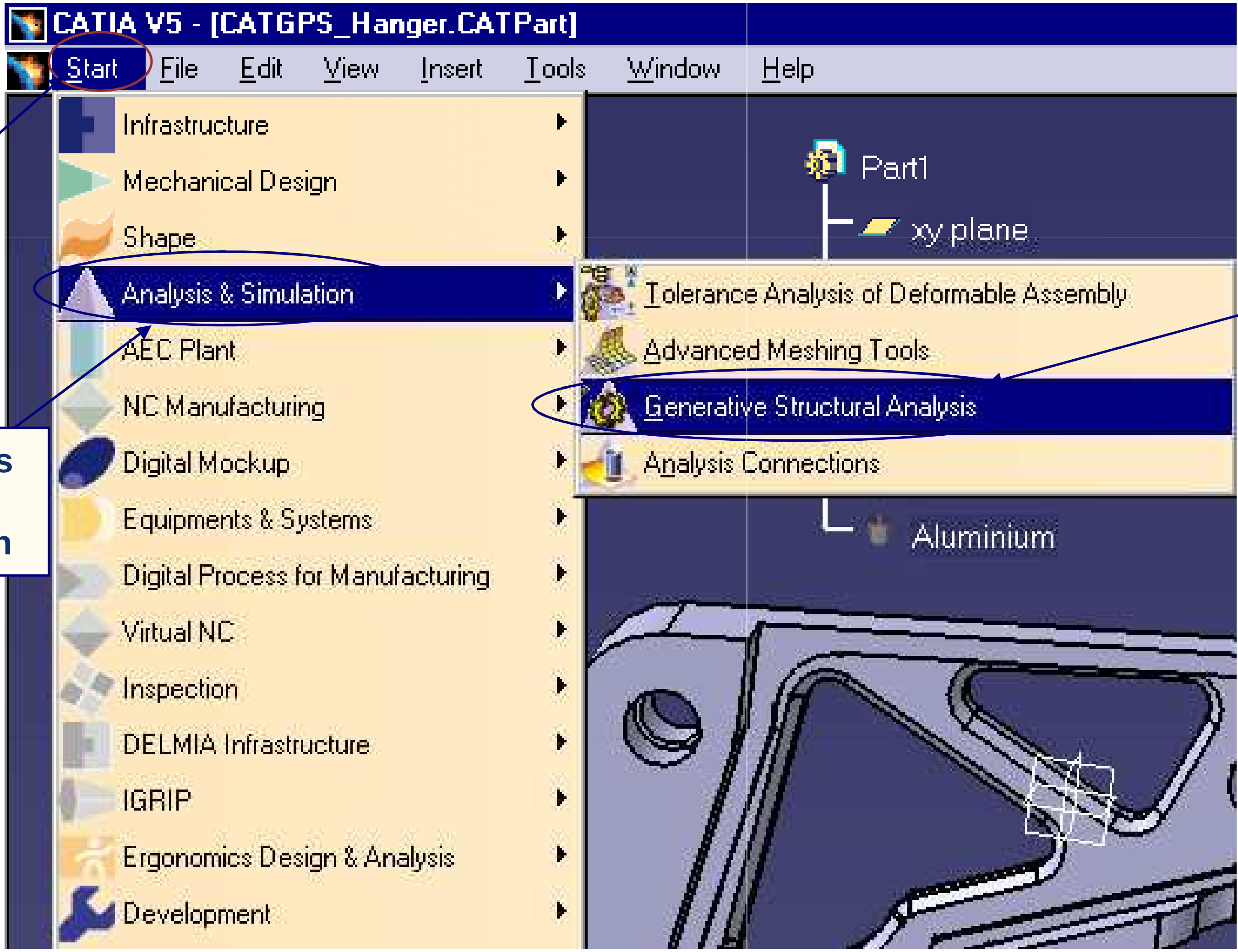
CATIA V5

- DMU ANALYSIS GPS -

Atelier GSA

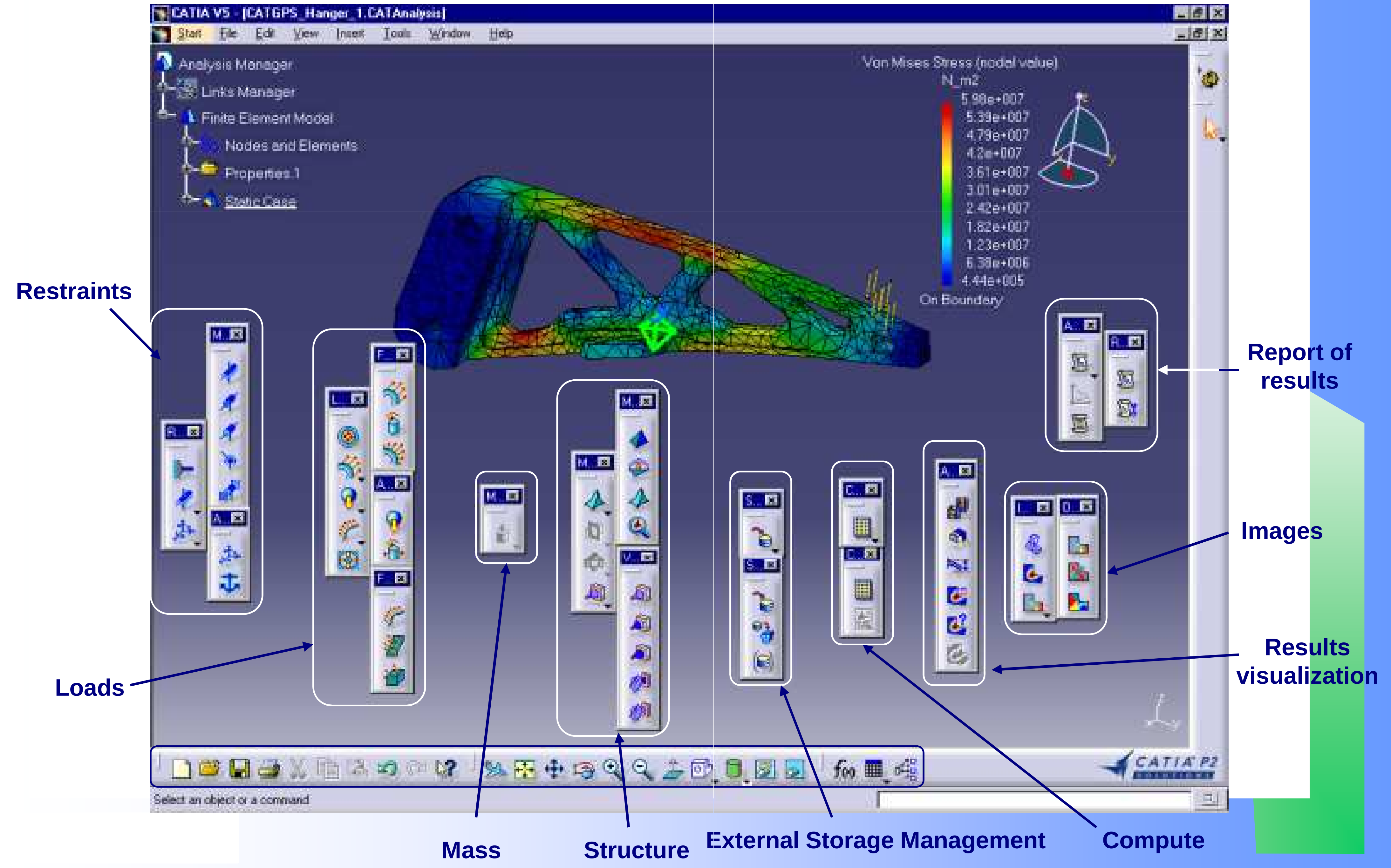


1- Start

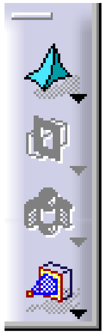
2- Analysis & Simulation

3- Modal or Static Analysis. A new CATAnalysis document is created.

Barres d'outils

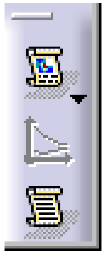


Icones



Structure

- Mesh Specification
- Connection
- Advanced Connection
- Virtual Part



Analysis Results

- Basic Analysis Report
- Historic Of Computations
- Listing

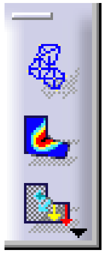


Image Creation

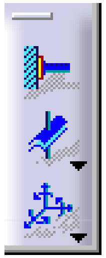
- Deformation
- Stress Von Mises
- Displacement

Solver Tools

- Storage Location
- Clear Storage
- Temporary Data Directory

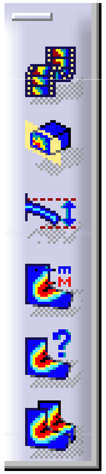
Mass Equipment

- Mass



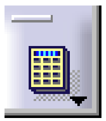
Restraints Application

- Clamp
- Mechanical Restraint
- Advanced Restraint



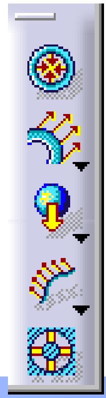
Results Visualization

- Animate
- Cut Plane Analysis
- Deformation Scale Factor
- Search Image Extrema
- Informations
- Image Layout



