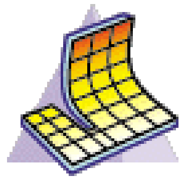
Advanced Meshing Tools

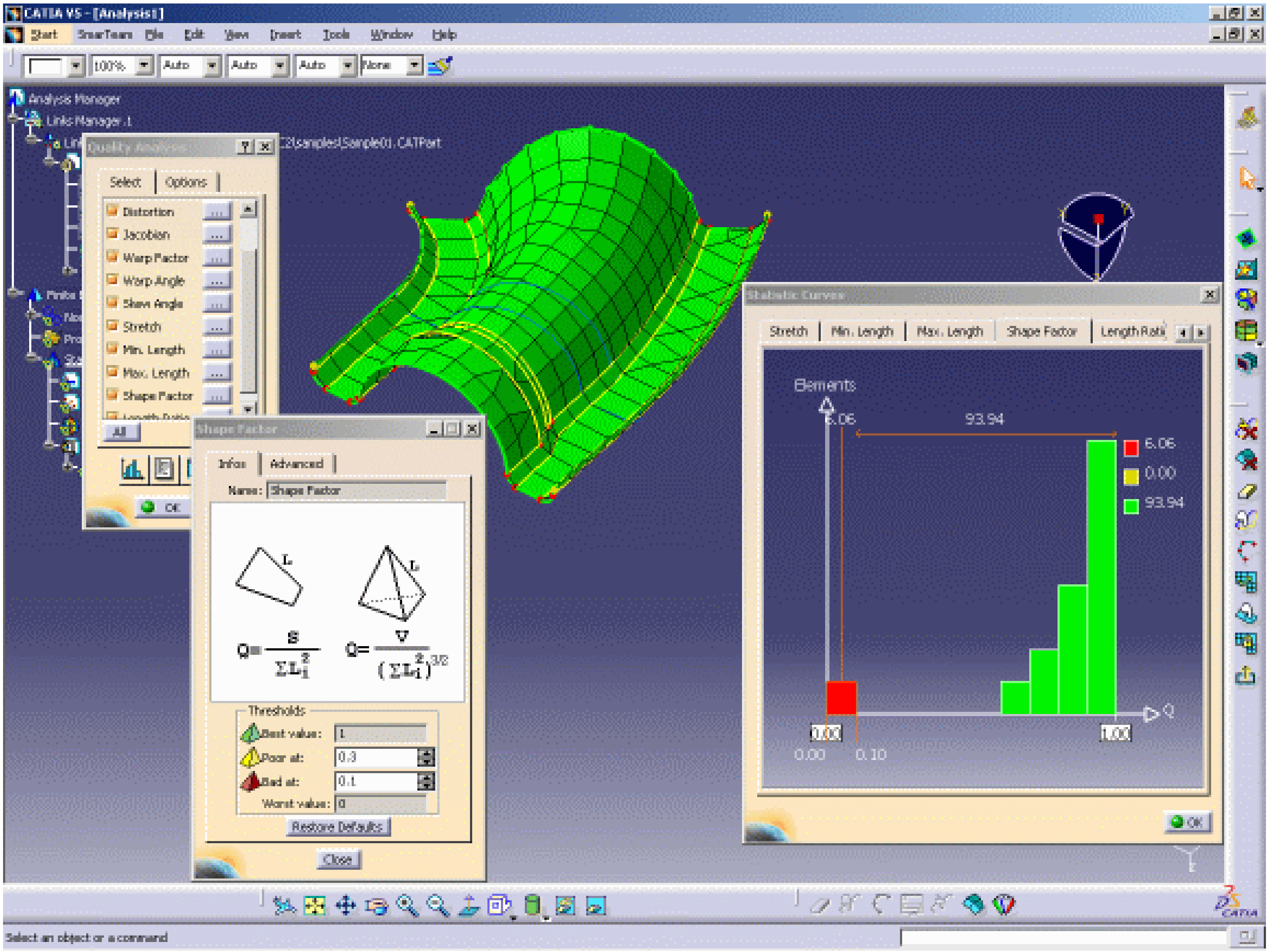


Preface

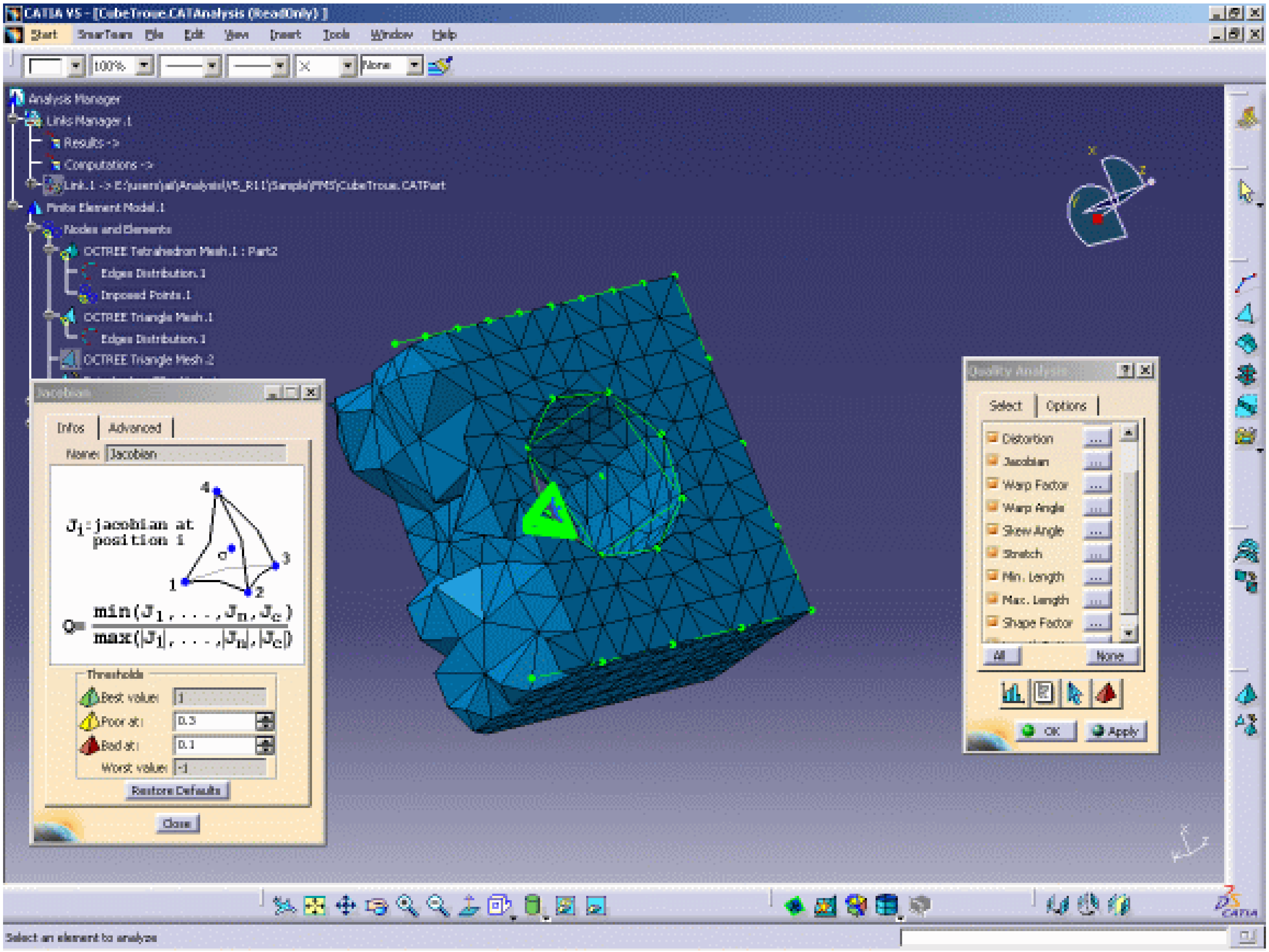
The Advanced Meshing Tools workbench allows you to rapidly generate a finite element model for complex parts whether they are surface or solid.

In other words, you will generate associative meshing from complex parts, with advanced control on mesh specifications.

To generate a finite element model for complex surface parts, you have to use the FEM Surface product.



To generate a finite element model for complex solid parts, you have to use the FEM Solid product.



Using This Guide
More Information
Conventions

Using This Guide

This book is intended for the user who needs to quickly become familiar with *FEM Surface* Version 5. The user should be familiar with basic concepts such as document windows, standard and view toolbars.

To get the most out of this guide, we suggest you start reading and performing the step-by-step tutorial "Getting Started". This tutorial will show you how to analyze a part from scratch.

The "Basic Tasks" section presents the main capabilities of the product. Each individual task is carefully defined and explained in detail.

It may also be a good idea to take a look at the "Workbench Description" section, presenting the menus and toolbars.

Finally, a "Glossary" has been provided to familiarize you with the terminology used in this guide.

Where to Find More Information

Prior to reading this book, we recommend that you read:

- Infrastructure User's Guide Version 5
- Part Design User's Guide
- Conventions chapter

Conventions

Certain conventions are used in CATIA, ENOVIA & DELMIA documentation to help you recognize and understand important concepts and specifications.

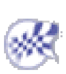
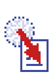
Graphic Conventions

The three categories of graphic conventions used are as follows:

- Graphic conventions structuring the tasks
- Graphic conventions indicating the configuration required
- Graphic conventions used in the table of contents

Graphic Conventions Structuring the Tasks

Graphic conventions structuring the tasks are denoted as follows:

This icon...	Identifies...
	estimated time to accomplish a task
	a target of a task
	the prerequisites
	the start of the scenario
	a tip
	a warning
	information
	basic concepts
	methodology
	reference information
	information regarding settings, customization, etc.
	the end of a task
	functionalities that are new or enhanced with this Release.
	allows you to switch back the full-window viewing mode.

Graphic Conventions Indicating the Configuration Required

Graphic conventions indicating the configuration required are denoted as follows:

This icon...	Indicates functions that are...

P1	specific to the P1 configuration
P2	specific to the P2 configuration
P3	specific to the P3 configuration

Graphic Conventions Used in the Table of Contents

Graphic conventions used in the table of contents are denoted as follows:

This icon...	Gives access to...
	Site Map
	Split View mode
	What's New?
	Overview
	Getting Started
	Basic Tasks
	User Tasks or the Advanced Tasks
	Workbench Description
	Customizing
	Reference
	Methodology
	Glossary
	Index

Text Conventions

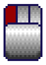
The following text conventions are used:

- The titles of CATIA, ENOVIA and DELMIA documents *appear in this manner* throughout the text.
- **File** -> **New** identifies the commands to be used.
- Enhancements are identified by a blue-colored background on the text.

How to Use the Mouse

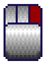
The use of the mouse differs according to the type of action you need to perform.

Use this mouse button...	Whenever you read...



- Select (menus, commands, geometry in graphics area, ...)
- Click (icons, dialog box buttons, tabs, selection of a location in the document window, ...)
- Double-click
- Shift-click
- Ctrl-click
- Check (check boxes)
- Drag
- Drag and drop (icons onto objects, objects onto objects)

- Drag
- Move



- Right-click (to select contextual menu)

What's New?

New Functionalities

Local Specifications

Modifying Local Specifications with Knowledgeware

You can modify local specifications using the **Formula** functionality.

Enhanced Functionalities

Beam Meshing

Beam Meshing

You can can mesh using parabolic 1D elements.

Surface Meshing

Setting Global Meshing Parameters

Global specifications are available at any time.

Meshing Connections

Meshing Spot Welding Connections

You can mesh hemmed spot weld.

Meshing Seam Welding Connections

You can mesh seams using compatible mode.

Quality Analysis

Checking Interferences / Intersections

You can focus on interference area.

Workbench Description

Toolbars

The interface of the Advanced Meshing Tools workbench has been improved.

The interface of the Surface Meshing workshop has been improved.

Getting Started

This tutorial will guide you step-by-step through your first Advanced Meshing Tools session, allowing you to get acquainted with the product. You just need to follow the instructions as you progress.

Before starting this tutorial, you should be familiar with the basic commands common to all workbenches.

The main tasks proposed in this section are:

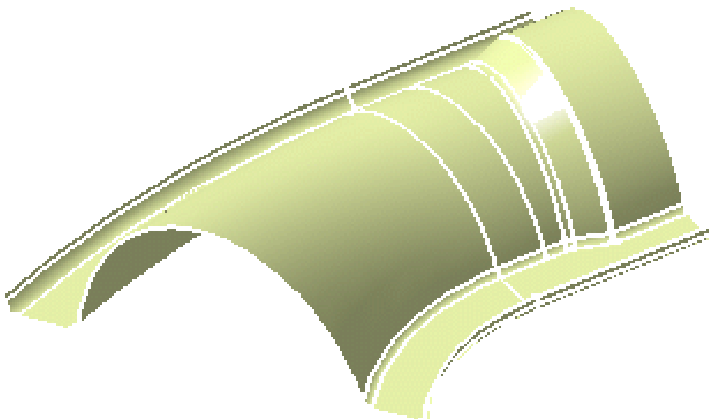
Entering the Advanced Meshing Tools Workbench
Defining the Surface Mesh Parameters
Setting Constraints and Nodes
Launching the Mesh Operation
Analyzing Element Quality
Mesh Editing
Re-meshing a Domain

Altogether, this scenario should take about 15 minutes to complete.

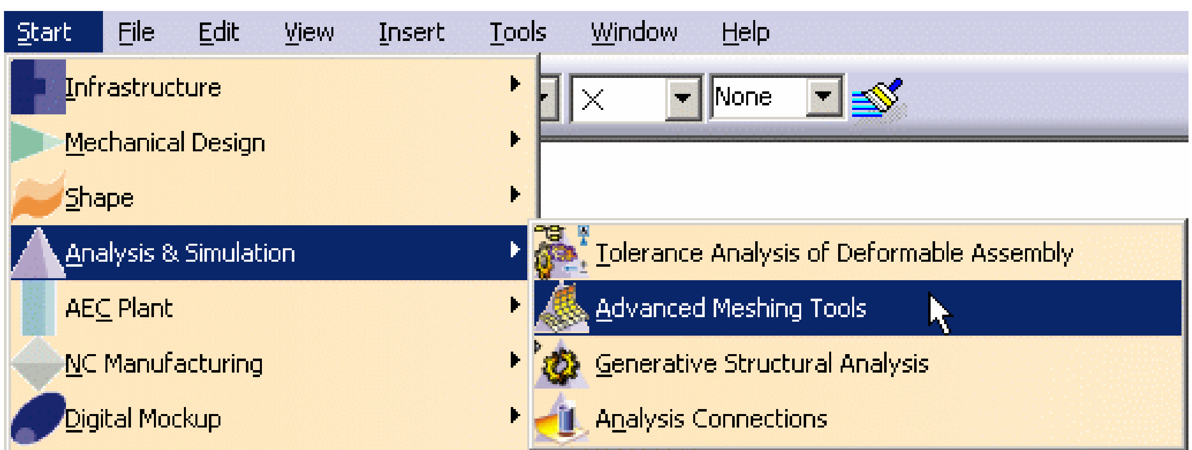
Entering the Advanced Meshing Tools Workbench

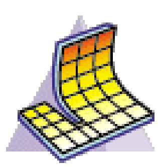
This task shows how to enter in the **Advanced Meshing Tools** workbench.

Open the Sample01.CATPart document from the sample directory.

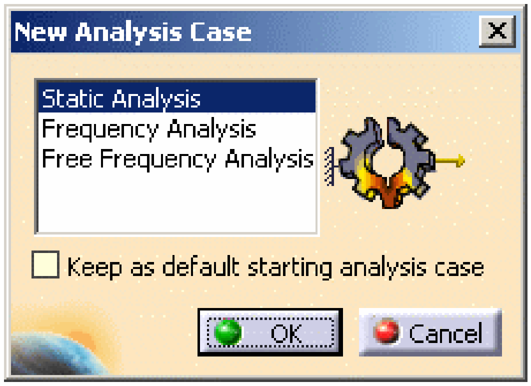


1. Select the **Start -> Analysis & Simulation -> Advanced Meshing Tools** menu.





You are now in the Advanced Meshing Tools workbench. An Analysis document is created and the New Analysis Case dialog box is displayed.



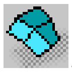
2. Select an analysis case type in the New Analysis Case dialog box. In this particular example, select **Static Analysis**.

   Optionally, you can activate the **Keep as default starting analysis case** option if you wish to have **Static Analysis Case** as default when launching the workbench again.

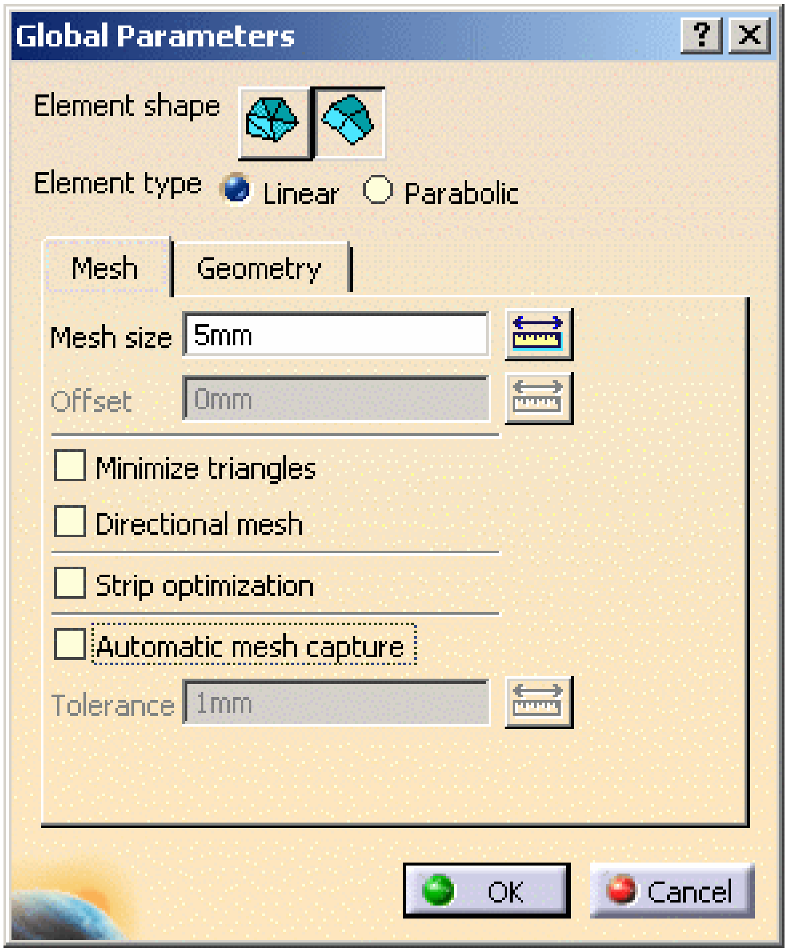
3. Click **OK** in the New Analysis Case dialog box.

Defining the Surface Mesh Parameters

This task will show you how to define, on a single surface part, a mesh type and global parameters.

1. Click the **Advanced Surface Mesher** icon .
2. Select the part.

   The Global Parameters dialog box appears.



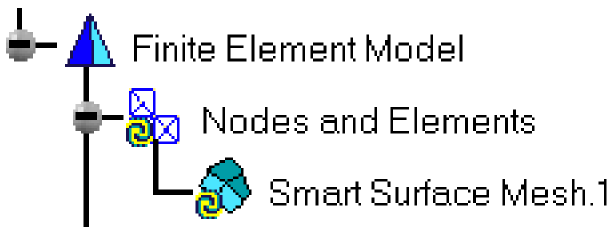
3. Define the desired mesh parameters in the Global Parameters dialog box.

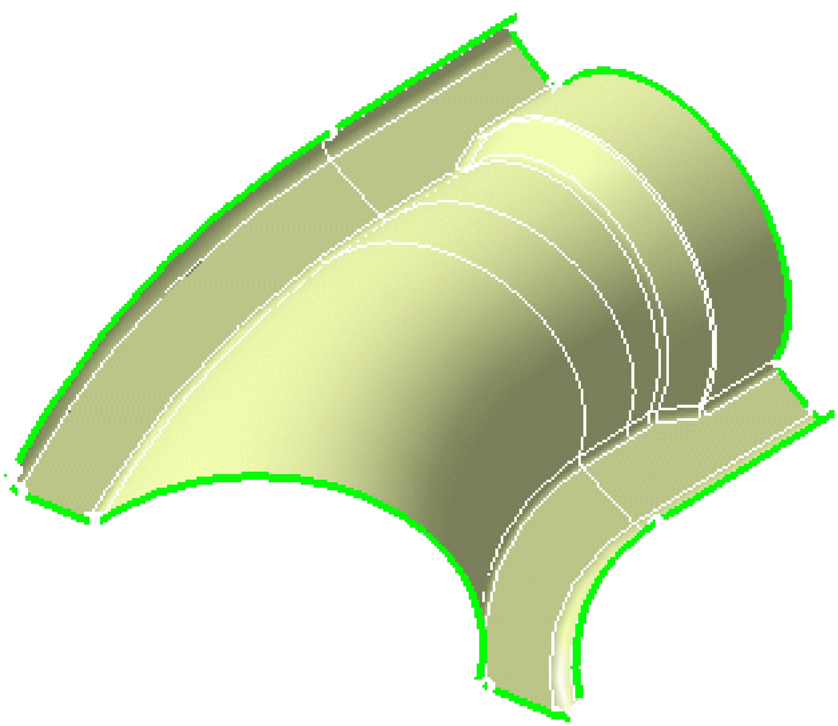
In this particular example, you will:

- select the **Set frontal quadrangle method** icon as **Element shape**
- in the **Mesh** tab of the Global Parameters dialog box:
  - enter **5 mm** as **Mesh size** value
  - enter **0 mm** as **Offset** value
- in the **Geometry** tab of the Global Parameters dialog box:
  - enter **1 mm** as **Constraint sag** value
  - enter **10 mm** as **Min holes size** value
  - select the **Merge during simplification** option
  - enter **2 mm** as **Min size**

4. Click **OK** in the Global Parameters dialog box.

A new **Smart Surface Mesh** feature is created in the specification tree.





- You now enter the **Surface Meshing** workbench and the following toolbars are available:
  - Local Specifications
  - Execution
  - Edition Tools

- At any time, you can visualize or modify the global parameters.

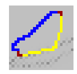
  For this, click the **Global Meshing Parameters** icon . The Global Parameters dialog box appears.

Setting Constraints and Nodes

This task will show you how to define constraints on edge nodes distributions.

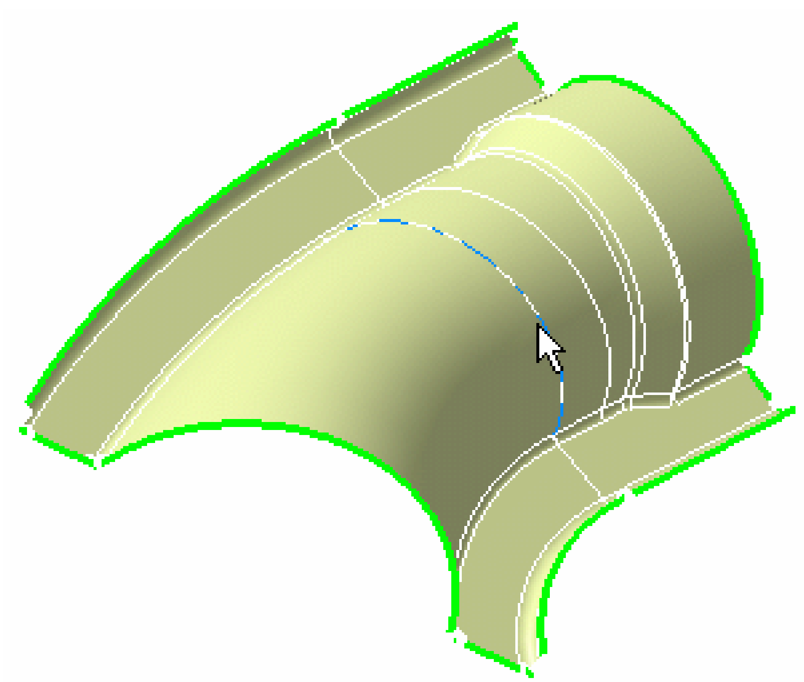
For this, you can select edges, vertices (on the geometrical simplification), curves or points (using the geometry).

You are still in the **Surface Meshing** workshop.

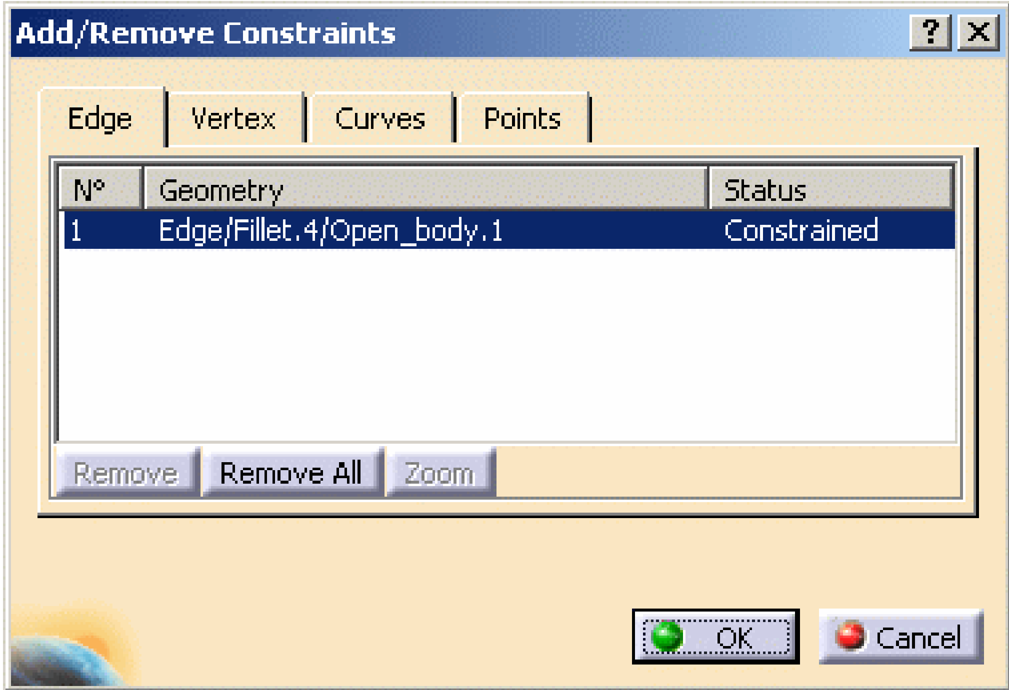
1. Click the **Add/Remove Constraints** icon  in the **Local Specifications** toolbar.

   Both the Add/Remove Constraints dialog box and the Trap Type dialog box appear.

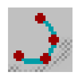
2. Select the edge to be constrained.



This edge is now yellow colored and the Add/Remove Constraints dialog box is updated.



3. Click **OK** in the Add/Remove Constraints dialog box.

4. Click the **Imposed Elements** icon  in the **Local Specifications** toolbar.

   The Imposed Elements dialog box appears.

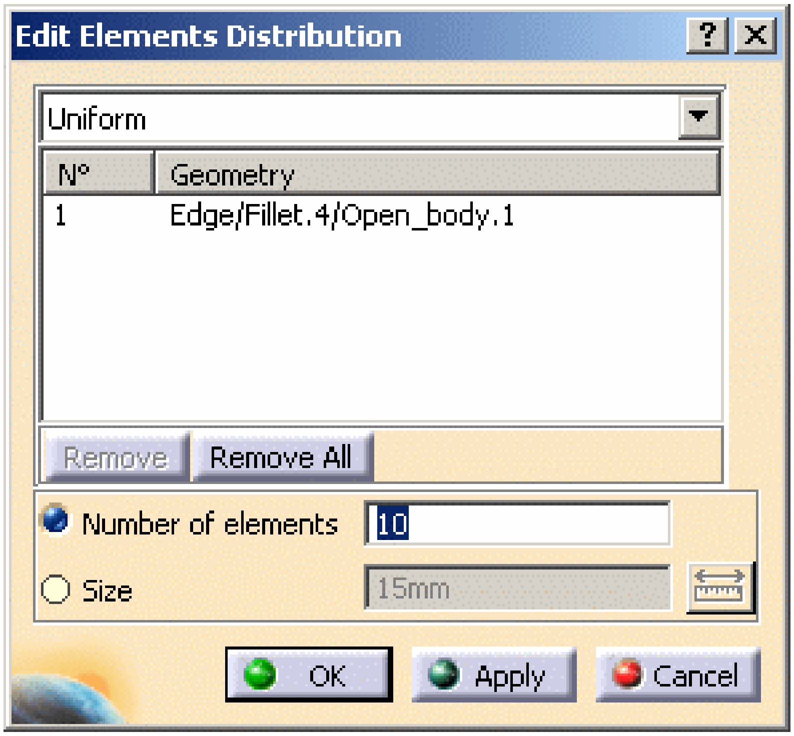
5. Select the edge you just have constrained.

   The Edit Elements Distribution dialog box appears.

6. Define the parameters of the new distribution.

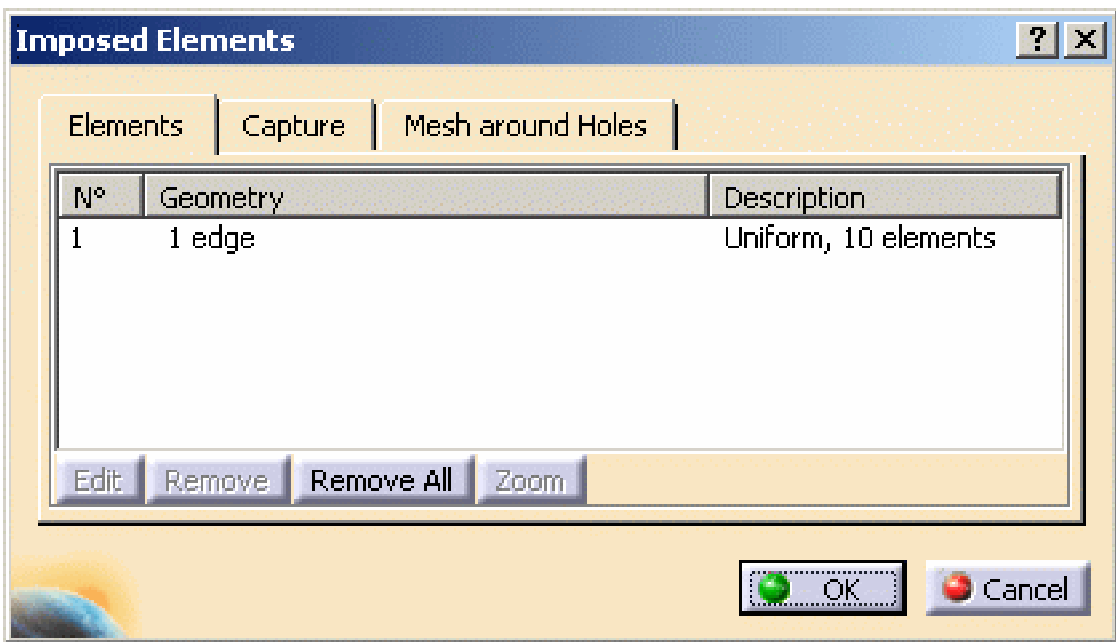
   In this particular case, you will:

   - select the **Uniform** distribution type
   - enter **10** as **Number of elements**

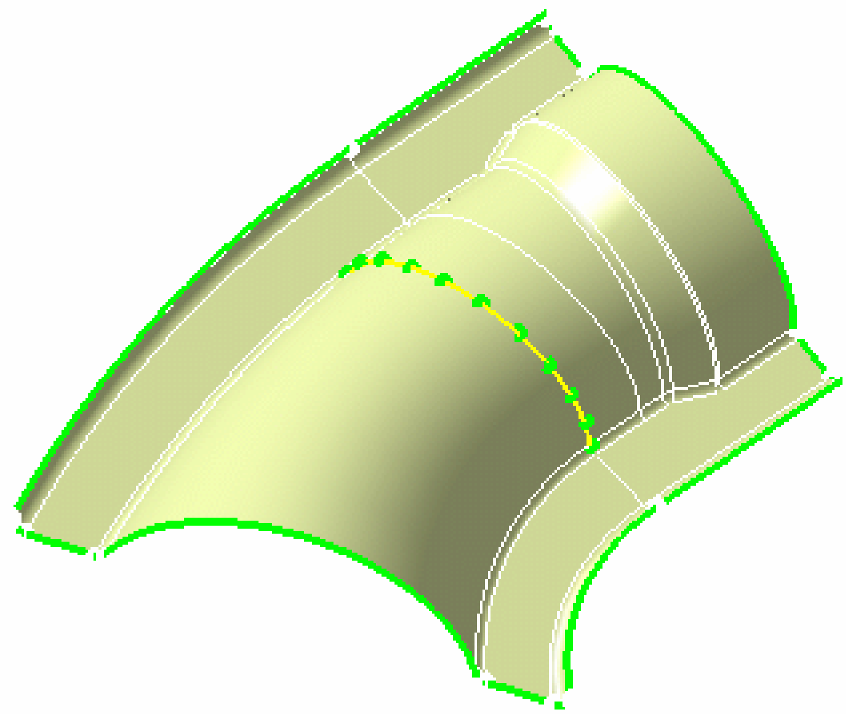


7. Click **OK** in the Edit Elements Distribution dialog box.

The nodes (or elements) are distributed on the selected edge. The node distribution description now appears in the Imposed Elements dialog box.



8. Click **OK** in the Imposed Elements dialog box.



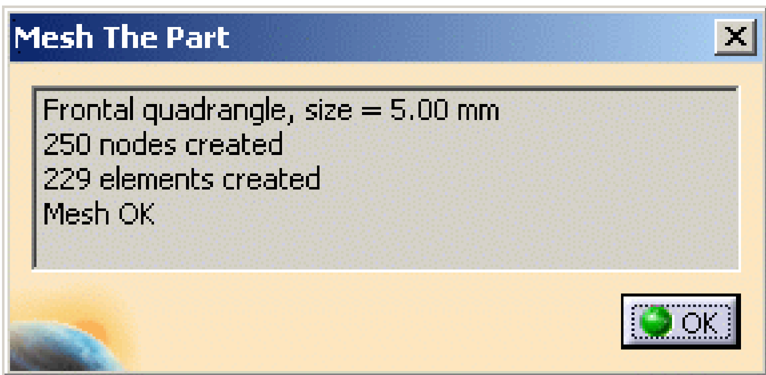
At this stage, distributions are stored in the mesh definition. You can now launch the mesh operation.

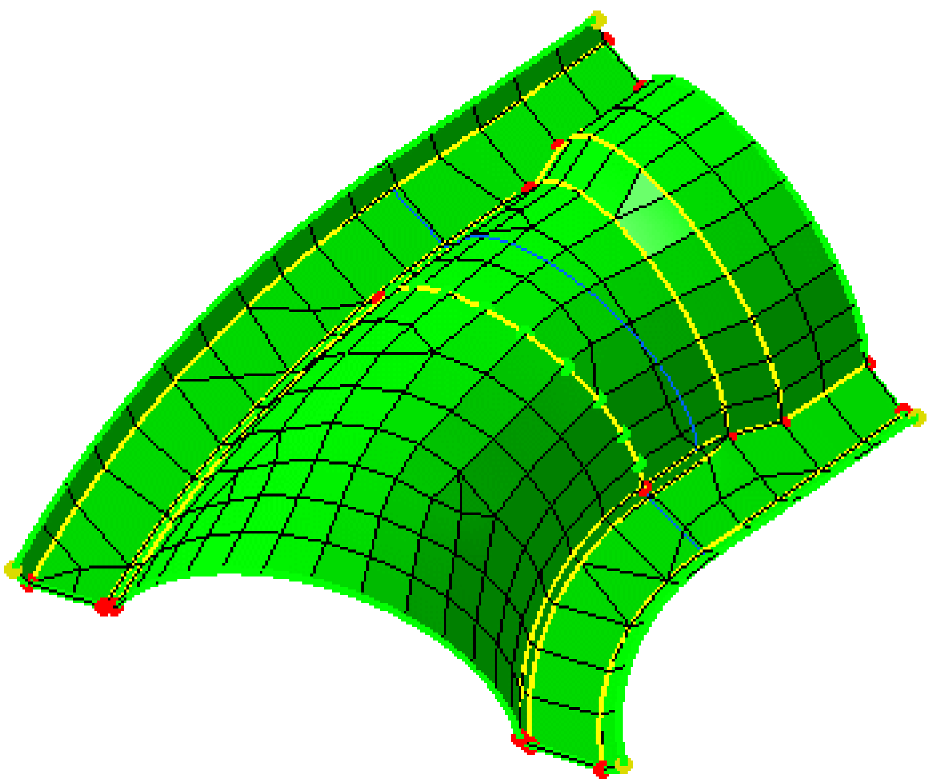
Launching the Mesh Operation

This task will show you how to generate the mesh, now that you defined the global parameters and the specifications.

1. Click the **Mesh The Part** icon in the **Execution** toolbar.

   The mesh is generated on the part and a little summary is provided in the Mesh The Part dialog box.





The visualization is switched to quality mode so that you can visualize the generated mesh and the quality of each element.

2. Click **OK** in the Mesh The Part dialog box.

Analyzing Element Quality

This task will show you how to use some basic quality analysis functionalities.

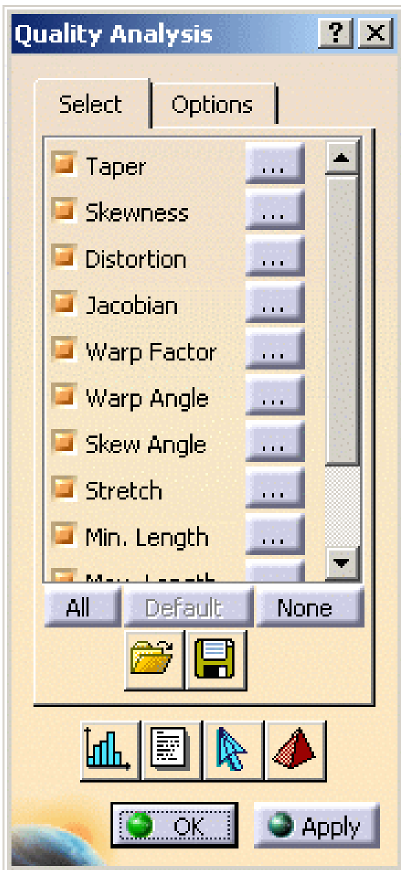
Quality Analysis functionalities are available at all steps of the meshing process.

You can analyze quality specifications of:

- all the elements
- one element

1. Click the **Quality Analysis** icon in the **Mesh Analysis Tools** toolbar.

   The Quality Analysis dialog box is displayed.

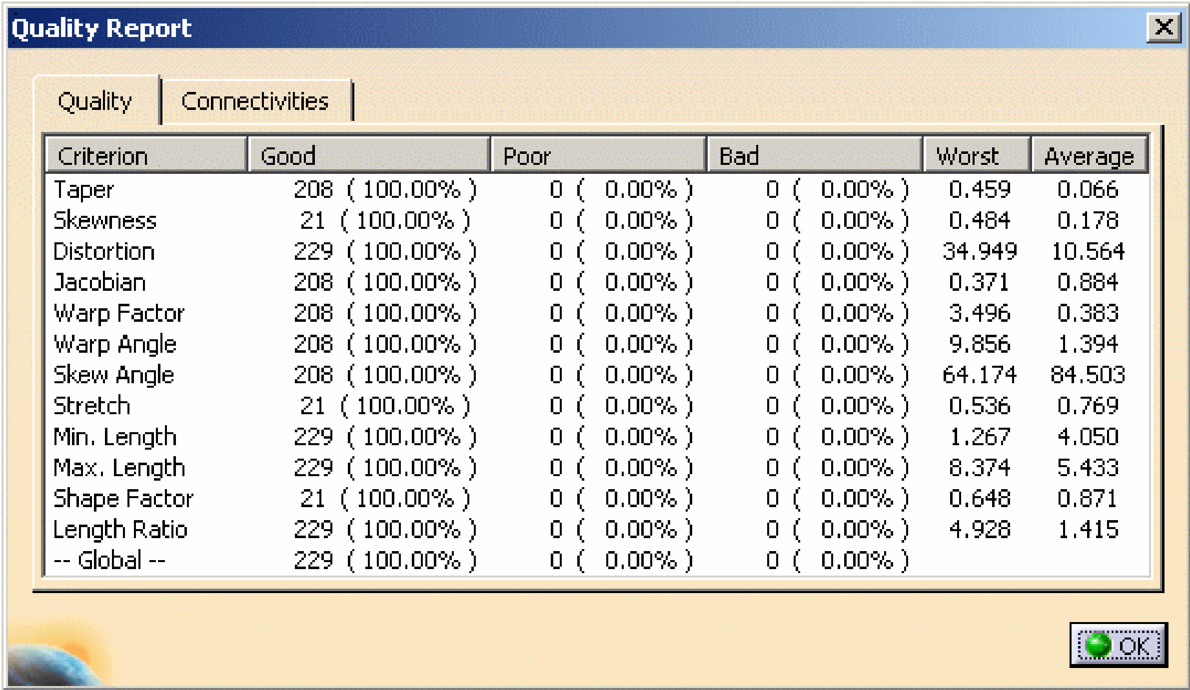


The Quality Analysis dialog box is a filter that lists the available quality specifications for visualizing and analyzing the mesh.
By selecting particular specifications, you can decide how you want to view the mesh. It also provides a set of functionalities for deeper analysis.

Quality Specifications of All the Elements

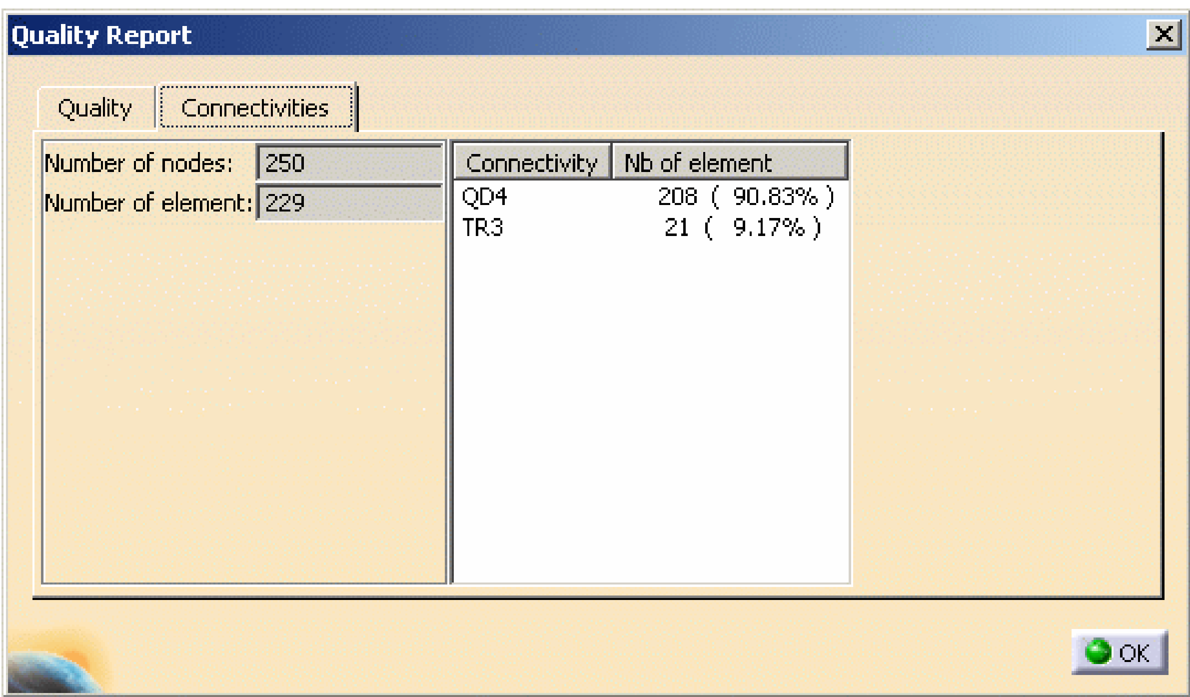
2. Click the **Show Quality Report** icon in the Quality Analysis dialog box.

The Quality Report dialog box presents the statistics corresponding to the selected quality specifications: **Good**, **Poor** , **Bad**, **Worst** and **Average**.



Criterion	Good	Poor	Bad	Worst	Average
Taper	208 (100.00%)	0 (0.00%)	0 (0.00%)	0.459	0.066
Skewness	21 (100.00%)	0 (0.00%)	0 (0.00%)	0.484	0.178
Distortion	229 (100.00%)	0 (0.00%)	0 (0.00%)	34.949	10.564
Jacobian	208 (100.00%)	0 (0.00%)	0 (0.00%)	0.371	0.884
Warp Factor	208 (100.00%)	0 (0.00%)	0 (0.00%)	3.496	0.383
Warp Angle	208 (100.00%)	0 (0.00%)	0 (0.00%)	9.856	1.394
Skew Angle	208 (100.00%)	0 (0.00%)	0 (0.00%)	64.174	84.503
Stretch	21 (100.00%)	0 (0.00%)	0 (0.00%)	0.536	0.769
Min. Length	229 (100.00%)	0 (0.00%)	0 (0.00%)	1.267	4.050
Max. Length	229 (100.00%)	0 (0.00%)	0 (0.00%)	8.374	5.433
Shape Factor	21 (100.00%)	0 (0.00%)	0 (0.00%)	0.648	0.871
Length Ratio	229 (100.00%)	0 (0.00%)	0 (0.00%)	4.928	1.415
-- Global --	229 (100.00%)	0 (0.00%)	0 (0.00%)		

3. Select the **Connectivities** tab to view mesh composition statistics (**Number of nodes**, **Number of element**, **Connectivity**, **Number of element per connectivity**).



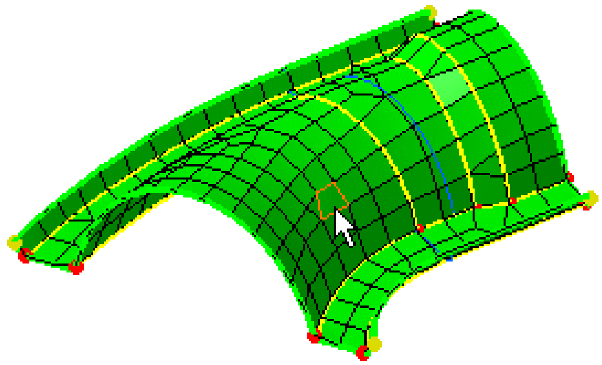
Connectivity	Nb of element
QD4	208 (90.83%)
TR3	21 (9.17%)

4. Click **OK** in the Quality Report dialog box.

Quality Specification of One Element

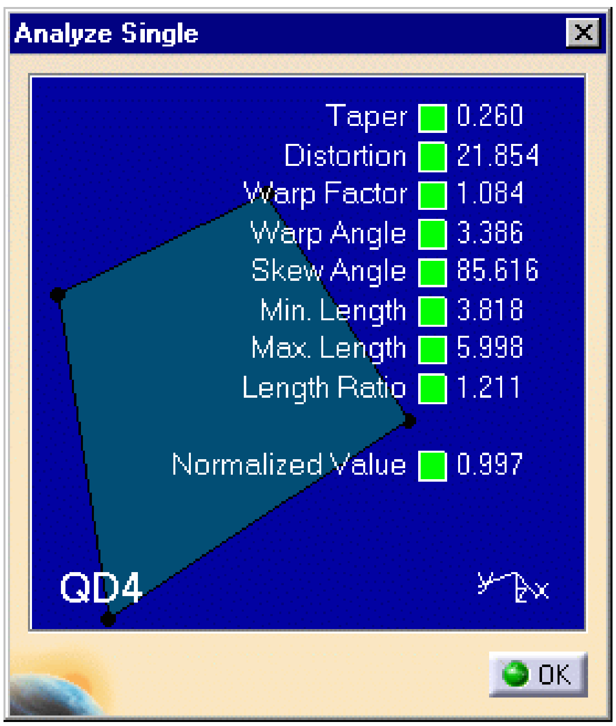
5. Click the **Analyze An Element** icon  in the Quality Analysis dialog box.

6. Select an element of the meshed part.



The Analyze Single dialog box now appears and gives you a detailed view of the quality of this element: **Taper**, **Distorsion**, **Warp Factor**, **Warp Angle**, **Skew Angle**, **Min. Length**, **Max. Length**, **Length Ratio** and **Normalized Value**.

In other words, you will check whether any of your specifications was not properly implemented.



7. Click **OK** in the Analyze Single dialog box.

8. Click **OK** in the Quality Analysis dialog box.

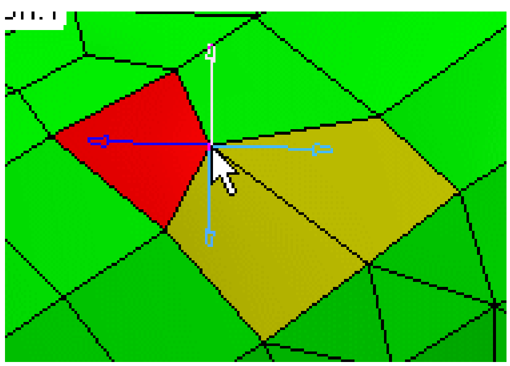
Mesh Editing

This task shows how to edit one mesh element by moving one node using auto smooth options and then cutting one element into two.

1. Click the **Edit Mesh** icon in the **Edition Tools** toolbar.

   The Edit Mesh dialog box appears.

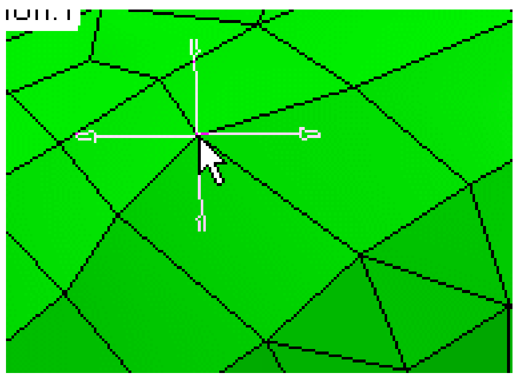
2. Select a node and move it to the desired location.



   The quality/visualization of the elements is updated according to the location you assign to the node.

3. Check the **Smooth around modifications** option from the Mesh Editing Options dialog box.

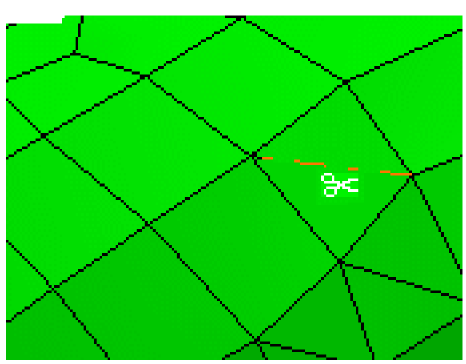
4. Select a node and move it to the desired location.

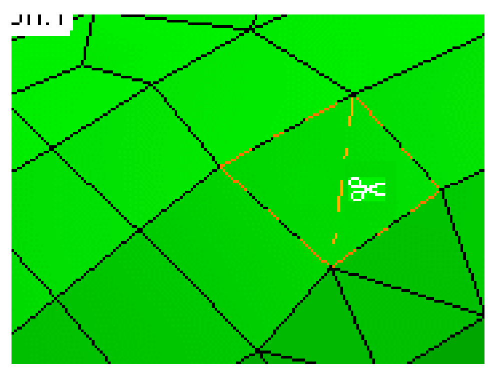


The quality of the elements is not modified whatever the location you assign to the node.

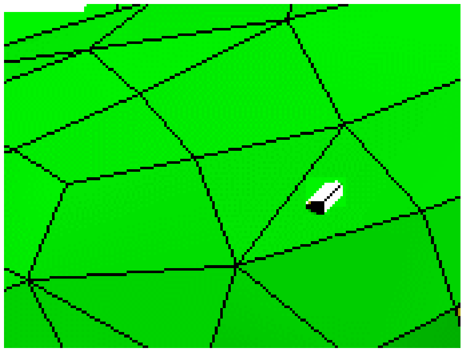
5. Position the cursor over one quadrangle element.

You can now cut the mesh element diagonally according to the position of the cursor.





6. Once you have cut one mesh element, position the cursor over the segment you just created and if you want to delete this segment, click on it using the rubber that appears.



7. Click **OK** in the Edit Mesh dialog box.

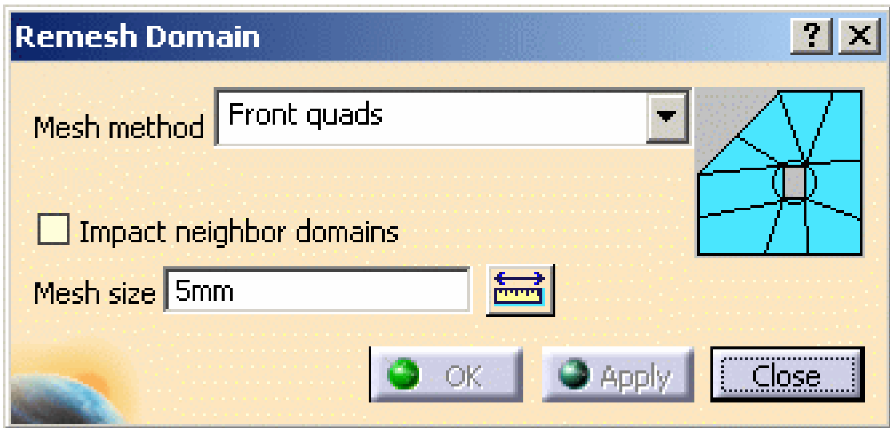
Note that as the **Smooth around modifications** option is still active in the Edit Mesh dialog box, the quality of the elements is not modified whatever the modifications you perform.

Re-meshing a Domain

This task will show you how to re-mesh a domain by modifying pre-defined local specifications such as mesh method and size.

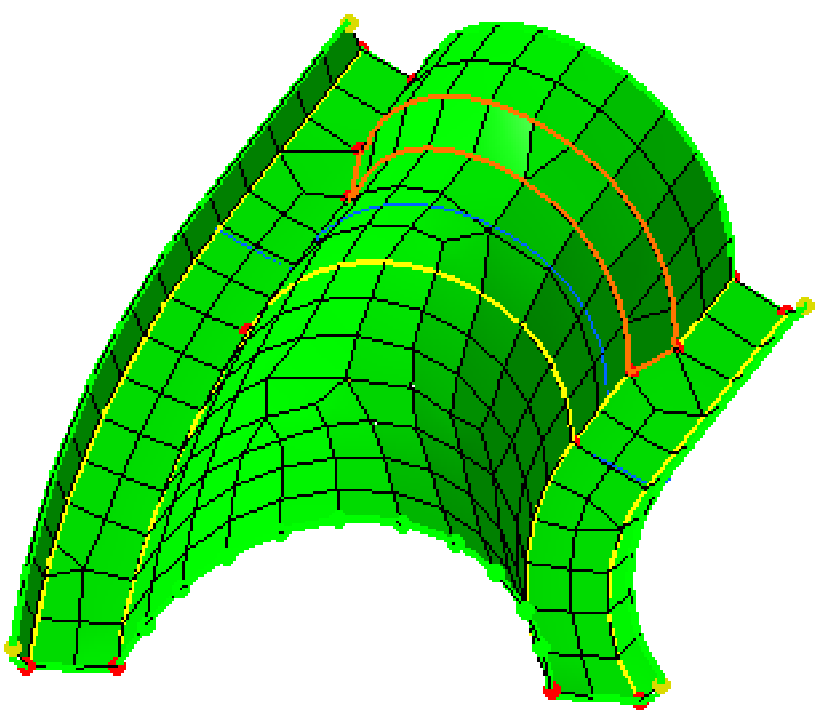
1. Click the **Remesh Domain** icon in the **Edition Tools** toolbar.

   The Remesh Domain dialog box appears.



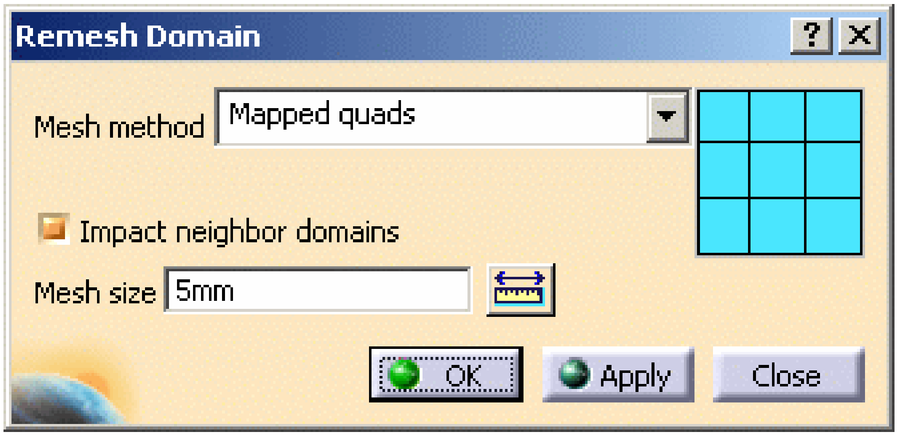
2. Select the domain to be re-meshed.

   As shown in the picture, the selected domain appears as limited in the red color.



We will try to remove the triangles by locally altering the mesh method.

3. Set the parameters for the selected domain from the Remesh Domain dialog box.



To do this, you can specify the:

- **Mesh method**
- **Impact neighbor domains**
- **Mesh size**

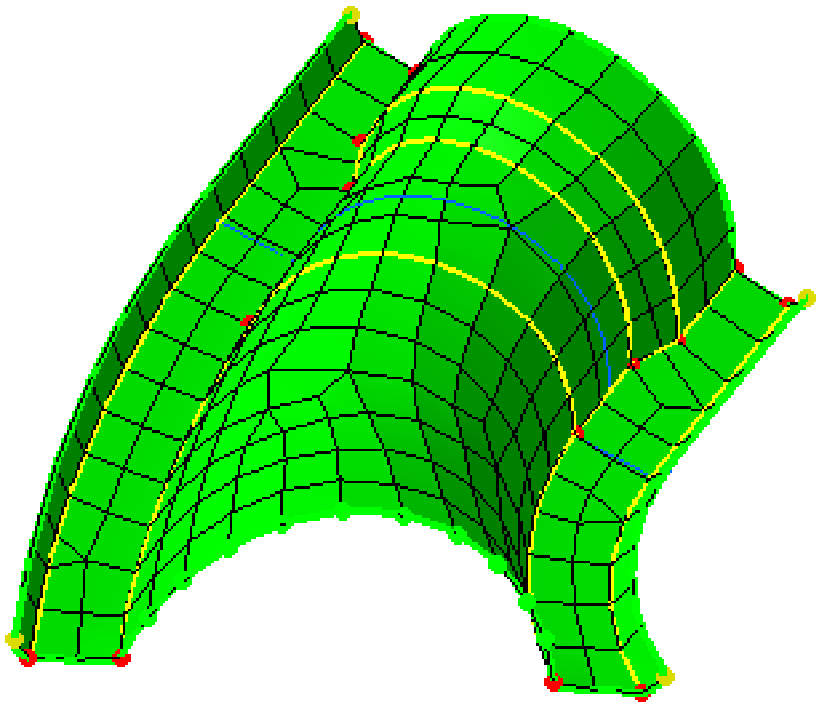
In this particular example:

- select the **Mapped quads** as **Mesh method**
- select the **Impact neighbor domains** option
- enter **5 mm** as **Mesh size**

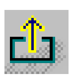
4. Click **OK** in the Remesh Domain dialog box.

The mesh is updated.

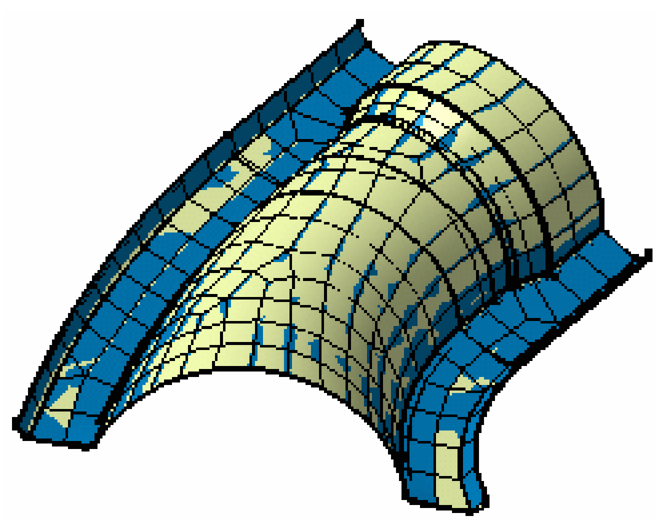
The selected domain is now meshed with quadrangles only.



Now that you have completed the tutorial, you can exit the **Surface Meshing** workshop.
For this:

5. Click the **Exit** icon  in the **Exit** toolbar.

The final meshed part can now be visualized and appears as shown here:



User Tasks

This section describes the User's Tasks that allow you to complete the mesh of a part using FEM Surface.

These tasks include:

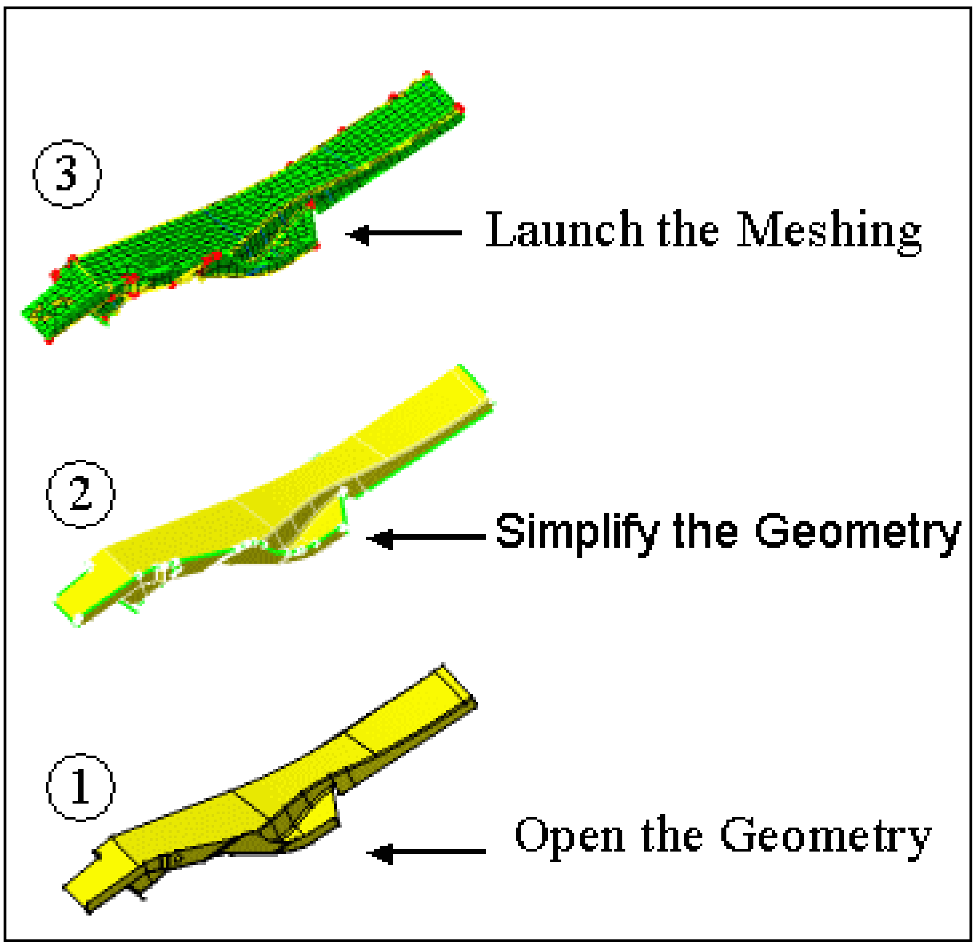
Before You Begin

You should be familiar with the following basic methodological approach and concepts:

- Smart Surface Triangle Quadrangle Mesher
- Methodology ...
- Colors Used for Elements
- Colors Used for Edges and Vertices
- Miscellaneous

Smart Surface Triangle Quadrangle Mesher

The Smart Surface Triangle Quadrangle Mesher works as shown here: you will open the geometry 1, launch the geometrical simplification 2 and then launch the meshing 3.



1 : Open the geometrical element on which you are going to generate geometrical simplification from parameters.

2 : The geometry is simplified in order to launch the meshing and manage constraints more easily. The level of the simplification depends on the mesh parameters previously defined.

3 : The surface mesh is created from the geometrical simplification previously generated.

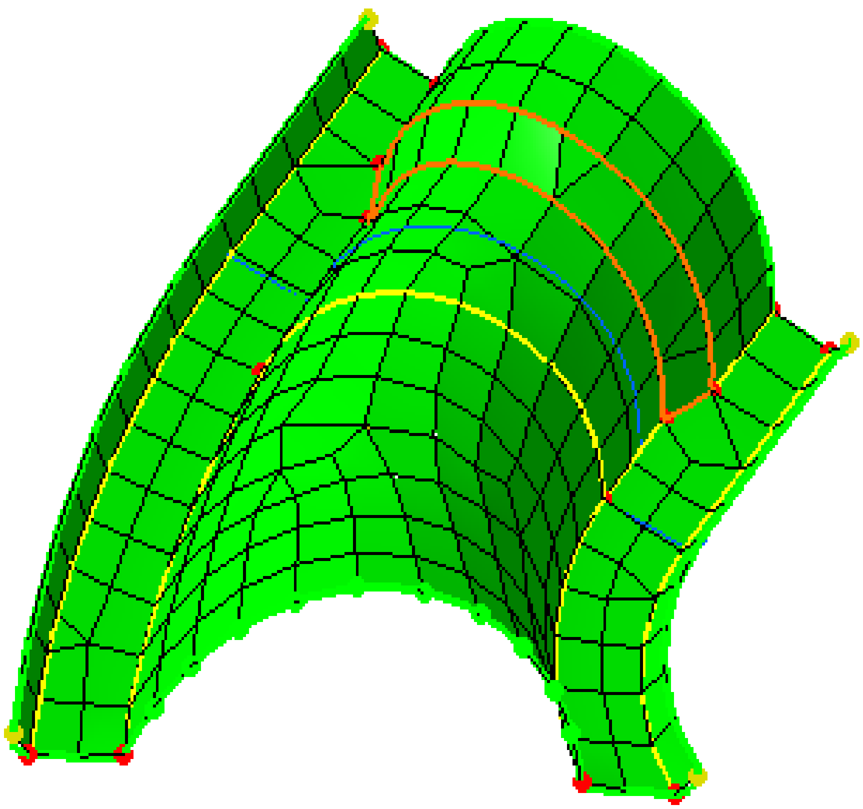
Methodology

Please, follow the below described methodological approach when using the **FEM Surface** product (**Advanced Meshing Tools** workbench).
Consider that the **FEM Surface** product was developed so that Meshing operation may be as user friendly and as automatic as possible.

- If you want your constraints to be associative with the resulting mesh, before you launch the Mesh operation apply as many **constraints** as possible and as **automatically** as possible.
- Still, try to regularly **check** how constraints result on the mesh.

Generally speaking, you will start defining parameters, cleaning the geometry according to the desired resulting meshing and specifying constraints as soon as possible. You will then launch the Geometrical Simplification and in one go the Mesh operation.

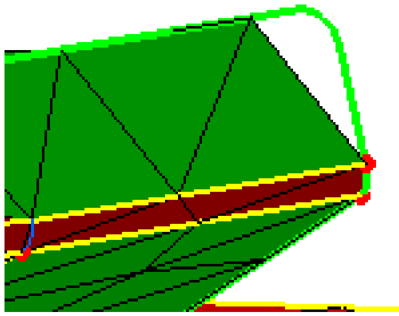
1. **define parameters**
2. **clean the geometry**
3. **specify constraints**
4. **launch the Geometrical Simplification**
5. **launch the Mesh operation**
6. **perform constraints modification according to the resulting mesh elements (not according to the topology)**
7. **if needed, edit the mesh elements**

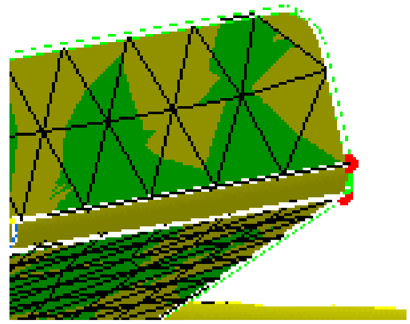


1. Define Parameters

From the very beginning, you need to specify global parameters: the shape of the elements, the size of the elements, the sag and the minimum size of these elements.

Before: *After:*



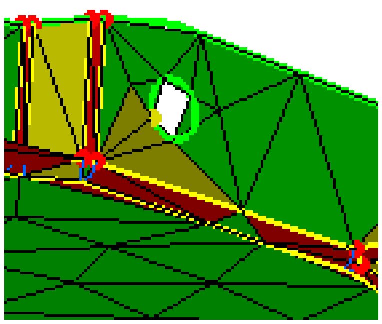


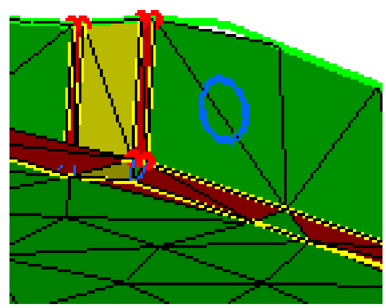
Use automatic algorithm and only define what the algorithm will not do properly for given cases.

2. Clean the Geometry

From the very beginning, you will also specify whether or not, you need given holes, button hole gaps (or cracks) and small faces to be taken into account by the Geometrical Simplification and therefore by the Mesher.

Before: *After:*



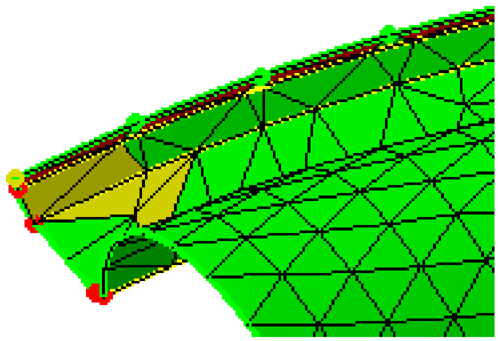


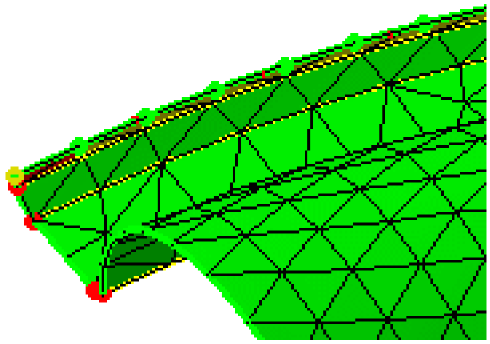
This must be performed BEFORE you launch the mesh operation. Once the part is meshed, the clean characteristics can no more be modified.

3. Specify Constraints

You will also specify the constraints that are absolutely necessary for performing the Analysis computation. For example, you will impose the desired constraints in order to generate connections between meshes and to create boundary conditions such as restraints and loads.

Before: *After:*

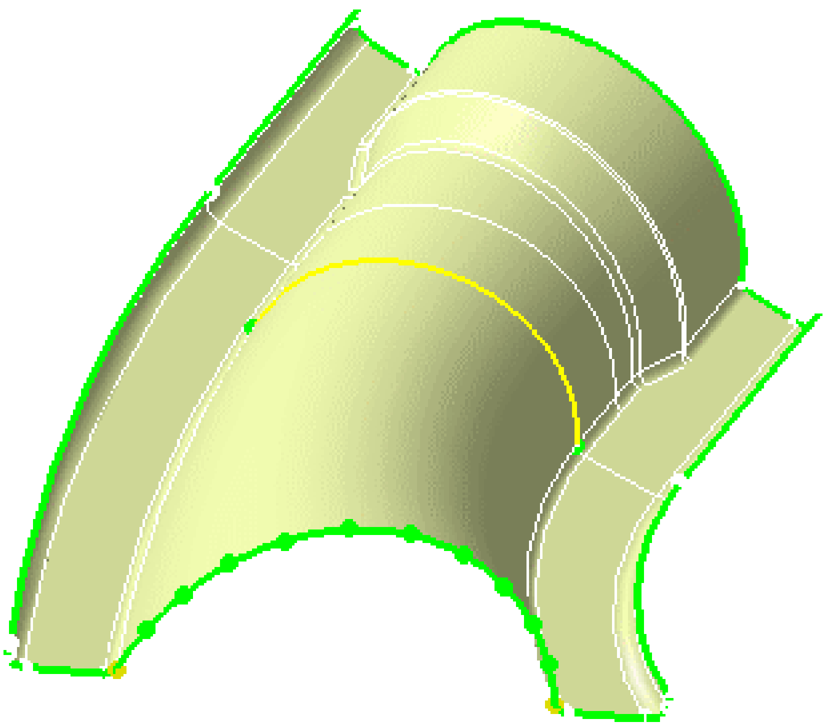




Specify as many constraints BEFORE you launch the mesh operation: these constraints will be associative. Specify these constraints as automatically as possible and avoid modifying them manually (for example dragging a node).

4. Simplify the Geometry

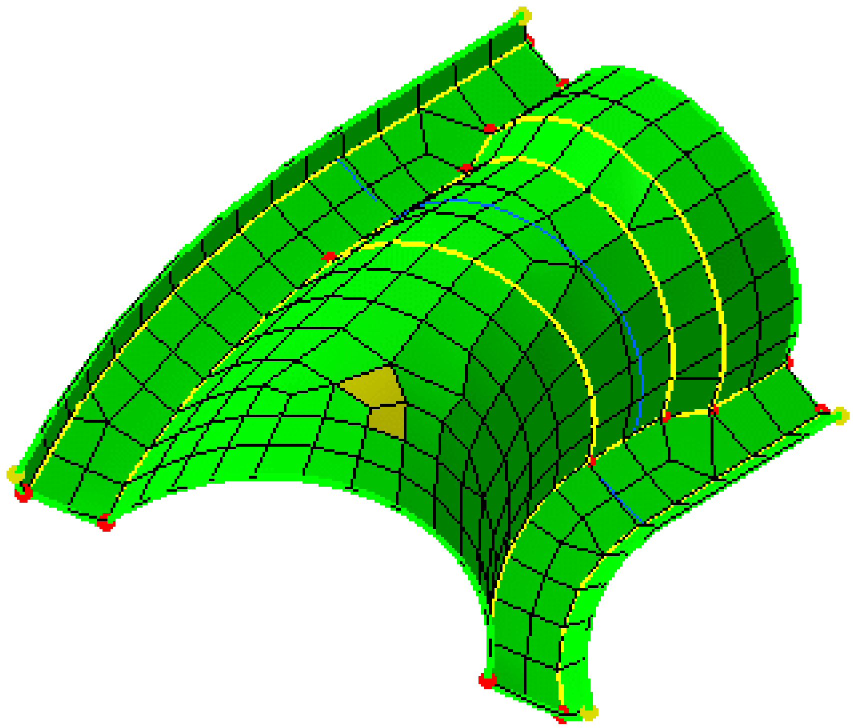
The Geometrical Simplification computation is based on the global parameters and the constraints imposed by the user. The system will create an additional set of new constraints that will automatically help the mesher in creating elements of a higher quality.



