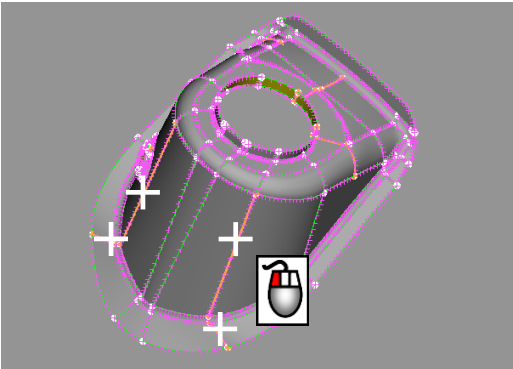
Simplify

Activate the Macros>Simplify function.

Keep the default options in the *Simplify* window that opens and click **Run**.



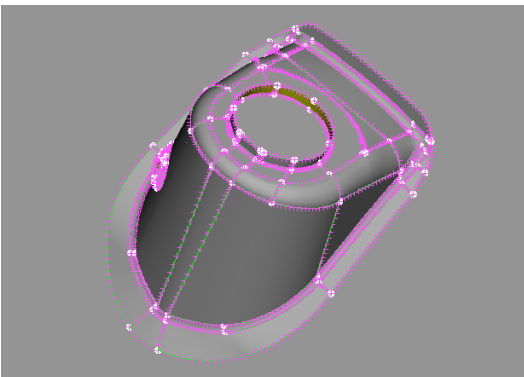
ANSA selects the Perimeters to be joined so there is no need to make any manual selections.



However, it is possible to manually add more Perimeters to be joined by selecting them with the left mouse button.

Select the four Perimeters as shown on the left.

Click **OK**.



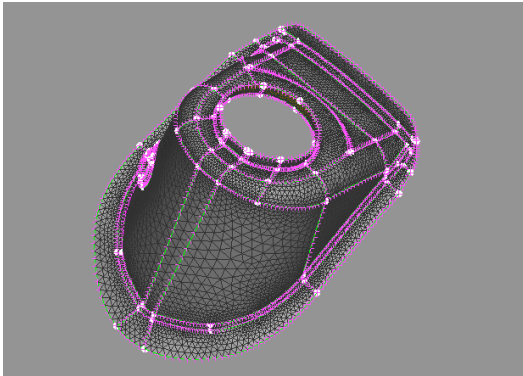
Spacing

Auto CFD

As after the Simplify function the Perimeters have been modified, the Perimeters>Spacing>[Auto CFD] function has to be re-applied.

Activate the function and in the *CFD Spacing Parameters* window press **OK** keeping the same options as before.

Now that the perimeters have been assigned the curvature dependent refinement, the mesh can be generated.



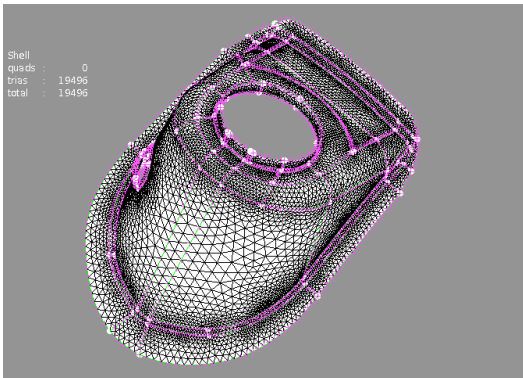
CFD

Visible

Activate the Mesh Generation>CFD>[Visible] function.

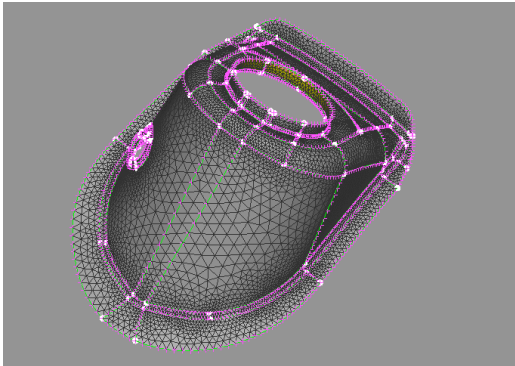
ANSA meshes the Macros with a variable size, curvature dependent mesh.

Rotate the part to observe the mesh refinement.



Switch to “Hidden” mode to check the quality of the mesh. No Off elements should be reported in the legend.

However if some Off elements exist, use the Shell Mesh>Reshape> [Advanced] function to fix the problematic areas.



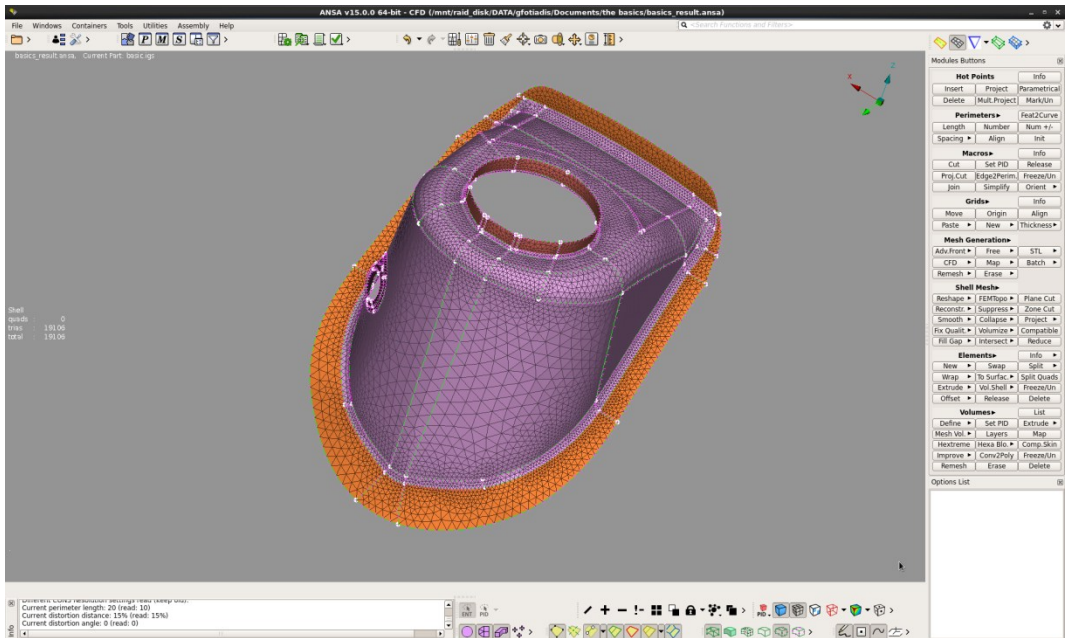
Enable back the “Shadow” mode.

The tutorial is completed.

Save the file from File>Save.

Quit ANSA from File>Quit.

8. Conclusion



In this tutorial you have followed all the basic steps to cleanup the geometry of a part, remove some small details and mesh it with shell elements, initially applying a uniform mesh and then a variable size, curvature dependent mesh. Not all capabilities were demonstrated. Refer also to the [ANSA for CFD Quick Start Guide.pdf](#) document and to the On-Line Help for the functionality of ANSA and detailed description of functions and procedures.