Computation

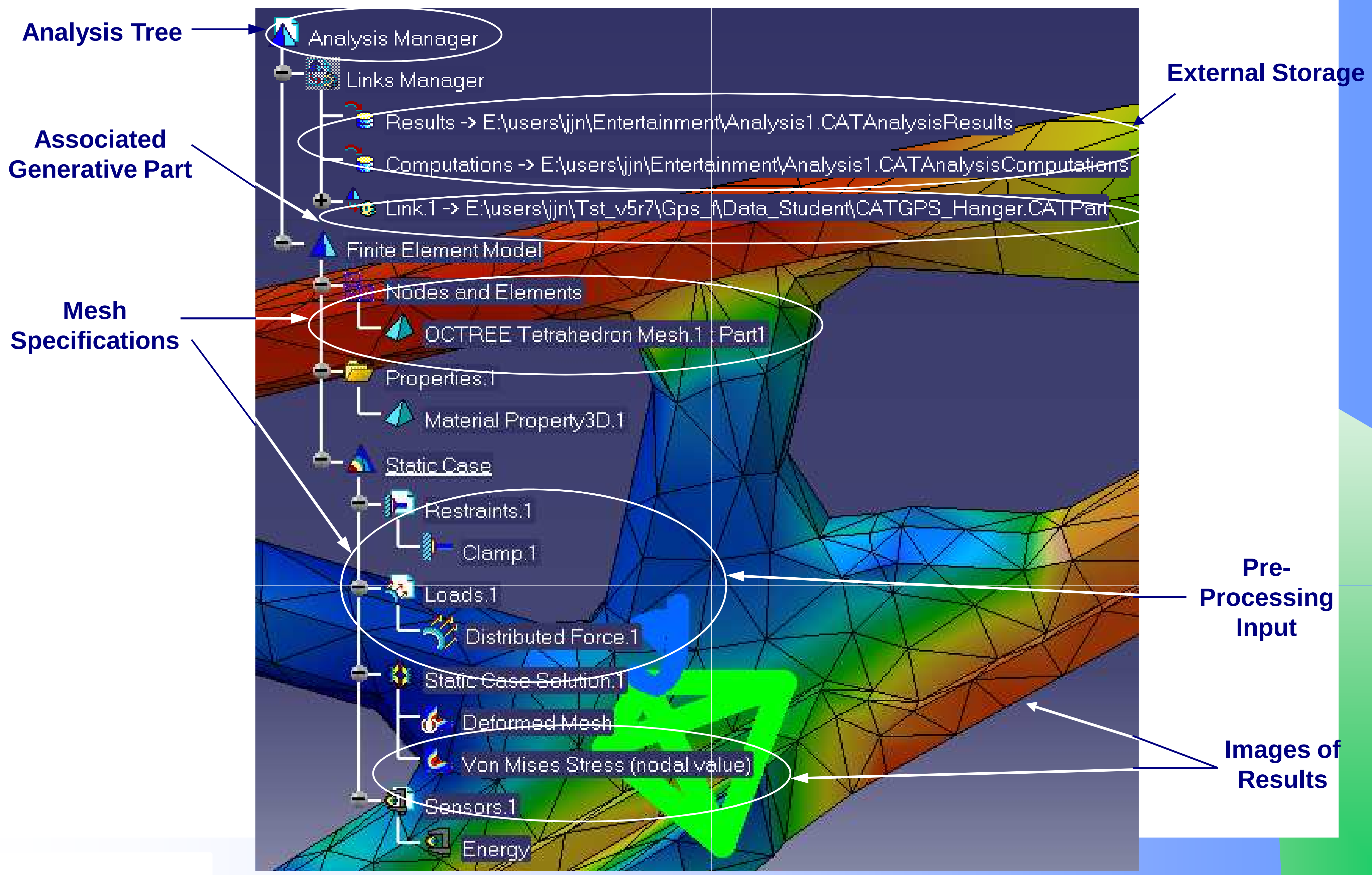
- Compute



Loads Application

- Pressure
- Force
- Acceleration
- Force Density
- Enforced Displacement

Arbre de Spécifications



Méthodologie

Ensure that part has material defined then Open Static Analysis Workbench

1 Apply Restraints to the model

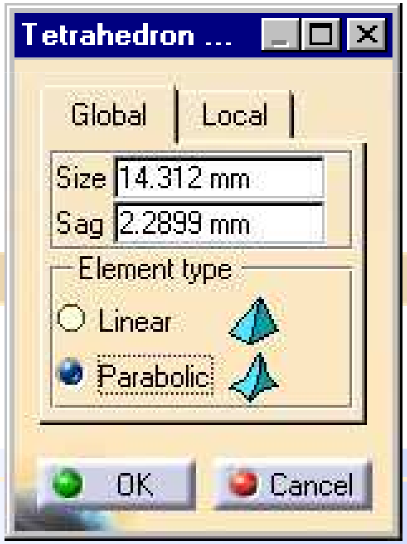
2 Apply Loads the model

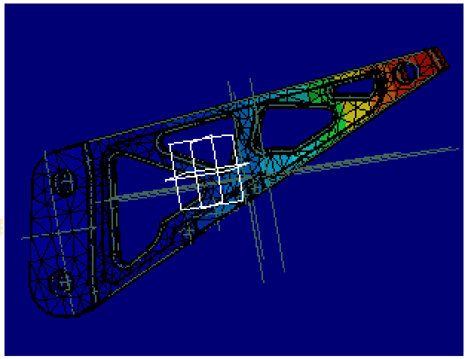
3 Perform Computation

4 Create Images

5 Analyze the Results

6 Refine the Analysis





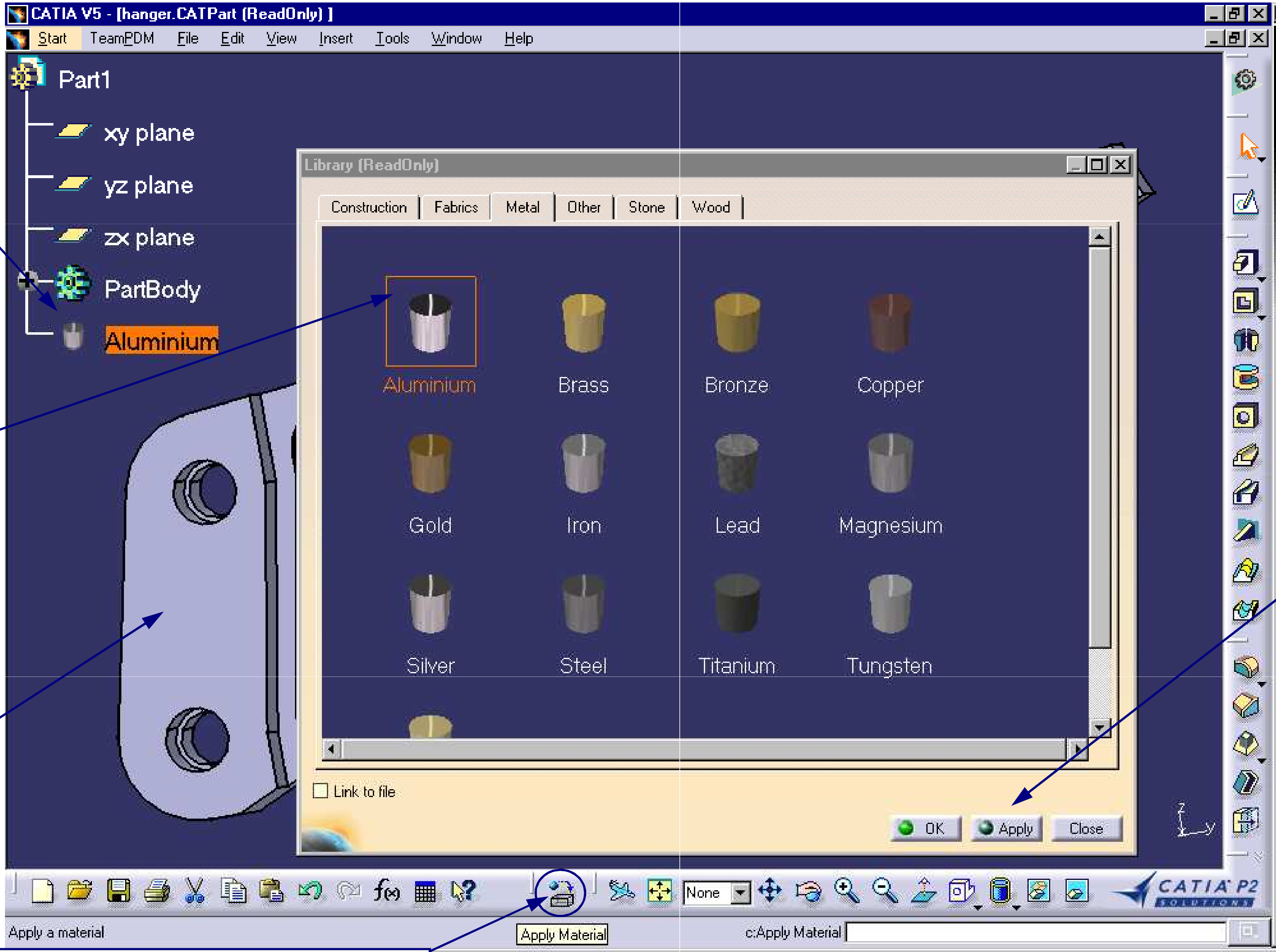
Affecter un Matériau

5- Check

2- Select a Material

3- Select a Part

4- Apply

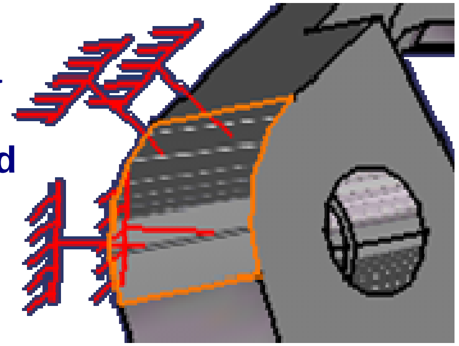


1- Click on the Material icon

Liaison - Clamps

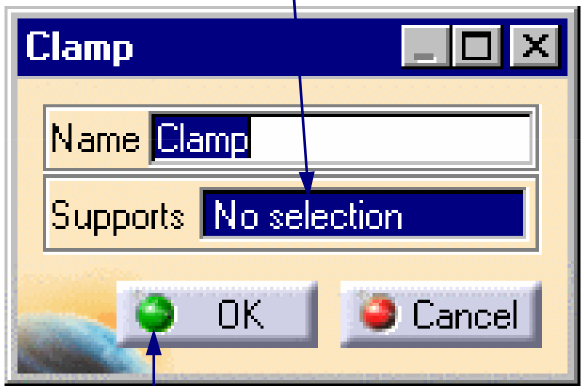
1- Click on the “Clamp” Icon in the “Restrain” Toolbar

Symbols associated to a null translation in all directions of the selected geometry are displayed.

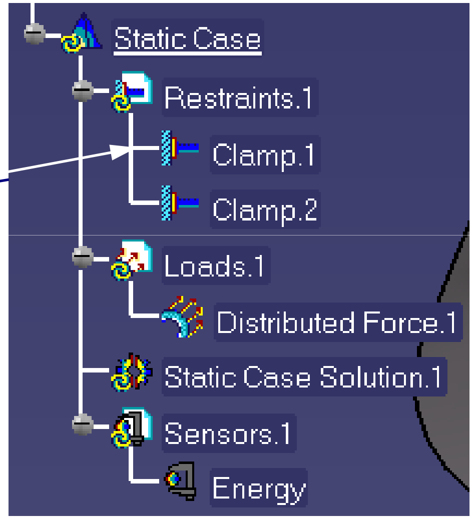


2- Select the geometry support(s) (Surfaces or Edges).

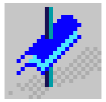
Any selectable geometry is highlighted when you drag the cursor over it.

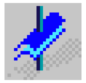


A Clamp object appears in the Specification Tree under the active Restraints objects set.