Avoid applying too many modifications before simplifying the geometry: launch the Geometrical Simplification and before modifying manually, check how the resulting mesh looks like.

5. Launch the Mesh Operation

As soon as the mesh elements are generated, a feedback on the quality is provided. You can then perform manual modifications on the mesh elements, if needed.

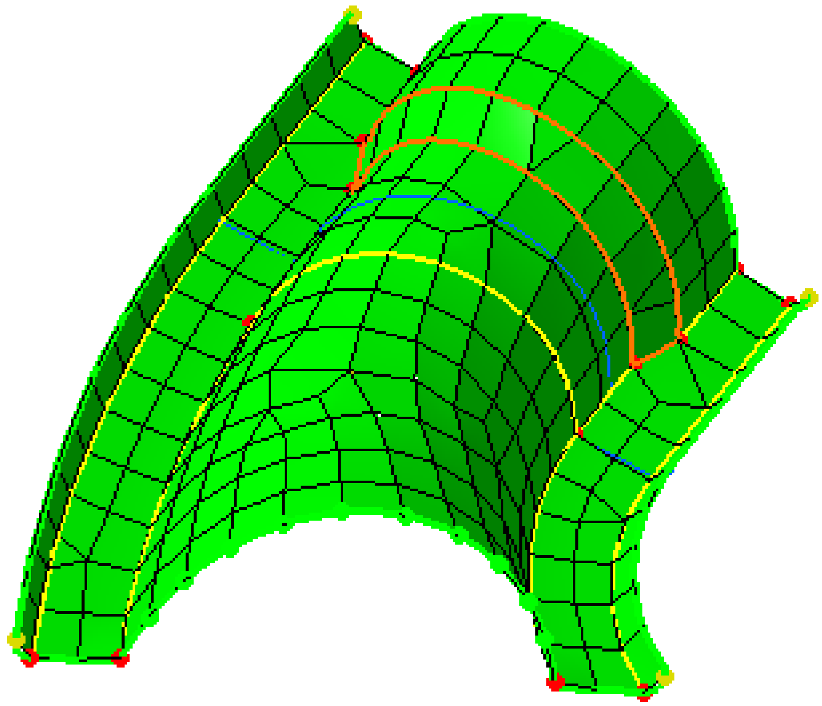


BE CAREFUL: this operation can be reversed. Even though you launch the mesh operation, you can apply modifications to the specifications. First have a look at the domains that seem to be problematic. Add more or delete existing constraints instead of modifying the mesh elements manually.

6. Modify the Meshing (or Re-Meshing)

Although the algorithms were developed in order to minimize user interactions, after the Mesh operation was performed, you can still modify the generated mesh elements. In other words, you can:

- modify the geometrical simplification generated by the system
- modify the nodes distribution
- apply local re-meshing (for example, the size or the type of the mesh elements)
- edit mesh elements and apply manual modifications.



Make sure you cannot remove the mesh and modify the constraints specifications instead. These modifications will not be associative.

For Advanced Users

If you are an advanced user and know very well how the Mesher behaves, you can launch the geometrical simplification, perform the above mentioned re-meshing modifications and then launch the mesh operation in order to fill the gaps.

Make sure you cannot remove the mesh and modify the constraints specifications instead. These modifications will not be associative.

Colors Used for Elements

Open the sample05.CATAnalysis document from the samples directory.

Before You Begin

- **Enter the smart surfacic triangle quadrangle mesher**.
  For this, double-click **Smart Surfacic Mesh.1** feature from the specification tree (below **Nodes and Elements** feature) and then click **OK** (**Continue anyway?**) in the warning box.
- **Mesh the surface.**

  For this, click the Mesh The Part icon .

Quality Visualization Mode

Color		Meaning
Green		Used when the element will be solved by the solver without any problem.
Yellow		Used when the element will be solved by the solver with very few possible problems.
Red		Used when the element will be hardly properly solved.

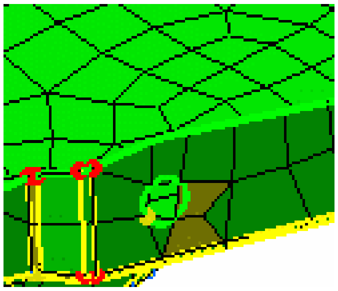
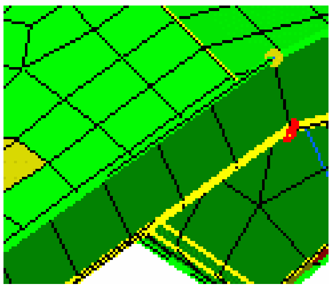
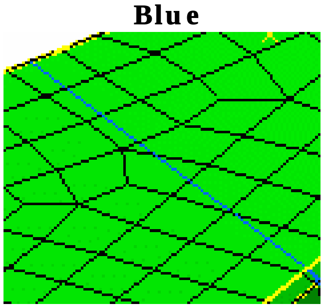
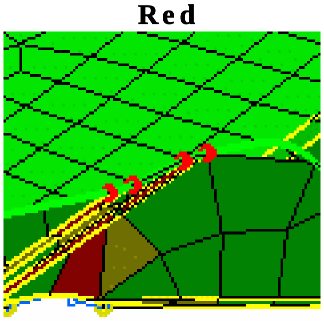
Colors Used for Edges and Vertices

Open the sample05.CATAnalysis document from the samples directory.

Before You Begin

- **Enter the smart surfacic triangle quadrangle mesher**.
  For this, double-click **Smart Surfacic Mesh.1** feature from the specification tree (below **Nodes and Elements** feature) and then click **OK** (**Continue anyway?**) in the warning box.
- **Mesh the surface.**

  For this, click the **Mesh The Part** icon .

Color	Meaning
Green 	Used for free edges and vertices, as well as holes.
Yellow 	Used for edges/vertices that are shared between two constrained faces.
Blue 	Used for edges/vertices that are shared between two non-constrained faces.
Red 	Used for edges/vertices that are shared between more than two constrained faces.

Miscellaneous

Resuming Editing on Mesh Part

Open the sample05.CATAnalysis document from the samples directory.

Before You Begin

- **Enter the smart surfacic triangle quadrangle mesher**.
  For this, double-click **Smart Surfacic Mesh.1** feature from the specification tree (below **Nodes and Elements** feature) and then click **OK** (**Continue anyway?**) in the warning box.
- **Mesh the surface.**

  For this, click the **Mesh The Part** icon .

You can now exit the **smart surfacic triangle quadrangle mesher**, and then decide to resume editing. For this, by double-click on the **Smart Surfacic Mesh.1** feature from the specification tree. A message will let you decide whether you want to edit the meshed CATAnalysis with:

- the initial meshing

OR

- the meshing last created.

Be careful: if the geometry was not properly updated, the CATAnalysis will be edited anyway with the initial meshing.

Beam Meshing

This task shows you how to add beam mesh to a Generative Shape Design CATPart, and if needed, edit the mesh.

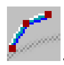
You can add a beam mesh on a Structure Design beam. So you will mesh it with 1D elements.

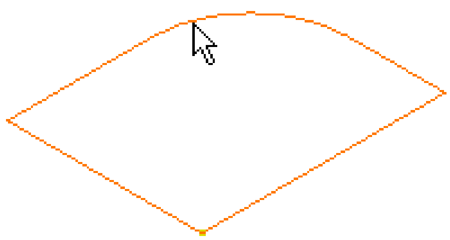
You cannot add beam mesh to a sketch geometry.

Open the sample47.CATAnalysis document from the sample directory.

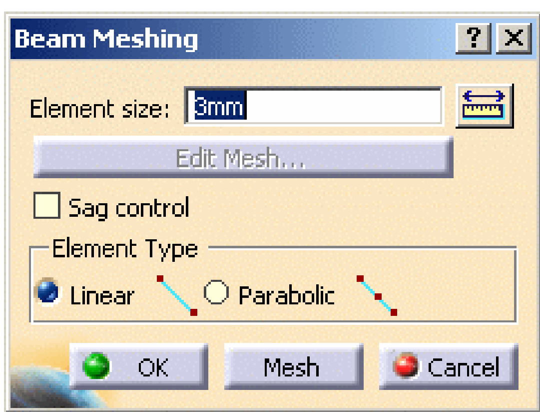
Before You Begin:

Make sure a material was applied to the geometry, first.

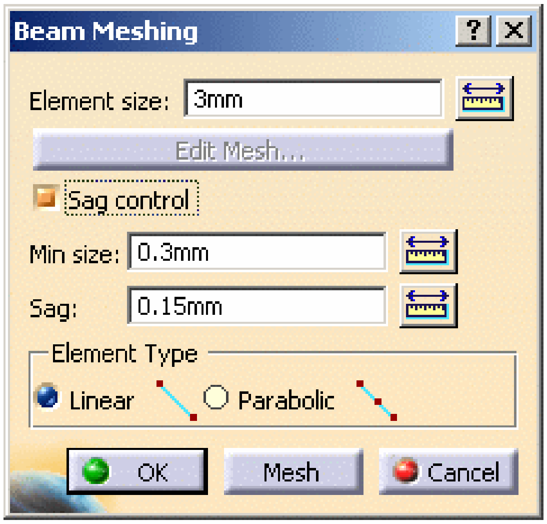
1. Click the **Beam Mesher** icon .
2. Select the feature corresponding to the geometry to be associated a mesh part.



The Beam Meshing dialog box appears.



- **Element size**: lets you set the global mesh size
- **Edit Mesh...**: lets you edit mesh manually to enhance quality
- **Sag control**: lets you set the element minimum size and sag. If checked, the corresponding values (**Min size** and **Sag**) appear in the dialog box
  Remember that the sag is the distance between the mesh elements and the geometry so that mesh refining is optimum in curve-type geometry:

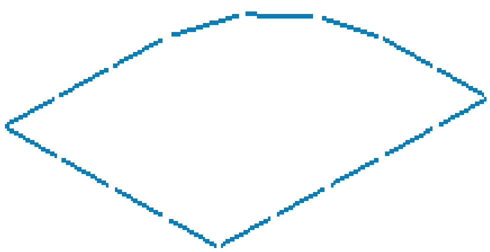


- **Element Type**: lets you choose the 1D element type
  - **Linear**: 1D element without intermediate node
  - **Parabolic**: 1D element with an intermediate node

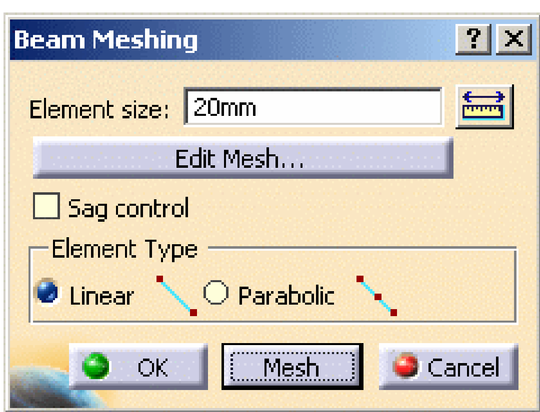
3. Enter the desired **Element size** in the Beam Meshing dialog box. In this example, enter a size of **20mm**.

4. Click the **Mesh** button in the Beam Meshing dialog box to launch computation.

   The mesh is displayed.

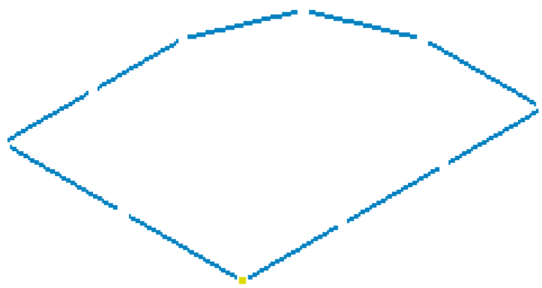


The Beam Meshing dialog box now appears as shown here: the **Edit Mesh...** button can be selected, if needed, to edit the mesh.

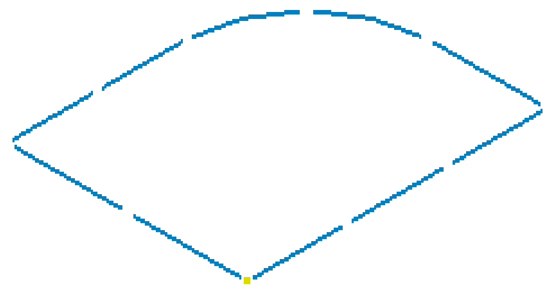


5. Select **Linear** or **Parabolic** as **Element Type** option and click the **Mesh** button in the Beam Meshing dialog box.

Linear option activated

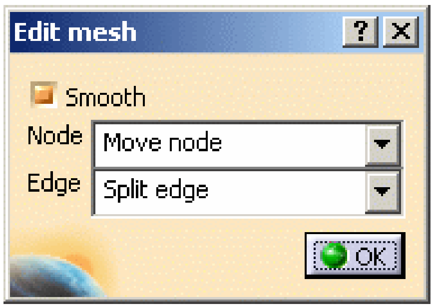


Parabolic option activated



6. Click the **Edit mesh** button in the Beam Meshing dialog box.

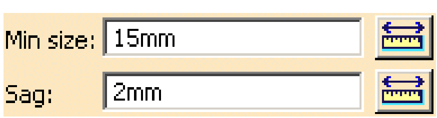
The Edit Mesh dialog box appears.



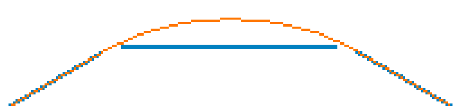
- **Smooth**: the neighbor elements are smoothed around the modified mesh.
- **Node**:
  - **Move node**: move a node belong the geometry.
  - **Freeze node**: fix a node whatever the modifications applied to the neighbor elements.
  - **Unfreeze node**: cancel the **Freeze node** option.
- **Edge**:
  - **Split edge**: create a node and split an edge into two edges.
  - **Condense edge**: suppress an edge by condensation of one node and the node opposite to the point on the selected edge.

7. Activate the **Sag control** option in the Beam Meshing dialog box and modify the **Min size** and **Sag** values so that the mesh is refined as desired.

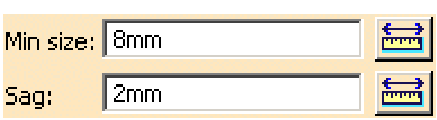
Selected Size and Sag values



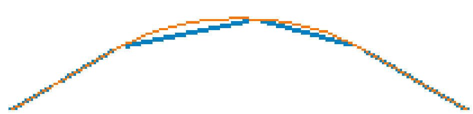
Resulting mesh



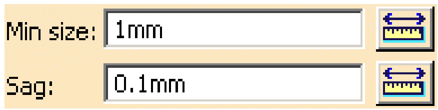
Selected Size and Sag values



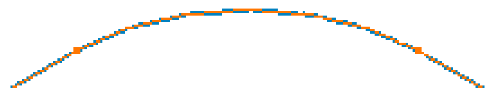
Resulting mesh



Selected Size and Sag values



Resulting mesh

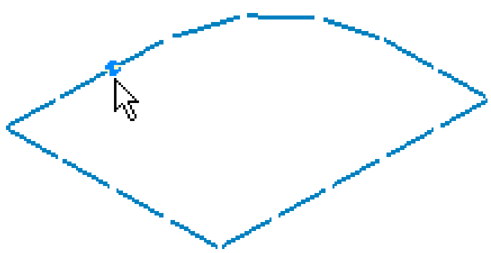


Moving a node:

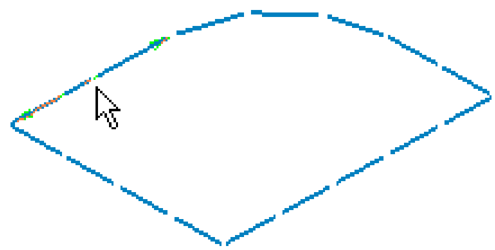
8. Select the **Move node** option in the **Node** combo box.

9. Select a node and move it to the desired location.

Select the node

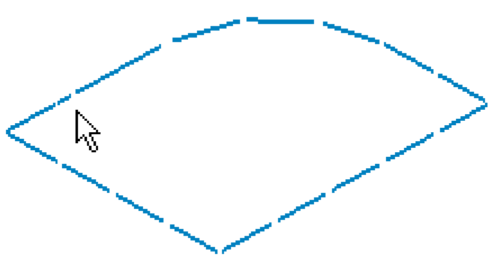


Move the node

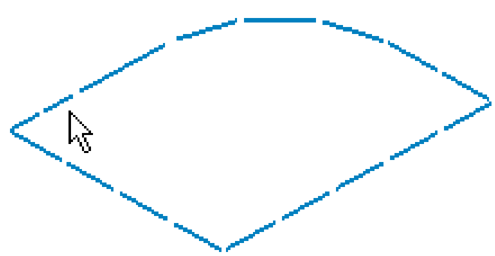


The result is:

Smooth option deactivated



Smooth option activated

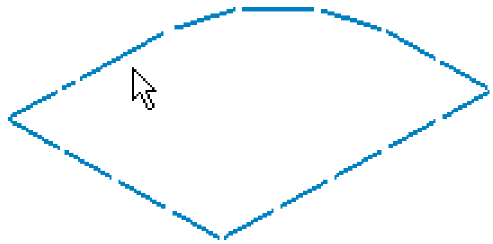


Splitting an edge:

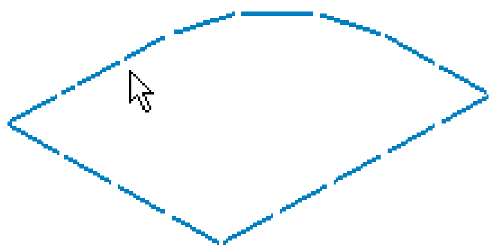
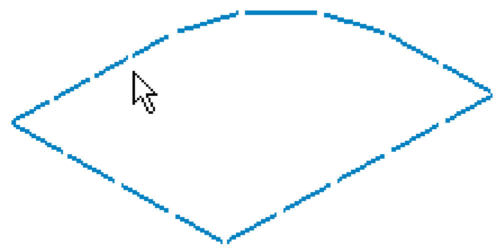
10. Select the **Split edge** option in the **Edge** combo box.

    To refine the mesh, you need to check the **Smooth** option in the Edit Mesh dialog box. The neighbor edges are updated.

11. Select the edge you want to split.



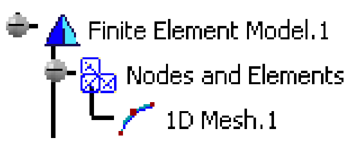
The result is:

Smooth option deactivated	**Smooth** option activated
	

12. Click **OK** in the Edit mesh dialog box.

13. Click **OK** in the Beam Meshing dialog box.

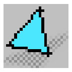
    The **1D Mesh.1** feature now appears in the specification tree.



1D, or beam mesh, can be deleted and/or added to parts manually.

Surface Meshing

This section deals with the 2D meshing methods.

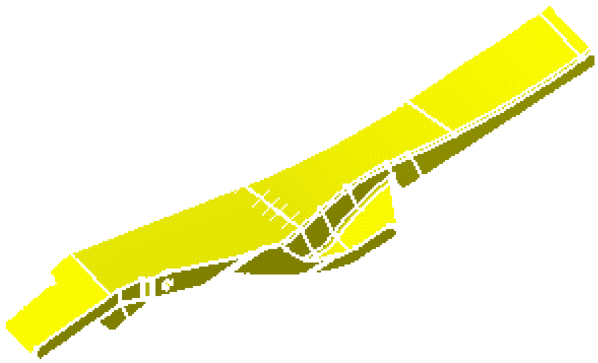
 Meshing using OCTREE Triangle: Assign linear or parabolic triangle elements to a surface.

 Advanced Surface Mesher: Mesh a surface part and let you enter the **Surface Meshing** workshop.

Meshing Using OCTREE Triangles

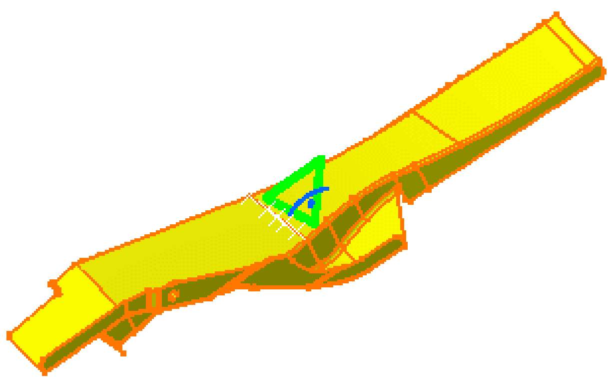
This task will show you how to mesh a part using OCTREE triangles.

Open the sample08.CATAnalysis document from the samples directory.



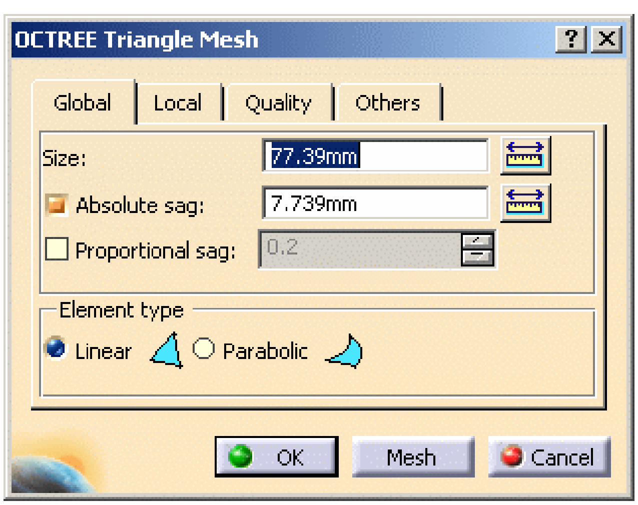
1. Click the **Octree Triangle Mesher** icon from the Meshing Methods toolbar.
2. Select the geometry to be meshed.

   For this, click either the geometry feature in the specification tree or click the part.

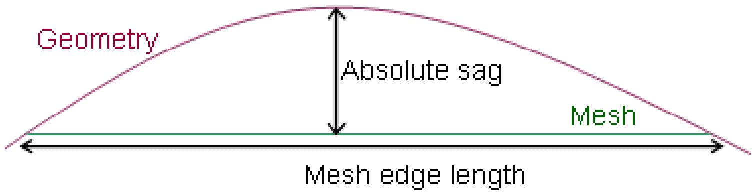


The OCTREE Triangle Mesh dialog box appears.

- **Global** tab:



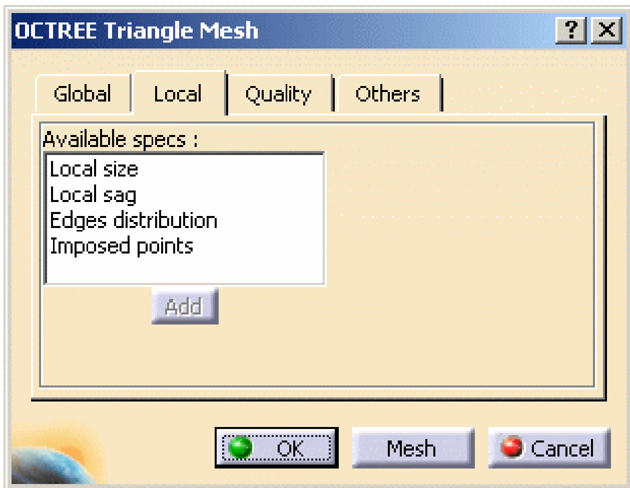
  - **Size**: lets you choose the size of the elements.
  - **Absolute sag**: maximal gap between the mesh and the geometry.



- **Proportional sag**: ratio between the **Absolute sag** and the local mesh edge length. **Proportional sag** value= (**Absolute sag** value) / (local mesh edge length value).
  - Note that **Proportional sag** and **Relative sag** could modify the local mesh edge length value.
  - You can use both **Proportional sag** and **Relative sag**, the most constraining of the two values will be used.
- **Element type**: lets you choose the type of element you want (**Linear** or **Parabolic**).

- **Local** tab:

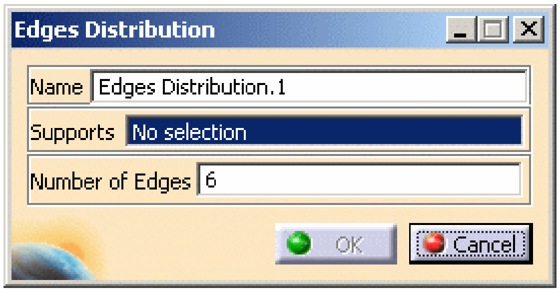
  You can also add local meshing parameters such as sag, size or distribution to the part.
  For this select the desired **Available specs** and then click the **Add** button.



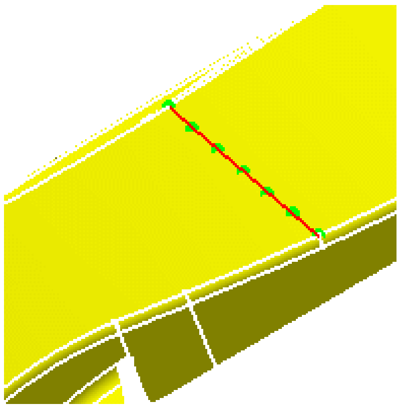
- **Local size**: you can modify the **Name, Support** and **Value.**

- **Local sag:** you can modify the **Name**, **Support** and **Value**.

- **Edges distribution**: lets you distribute local nodes on a edge.
  For this:
  - Select the **Edges distribution** option and click **Add.** The Edges Distribution dialog box appears.



- Select the edge on which you want to assign nodes (**Supports**) as well as the **Number of Edges** to be created.
  The **Edges Distribution.1** feature now appears in the specification tree.
- Click **OK** in the Local Mesh Distribution dialog box.
  In this particular case, 7 nodes are generated, i.e. 6 edges will be generated after the meshing.

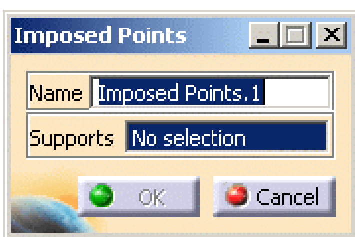


- **Imposed points**: lets you select the points that will be taken into account when meshing.

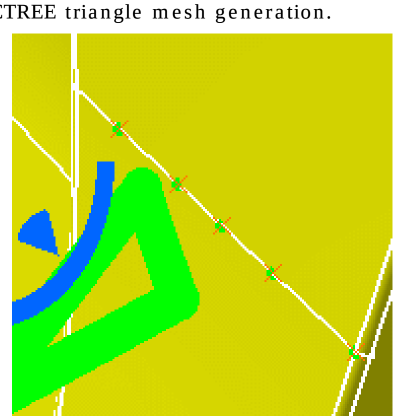
  In this case, the points you have to select must have been created via Shape Design or Part Design. Only points on curve or points on surface are supported. The points support must be a member of the meshed geometry.

For this:

- Select the **Local imposed points** option and click **Add.**
  The Imposed Points dialog box appears.



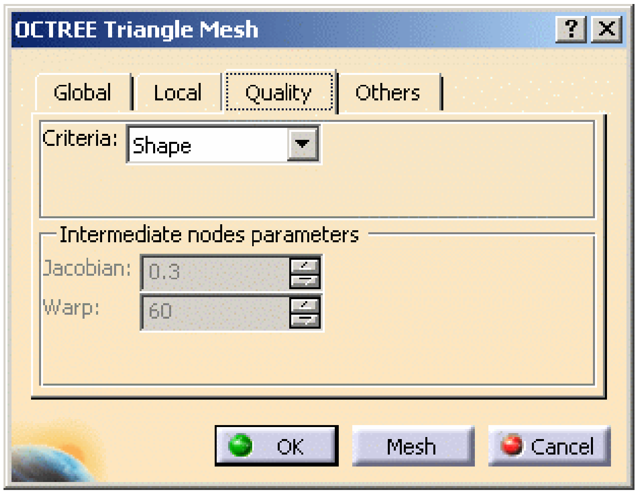
- Select from the specification tree (under **Open Body** feature) the points (**Supports**) you will impose for OCTREE triangle mesh generation.



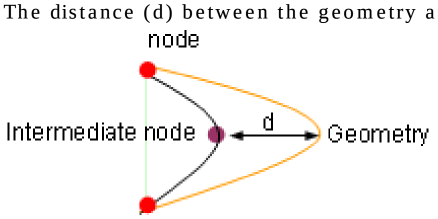
- Click **OK** in the Imposed Points dialog box.

To edit the local mesh distribution that has just been created, you need to double-click the **Local Nodes Distribution** object in the specification tree and modify the desired options from the Local Mesh Distribution dialog box that appears.

- **Quality** tab:

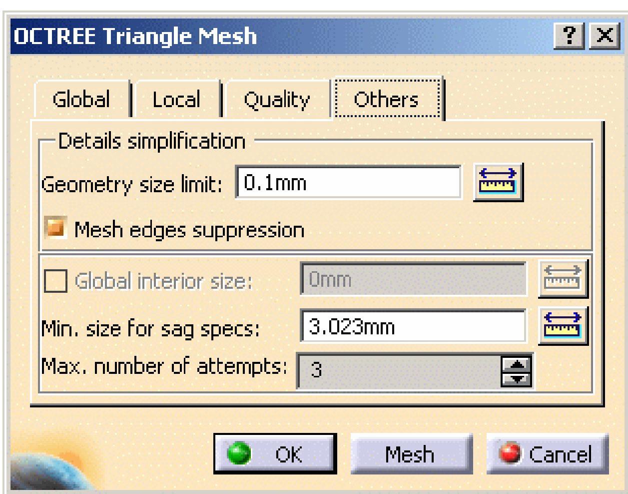


  - **Criteria**: lets you choose a criterion (**Shape**, **Skweness** or **Strech**) to optimize the mesh quality.
  - **Intermediate nodes parameters**: only available if you have chosen a **Parabolic** element type. This option lets you choose the position of parabolic tetrahedron intermediate nodes (Jacobian or Warp).
    The distance (d) between the geometry and the intermediate node is function of **Jacobian** and **Warp** values.



For more details about mesh quality analysis, please refer to Analyzing Element Quality.

- **Others** tab:

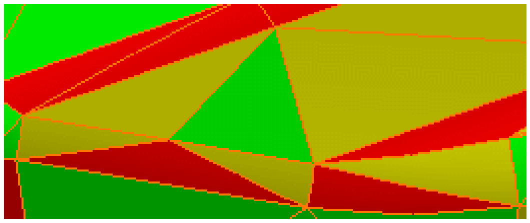


  - **Details simplification**: lets you remove small mesh.
    - **Geometry size limit**: lets you specify the maximum size of the elements ignored by the mesher (before the meshing).

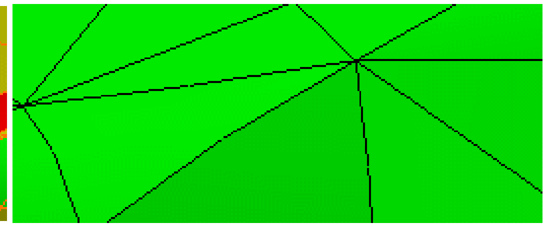
      If all the edges of a surface are smaller than the **Geometry size limit** value, this surface will be ignored by the mesher.

    - **Mesh edges suppression**: removes small edges (after the meshing).

Without **Mesh edge suppression**:



With **Mesh edge suppression**:



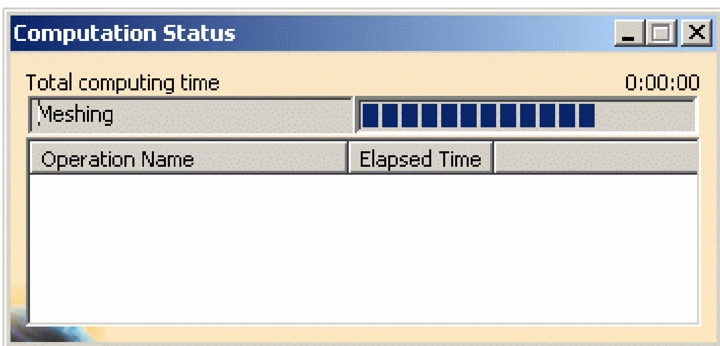
It may happen that **Mesh edge suppression** involve constraints violation.

- **Global interior size**: not available in Octree triangle mesh.
- **Min. size for sag specs**: lets you specify the minimum size of the mesh refining due to sags specifications.
- **Max. number of attempts**: lets you impose a maximum number of attempts, if several attempts are needed to succeed in meshing, in the case of a complex geometry.

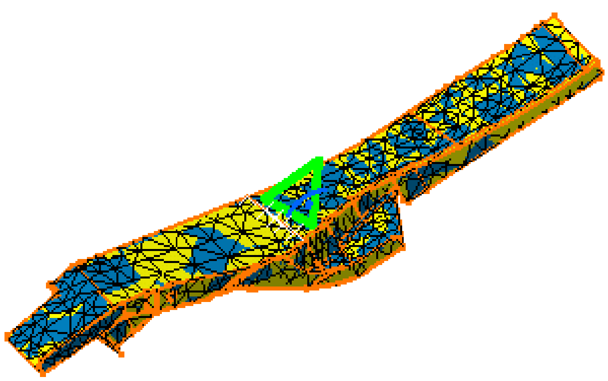
3. Select the desired parameters in the OCTREE Tetrahedron Mesh dialog box.

4. Click **Mesh** in the OCTREE Triangle Mesh dialog box.

   The Computation Status dialog box appears.



The OCTREE Triangle mesh is generated on the part.



To edit the Octree triangle mesh, double-click the **OCTREE Triangle Mesh.1** object in the specification tree. The OCTREE Triangle Mesh dialog box reappears.

Advanced Surface Mesher

This section shows you how to use the advanced surface mesher functionality and how to access the **Surface Meshing** workshop.

1. Enter the **Surface Meshing** workshop.
2. If needed, define local specifications of the surface mesher (using the Local Specifications toolbar).
3. Launch or remove the simplification geometry and/or the mesh (using the Execution toolbar).
4. If needed, perform manually modifications (using the Edition Tools toolbar).

At any time, you can:

a. access the global parameters if you want to visualize parameters you have defined or if you want to modify some of the global specifications.

b. exit the **Surface Meshing** workshop.

Entering the Surface Meshing Workshop

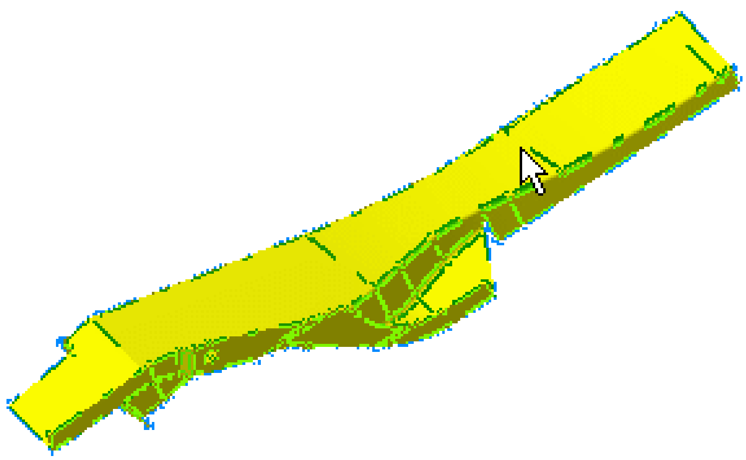
This task shows you how to enter the **Surface Meshing** workshop by:

- creating a new mesh part (using the **Advanced Surface Mesher** functionality)
- editing an existing mesh part

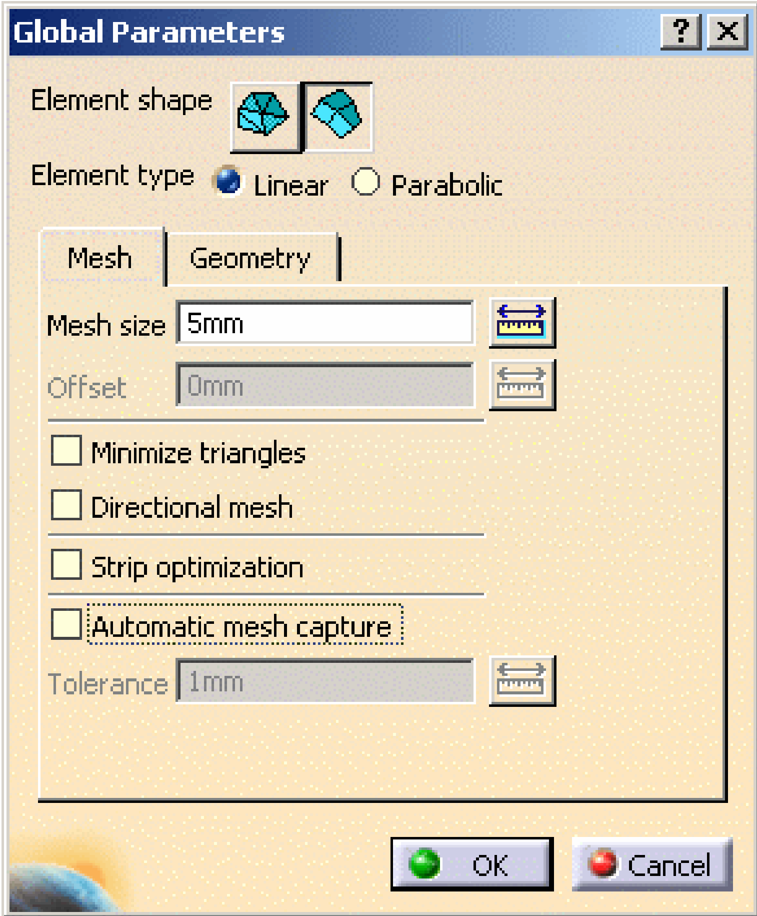
Creating a Mesh Part

Open the sample06.CATAnalysis document from the samples directory.

1. Click the **Advanced Surface Mesher** icon .
2. Select the geometry to be meshed by clicking on the part.



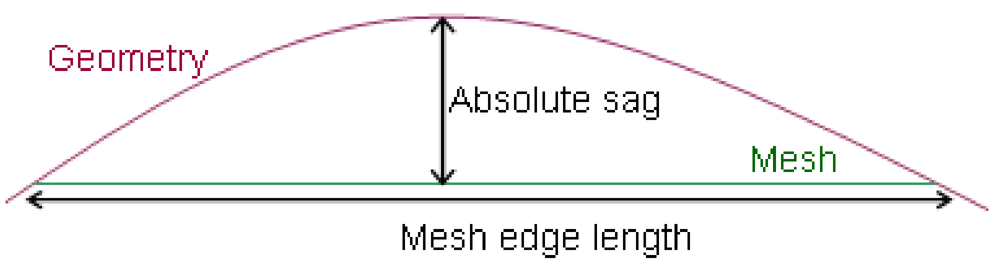
The Global Parameters dialog box appears.



- **Element type:** lets you specify if you want **Linear** or **Parabolic** elements.
- **Mesh** tab:

  Triangle Method 

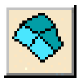
  - **Mesh size**: global size assigned to the mesh.
  - **Offset**: value according to which both the geometrical simplification and meshing will be offset.
  - **Absolute sag**: maximal gap between the mesh and the geometry.



  - **Relative sag**: ratio between the **Absolute sag** and the local mesh edge length. **Relative sag** value= (**Absolute sag** value) / (local mesh edge length value).

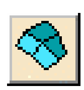
    - Note that **Absolute sag** and **Relative sag** could modify the local mesh edge length value.
    - You can use both **Absolute sag** and **Relative sag**, the most constraining of the two values will be used.

- **Min size**: minimum value of the mesh size. Only available when you want to use **Absolute sag** or **Relative sag**.
- **Automatic mesh capture**: when activated, mesh capture is performed dynamically on all the constraints (free edges, internal edges, external edges) and after all constraints modifications. You do not need to select all the constraints one after the others. Note that there is a capture tolerance, you can decide to impose or not a limitation to edge control neighborhood. Automatic capture is automatically performed using condensation. Meshing is then captured within the mesh part that belongs to the same CATAnalysis document, geometrically speaking.
  Be careful: mesh can only be captured on updated mesh part.
  - **Tolerance**

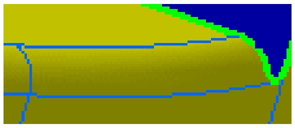
Quadrangle Method 

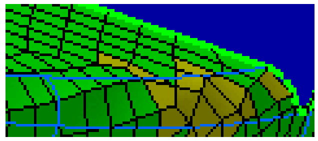
- **Mesh size**: global size assigned to the mesh
- **Offset**: value according to which both the geometrical simplification and meshing will be offset
- **Minimize triangles**: minimum number of triangles in the mesh
- **Directional mesh**: produces a regular mesh based on a privileged direction of the domain.
- **Strip optimization:** optimization the nodes position along the strips
- **Automatic mesh capture**: when activated, mesh capture is performed dynamically on all the constraints (free edges, internal edges, external edges) and after all constraints modifications. You do not need to select all the constraints one after the others. Note that there is a capture tolerance, you can decide to impose or not a limitation to edge control neighborhood. Automatic capture is automatically performed using condensation. Meshing is then captured within the mesh part that belongs to the same CATAnalysis document, geometrically speaking.
  Be careful: mesh can only be captured on updated mesh part.
  - **Tolerance**

- **Geometry** tab:

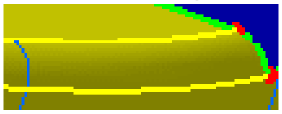
Triangle  *and Quadrangle*  *Methods*

- **Constraint sag**: constraint is created along the edge of a face to avoid creating elements across this edge (the element sag would be higher than the specified value). This does not guarantee that the whole mesh respects the sag value but helps creating constraints. For a given mesh size, the lower the constraint sag value, the more numerous the constraints are created, and vice versa.
  For example:
  Due to the sag value (too high), the edges are not constrained (blue colored).





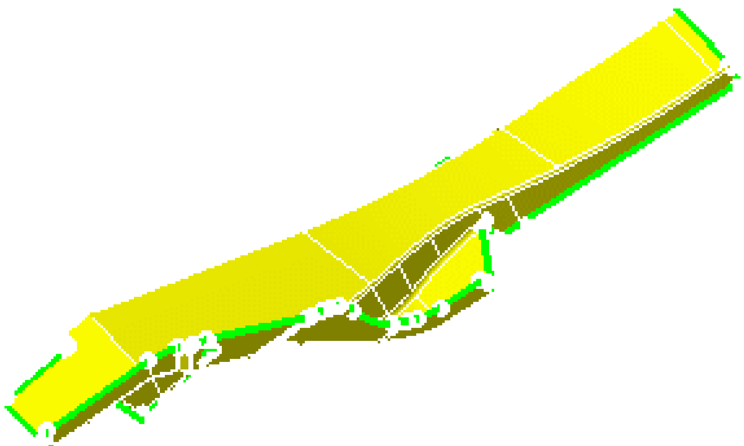
Due to the sag value (low enough), the edges are constrained (yellow colored).



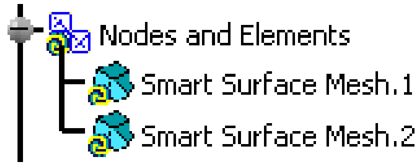
- **Angle between faces**: angle computed between the two normals corresponding to neighbor faces
- **Angle between curves**: angle computed between two tangents on a contour
- **Min holes size**: sets the diameter for automatic hole deletion
  - **Merge during simplification** option: allows optimizing the position of the nodes in tight zones in order to improve the quality of the elements
  - **Min size**
- **Automatic curve capture**: when activated, mesh capture is performed dynamically on all the constraints (free edges, internal edges, external edges) and after all constraints modifications. You do not need to select all the constraints one after the others. Note that there is a capture tolerance, you can decide to impose or not a limitation to edge control neighborhood. Automatic capture is automatically performed using condensation. Meshing is then captured within the mesh part that belongs to the same CATAnalysis document, geometrically speaking.
  Be careful: mesh can only be captured on updated mesh part.
  - **Tolerance**

3. Select the desired parameters in the Global Parameters dialog box.

The geometrical simplification is now launched.



A **Smart Surface Mesh** object appears in the specification tree.



4. Click **OK** in the Global Parameters dialog box.

You enter the **Surface Meshing** workshop.

You can now:

a. access the global parameters at any time,

b. define local specifications of the surface mesher,

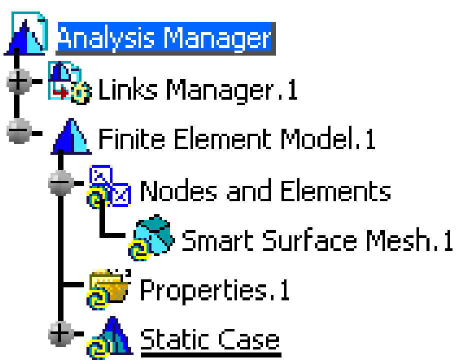
c. launch the simplification geometry or the mesh execution,

d. perform manually modifications,

e. exit the **Surface Meshing** workshop at any time.

Editing an Existing Mesh Part

Open the Sample06.CATAnalysis document from the samples directory.



1. Update the **Smart Surface Mesh.1** mesh part.

   For this, right-click the **Smart Surface Mesh.1** mesh part and select the **Update Mesh** contextual menu 

.

   This will launch both the geometry simplification and the mesh execution.

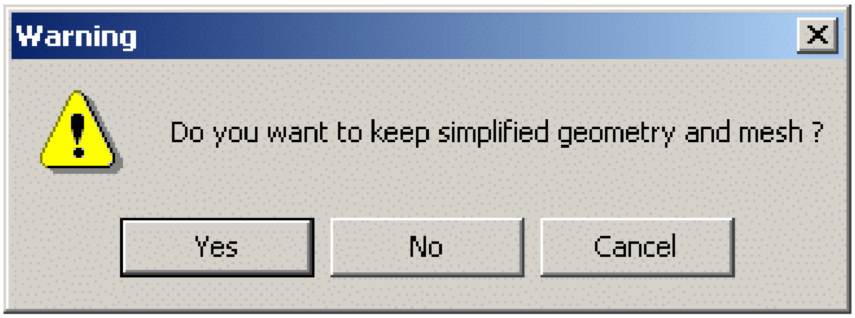
2. Double-click the **Smart Surface Mesh.1** mesh part in the specification tree.

   The Global Parameters dialog box appears.

   For more details about this dialog box, please click here.

3. Click **OK** in the Global Parameters dialog box without any modification.

   The following Warning message appears:



4. Click **Yes** in the Warning message.

   You enter the **Surface Meshing** workshop.

   You can now:

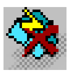
   a. access the global parameters at any time,

   b. define local specifications of the surface mesher,

   c. launch the simplification geometry or the mesh execution,

   d. perform manually modifications,

   e. exit the **Surface Meshing** workshop at any time.

5. Click the **Remove Mesh** icon  and leave the Surface Meshing workshop.

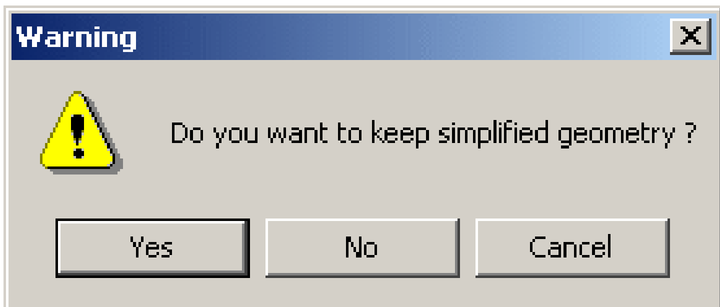
You return in the Advanced Meshing Tools workbench.

6. Double-click the **Smart Surface Mesh.1** mesh part in the specification tree.

   The Global Parameters dialog box appears.

7. Click **OK** in the Global Parameters dialog box without any modification.

   The following Warning message appears:



8. Click **Yes** in the Warning message.

   You enter again the **Surface Meshing** workshop.

Setting Global Meshing Parameters

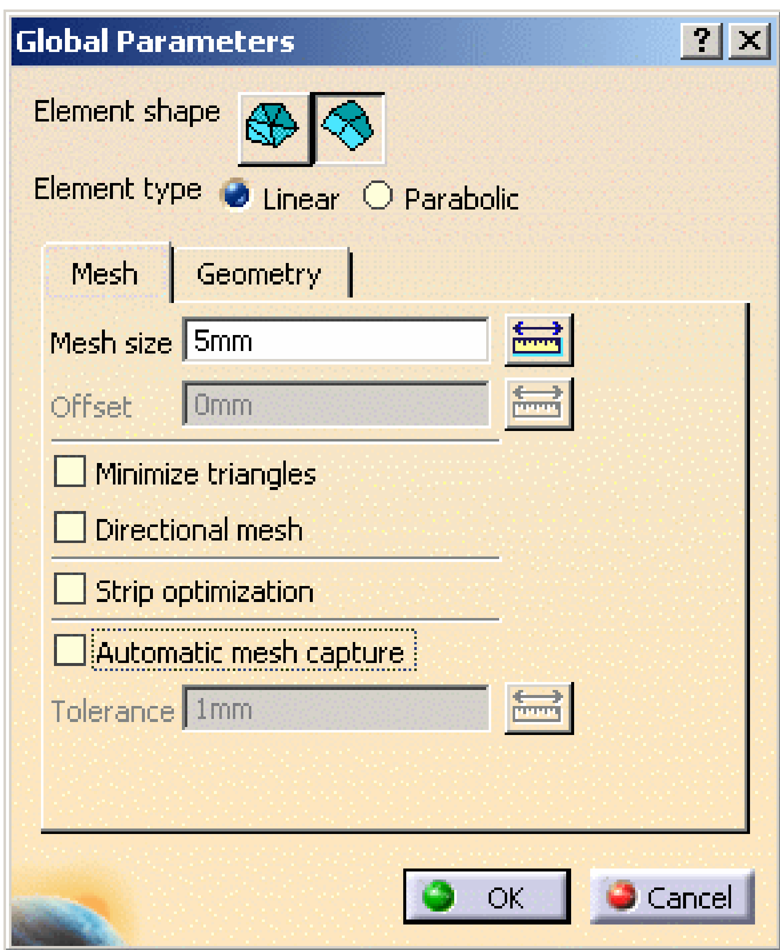
This task shows you how to visualize or modify the global meshing parameters at any time in the **Surface Meshing** workshop (before and after simplifying the geometry, before and after meshing).

- Open the sample06.CATAnalysis document from the samples directory.
- Enter the **Surface Meshing** workshop.

  For more details, please refer to Entering the Surface Meshing Workshop.

1. Click the **Global Meshing Parameters** icon  from the **Global Specifications** toolbar.

   The Global Parameters dialog box appears.



For more details about the description of the Global Parameters dialog box, please refer to Entering the Surface Meshing Workshop.

2. Change the desired parameters in the Global Parameters dialog box.

3. Click **OK** in the Global Parameters dialog box.

Local Specifications

Removing Holes: Ignore holes in the geometry.

Removing Cracks: Ignore cracks in the geometry.

Removing Faces: Ignore faces in the geometry.

Adding/Removing Constraints (Specifications): Add or remove constraints applied to vertices or curves.

Imposing Nodes (Specifications): Distribute nodes on edges.

Specifying a domain: Impose a domain for the geometry simplification step.

Removing Holes

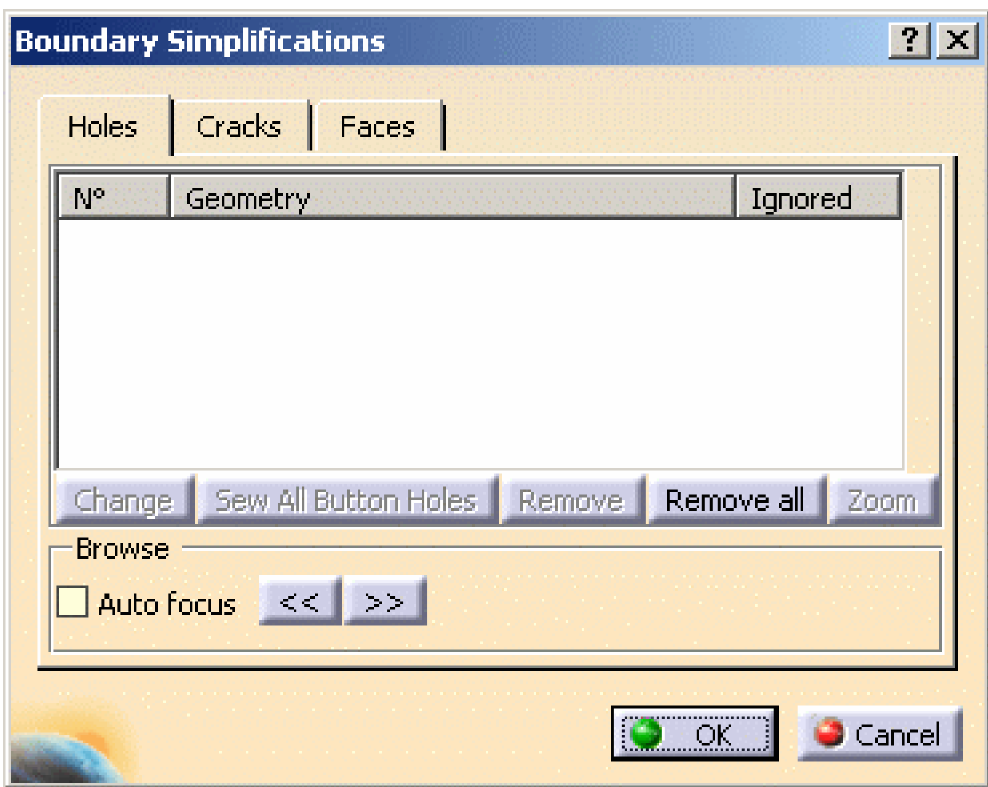
This task shows you how to ignore holes in the geometry which you consider as unnecessary holes.

- Open the Sample03.CATAnalysis document from the samples directory.
- Enter the **Surface Meshing** workshop.

  For more details, please refer to Entering the Surface Meshing Workshop.

1. Click the **Boundary Simplifications** icon from the **Local Specifications** toolbar.

The Boundary Simplifications dialog box appears:



- **Holes**:
  - **Browse:** browse the remaining holes
  - **Auto Focus:** zoom in given holes.
- **Cracks**: please refer to Removing Cracks.

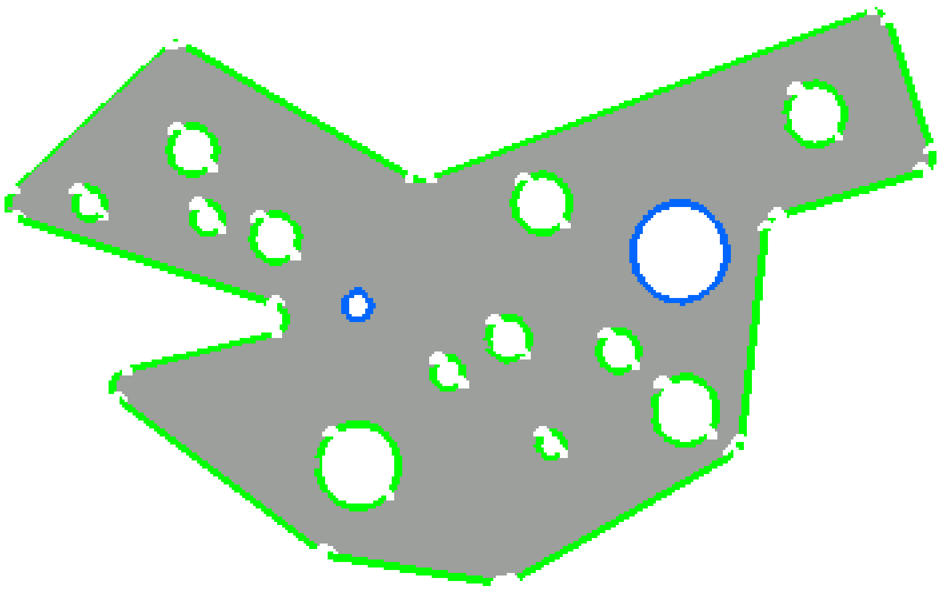
- **Faces**: please refer to Removing Faces.

2. Click a green hole.

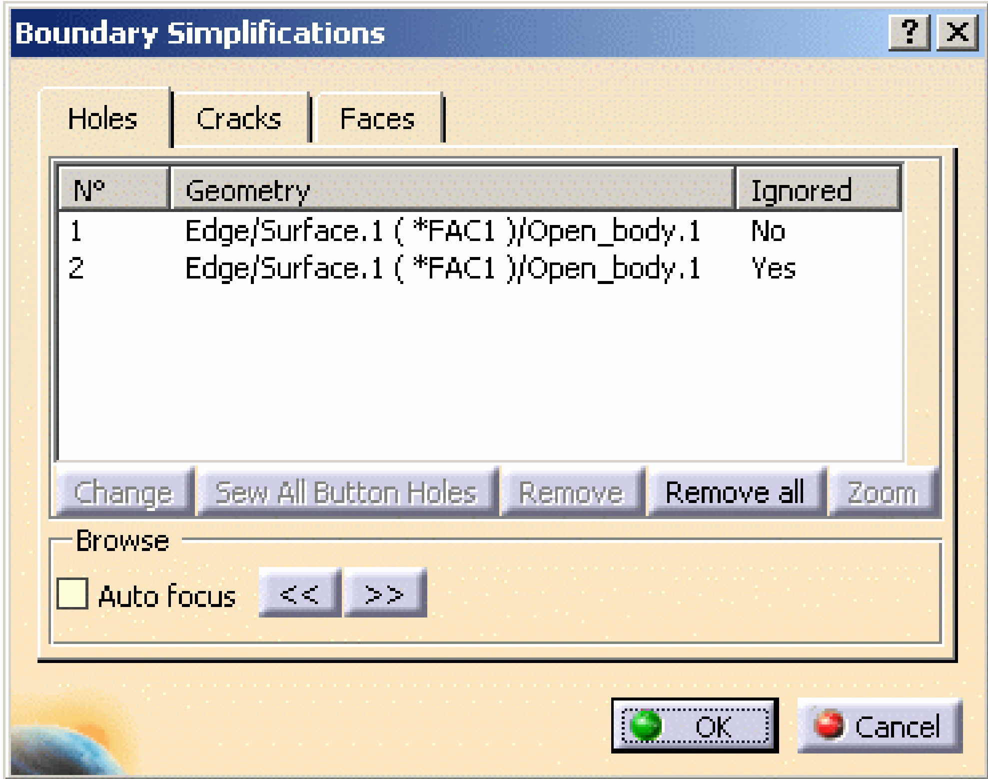
   It is turned blue and will then be ignored by the mesher.

3. Click a blue hole.

   It is reactivated and will be taken into account by the mesher.

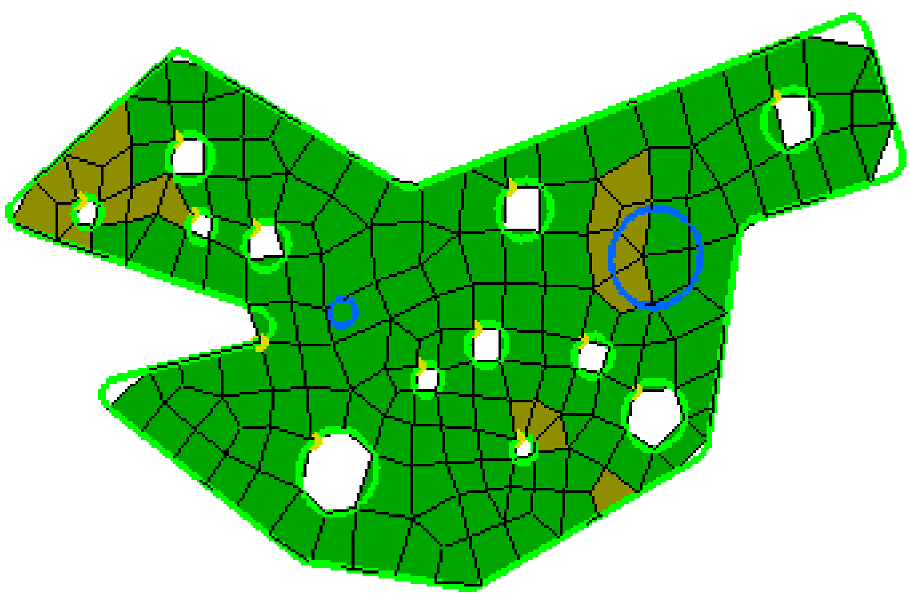


The Boundary Simplifications dialog box is updated in accordance with the hole(s) you selected or de-selected:



4. Click the **Mesh The Part** icon .

The meshed part now appears with ignored and non-ignored holes.



5. Click **OK** in the Mesh The Part dialog box.

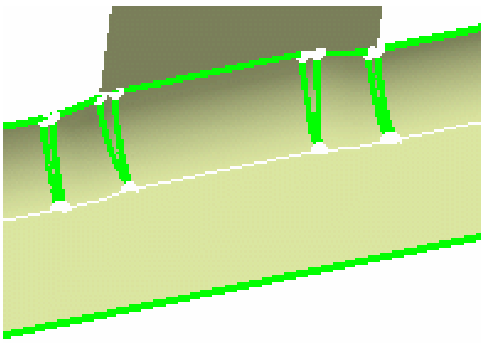
Removing Cracks

This task shows you how to ignore cracks in the geometry.

- Open the Sample31.CATAnalysis document from the samples directory.
- Enter the **Surface Meshing** workshop.

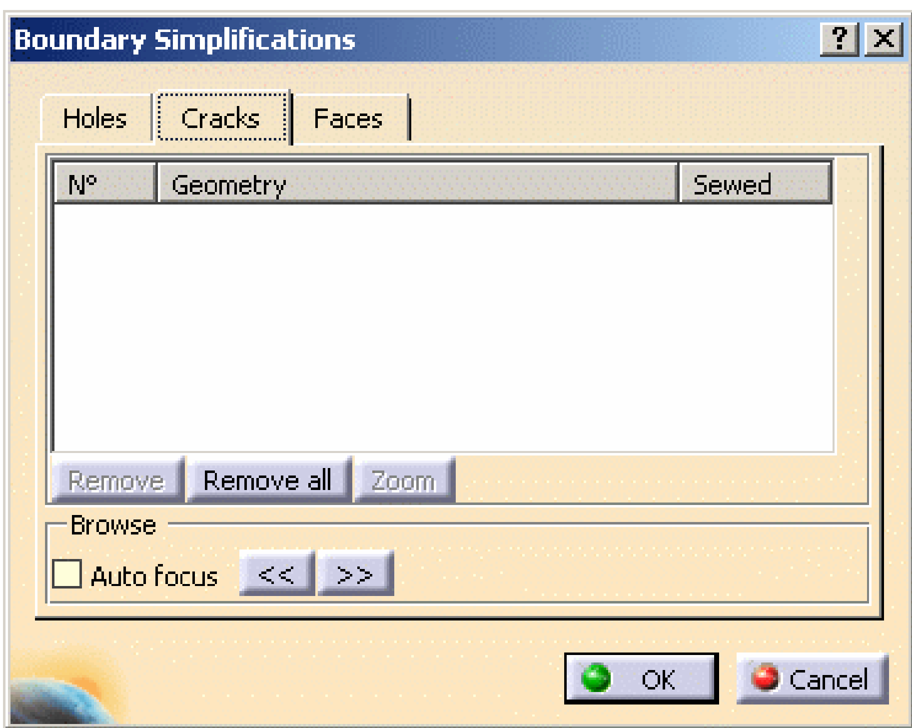
  For more details, please refer to Entering the Surface Meshing Workshop.

The CATAnalysis appears as shown here.



1. Click the **Boundary Simplifications** icon from the **Local Specifications** toolbar.

   The Boundary Simplifications dialog box appears:

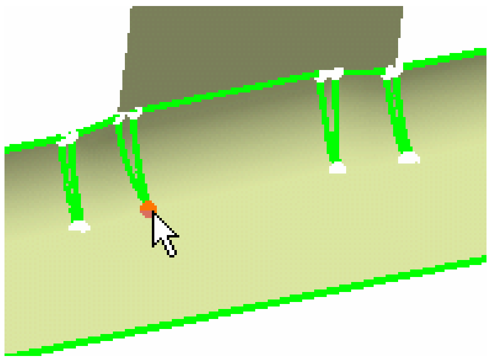


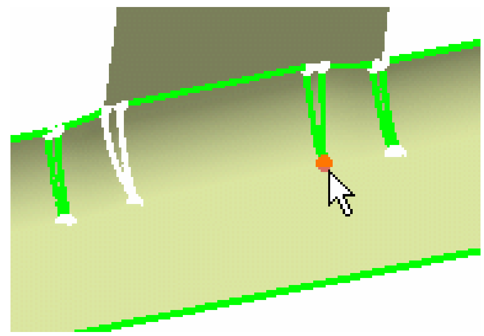
- **Holes**: please refer to Removing Holes.

- **Cracks**:
    - **Browse:** browse the remaining cracks
    - **Auto Focus:** zoom in given cracks.
- **Faces**: please refer to Removing Faces.

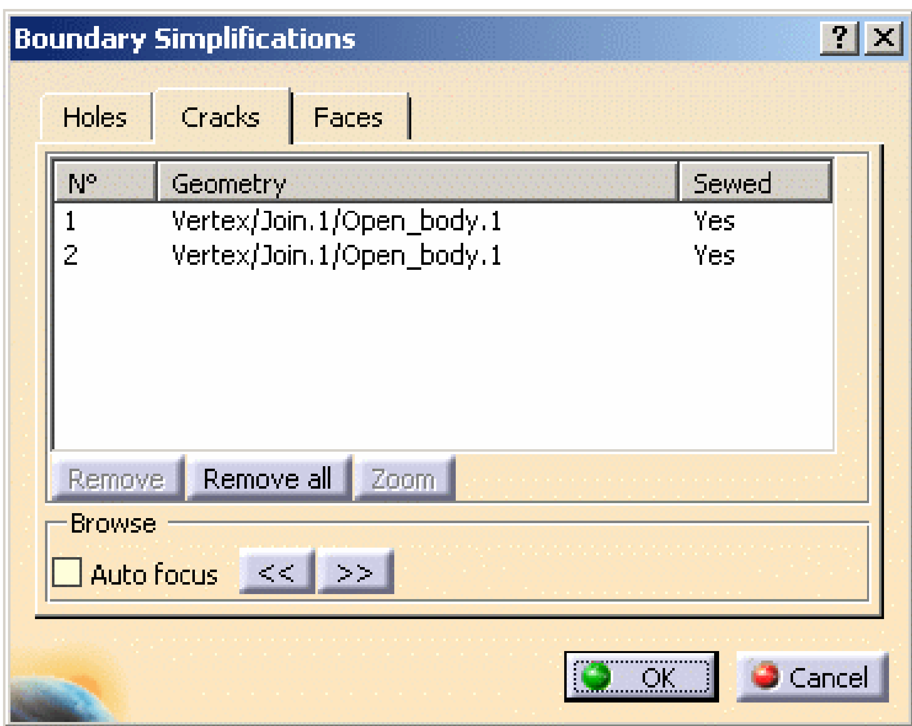
2. Select the **Cracks** tab in the Boundary Simplifications dialog box.

3. Click the cracks to be removed.





The Boundary Simplifications dialog box is updated accordingly:



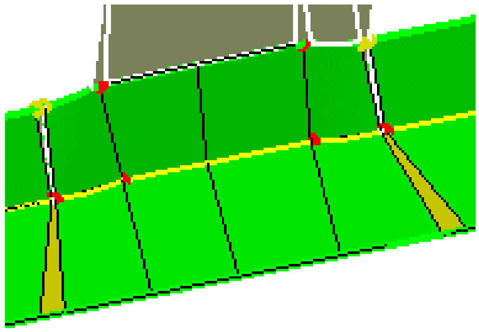
4. Click **OK** in the Boundary Simplifications dialog box.

You can now launch the Mesh operation. For this:

5. Click the **Mesh the Part** icon .

The Mesh The Part dialog box appears.

The boundary simplifications are taken into account.



If needed, you can now apply manual simplifications.

For this, click the **Edit Simplification** icon from the **Edition Tools** toolbar.

Removing Faces

This task shows how to ignore faces in the geometry.

- Open the Sample04.CATAnalysis document from the samples directory.
- Enter the **Surface Meshing** workshop.

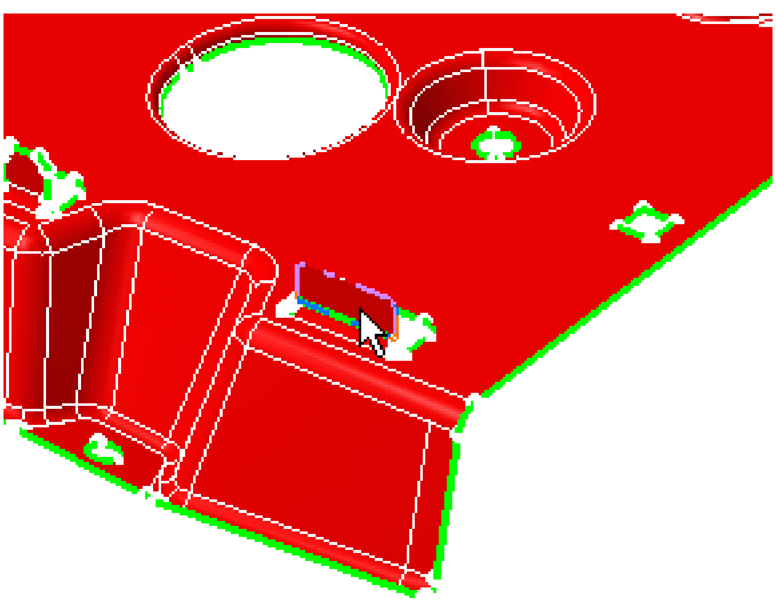
  For more details, please refer to Entering the Surface Meshing Workshop.
- Mesh the part.

  For this, click the **Mesh The Part** icon from the **Execution** toolbar.
  You will then click **OK** in the Mesh The part dialog box.

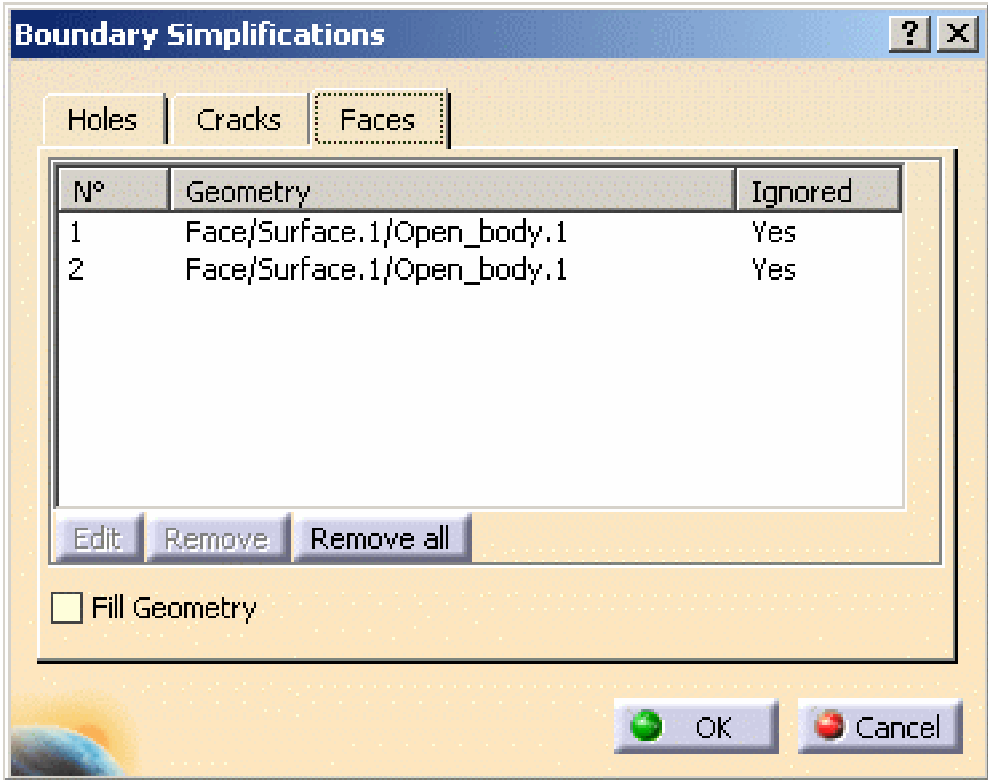
1. Click the **Boundary Simplifications** icon from the **Local Specifications** toolbar.

   The Boundary Simplifications dialog box appears. You can configure the mesher to ignore faces and holes by simply clicking on these faces and holes.

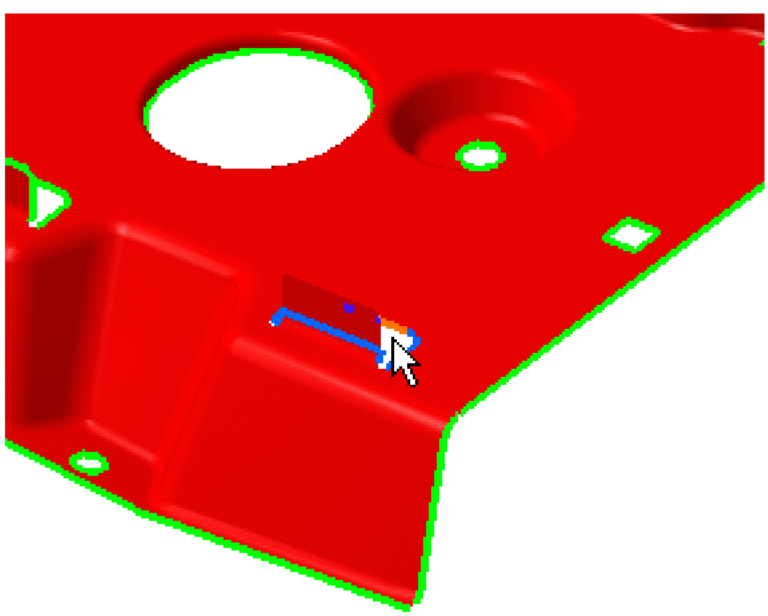
2. Select the **Faces** tab in the Boundary Simplifications dialog box and then the faces to be removed from the model.



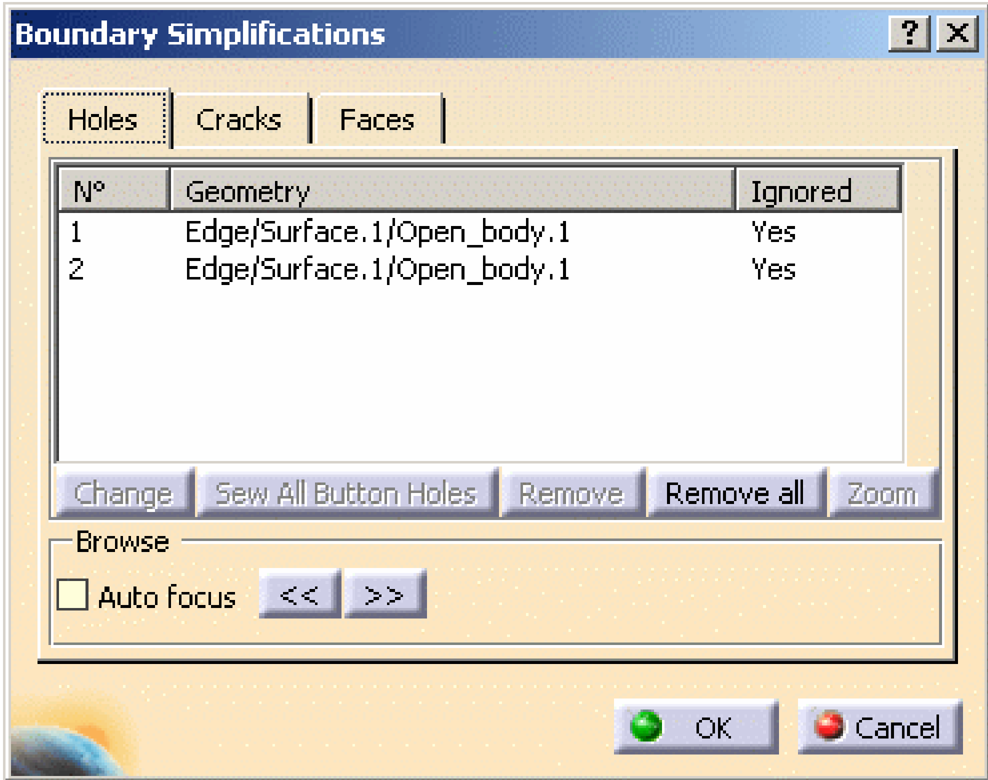
The selected faces automatically appear in the Boundary Simplifications dialog box.