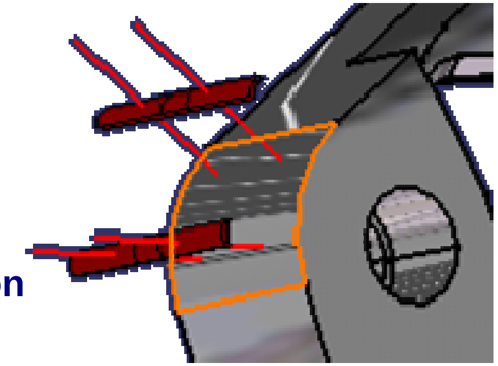
3- Click “OK”

Liaison - Surface Slider 

1- Click on Surface Slider Icon 

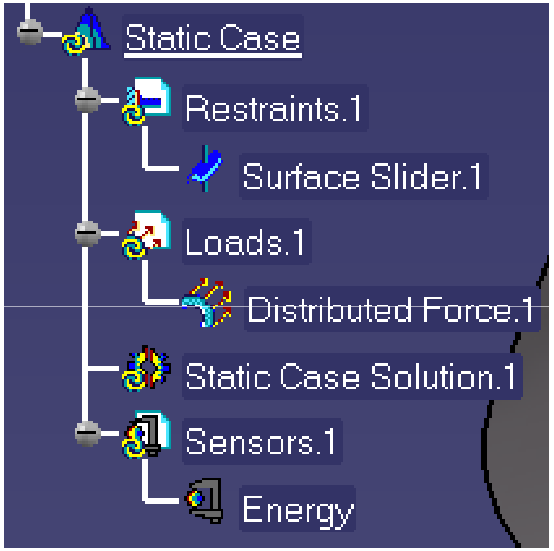
2- Select the Geometry Support(s) (Surfaces)

3- Click “OK”



Symbols on Geometry

Features Tree

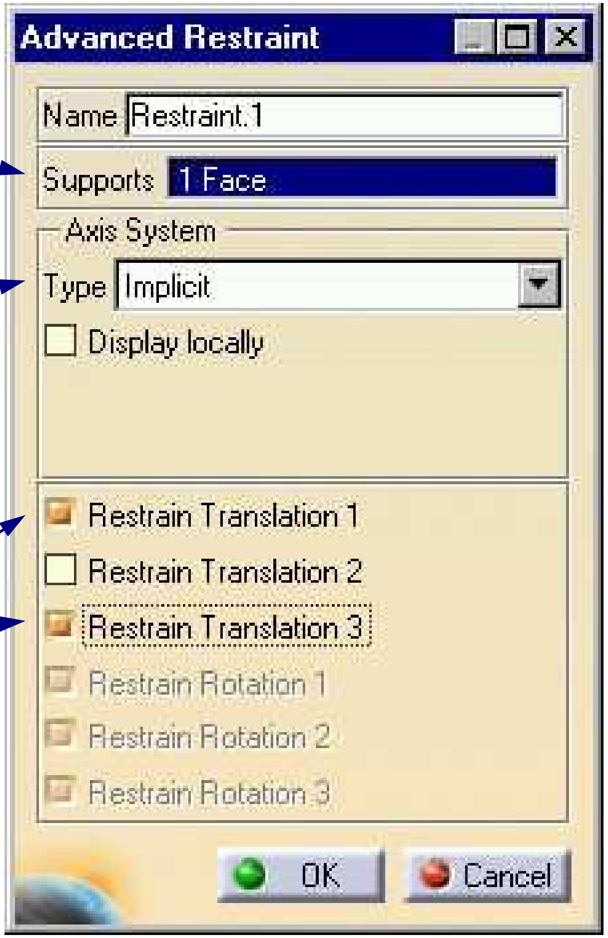


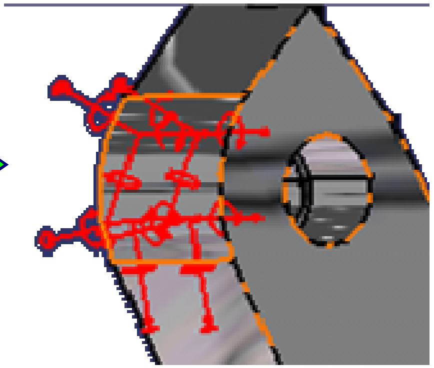
Liaison - Advanced Restraint

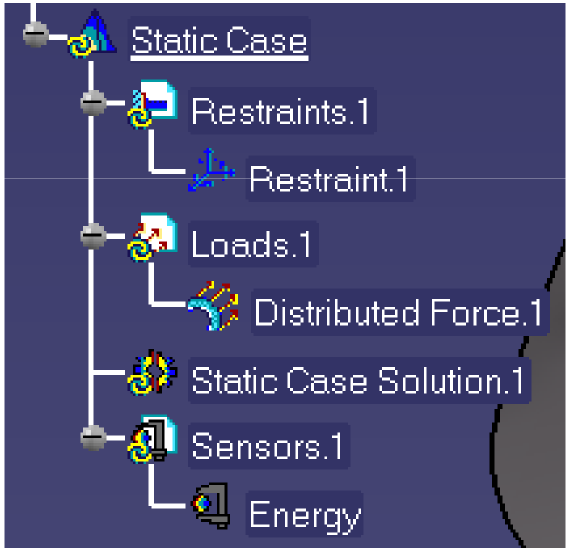
1- Select the support(s) (Surfaces or Edges)

2- Select axis type

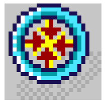
3- Activate degrees to be fixed







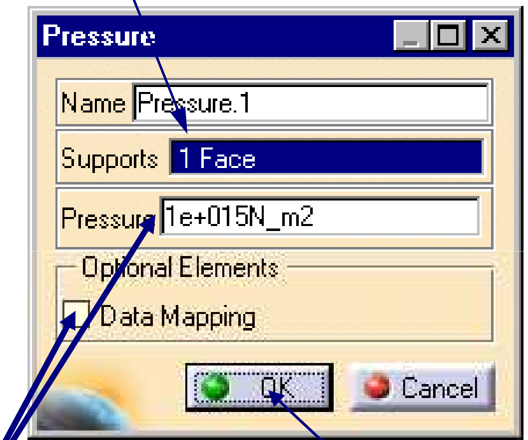
The rotation degrees are relevant only for structural element meshes (i.e. shell elements), or Virtual Parts.

Chargement - Pressure Load 

Pressures are intensive loads representing uniform scalar pressure fields applied to surface geometries, hence the force direction is everywhere normal to the surface.

1- Click on the "Pressure" Icon

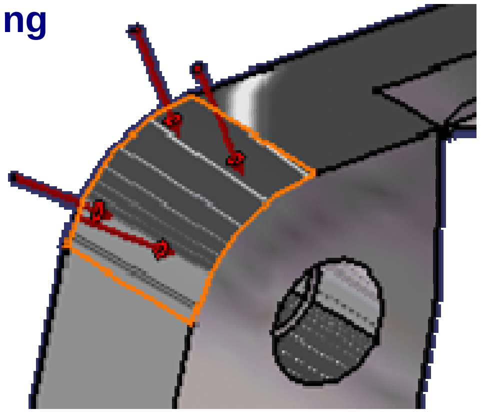
2- Select the geometry support(s) (Surfaces). Any selectable geometry is highlighted when you drag the cursor over it.



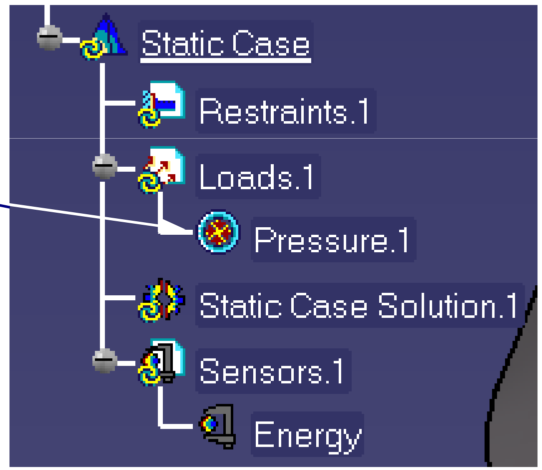
3- Specify a pressure value or Open a Data File for mapping

4- Click "OK"

Symbols representing the Pressure Loads are displayed.



A Loads object appears in the Features Tree under the active Loads objects set.

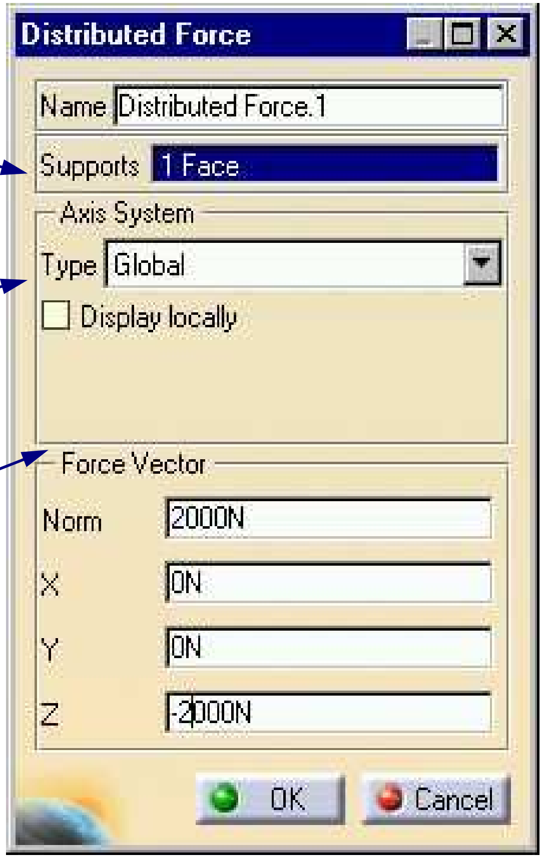


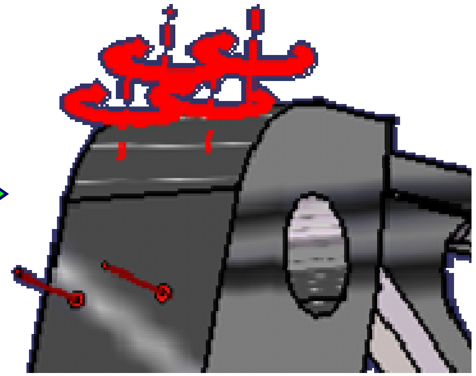
Chargement - Distributed Force & Moment

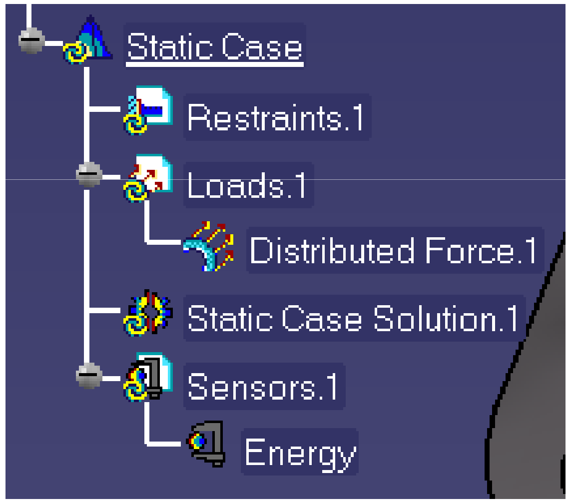
1- Select support(s) (Surfaces or Edges)

2- Select axis type

3- Specify force







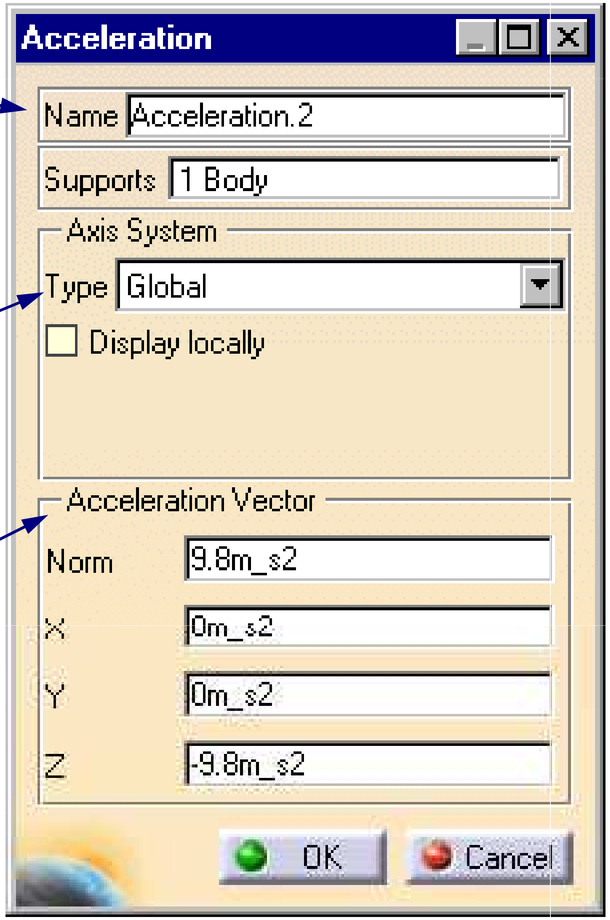
Chargement - Acceleration

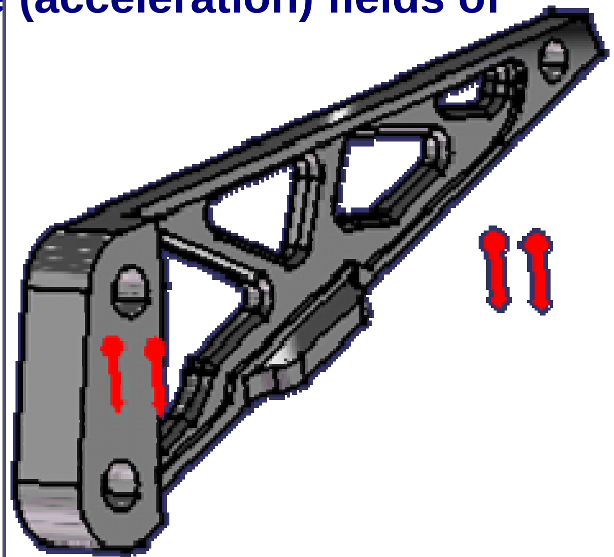
Accelerations are intensive loads representing mass body force (acceleration) fields of uniform magnitude applied to parts.

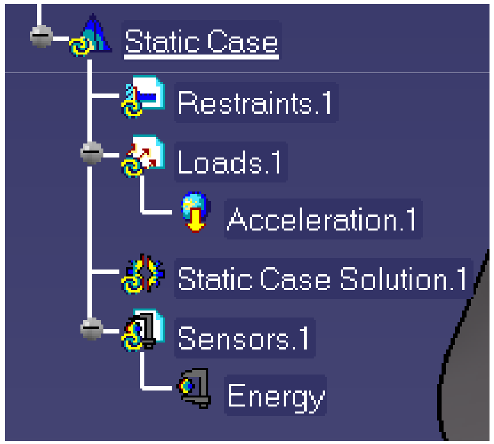
1- Select support(s) (Surfaces or Edges)

2- Select axis type

3- Specify vector







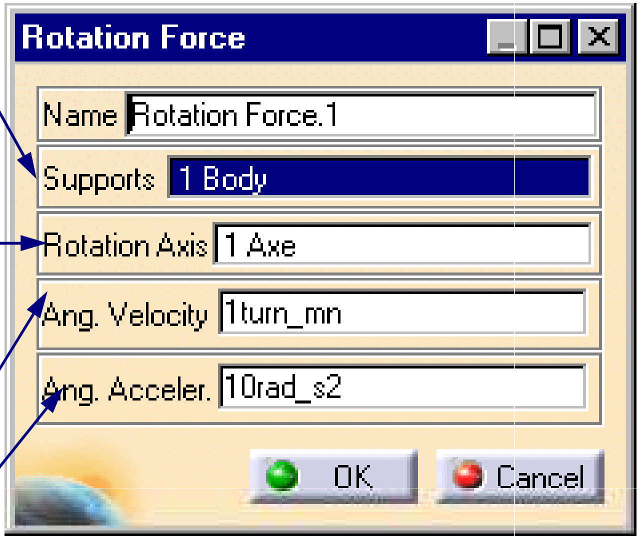
Chargement - Rotation Forces

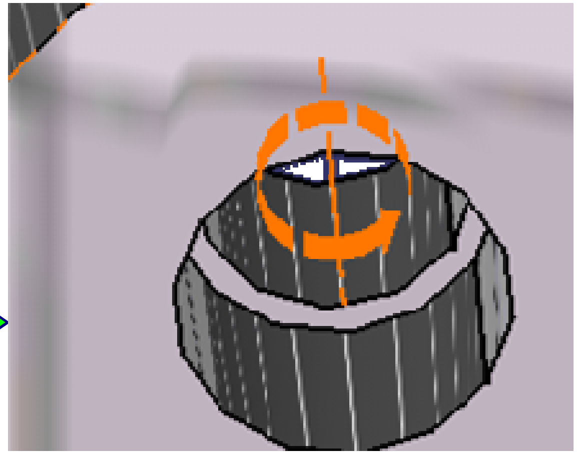
Rotation Forces are intensive loads representing mass body force (acceleration) fields induced by rotational motion applied to parts.

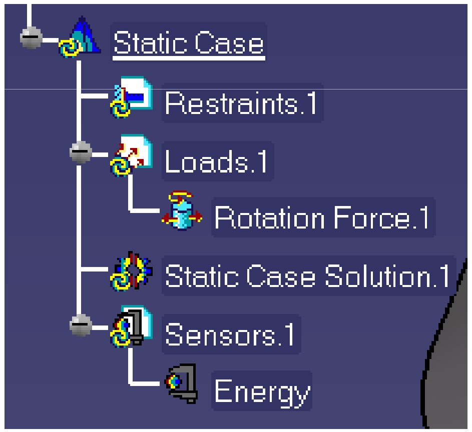
1- Select support(s) (Surfaces or Edges)

2- Select axis

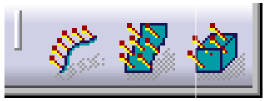
3- Specify angle velocity and acceleration (if necessary)







Chargement - Force Density



Force Densities are intensive loads representing line (surface) traction fields or volume body force fields, of uniform magnitude, applied to either curve (surface) geometries, or to parts.

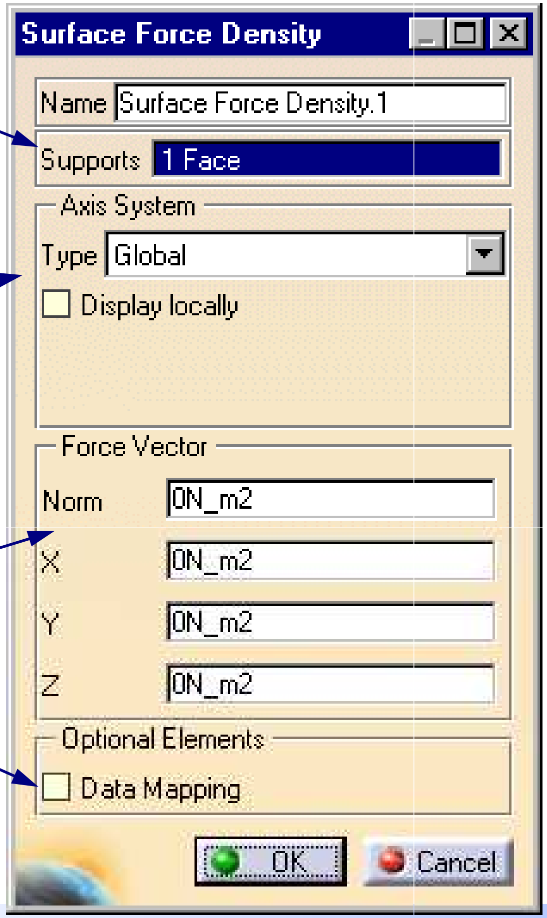
1- Select support(s)