3. If needed, select the **Holes** tab in the Boundary Simplifications dialog box and then the hole to be removed from the model.



The selected hole automatically appears in the Boundary Simplifications dialog box.

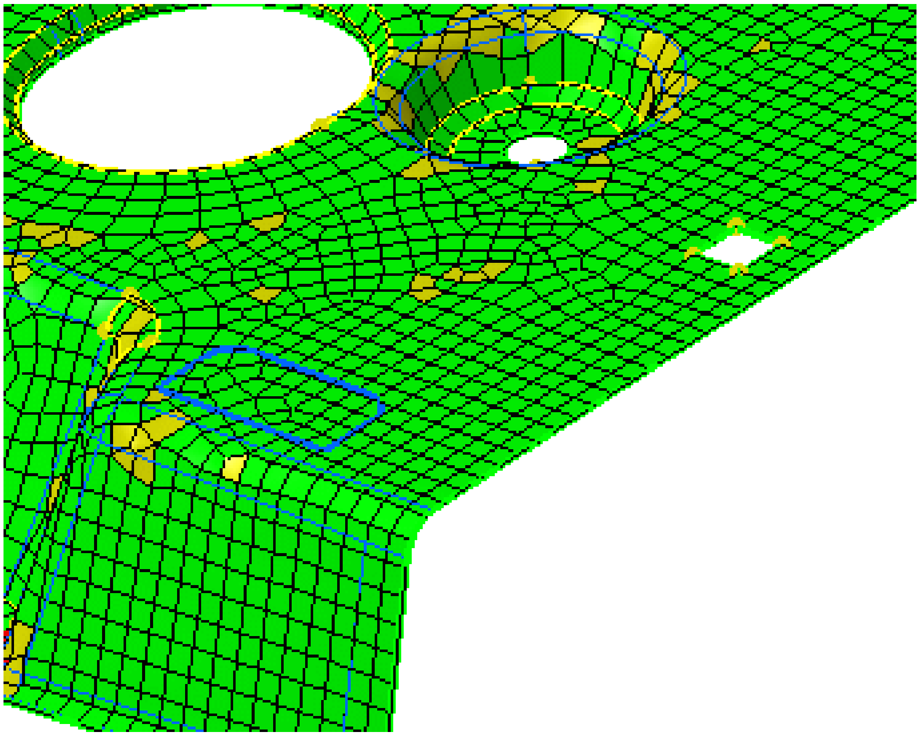


Now, the faces and holes will be ignored by the mesher.

4. Click **OK** in the Boundary Simplifications dialog box.

You can now launch the Mesh operation.

For this, click the **Mesh the Part** icon .



Adding/Removing Constraints (Specifications)

This task shows how to add/remove constraints as mesh specifications either using the geometry (curves or point) or directly on the geometrical simplification (edges or vertices).

Adding/Removing Constraints (Specifications):

- On/From The Geometrical Simplification
- On/From The Geometry
- Using edge/external curve selection by path

Two types of constraints exist:

1. a constraint applied to a **vertex/point**: as a result, a node will be created on this vertex.
2. a constraint applied to a **edge/curve**: as a result, all the element edges will be aligned on this curve.

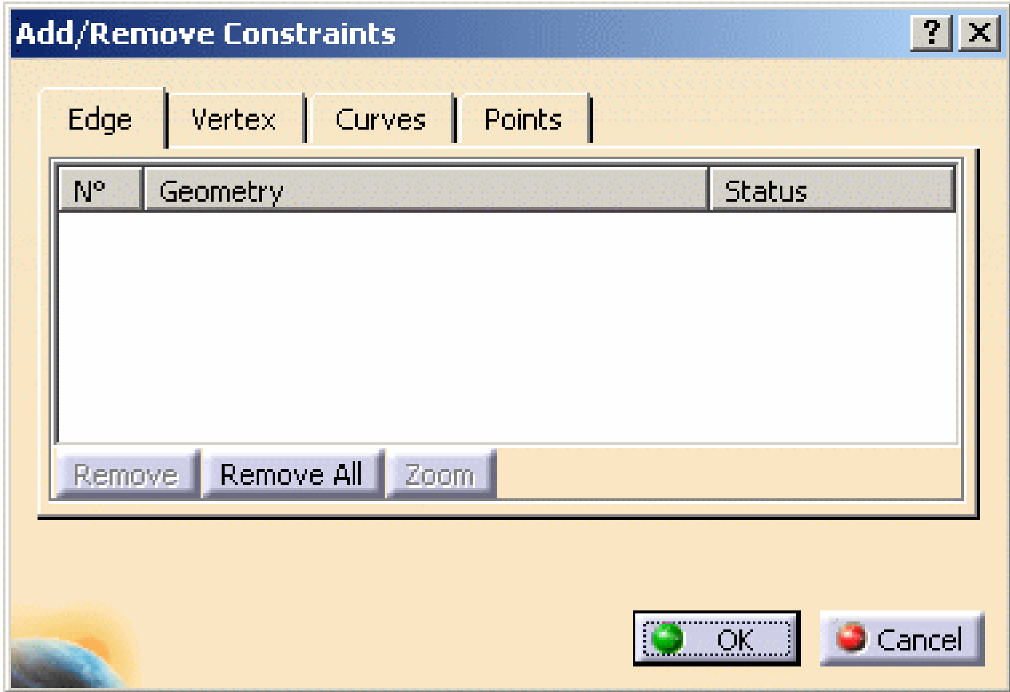
On/From The Geometrical Simplification

- Open the sample05.CATAnalysis document from the samples directory.
- Enter the **Surface Meshing** workshop.

  For more details, please refer to Entering the Surface Meshing Workshop.

1. Click the **Add/Remove Constraints** icon from the Local Specifications toolbar.

The Add/Remove Constraints dialog box appears with tabs that will allow you assigning constraints to edges, vertices / curves, points.

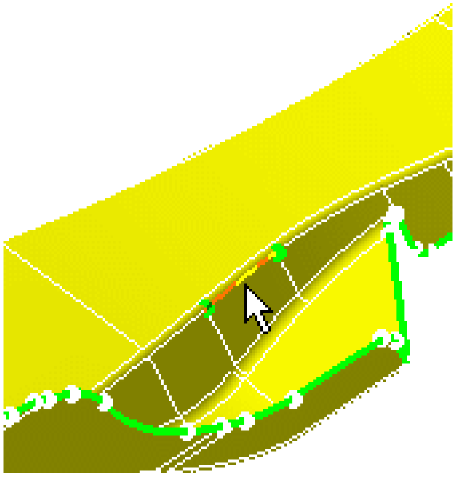


The Trap Type dialog box also appears and remains displayed as long as you keep the **Edge** tab active:

You can use multi-selection:

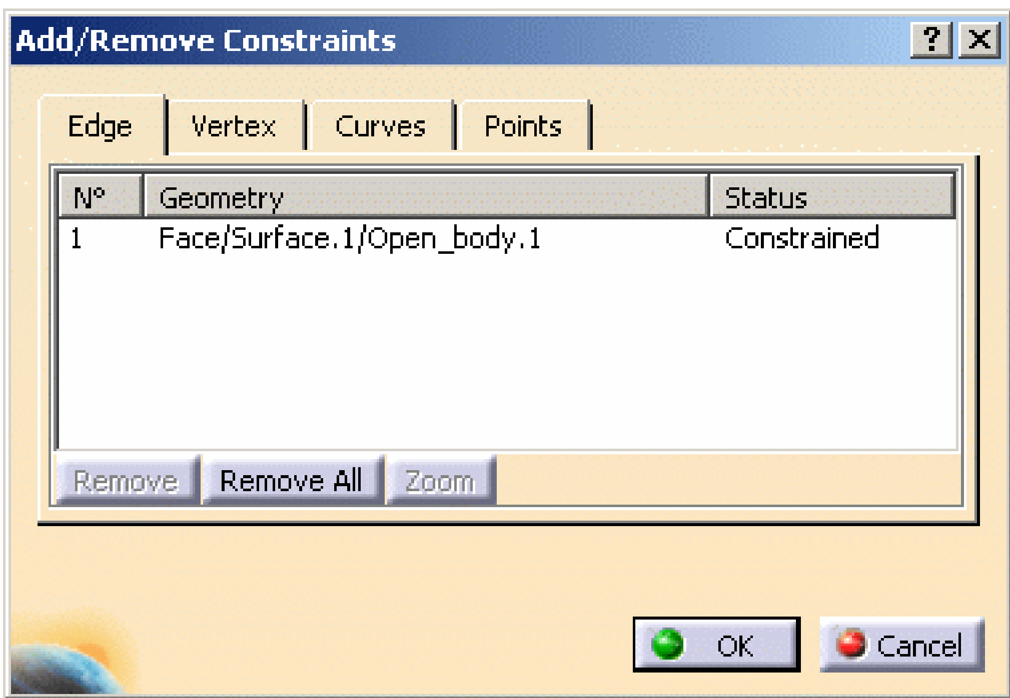
- **using an intersection polygon trap**: all the domains which have non-empty intersection with the trap will be selected.
- **using an inclusive polygon trap**: all the domains that are completely included within the trap will be selected.

2. Select the edge which you want to constrain.

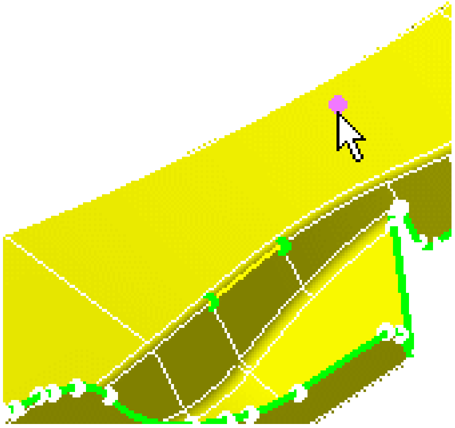


Note that you can use edge selection by path.

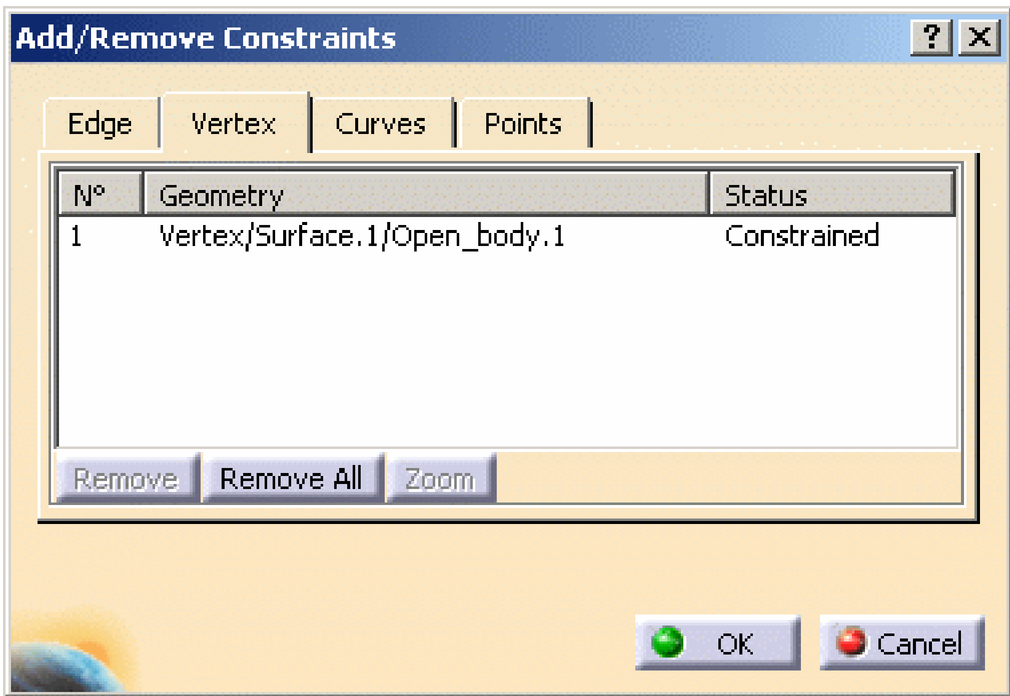
The Add/Remove Constraints dialog box now displays information on the element just selected.



3. Select the vertex which you want to constrain.



The dialog box now displays information on the element just selected.



If you select one element in this dialog box buttons become selectable:

- **Remove**: you can remove one constraint you previously created using this dialog box.
- **Remove All**: you can remove all the constraints you previously created using this dialog box.
- **Zoom**: you can zoom in on the constraint that is currently selected in this dialog box.

4. Click **OK** in the Add/Remove Constraints dialog box.

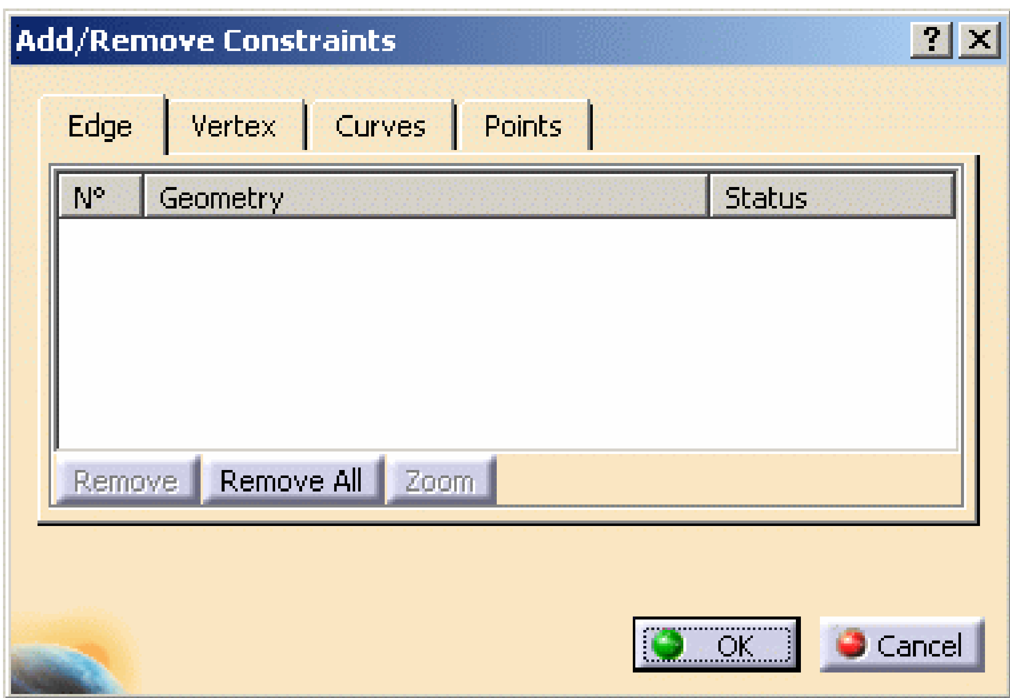
On/From The Geometry

- Open the sample17.CATAnalysis document from the samples directory.
- Enter the **Surface Meshing** workshop.

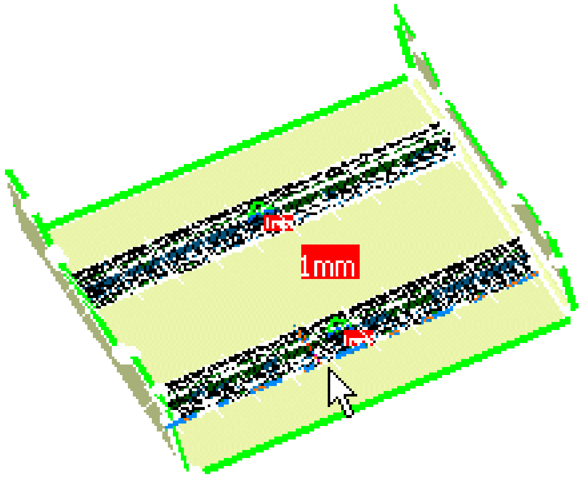
  For more details, please refer to Entering the Surface Meshing Workshop.

1. Select the **Add/Remove Constraints** icon from the Local Specifications toolbar.

   The Add/Remove Constraints dialog box appears with tabs that will allow you assigning constraints to edges, vertices / curves, points.

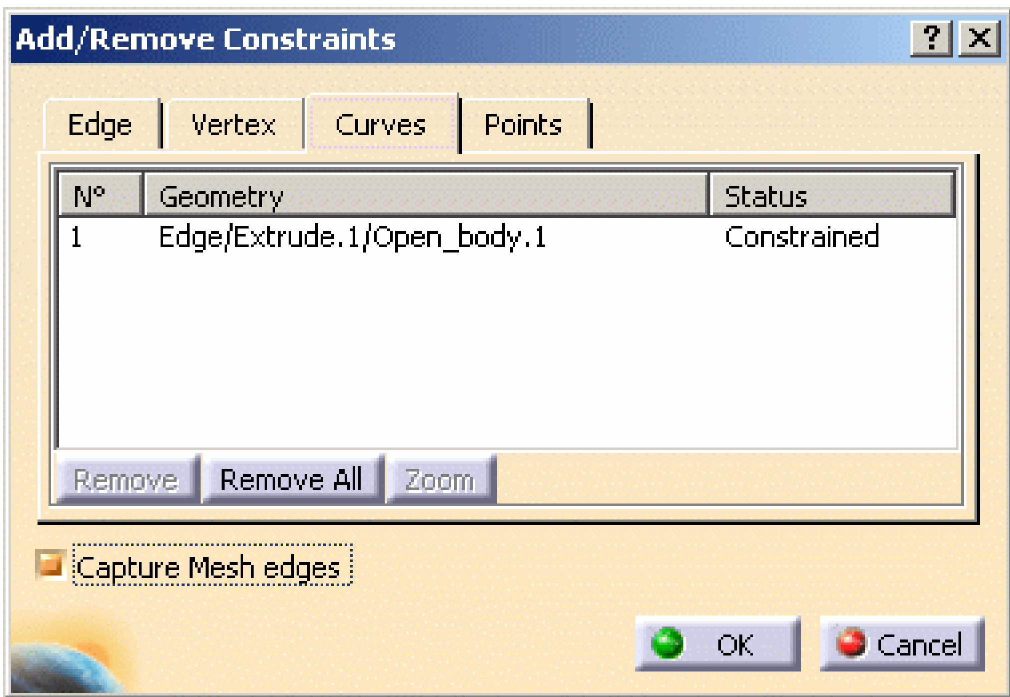


2. Select the **Curve** tab in the Add/Remove Constraints dialog box and the desired geometry on the CATAnalysis document.



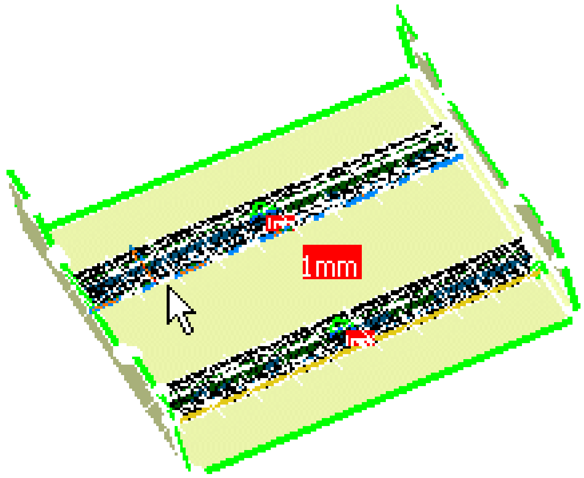
Note that you can select the mono-dimension feature of a face or an edge. When selecting a face, the edges of the face are projected. For mono-dimension features (for example, intersection features), all the edges of the feature are projected.

The Add/Remove Constraints dialog box now displays information on the elements just selected as well as the possibility to **Capture Mesh edges** from one curve to another.

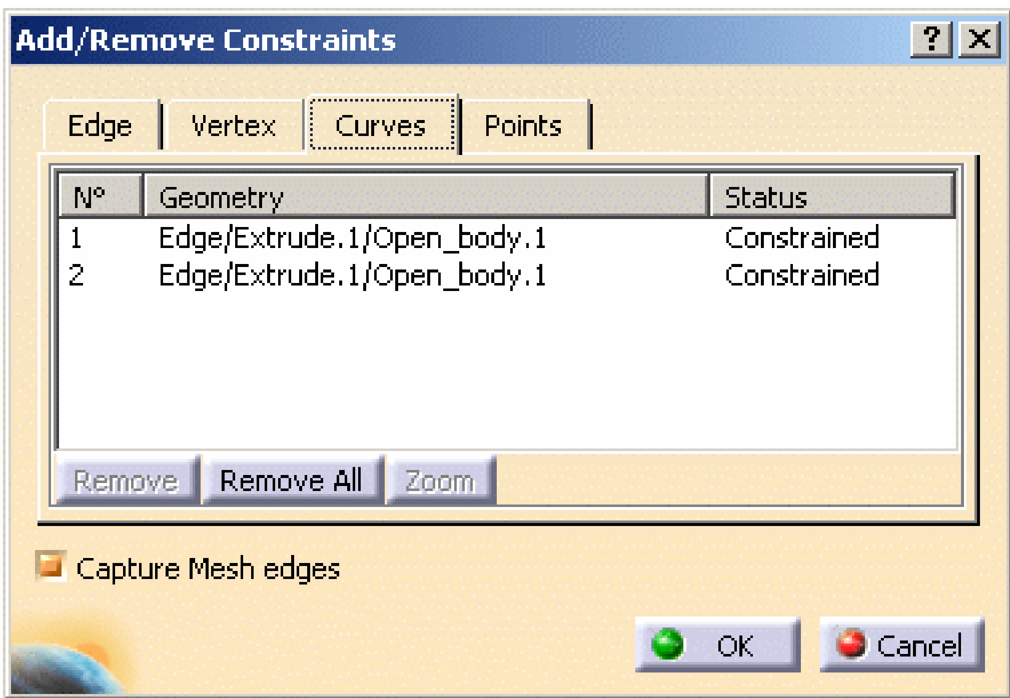


Note that you can use external curve selection by path.

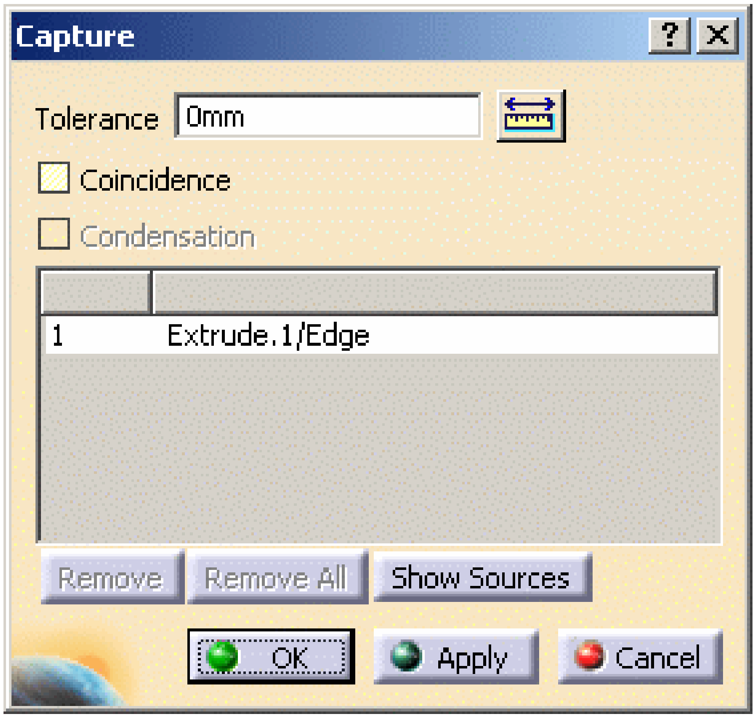
3. Select more curves on the CATAnalysis document.



The dialog box is now as shown here:



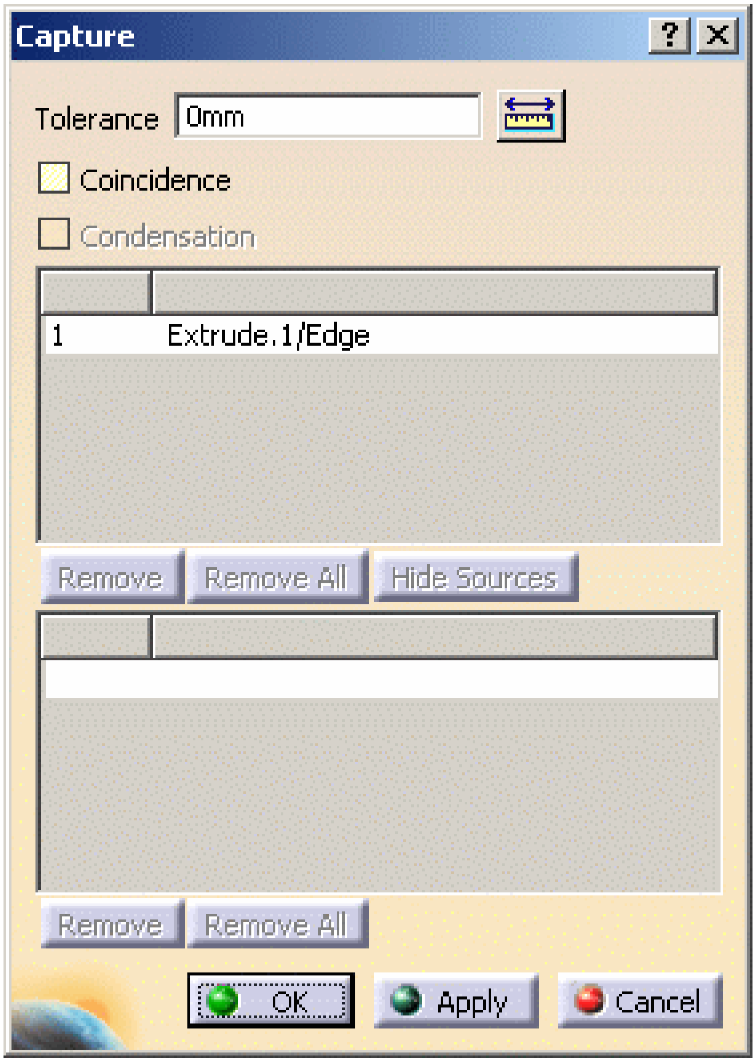
The Capture dialog box appears to let you select the destination curve edge and, if needed, (**Show Sources** button) a source curve edge. In other words, you will select both edges one after the other.



- **Tolerance**:
  Mesh will be found automatically relatively to the selected tolerance.
- **Coincidence**:
  You can decide that you will have the nodes from both edges superimposed (nodes created in coincidence)
- **Condensation**:
  You can decide that the nodes from both edges (receiver and source) are single nodes. The nodes of the source will be taken into account.
- **Receivers**:
  The edge on which the source will be projected.
- **Show Sources**:
  A switch that lets you select the curve to be projected onto the receiver.

4. Click **Show Sources** button in the Capture dialog box.

You can now select the curve to be projected onto the receiver

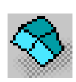


5. Click **OK** in the Capture dialog box.

6. Click **OK** in the Add/Remove Constraints dialog box.

7. Mesh the surface.

   For this:

   - click the **Mesh The Part** icon  from the Local Specifications toolbar.
   - You will then click **OK** in the Mesh The part dialog box.
   - The part is meshed accordingly.

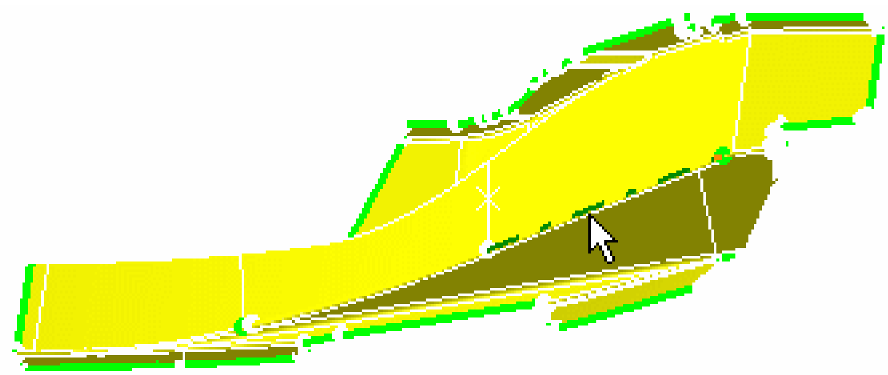
Using Edge/External Curve Selection by Path

- Open the sample05.CATAnalysis document from the samples directory.
- Enter the **Surface Meshing** workshop.

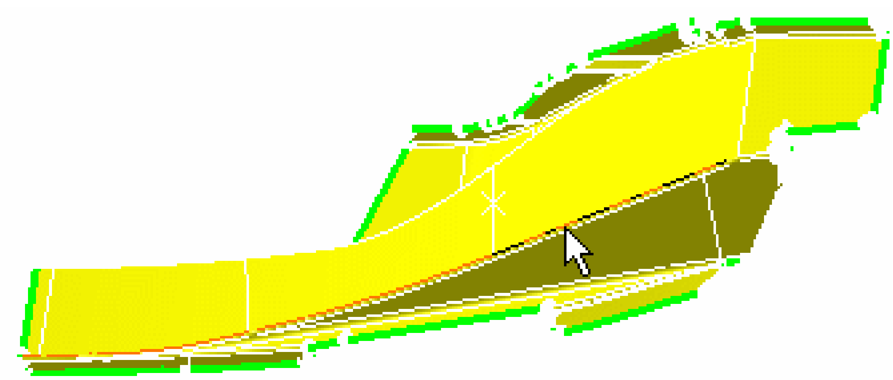
  For more details, please refer to Entering the Surface Meshing Workshop.

1. Click the **Add/Remove Constraints** icon from the Local Specifications toolbar.

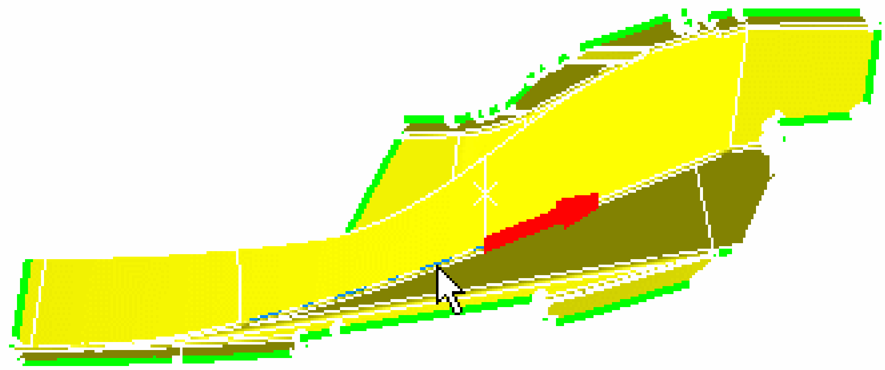
   The Add/Remove Constraints dialog box appears.
2. Go over an edge with the cursor.

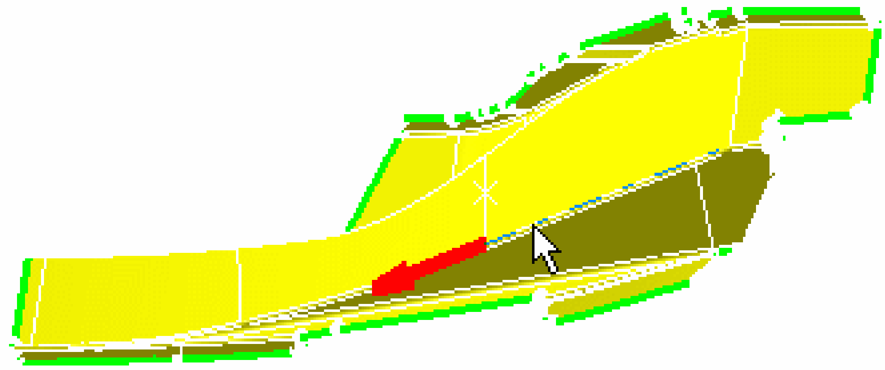


3. Press the **Shift** key.

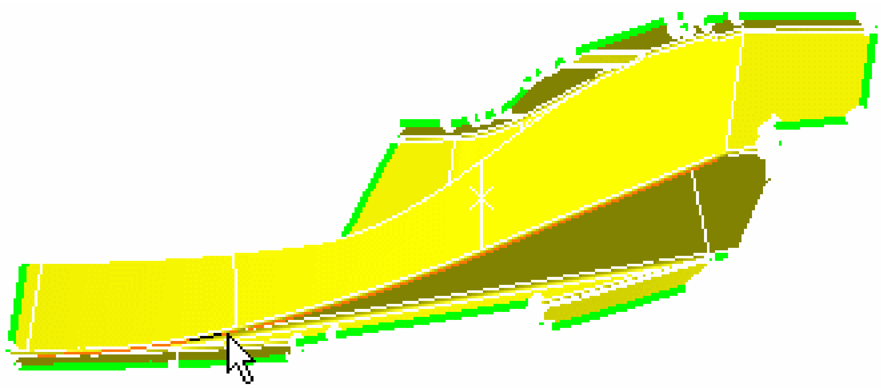


   The path corresponding to the edge you went over with the cursor is highlighted.
4. Choose the direction of the path by positioning the cursor on the path accordingly.

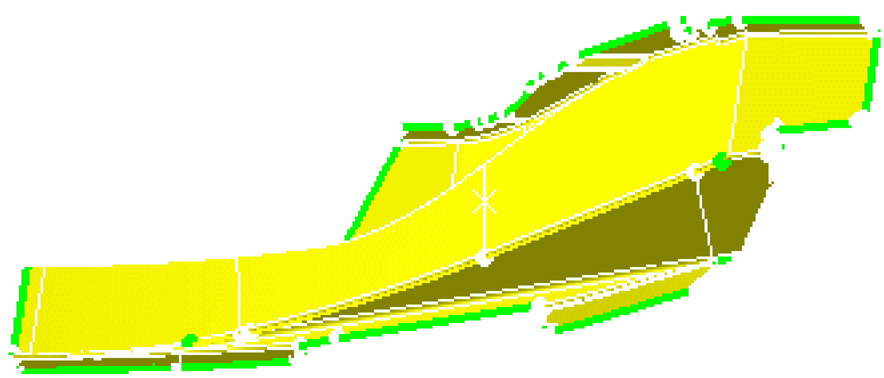




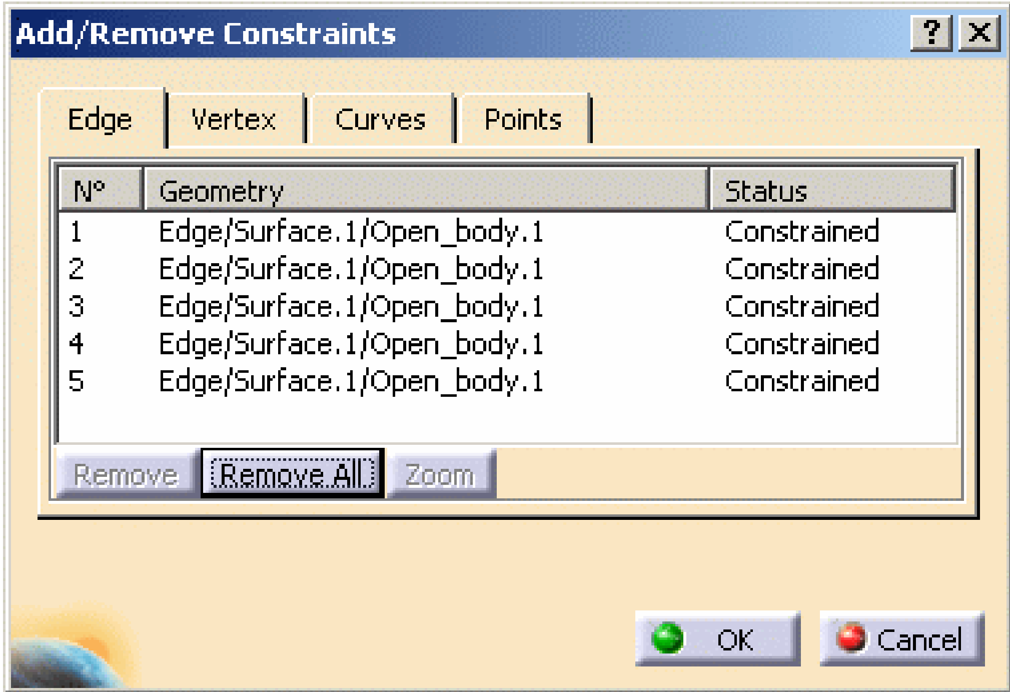
5. When you are satisfied with the operation, click on the path at the very point where you want the edge path to end.



The edge path is defined.



The Add/Remove Constraints dialog box is updated:



Only updated mesh parts will be constrained. In others words, when you constrain the edge of a part, if you modify the associated geometry (external edges), you need to force an update on this part to have this part up to date.

Imposing Nodes (Specifications)

This task shows how to impose nodes (as mesh specifications) either on free edges or on constrained edges.

- Distribute nodes
- Capture nodes
- Mesh around holes

- Open the sample07.CATAnalysis document from the samples directory.

- Enter the **Surface Meshing** workshop.

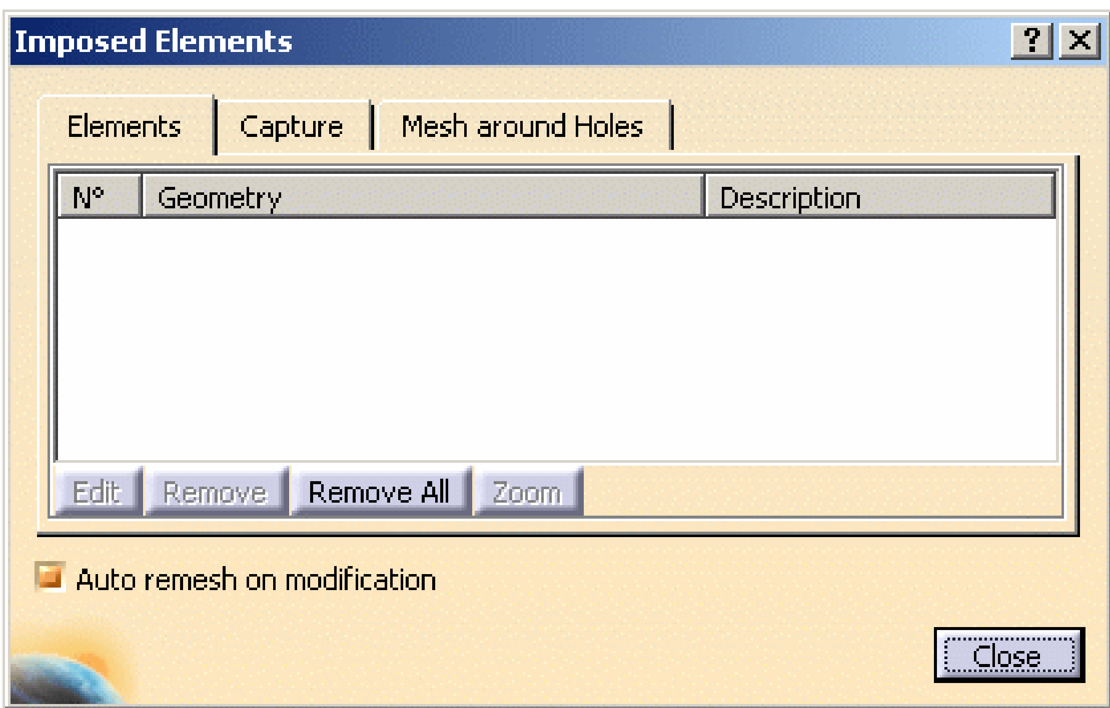
  For more details, please refer to Entering the Surface Meshing Workshop.

1. Click the **Imposed Elements** icon from the Local Specifications toolbar.

   The Imposed Elements dialog box appears.

Distribute Nodes

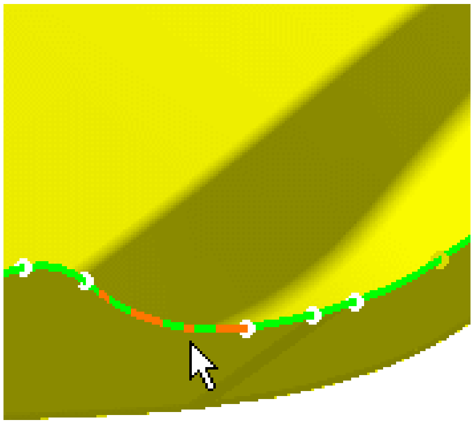
This Imposed Elements dialog box will display information on the nodes you are going to distribute.
Nodes can be distributed both on external and internal curves.



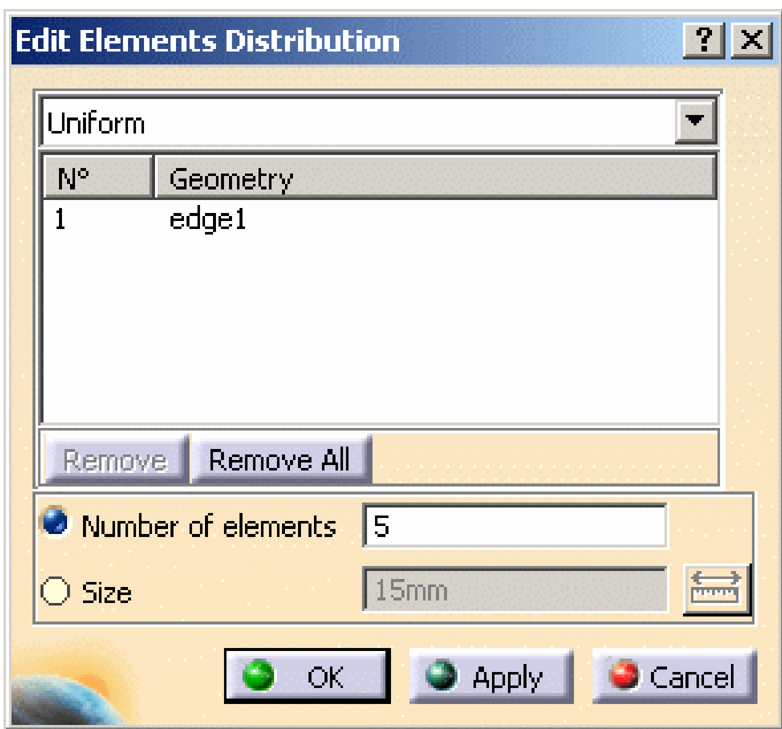
Note that you can use edge selection by path.

2. Select the geometry on which you want to distribute nodes.

   You can select several edges on the condition they are continuous to each others.



The Edit Elements Distribution dialog box now appears with given default values.



- Distribution type:
  - **Uniform**: the distance between all the distributed nodes will be the same.
  - **Arithmetic:** the distance between the distributed nodes will be defined by a common difference computed with the following parameters.

    Note that you have to specify two and only two parameters among the four.

    - **Number of Edges**: lets you specify how many edges you want.
    - **Size 2 / Size 1**: lets you specify the ratio between the lengths of the

last and the first distribution edge.

- **Size at node 1**: lets you specify the length of the first edge of the distribution.
- **Size at node 2**: lets you specify the length of the last edge of the distribution.

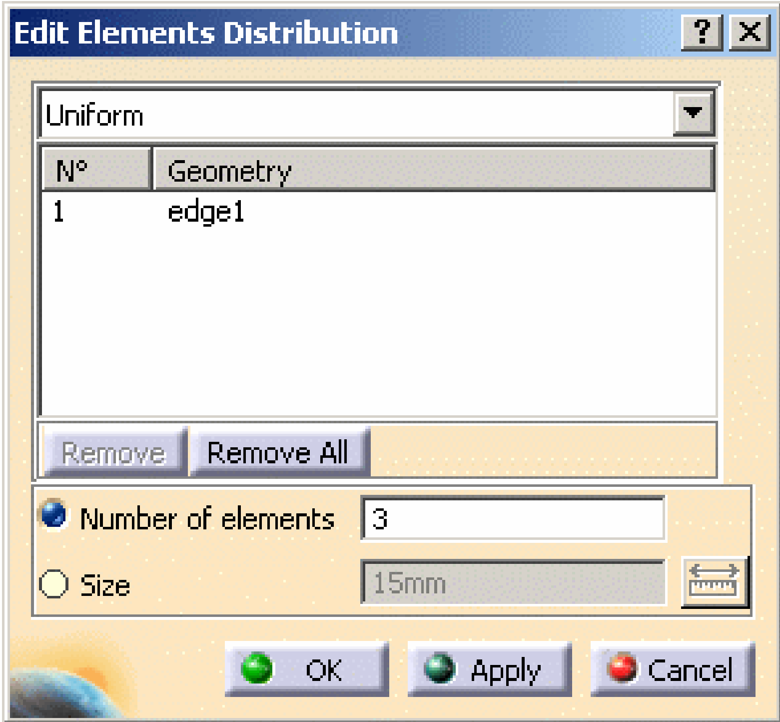
- **Geometric**: the distance between the distributed nodes will be defined by a common ratio computed with the following parameters.

  Note that you have to specify two and only two parameters among the four.

  - **Number of Edges**: lets you specify how many edges you want.
  - **Size 2 / Size 1**: lets you specify the ratio between the lengths of the last and the first distribution edge.
  - **Size at node 1**: lets you specify the distance between the two first nodes of the distribution.
  - **Size at node 2**: lets you specify the distance between the two last nodes of the distribution.

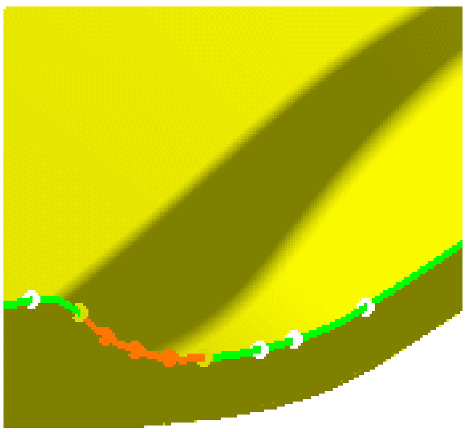
- Selected element(s): multi selection is available
- **Number of elements**: check this option to define the number of the nodes you want to distribute on the currently selected geometry.
- **Size**: check this option to define the size on which a given nodes will be uniformly distributed.

3. If needed, modify the number of the nodes or size. In this case, enter **3** as new value (**Number of elements**).

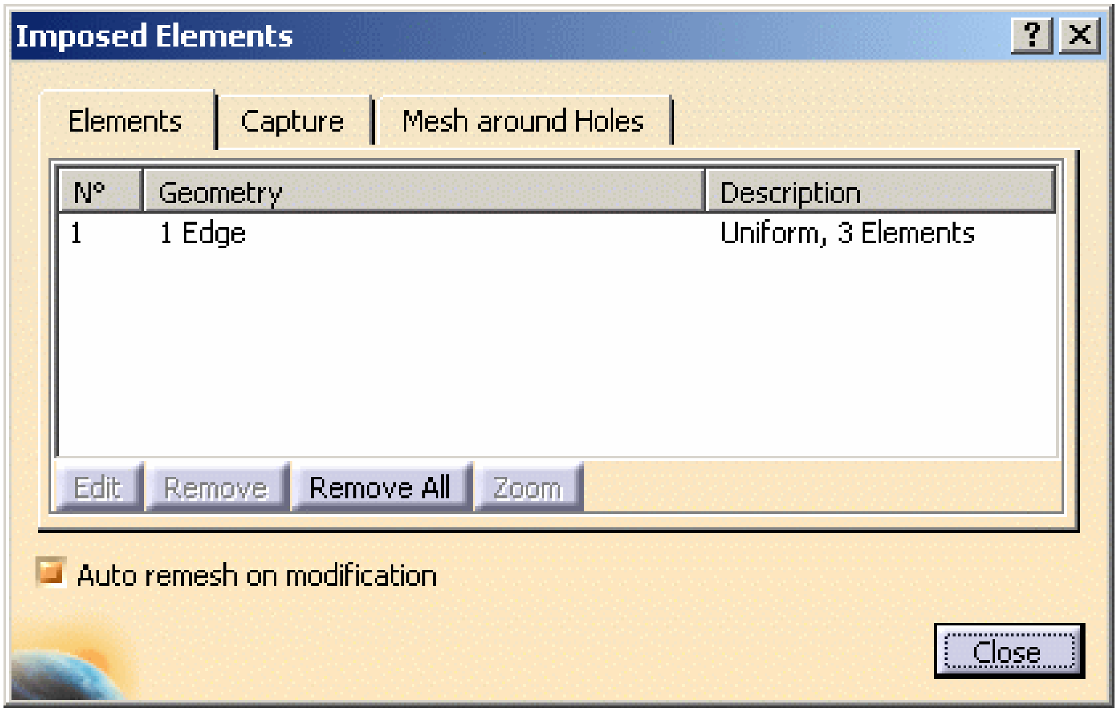


4. Click **OK** in the Edit Elements Distribution dialog box.

The nodes appear on the geometry.

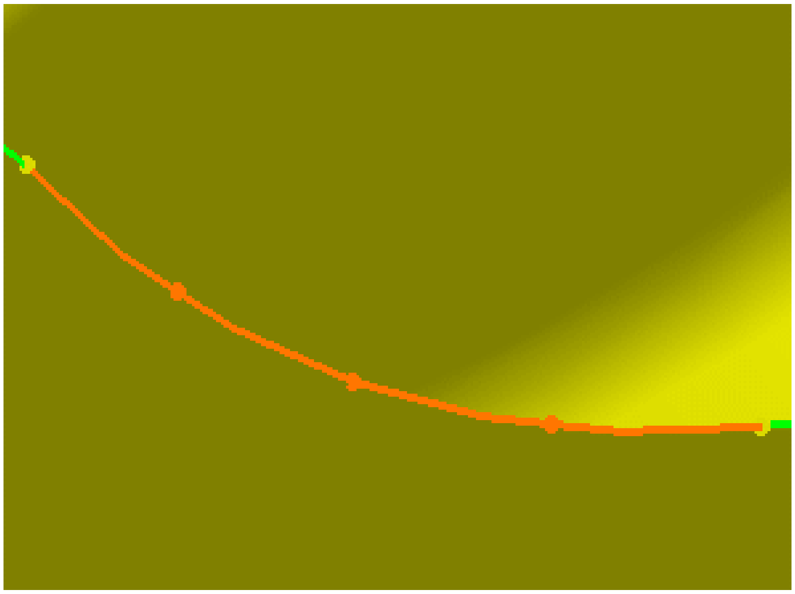


The Imposed Elements dialog box is automatically updated.

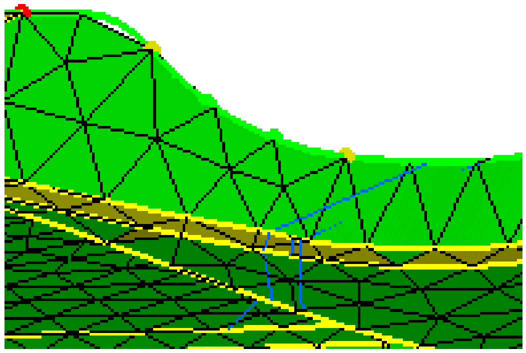


If you select one element in this dialog box switch buttons become selectable:

- **Edit**: you can display the Edit Elements Distribution dialog box and, if needed, modify the values.
- **Remove**: you can remove one node distribution you previously created using this dialog box.
- **Remove All**: you can remove all the node distributions you previously created using this dialog box.
- **Zoom**: you can zoom in on the node distribution that is currently selected in this dialog box.



The new nodes appear on the geometry which is automatically re-meshed.



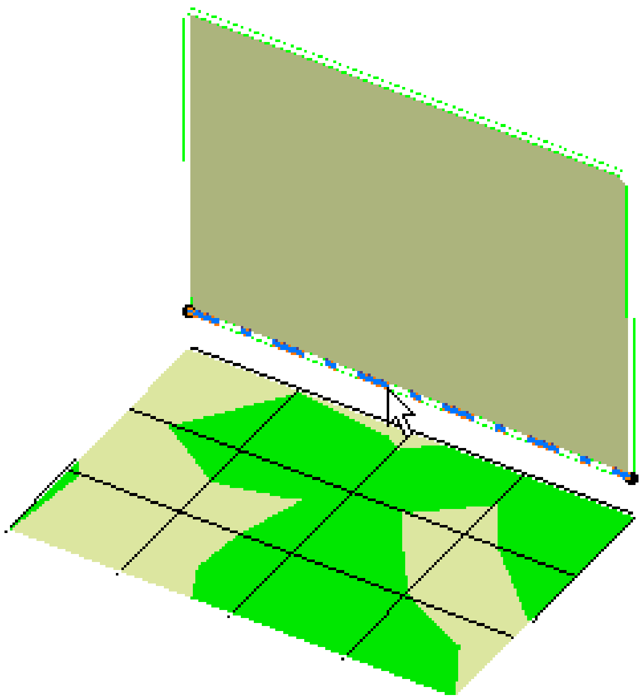
5. Click **OK** in the Imposed Elements dialog box once you are satisfied with you operations.

Capture Nodes

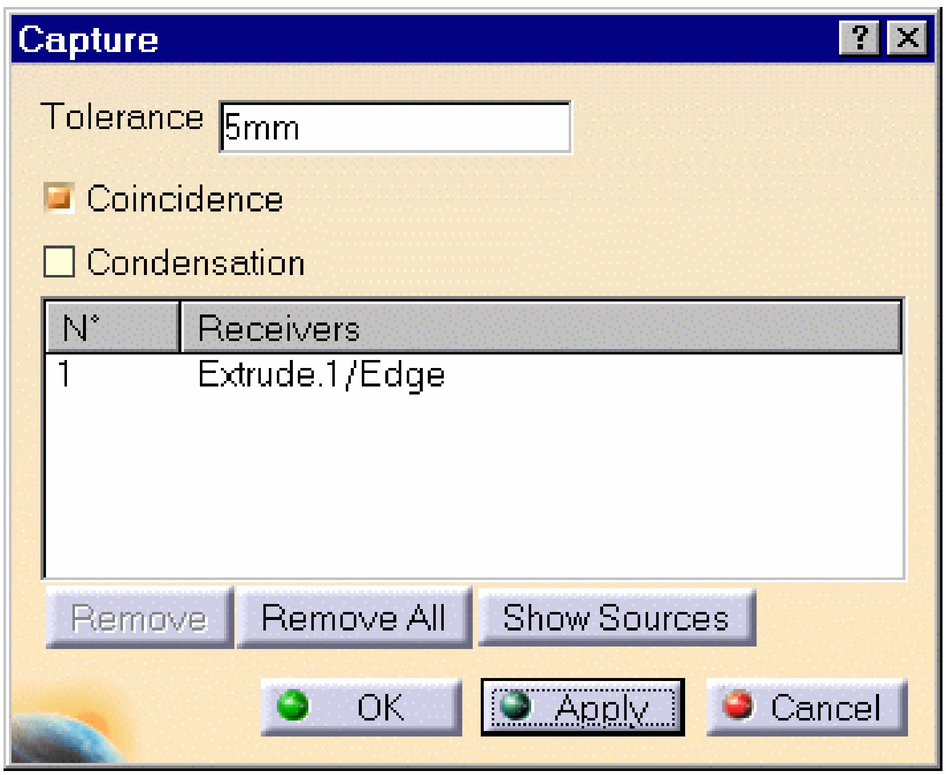
- Open the sample30.CATAnalysis document from the samples directory.
- Enter the **Surface Meshing** workshop.

  For more details, please refer to Entering the Surface Meshing Workshop.

1. Select the **Capture** tab in the Imposed Elements dialog box and the edge you want to be imposed nodes (also called Receiver edge).



The Capture dialog box now appears as shown here:

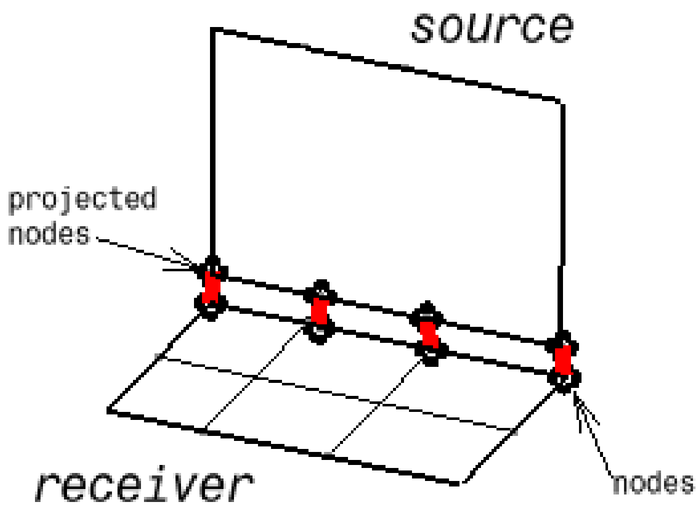


Note that you can use edge selection by path.

- **Tolerance**:
  The source edge will be found automatically relatively to the selected tolerance.
- **Coincidence**:

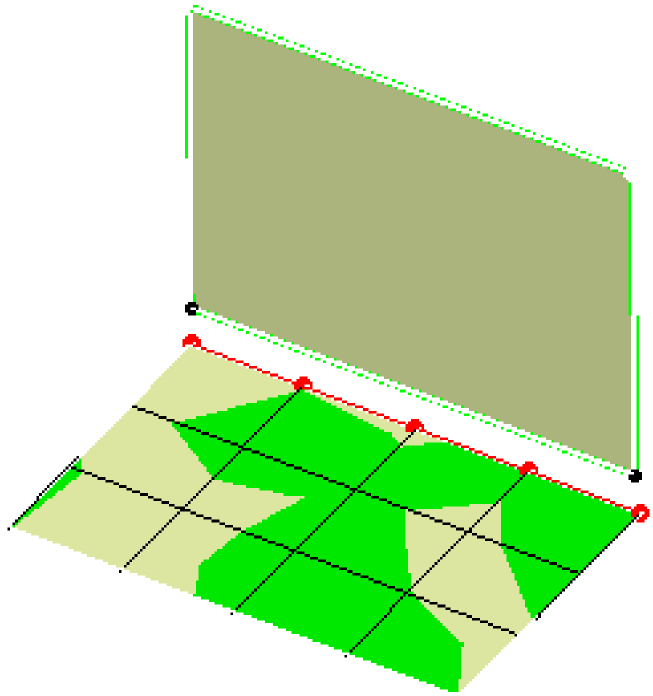
You can decide that you will have the nodes from both edges superimposed

- **Condensation**:
  You can decide that the nodes from both edges (receiver and source) are single nodes.
- **Receivers**:
  The edge on which the source will be projected.
- **Show Sources**:
  A switch that let's you select the mesh edges to be projected onto the receiver.
  Be careful: mesh can only be captured on updated mesh part.



2. Enter the desired **Tolerance** value, for example **5mm**, and click **OK** or **Apply** from the Capture dialog box.

   The nodes from a source edge at a 5mm from the receiver edge now appear.

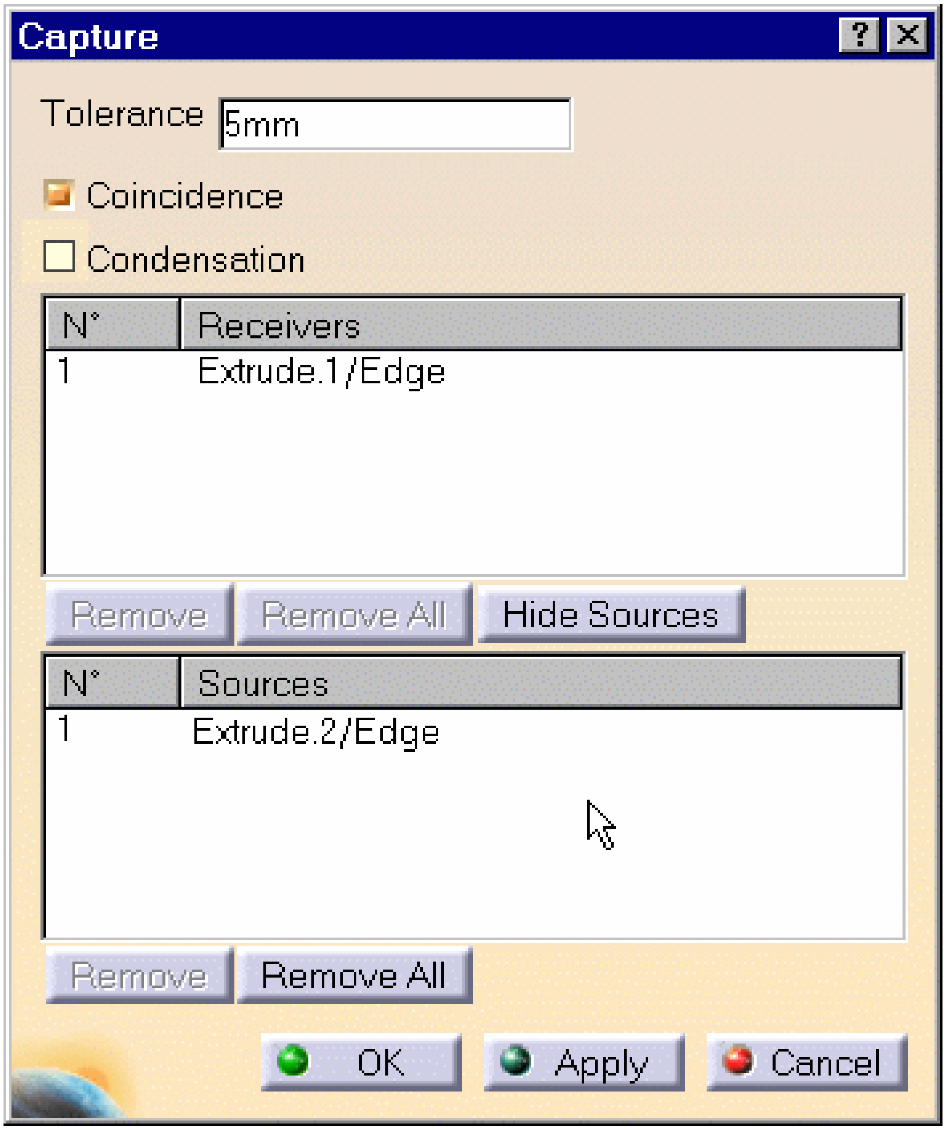


You might also decide that you want to explicitly select the source edge. For this:

3. Click the **Show Sources** switch button in the Capture dialog box.

4. Click in the **Sources** field with the mouse and then select the desired edge on the model.

The Capture dialog box now appears as shown here:



The nodes from the selected source edge now appear on the receiver edge.

5. Click **OK** in the Capture dialog box

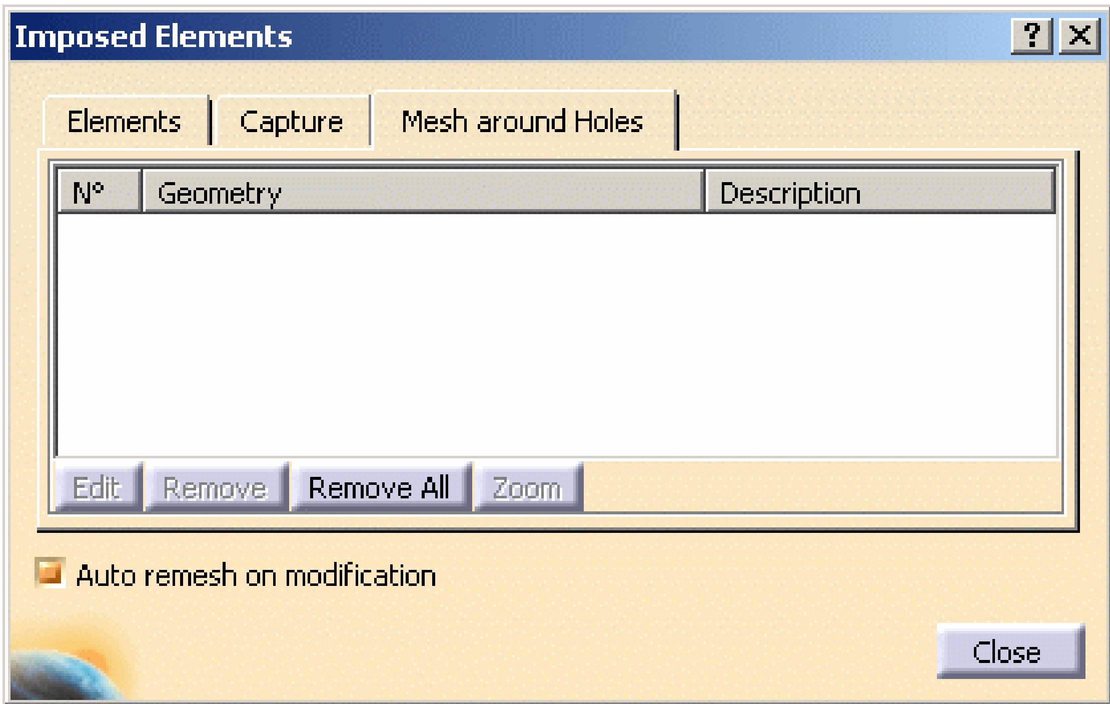
6. Click **OK** in the Imposed Elements dialog box once you are satisfied with you operations.

Mesh around Holes

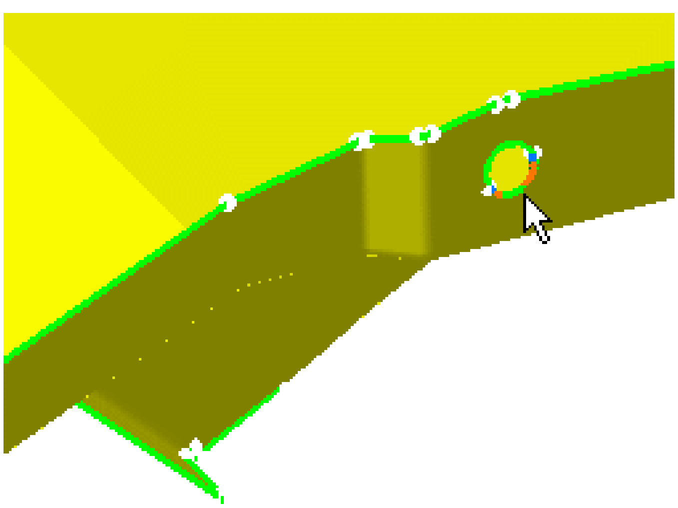
- Open the sample07.CATAnalysis document from the samples directory.
- Enter the **Surface Meshing** workshop.

  For more details, please refer to Entering the Surface Meshing Workshop.

1. Select the **Mesh around Holes** tab and the hole you want to be imposed nodes.

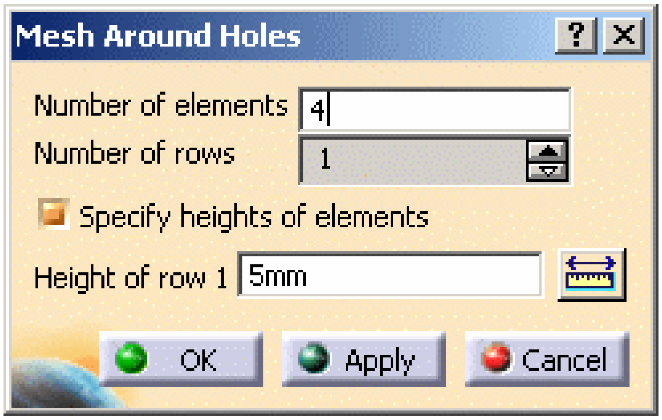


Note that whatever the number of curves making out the holes, multi-selection is automatic. you do not need to select each curve making out the hole.



The Mesh Around Holes dialog box now appears.

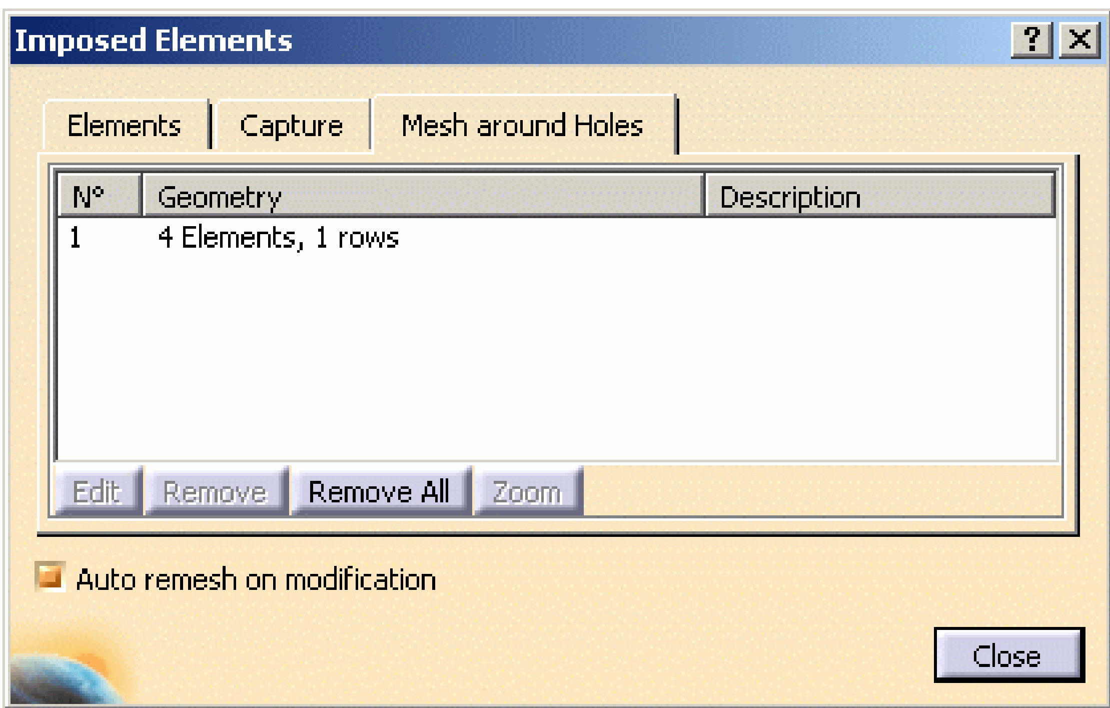
2. Enter the desired number of elements in the Mesh Around Holes dialog box, if needed.



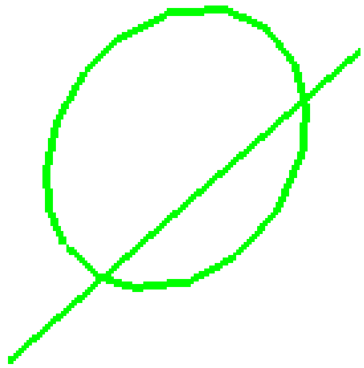
Note that you can specify the height of the row, if needed.

3. Click **OK** in the Mesh Around Holes dialog box.

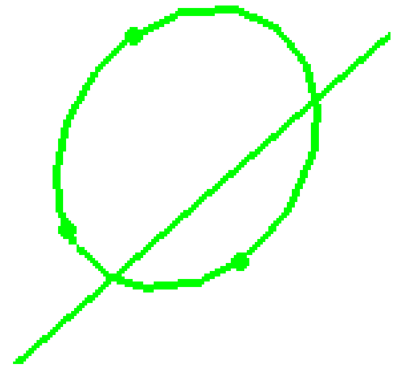
Both the Imposed Elements dialog box and the model are updated.



Hole and Nodes Before



Hole and Nodes After:



4. Click **OK** in the Imposed Elements dialog box once you are satisfied with you operations.

Specifying a Domain

This task shows you how to impose a domain for the geometry simplification step.

- Open the sample08.CATAnalysis document from the samples directory.

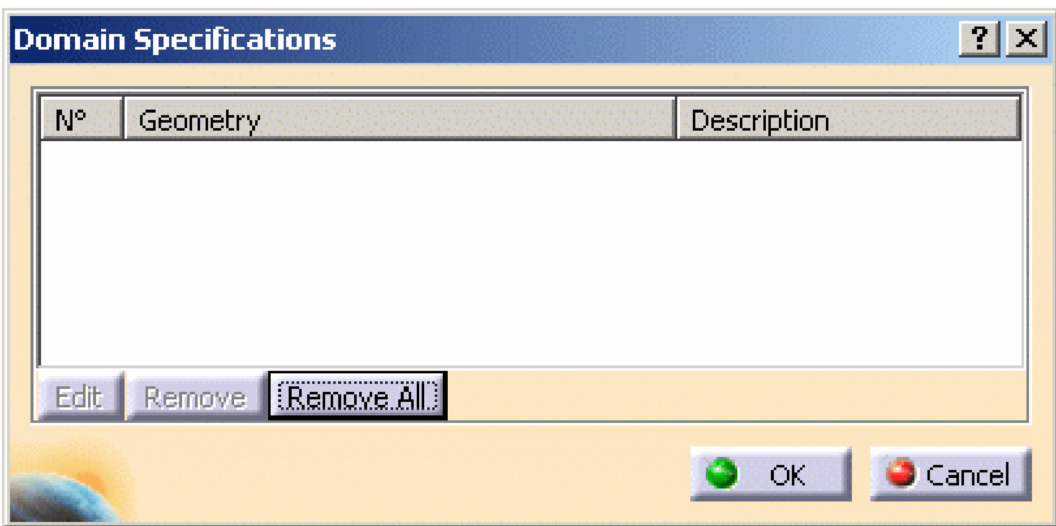
- Enter the **Surface Meshing** workshop.

  For more details, please refer to Entering the Surface Meshing Workshop.

In this particular example, choose the **Frontal Quadrangles** method and enter **10mm** as **Mesh size** value.

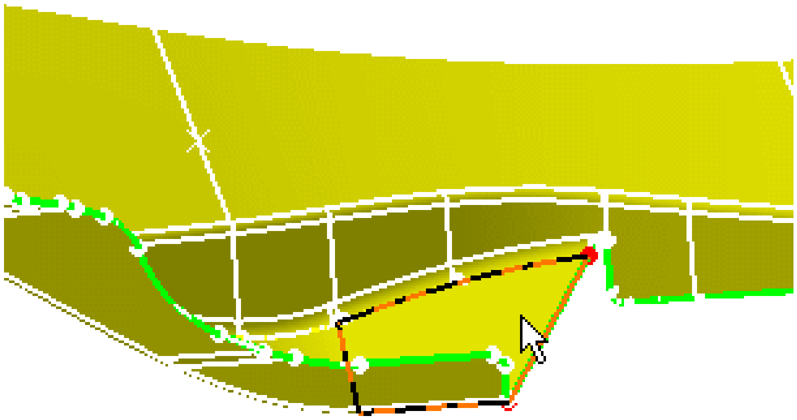
1. Click the **Domain Specifications** icon from the **Local Specifications** toolbar.

   The Domain Specifications dialog box appears.

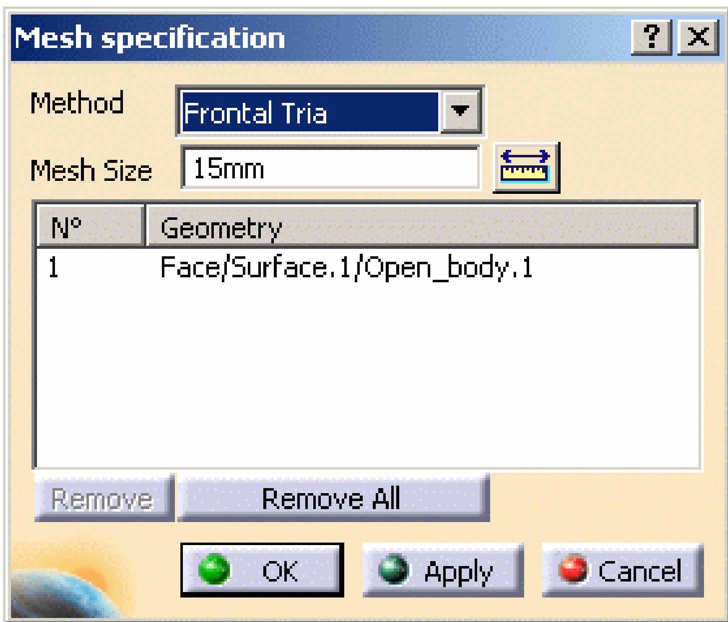


   - **Edit:** lets you modify a domain specification.
   - **Remove:** lets you remove a domain specification.
   - **Remove all:** lets you remove all the domain specifications.

2. Select a domain as shown below.



The Mesh Specification dialog box appears.

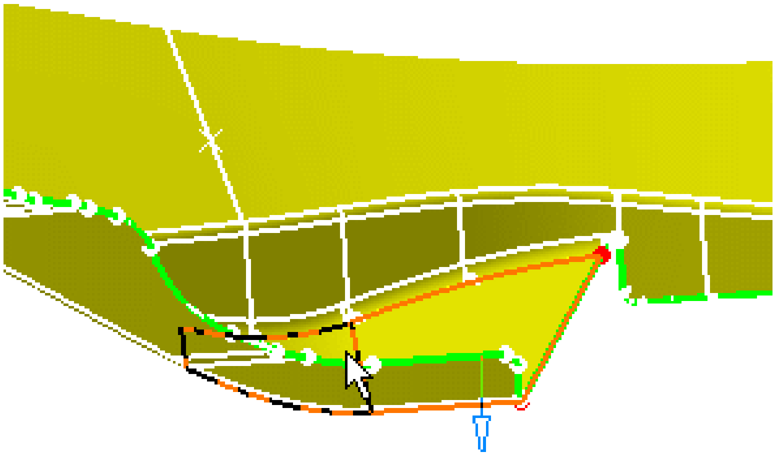


- **Method:** lets you specify the meshing method you want for this domain
- **Mesh Size:** lets you specify the mesh size value for this domain
- **Remove:** lets you remove an element of the selection
- **Remove All:** lets you remove all the elements of the selection.

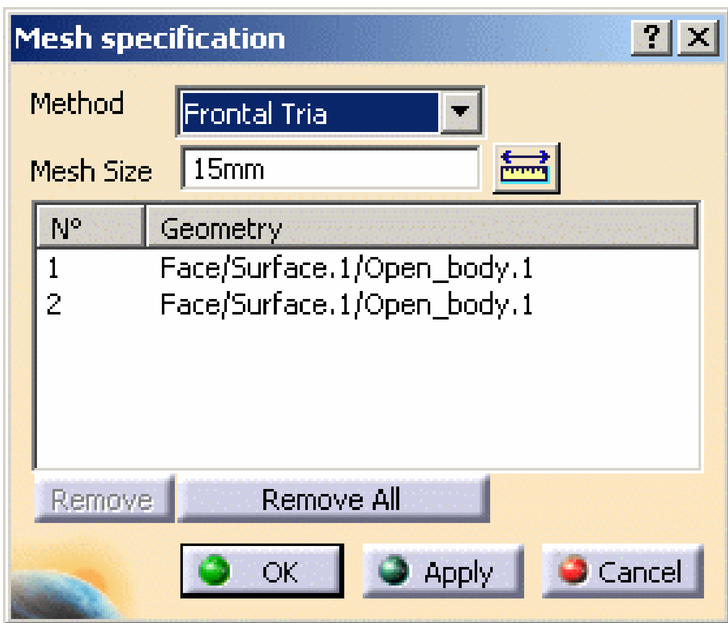
In this particular case:

- select **Frontal Tria** as **Method** option
- enter **15mm** as **Mesh Size** value.

3. Select a continuous domain as shown below.



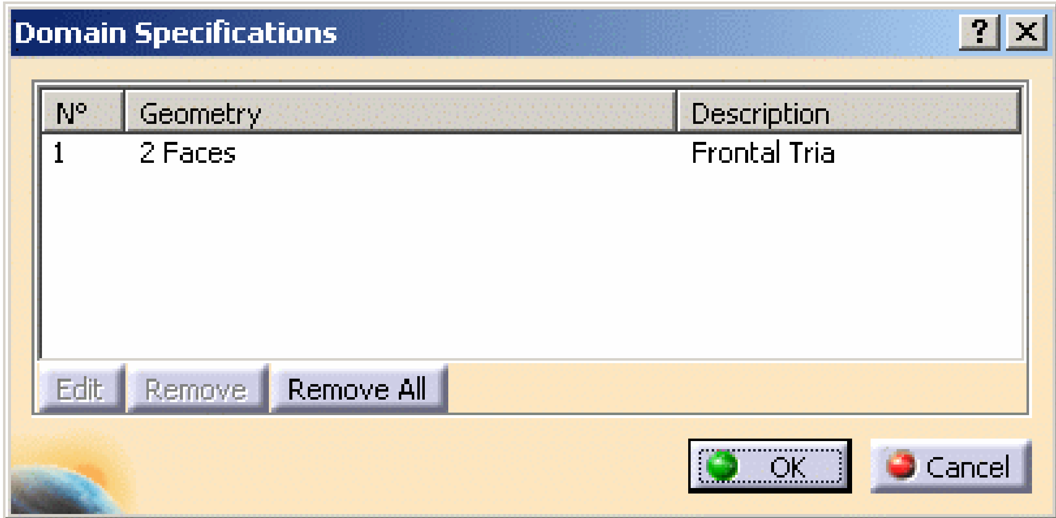
The Mesh Specification dialog box is updated.



Note that you cannot define a discontinuous domain.

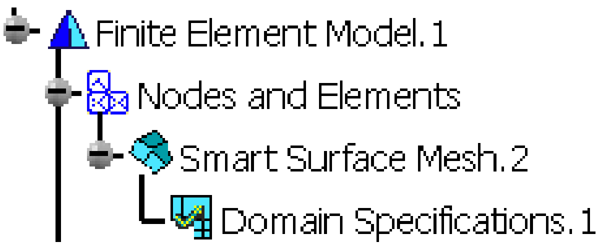
4. Click **OK** in the Mesh Specification dialog box.

The Domain Specifications dialog box is updated as shown below:



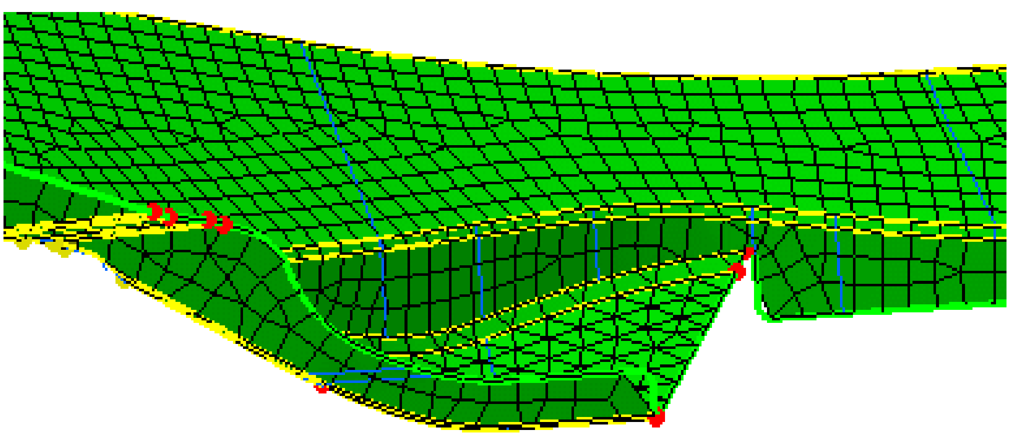
5. Click **OK** in the Domain Specifications dialog box.

A **Domain Specifications.1** object appears in the specification tree:



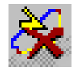
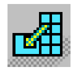
6. Click the **Mesh the Part** icon  from the **Execution** toolbar.

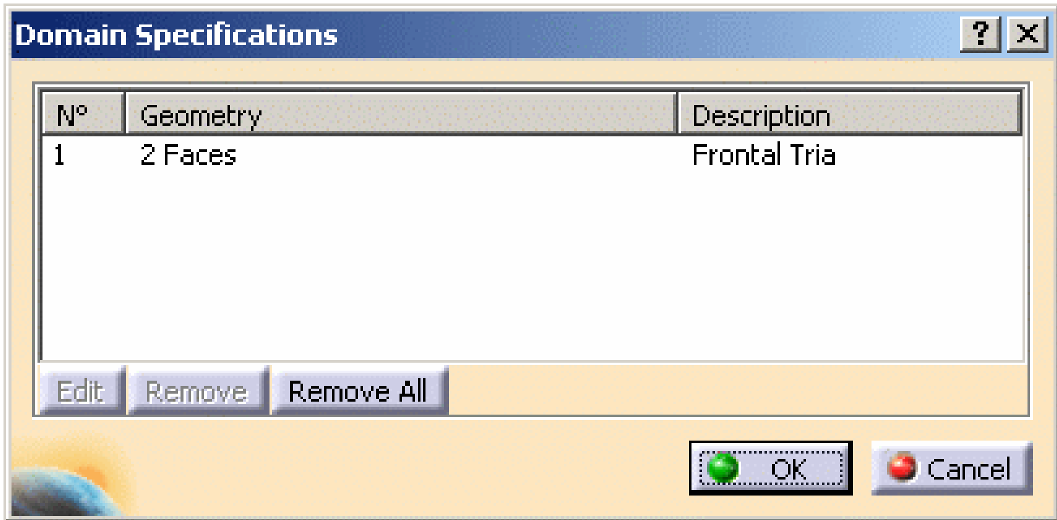
The part is meshed as shown below:



7. Edit the domain specification.

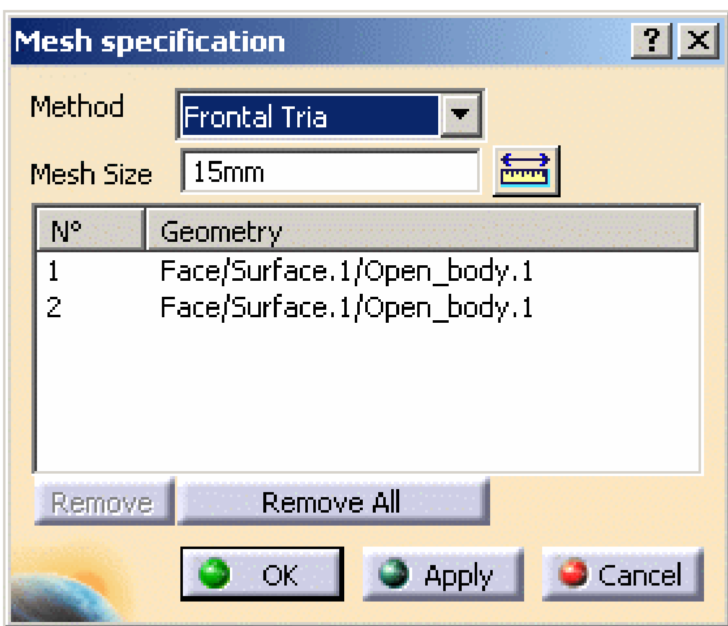
For this:

- click the **Remove Simplification** icon  from the **Execution** toolbar to remove the geometrical simplification
- click the **Domain Specifications** icon  from the **Local Specifications** toolbar.
  The Domain Specifications dialog box appears.



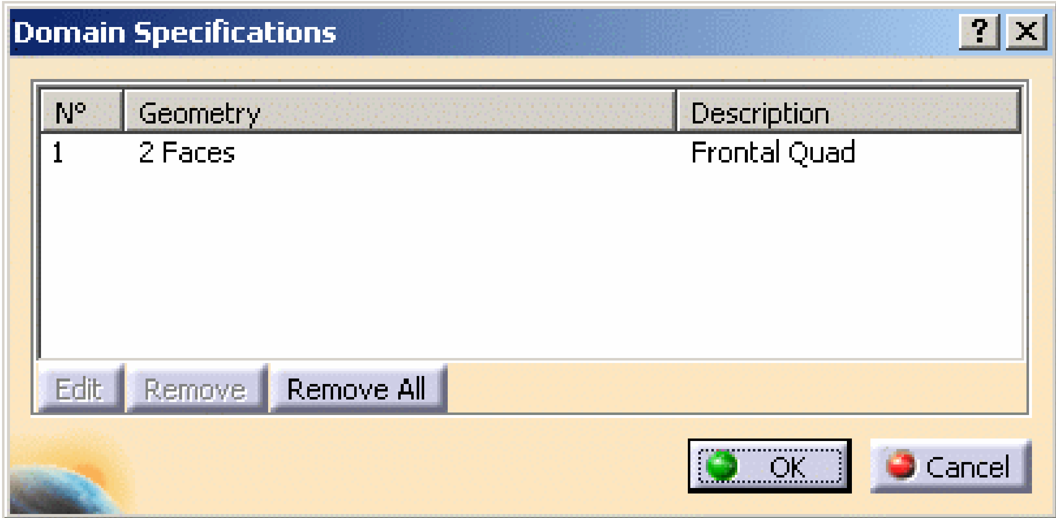
8. Click the Edit button from the Domain Specifications dialog box.

The Mesh Specification dialog box appears.



9. Select **Frontal Quad** as **Method** option from the Mesh Specification dialog box and click **OK**.

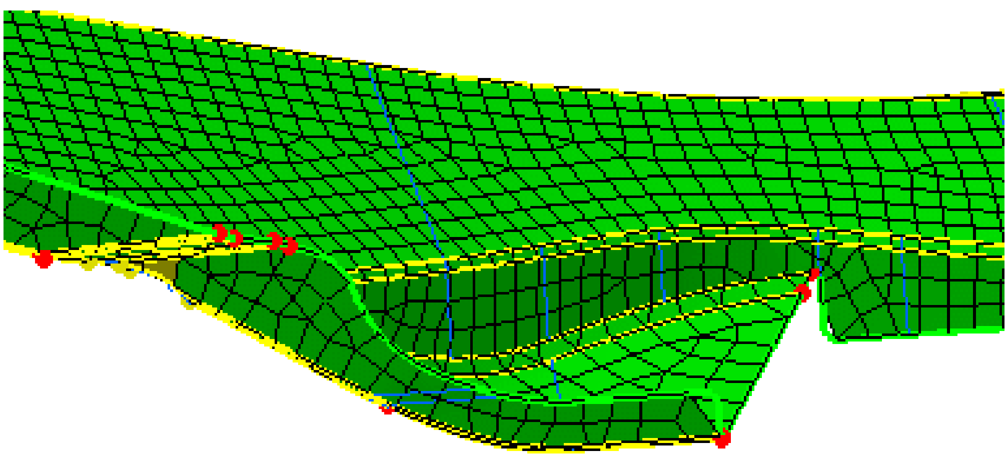
The Domain Specifications dialog box appears and is updated as shown here:



10. Click **OK** in the Domain Specifications dialog box.

11. Click the **Mesh the Part** icon  from the **Execution** toolbar.

The mesh part is updated as shown below:



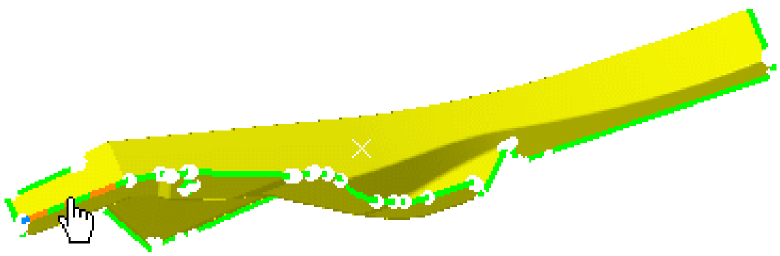
Modifying Local Specifications with Knowledgeware

This task shows you how to modify local specifications using the knowledge formula functionality.

- Open the sample07.CATAnalysis document from the samples directory.
- Enter the **Surface Meshing** workshop.

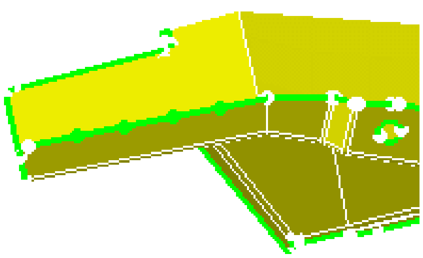
  For more details, please refer to Entering the Surface Meshing Workshop.

1. Click the **Imposed Elements** icon from the **Local Specifications** toolbar.
2. Select an edge as shown bellow:



3. Enter **5** as **Number of elements** in the Edit Elements Distribution dialog box.
4. Click **OK** in the Edit Elements Distribution dialog box and then in the Imposed Elements dialog box.

   The imposed nodes are displayed:



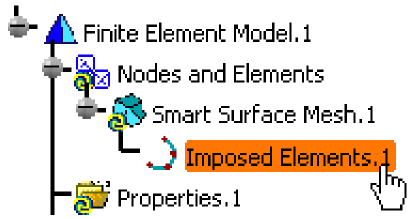
5. Click the **Exit** icon .
6. Click the **Formula** icon from the **Knowledge** toolbar.

   The Formulas: Analysis Manager dialog box appears.

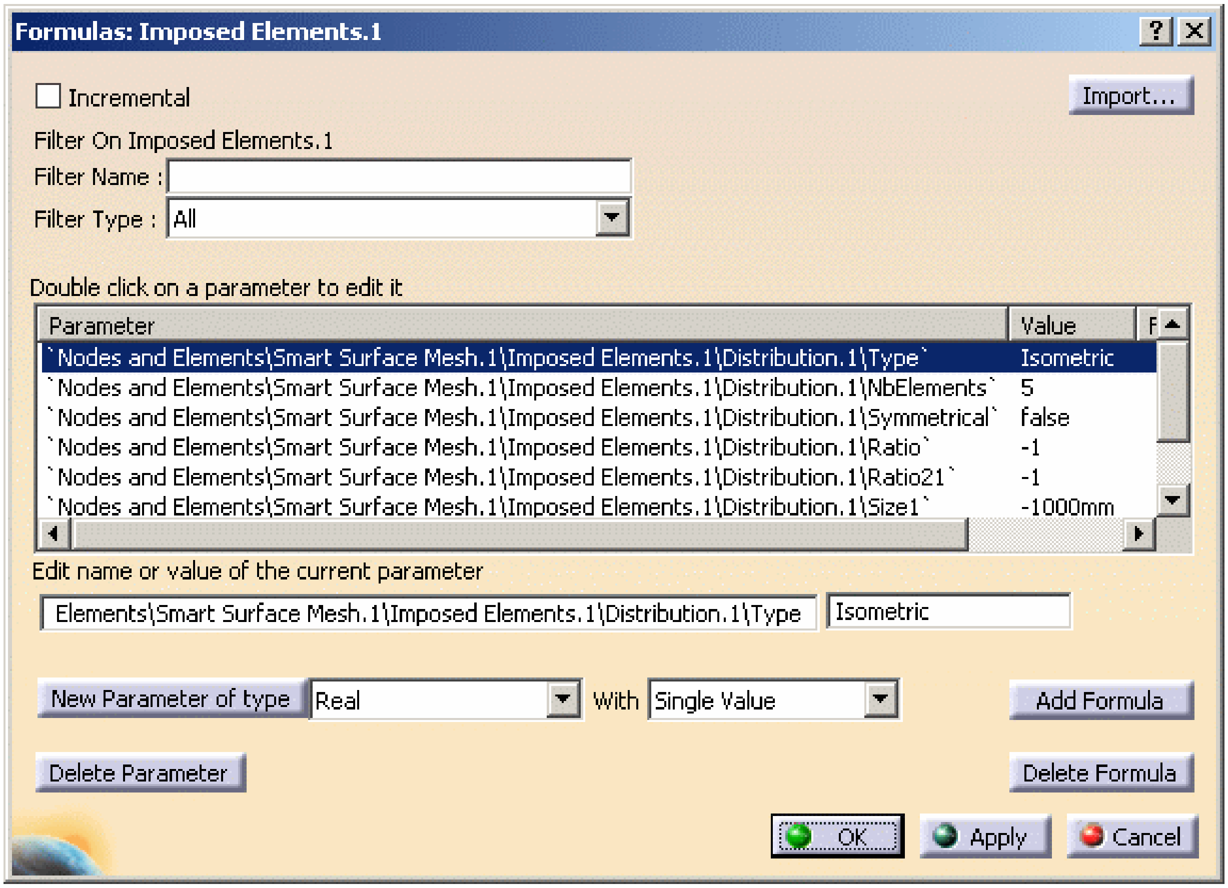
   For more details about the **Formula** functionality and the Formulas dialog box, please refer to Formulas in the *Infrastructure User's Guide*.

7. Select the local specification you want to modify.

In this particular example, select the **Imposed Elements.1** specification.

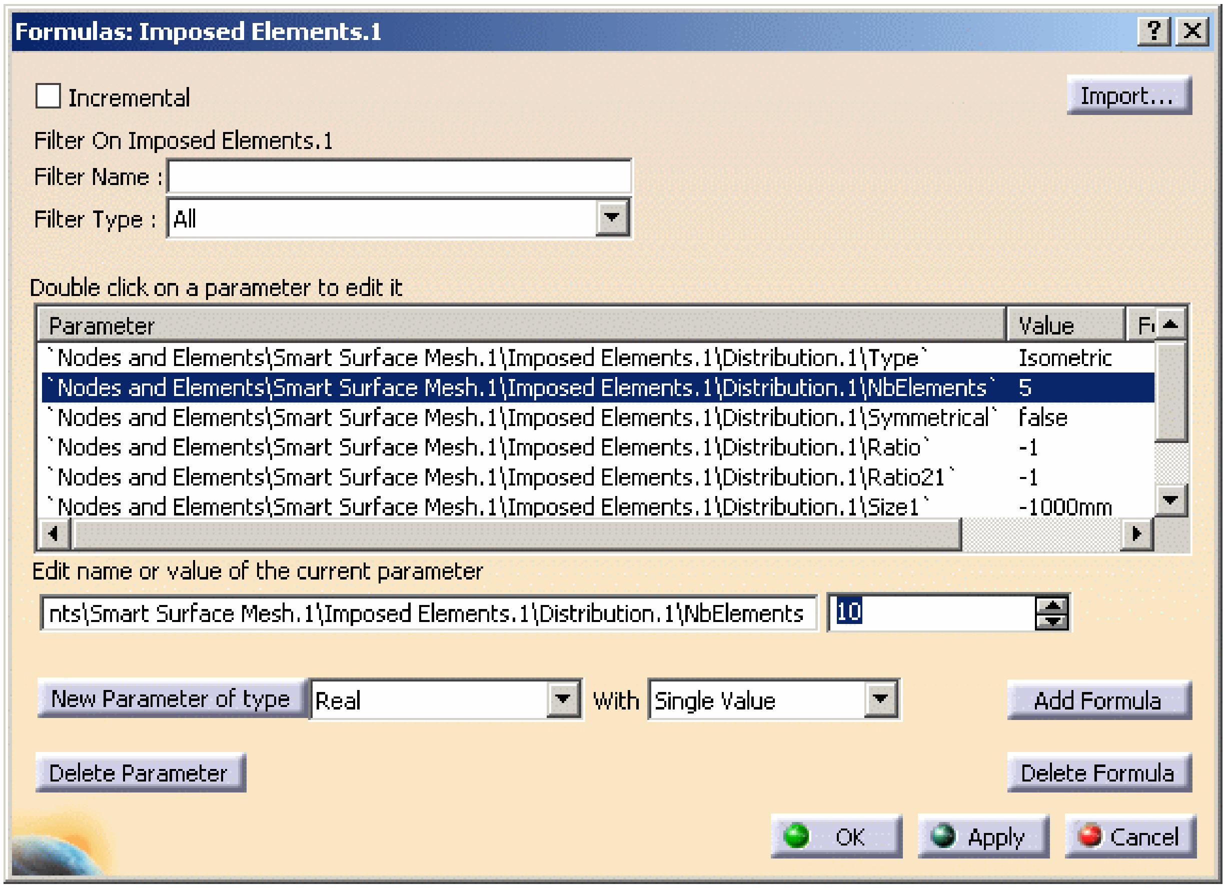


The Formulas dialog box is updated.



8. Select the second line.

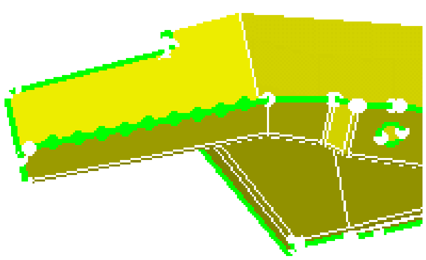
9. Enter **10** as new value as shown bellow:



10. Click **OK** in the Formulas: Imposed Elements.1 dialog box.

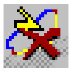
11. Enter again the **Surface Meshing** workshop.

Note that the number of element has been increased:

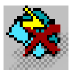


Execution

 Simplifying the Geometry (Specification): Improve the meshing.

 Removing the Geometrical Simplification: Remove the geometrical simplification you applied to geometry.

 Meshing the Part: Launch the surface mesher execution.

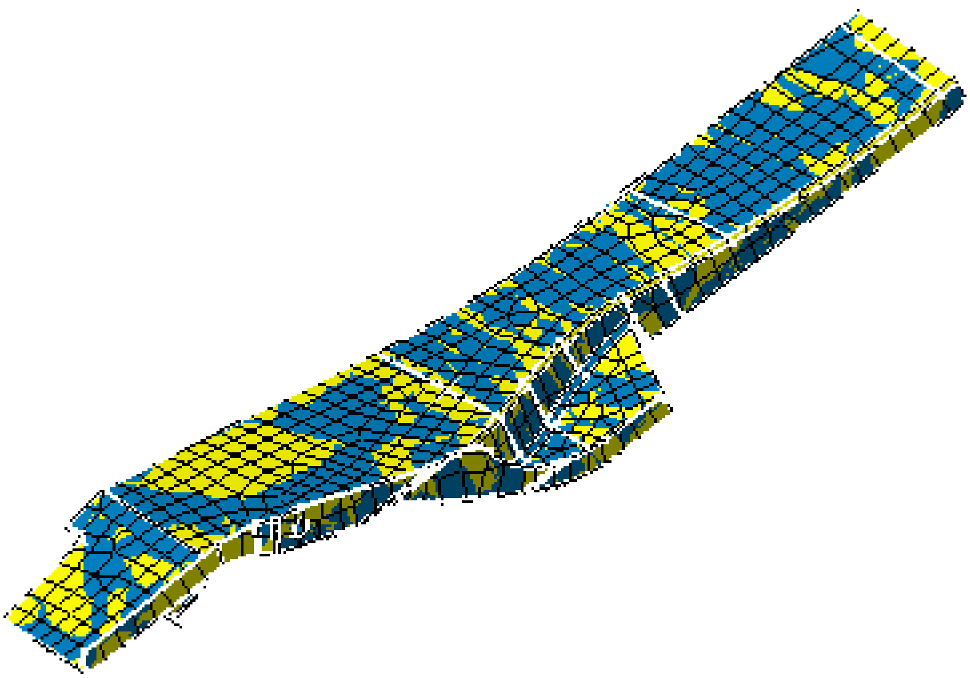
 Removing the Mesh: Remove the mesh you generated on the geometry.

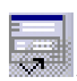
Simplifying the Geometry

This task shows how to simplify the geometry in order to improve the quality of the mesh.

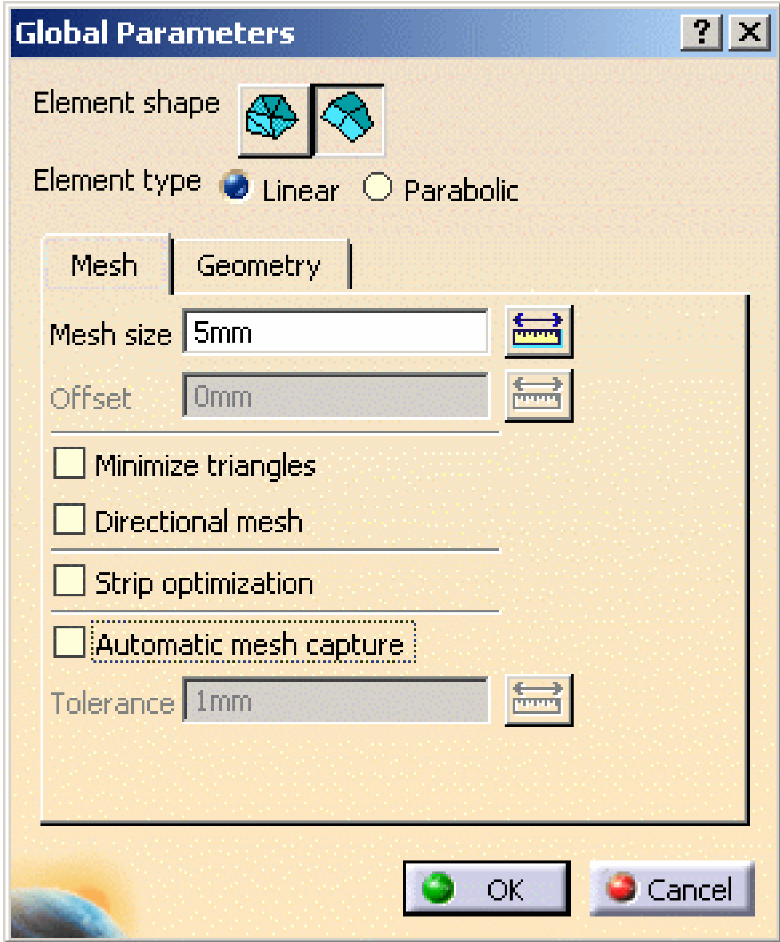
- Open the sample05.CATAnalysis document from the samples directory.
- Enter the **Surface Meshing** workshop.

  For more details, please refer to Entering the Surface Meshing Workshop.



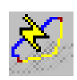
1. Click the **Global Meshing Parameters** icon .

   The Global Parameters dialog box appears.

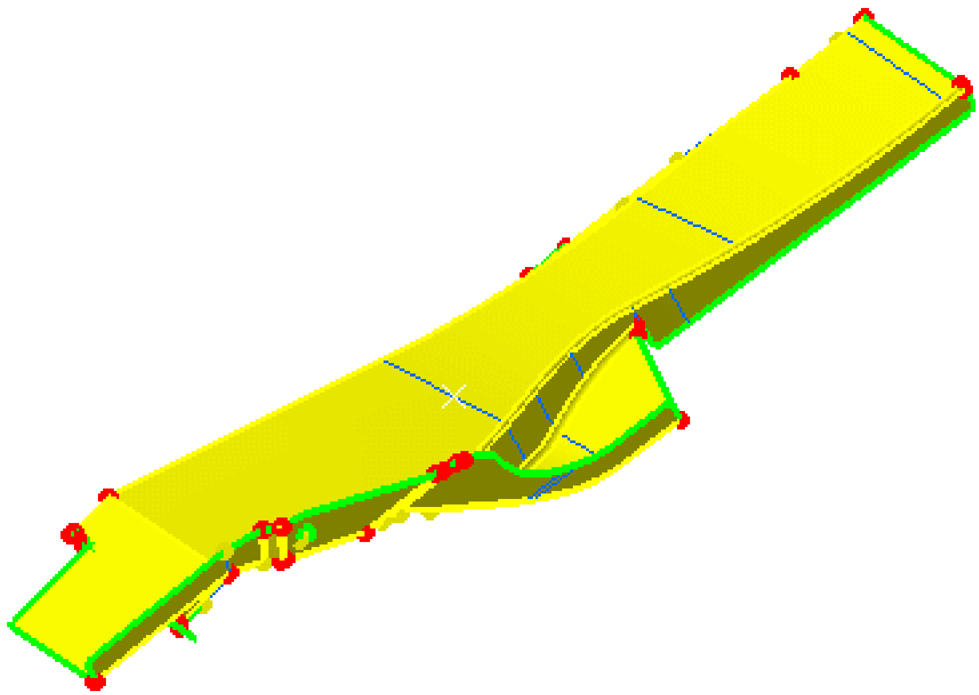


2. Select the desired parameters and click **OK** in the Global Parameters dialog box.

   For more details about this dialog box, please refer to Setting Global Meshing Parameters.

3. Click the **Geometry Simplification** icon  from the **Execution** toolbar.

   A set of optimized constraints is computed.



- Geometrical simplification is always done, before meshing, even if you do not launch it.
- determine if you have to modify manually the specifications before the meshing.

- Simplify the geometry to preview the domains.
- If needed, modify the specifications:
  - Remove the geometrical simplification.
    For more information, please refer to Removing the Geometrical Simplification.
  - Define the desired specifications.
    For more information, please refer to Local Specifications toolbar.

Removing the Geometrical Simplification

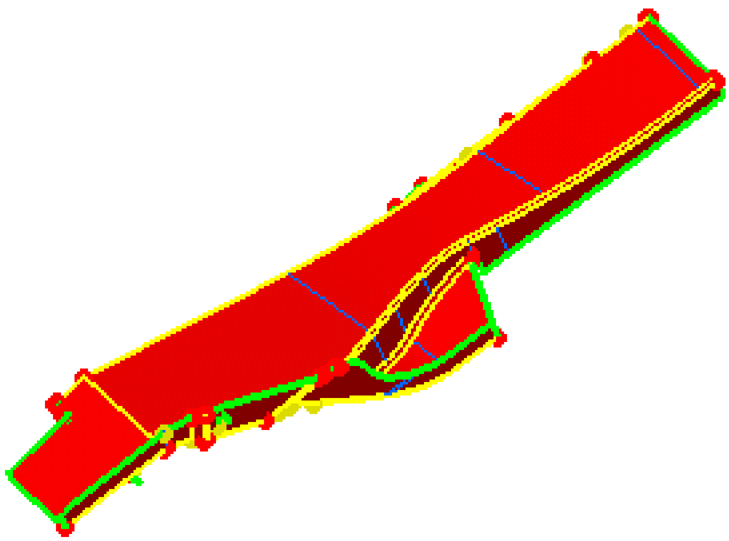
This task shows how to remove the geometrical simplification you applied to geometry.

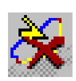
The mesh elements also are removed.

- Open the sample08.CATAnalysis document from the samples directory.
- Enter the **Surface Meshing** workshop.

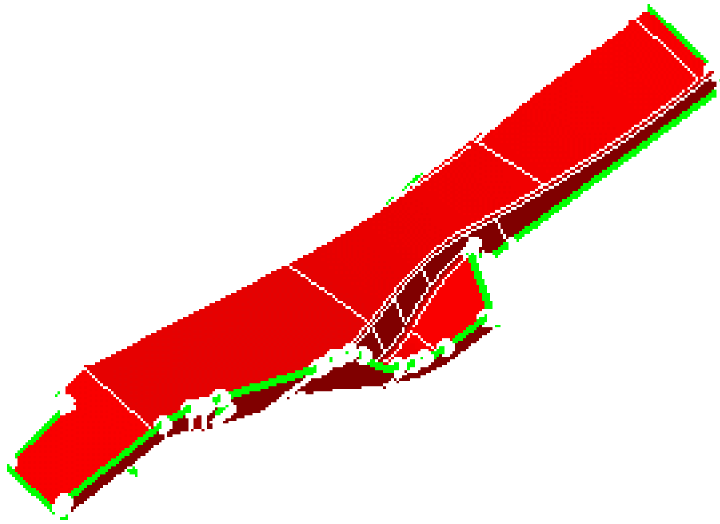
  For more details, please refer to Entering the Surface Meshing Workshop.
- Launch the geometrical simplification.

  For more details, please refer to Simplifying the Geometry.



1. Click the **Remove Simplification** icon  from the **Execution** toolbar.

The geometrical simplification is automatically removed after you confirmed the operation in a box that appears.



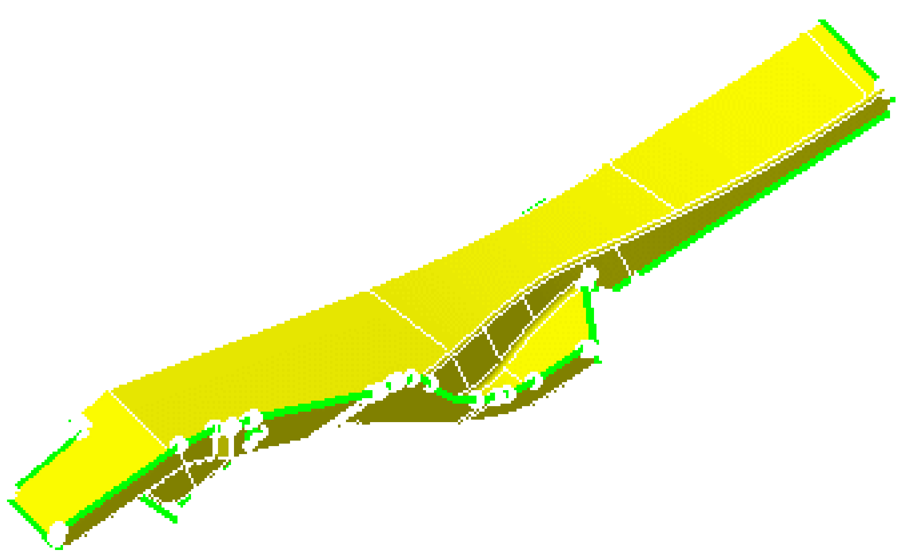
Meshing the Part

This task demonstrates how to launch the mesh of a surface part.

FEM Surface lets you automatically handle the mesh of complex geometries with advanced control on specifications. The meshes are fully associative with design changes. You can define specifications in order to simplify the geometry. However, the referenced geometry is never modified in the whole meshing process. The mesh deals with the exact replica of the geometry as a clone: it respects all the geometry characteristics, and adapt those to the mesh needs, without impacting the original geometry design. You get an automatic simplified interpretation of the meshed geometry. For example, you can mesh over holes and gaps. In the same way, it is possible to eliminate small faces such as stiffeners or flanges for meshing. You can achieve an accurate and smart elimination of details such as fillets. Different types o f finite elements and meshing methods are available: quadrangles or triangles, advancing-front or mapped meshers.

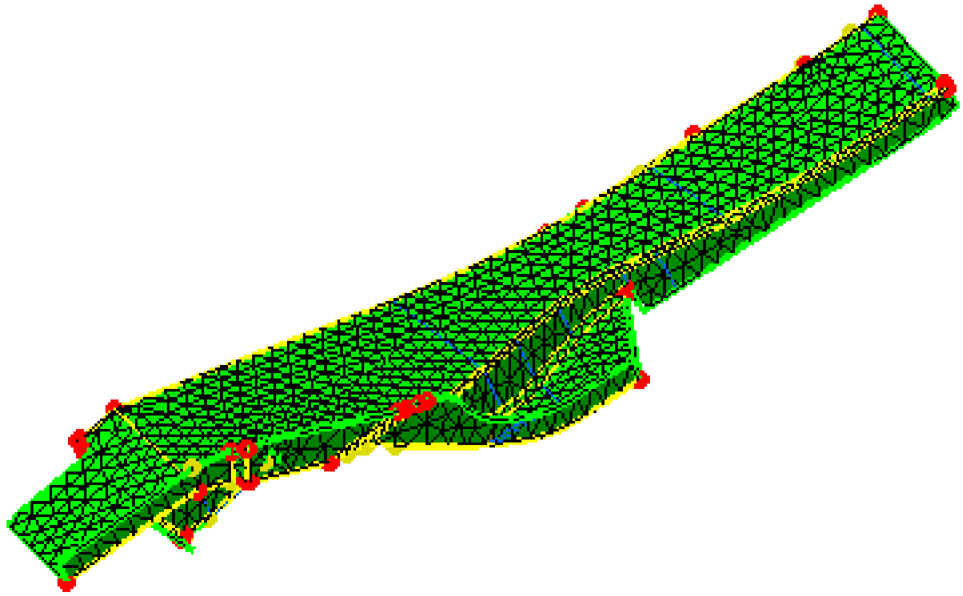
- Open the sample08.CATAnalysis document from the samples directory.
- Enter the **Surface Meshing** workshop.
  For more details, please refer to Entering the Surface Meshing Workshop.

Constraints and nodes appear.

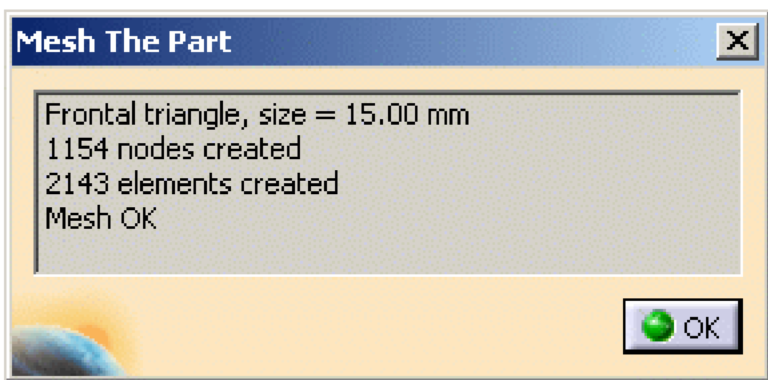


1. Click the **Mesh The Part** icon from the **Execution** toolbar.

The mesh is generated on the part.



A little summary is provided in the Mesh The Part dialog box.



2. Click **OK** in the Mesh The Part dialog box.

Any modification you will then manually apply to the meshed surface will not be saved for example if you edit the mesh. In other words, each time you will apply manual modifications (Adding/Removing Constraints (Modifications), Imposing Nodes (Modifications), Re-meshing Domains, Mesh Editing), these modifications will only be saved on the condition you launch the mesh operation again.

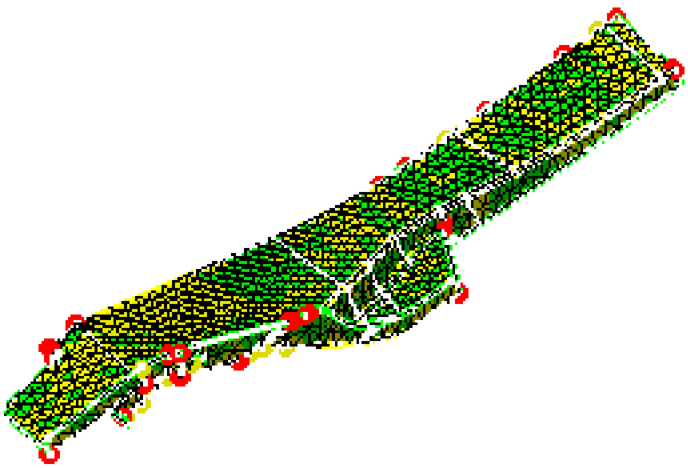
Removing the Mesh

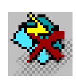
This task shows how to remove the mesh you generated on the geometry.

- Open the sample08.CATAnalysis document from the samples directory.
- Enter the **Surface Meshing** workshop.

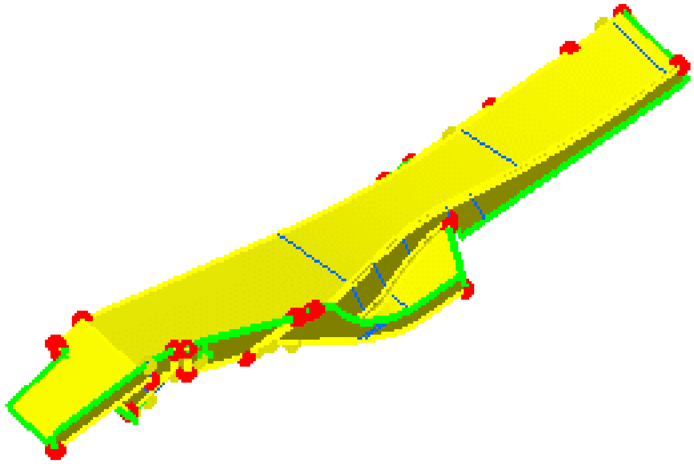
  For more details, please refer to Entering the Surface Meshing Workshop.
- Mesh the surface.

  For more details, please refer to Meshing the Part.



1. Click the **Remove Mesh** icon  from the **Execution** toolbar.

   The mesh is automatically removed.



Edition Tools

Cleaning Holes: Ignore holes in the mesh.

Adding/Removing Constraints (Modifications): Add or remove two types of constraints: constraints applied to vertices ; constraints applied to curves.

Imposing Nodes (Modifications): Distribute nodes on edges.

Domain Edition

Re-meshing a Domain: Re-mesh a domain using new parameters.

Removing the Mesh by Domain: Remove the mesh you generated on a domain of the geometry.

Locking a Domain: Lock a domain by selecting it.

Modify Mesh

Mesh Editing: Edit a mesh to provide higher quality.

Splitting Quadrangles: Split quadrangle elements in order to improve the mesh quality.

Cleaning Holes

This task shows you how to specify manually which hole you want to ignore during the mesh process.

- Open the Sample03.CATAnalysis document from the samples directory.

- Enter the **Surface Meshing** workshop.

  For more details, please refer to Entering the Surface Meshing Workshop.

- Mesh the surface.

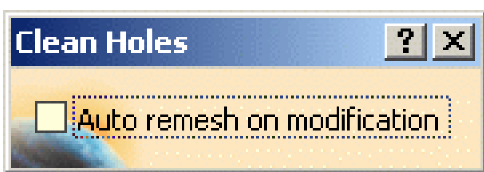
  For more details, please refer to Meshing the Part.

- Hide the geometry for a better visualization of the mesh.

  For this, right-click the **Part.1** in the specification tree and select the **Hide / Show** contextual menu.

1. Click the **Clean Holes** icon from the **Edition Tools** toolbar.

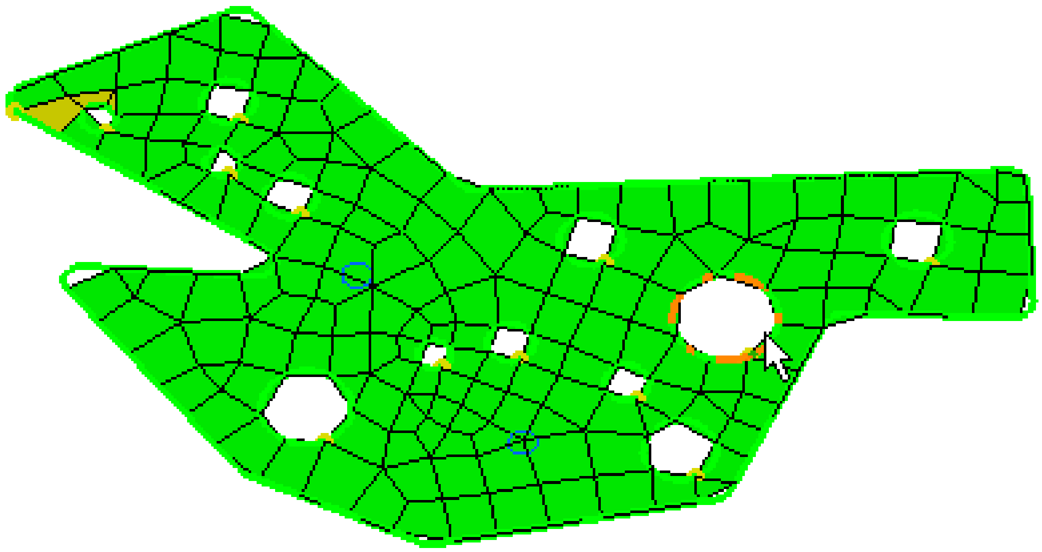
   The Boundary Simplifications dialog box appears:

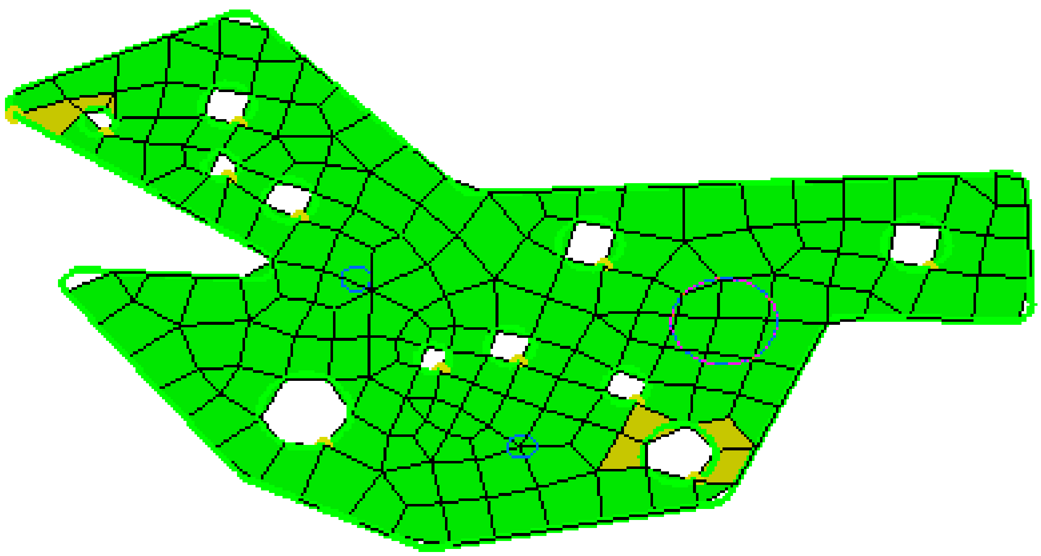
   - **Auto remesh on modification**:
     - if you activate this option, an automatic re-mesh is performed on the domain.
     - if you deactivate this option, the mesh will be removed on the domain.

2. Select the **Auto remesh on modification** option in the Clean Holes dialog box.

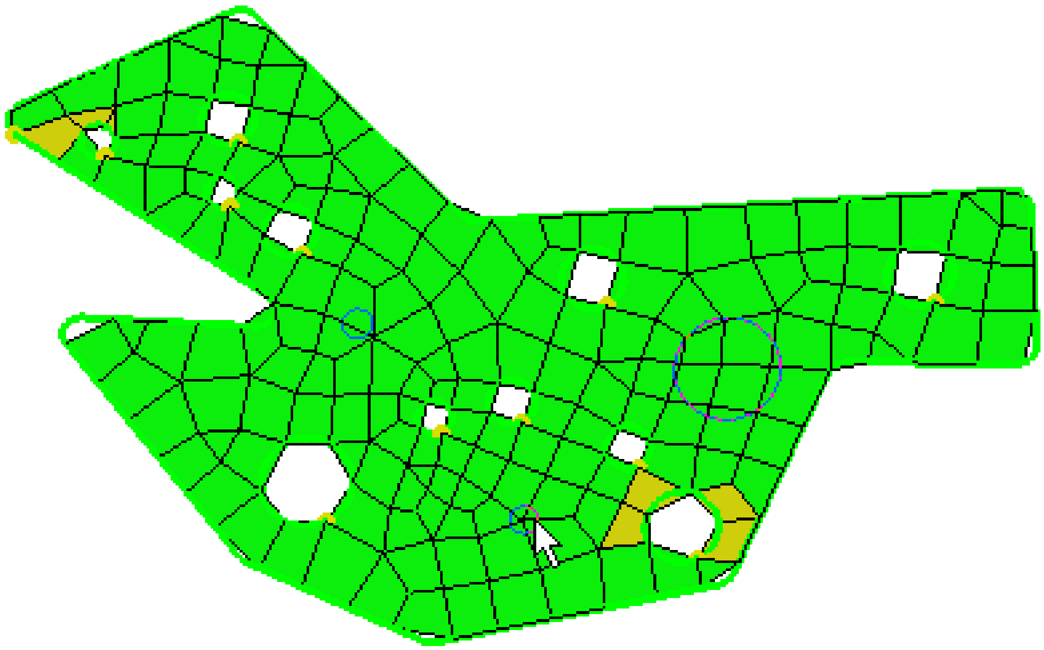
3. Select a hole as shown bellow.



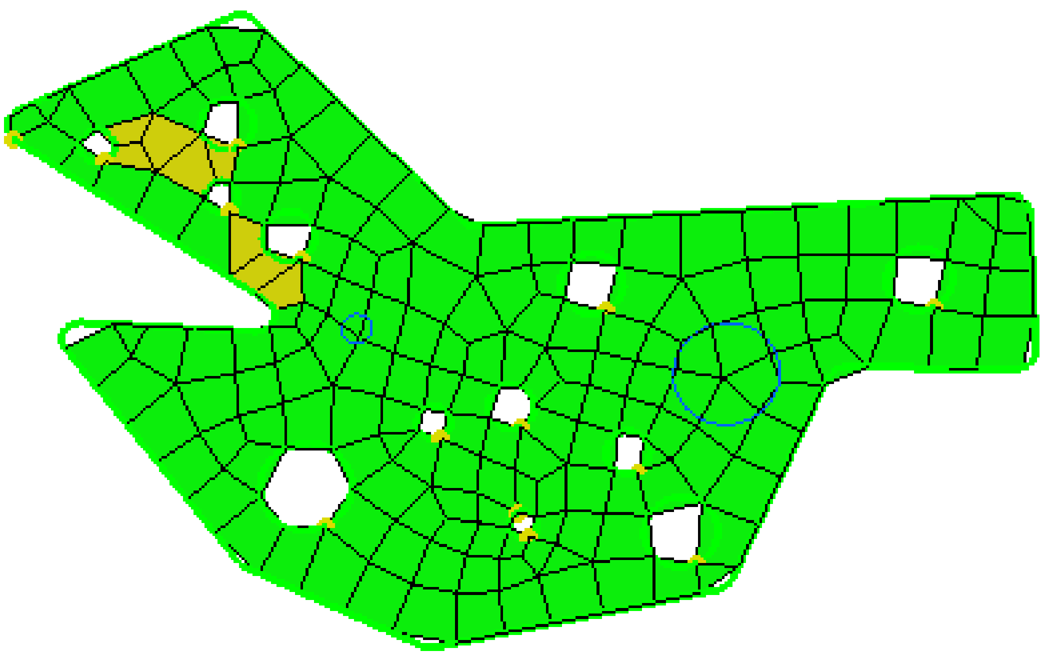
As a result, an automatic re-mesh is performed around the selected hole:



4. Select a hole as shown bellow.

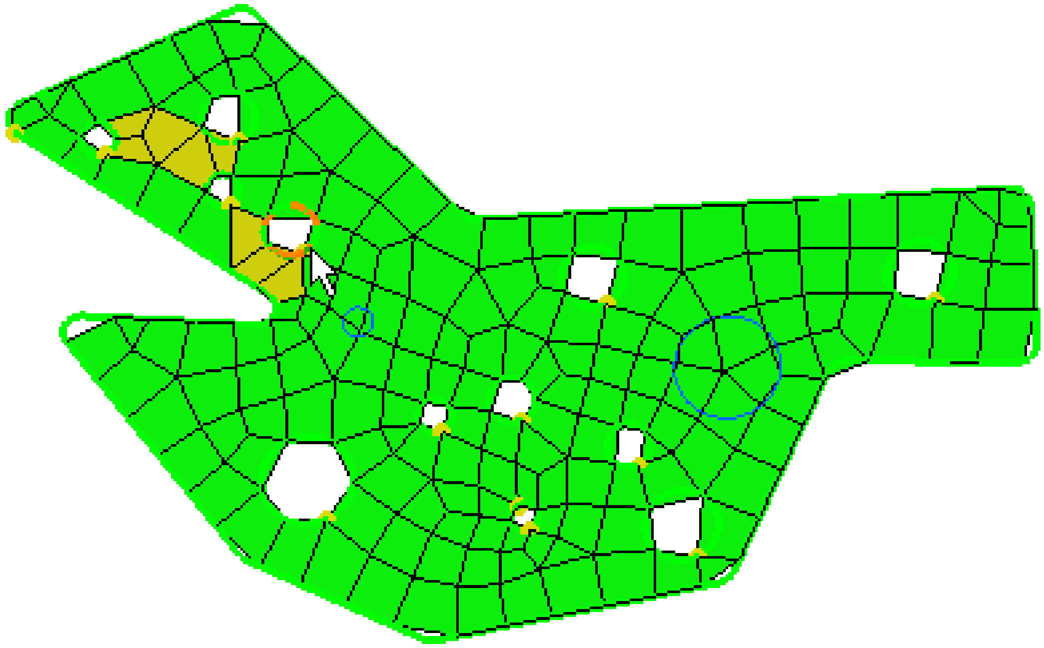


As a result, an automatic re-mesh is performed around the selected hole:

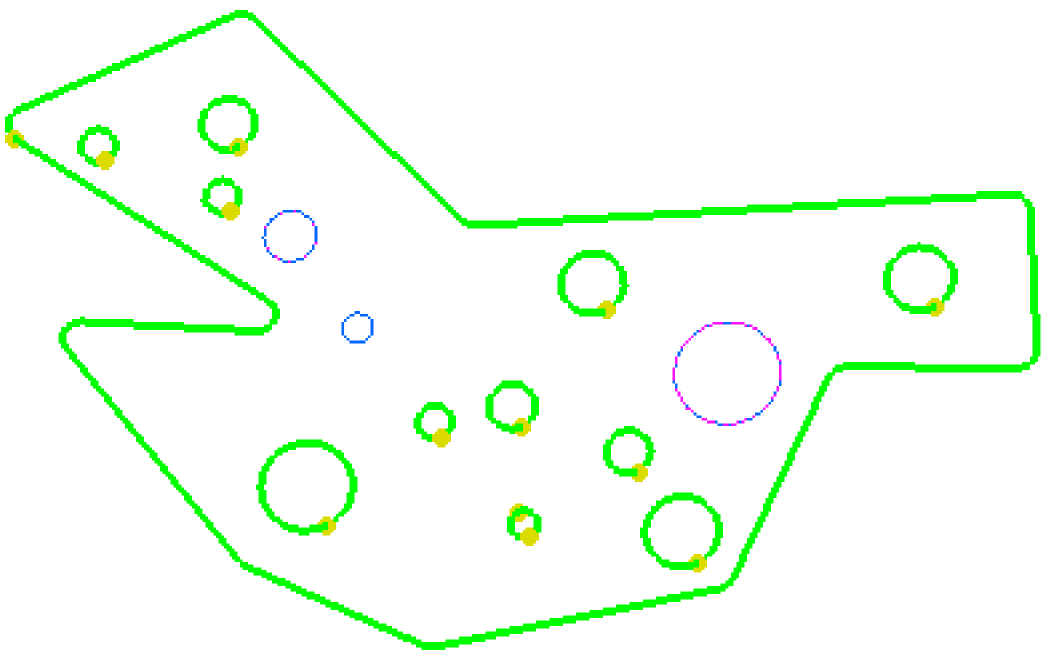


5. Select the **Auto remesh on modification** option in the Clean Holes dialog box.

6. Select a hole as shown bellow.

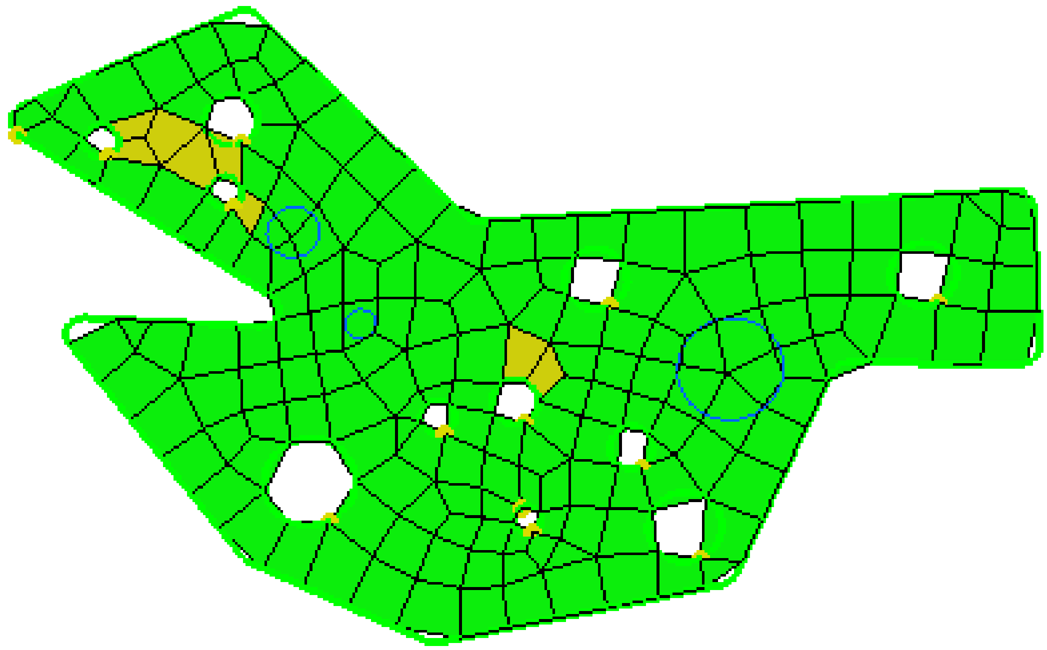


As a result, the mesh is removed on the domain:



7. Click the **Mesh the Part** icon  from the **Execution** toolbar.

As a result the meshing process is launched and all the modifications you have previously performed are taken into account.



8. Click **OK** in the Mesh The Part dialog box.

9. Click again the **Clean Holes** icon  from the **Edition Tools** toolbar.

Adding/Removing Constraints (Modifications)

This task shows how to apply manual topological modifications on existing constraints.

There are two types of constraints.

1. a constraint applied to a vertex: as a result, a node will be created on this vertex.
2. a constraint applied to a curve: as a result, all the element edges will be aligned on this curve.

Adding/Removing Constraints using:

- Swap constrained state option
- Split a domain option
- Collapse edge option
- Cut edge option
- Merge edges option
- Undo simplifications option
- Drag option

- Open the sample05.CATAnalysis document from the samples directory.
- Enter the **Surface Meshing** workshop.

  For more details, please refer to Entering the Surface Meshing Workshop.
- Launch the geometrical simplification.

  For more details, please refer to Simplifying the Geometry.

1. Click the **Edit Simplification** icon from the **Edition Tools** toolbar.

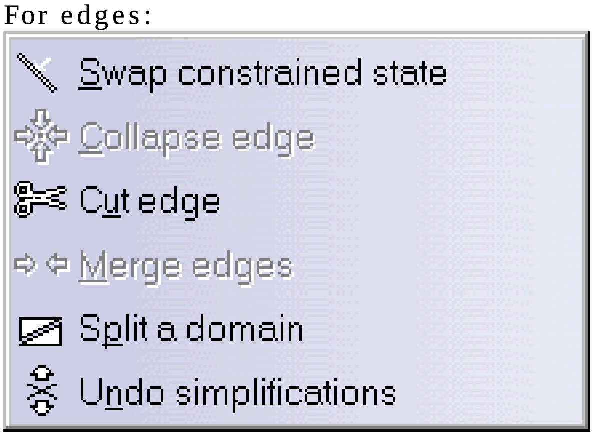
   The Edit Simplification dialog box appears with one option for re-meshing automatically as you modify the geometrical simplification.

The Undo command is not available.

2. Right-click the elements required for modifying edges, vertices or still for performing split, collapse or merge operations.

3. Select the desired options from the available contextual menu.

   For edges:

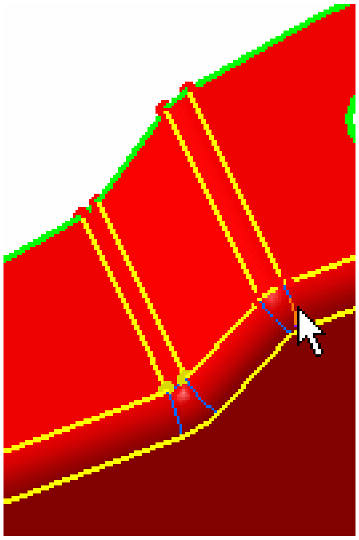


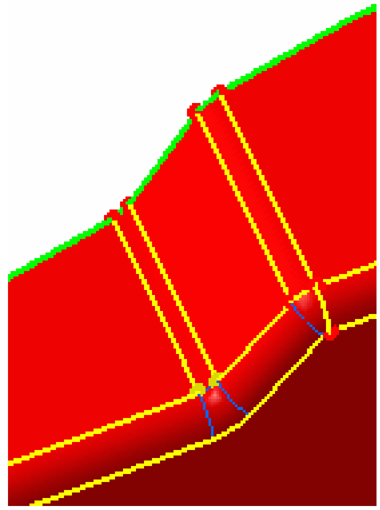
For vertices:

The desired option (contextual menu), once selected, remains active. You then simply need to select the elements to be modified.

Swap constrained state option

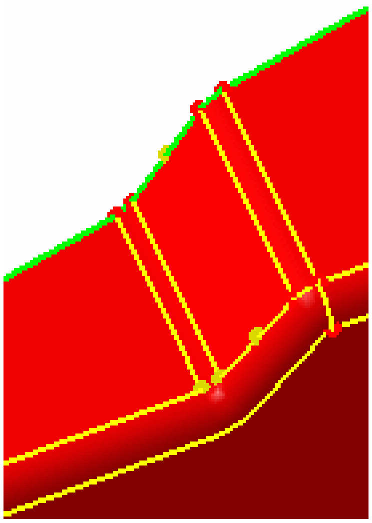
If you right-click on this part: *You get this constrained state edge:*

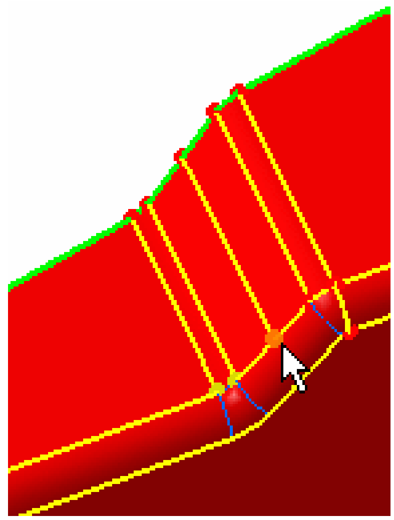




Split a domain option

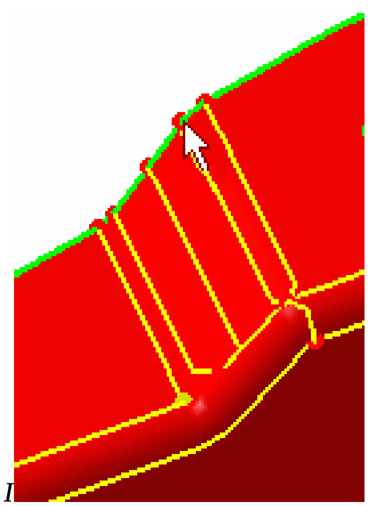
If you right-click on two existing vertices: *You get this split domain:*



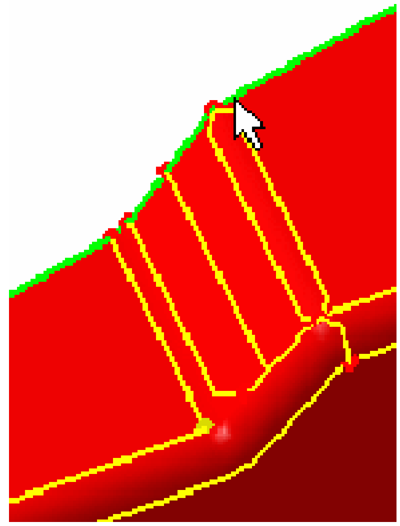


Collapse edge option

If you right-click on an edge: *You get this collapsed-edge element:*

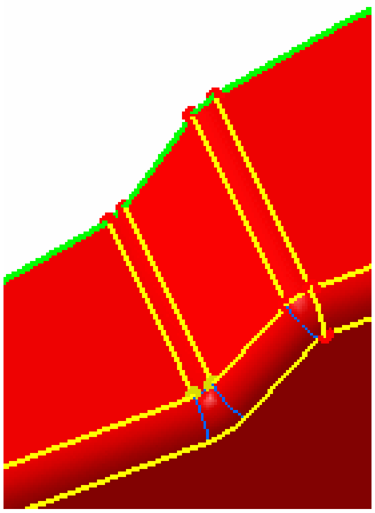


I



Cut edge option

If you right-click on an edge:



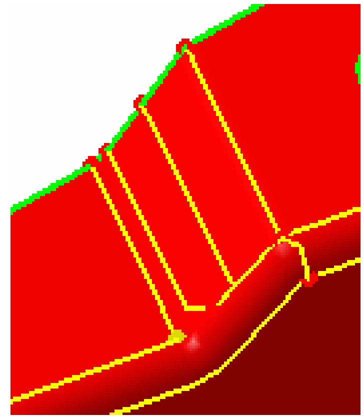
You get this cut edge (and new vertex):

Merge edges option

If you right-click on two collapsed edges:

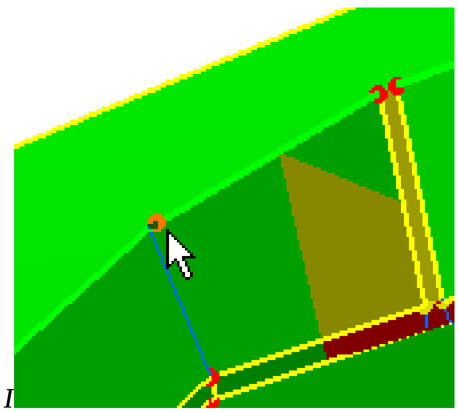
One after the other:

You get this merged edge:

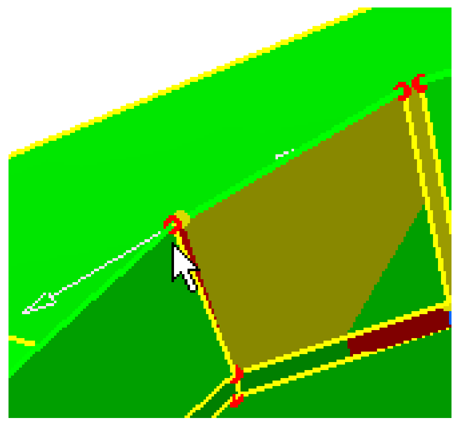


Drag option

If you right-click on a vertex on a constrained edge



You can move the vertex over the constrained edge:

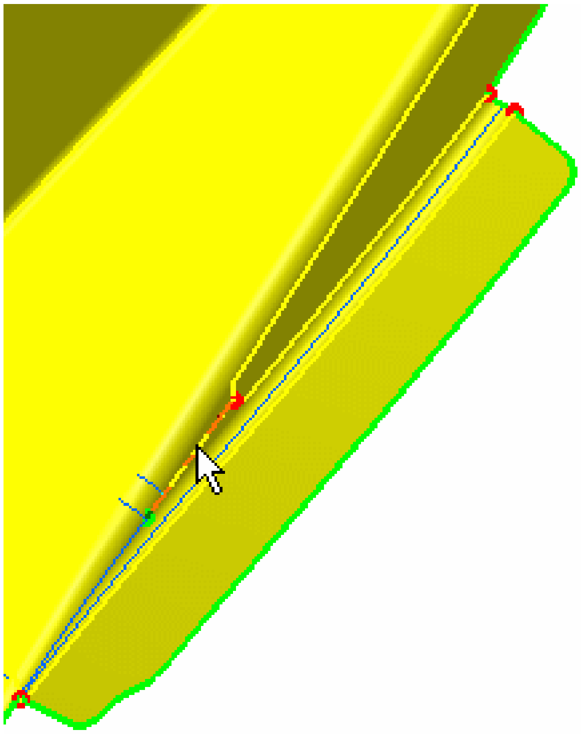


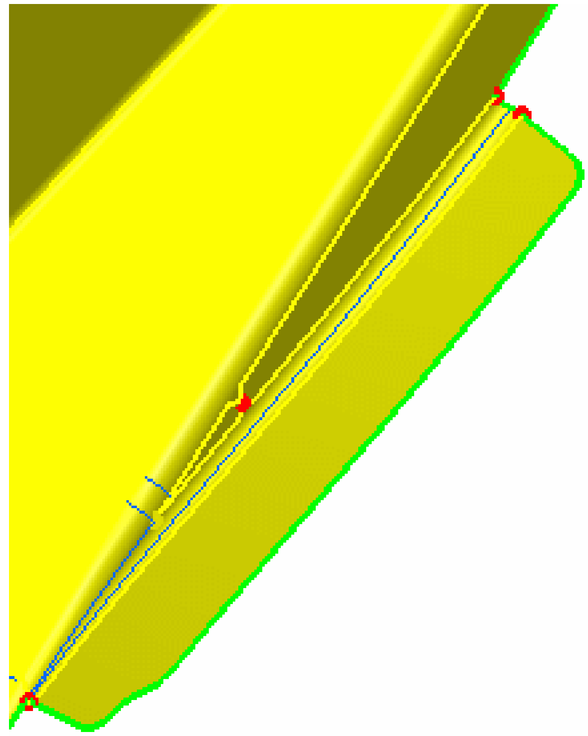
Undo simplifications option

This option can be applied either on simplifications generated by the system or on simplifications you applied manually (see above).

If you right-click on an edge:

You get this unmerged mesh element:





Imposing Nodes (Modifications)

This task shows how to impose node distribution on geometry.

- Open the sample08_1.CATAnalysis document from the samples directory.
- Enter the **Surface Meshing** workshop.