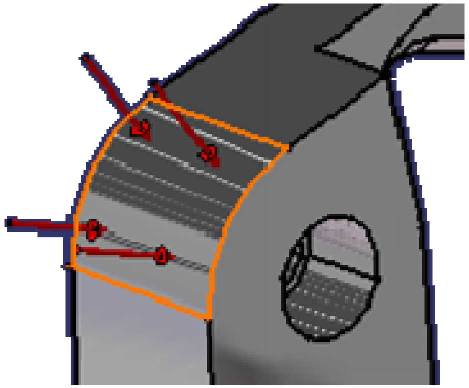
2- Select axis type

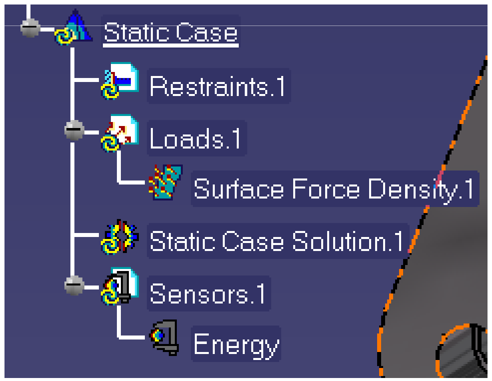
3- Specify vector

Or

Open a data file for mapping an external load



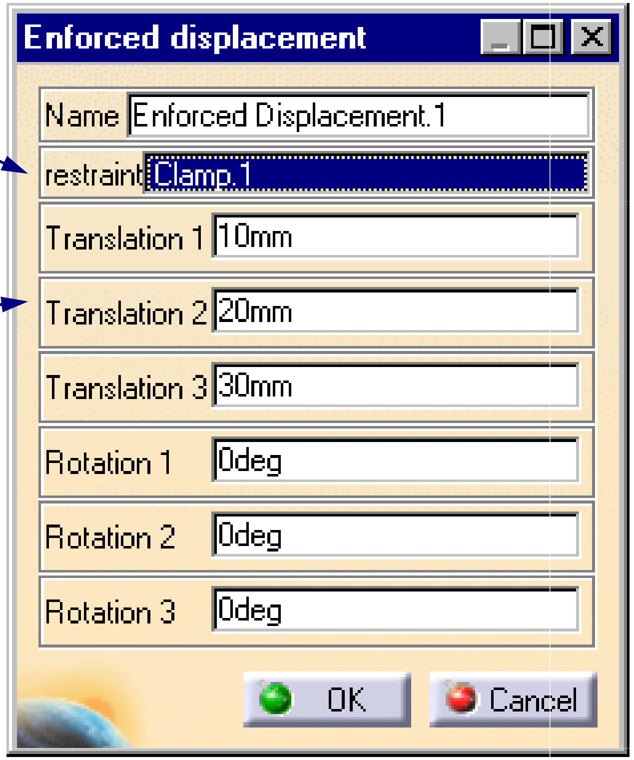


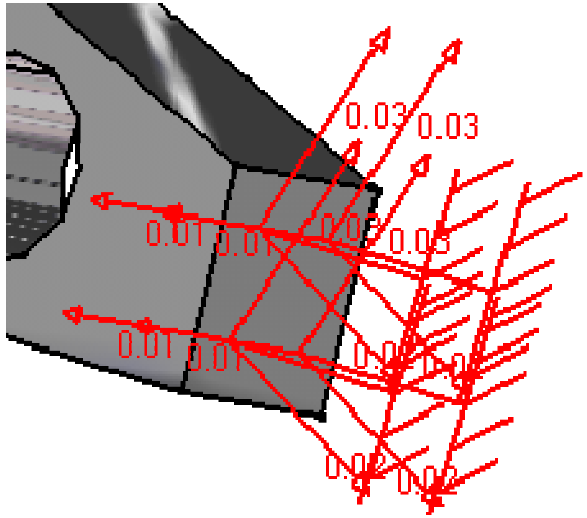


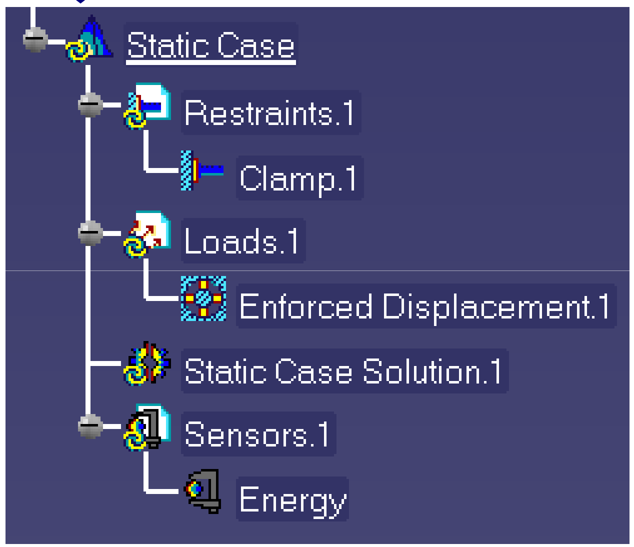
Chargement - Enforced Displacements

1- Select restraint

2- Enter values for each restrained dof





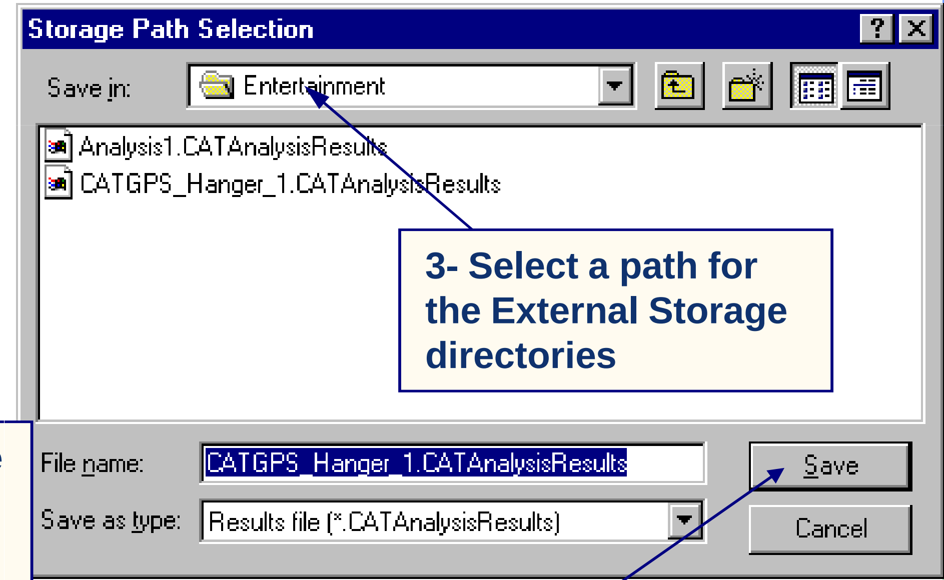


Données Externes

1- Click on the ‘Storage” icon in the Solver Tools toolbar

The “Elfini Storage Location” dialog box is displayed

2- Click the Modify button

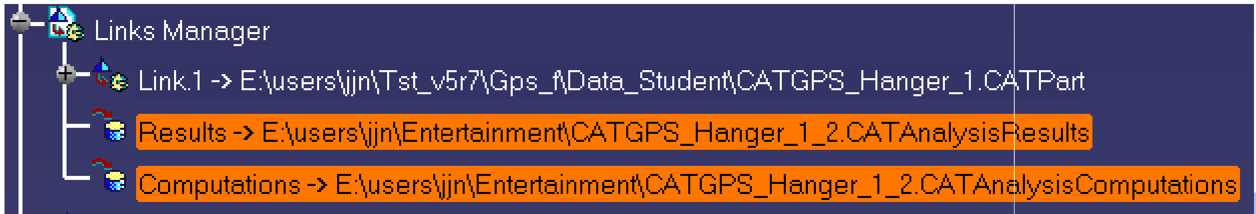


3- Select a path for the External Storage directories

4- Click Save

5- Click OK

The other method is to use the tree specifications.



Computing

Once you have successfully defined Restraints and Loads in your Static Analysis Case, you can undertake the actual results computation of that case.

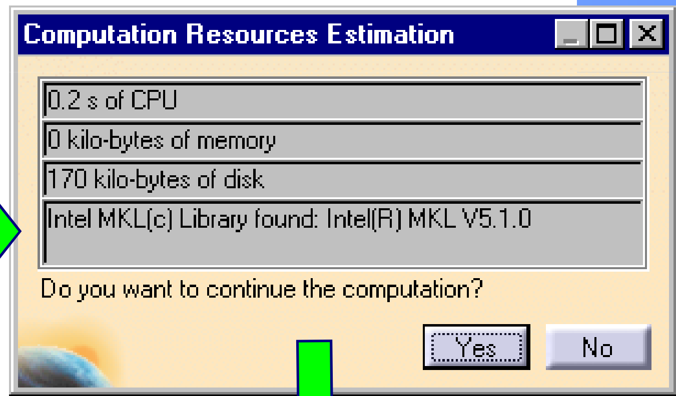
1- Click the Compute icon.

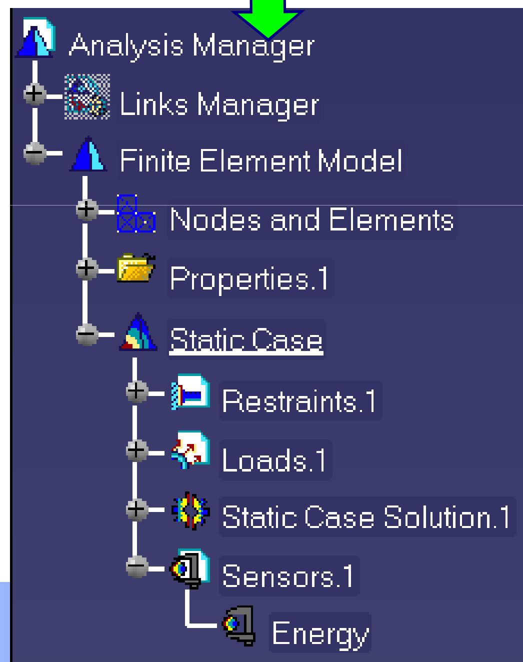
2- Activate the All (default) option.

Activate "Preview" if you want an estimation of the computation time.

3- Click OK (or Yes) to launch the computation.

The Compute dialog box is displayed.





Upon successful completion of the computation the status of all objects in the analysis features tree is changed to valid.

A series of status messages (Meshing, Factorization, Solution) informs you about the progress of the computing process.

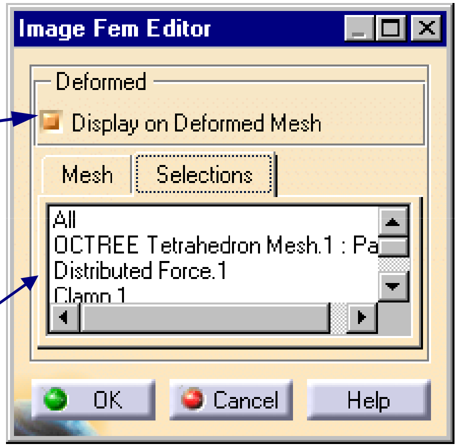
Résultats - Image Déformation

Deformed Mesh images are used to visualize the finite element model in its deflected configuration, as a result of the environmental action (loading).

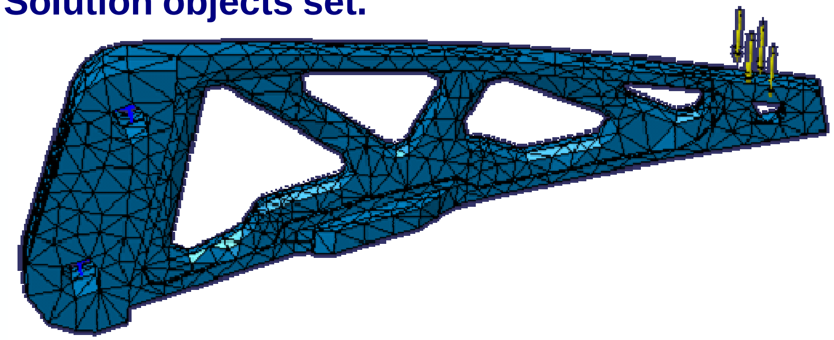
1- Click the Deformation icon.

If you de-activate this button you get the initial, i.e. undeformed, mesh.