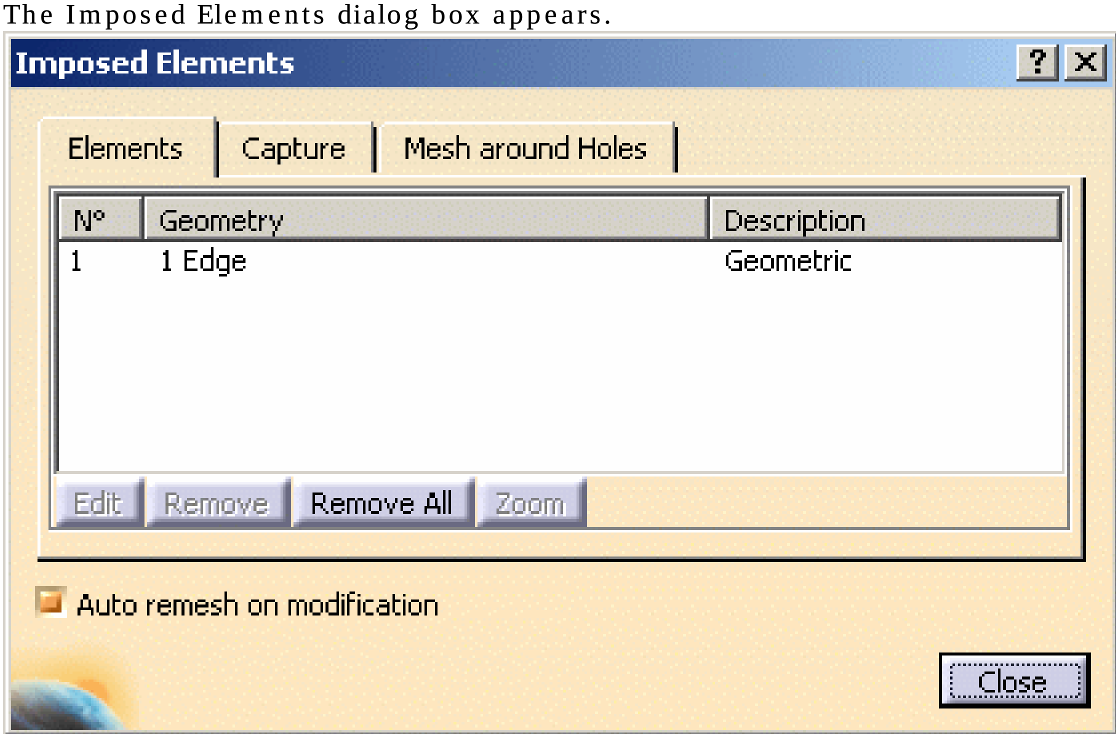
  For more details, please refer to Entering the Surface Meshing Workshop.
- Mesh the surface.

  For more details, please refer to Meshing the Part.
- Display the geometry in shading mode.

  For this, click the **Shading (SHD)** icon from the View toolbar.

1. Click the **Imposed Elements** icon from the **Edition Tools** toolbar.

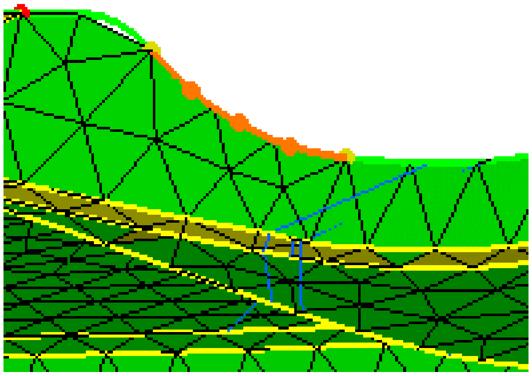
   The Imposed Elements dialog box appears.



The Undo command is not available.

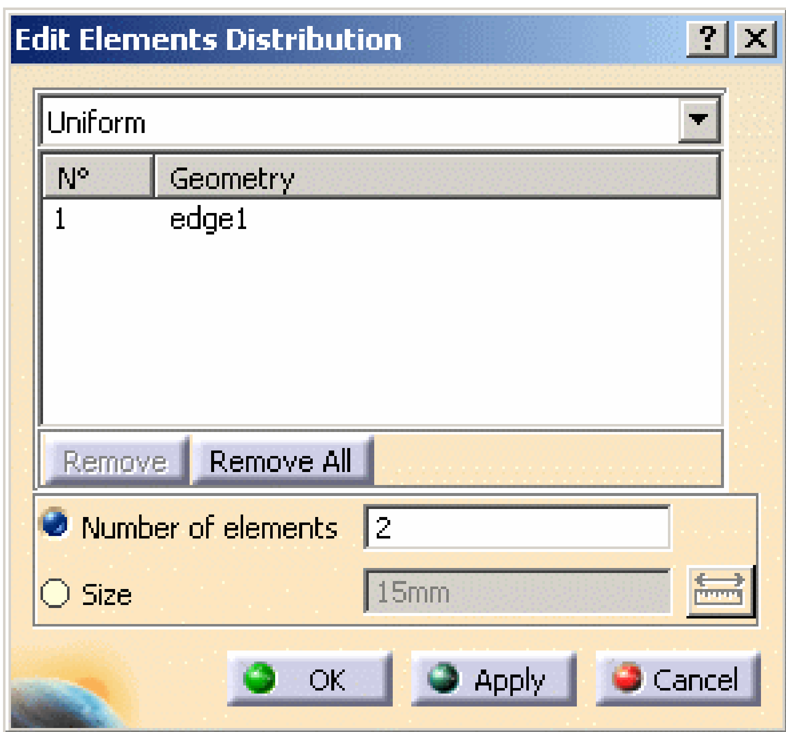
Note That you can now edit the nodes distribution done before the meshing. For more information on the Edit Elements Distribution dialog box, please refer to Imposing Nodes (Specifications).

2. Select the geometry on which you want to modify nodes distribution.



The Edit Elements Distribution dialog box now appears with default values corresponding to the edge you just selected.

3. Modify the number of the nodes. In this case, enter **2** as new value (**Number of elements**).

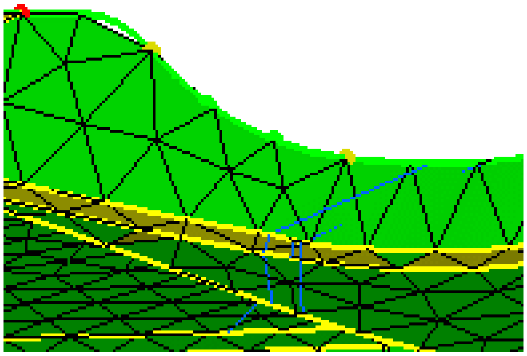


For more information about the Edit Elements Distribution dialog box, please refer to Imposing Nodes (Specifications).

4. Click **OK** in the Edit Elements Distribution dialog box.

5. In the Imposed Elements dialog box, activate the **Auto remesh on modification** option and click **OK**.

The new nodes appear on the geometry which is automatically re-meshed.



Re-meshing a Domain

This task shows how to re-mesh a domain using new specifications such as mesh method and size.

You will see here only three particular mesh methods:

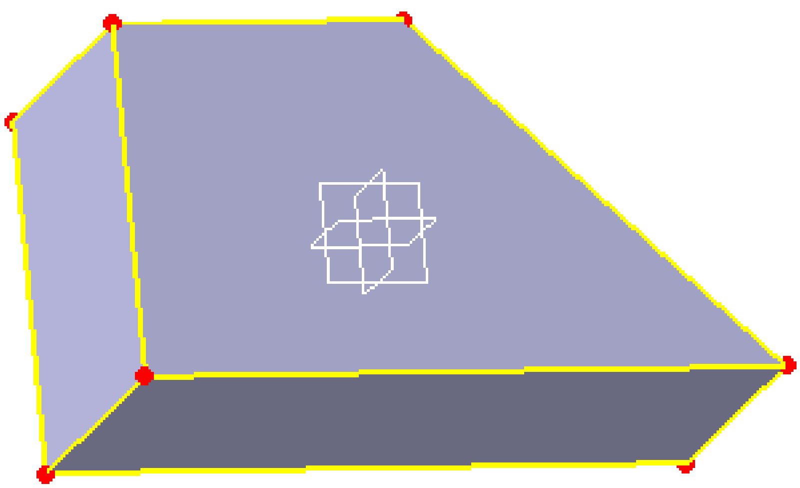
- mapped quads
- projection
- mapping

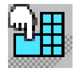
Mapped Quads Mesh Method

- Open the sample22.CATAnalysis document from the samples directory.

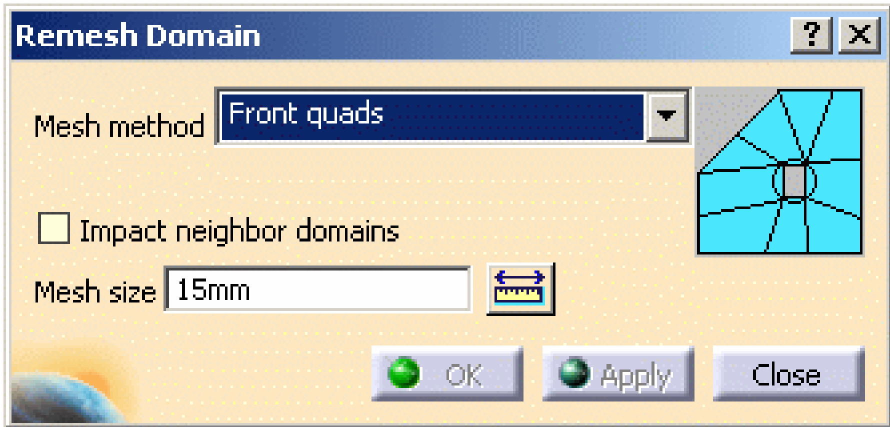
- Enter the **Surface Meshing** workshop.

  For more details, please refer to Entering the Surface Meshing Workshop.

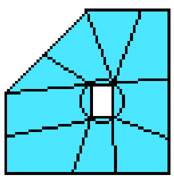


1. Click the **Remesh Domain** icon  in the **Edition Tools** toolbar.

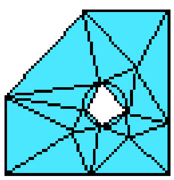
The Remesh Domain dialog box appears.



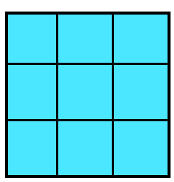
- **Mesh method**:



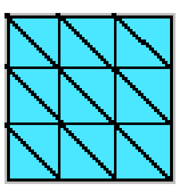
Front quads (*)



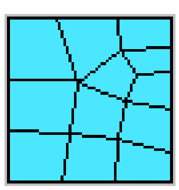
Front trias (*)



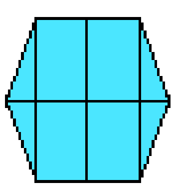
Mapped quads



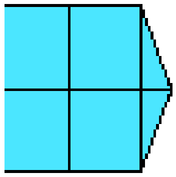
Mapped trias

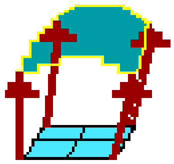


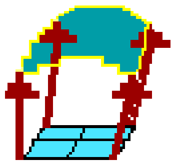
Mapped Free quads



Bead quads


Half Bead quads

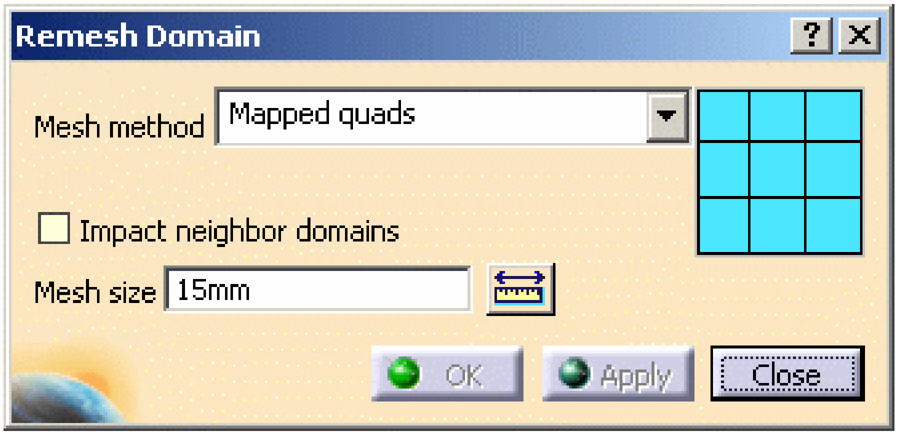

Projection


Mapping (*)

(*) indicates that the Trap Type dialog box appears at the same time you choose the option.

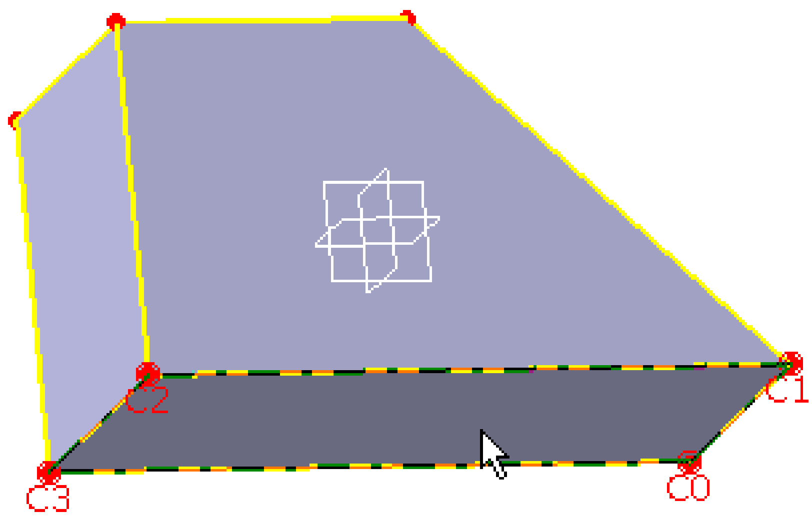
- You can use multi-selection:
  - **using an intersection polygon trap**: all the domains which have non-empty intersection with the trap will be selected.
  - **using an inclusive polygon trap**: all the domains that are completely included within the trap will be selected.
- **Impact neighbor domains**: lets you define whether you wish to apply the new mesh method to the neighboring domains. If the option is de-activated, the nodes on domain edges will not be modified.
- **Smooth projected mesh**: allows you to optimize the quality of the projected mesh (this option is only available with the **Projection** and the **Mapping** options).
- **Mesh size:** lets you enter the desired mesh size (this option is not available with the **Projection** and the **Mapping** options).

2. Select the **Mapped quads** option in the Remesh Domain dialog box.



3. Select the desired domain on the surface.

The edge of the selected domain can be either green (constrained) or yellow (free).

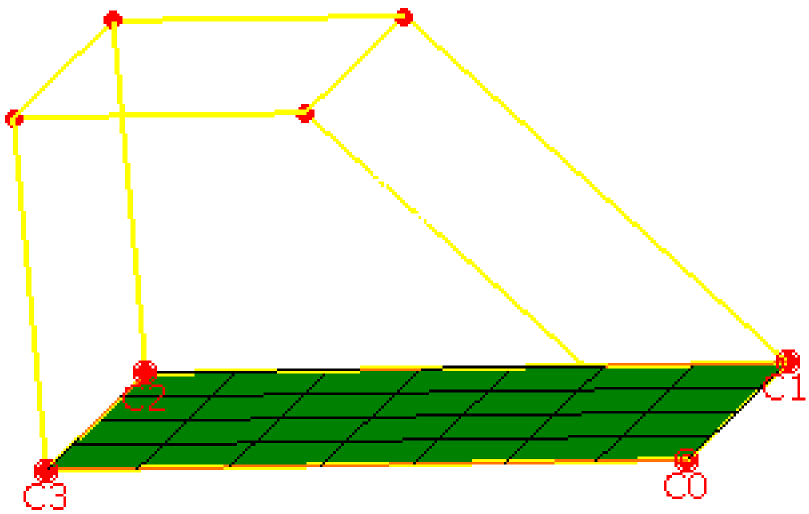


4. Enter the desired **Mesh size** value.

In this particular example, enter **15mm**.

5. Click **Apply** in the Remesh Domain dialog box.

The selected domain is re-meshed.



You can see that the corners (C0, C1, C2 and C3) of the domain are displayed.

You can select a corner and change its position by clicking another vertex.

6. Click **Close** in the Remesh Domain dialog box.

Note that: you can remove the mesh if you are not satisfied.

For more details, please refer to Removing the Mesh.

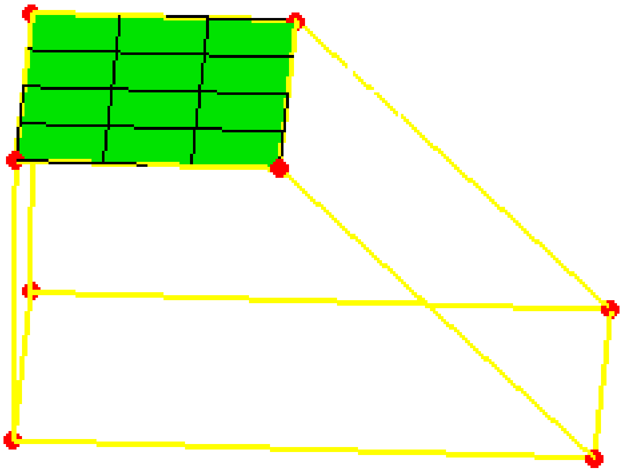
Projection Mesh Method

All the mesh elements are projected on the receiving domain and the projected mesh respects the initial mesh topology.

The projection mesh method automatically makes the correspondence between vertex and node (by projecting a vertex on a node).

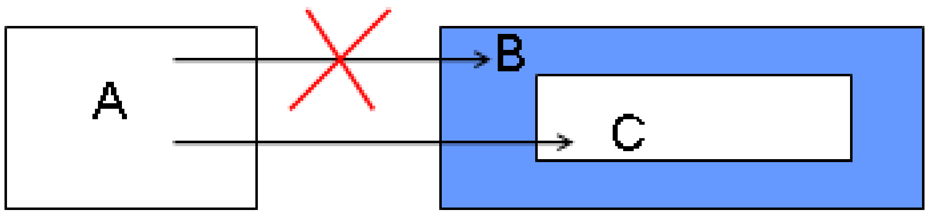
- Open the sample22_1.CATAnalysis document from the samples directory.
- Enter the **Surface Meshing** workshop.

  For more details, please refer to Entering the Surface Meshing Workshop.



The two domains (the meshed one and the receiving one) must have the same number of boundaries.

In the following example:



- you can project the mesh of the domain A to the domain C.
- you cannot project the mesh of the domain A to the domain B.
  If you try to do this, an error message appears to inform you that the number of boundaries of the two domains is different.

1. Click the **Remesh Domain** icon in the **Edition Tools** toolbar.

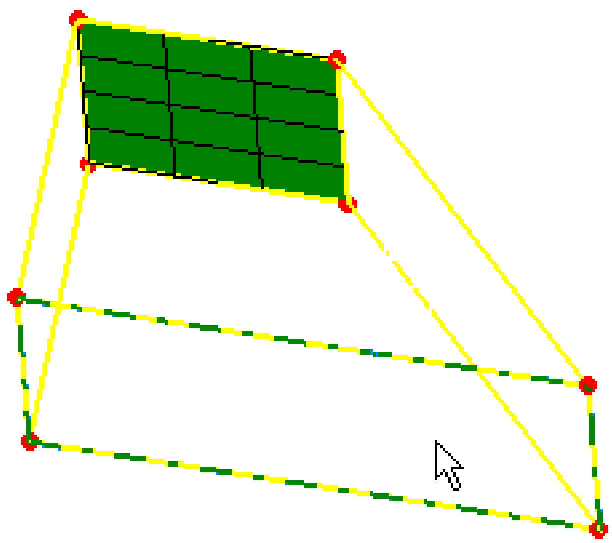
   The Remesh Domain dialog box appears.

   For more details about this dialog box, please refer to Mapped Quads Mesh Method.

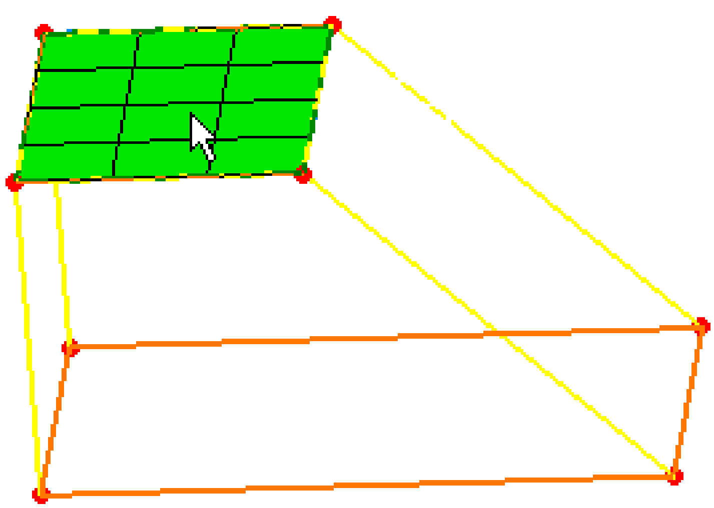
2. Select the **Projection** option in the Remesh Domain dialog box.

3. Select the domain you want to mesh.

   The meshed domain must be towards the receiving domain.



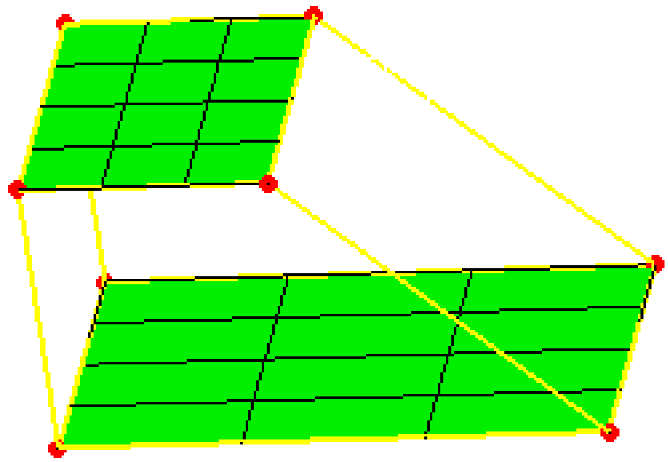
4. Select the mesh domain.



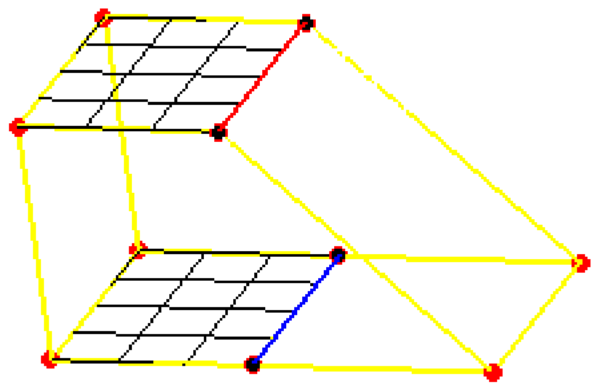
5. Click **OK** in the Remesh Domain dialog box.

The projected domain is updated.

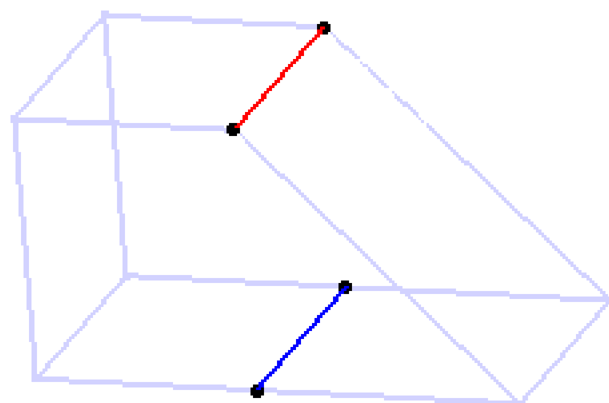
The projected mesh entirely fills the receiving domain.
The projected mesh is not partial.



The only way to preserve the mesh elements size between the meshed domain and the receiving domain



is to modify the receiving domain (by splitting it, for example)



Mapping Mesh Method

All the mesh elements are projected on the receiving domain and the projected mesh respects the initial mesh topology.

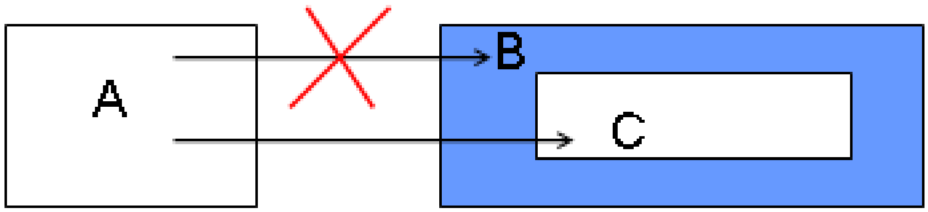
Contrary to the projection mesh method, you have to define the correspondence between vertex and node in case of mapping mesh method.

- Open the sample22.CATAnalysis document from the samples directory.
- Enter the **Surface Meshing** workshop.

  For more details, please refer to Entering the Surface Meshing Workshop.

The two domains (the meshed one and the receiving one) must have the same number of boundaries.

In the following example:



- you can project the mesh of the domain A to the domain C.
- you cannot project the mesh of the domain A to the domain B.
  If you try to do this, an error message appears to inform you that the number of boundaries of the two domains is different.

1. Click the **Remesh Domain** icon in the **Edition Tools** toolbar.

   The Remesh Domain dialog box appears.

   For more details about this dialog box, please refer to Mapped Quads Mesh Method.

2. Select the **Mapped Quads** option in the Remesh Domain dialog box.

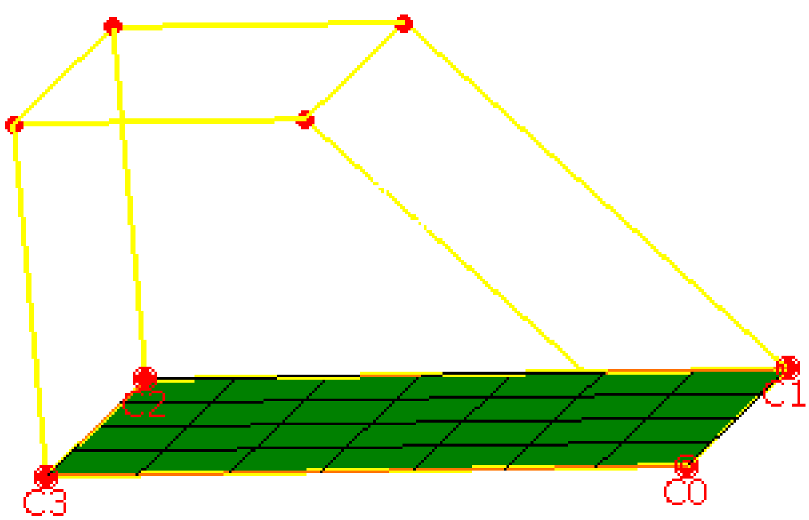
   Multi-selection is not available.

3. Select the desired domain.

4. Enter the desired **Mesh size** value.

   In this particular example, enter **15mm**.

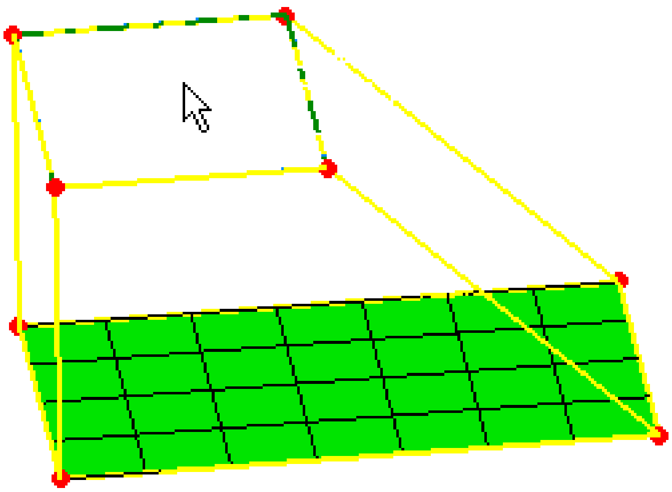
5. Click **OK** in the Remesh Domain dialog box.



6. Click the **Remesh Domain** icon in the Modification Tools toolbar.

7. Select the **Mapping** option in the Remesh Domain dialog box.

8. Select the domain you want to mesh.

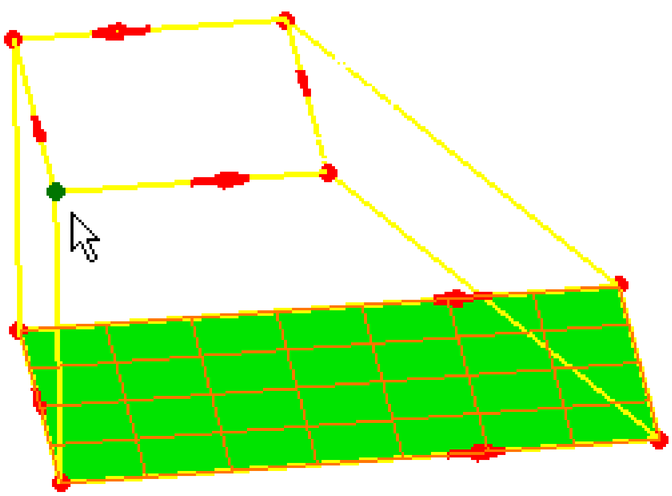


The Trap Type dialog box appears.
For more details about this diolog box, please refer to Mapped Quads Mesh Method.

Multi-selection is available on the condition you multi-select connected mesh domains.

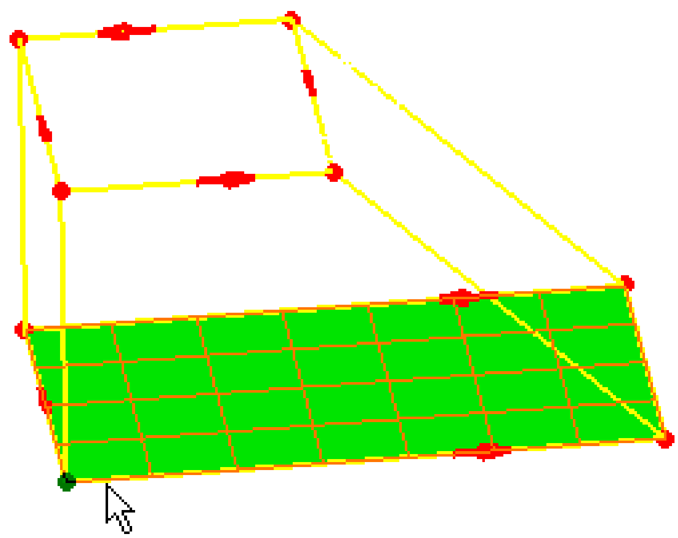
9. Select the meshed domain to be projected on the previously selected domain.

10. Select a vertex on the domain that will receive the projected mesh to identify it with the nearest node of the mesh domain.

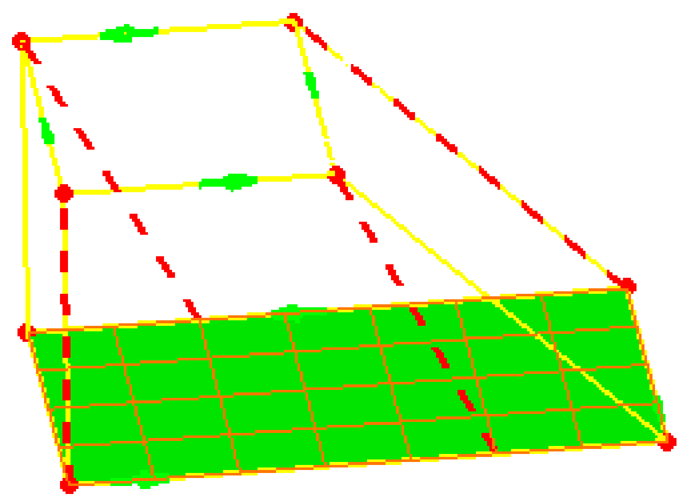


You now have to set up all the needed correspondence between vertex (of the domain to be meshed) and the nodes (of the mesh domain).

11. Select the corresponding node on the mesh domain.



As soon as you have selected the node, the geometry appears as shown here:

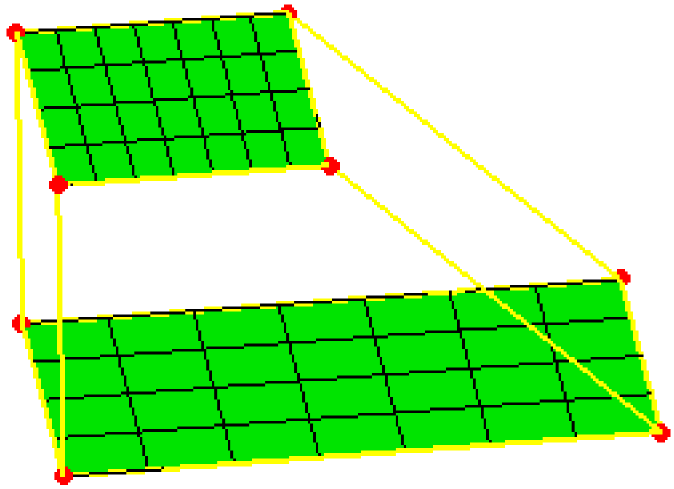


You can see that some vertex are not corresponding with nodes of the mesh domain. You can manually establish the correspondence.

For this, select a vertex and then the corresponding node on the mesh domain.

12. Click **OK** in the Remesh Domain dialog box.

The mapped domain is updated as shown here:



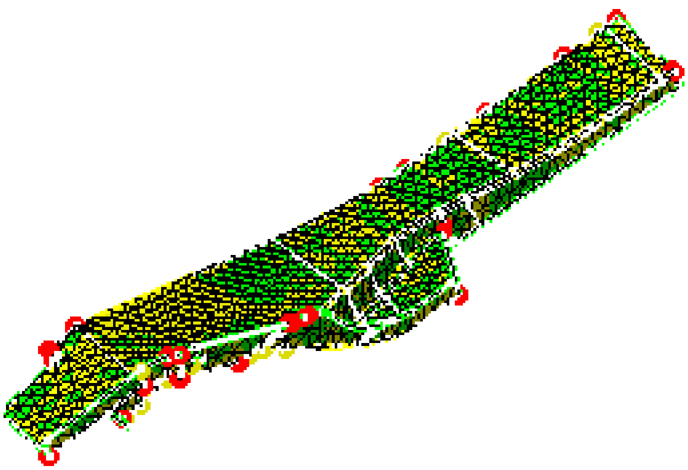
Removing the Mesh by Domain

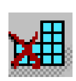
This task shows how to remove the mesh you generated on a single or several domains of the geometry.

- Open the sample08.CATAnalysis document from the samples directory.
- Enter the **Surface Meshing** workshop.

  For more details, please refer to Entering the Surface Meshing Workshop.
- Mesh the surface.

  For more details, please refer to Meshing the Part.



1. Click the **Remove Mesh by Domain** icon  from the **Edition Tools** toolbar to activate the remove mesh by domain mode.

   The Trap Type dialog box appears.

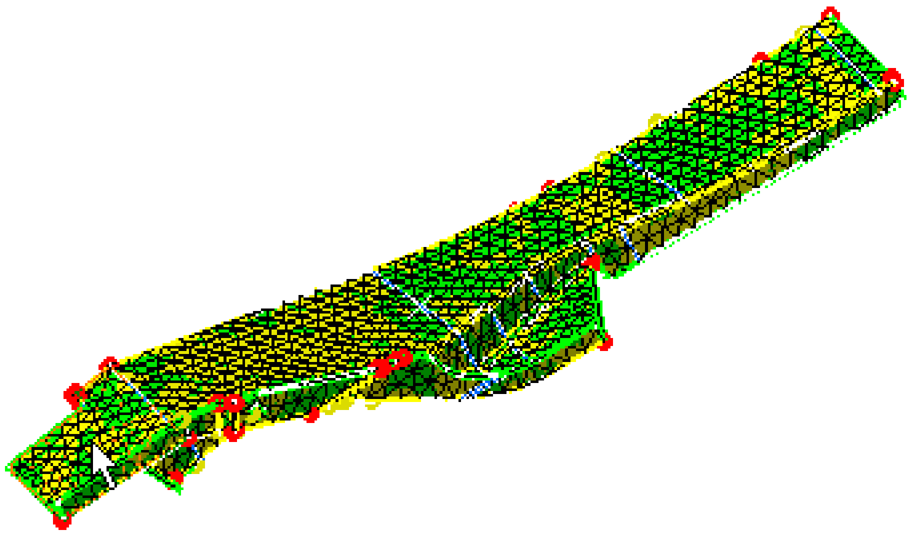
   

2. Select the mesh of the single domain or domains you want to remove.

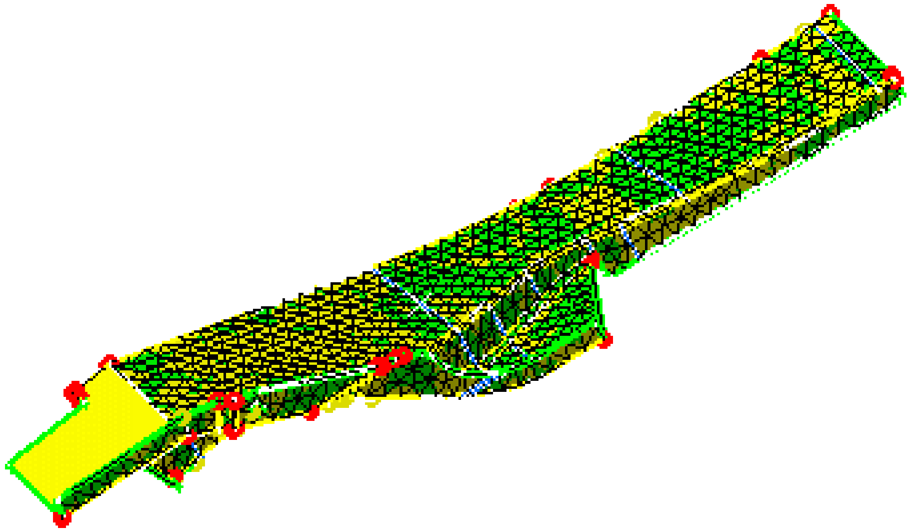
For this You can use:

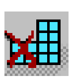
- Mono selection to select a single domain
- Multi-selection using the Trap Type dialog box
  - using an intersection polygon trap: all the domains which have non-empty intersection with the trap will be selected.
  - using an inclusive polygon trap: all the domains that are completely included within the trap will be selected.

In this particular case click a domain as shown below:



As a result the mesh of the selected domain is removed:



3. Click the **Remove Mesh by Domain** icon  from the **Edition Tools** toolbar to deactivate the remove mesh by domain mode.

The Trap Type dialog box disappears.

Note that removing the mesh by domain is useful before Re-meshing a Domain.

You can improve the mesh of a domain without removing the mesh of all the part.

Locking a Domain

This task shows how to lock a domain.

- Open the sample05.CATAnalysis document from the samples directory.
- Enter the **Surface Meshing** workshop.

  For more details, please refer to Entering the Surface Meshing Workshop.
- Mesh the surface.

  For more details, please refer to Meshing the Part.

1. Select the **Lock Domain** icon from the **Edition Tools** toolbar.

   The Trap Type dialog box appears

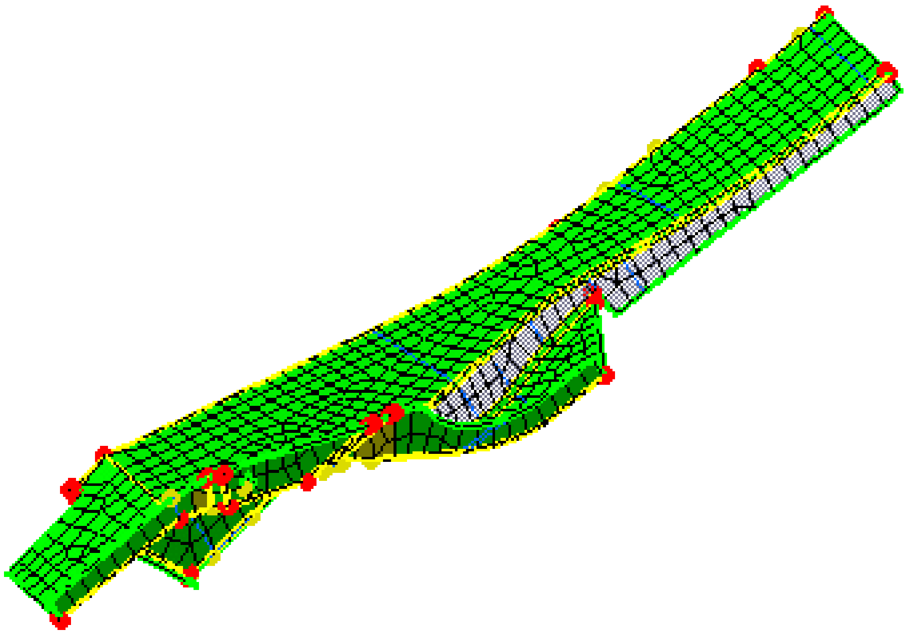
   You can use multi-selection:

   - **using an intersection polygon trap**: all the domains which have non-empty intersection with the trap will be selected.
   - **using an inclusive polygon trap**: all the domains that are completely included within the trap will be selected.

2. Select the domain to be locked.

This domain is now transparent:

- any modification applied to neighboring domain will not impact the locked domain.
- the locked domain cannot be re-meshed.



If you select two domains which are not homogeneous, a warning box appears: **Do you want to lock the selected domains?** If you click **No**, the domains will be unlocked.

Mesh Editing

This task shows how to edit surface elements. Thanks to the full cursor editing, you can preview and then update dynamically the quality analysis results on the mesh. When quality criteria are respected, you will see the change in finite element color. You can also move nodes on geometry, edit finite elements (splitting, swapping, and so forth) or smooth the mesh.

Mesh Editing on:

- Surface Elements
- Edges

using either the Contextual menu or:

- Dynamic Update of Element Quality Analysis (colors change, if needed)
- Full Mouse Mesh Editing
- Automatic Smooth on Nodes
- Contextual Menu

- Open the sample05.CATAnalysis document from the samples directory.
- Enter the **Surface Meshing** workshop.

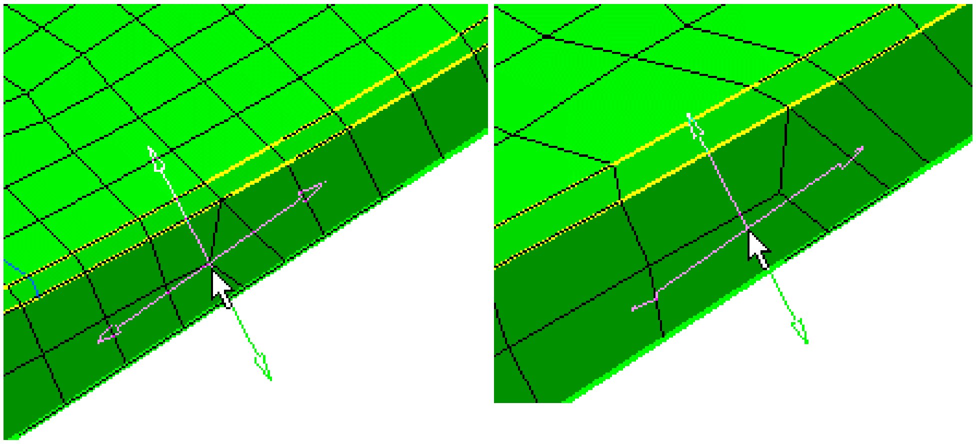
  For more details, please refer to Entering the Surface Meshing Workshop.
- Mesh the surface.

  For more details, please refer to Meshing the Part.

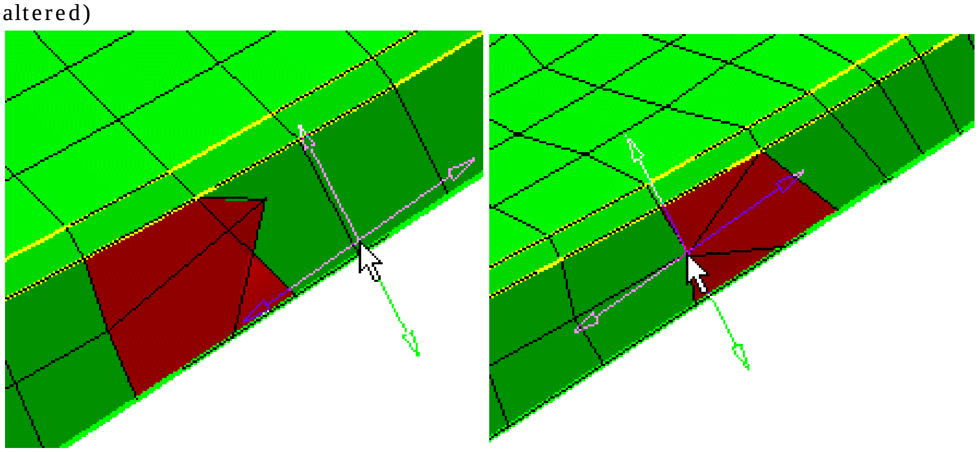
1. Click the **Edit Mesh** icon from the **Edition Tools** toolbar.

   The Edit Mesh dialog box appears.

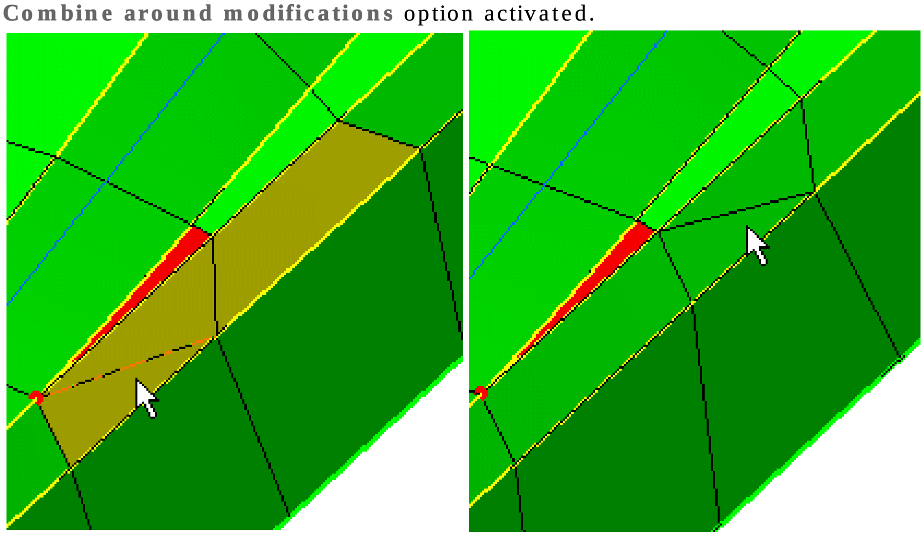
- **Smooth around modifications**: allows optimizing the shape of the element so that split elements meet desired Analysis criteria:
  Note that as the **Auto smooth** option is inactive in the Edit Mesh dialog box, the surrounding elements are not modified and therefore do not enhance the quality whatever the modifications you perform.
  - **Smooth around modifications** option activated (the shape of the neighbor elements is altered in order to have the best possible quality of the elements).



- **Smooth around modifications** option de-activated (the shape of the neighbor elements is not altered)



- **Combine around modifications**: allows combining two triangles into a quadrangle of good quality (when possible).
  - **Combine around modifications** option activated.

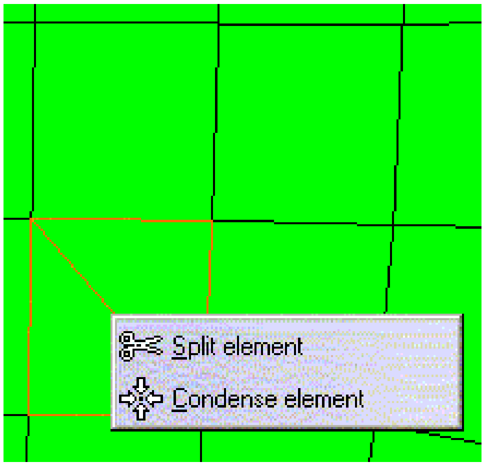


- **Swap around modifications**: allows dragging a node on a surface.
- **Propagate to neighbour domains**: allows propagating smoothing modifications from one domain to a neighboring one.
- **Global Optimization**: lets you launch again the Quality Analysis.

Surface Elements

Using the Contextual Menu

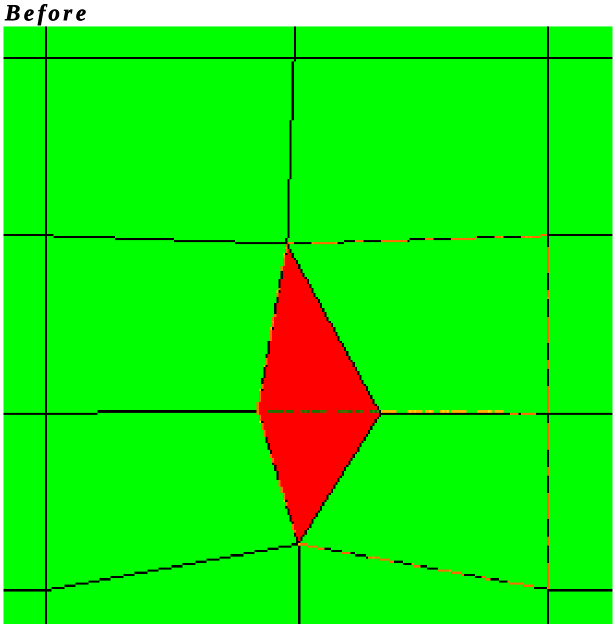
You can split or condense elements:



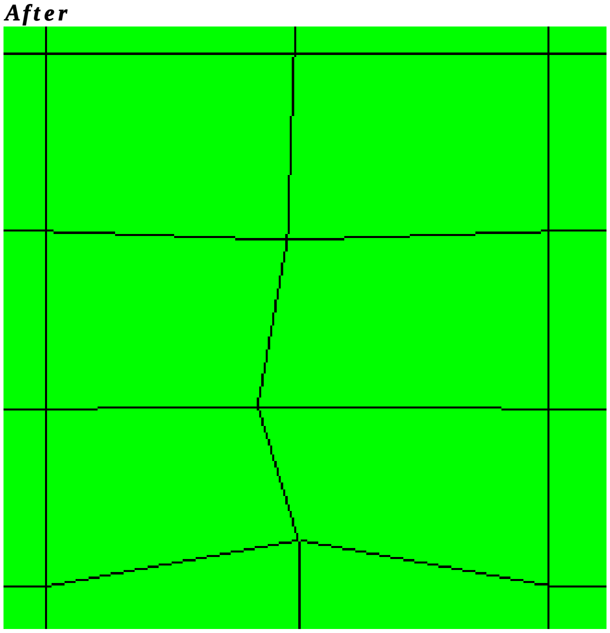
2. Position the **Condense element** symbol very close to a node and right-click.

In this case, we de-activated all the options from the Edit Mesh dialog box.

Target vs opposite node
Before



Condensed node
After

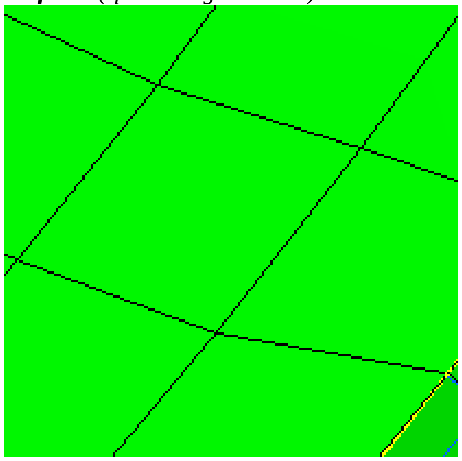


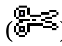
Dynamic Update of Element Quality Analysis (colors change, if needed)

3. Position the cursor over one quadrangle element.

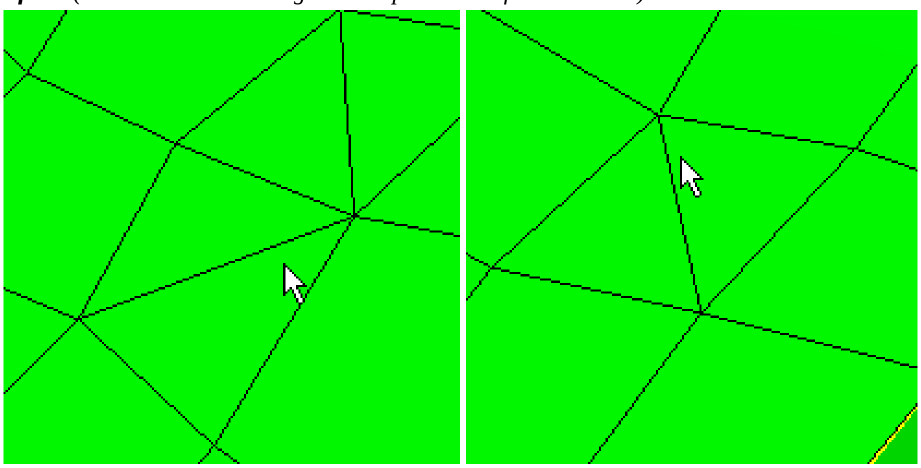
You can now cut the mesh element diagonally according to the position of the cursor.

- ***Before*** *(quadrangle mesh)*



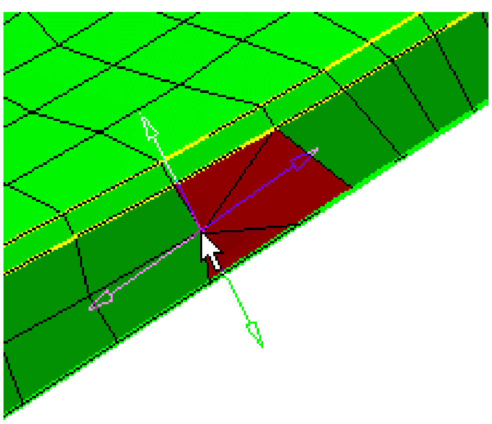
When you go over the quadrangle mesh you want to cut, both a symbol () and orange-highlighted cut mesh preview appear on this mesh.

- ***After*** *(cut mesh according to the position of the cursor)*



Full Mouse Mesh Editing

You can drag nodes on surfaces



4. Select a node and move it to the desired location.

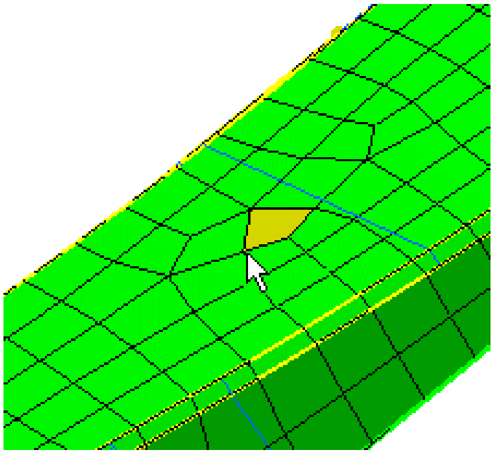
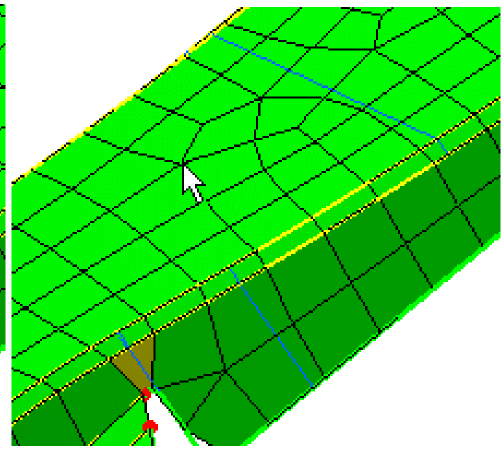
   The quality of the elements updates according to the location you assign to the node.

Automatic Smooth on Nodes

You can smooth elements by letting the algorithm update the mesh elements location.

5. Click on a node.

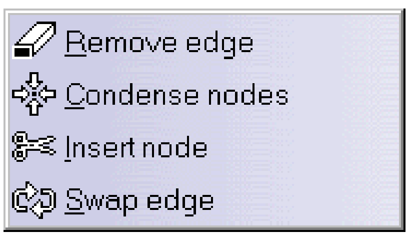
The node is automatically moved to the optimum location. The quality of the elements updates according to this new location.

Edges

Contextual Menu

6. Right-click a segment.

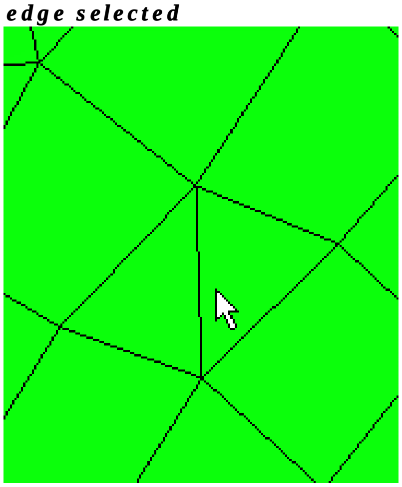
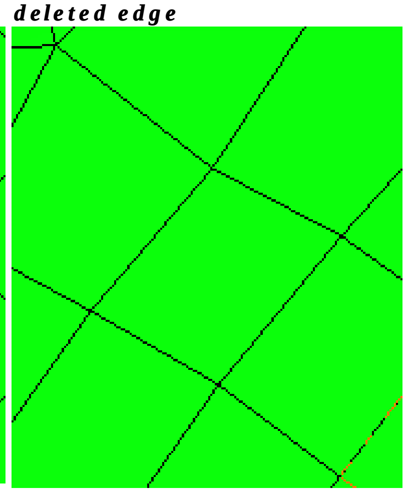


Once you selected an option from the contextual menu, when you go over the quadrangle mesh you want to edit, both the corresponding symbol and orange-highlighted edited mesh preview appear on this mesh.

- 

: allows removing edges you previously created manually when splitting a quadrangle element
In this case, we activated all the options, right-clicked an edge and activated the **Remove edge** contextual menu.

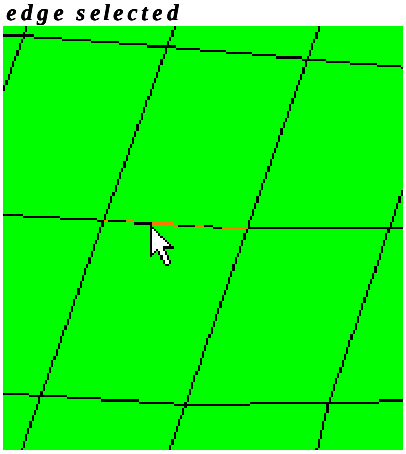
edge selected ***deleted edge***

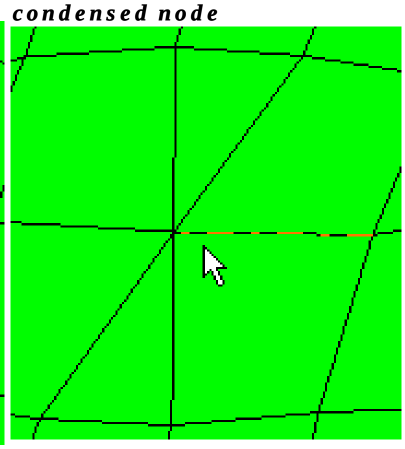
- 

: allows condensing nodes on edges or a node on another node.
  - In this case, we activated all the options, right-clicked an edge and activated the **Condense nodes** contextual menu.

edge selected



condensed node

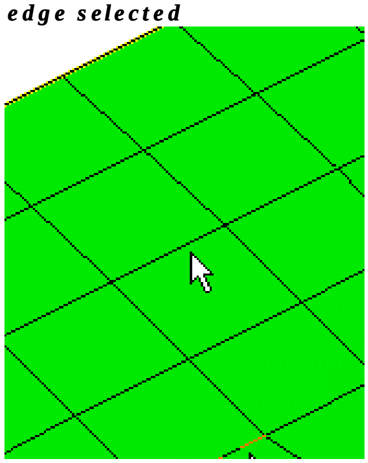


  - In this new case, we de-activated all the options, positioned the Condense symbol very close to a node and right-clicked.

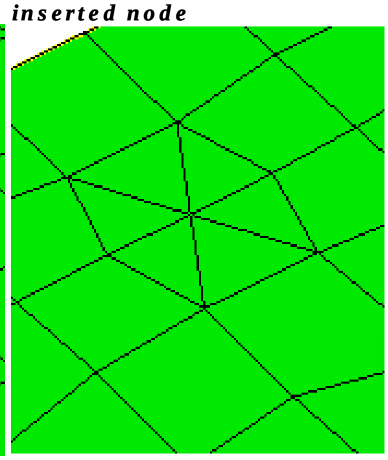
- 

: allows inserting a node on an edge
  - In the case below, we de-activated the **Auto combine** and **Auto swap** options, right-clicked an edge and activated the **Insert node** contextual menu.

edge selected

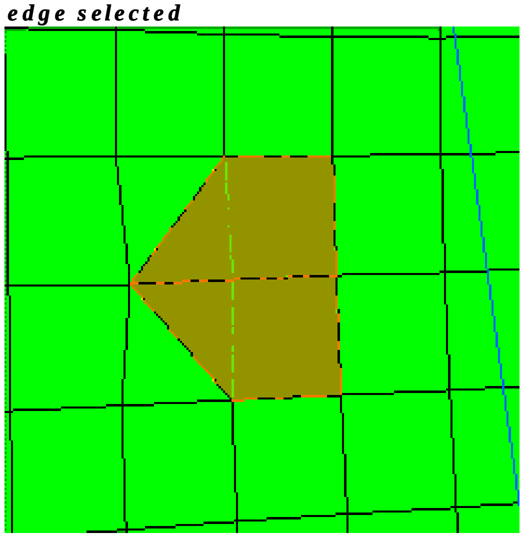


inserted node

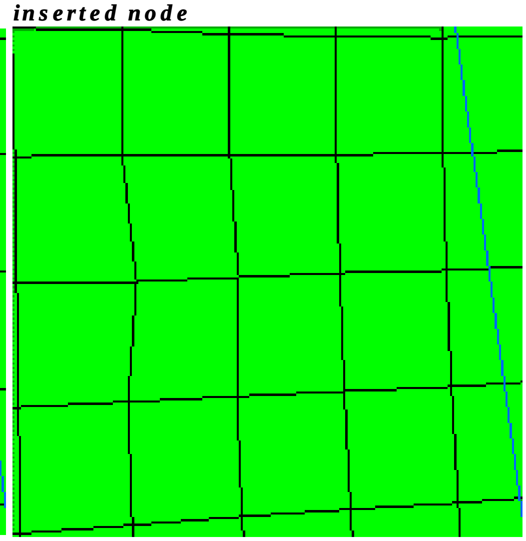


  - In the case below, we activated the **Auto combine** options, right-clicked an edge and activated the **Insert node** contextual menu.

edge selected



inserted node



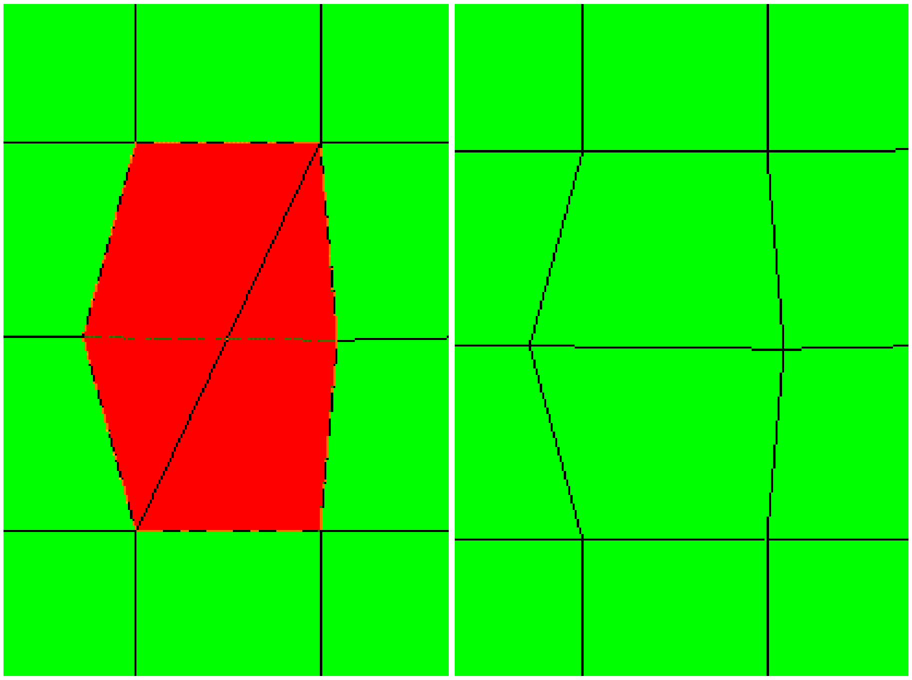
- 

: allows swapping between two edges.

  When you make a swap between a triangle and a quadrangle, whatever the edge new position, the resulting mesh will always be a triangle and a quadrangle. The preview will be a good help.

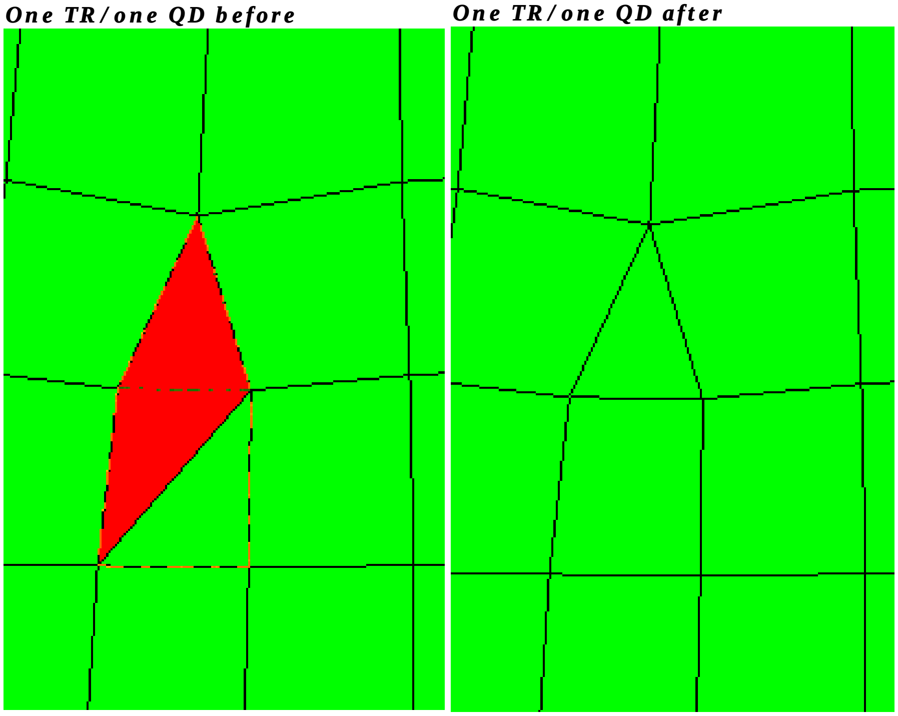
  - Two Quadrangle Elements:

    edge selected ***swapped edge***

  - A Quadrangle and a Triangle:

    One TR/one QD before ***One TR/one QD after***

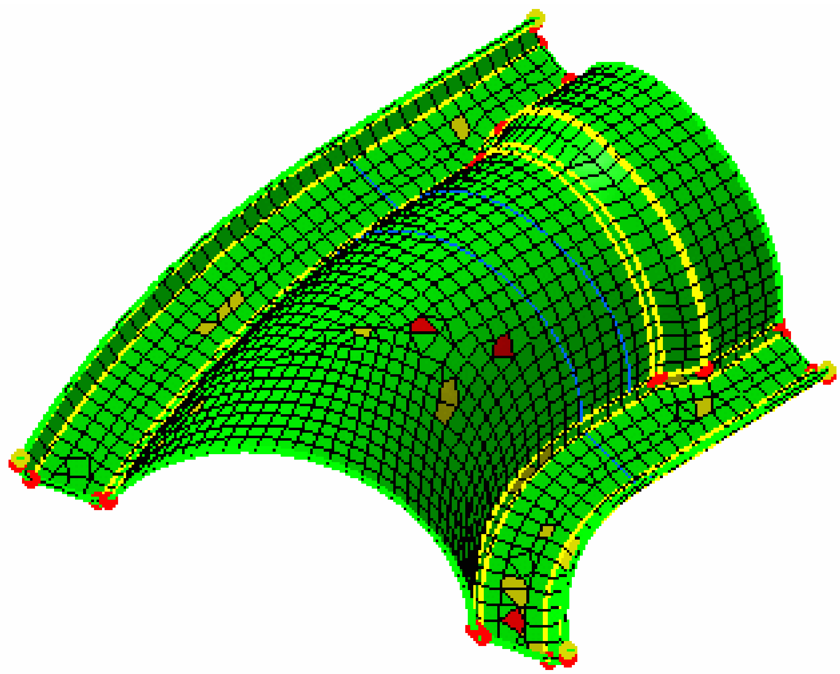
    

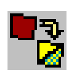
Splitting Quadrangles

This task shows you how to improve the element quality by splitting the quadrangles.

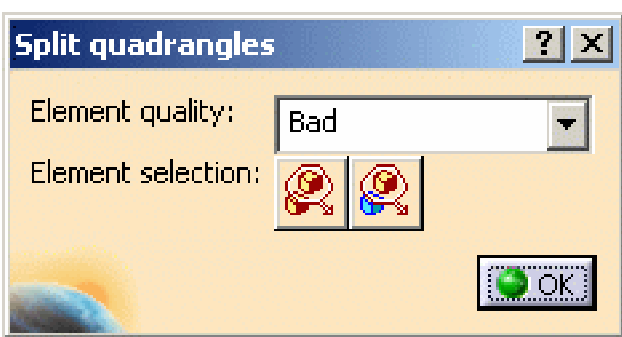
- Open the Sample01_1.CATAnalysis document from the samples directory.
- Enter the **Surface Meshing** workshop.

  For more details, please refer to Entering the Surface Meshing Workshop.



1. Click the **Split Quadrangles** icon  from the **Edition Tools** toolbar to activate the split quadrangles mode.

   The Split Quadrangles dialog box appears:

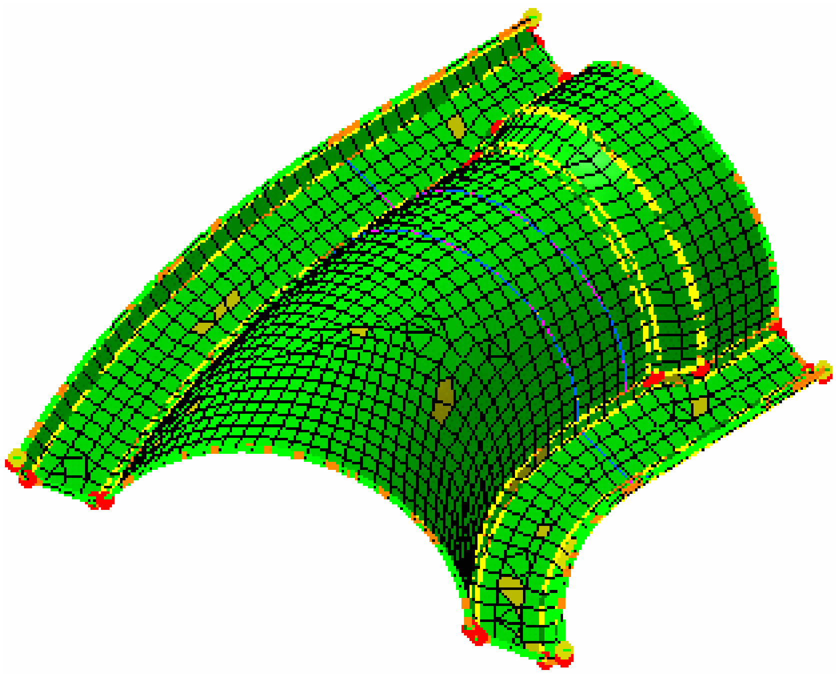
   - **Element quality**: lets you specify the elements to split depending on their quality. You can choose **Bad**, **Bad and poor**, **Poor** and **All** elements qualities
   - **Element selection**: lets you specify the elements to split depending on their

location using the trap type elements selection method

Do not use **Element Selection** parameter if you want to perform the split quadrangles functionality on all the mesh part. You have only to click on it.

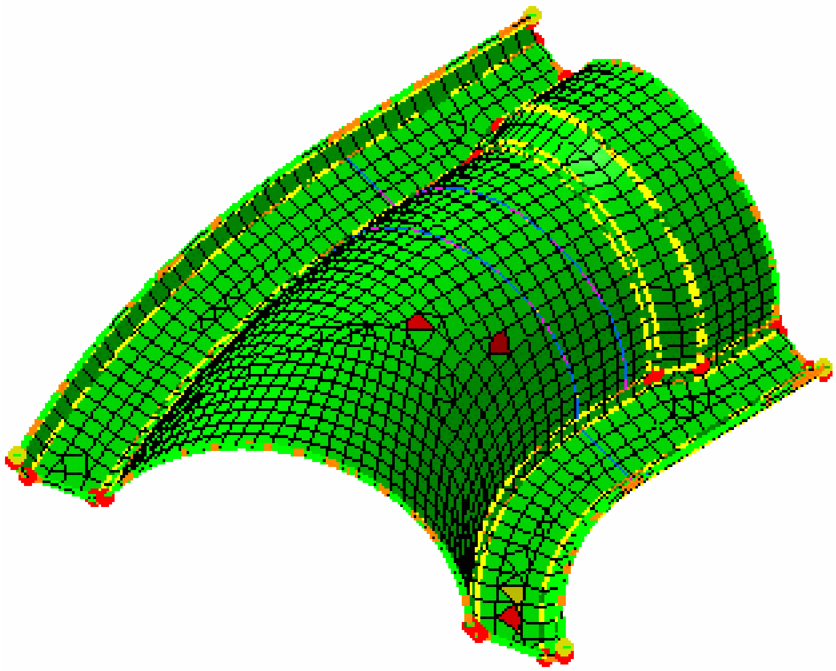
2. Select **Bad** as **Element quality** option and click the mesh part.

   Bad quality elements are split, if necessary, in order to improve their quality as shown below:



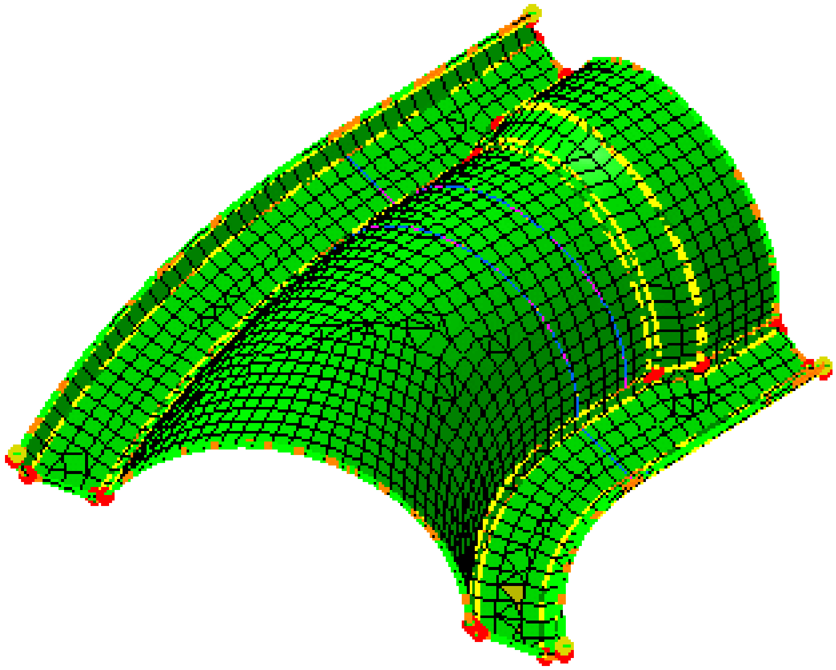
3. Click the **Undo** icon  from the Standard toolbar to undo your modifications, select **Poor** as **Element quality** option and click the mesh part.

   Poor quality elements are split, if necessary, in order to improve their quality as shown below:



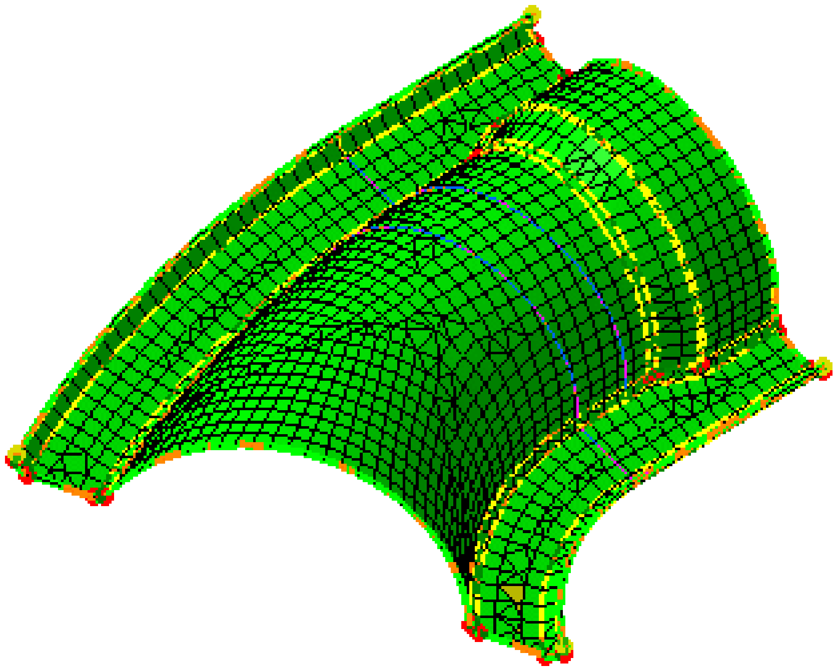
4. Click the **Undo** icon  from the **Standard** toolbar to undo your modifications, select **Bad and poor** as **Element quality** option and click the mesh part.

   Bad and poor quality elements are split, if necessary, in order to improve their quality as shown below:



5. Click the **Undo** icon  from the **Standard** toolbar to undo your modifications, select **All** as **Element quality** option and click the mesh part.

All quality elements are split, if necessary, in order to improve their quality as shown below:



6. Click **OK** in the Split quadrangles dialog box to deactivate the split quadrangles mode.

Note that each selected element is split only if it will improve the mesh quality.

Leaving the Surface Meshing Workshop

This task shows you how to exit the Surface Meshing workshop.

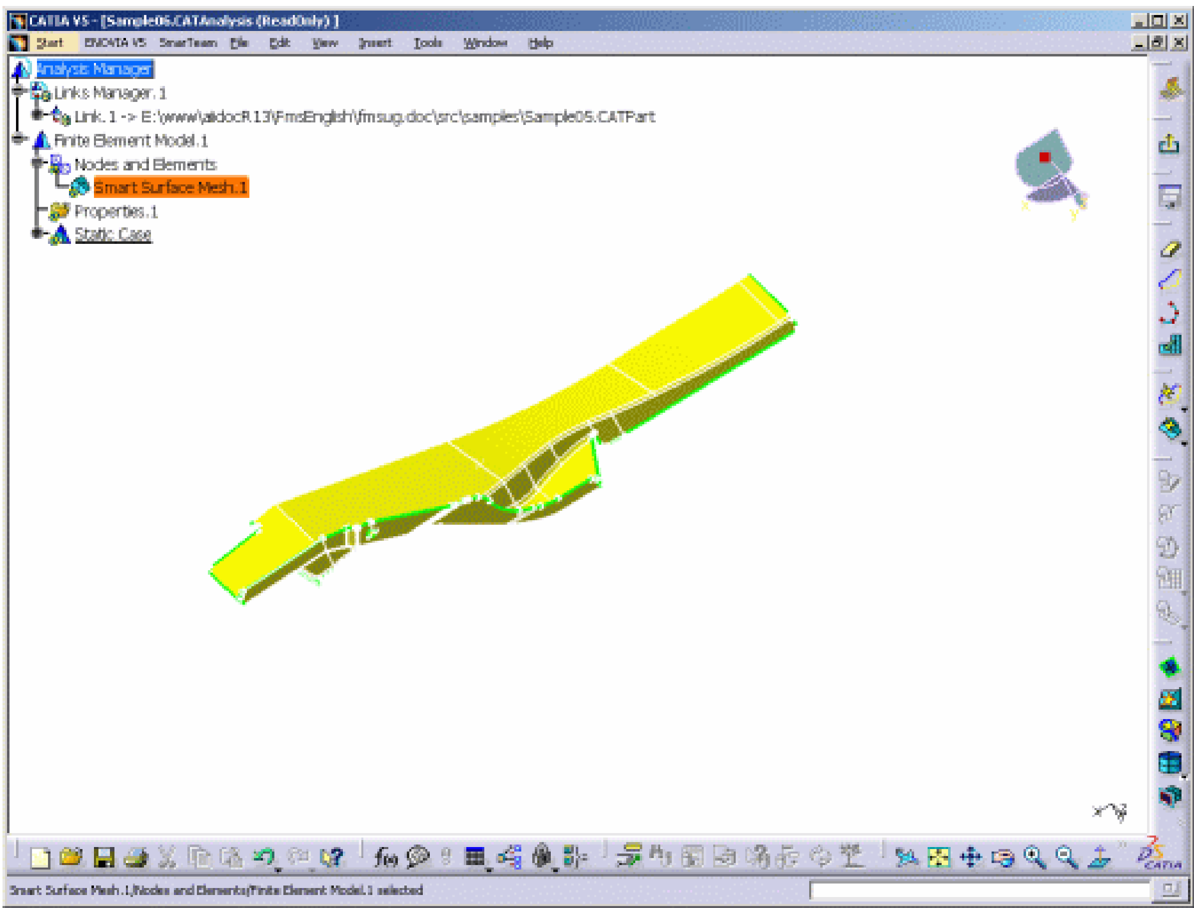
Open the sample06.CATAnalysis document from the samples directory.

1. Double-click the **Smart Surface Mesh.1** in the specification tree.

   The Global Parameters dialog box appears.

2. Click **OK** in the Global Parameters dialog box.

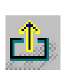
   You now enter the **Surface Meshing** workshop:



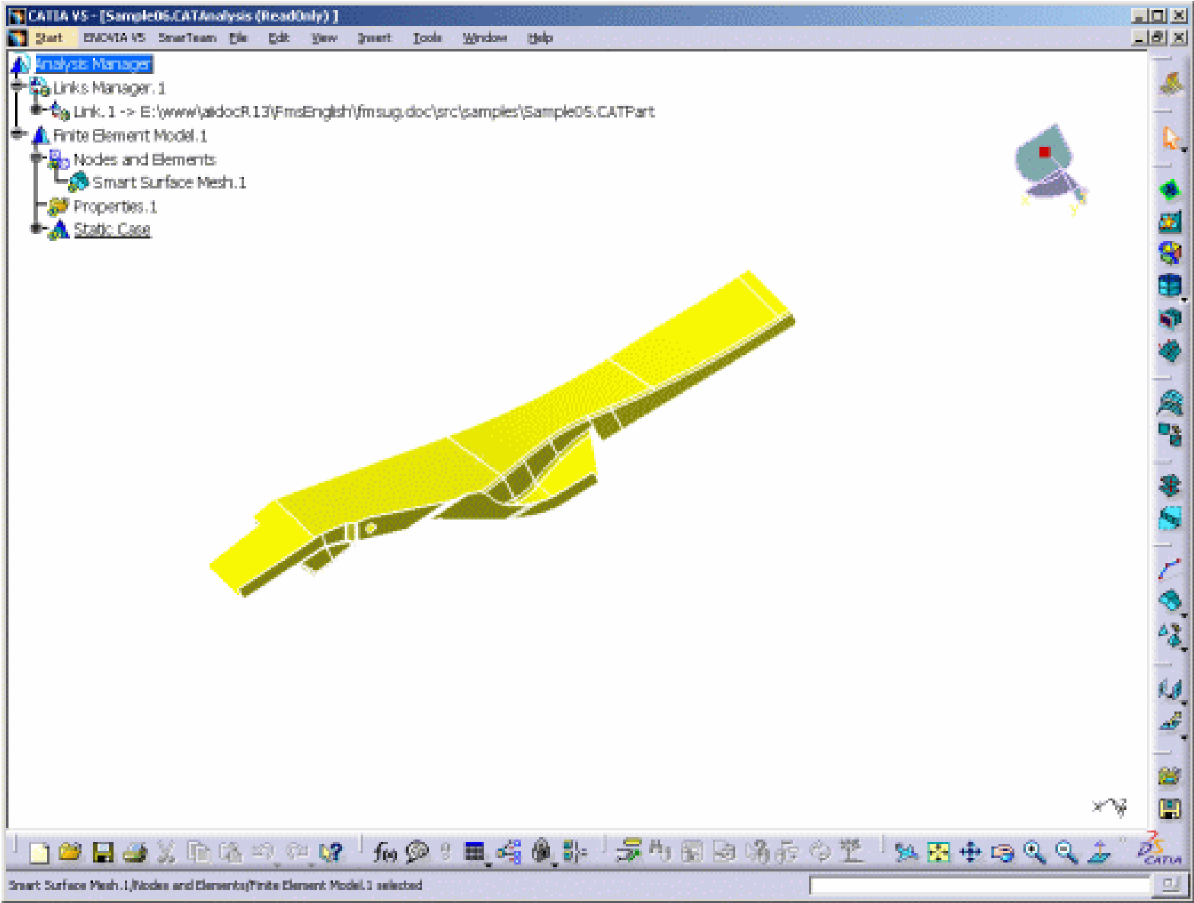
   You can access the following toolbars: Global Specifications, Local Specifications, Execution and Edit Tools.

   At any time you can exit the **Surface Meshing** workshop.

   For this:

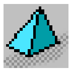
3. Click the **Exit** icon  from the **Exit** toolbar.

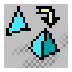
You retrieve the **Advanced Meshing Tools** workbench.



Solid Meshing

This section deals with the 3D meshing methods.

 OCTREE Tetrahedron Mesher: Assign linear or parabolic tetrahedrons elements to a solid.

 Tetrahedron filler: Mesh a solid part by generating tetrahedron meshes from surface meshes.

OCTREE Tetrahedron Mesher

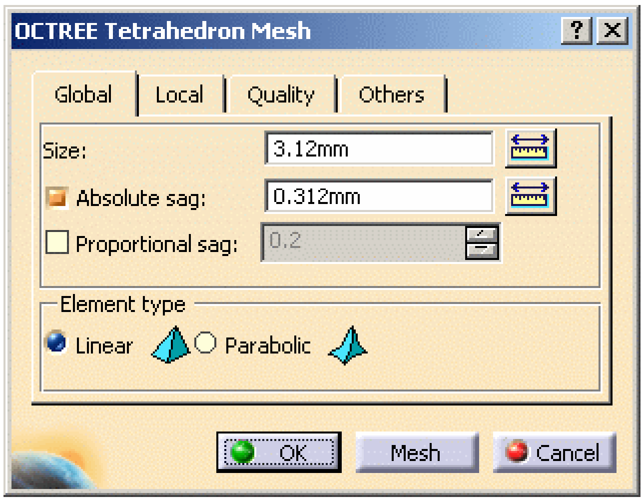
This task will show you how to mesh a solid part using the OCTREE Tetrahedron Mesher.

Open the sample41.CATAnalysis document from the samples directory.

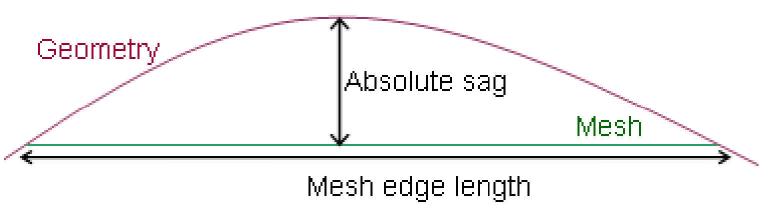
1. Click the **Octree Tetrahedron Mesher** icon from the **Meshing Methods** toolbar.
2. Select the solid you want to mesh.

The OCTREE Tetrahedron Mesh dialog box appears.

- **Global** tab:

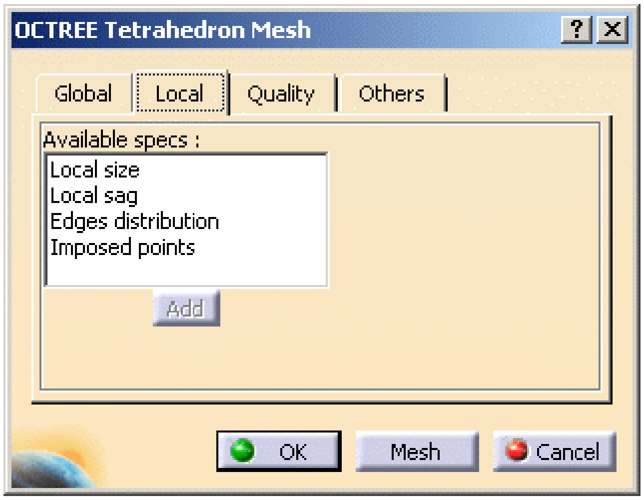


- **Size**: lets you choose the size of the elements (in mm).
- **Absolute sag**: maximal gap between the mesh and the geometry.



- **Proportional sag**: ratio between the **Absolute sag** and the local mesh edge length. **Proportional sag** value= (**Absolute sag** value) / (local mesh edge length value).
  - Note that **Proportional sag** and **Relative sag** could modify the local mesh edge length value.
  - You can use both **Proportional sag** and **Relative sag**, the most constraining of the two values will be used.
- **Element type**: lets you choose the type of element you want (**Linear** or **Parabolic**).

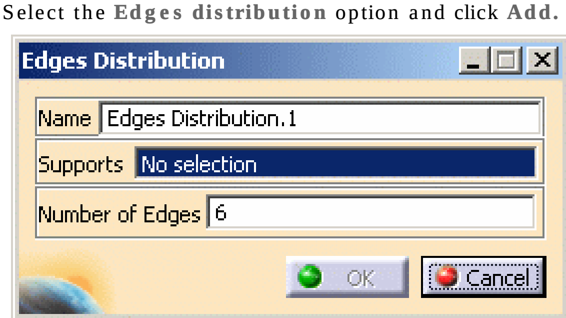
- **Local** tab:
  You can also add local meshing parameters such as sag, size or distribution to the part.
  For this select the desired **Available specs** and then click the **Add** button.



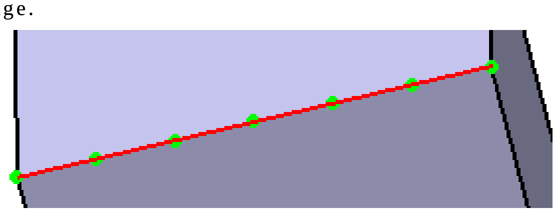
- **Local size**: you can modify the **Name**, **Support** and **Value**.

- **Local sag**: you can modify the **Name**, **Support** and **Value**.

- **Edges distribution**: lets you distribute local nodes on a edge.
  For this:
  - Select the **Edges distribution** option and click **Add.** The Edges Distribution dialog box appears.



  - Select the edge on which you want to assign nodes (**Supports**) as well as the **Number of Edges** to be created.
    The **Edges Distribution.1** feature now appears in the specification tree as well as the nodes on the selected Edge.



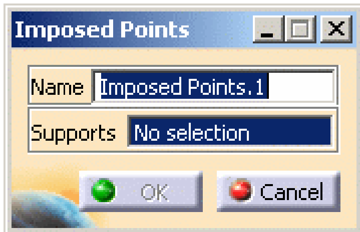
  - Click **OK** in the Local Mesh Distribution dialog box.

- **Imposed points**: lets you select the points that will be taken into account when meshing.

  In this case, the points you have to select must have been created via Shape Design or Part Design. Only points on curve or points on surface are supported. The points support must be a member of the meshed geometry.

  For this:

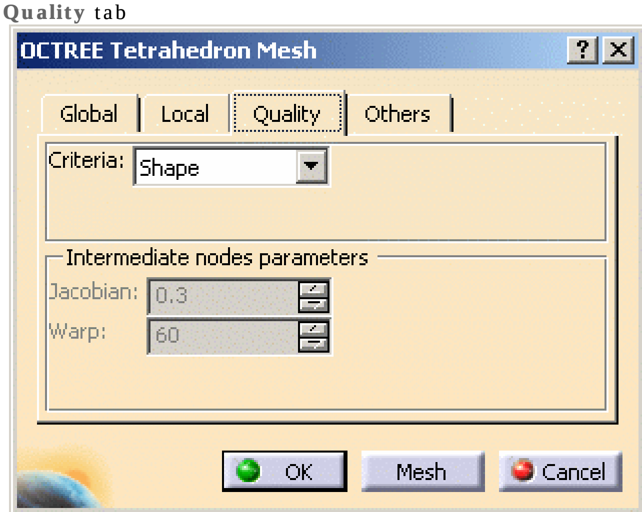
  - Select the **Local imposed points** option and click **Add**.
    The Imposed Points dialog box appears.

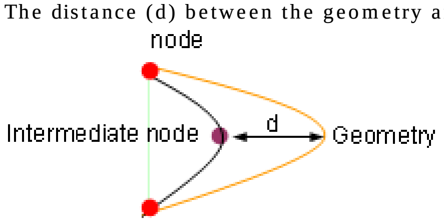
  - Select from the specification tree (under **Open Body** feature) the points (**Supports**) you will impose for OCTREE tetrahedron mesh generation.
  - Click **OK** in the Imposed Points dialog box.

    To edit the local mesh distribution that has just been created, you need to double-click the **Local Nodes Distribution** object in the specification tree and modify the desired options from the Local Mesh Distribution dialog box that appears.

○ **Quality** tab

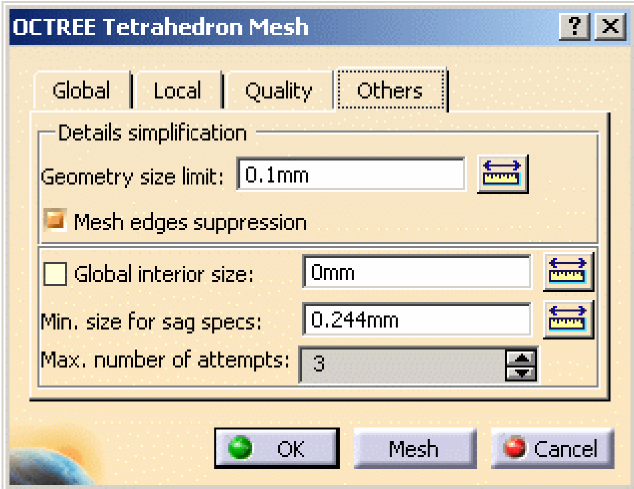


- **Criteria**: lets you choose a criterion (**Shape**, **Skweness** or **Strech**) to optimize the mesh quality.
- **Intermediate nodes parameters**: only available if you have chosen a **Parabolic** element type. This option lets you choose the position of parabolic tetrahedron intermediate nodes (Jacobian or Warp).
  The distance (d) between the geometry and the intermediate node is function of **Jacobian** and **Warp** values.



For more details about mesh quality analysis, please refer to Analyzing Element Quality.

- **Others** tab:

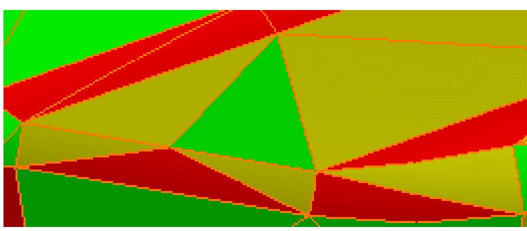


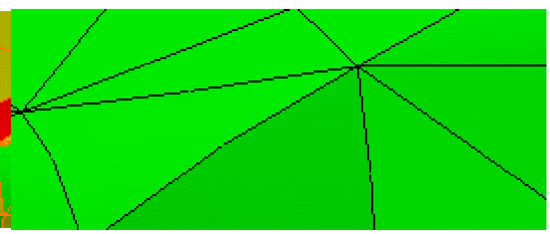
- **Details simplification**: lets you remove small mesh.
  - **Geometry size limit**: lets you specify the maximum size of the elements ignored by the mesher (before meshing).

    If all the edges of a surface are smaller than the **Geometry size limit** value, this surface will be ignored by the mesher.

  - **Mesh edges suppression**: removes small edges (after meshing).

    Without **Mesh edge suppression**: With **Mesh edge suppression**:

    It may happen that **Mesh edge suppression** involve constraints violation.

  - **Global interior size**: lets you specify the maximum interior size of the mesh.

    If the value of the **Global interior size** is smaller than the value of the **Size**, the value of the **Size** will be reduced to the value of the **Global interior size**.

  - **Min. size for sag specs**: lets you specify the minimum size of the mesh refining due to sags specifications.
  - **Max. number of attempts**: lets you impose a maximum number of attempts, if several attempts are needed to succeed in meshing, in the case of a complex geometry.

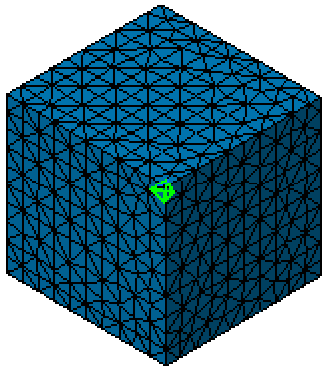
3. Select the desired parameters in the OCTREE Tetrahedron Mesh dialog box.

   In this particular example, preserve the default parameters.

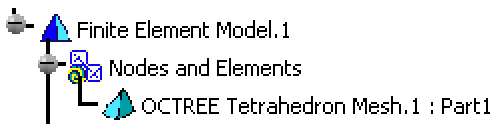
4. Click **Mesh** in the OCTREE Tetrahedron Mesh dialog box.

The Computation Status dialog box appears and the mesh is generated on the part.

For a better visualization, hide the geometry.
For this, right-click the **Links Manager.1** object and select the **Hide/Show** contextual menu.



The **OCTREE Tetrahedron Mesh.1: Part.1** object appears in the specification tree.



To edit the Octree tetrahedron mesh, double-click the **OCTREE Tetrahedron Mesh.1: Part.1** object in the specification tree.

The OCTREE Tetrahedron Mesh dialog box reappears.

Tetrahedron Filler

This task will show you how to mesh a solid part by generating tetrahedron meshes from surface meshes.

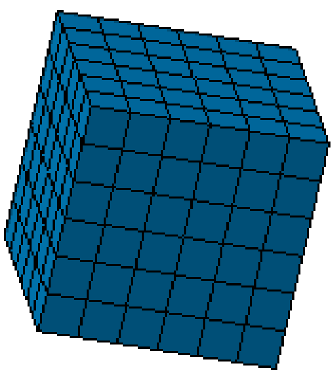
The tetrahedron filler provides volume mesh (Linear Tetrahedron or Parabolic Tetrahedron - cf. *Finite Element Reference Guide*) from surface mesh (Linear Triangle Shell or Linear Quadrangle - cf. *Finite Element Reference Guide*).

There are two necessary conditions to generate a solid mesh:

- Make sure that the mesh surface is closed in terms of connectivity.
- Make sure that the surface mesh has no intersection.
  For this, please refer to Checking Intersections / Interferences.

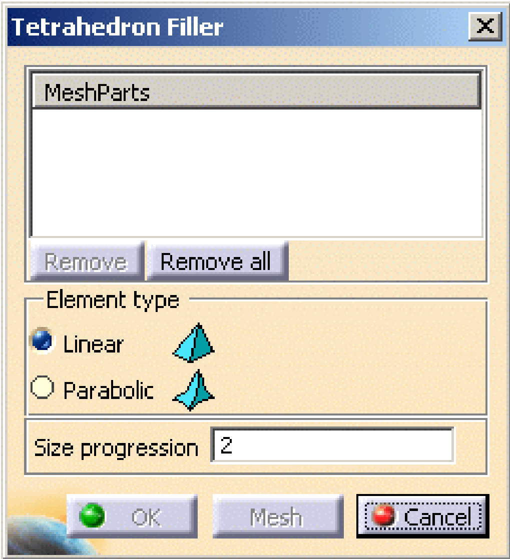
Before You Begin:

- A surface mesh must exist and can be associated or not to a geometry.
  If the surface mesh is associated to a geometry, this geometry can be either a solid or a set of connected faces.
- Open the sample40.CATAnalysis document from the samples directory.
  In this particular example, a surface mesh has been already created.



1. Click the **Tetrahedron Filler** icon from the **Meshing Methods** toolbar.

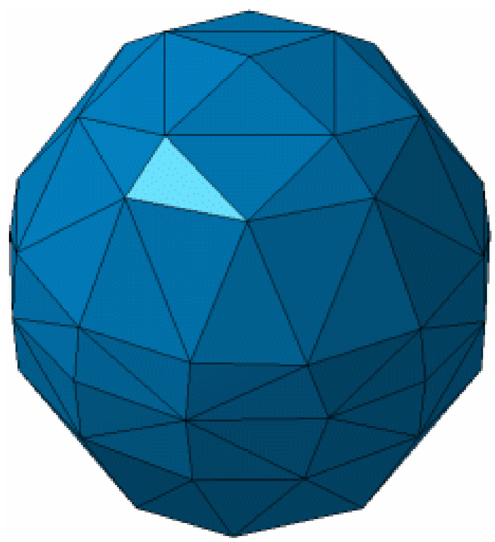
   The Tetrahedron Filler dialog box appears.



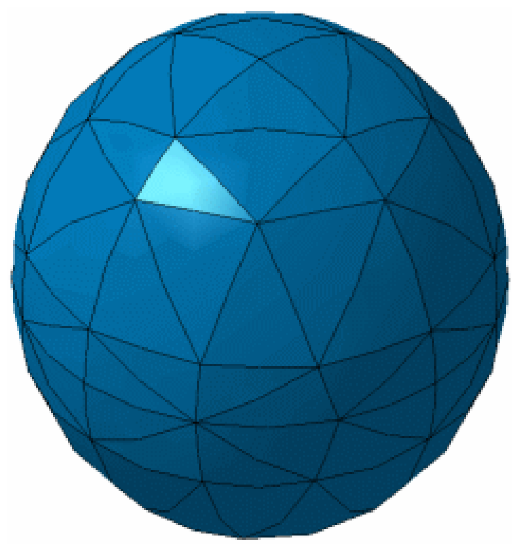
- **MeshParts**
  - **Remove**: lets you remove a selected mesh part.
  - **Remove all**: lets you remove all the mesh parts.

- **Element Type**: lets you choose the type of solid mesh elements. They are independent of the surface mesh elements degree.
  - **Linear**: elements without intermediate nodes, useful to fill a solid with only straight edges.
  - **Parabolic**: elements with intermediate nodes on the geometry, useful to fill a solid with curve edges.

Linear elements type:



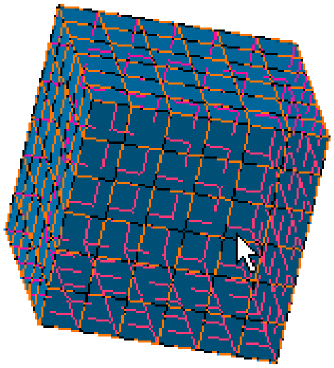
Parabolic elements type:



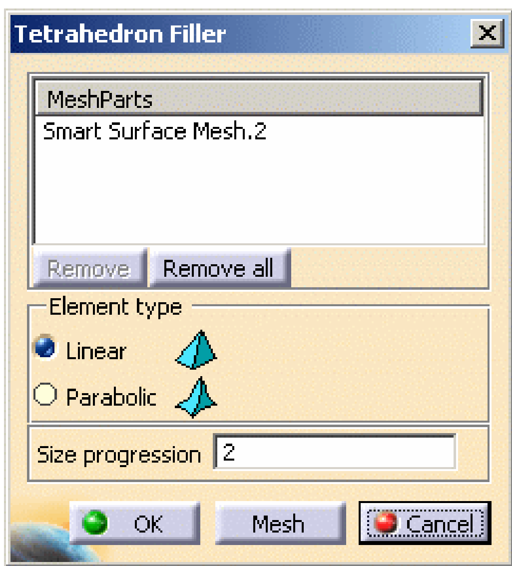
- **Size progression**: factor which lets you dilute the mesh elements inside the solid (if this factor is equal to one, the internal edge sizes are the sizes induced from the surface edge sizes).

2. Select the desired options.

In this particular example, select the **Smart Surface Mesh.2**.



The dialog box is updated:



In the case of a set of connected surface meshes, you have to select different surface mesh part until the global surface mesh is closed (free edges appear in green while the global surface mesh is not closed).

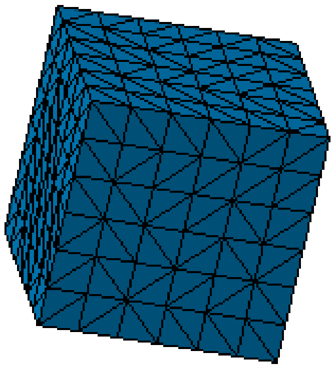
3. Select the **Element type** option and the **Size progression** factor.

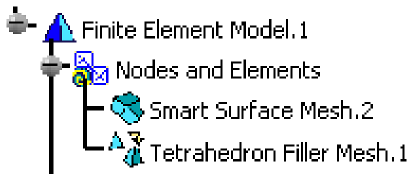
   In this particular example, select the **Linear** option and **2** as **Size progression** factor.

4. Click **Mesh** button in the Tetrahedron Filler dialog box.

   The corresponding quads are automatically split into two triangles using the shortest diagonal. This operation is only a preprocessing of Tetrahedron Filler: the original quads are preserved in the mesh model.

   The mesh is generated on the part and the **Tetrahedron Filler Mesh.1** appears in the specification tree.

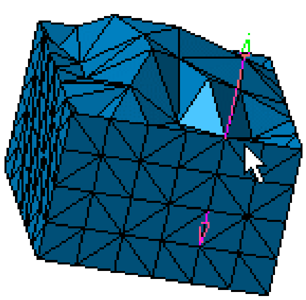




You can visualize the solid mesh using the Cutting Plane functionality
For this:

- Hide the surface mesh.
  For this, right-click the **Smart Surface Mesh.2** object and select the **Hide/Show** contextual menu.

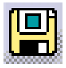
- Select the **Cutting plane** icon  from the Quality Analysis toolbar.
  The Cutting Plane Definition dialog box appears.
  - Select the **Z** options, click the **Reverse** button.
  - Deactivate the **Exact mesh cut** option.
    For more details, please refer to Cutting Plane.



When Parabolic Elements are chosen in the Tetrahedron Filler dialog box, the mesh provided (by clicking the **Mesh** button or by updating the Tetrahedron Filler mesh part) contains elements with intermediate nodes (parabolic tetrahedron). At the moment, those nodes are located in the middle of the corresponding linear tetrahedron edges (i.e. not always on the geometry).

Import / Export Mesh

 Importing the Mesh: Import an existing mesh.

 Exporting the Mesh: Save the mesh into a bulk data file or CATIA V4 file.

Importing the Mesh

This task shows how to import one or more existing Surface meshes from a .dat document.

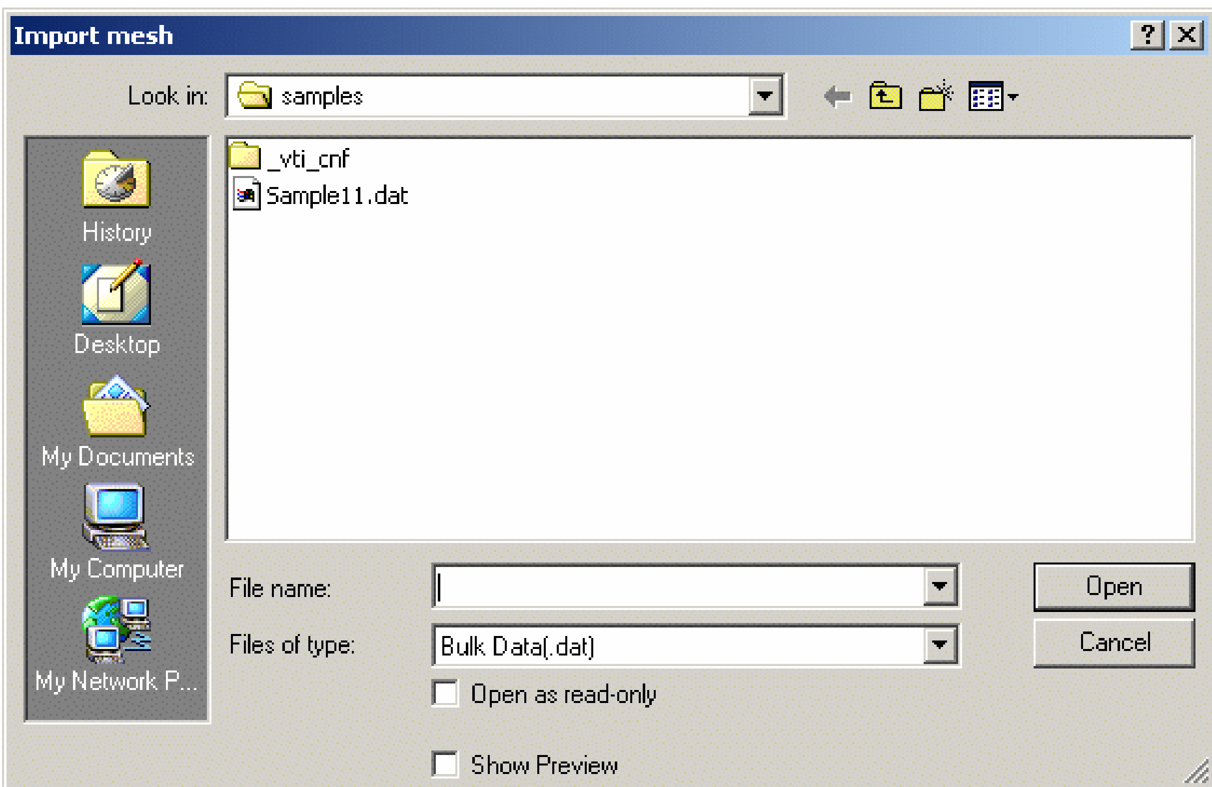
1. Select the **Start -> Analysis -> Advanced Meshing Tools** menu.

   The New Analysis Case dialog box appears.

2. Click **OK** in the New Analysis Case dialog box.

3. Click the **Import Mesh** icon from the **Import/Export** toolbar.

   The Import mesh dialog box appears.

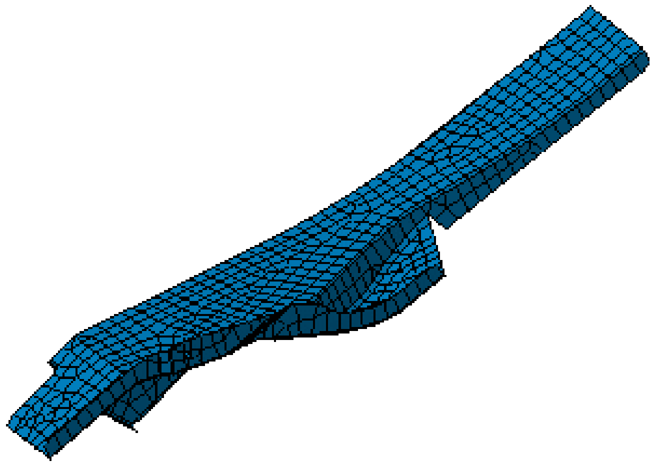


4. Select the **Sample11.dat** document from the samples directory.

   Multi-selection is available in the Import mesh dialog box.

5. Click the **Open** button in the Import mesh dialog box.

The imported mesh appears as shown here:

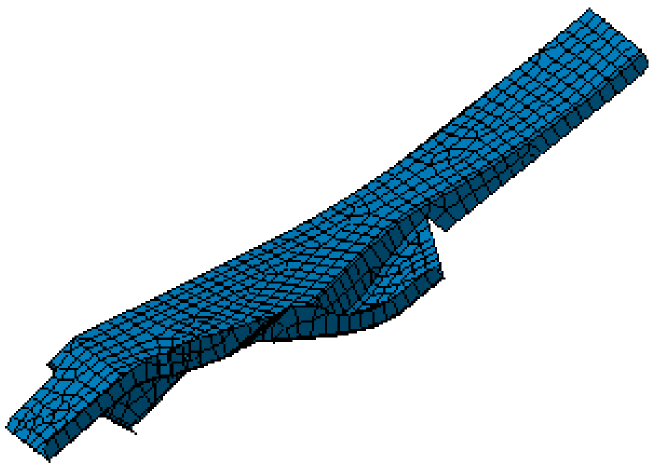


Exporting the Mesh

This task shows how to store a mesh with a new format. These formats can be either **.model** (CATIA V4 files) or **.dat** (Bulk data files).

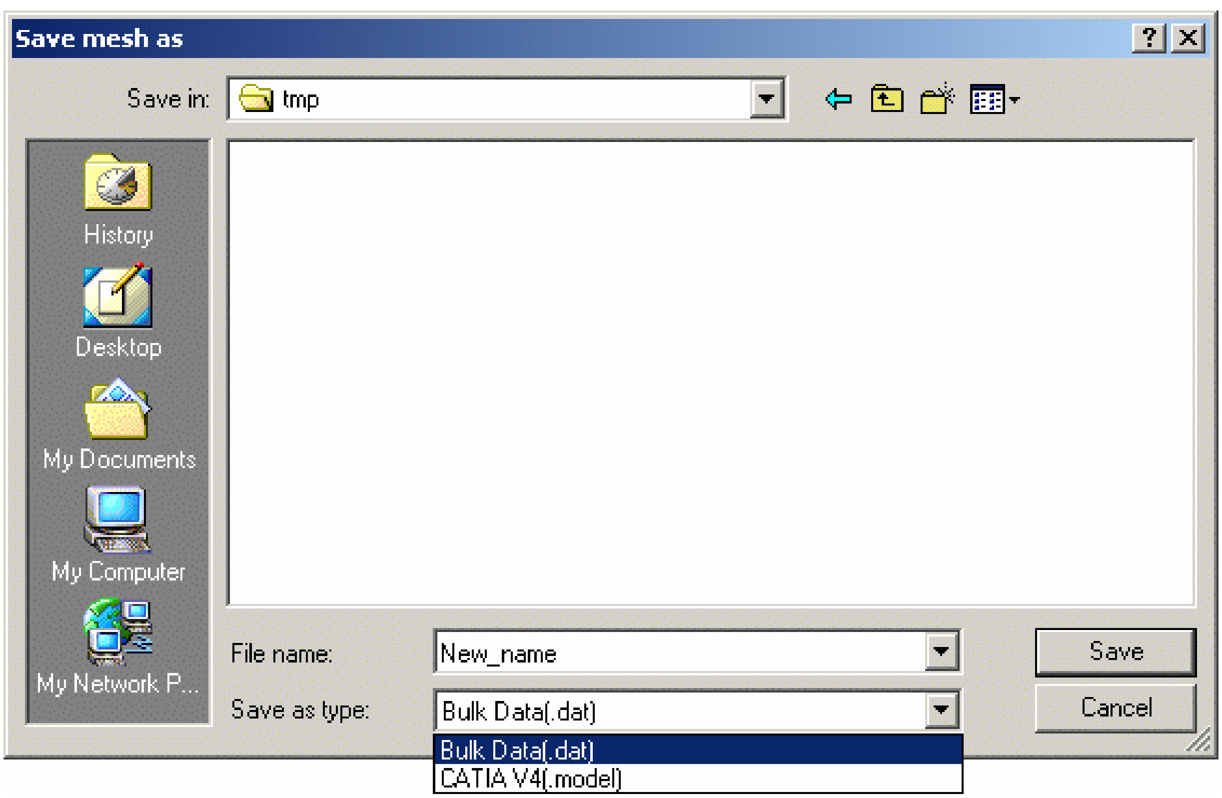
- **.model**: all the connectivities (geometrical shapes) are transferred.
- **.dat**: OCTREE (GPS/EST) and Surface mesh (FMS) parts are transferred.

Open the Sample11.CATAnalysis document from the samples directory.



1. Click the **Export Mesh** icon from the **Import/Export** toolbar.

   The Save mesh as dialog box appears.

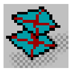


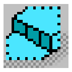
2. Enter the desired options in the dialog box: **File name** and **Save as type**.

3. Click the **Save** button in the Save mesh as dialog box.

The mesh will now be exported as **New_name.dat** in the samples directory.

Meshing Connections

 Meshing Spot Welding Connections: Mesh spot welds that were previously defined in the Generative Structural Analysis workbench.

 Meshing Seam Welding Connections: Mesh a Seam Welding Connection that was created between two parts.

Meshing Spot Welding Connections

This task shows how to mesh spot welding connections.

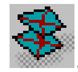
You can mesh the following connections:

- **Point Analysis Connection** and **Point Analysis Connection within one Part** created in the Generative Structural Analysis workbench
- **Joint** created in the Body in White Fastening workbench (except for the joint created using the datum mode)
- **Spot Welding Analysis Connection** created before the V5R12 in the Analysis Connections workbench

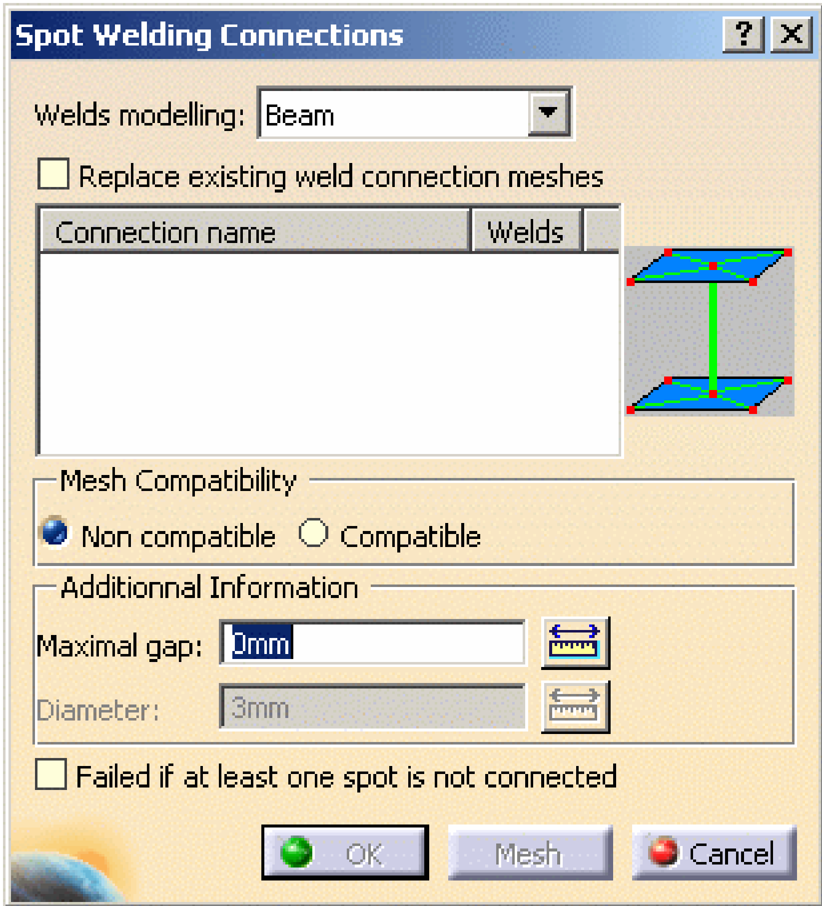
Connections	**Support**
Analysis Connections (from R12) **Point Analysis Connection** **Point Analysis Connection within one Part**	One or two
Body in White Fastener Connections **Joint** (containing or not hemming information)	One or more
Analysis Connections (before R12) **Spot Welding Analysis Connection**	Two

For more details about joint containing hemming information, please refer to the *Automotive Body in White Fastening User's Guide - Creating Joint Bodies*.

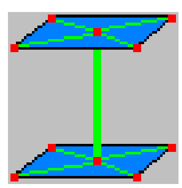
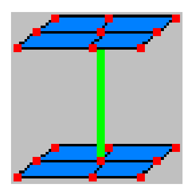
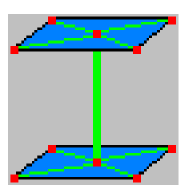
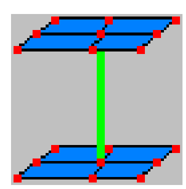
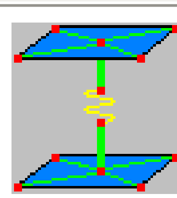
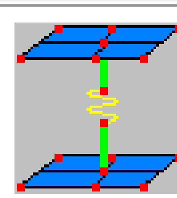
Open the sample26.CATAnalysis document from the samples directory.

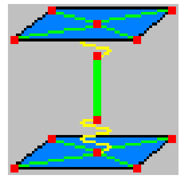
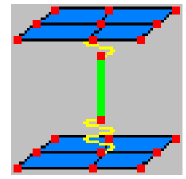
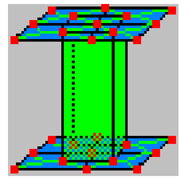
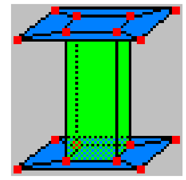
1. Click the **Spot Welding Connections** icon .

The Spot Welding Connections dialog box appears.



- **Welds modelling**:

Welds modelling	Non compatible mesh	Compatible mesh
▪ **Beam**		
▪ **Rigid**		
▪ **Rigid-Spring-Rigid**		

▪ **Spring-Rigid-Spring**		
▪ **Hexahedron**		

- **Replace existing weld connection meshes**: allows you to select a weld spot connection that has been already associated to a mesh part.

  You must activate this option before selecting the weld spot connection.
  If not, a warning message informs you that the weld spot connection has been already meshed.

- **Mesh Compatibility**:
  - **Non compatible**: lets you create nodes by orthogonal projection of the welding points on the two mesh parts.
  - **Compatible**: lets you use existing nodes.

    The welding points must have been previously projected on the two mesh parts.
    For more details, please refer to Adding/Removing Constraints (Specifications).

- **Additional Information**:
  - **Maximal gap:** lets you specify the radius value of a sphere which has the welding spot as center. It must be an intersection between each face and the solid thus defined.
    - The value must be strictly positive.
    - While the **Maximal gap** value is null, the **OK** and the **Mesh** buttons will be not available.
    - If you update spot welding connections created before the V5R12 level without modifying them, the data used to mesh are the same as those used up to the V5R11 level. Errors could be generated (for example: the maximal gap value is too small).
      In this case, you have to edit the connection mesh and to increase the **Maximal Gap** value.
  - **Diameter**: lets you specify the diameter of the hexahedron.

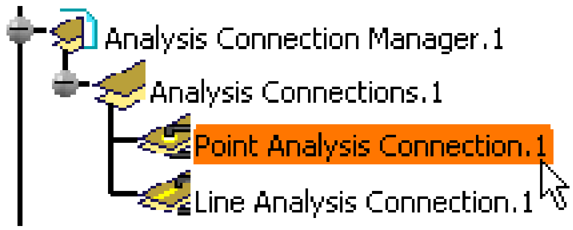
    This option is only available if you selected the **Hexahedron** and the **Non compatible** options.

- **Failed if at least one spot is not connected**: lets you stop the **Update all meshes** process.
  If this option is activated and if you use the **Update all meshes** contextual menu, only the meshes created before the weld spot connection mesh will be

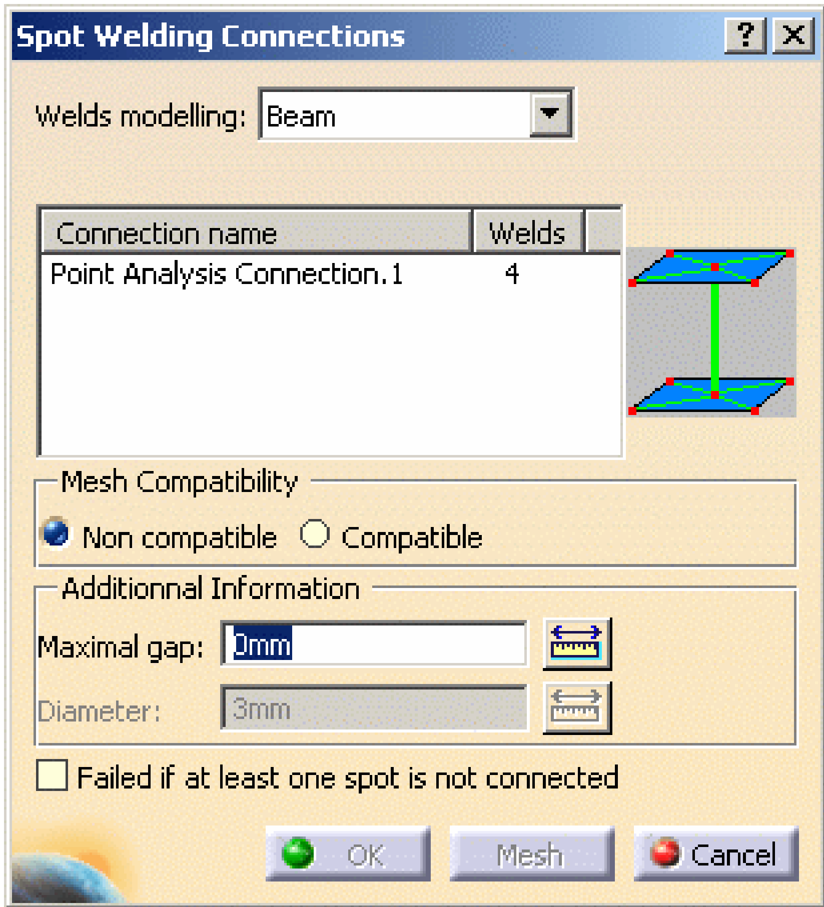
updated.

2. Select the connection you want to mesh in the specification tree.

In this particular example, select the **Point Analysis Connection.1** object in the specification tree (under the **Analysis Connections Manager.1** set).



The Spot Welding Connection dialog box is updated.



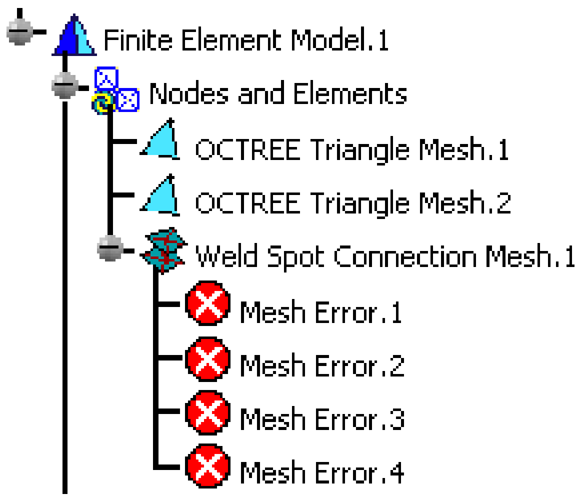
3. Set the desired parameters.

In this particular example:

- select **Beam** as **Welds modelling** option
- select **Non compatible** as **Mesh Compatibility** option
- enter **2mm** as **Maximal gap** value

4. Click **Mesh** in the Spot Welding Connections dialog box.

   Several errors are displayed in the specification tree under the **Weld Spot Connection Mesh.1** object.



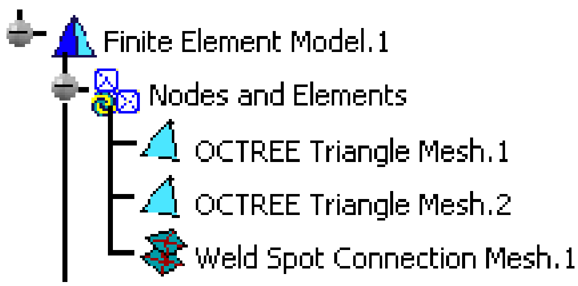
   You have to change the **Maximal gap** value.

5. Double-click the **Weld Spot Connection Mesh.1** object in the specification tree to edit it.

   The Spot Welding Connections dialog box appears.

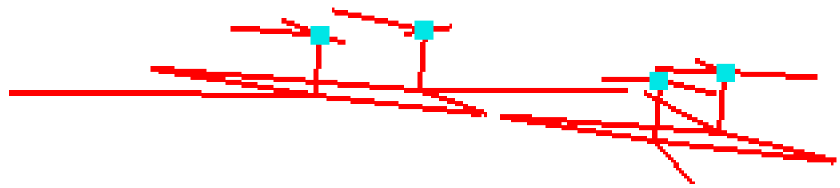
6. Enter **5mm** as **Maximal gap** value and click **Mesh** in the Spot Welding Connections dialog box.

   The connection mesh is now correct.



For a better visualization, you can change the mesh color using the **Properties** contextual menu.

You can also hide the OCTREE Triangle Mesh.1 and OCTREE Triangle Mesh.2 using the **Hide/Show** contextual menu:



7. Change the weld spot connection mesh parameters.

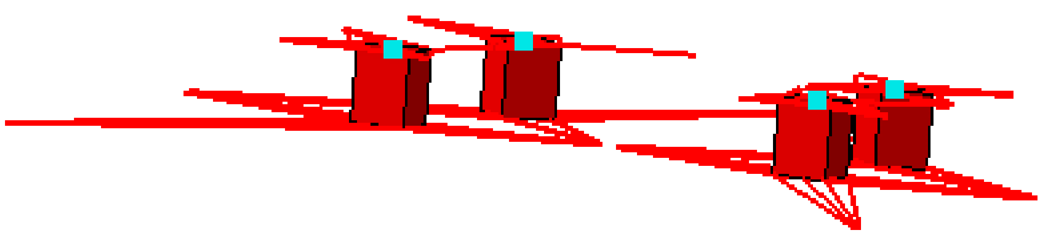
For this, double-click the **Weld Spot Connection Mesh.1** object in the specification tree.

The Spot Welding Connection dialog box appears.

In this particular example:

- select **Hexahedron** as **Welds modelling** option
- enter **3mm** as **Diameter** value
- click the **Mesh** button

If you use the **Hide/Show** contextual menu to hide the OCTREE Triangle meshes, the result will be:



Meshing Seam Welding Connections

This task shows how to mesh a Seam Welding Connection that was created between two parts.

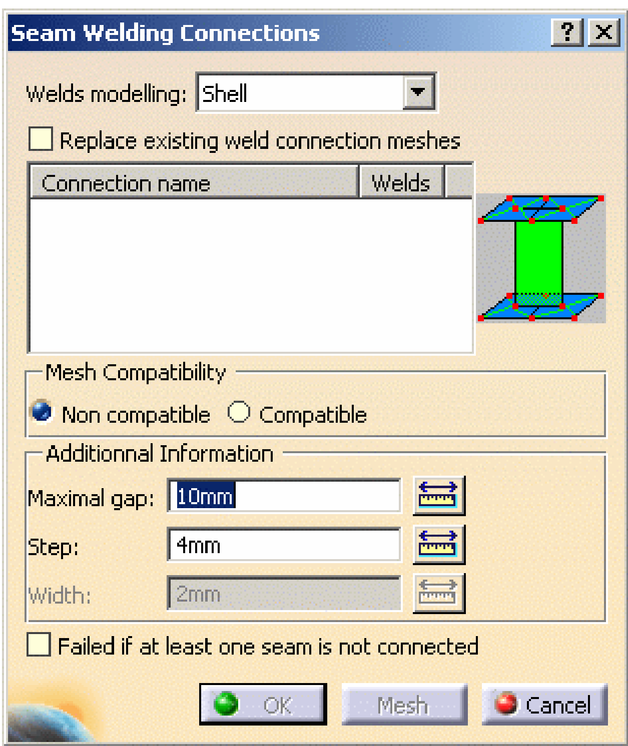
Seam weld can be defined as a wire frame (or 1D) feature.

Connections	**Support**
Analysis Connections (from R12) **Line Analysis Connection** **Line Analysis Connection within one Part**	One or two
Body in White Fastener Connections **Joint**	One or more
Analysis Connections (before R12) **Seam Weld Analysis Connection**	Two

Open the sample26.CATAnalysis document from the sample directory.

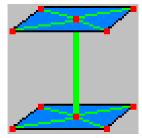
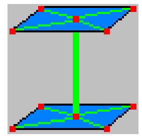
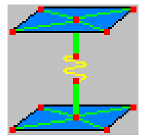
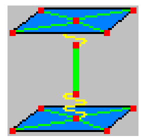
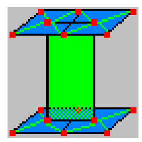
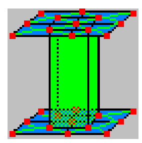
1. Select the **Seam Welding Connections** icon.

The Seam Welding Connections dialog box appears.



- **Welds modelling**:

Welds modelling	Non compatible mesh (default behavior)

▪ **Contact**	
▪ **Rigid**	
▪ **Rigid-Spring-Rigid**	
▪ **Spring-Rigid-Spring**	
▪ **Shell**	
▪ **Hexahedron**	

- **Replace existing weld connection meshes**: allows you to select a weld seam connection that has been already associated to a mesh part.

  You must activate this option before selecting the weld seam connection.
  If not, a warning message informs you that the weld seam connection has been already meshed.

- **Mesh Compatibility**:
  - **Non compatible**: lets you create nodes by orthogonal projection of the seam on the two mesh parts.
  - **Compatible**: lets you use existing nodes.

    The welding seams must have been previously projected on the two mesh parts.
    For more details, please refer to Adding/Removing Constraints (Specifications).

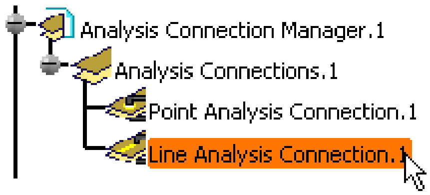
- **Additional Information**:
  - **Maximal gap:** lets you specify the radius value of each sphere created along the welding seam. It must be an intersection between each face and the solid thus defined.
    - The value must be strictly positive.
    - While the **Maximal gap** value is null, the **OK** and the **Mesh** buttons will be not available.
    - If you update seam welding connections created before the V5R12 level without modifying them, the data used to mesh are the same as those used up to the V5R11 level. Errors could be generated (for example: the maximal gap value is too small).
      In this case, you have to edit the connection mesh and to increase the **Maximal Gap** value.
  - **Step**: lets you specify the mesh progression along the welding seam.
  - **Width**: lets you specify the width of the hexahedron.

    This option is only available if you selected the **Hexahedron** option.

- **Failed if at least one spot is not connected**: lets you stop the **Update all meshes** process.
  If this option is activated and if you use the **Update all meshes** contextual menu, only the meshes created before the weld seam connection mesh will be updated.

2. Select the connection you want to mesh in the specification tree.

In this particular example, select the **Line Analysis Connection.1** object in the specification tree (under the **Analysis Connection Manager** set).



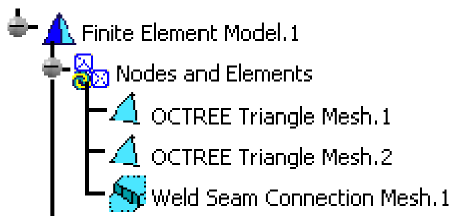
3. Set the desired parameters.

In this particular example:

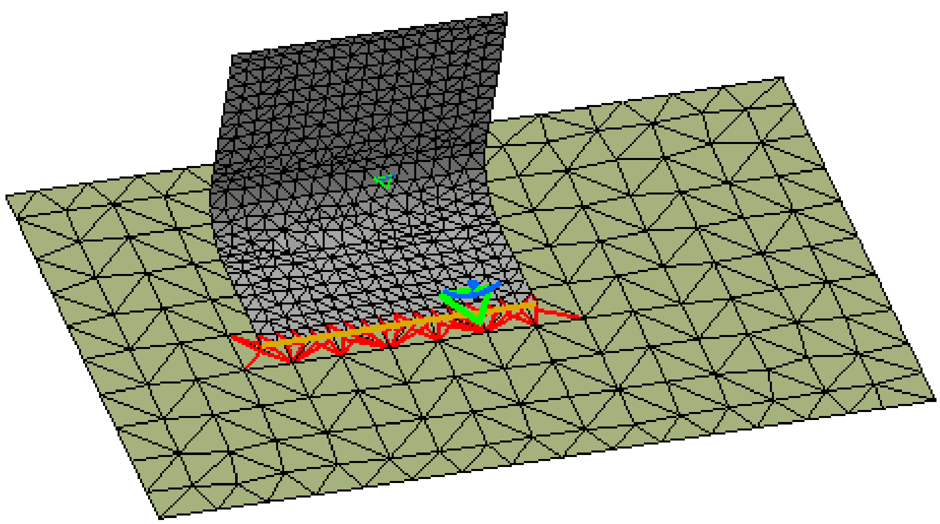
- select **Contact** as **Welds modelling** option
- select **Non compatible** as **Mesh Compatibility** option
- enter **3mm** as **Maximal gap** value
- enter **5mm** as **Step** value

4. Click **Mesh** in the Seam Welding Connections dialog box.

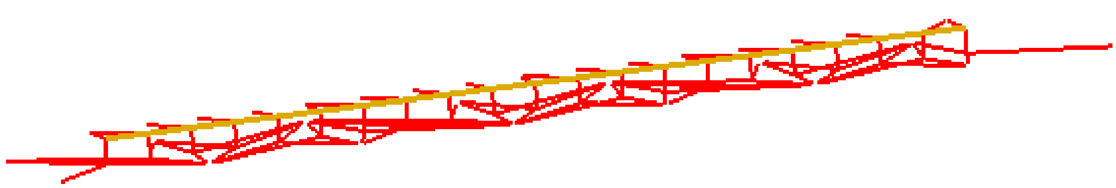
A **Weld Seam Connection Mesh.1** object is displayed in the specification tree.



For a better visualization, you can change the mesh color using the **Properties** contextual menu.



You can also hide the OCTREE Triangle Mesh.1 and OCTREE Triangle Mesh.2 using the **Hide/Show** contextual menu:



5. Double-click the **Weld Seam Connection Mesh.1** object in the specification tree to edit it.

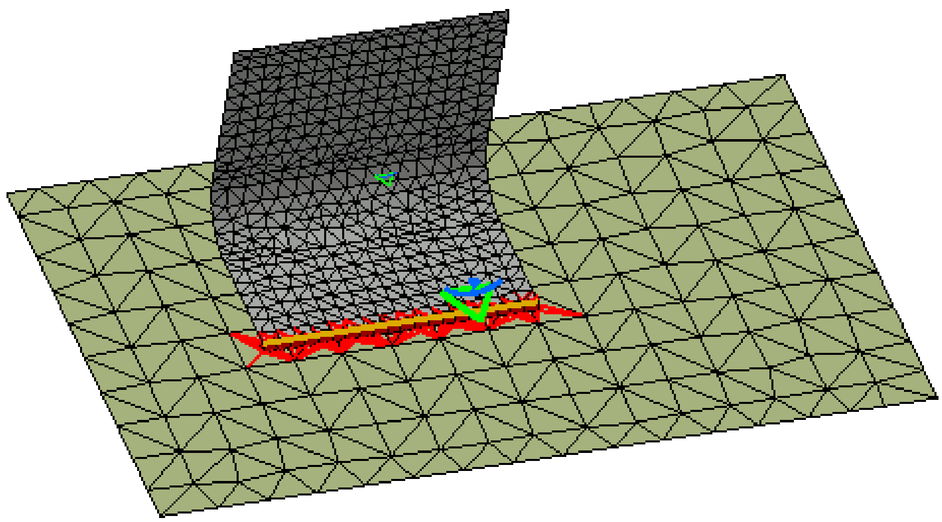
   The Seam Welding Connections dialog box appears.

6. Change the seam weld connection mesh parameters.

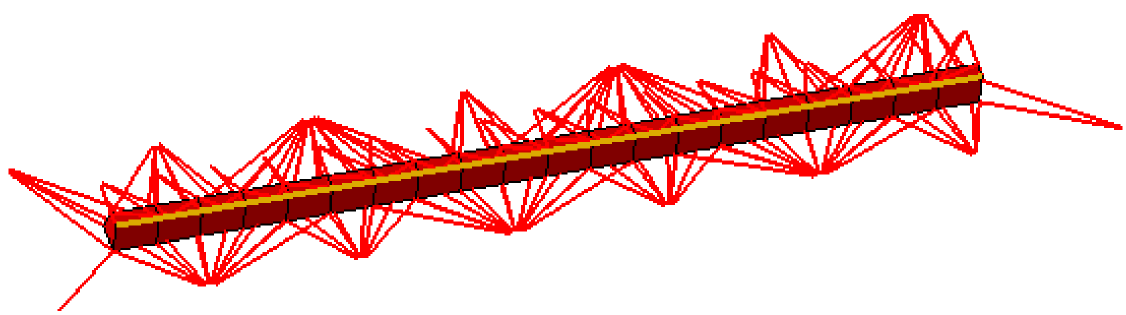
   In this particular example:

   - select **Hexahedron** as **Welds modelling** option
   - enter:
     - **3mm** as **Maximal gap** value
     - **5mm** as **Step** value
     - **3mm** as **Width** value

7. Click **Mesh** in the Seam Welding Connections dialog box.



You can hide the OCTREE Triangle Mesh.1 and OCTREE Triangle Mesh.2 using the **Hide/Show** contextual menu:



Quality Analysis

The quality analysis tools are available at any time in the **Advanced Meshing Tools** workbench and the **Surface Meshing** workshop.

Displaying Free Edges: Perform quick visualization of incompatible mesh

Checking Intersections / Interferences: Check meshes to detect intersections and/or interferences.

Switching to Standard/Quality Visualization Mode: Switch to element standard/quality visualization mode.

Analyzing Element Quality: Analyze element quality, select and tune quality criteria.

Cutting Plane: Cut plane visualization mode.

Elements Orientation: Elements Orientation visualization mode.

Returning Mesh Part Statistics: Return mesh part statistics dynamically (only available in the **Surface Meshing** workshop)

Displaying Free Edges

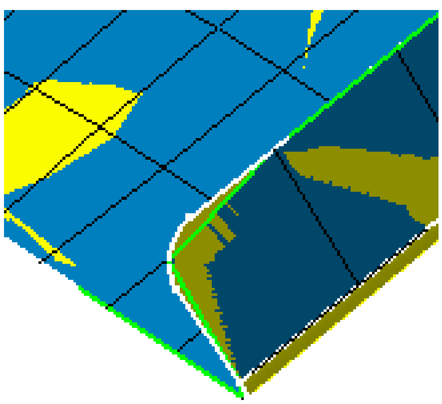
This task shows how to display the free edges of the mesh and thereby quickly visualize incompatible mesh.

Open the Sample05.CATAnalysis document from the samples directory.

The mesh elements are assigned green, yellow or red colors.
For more information, please refer to Before You Begin.

1. Click the **Free Edges** icon .

   The free edges are automatically drawn in green and allow quickly checking mesh validity.



Checking Intersections / Interferences

This task shows how to check intersections / interferences on meshed 2D geometry (surface element).

Intersection means standard geometrical intersection.

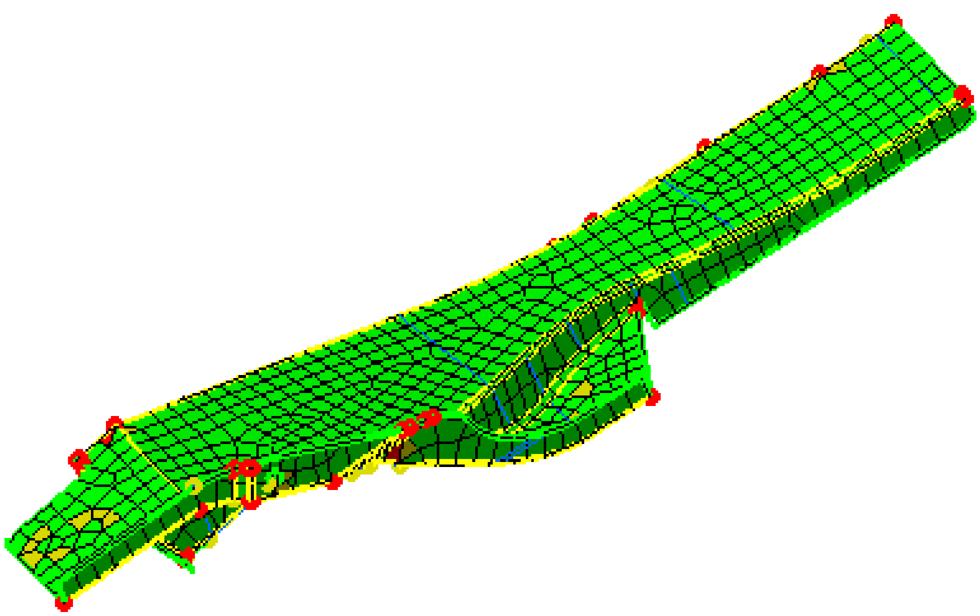
Interference means that the elements generated are positioned beyond a given distance (or clearance). This let's you take into account the real width of the geometry when detecting interferences.

You can use the sample05.CATAnalysis document from the samples directory.

Before You Begin:

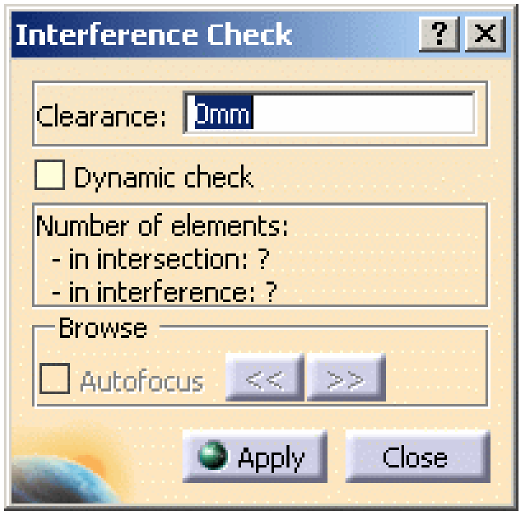
- You have to enter the Surface Meshing workshop.
  For this, double-click **Smart Surface Mesh.1** feature from the specification tree (below **Nodes and Elements** feature) and then click **YES** (**Continue anyway?**) in the warning box.
- You have to mesh the surface.

  For this, click the **Mesh The Part** icon from the **Execution** toolbar.
  You will then click **OK** in the Mesh The Part dialog box.
  For more information, please refer to Surface Mesher.



1. Click the **Intersections / Interferences** icon in the **Mesh Analysis Tools** toolbar.

The Interference Check dialog box appears.



- **Clearance**: lets you define the clearance from which you decide that an interference exists.
- **Dynamic check**: lets you modify the mesh and dynamically visualize the corresponding interferences.
- **Number of elements**:
  - **in intersection**:
  - **in interference**:
- **Autofocus**: lets you focus on interference area.

Note that the Interference visualization mode is different.

The **Clearance** field allows you defining the clearance from which you decide that an interference exists.

The clearance maximum value should not be greater than the mesh size.

Note that:

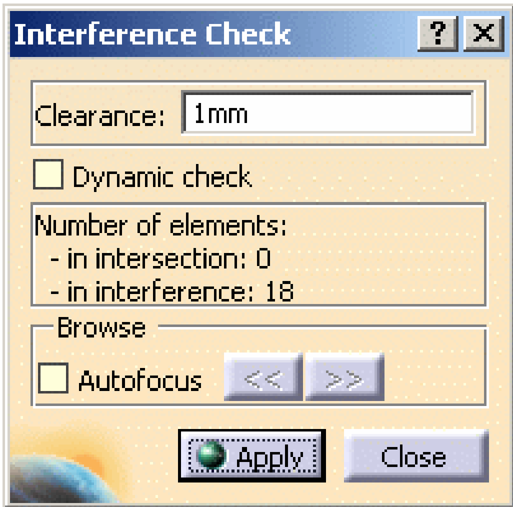
- If the value is equal to zero, only the existing intersections will be visualized.
- If the value is greater than zero, both the existing intersections and interferences will be visualized
- The **Dynamic check** field allows you modifying the mesh and dynamically visualize the corresponding interferences.

2. Enter the desired **Clearance** value in the Interference Check dialog box.

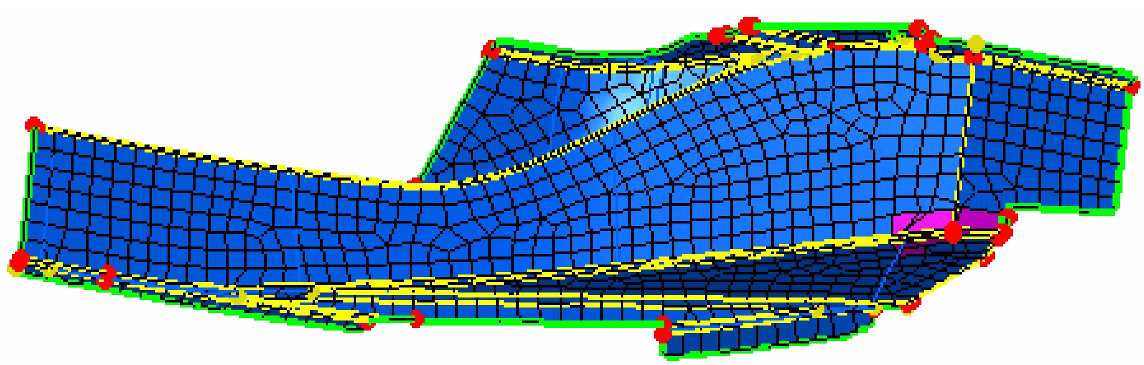
   In this particular example, enter **1mm**.

3. Click **Apply** in the Interference Check dialog box.

Information on the number of the intersection elements and interference elements automatically appears in the dialog box.



Note that the **Autofocus** option is available.

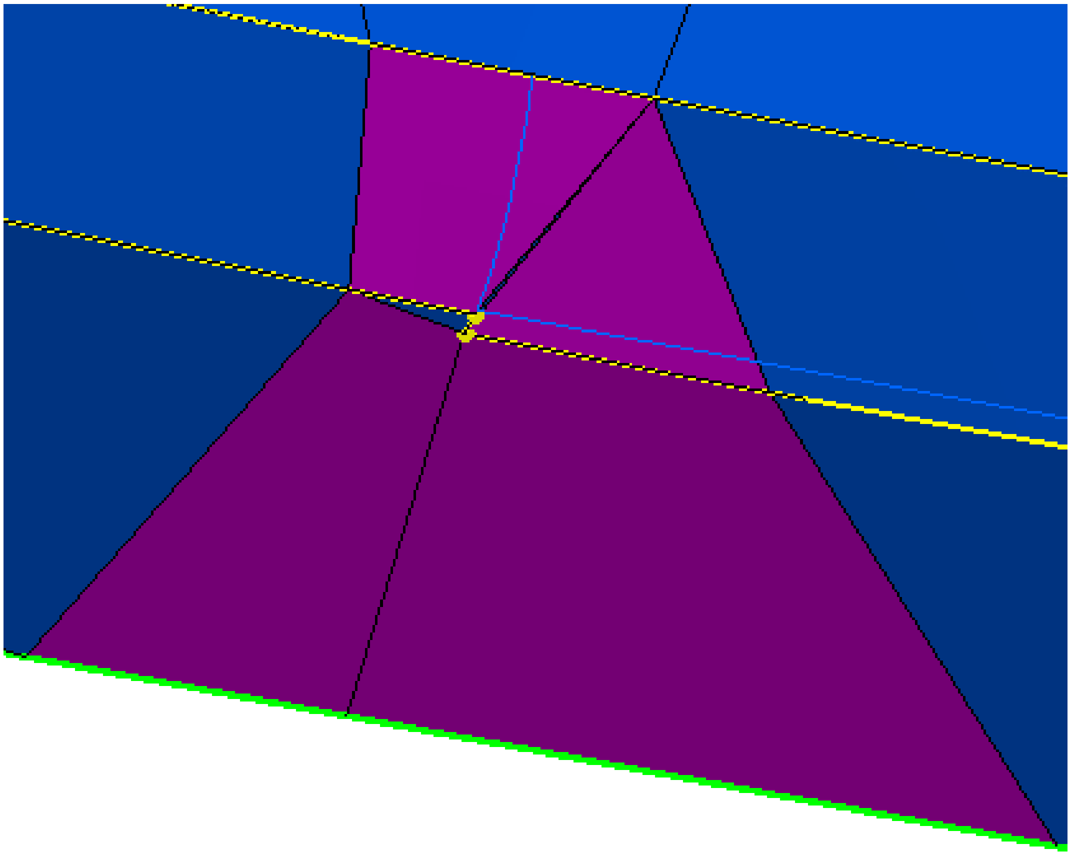


These intersection elements and interference elements also appear on the geometry:

- the red color is assigned to intersection elements.
- the pink color is assigned to interference elements.