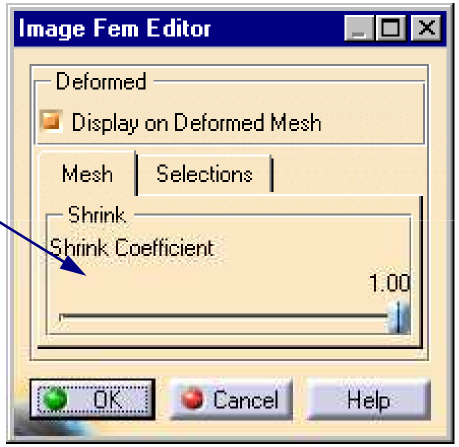
You can choose to see just one entity.



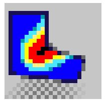
The Deformed Mesh image is displayed and a Deformed Mesh Image object appears in the feature tree under the active Static Case Solution objects set.



You can also set a shrink coefficient for all the elements of your mesh.



2- To customize the visualization, double-click the Deformed Mesh Image object in the feature tree to edit the image. The "Image Fem Editor" dialog box is displayed.

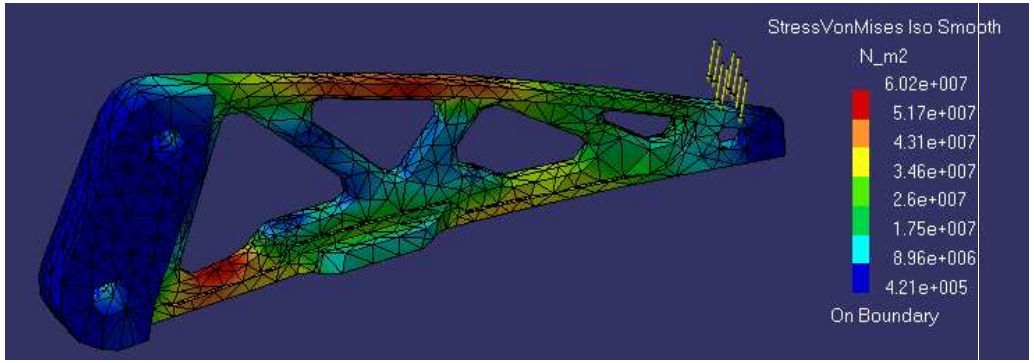
Résultats - Image Von Mises 

Von Mises Stress images are used to visualize Von Mises stress field patterns, which represent a scalar field quantity obtained from the volume distortion energy density and used to assess the state of stress.

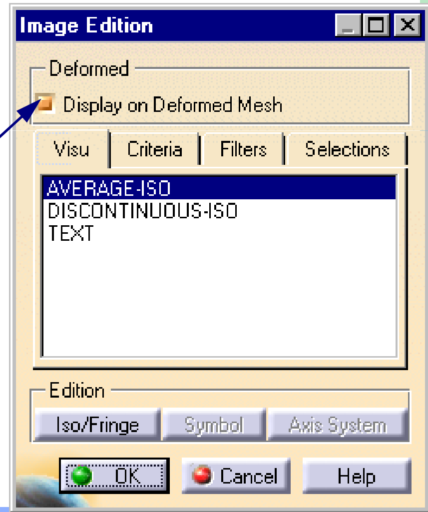
1- Click the "Von Mises' icon 

2- To customize the visualization, double-click the Von Mises Stress Image object in the feature tree to edit the image. The "Image Editor" dialog box is displayed.

The Von Mises stress distribution on the part is visualized in iso-value mode, along with a color palette, and a Stress Von Mises Image object appears in the feature tree.



If you de-activate this toggle button the Von Mises stress image is displayed on the undeformed mesh.



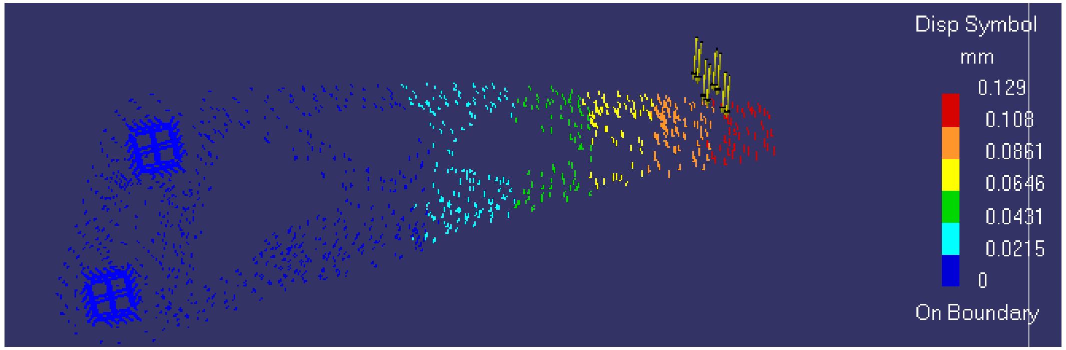
For a sound structural design, the maximum value of the Von Mises stress is generally considered to be less than the material yield stress value.

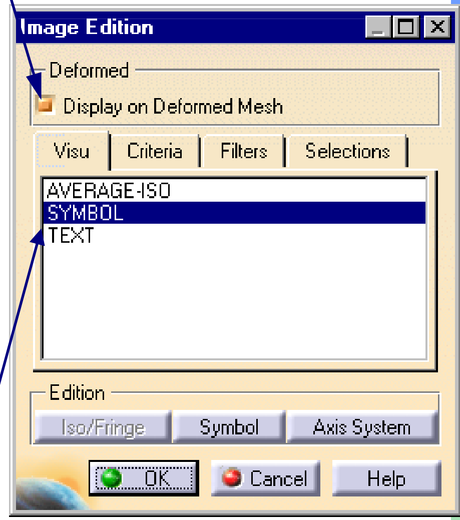
Résultats - Image Déplacements

1- Click the “Displacements” icon

If you de-activate this button the Displacements image is displayed on the undeformed mesh.

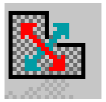
The Displacements distribution on the part is visualized in arrow symbol mode, along with a color palette.



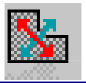


2- To customize the visualization, double-click the Displacements Image object in the feature tree to edit the image. The “Image Editor” dialog box is displayed.

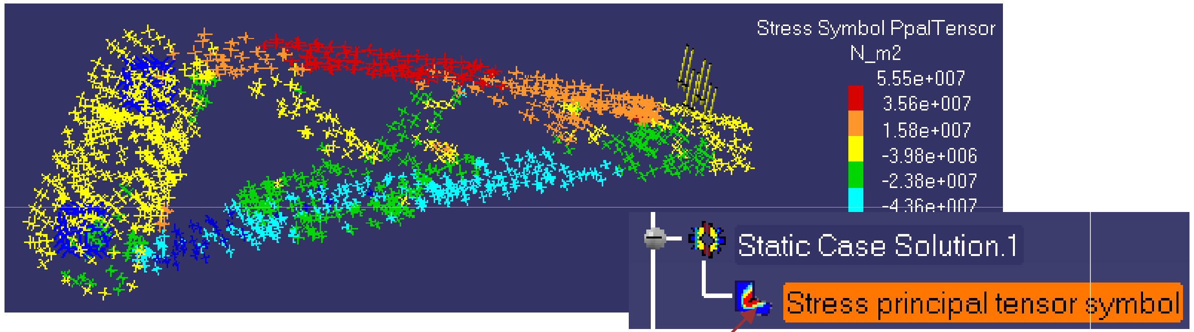
You can choose between a symbolic view (vectors) or an average-iso view (colors), and filter the desired displacement vector’s components.

Résultats - Contraintes Principales 

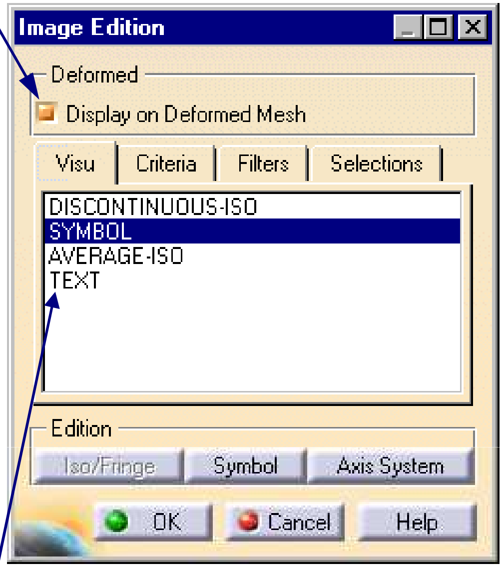
At each node, the principal stress tensor shows the directions along which the part is in a state of pure tension/compression and the corresponding tensile/compressive stresses.

1- Click the “Principal Stress“ icon 

The Principal Values Stress Tensor distribution on the part is visualized in symbol mode, along with a color palette : at each point, a set of three directions is represented by line symbols (principal directions of stress). Arrow directions (inwards / outwards) indicate the sign of the principal stress. The color code provides quantitative information.



If you de-activate this button the Principal Stress image is displayed on the undeformed mesh.



2- To customize the visualization, double-click the Principal Stress Image object in the feature tree to edit the image. The “Image Editor” dialog box is displayed.

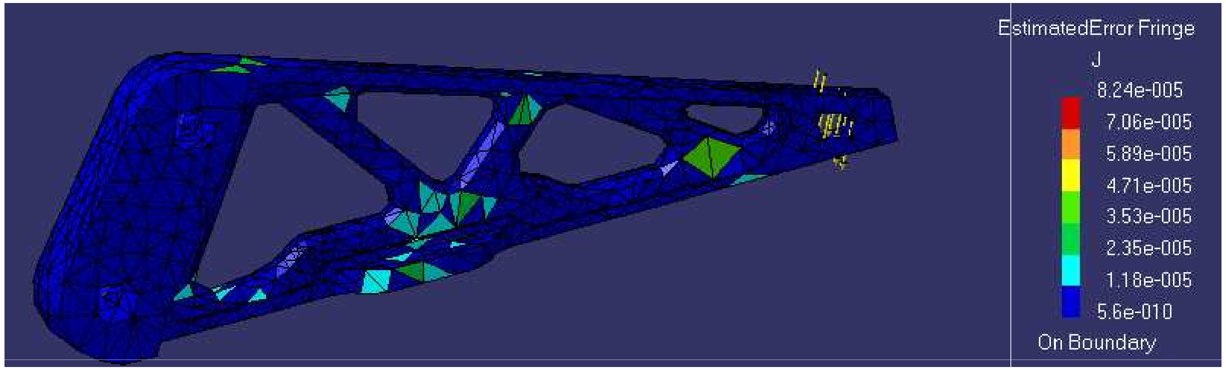
You can choose between a symbolic view or discontinuous-iso view (colors), and filter the desired principal stress tensor’s components.