4. Activate the **Autofocus** option and click **Apply** in the Interference Check dialog box.

The first interference area is displayed:



5. Click the >> button to display an other area.

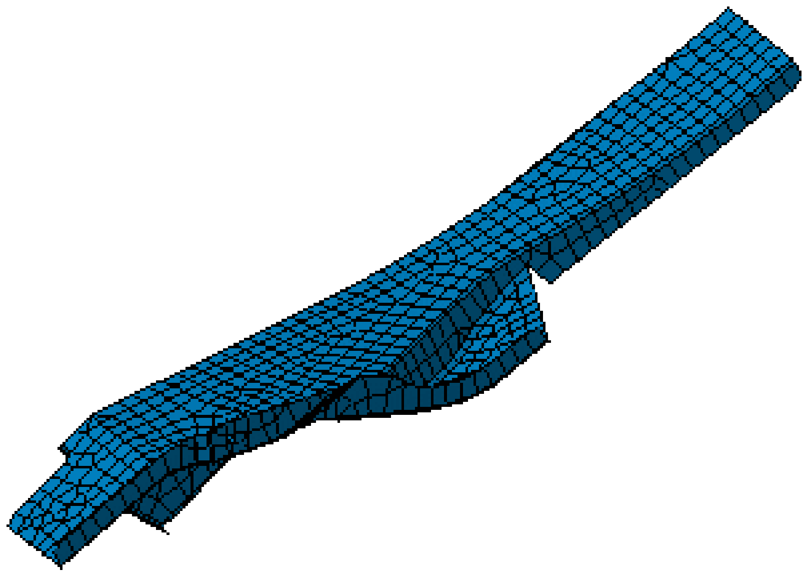
6. Click **Close** in the Interference Check dialog box.

Note that if you need to visualize the intersection elements exclusively, use the Wireframe (NHR) visualization type as the command is still active.

Switching to Standard/Quality Visualization Mode

This task shows how to switch either to the Standard or Quality visualization mode.

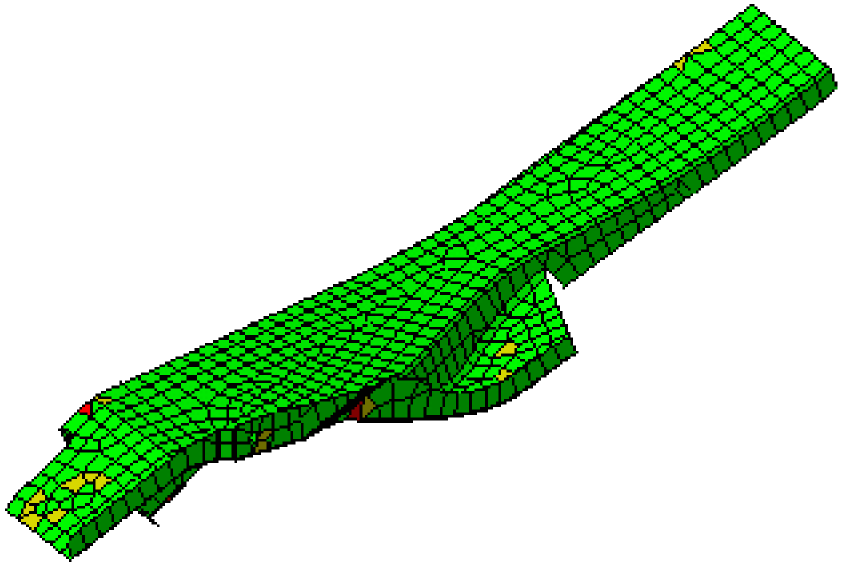
Open the Sample11.CATAnalysis document from the samples directory.



The default visualization mode is the Standard Visualization mode.

1. Click the **Quality Visualization** icon in the Visu Mode toolbar.

   The CATAnalysis document now appears as shown here:



The mesh elements are assigned green, yellow or red colors.

For more information, please refer to Before You Begin.

Analyzing Element Quality

This task shows how to analyze the quality of the elements relatively to given criteria, on one or more parts.

Open the Sample11.CATAnalysis document from the samples directory.

The mesh elements are assigned green, yellow or red colors.
For more details, please refer to Before You Begin.

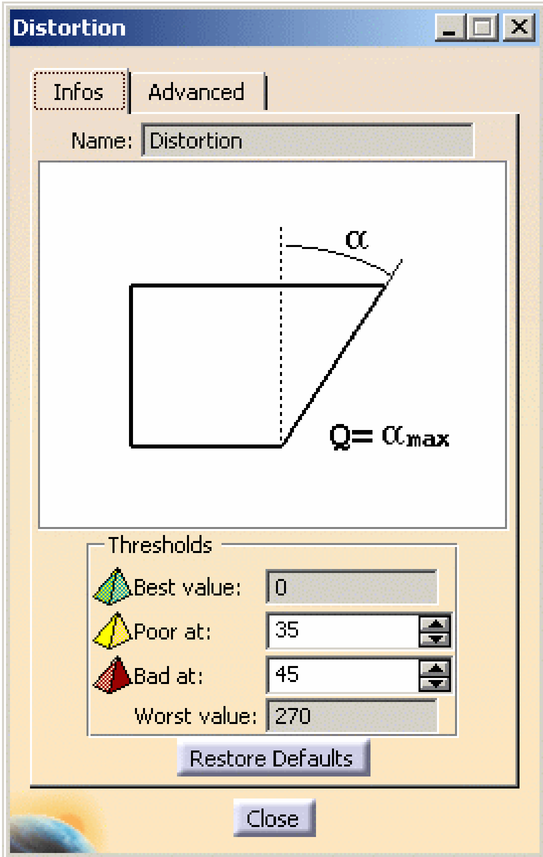
1. Click the **Quality Analysis** icon in the Quality Analysis toolbar.

   The Quality Analysis dialog box appears for visualizing element quality in accordance with selected criteria.

   Note that each time you will modify the criteria list, you need to select the **Apply** button to have the below detailed information (Statistic, Report, Select, Worst element) as well as the Finite Element Model visualization updated.

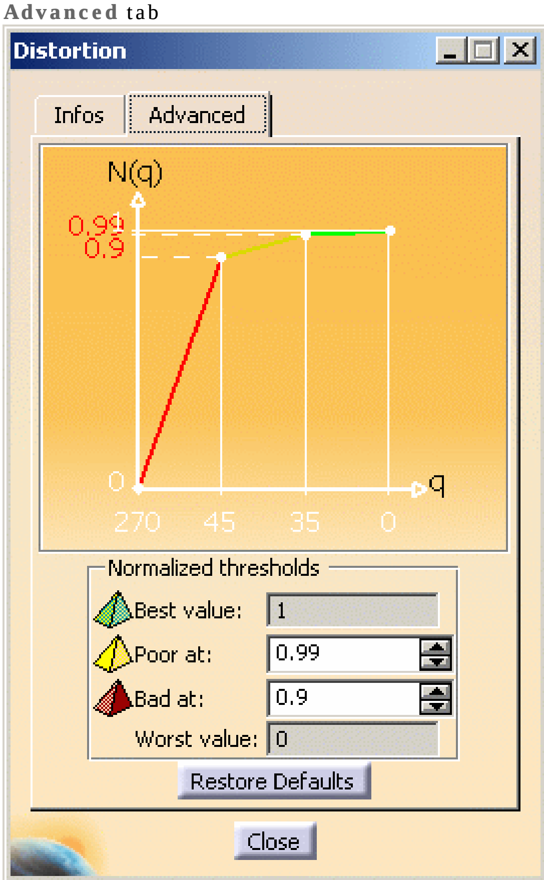
- **Select** tab
  A filter that let you select the criteria you will use for element analysis.

  Clicking the ... switch button, you will get the below basic and advanced information about a given criterion.
  - **Infos** tab:



- **Name:** the name of the selected criterion.
  A drawing as well as a definition is also provided.
- **Threshold:** the values at which you decide the element is good (green color), poor (yellow color) or bad (red color).
  You can **Restore Defaults** you previously entered.

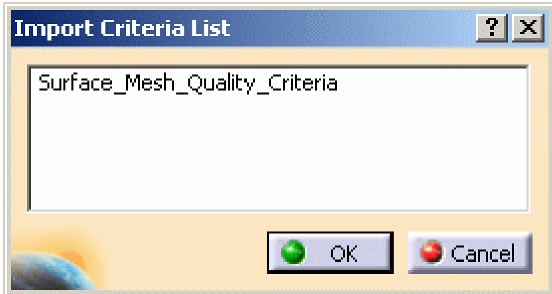
- **Advanced** tab



- A graphic is provided that illustrate the criteria normalized threshold.

- **Normalized thresholds:** allows modulating the weight of a criterion. The computation of the normalized quality (criteria normalized function) results from the interpolation of the quality with both the Poor and the Bad weight that can be customized.
  You can **Restore Defaults** you previously entered.

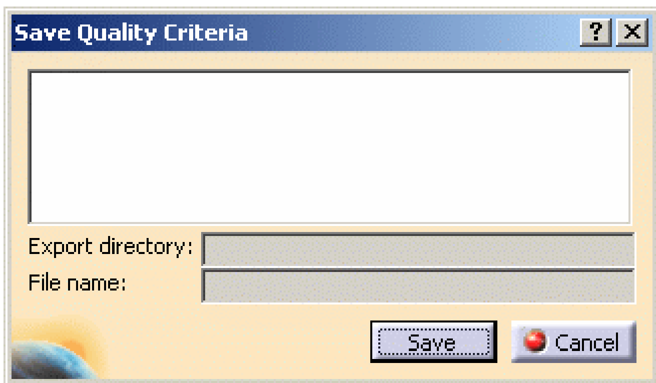
- **Import Criteria** 
  Lets you choose a criteria configuration from a predefined list.



For more details about managing the **Import Criteria** predefined list, please refer to Quality.

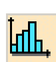
- **Export Criteria** 
  Lets you save a criteria configuration to a predefined directory.

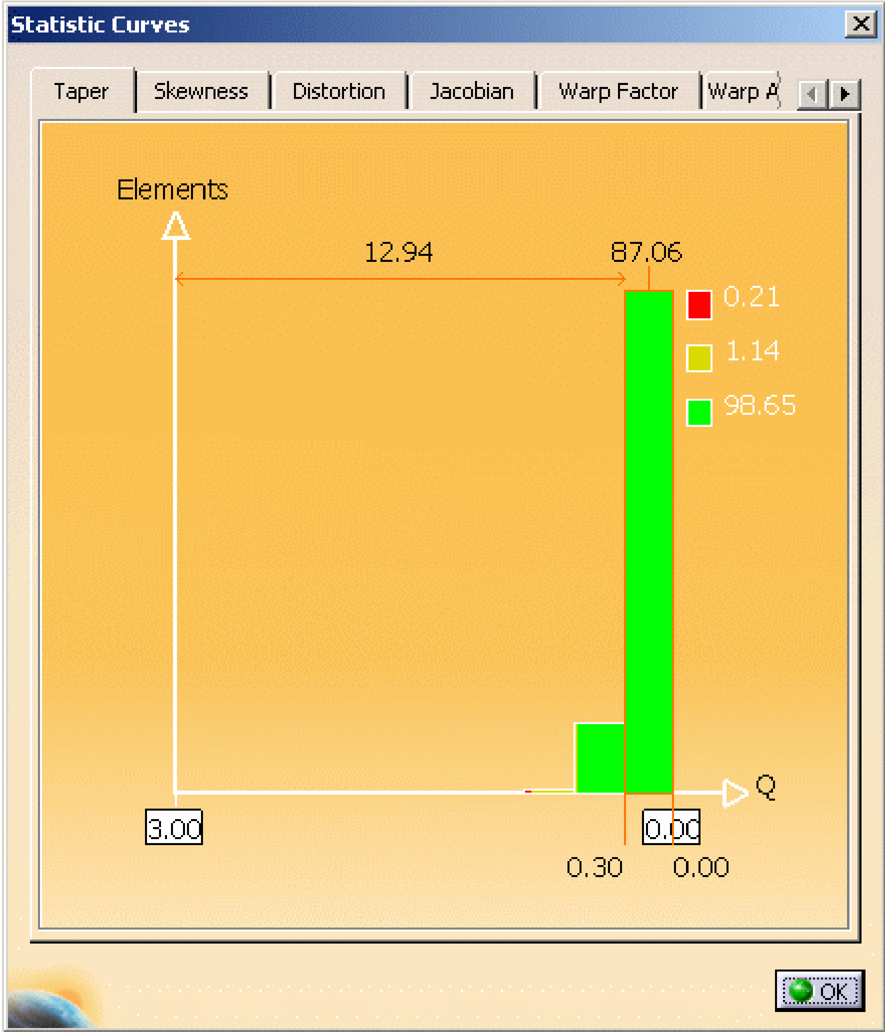
By default, there is no pre-defined directory.



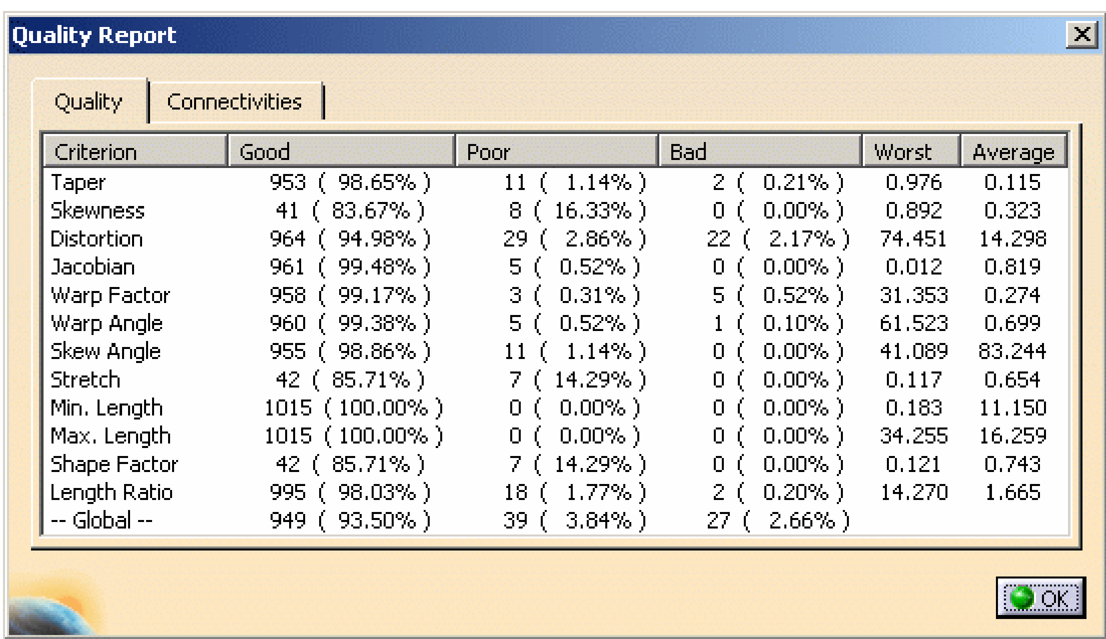
You have to customize the quality settings: you can use the **Export Criteria** option only if a default directory is defined in the Options dialog box.

For more details about managing the **Export Criteria** default directory, please refer to Quality.

- **Show Statistic Curves** 
  In this example, let's choose the **Distortion** tab: you use the arrow to visualize the stated percentage for a given element threshold.
  Note that the information displayed in the Statistic Curves box depends on the options you pre-defined in the Quality Analysis dialog box (**Options** tab).



- **Show Quality Report**
  If you activate this button, the Quality Report dialog box will give you access to quality or connectivity information.
  - **Quality** tab
    Information on **Good**, **Poor**, **Bad**, **Worst** or **Average** statistic for all the previously selected criteria.


Quality Report

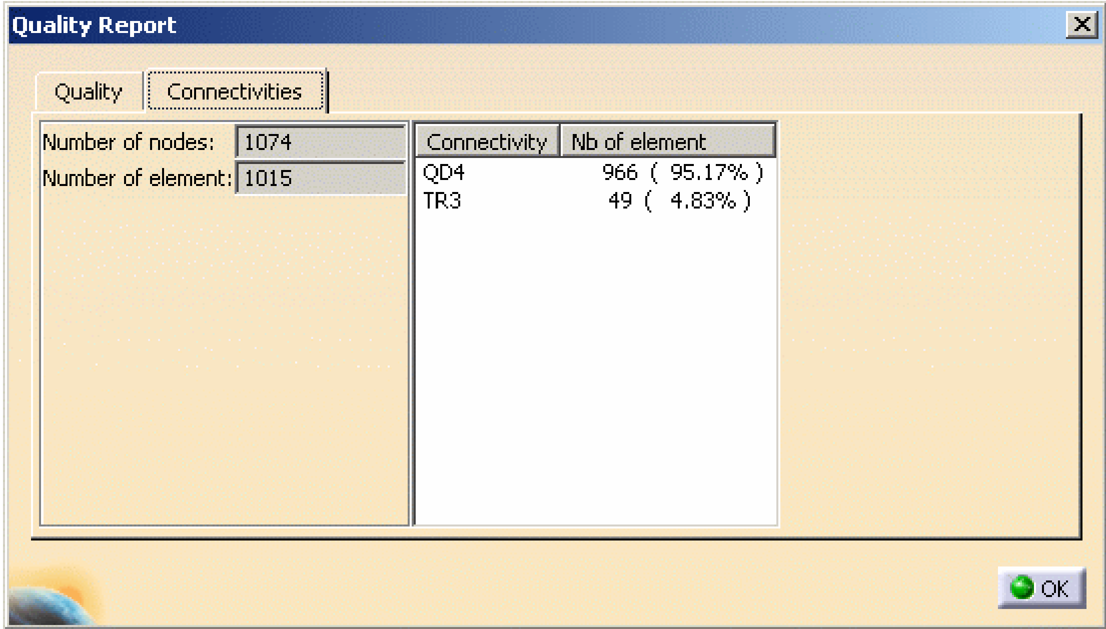
Quality | Connectivities

Criterion	Good	Poor	Bad	Worst	Average
Taper	953 (98.65%)	11 (1.14%)	2 (0.21%)	0.976	0.115
Skewness	41 (83.67%)	8 (16.33%)	0 (0.00%)	0.892	0.323
Distortion	964 (94.98%)	29 (2.86%)	22 (2.17%)	74.451	14.298
Jacobian	961 (99.48%)	5 (0.52%)	0 (0.00%)	0.012	0.819
Warp Factor	958 (99.17%)	3 (0.31%)	5 (0.52%)	31.353	0.274
Warp Angle	960 (99.38%)	5 (0.52%)	1 (0.10%)	61.523	0.699
Skew Angle	955 (98.86%)	11 (1.14%)	0 (0.00%)	41.089	83.244
Stretch	42 (85.71%)	7 (14.29%)	0 (0.00%)	0.117	0.654
Min. Length	1015 (100.00%)	0 (0.00%)	0 (0.00%)	0.183	11.150
Max. Length	1015 (100.00%)	0 (0.00%)	0 (0.00%)	34.255	16.259
Shape Factor	42 (85.71%)	7 (14.29%)	0 (0.00%)	0.121	0.743
Length Ratio	995 (98.03%)	18 (1.77%)	2 (0.20%)	14.270	1.665
-- Global --	949 (93.50%)	39 (3.84%)	27 (2.66%)		

OK

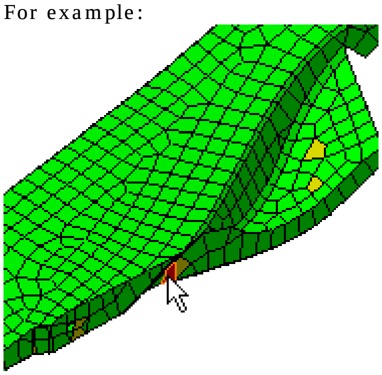
  - **Connectivities** tab

Information on the mesh elements number, type, node in the current Finite Element model:

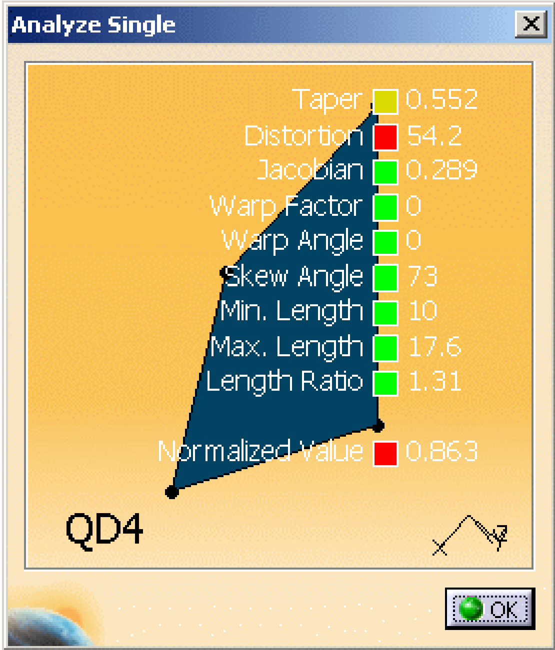


- **Analyse An Element** 
  You can also select one element have this element analyzed.

  For example:

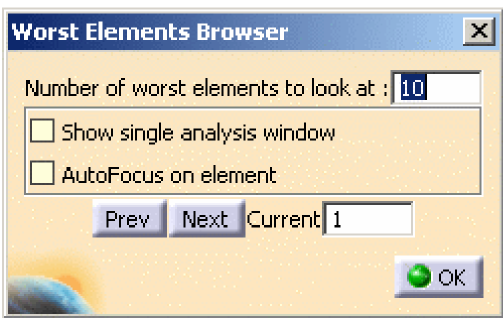
  

  The Analyze Single box appears with the quality type (**Good**, **Poor**, **Bad**) and value for all the selected criteria assigned to this particular element.



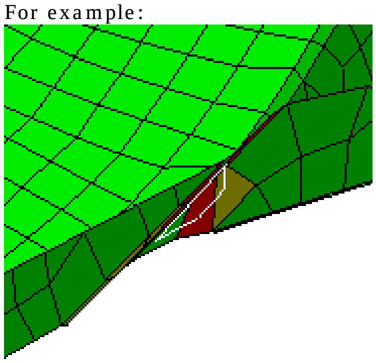
- **Worst Elements Browser** 

  The Worst Elements Browser dialog box appears. You just need to click the **Next** button to have the worst elements zoomed.



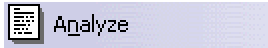
  - **Show single analysis window**: the Single Analysis window of the current worst element remains displays and updates as you browse a Next element.
  - **AutoFocus on element**: the worst browsed element is automatically auto focused.

  For example:



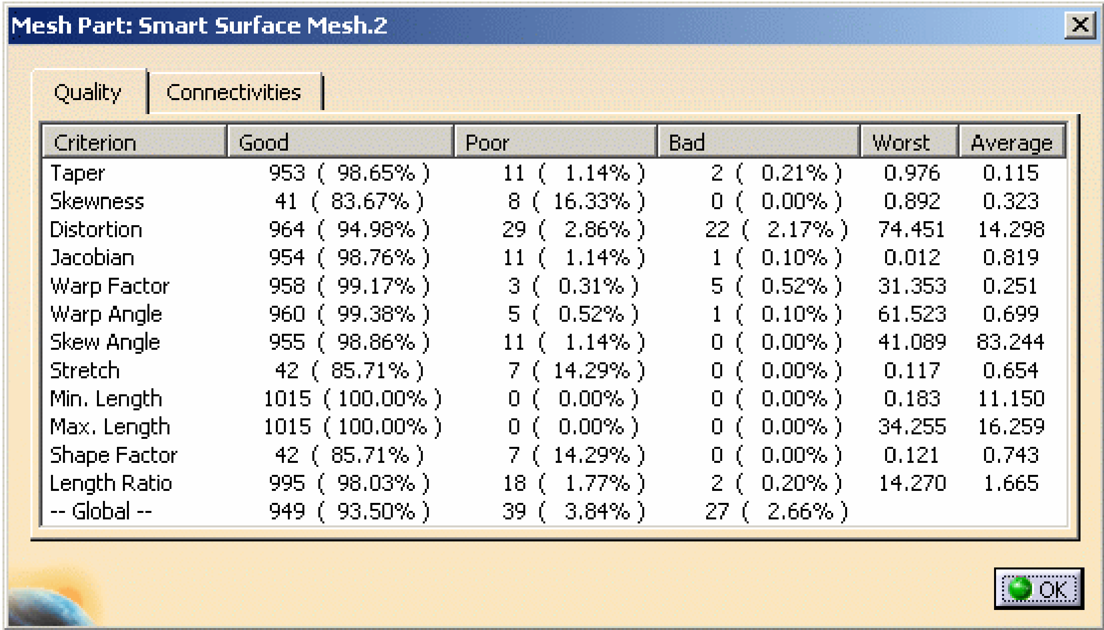
2. Select the desired criteria from the Quality Analysis dialog box.

3. Analyze the quality of the other mesh part.

For this, right-click on the **Smart Surface Mesh Part.2** feature in the specification tree and select the **Analyze** contextual menu 

.

The Mesh Part dialog box now appears with details on Quality and Connectivity report for the selected part:



Criterion	Good	Poor	Bad	Worst	Average
Taper	953 (98.65%)	11 (1.14%)	2 (0.21%)	0.976	0.115
Skewness	41 (83.67%)	8 (16.33%)	0 (0.00%)	0.892	0.323
Distortion	964 (94.98%)	29 (2.86%)	22 (2.17%)	74.451	14.298
Jacobian	954 (98.76%)	11 (1.14%)	1 (0.10%)	0.012	0.819
Warp Factor	958 (99.17%)	3 (0.31%)	5 (0.52%)	31.353	0.251
Warp Angle	960 (99.38%)	5 (0.52%)	1 (0.10%)	61.523	0.699
Skew Angle	955 (98.86%)	11 (1.14%)	0 (0.00%)	41.089	83.244
Stretch	42 (85.71%)	7 (14.29%)	0 (0.00%)	0.117	0.654
Min. Length	1015 (100.00%)	0 (0.00%)	0 (0.00%)	0.183	11.150
Max. Length	1015 (100.00%)	0 (0.00%)	0 (0.00%)	34.255	16.259
Shape Factor	42 (85.71%)	7 (14.29%)	0 (0.00%)	0.121	0.743
Length Ratio	995 (98.03%)	18 (1.77%)	2 (0.20%)	14.270	1.665
-- Global --	949 (93.50%)	39 (3.84%)	27 (2.66%)		

4. Simultaneously perform the operation on all the parts you want to analyze.

   You will open as many Mesh Part dialog boxes as desired.

5. Click **OK** in the Mesh Part when you are satisfied with the information.

   You can perform an independent check and validation for each part in an assembly.

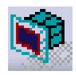
Cutting Plane

This task will show you how to cut a surface or a solid mesh part.

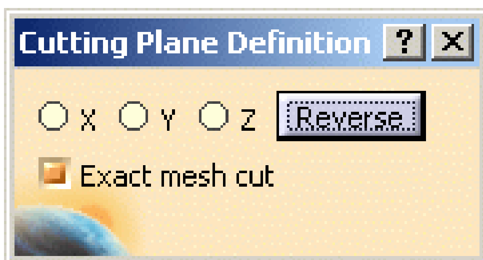
You have to hide the geometry.

Open the sample40.CATAnalysis document from the samples directory.

Before You Begin:
Hide the geometry.
For this, right-click **Link Manager.1** in the specification tree and select the **Hide/Show** contextual menu.

1. Click the **Cutting plane** icon .

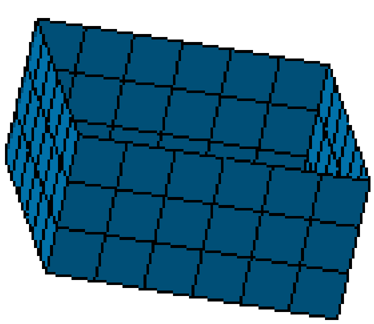
   The Cutting Plane Definition dialog box appears.



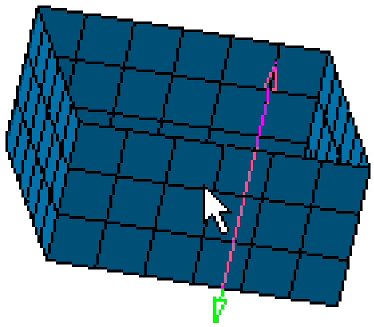
- **X, Y, Z** options: lets you select the normal of which the direction is the X, Y or Z axis
- **Reverse**: lets you reverse the cutting part
- **Exact mesh cut**: if this option is activated, you can cut the mesh exactly where you drag the cursor. If this option is not activated, the cutting follows the mesh elements.

2. Select the desired parameters.

   In this particular example, select **Z** option and click the **Reverse** button.

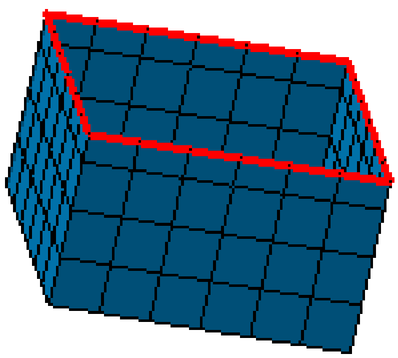


3. Select the section and drag the cursor to the desired position.

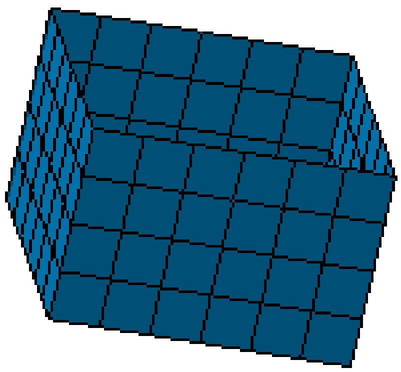


With the same cursor displacement:

Exact mesh cut activated



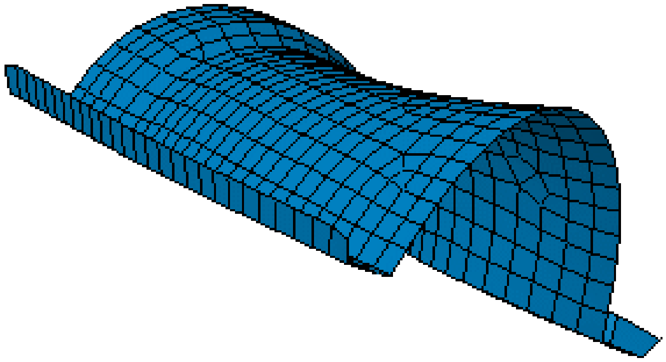
Exact mesh cut not activated

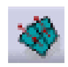


Displaying Elements Orientation

This task shows you how to visualize the elements orientation according to the normal direction of the finite element.

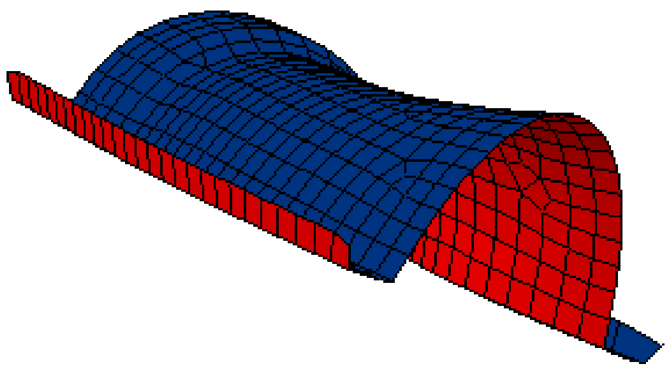
Open the Sample01.CATAnalysis document from the samples directory.



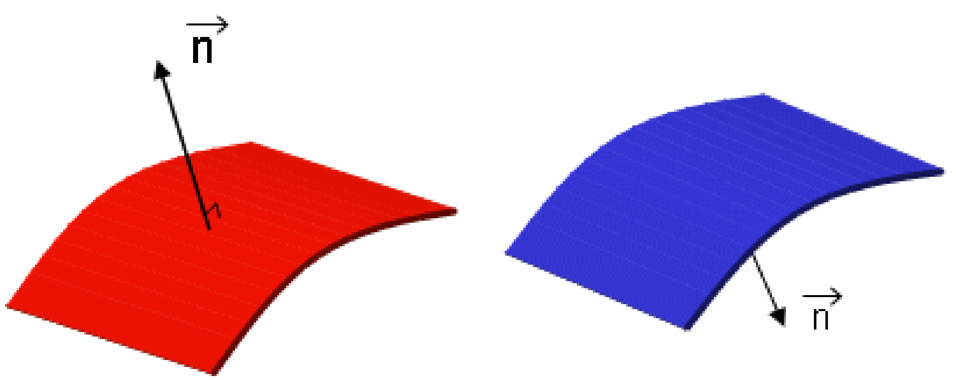
1. Click the **Element Orientation** icon  from the Quality Analysis toolbar.

   The elements orientation appears as shown below.

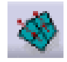
   - red color indicates the positive orientation
   - dark blue color indicates the negative orientation



- The positive side supports the finite element normal.



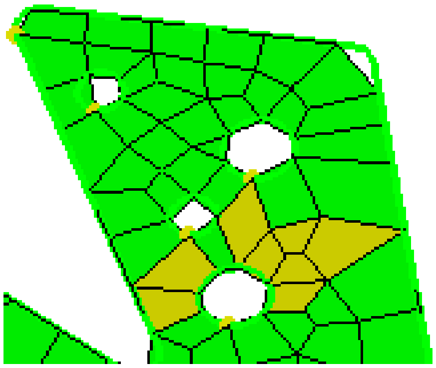
- The generation ordering of the nodes determines which side of the element is the positive one and which is the negative one.

2. Click the **Element Orientation** icon  from the Quality Analysis toolbar to exit the element orientation visualization mode.

Returning Mesh Part Statistics

This task shows how to return mesh part statistics dynamically.

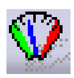
Open the sample03.CATAnalysis document from the samples directory.



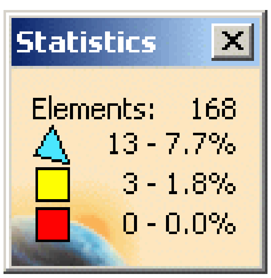
Before You Begin:

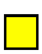
- **Enter the smart surface triangle quadrangle mesher**.
  For this, double-click **Smart Surface Mesh.1** feature from the specification tree (below **Nodes and Elements** feature) and then click **YES** (**Continue anyway?**) in the warning box.
- **Mesh the surface.**

  For this, click the **Mesh The Part** icon  from the Specification Tools toolbar.
  You will then click **OK** in the Mesh The Part dialog box. See Setting Global Parameters for more information on this step.

1. Click the **Mesh Part Statistics** icon  from the Specification Tools toolbar.

   The Statistics dialog box appears.

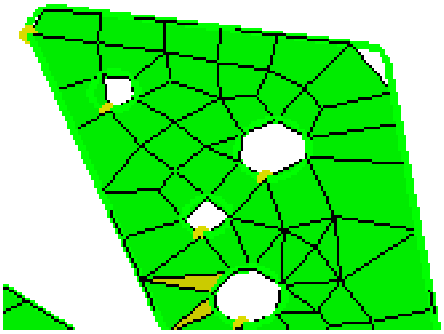
   

   - **Elements**: number of the mesh part triangle elements.
   -  : Mesh part triangle number and percentage
   -  : Mesh part poor element number and percentage

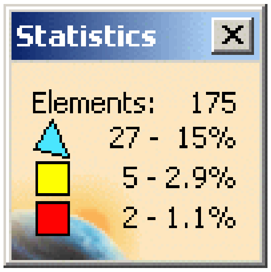
-  : Mesh part bad element number and percentage

2. Modify the mesh. For this, click the **Edit Mesh** icon  from the Modification Tools toolbar and start modifying the mesh.

See Mesh Editing for more information on this step

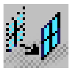


The Statistics dialog box is dynamically updated as you edit the mesh:

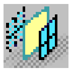


You can keep the Statistics dialog box displayed as you perform modifications on the mesh, whatever, using the Modification Tools commands.

Mesh Transformations


Translation: Create a translation mesh part from a parent mesh part.


Rotation: Create a rotation mesh part from a parent mesh part.


Symmetry: Create a symmetry mesh part from a parent mesh part.


Extrusion by Translation: Create an extrusion by translation mesh part from a parent mesh part.


Extrusion by Rotation: Create an extrusion by rotation mesh part from a parent mesh part.

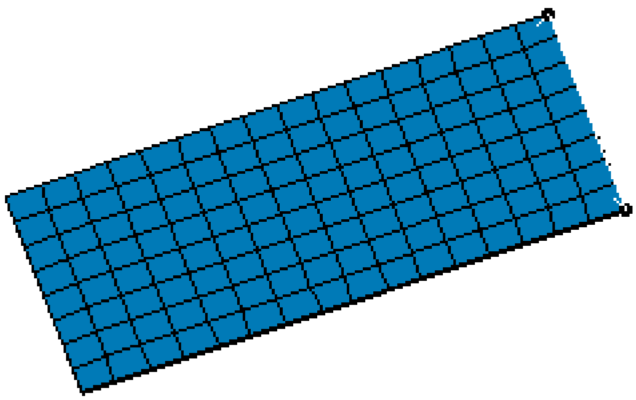

Extrusion by Symmetry: Create an extrusion by symmetry mesh part from a parent mesh part.

Translation

This task shows how to create a translation mesh part from a parent mesh part.

This transformation is available in case of 1D, 2D and 3D mesh parts.

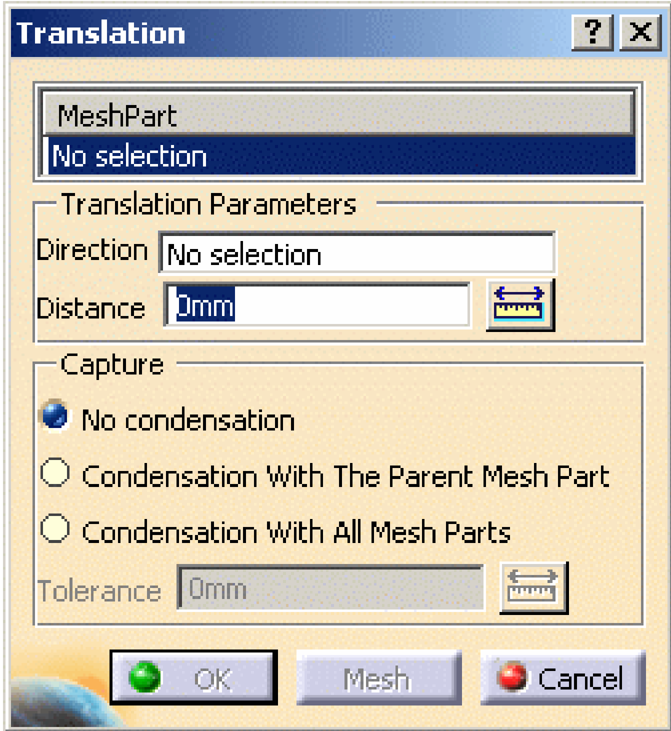
Open the Sample09.CATAnalysis document from the samples directory.



All the mesh parts must have been updated before creating a translation mesh part.

1. Click the **Translation Mesher** icon .

   The Translation dialog box appears.



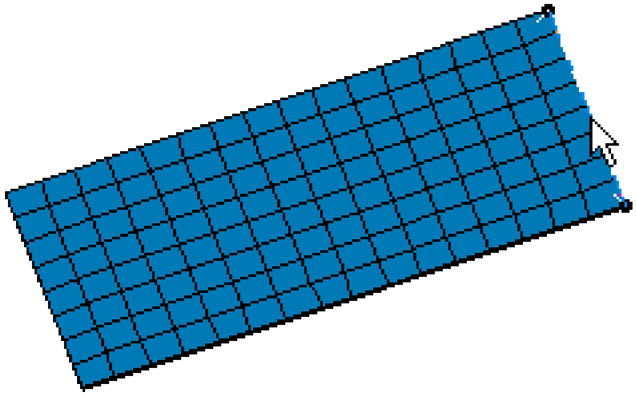
- **MeshPart**: lets you select the parent mesh part
- **Translation Parameters**
  - **Direction**: lets you choose the direction of the translation
  - **Distance**: lets you choose the value and the orientation of the translation (in millimeters)
- **Capture**: lets you capture the existing mesh
  - **No condensation**: lets you decide to not condense the nodes of the transformed and the parent mesh part
  - **Condensation With The Parent Mesh Part**: lets you condense the nodes of the transformed and the parent mesh part
    - **Tolerance**: lets you specify the tolerance of the condensation
  - **Condensation With All Mesh Parts**: lets you condense the nodes of the transformed mesh part and all the neighboring mesh parts
    - **Tolerance**: lets you specify the tolerance of the condensation

Note that: This transformation respects associativity.

In other words, if a load is applied to the parent mesh part, the same load will be applied to the transformed mesh part.

2. Select the **Smart Surface Mesh.2** object.

3. Select the desired direction.



4. Type the desired value of translation in the **Distance** field.

   In this particular case, enter **50mm**.

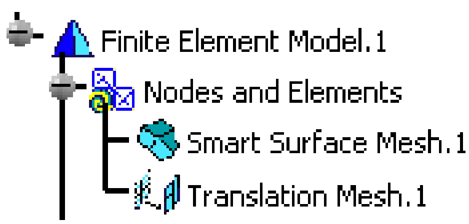
   

   You have the possibility to enter a positive or negative value to define the orientation of the translation.

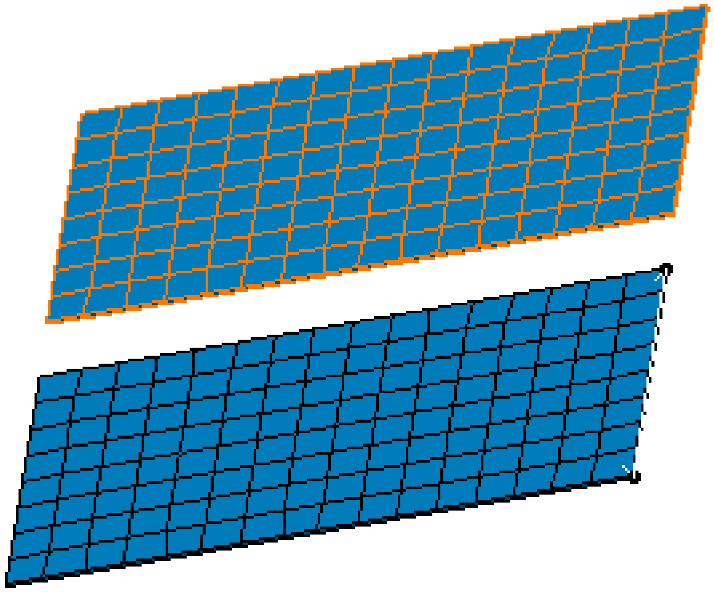
5. Select the **No condensation** option in the Translation dialog box.

6. Click the **Mesh** button in the Translation dialog box.

   A **Translation Mesh.1** object appears in the specification tree and the translation mesh part is created.

   For a better visualization, select the **Translation Mesh.1** object in the specification tree.



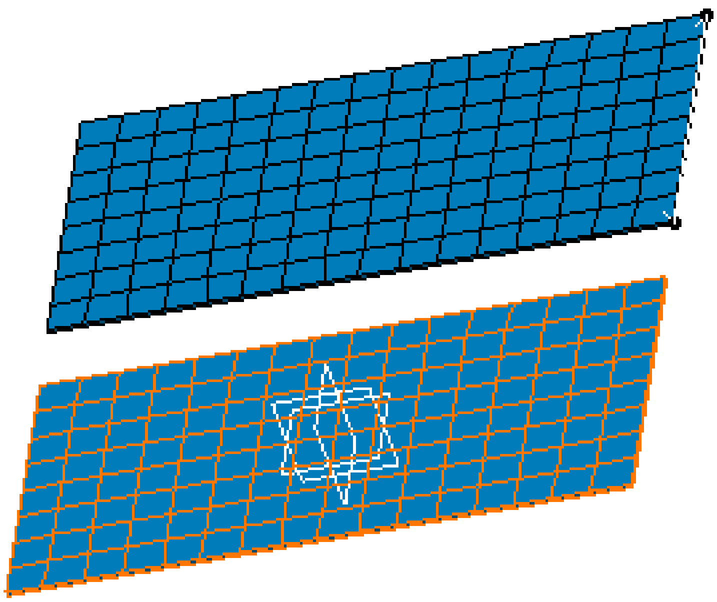
7. Edit the translation mesh.

For this, double-click the **Translation Mesh.1** object in the specification tree. The Translation dialog box appears.

8. Enter **-50mm** as **Distance** value and click **Mesh** in the Translation dialog box.

As a result, the translation mesh part appears in the opposite direction.

For a better visualization, select the **Translation Mesh.1** object in the specification tree.

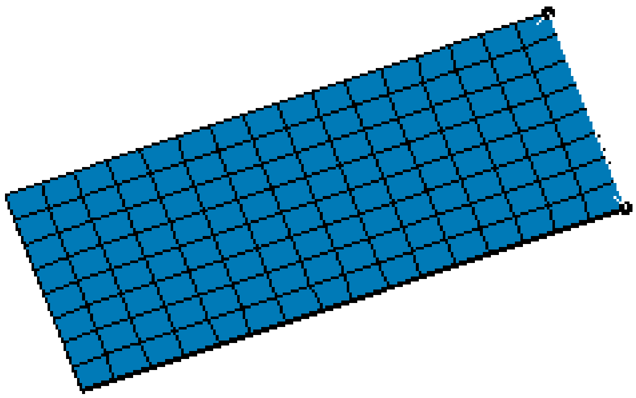


Rotation

This task shows you how to create a rotation mesh part from a parent mesh part.

This transformation is available in case of 1D, 2D and 3D mesh parts.

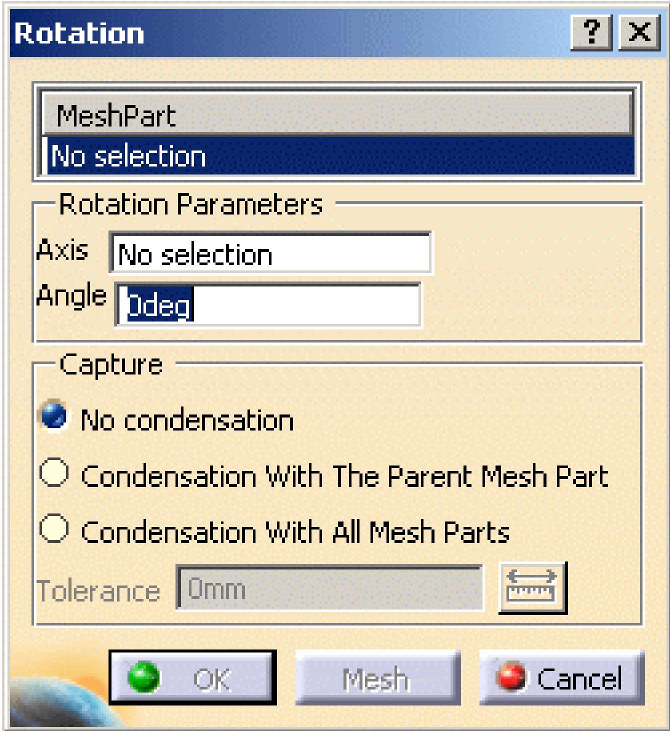
Open the Sample09.CATAnalysis document from the samples directory.



All the mesh parts must have been updated before creating a rotation mesh part.

1. Click the **Rotation Mesher** icon .

   The Rotation dialog box appears.



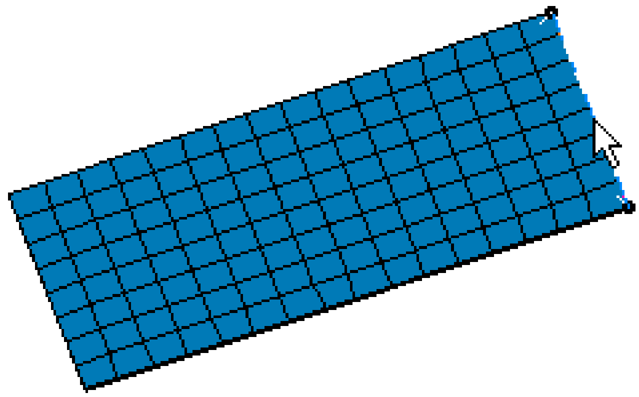
- **MeshPart**: lets you select the parent mesh part
- **Rotation Parameters**
  - **Axis**: lets you select the axis of rotation
  - **Angle**: lets you select the angle of rotation (in degrees)
- **Capture**: lets you capture the existing mesh
  - **No condensation**: lets you decide to not condense the nodes of the transformed and the parent mesh part
  - **Condensation With The Parent Mesh Part**: lets you condense the nodes of the transformed and the parent mesh part
    - **Tolerance**: lets you specify the tolerance of the condensation
  - **Condensation With All Mesh Parts**: lets you condense the nodes of the transformed mesh part and all the neighboring mesh parts
    - **Tolerance**: lets you specify the tolerance of the condensation

Note that: This transformation respects associativity.

In other words, if a load is applied to the parent mesh part, the same load will be applied to the transformed mesh part.

2. Select the **Smart Surface Mesh.2** object.

3. Select the axis of rotation.



4. Enter the desired value of rotation angle.

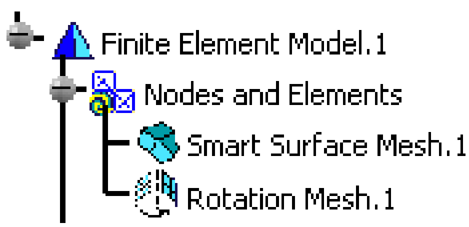
   In this particular case, enter **90 deg** as **Angle** value.

   You have the possibility to enter a positive or negative value to define the orientation of the rotation.

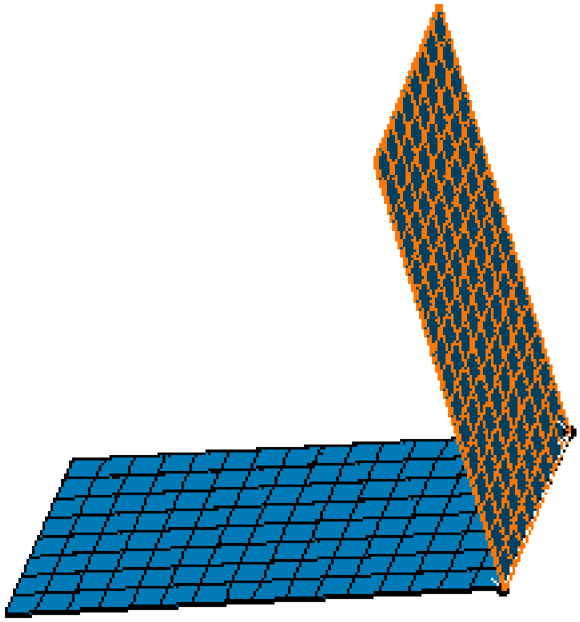
5. Select the **No condensation** option in the Rotation dialog box.

6. Click the **Mesh** button in the Rotation dialog box.

   A **Rotation Mesh.1** object appears in the specification tree and the rotation mesh part is created.



For a better visualization, select the **Rotation Mesh.1** object in the specification tree.



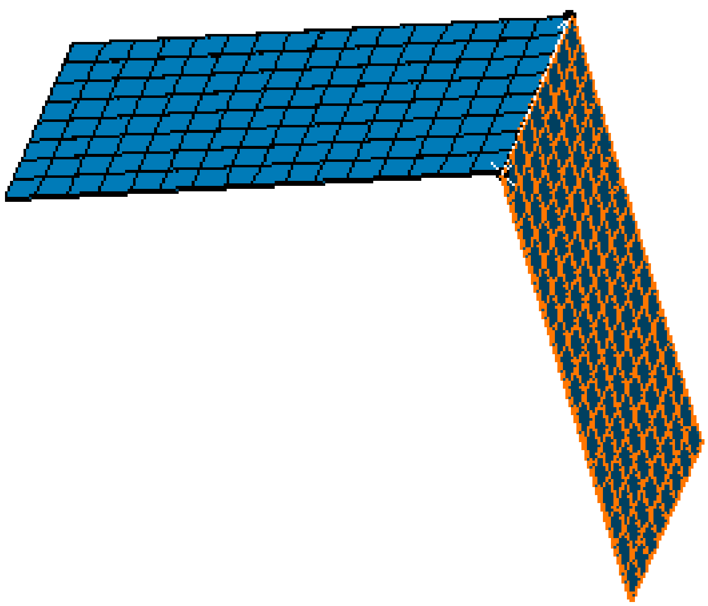
7. Edit the rotation mesh part.

For this, double-click the **Rotation Mesh.1** object in the specification tree. The Rotation dialog box appears.

8. Enter **-90 deg** as **Angle** value and click **Mesh** in the Rotation dialog box.

As a result, the rotation mesh appears in the opposite direction.

For a better visualization, select the **Rotation Mesh.1** object in the specification tree.

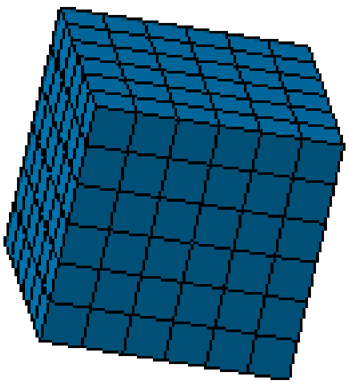


Symmetry

This task shows you how to create a symmetric mesh part from a parent mesh part.

This transformation is available in case of 1D, 2D and 3D mesh parts.

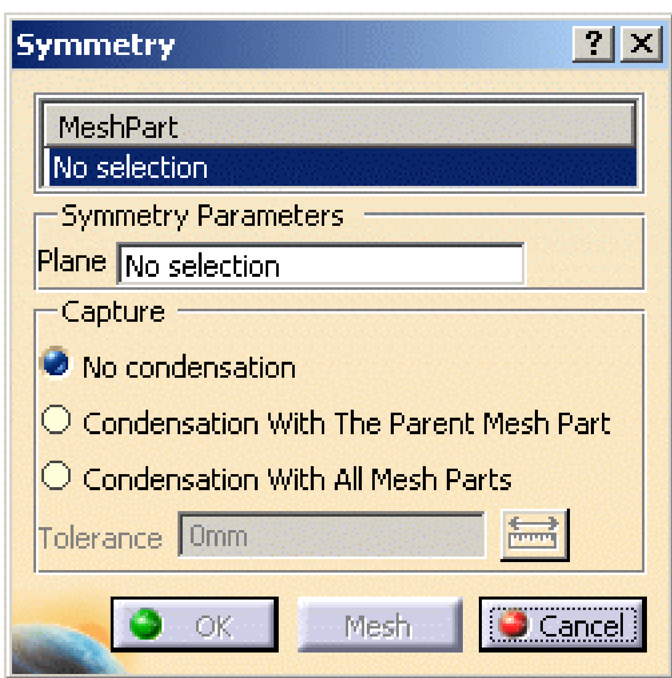
Open the Sample40.CATAnalysis document from the samples directory.



All the mesh parts must have been updated before creating a symmetry mesh part.

1. Click the **Symmetry Mesher** icon .

   The Symmetry dialog box appears.



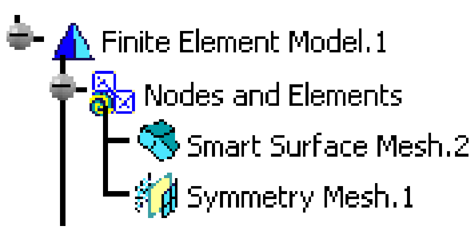
- **MeshPart**: lets you select the parent mesh part
- **Symmetry Parameters**
  - **Plane**: lets you select the plane of symmetry
- **Capture**: lets you capture the existing mesh
  - **No condensation**: lets you decide to not condense the nodes of the transformed and the parent mesh part
  - **Condensation With The Parent Mesh Part**: lets you condense the nodes of the transformed and the parent mesh part
    - **Tolerance**: lets you specify the tolerance of the condensation
  - **Condensation With All Mesh Parts**: lets you condense the nodes of the transformed mesh part and all the neighboring mesh parts
    - **Tolerance**: lets you specify the tolerance of the condensation

Note that: This transformation respects associativity.

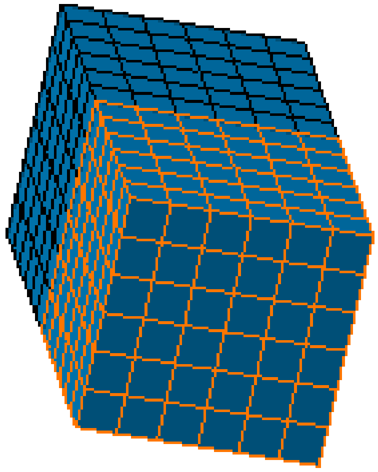
In other words, if a load is applied to the parent mesh part, the same load will be applied to the transformed mesh part.

2. Select the **Smart Surface Mesh.2** object.
3. Select the **zx plane** in the specification tree.
4. Select the **No condensation** option in the Symmetry dialog box.
5. Click the **Mesh** button in the Symmetry dialog box.

   A **Symmetry Mesh.1** object appears in the specification tree and the symmetric mesh part is created.

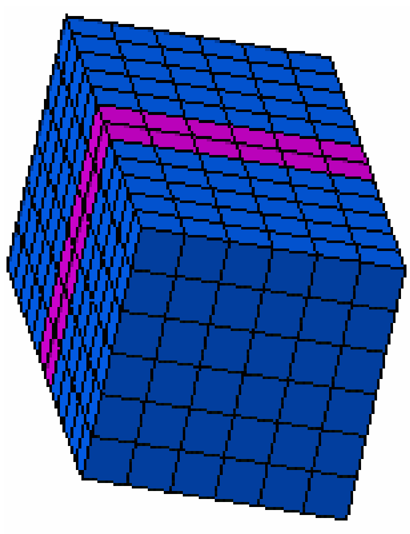


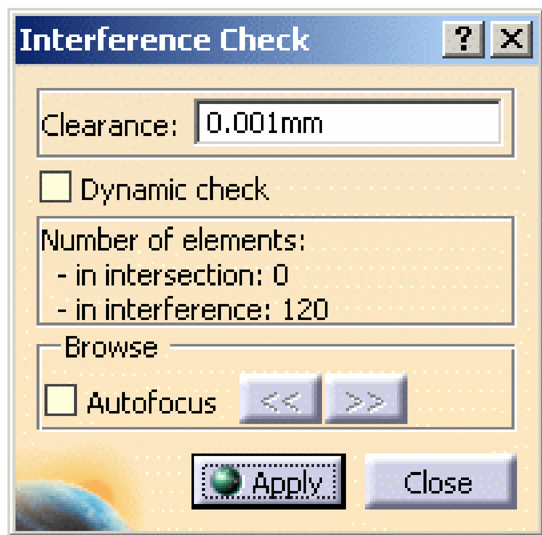
For a better visualization, select the **Symmetry Mesh.1** object in the specification tree.



6. Verify the intersections.

   For this, click the **Intersections / Interferences** icon, choose **0.001 mm** as **Clearance** value and click **Apply** in the Interference Check dialog box.





   You can see that there are 120 elements in interference because the nodes of the symmetric mesh part and the parent mesh part have not been condensed.
   Click **Close** in the Interference Check dialog box.

7. Edit the symmetric mesh.

   For this, double-click the **Symmetry Mesh.1** object in the specification tree. The Symmetry dialog box appears.

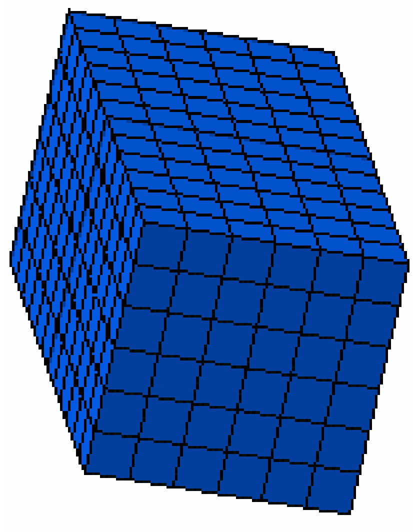
8. Select the **Condensation With The Parent Mesh Part** option.

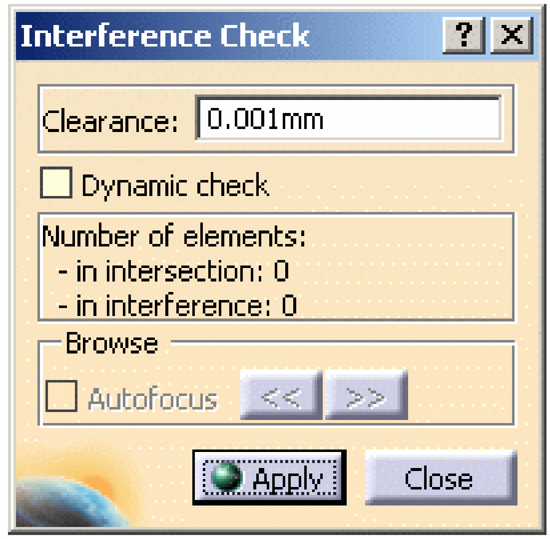
9. Enter **1mm** as **Tolerance** value.

10. Click **Mesh** in the Symmetry dialog box.

    The symmetric mesh seems to be identical to the previous one.

11. Verify the intersections.





In this case, there is no element in interference because the nodes of the symmetric mesh part and the nodes of the parent mesh part have been condensed.
Click **Close** in the Interference Check dialog box.

Mesh Extrusion by Translation

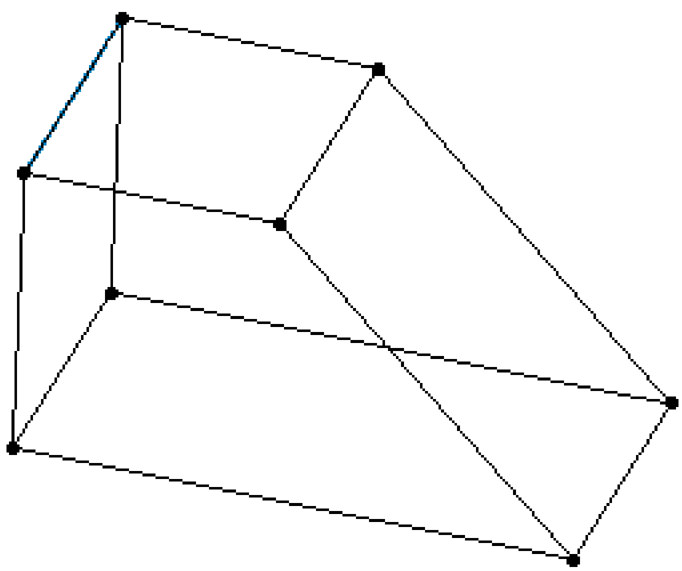
This task shows you how to extrude a mesh by translation.

You will see here the two possibilities of the extrusion by translation:

- create a 2D mesh from a 1D mesh
- create a 3D mesh from a 2D mesh

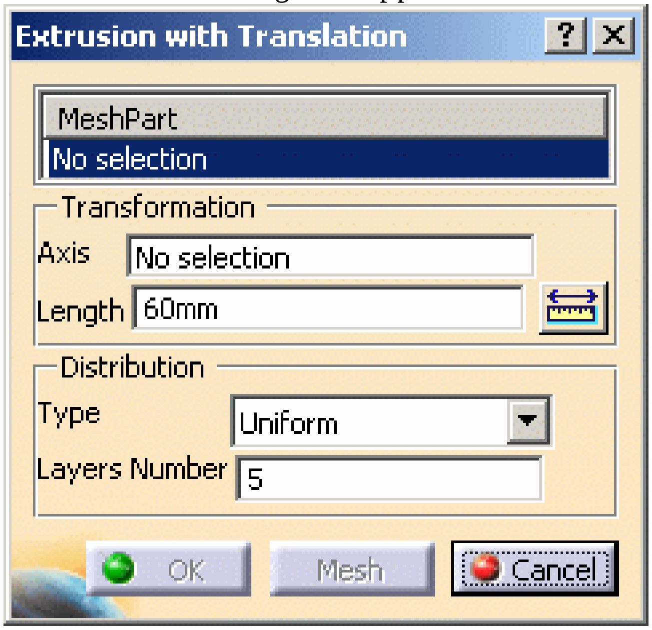
Create a 2D mesh from a 1D mesh

Open the Sample10.CATAnalysis document from the samples directory.



1. Click the **Extrude Mesher with Translation** icon from the Mesh Transformations toolbar.

The Extrusion dialog box appears:

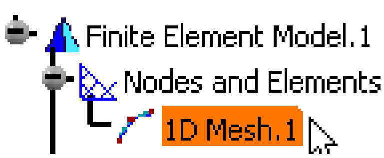


- **MeshPart**: lets you specify the mesh to extrude.
- **Transformation:**
  - **Axis**: lets you specify the direction of the extrusion.
  - **Length**: lets you specify the length of the extrusion.

    Note that you can reverse the extrusion direction if you enter a negative **Length**.
- **Distribution**:
  - **Type**: Indicate the node distribution type.
  - **Layers Number**: lets you specify the number of layers you want.

    Note that this value determines the mesh size.
    For example, if the extrusion **Length** value is **30mm** and the **Layers number** value is **6**, the mesh size value will be : 30mm / 6 = 5mm.

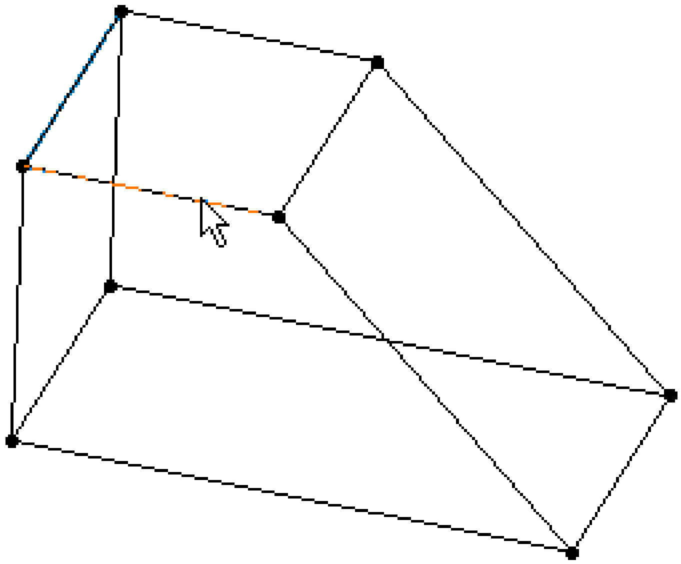
2. Select the 1D Mesh.1 feature from the specification tree.



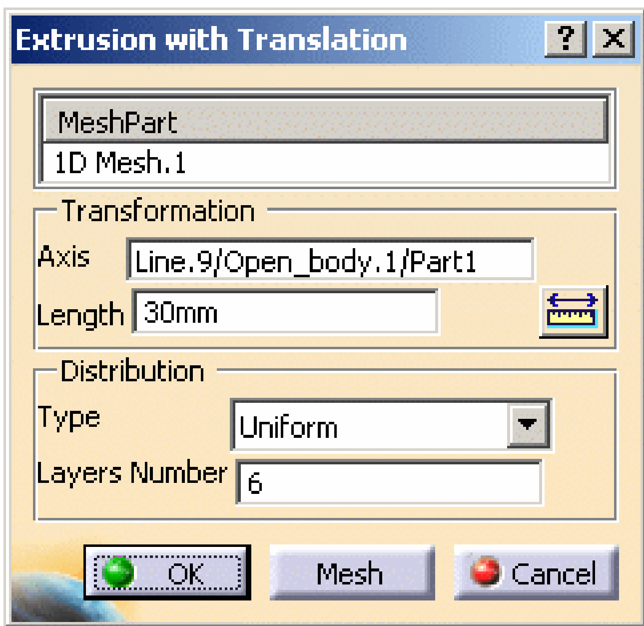
3. Specify the properties of the extrusion in the Extrusion dialog box.

In this particular case:

- Select **Line.9** as **Axis** as shown below:

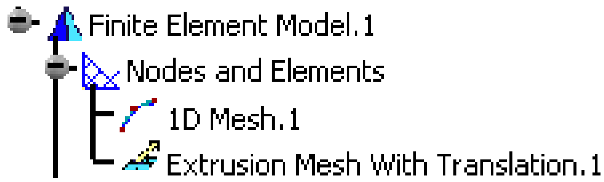


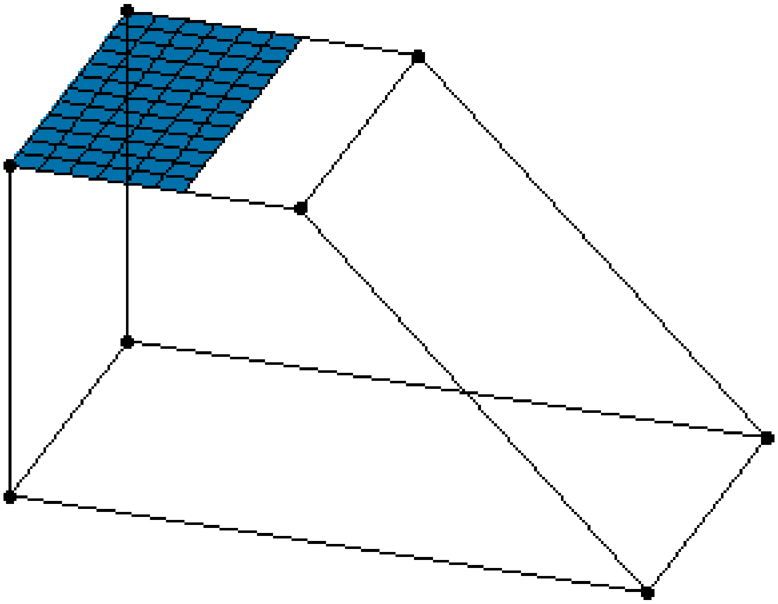
- Enter **30mm** as **Length** value
- Select **Uniform** as **Type** option
- Enter **6** as **Layers number** value



4. Click the **Mesh** button.

An **Extrusion Mesh with Translation.1** object appears in the specification tree and a 2D extrusion by translation mesh part is created.



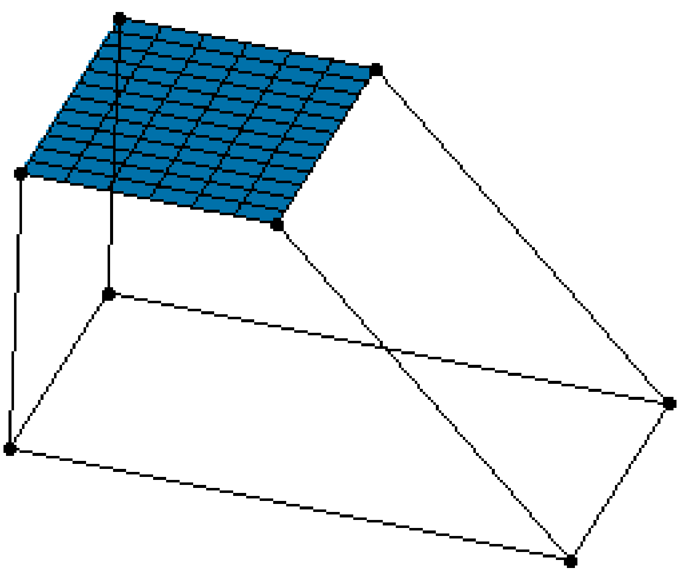


5. Edit the 2D extrusion by translation mesh.

   For this, double-click on the **Extrusion Mesh with Translation.1** object from the specification tree. The Extrusion dialog box appears.

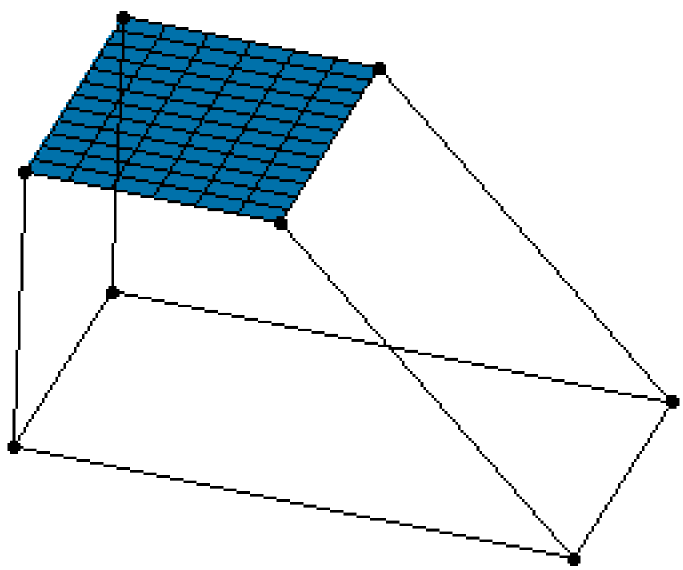
6. Enter **50mm** as **Length** value and click **Mesh** in the Extrusion dialog box.

   As a result the 2D extrusion by translation mesh part is updated as shown below:



Create a 3D mesh from a 2D mesh

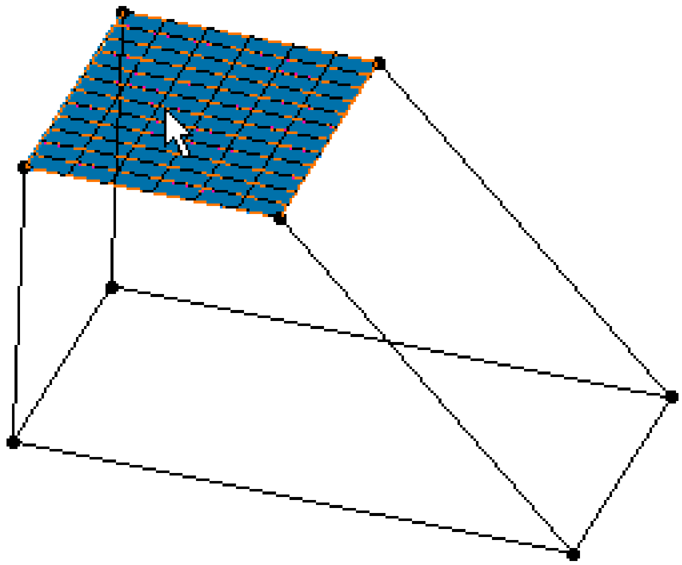
Use the CATAnalysis document you have obtained at the end of the Create a 2D mesh from a 1D mesh scenario or open the Sample10_1.CATAnalysis document from the samples directory.



1. Click the **Extrude Mesher with Translation** icon from the Mesh Transformations toolbar.

   The Extrusion dialog box appears.
   For more information about the Extrusion dialog box please refer to Create a 2D mesh from a 1D mesh.

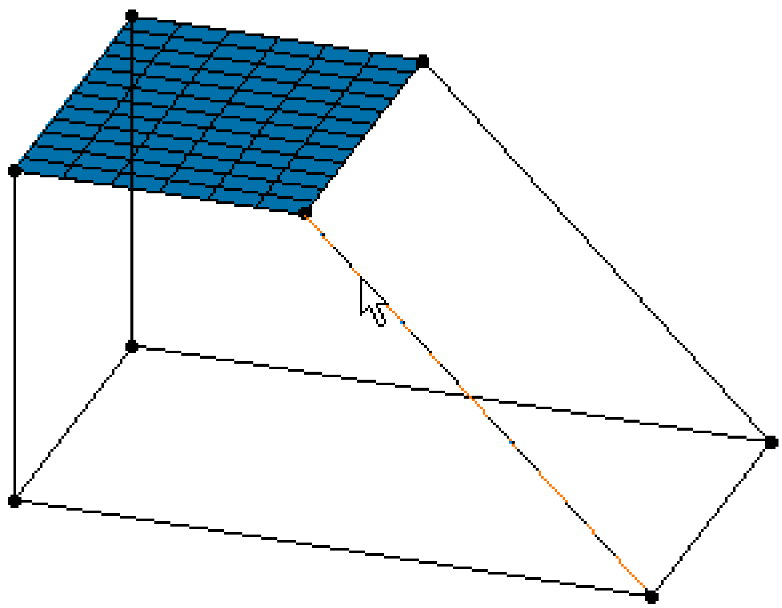
2. Click the 2D mesh you want to extrude.



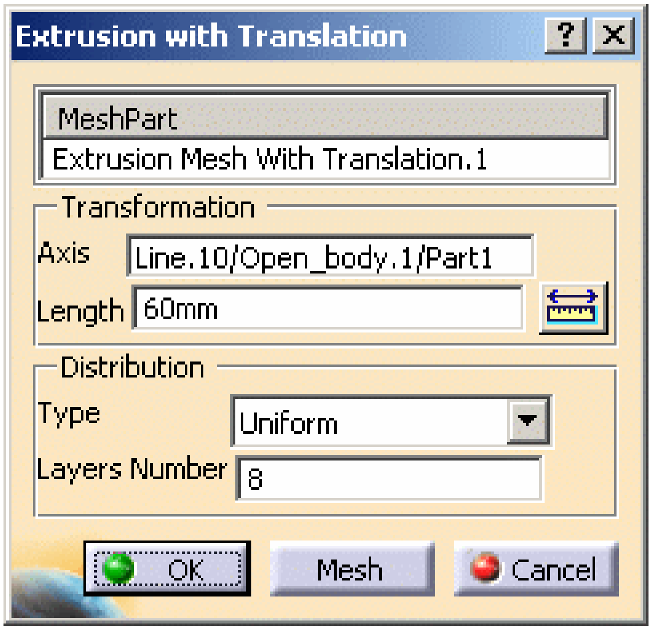
3. Specify the properties of the extrusion in the Extrusion dialog box.

In this particular case:

- Select **Line.10** as **Axis** as shown below:

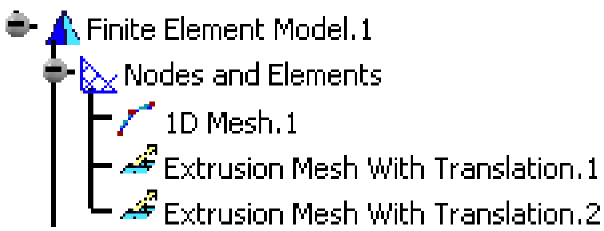


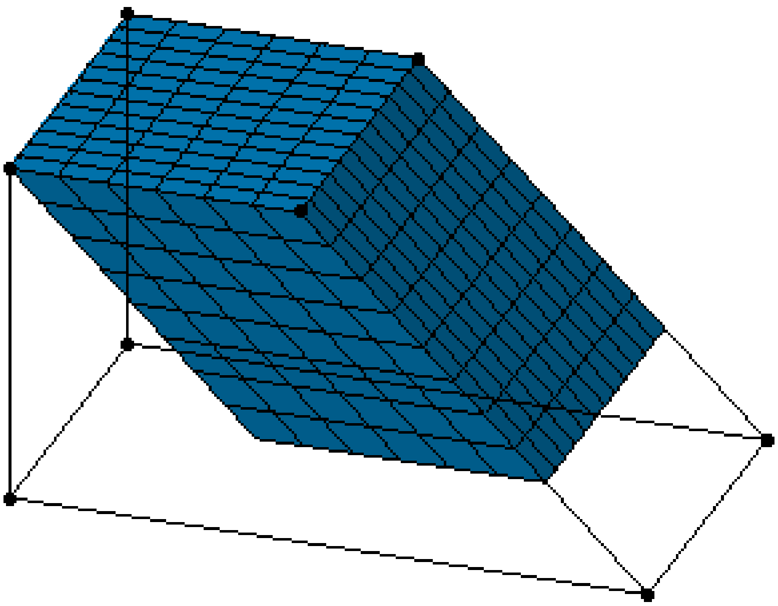
- Enter **60mm** as **Length** value
- Select **Uniform** as **Type** option
- Enter **8** as **Layers number** value



4. Click the **Mesh** button.

An **Extrusion Mesh with Translation.2** object appears in the specification tree and a 3D extrusion by translation mesh part is created.

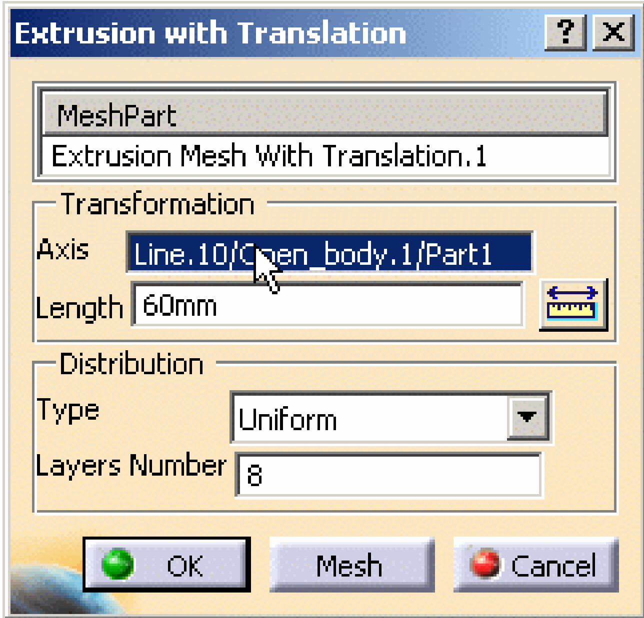




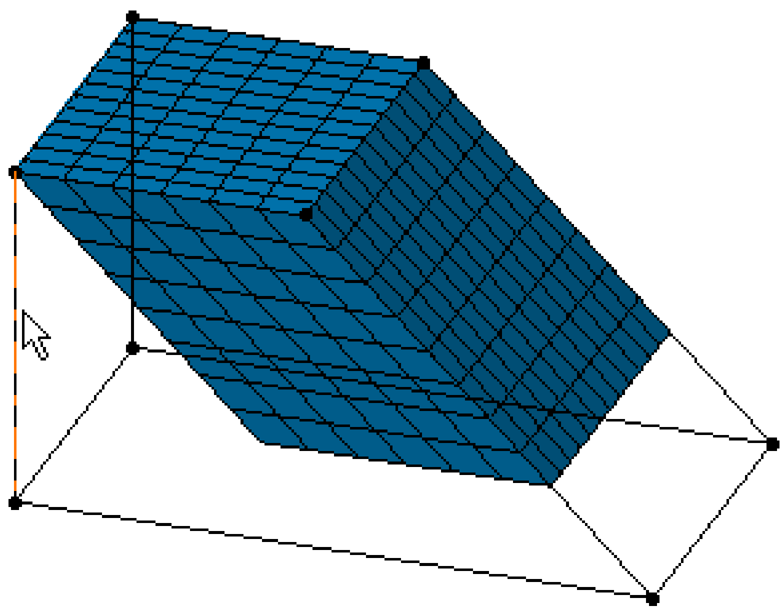
5. Edit the 3D extrusion by translation mesh.

For this, double-click on the **Extrusion Mesh with Translation.2** object from the specification tree. The Extrusion dialog box appears.

6. Click the **Axis** input field in the Extrusion dialog box.

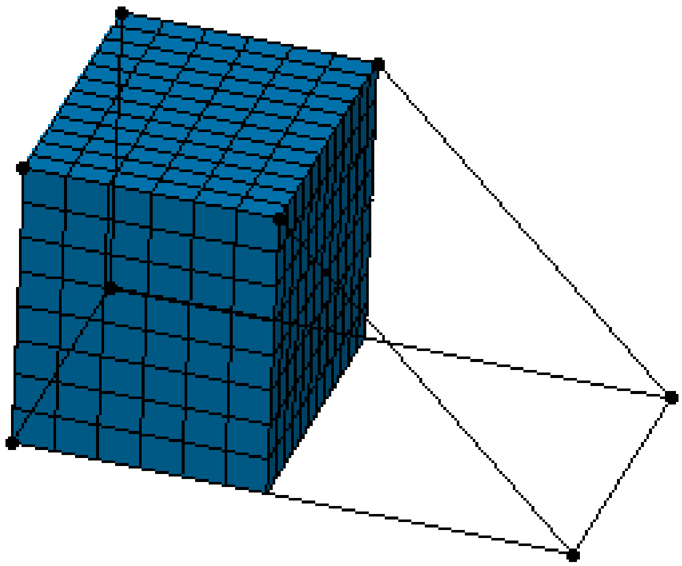


7. Select **Line.2** as **Axis** as shown below:



8. Click **Mesh** in the Extrusion dialog box.

As a result the 3D extrusion by translation mesh part is updated as shown below:



Mesh Extrusion by Rotation

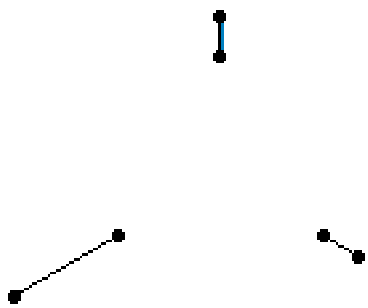
This task shows you how to extrude a mesh by rotation.

You will see here the two possibilities of the extrusion by rotation:

- create a 2D mesh from a 1D mesh
- create a 3D mesh from a 2D mesh

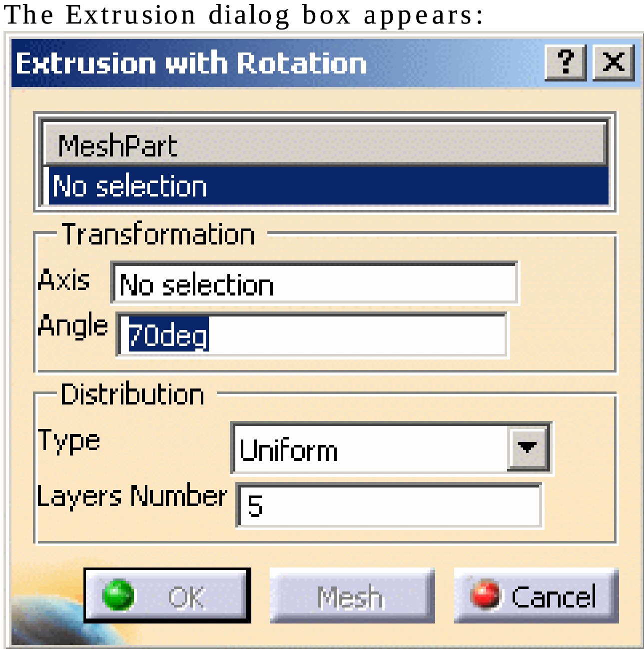
Create a 2D mesh from a 1D mesh

Open the Sample21.CATAnalysis document from the samples directory.



1. Click the **Extrude Mesher with Rotation** icon  from the Mesh Transformations toolbar.

   The Extrusion dialog box appears:

   - **MeshPart**: lets you specify the mesh to extrude.

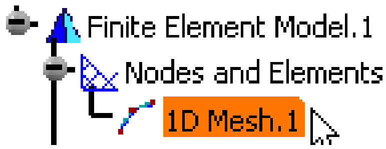
- **Transformation**
  - **Axis**: lets you specify the axis of extrusion by rotation.
  - **Angle**: lets you specify the angle of the extrusion by rotation.

    Note that you can reverse the direction of the extrusion by rotation if you enter a negative **Angle**.

- **Distribution**:
  - **Type**: Indicate the node distribution type.
  - **Layers number**: lets you specify the number of layers you want.

    Note that the mesh size is variable and depends on this value and the distance between the source node and the axis of extrusion by rotation.
    For example, if the extrusion radius value between the source node and the axis of extrusion by translation is 40mm, the angle is 45 degree and the **Layers number** value is 10, the mesh size value will be :
    (**Angle*** Radius) / **Layers number** = ((PI/4) * 40mm) / 10 = 3.14 mm

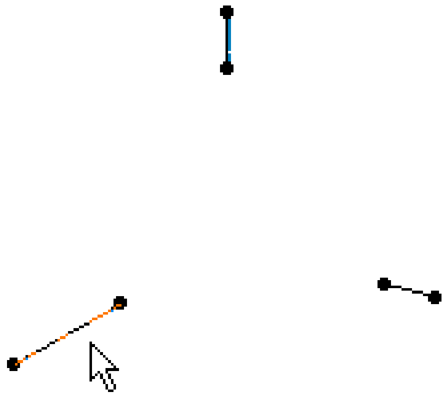
2. Select the 1D Mesh.1 feature from the specification tree.



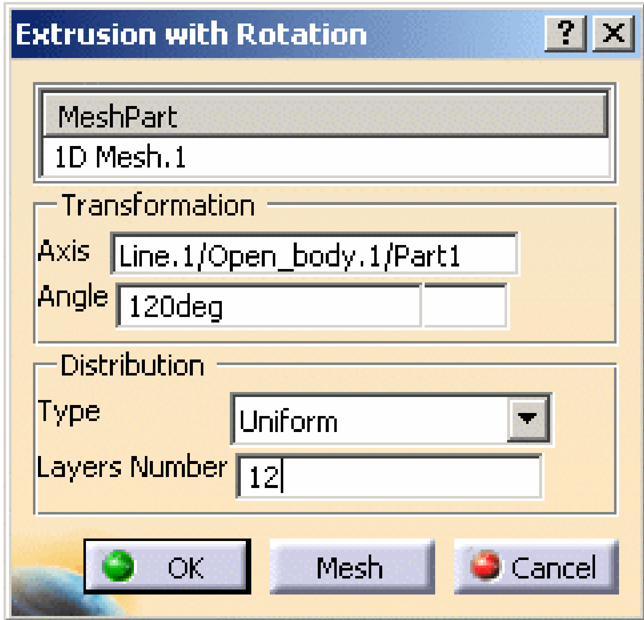
3. Specify the properties of the extrusion in the Extrusion dialog box.

In this particular case:

- Select **Line.1** as **Axis** as shown below

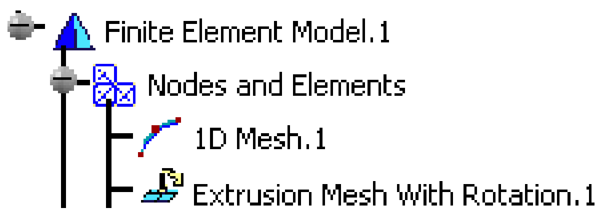


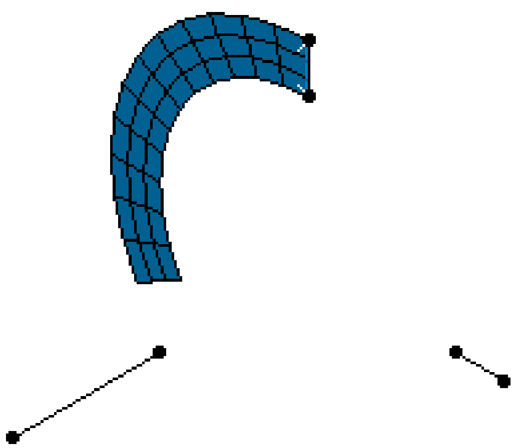
- Enter **120deg** as **Angle** value
- Select **Uniform** as **Type** option
- Enter **12** as **Layers number** value



4. Click the **Mesh** button.

   An **Extrusion Mesh with Rotation.1** object appears in the specification tree and a 2D extrusion by rotation mesh part is created.





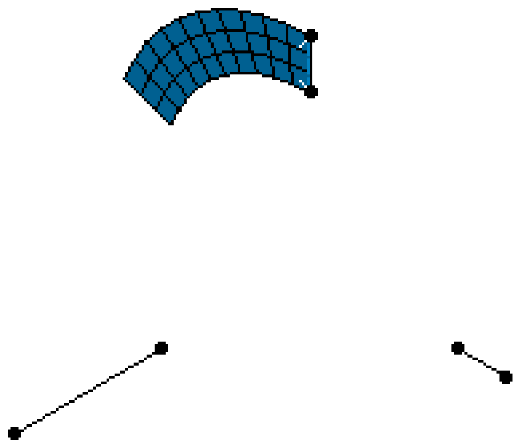
5. Edit the 2D extrusion by rotation mesh.

   For this, double-click on the **Extrusion Mesh with Rotation.1** object from the specification tree. The Extrusion dialog box appears.

6. Enter **70deg** as **Angle** value, **10** as **Layers number** value and click **Mesh** in the

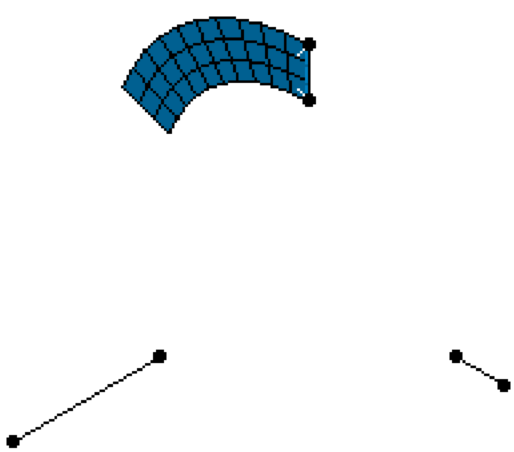
   Extrusion dialog box.

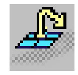
As a result the 2D extrusion by rotation mesh part is updated as shown below:



Create a 3D mesh from a 2D mesh

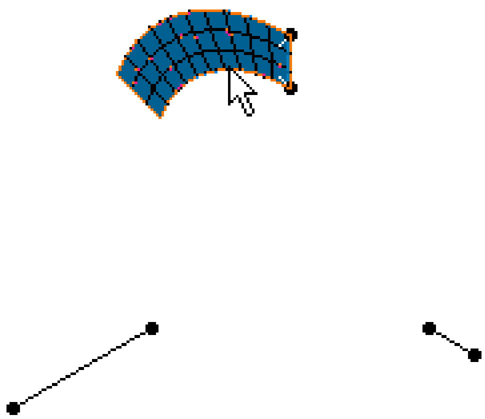
Use the CATAnalysis document you have obtained at the end of the Create a 2D mesh from a 1D mesh scenario or open the Sample21_1.CATAnalysis document from the samples directory.



1. Click the **Extrude Mesher with Rotation** icon  from the Mesh Transformations toolbar.

   The Extrusion dialog box appears.
   For more information about the Extrusion dialog box please refer to Create a 2D mesh from a 1D mesh.

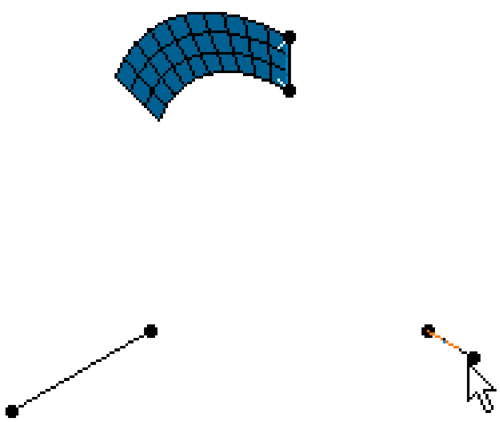
2. Click the 2D mesh you want to extrude.



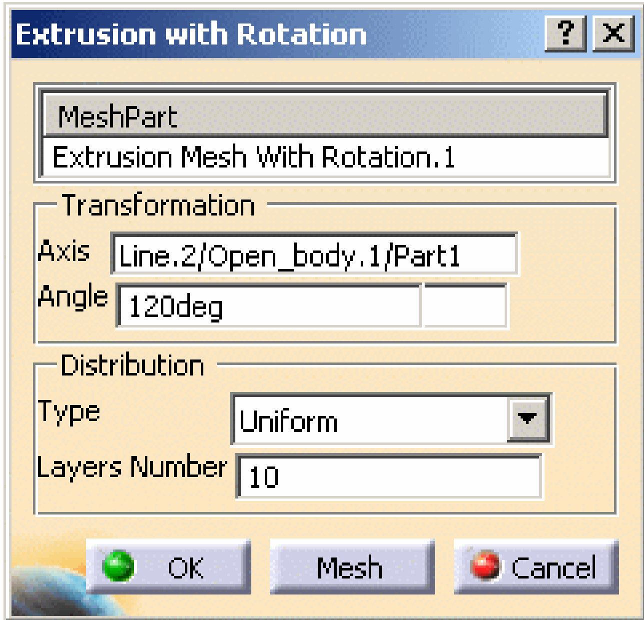
3. Specify the properties of the extrusion in the Extrusion dialog box.

   In this particular case:

   - Select **Line.12** as **Axis** as shown below

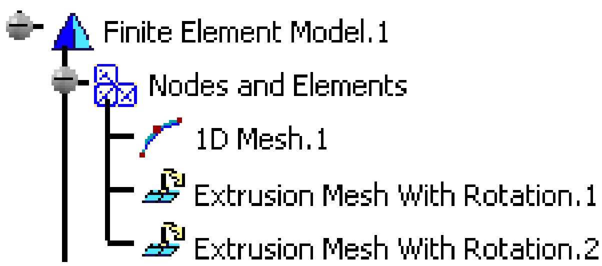


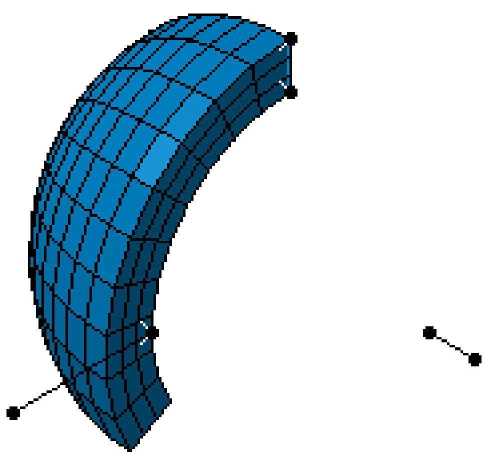
   - Enter **120deg** as **Angle** value
   - Select **Uniform** as **Type** option
   - Enter **10** as **Layers number** value



4. Click the **Mesh** button.

An **Extrusion Mesh with Rotation.2** object appears in the specification tree and a 3D extrusion by rotation mesh part is created.



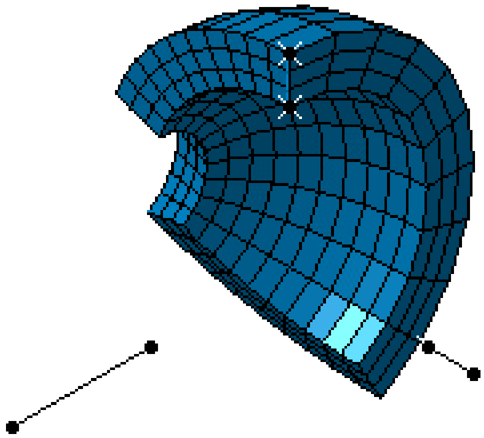


5. Edit the 3D extrusion by rotation mesh.

For this, double-click on the **Extrusion Mesh with Rotation .2** object from the specification tree. The Extrusion dialog box appears.

6. Enter **-150deg** as **Angle** value and click **Mesh** in the Extrusion dialog box.

As a result the 3D extrusion by rotation mesh part is updated as shown below:



Mesh Extrusion by Symmetry

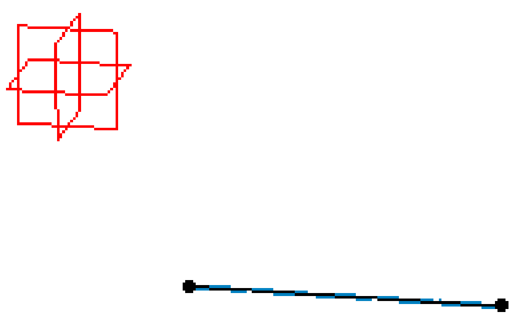
This task shows you how to extrude a mesh by symmetry.

You will see here the two possibilities of the extrusion by symmetry:

- create a 2D mesh from a 1D mesh
- create a 3D mesh from a 2D mesh

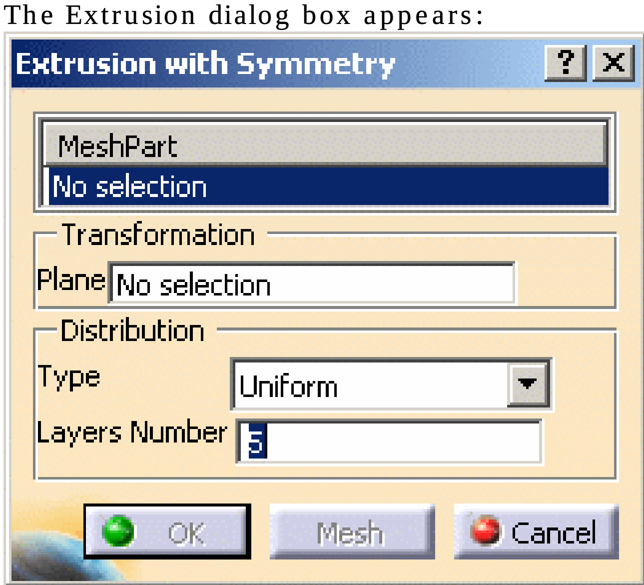
Create a 2D mesh from a 1D mesh

Open the Sample24.CATAnalysis document from the samples directory.



1. Click the **Extrude Mesher with Symmetry** icon from the Mesh Transformations toolbar.

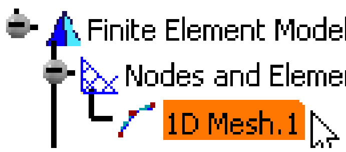
   The Extrusion dialog box appears:



   - **MeshPart**: lets you specify the mesh to extrude.
   - **Transformation**
     - **Plane**: lets you specify the plane of symmetry of the extrusion

- **Distribution**:
  - **Type**: Indicate the node distribution type.
  - **Layers number**: lets you specify the number of layers you want.
    - The extrusion length is twice the distance between the source mesh and the plane of symmetry.
    - Note that this value determines the mesh size. For example, if the extrusion **Length** value is **30mm** and the **Layers number** value is **6**, the mesh size value will be : 30mm / 6 = 5mm.

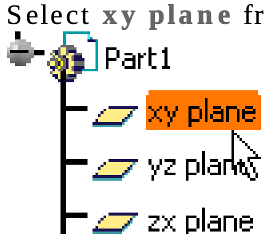
2. Select the 1D Mesh.1 feature from the specification tree.



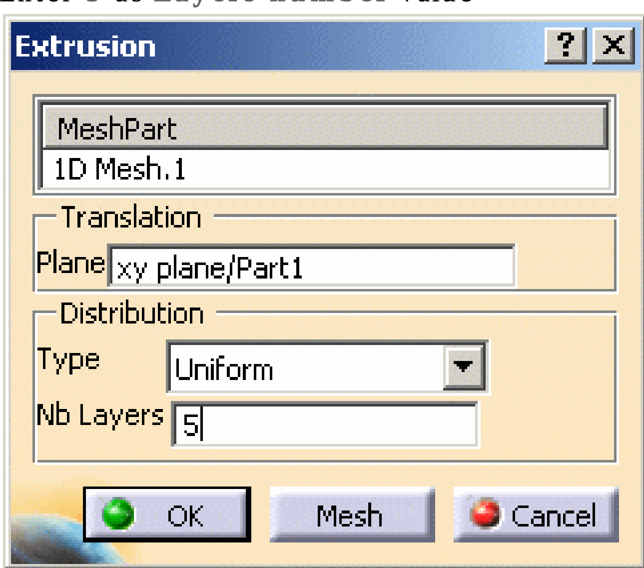
3. Specify the properties of the extrusion in the Extrusion dialog box.

In this particular case:

- Select **xy plane** from the specification tree as **Plane** option as shown below:

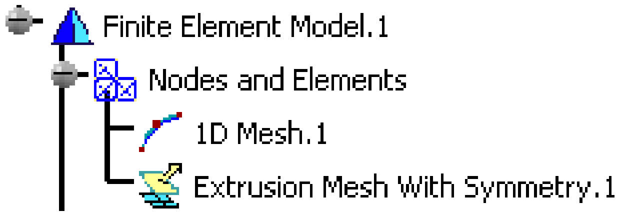


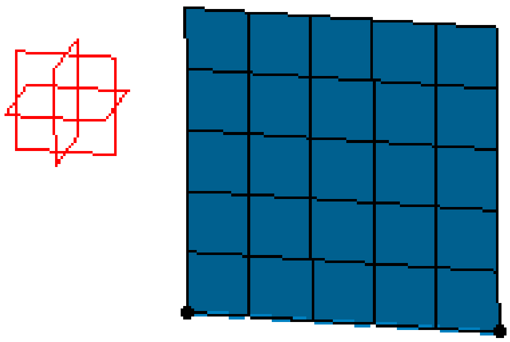
- Select **Uniform** as **Type** option
- Enter **5** as **Layers number** value



4. Click the **Mesh** button.

An **Extrusion Mesh with Symmetry.1** object appears in the specification tree and a 2D extrusion by symmetry mesh part is created.

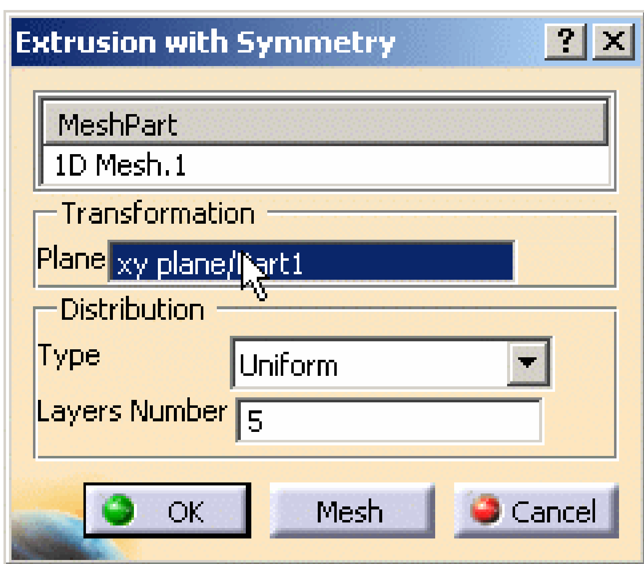




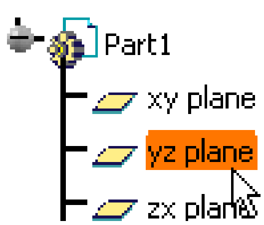
5. Edit the 2D extrusion by symmetry mesh.

   For this, double-click on the **Extrusion Mesh with Symmetry.1** object from the specification tree. The Extrusion dialog box appears.

6. Click the **Plane** input field in the Extrusion dialog box.

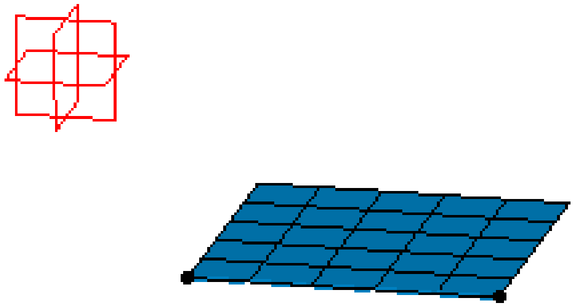


7. Select **yz plane** from the specification tree as **Plane** option as shown below:



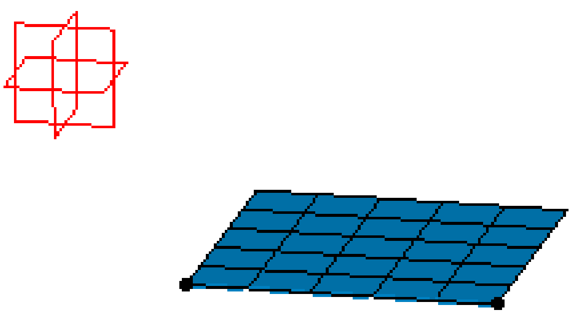
8. Click **Mesh** in the Extrusion dialog box.

As a result the 2D extrusion by symmetry mesh part is updated as shown below:



Create a 3D mesh from a 2D mesh

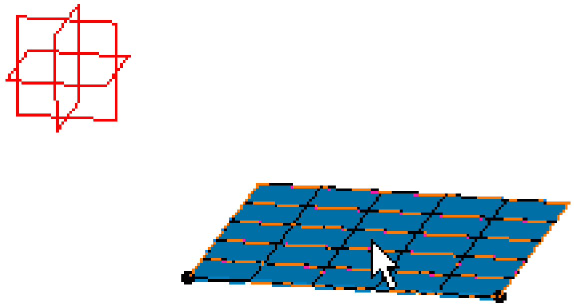
Use the CATAnalysis document you have obtained at the end of the Create a 2D mesh from a 1D mesh scenario or open the Sample24_1.CATAnalysis document from the samples directory.



1. Click the **Extrude Mesher with Symmetry** icon from the Mesh Transformations toolbar.

   The Extrusion dialog box appears.
   For more information about the Extrusion dialog box please refer to Create a 2D mesh from a 1D mesh.

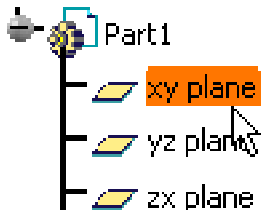
2. Click the 2D mesh you want to extrude.



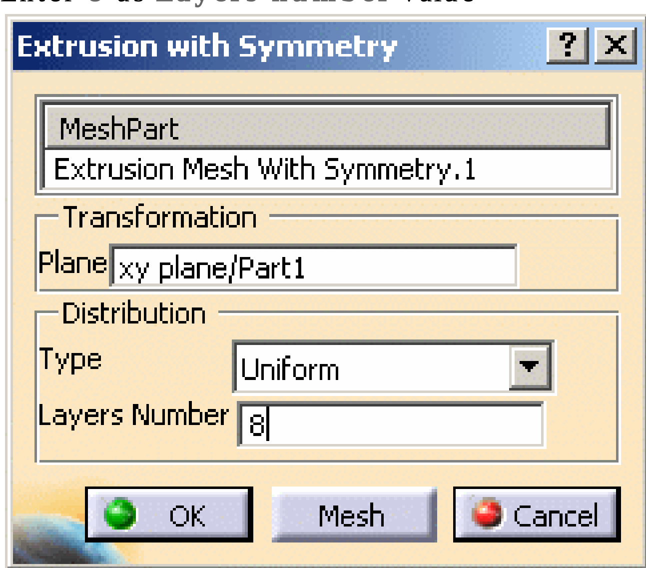
3. Specify the properties of the extrusion in the Extrusion dialog box.

In this particular case:

- Select **xy plane** from the specification tree as **Plane** option as shown below:

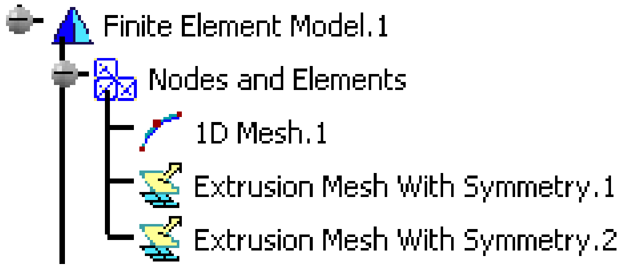


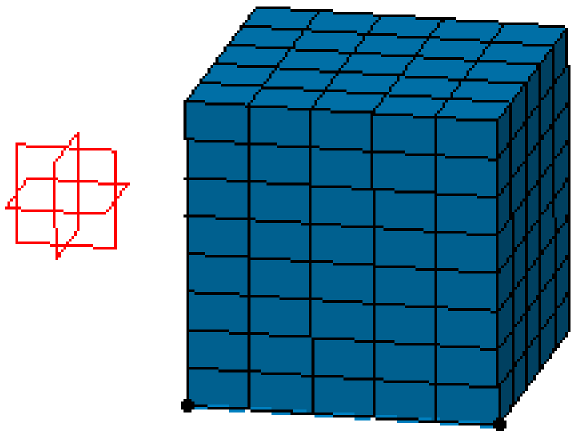
- Select **Uniform** as **Type** option
- Enter **8** as **Layers number** value



4. Click the **Mesh** button.

An **Extrusion Mesh with Symmetry.2** object appears in the specification tree and a 3D extrusion by symmetry mesh part is created.



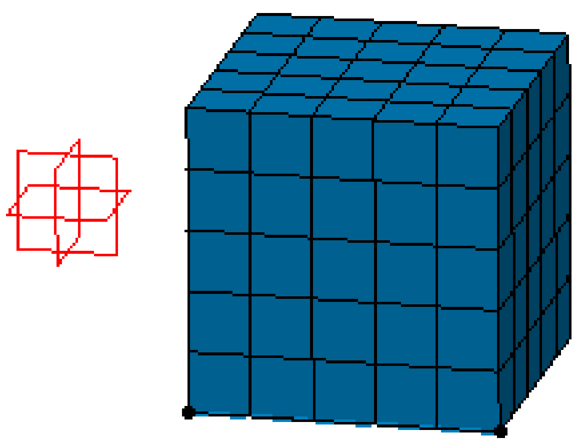


5. Edit the 3D extrusion by symmetry mesh.

   For this, double-click on the **Extrusion Mesh with Symmetry.2** object from the specification tree. The Extrusion dialog box appears.

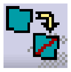
6. Enter **5** as **Layers number** value and click **Mesh** in the extrusion dialog box.

   As a result the 3D extrusion by symmetry mesh part is updated as shown below:



Mesh Operators

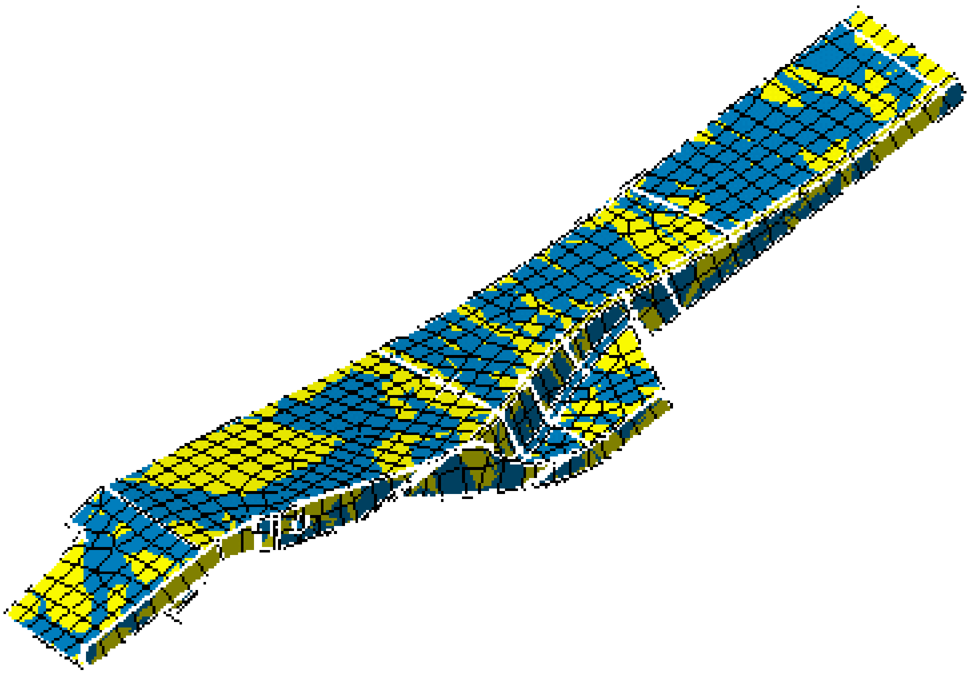
 Offsetting the Mesh: Apply an offset to a mesh.

 Splitting Quads: Split quadrangle mesh elements into triangle mesh elements.

Offsetting the Mesh

This task shows how to apply an offset to a 2D mesh (surface element).

Open the sample05.CATAnalysis document from the samples directory.

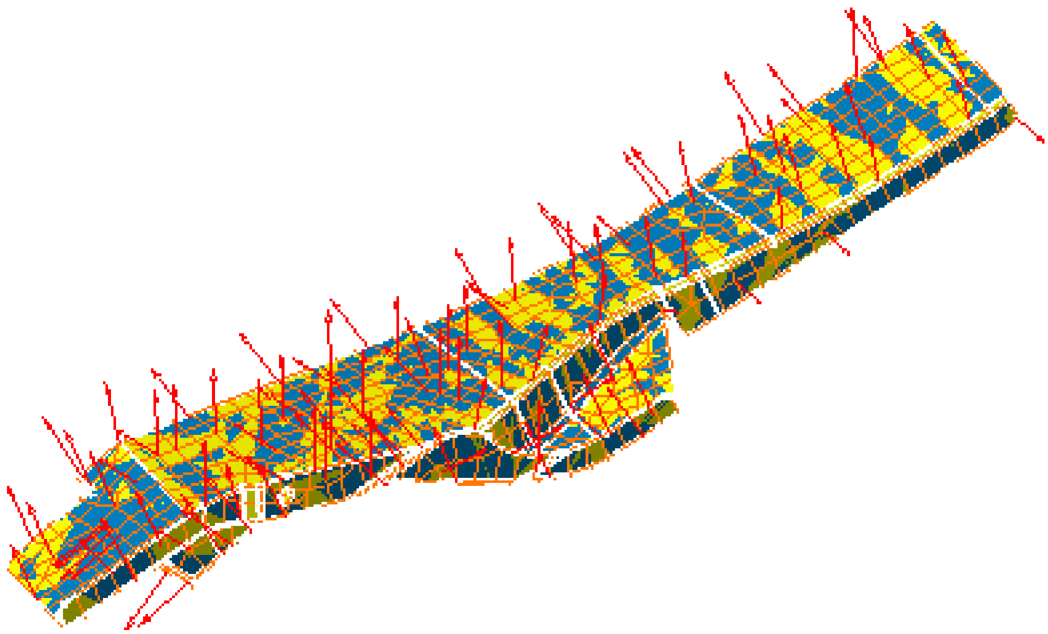


1. Click the **Mesh Offset** icon from the **Mesh Operators** toolbar.

   The Mesh Offset dialog box appears.

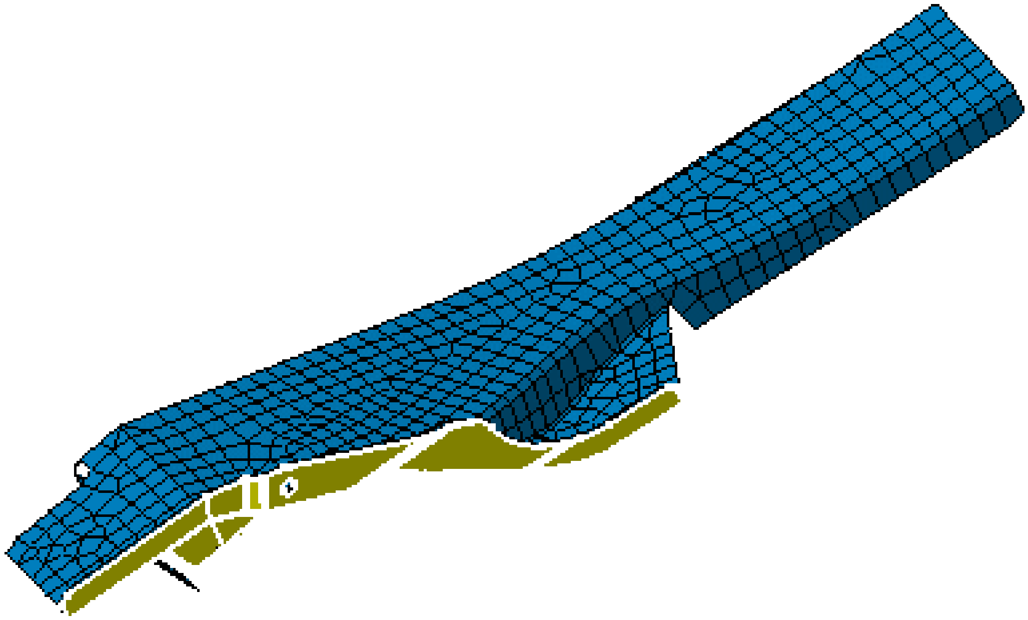
2. Select the mesh.

The **Offset** field in the Mesh Offset dialog box allows you defining which offset you want to apply relatively to a given normal. To modify the direction of the normal, click on it. As a result, the **Offset** value can only be a positive value.



3. Enter the **Offset** value in the Mesh Offset dialog box. In this example, enter **5mm** and click **OK**.

The mesh is now offset relatively to its previous position:



Splitting Quads

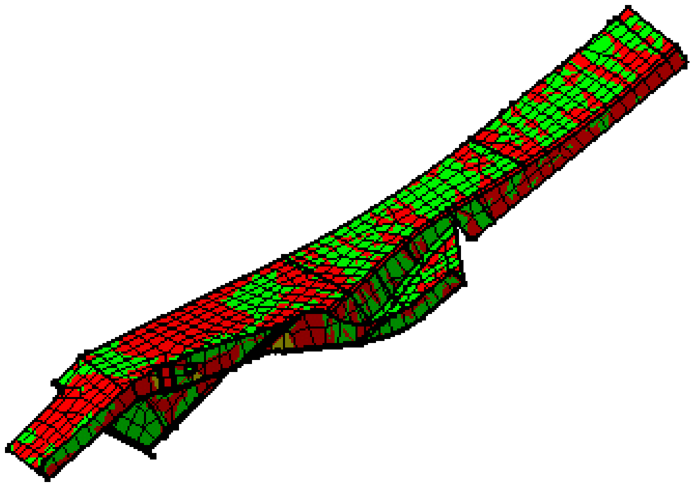
This task demonstrates how to split quadrangle mesh elements into triangle mesh elements.

Open the sample05.CATAnalysis document from the samples directory.

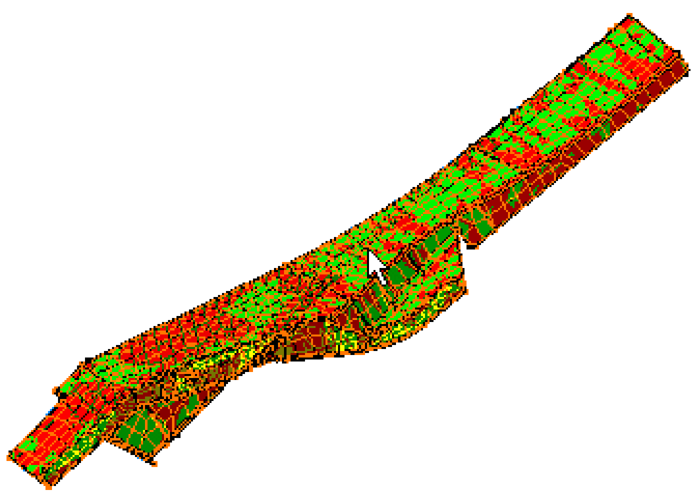
Before You Begin:

- You have to enter the Surface Meshing product
  For this, double-click **Smart Surface Mesh.1** feature from the specification tree (below **Nodes and Elements** feature) and then click **YES** (**Continue anyway?**) in the Warning box.
- You have to mesh the surface.

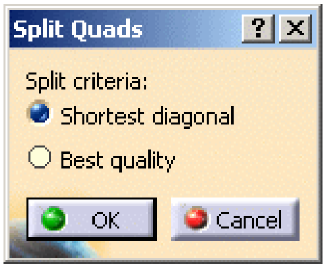
  For this, click the **Mesh the Part** icon  from the **Execution** toolbar.
  You will then click **OK** in the Mesh the Part dialog box.



1. Click the **Split Quads** icon  from the **Mesh Operators** toolbar.
2. Select the geometry to be meshed by clicking on the part.

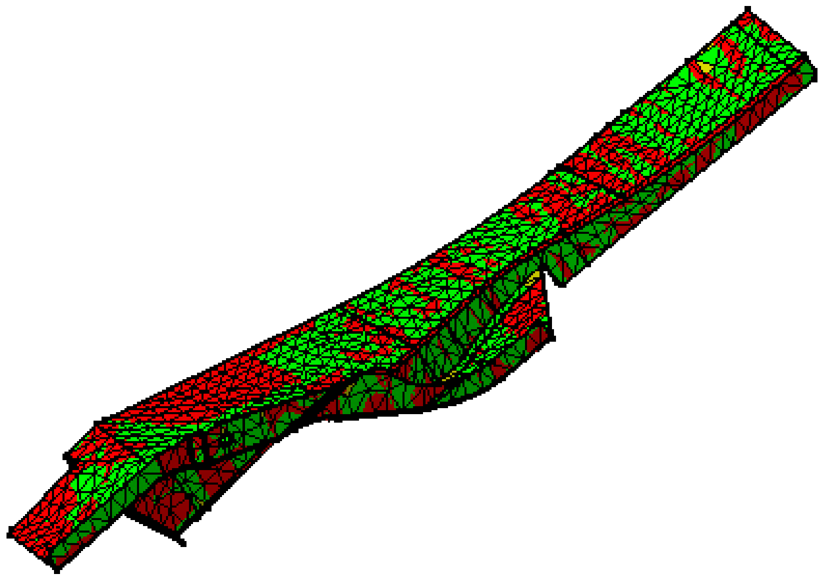


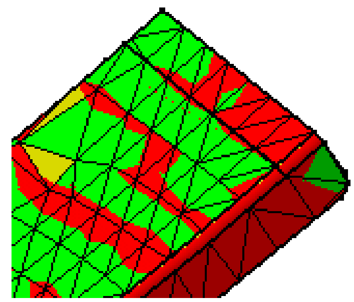
The Split Quads dialog box appears.



- **Shortest diagonal:** Choosing between both diagonals the shortest.
- **Best quality:** Considering all the active quality criteria and choosing the diagonal with the best quality elements.

The quadrangle mesh elements are split into triangle mesh elements.



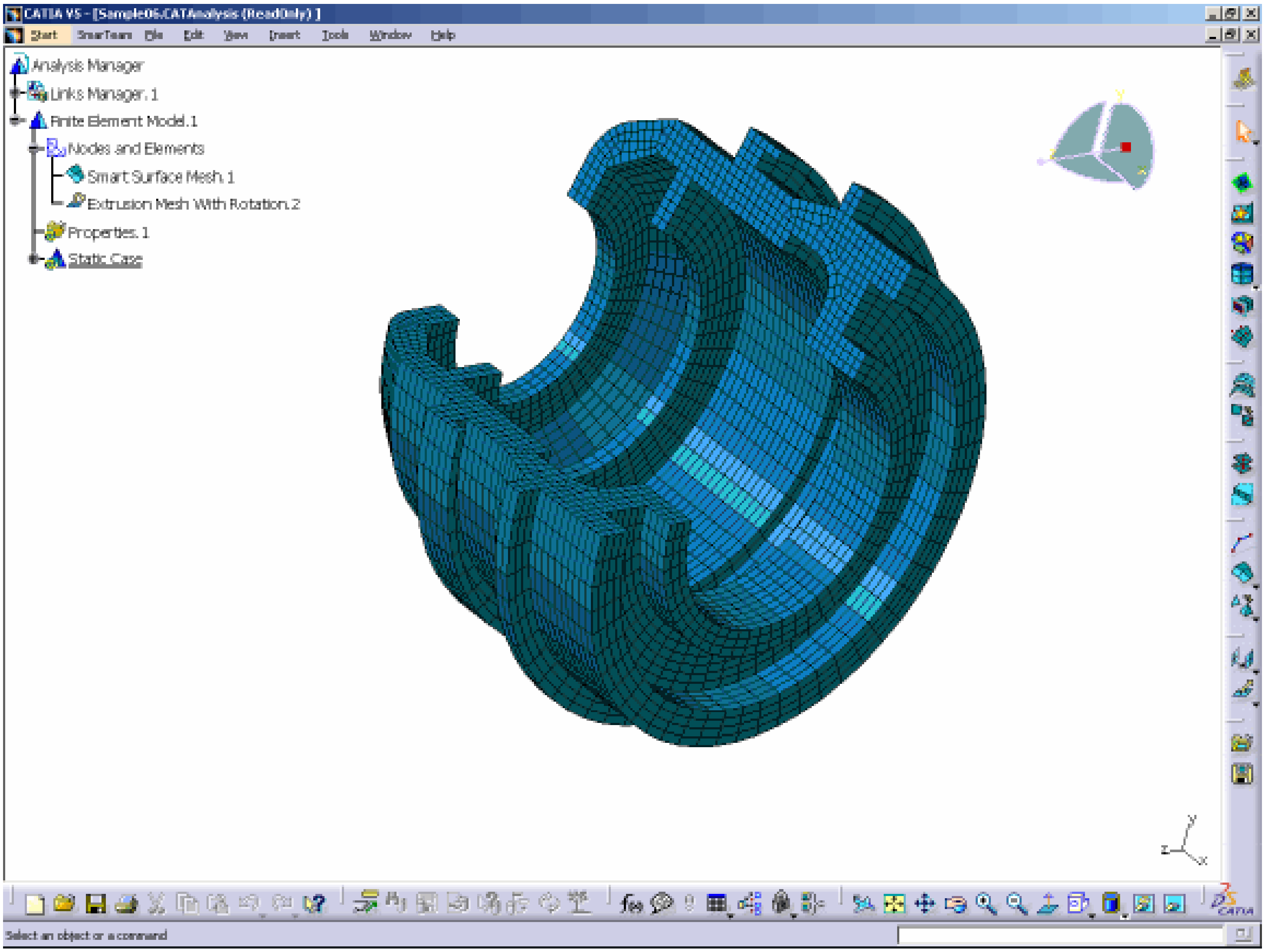


3. Click **OK** in the Split Quads dialog box.

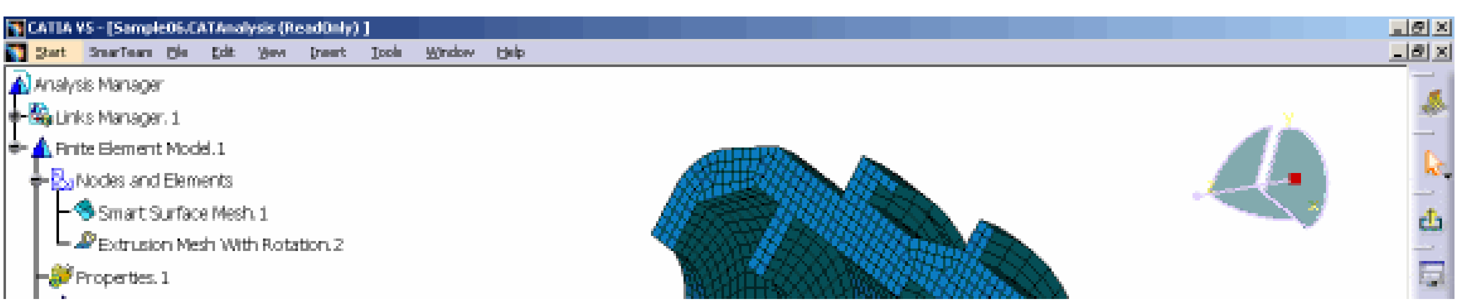
Workbench Description

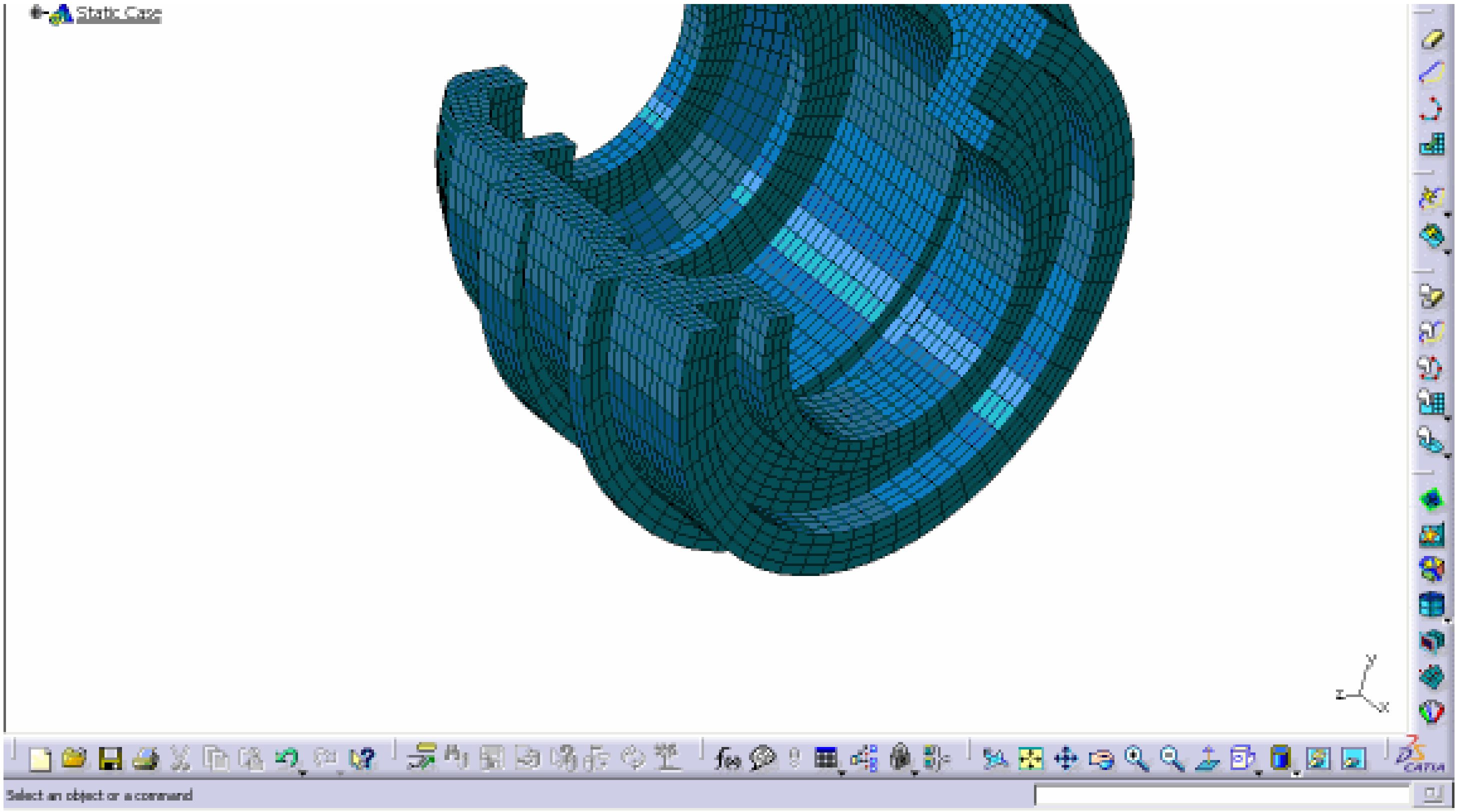
This section presents a description of the toolbars and specification tree of the Advanced Meshing Methods workbench.
You can click the sensitive areas on these images to see related documentation.

Advanced Meshing Tools Toolbars



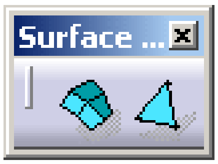
Surface Meshing Workshop Toolbars

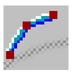




Meshing Methods Toolbar
Global Specifications Toolbar
Local Specifications Toolbar
Execution Toolbar
Edition Tools Toolbar
Exit Toolbar
Import/Export Toolbar
Welding Meshing Methods Toolbar
Mesh Analysis Tools Toolbar
Mesh Transformations Toolbar
Mesh Operators Toolbar
CATAnalysis

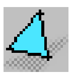
Meshing Methods Toolbar



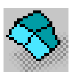


See Meshing using Beams

Surface Meshing Methods



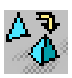
See Meshing using OCTREE Triangles



See Meshing the Surface Part

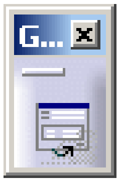
Solid Meshing Methods

See OCTREE Tetrahedron Mesher



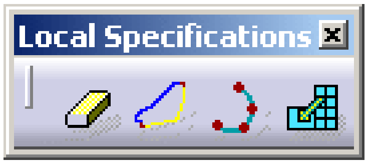
See Tetrahedron Filler

Global Specifications Toolbar

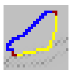


 See Setting Global Meshing Parameters

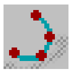
Local Specifications Toolbar



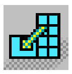
See Removing Holes
See Removing Cracks
See Removing Faces



See Adding/Removing Constraints (Specifications)

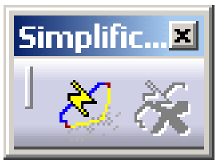


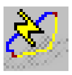
See Imposing Nodes (Specifications)

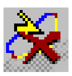


See Specify a domain

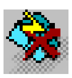
Execution Toolbar



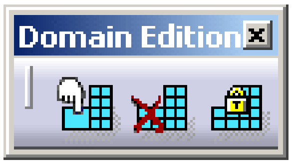
 See Simplifying the Geometry

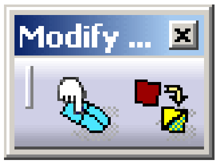
 See Removing the Geometrical Simplification

 See Meshing the Part

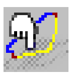
 See Removing the Mesh

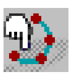
Edition Tools Toolbar

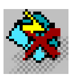




 See Cleaning Holes

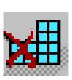
 See Adding/Removing Constraints (Modifications)

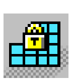
 See Imposing Nodes (Modifications)

 See Removing the Mesh

Domain Edition

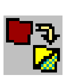
 See Re-meshing a Domain

 See Removing the Mesh by Domain

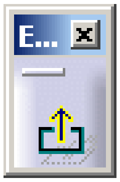
 See Locking a Domain

Modify Mesh

 See Mesh Editing

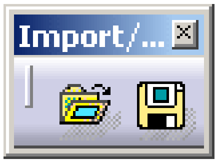
 See Splitting Quadrangles

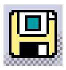
Exit Toolbar



 See Leaving the Surface Meshing Workshop

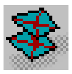
Import/Export Toolbar

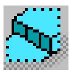


 See Exporting the Mesh

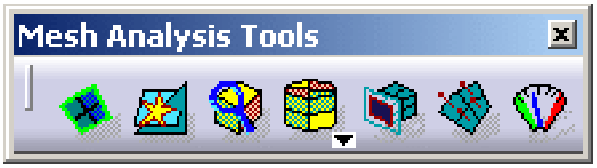
 See Importing the Mesh

Welding Meshing Methods Toolbar

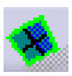
 See Meshing Spot Welds

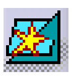
 See Meshing Seam Welding Connections

Mesh Analysis Tools Toolbar

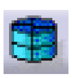


(in the **Surface Meshing** workshop)

 See Displaying Free Edges

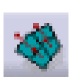
 See Checking Intersections / Interferences

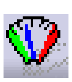
 See Analyzing Element Quality



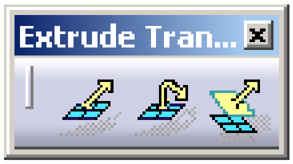
See Switching to Standard/Quality Visualization Mode

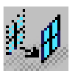
 See Cutting Plane

 See Displaying Element Orientation

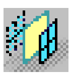
 See Returning Mesh Part Statistics (only available in the **Surface Meshing** workshop)

Mesh Transformations Toolbar



 See Translation

 See Rotation

 See Symmetry

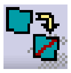
 See Extrusion by Translation

 See Extrusion by Rotation

 See Extrusion by Symmetry

Mesh Operators Toolbar

 See Offsetting the Mesh

 See Splitting Quads

CATAnalysis

A CATAnalysis file is composed of:

1. Links Manager, which references the part or the product to be analyzed.
2. Connection Design Manager, which contains the analysis design connections.
3. Finite Element Model, which contains the specifications of finite element model.

Analysis Manager

Links Manager

The **Links Manager** gives you the directory path and the main information on the linked documents or files.

- Product
- Part
- Results and Computations: gives you the directory path of the CATAnalysisComputations and the CATAnalysisResults files .

Connection Design Manager

The **Connection Design Manager** is composed of:

- General Design Connection
- Point Design Connection
- Point Design Connection within one Part
- Line Design Connection
- Line Design Connection within one Part

Finite Element Model

The **Finite Element Model** is composed of:

- Nodes and Elements
- Properties
- Modulation
- Group
- Analysis Case

Customizing

This section describes the different type of setting customization you can perform in the Analysis workbenches using the **Tools -> Options...** submenu.

This type of customization is stored in permanent setting files: these settings will not be lost if you end your session.

1. Select the **Tools -> Options...** submenu.

   The Options dialog box appears.

2. Select the **Analysis & Simulation** category.

   The following tab appears:

   These tabs lets you define the:

   - general settings
   - graphic settings
   - post processing settings
   - quality settings
   - storage settings

3. Change the desired parameters.

4. Click **OK** in the Options dialog box when done.

General

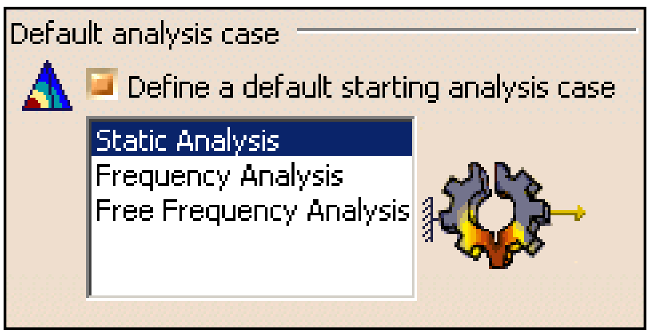
This task explains how to customize Analysis & Simulation general settings.

The General tab deals with the following settings:

- the default analysis case choice
- the specification tree display

Click here to see the parent page.

Default Analysis Case



Define a default starting analysis case

This option lets you define a default analysis case that will be inserted each time you enter the Generative Structural Analysis workbench or the Advanced Meshing Tools workbench.

Before defining a default analysis case using **Tools->Options** command, make sure you started the Analysis & Simulation (Generative Structural Analysis or Advanced Meshing Tools) workbench at least once.

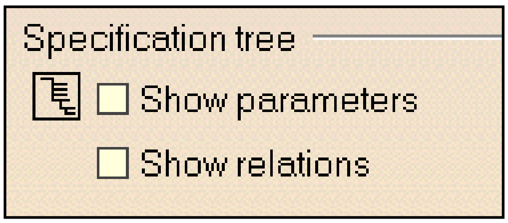
The default starting analysis case is Static Analysis. You can decide that the new default case will be:

- **Static Analysis**
- **Frequency Analysis**
- **Free Frequency Analysis**

The cases will only be displayed if an analysis workbench has been loaded at least once because the listed cases are linked to the Analysis workbenches last loaded.

By default, this option is deactivated.

Specification Tree



Show parameters

This option lets you display parameters in the specification tree.

By default, this option is deactivated.

Show relations

This option lets you display relations in the specification tree.

By default, this option is deactivated.

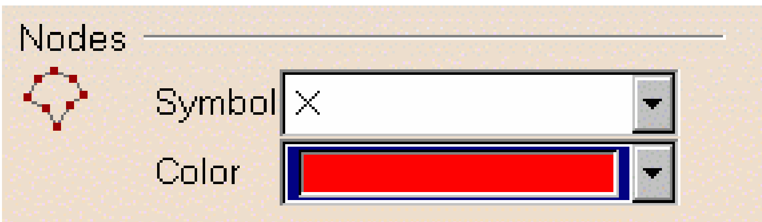
Graphics

This page deals with the following options:

- Nodes
- Elements

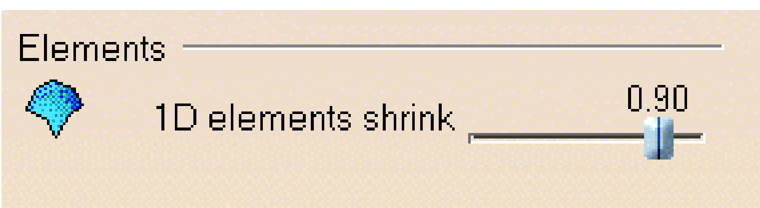
Click here to see the parent page.

Nodes



This option lets you select the symbol and color you wish to assign to the nodes.

Elements



This option lets you define the shrink of 1D elements.

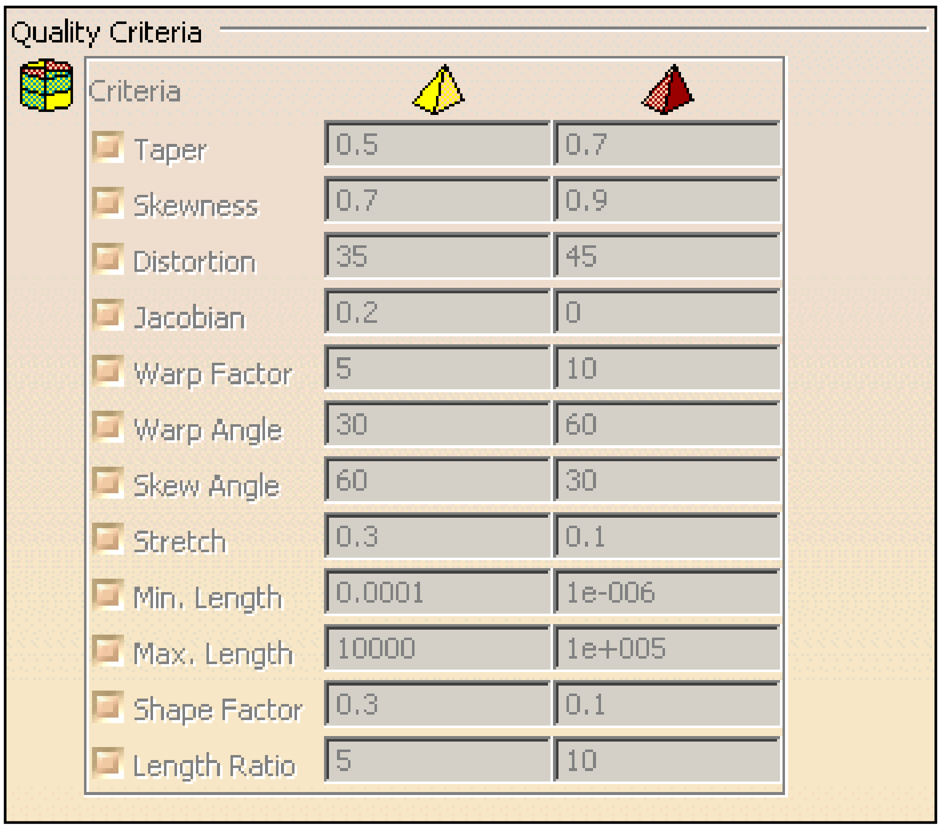
Quality

This page deals with the following options:

- Quality Criteria
- Default Standard File
- Export Default Format

Click here to see the parent page.

Quality Criteria



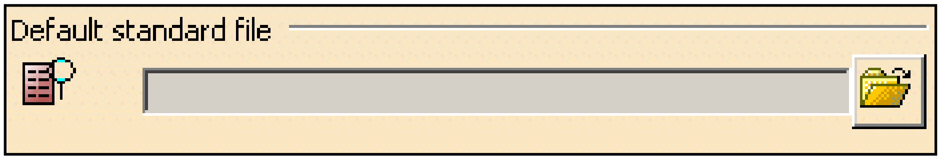
This frame lets you visualize the quality criteria that are taken into account and their limit values between:

- good and poor elements
- poor and bad elements

By default, all the **Quality Criteria** are taken into account.

The limit values change as you define the **Default Standard File** option.

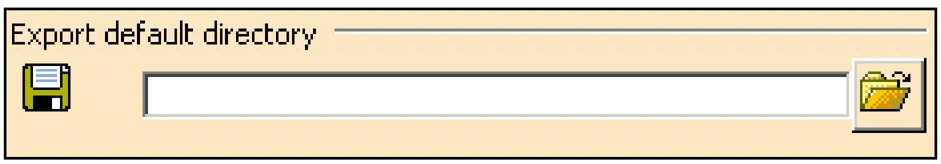
Default Standard File



This option lets you define the list of quality criteria that will be used by default.

By default, the **Default Standard File** field is empty and so all the **Quality Criteria** are taken into account.

Export Default Directory



This option lets you define the default directory in which the criteria configuration have been saved.

By default, the **Export Default Directory** field is empty.

While a default directory is not defined, you cannot use the **Export Criteria** option in the **Quality Analysis** functionality in the Advanced Meshing Tools workbench (for more details, please refer to the *Advanced Meshing Tools User's Guide - Analyzing Element Quality*)

Glossary

C D G I M P S

C

crack	A geometry defect that occurs when two adjacent faces, near the free edges, are not topologically linked.
constraint sag	A constraint is created along the edge of a face to avoid creating elements across this edge (the sag would be higher than the mesh size value. This does not guarantee that the whole mesh respects the sag value but helps creating constraints. For a given mesh size, the lower the constraint sag value, the more numerous the constraints are created, and vice versa.

D

domain	A sub part on a surface.

G

global size	The target size for element edges.

I

interference	A standard geometrical intersection.
intersection	A given distance (or clearance) beyond which the elements generated are positioned. This let's you take into account the real width of the geometry when detecting interferences.

M

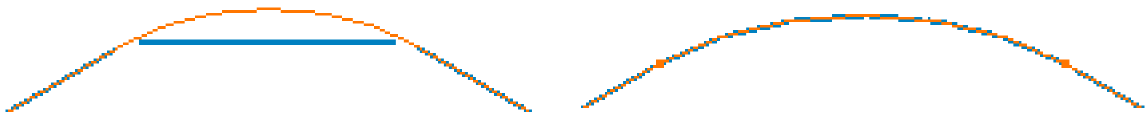
minimum size	The minimum size of an edge of an element. When the detail elimination option is active, the mesher does not generate elements with edges shorter than the minimum size.

P

part	A 3D entity obtained by combining different features in the Part Design workbench. Please see *Part Design User's Guide* for further information.

S

sag	The distance between the mesh elements and the geometry so that mesh refining is optimum in curve-type geometry.



Index

9 A B C D E F G H I L M N O Q R S T U V W

Numerics

A

B

C

D

E

F

L

M

N

O

Q

R

S

T