Résultats - Image Précision

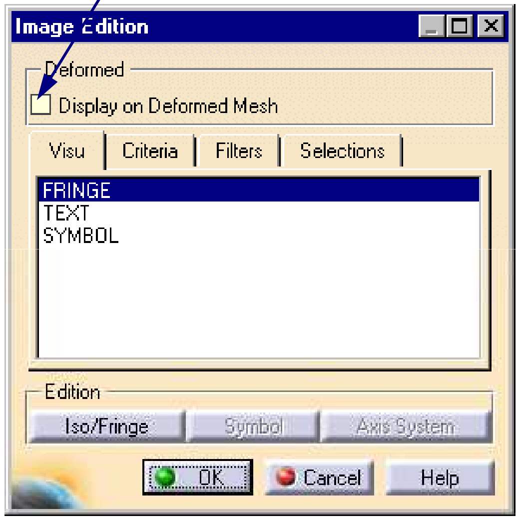
Estimated Error images are used to visualize computation error maps, and evaluate the validity of the computation. It displays a predicted energy error norm map which gives qualitative insight about the error distribution on the part.

1- Click the “Precision” icon

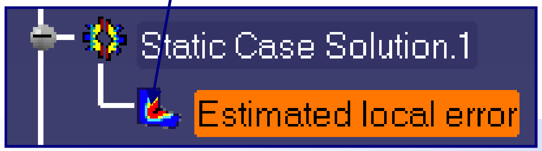
The Estimated Error distribution on the part is visualized in fringe pattern mode, along with a color palette.



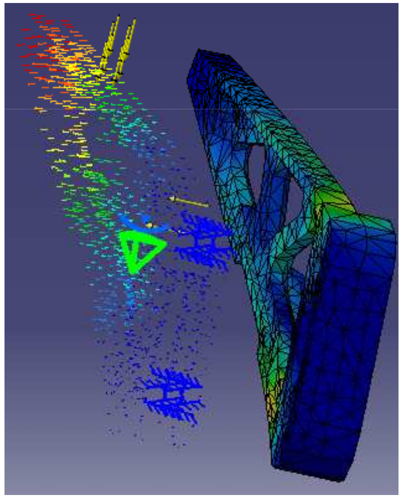
If you activate this button the precision image is displayed on the deformed mesh.

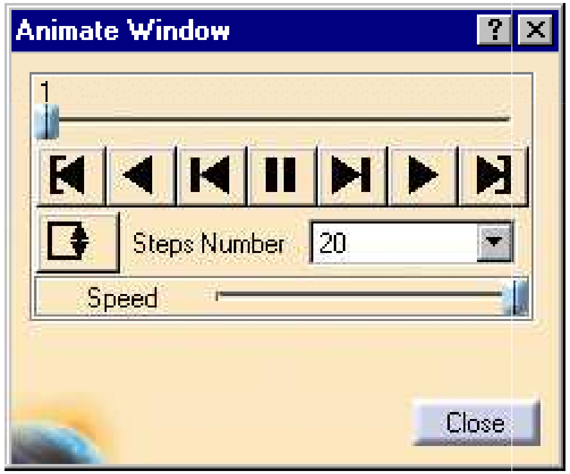


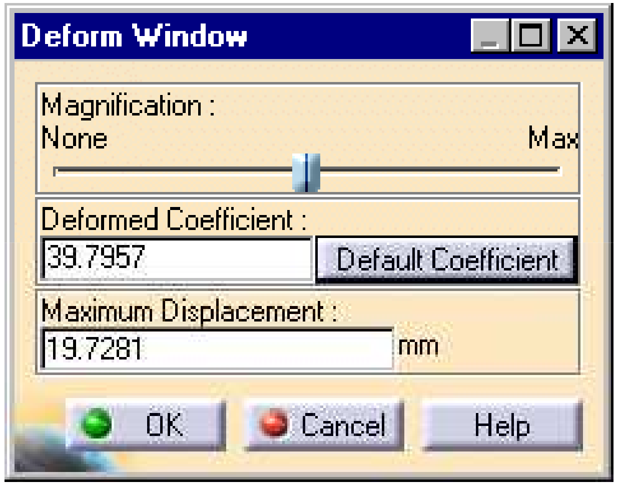
2- To customize the visualization, double-click the precision Image object in the feature tree to edit the image. The “Image Editor” dialog box is displayed.

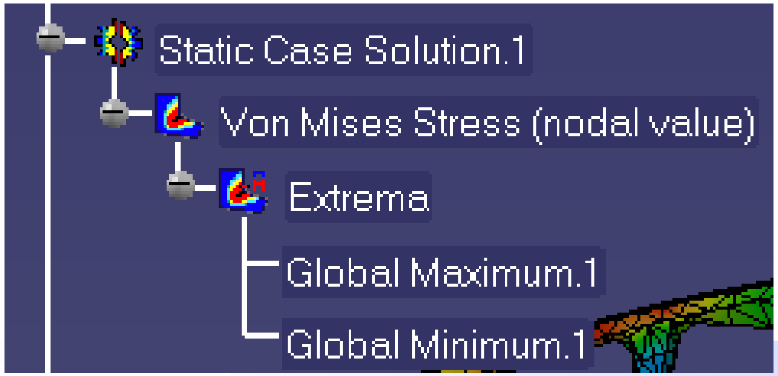


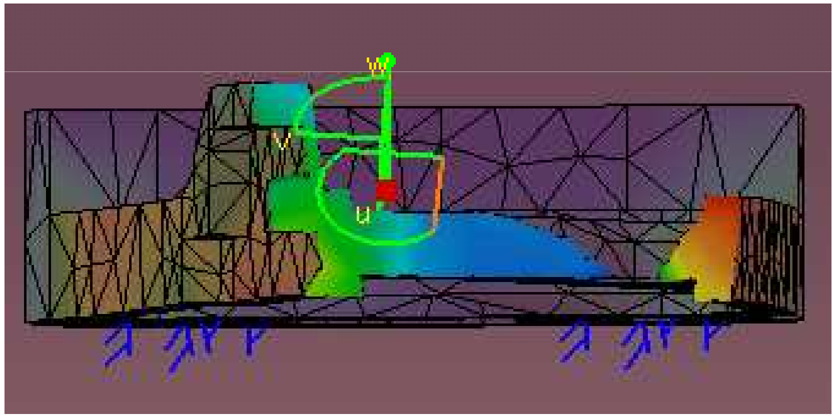
Résultats - Image Management









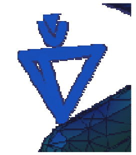


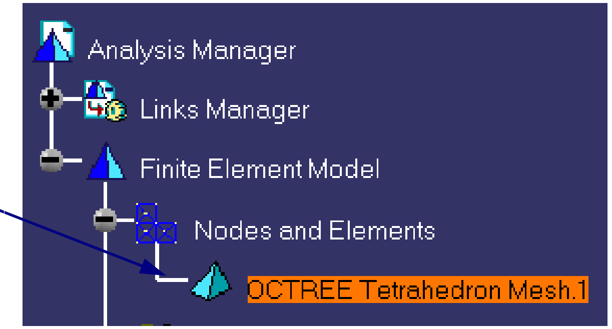
Remaillage Global & Local

The second step when you want to improve the precision of your analysis results is to refine the mesh of your part. You can refine both the Size of a mesh, and the Sag (chord error). This can be performed both globally or locally.

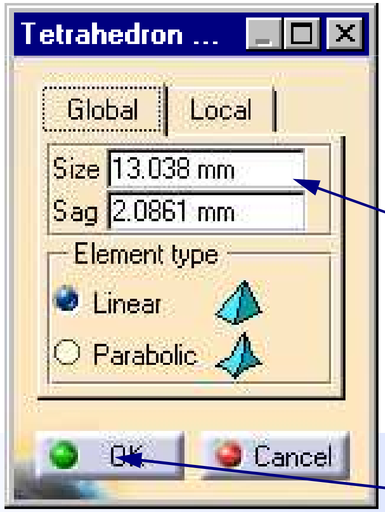
The mesh size is the dimension of the element edge and the sag is a measure of how closely the element boundaries follow the geometrical support. The smaller the mesh size and sag, the more accurate your analysis results will be.

1- Click either on the mesh specifications symbol or on the corresponding feature in the analysis tree.

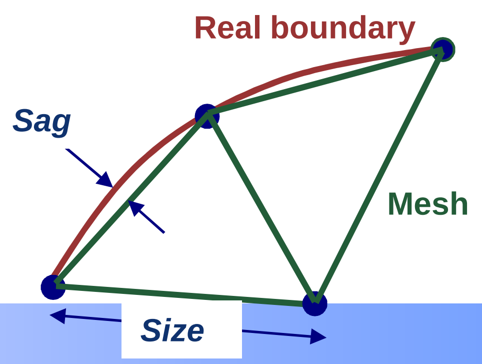




The mesh specifications box is displayed.



2- Reduce the global size and/or sag.



3- Click OK

Remaillage local - Adaptivity Boxes

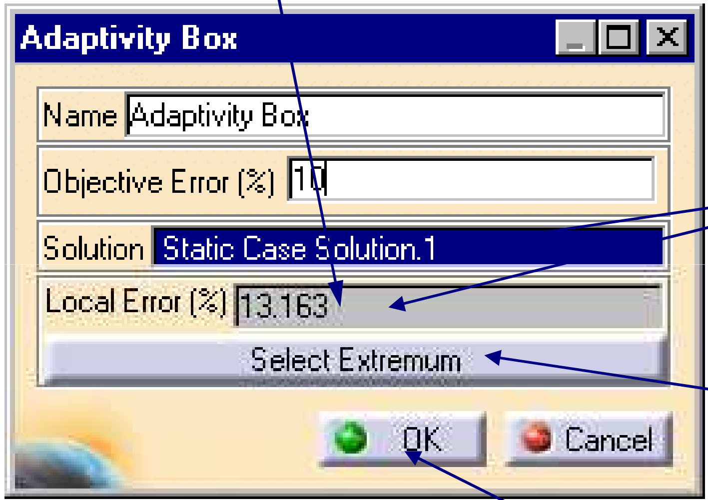
A powerful tool of GPS is the possibility to refine a given mesh only in areas of interest.

You specify the areas where you want refinement to be managed with the so called Adaptativity Boxes.

1- Perform a static analysis and compute a Static Case Solution

2- Click the first Adaptivity Box icon.

The Local Adaptivity Box dialog box is displayed, and current local error inside the box is indicated.

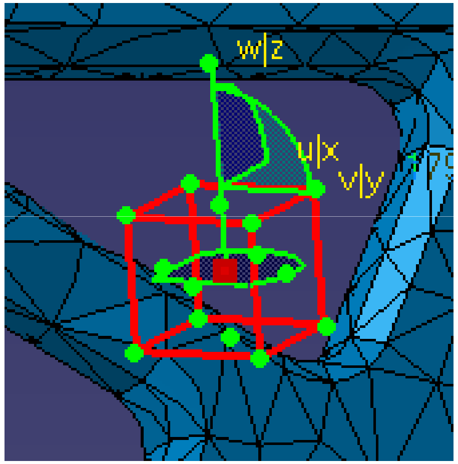


3- Select existing Static Case Solution and specify target percentage error in Objective Error box.

Click to centralize the box on the extremum

4- Click OK

A cuboid symbol representing the Adaptivity Box is displayed on the part.

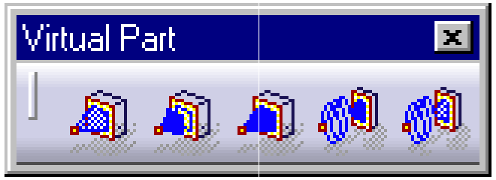


Piece Virtuelle

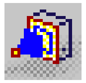
Virtual Parts transmit actions (masses, restraints and loads) applied at the handler point, to the geometries to which they are attached.

The handler point is either user-specified, or automatically defined as the centroid of the targeted geometry.

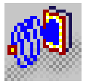
- Each Virtual Part type transmits its action to the real Part to which it is attached in a specific way.

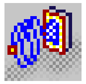


 **Smooth Virtual Parts softly transmit their actions : they don’t stiffen the deformable body.**

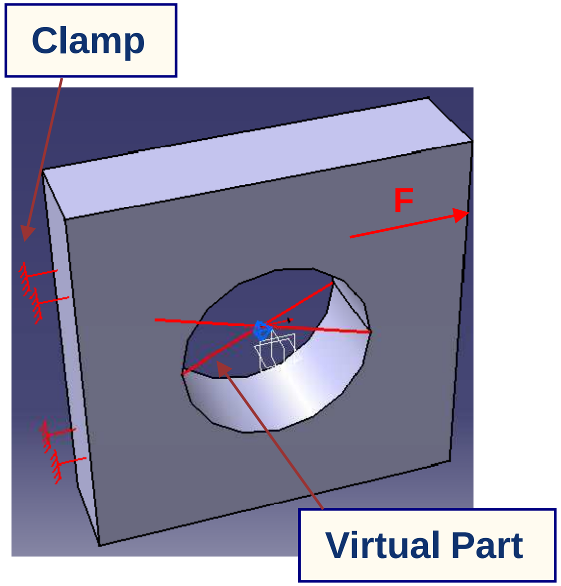
 **Contact Virtual Parts softly transmit their actions while preventing from body inter-penetration.**

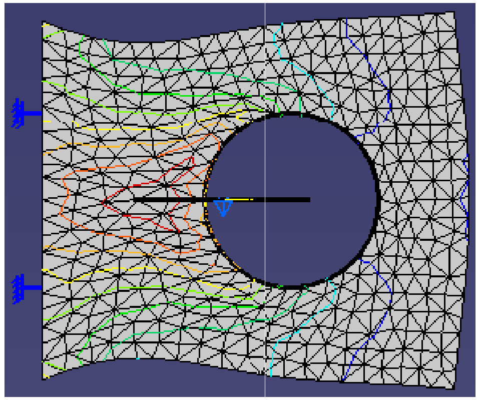
 **Rigid Virtual Parts stiffly transmit their actions : they locally stiffen the deformable body.**

 **Rigid Spring Virtual Parts stiffly transmit their actions and behave like a 6-dof spring .**

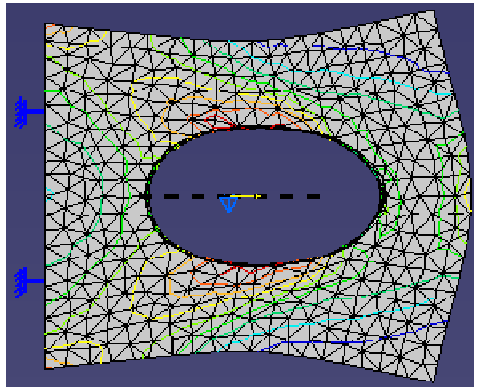
 **Smooth Spring Virtual Parts softly transmit their actions and behave like a 6-dof spring .**

Piece Virtuelle - Exemple





Rigid Virtual Part



Smooth Virtual Part

Piece Virtuelle - Liaisons & Contraintes

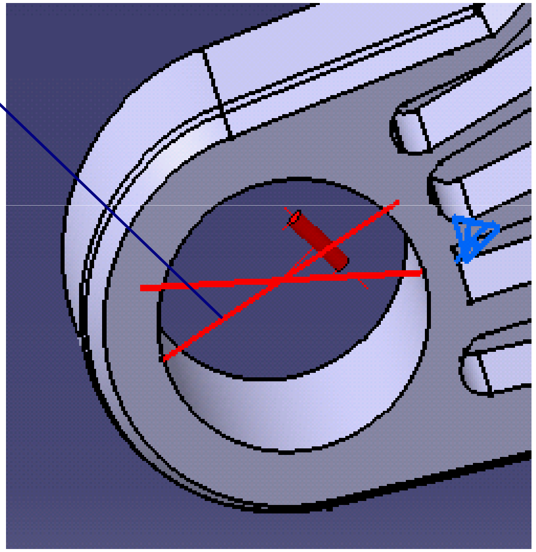
There are four types of restraints that you can apply to a virtual part's handler point :

- **Ball joints**
- **Pivots**
- **Sliders**
- **Sliding Pivots**

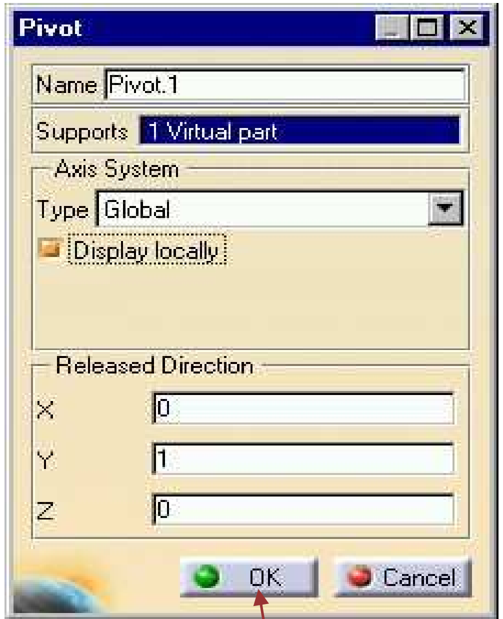
1- Click on one of the technological restraints

2- Select the Virtual Part.

The restraint will automatically be applied to its handler point

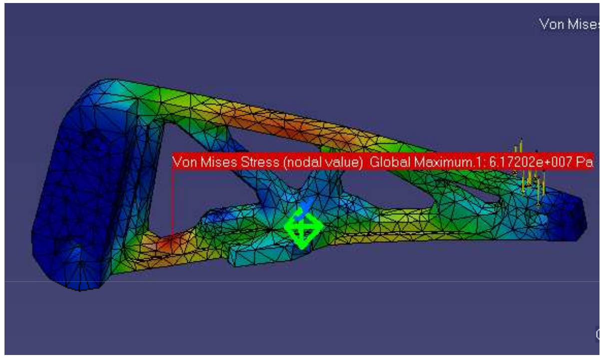


3- Define an axis (except for ball joints)



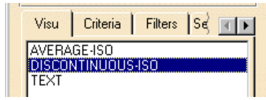
4- Click OK

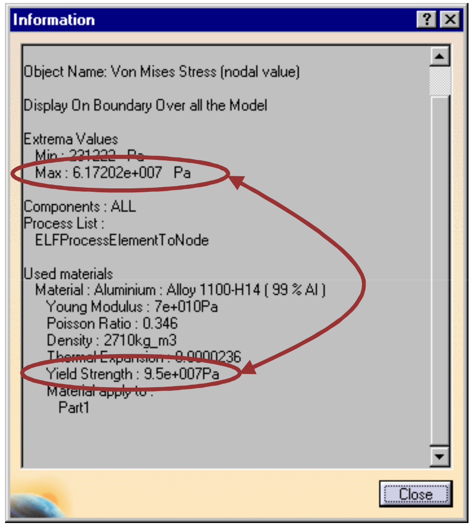
Validation des résultats

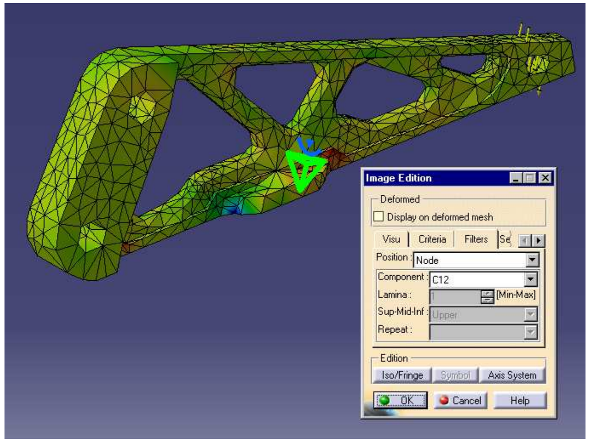


1. Von Mises result : It's the most used stress state indicator (for ductile materials, like aluminum and steel).

Here, the max stress value is 61.7 MPa which is less than the 95 Mpa tensile yield strength : the part is deformed elastically.

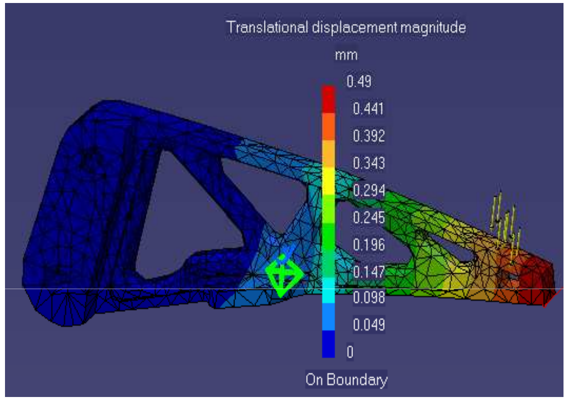


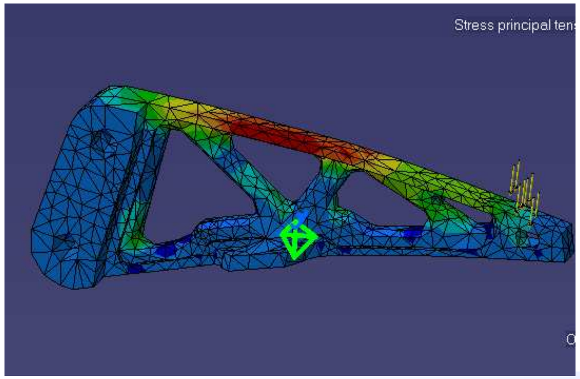




2. Full stress tensor result : We can visualize normal (C11, C22, C33) and shear stresses (C12,…), which must be less than respectively the tensile and the shear Yield Strength.

Here, shear stresses, so twisting effects, are not predominant (compare to bending effects).





3. Principal stresses result : We can see the pure tension and compression effects (by convention, C1>C2>C3).

Here, the upper face is submitted to pure tension, whereas the bottom area shows compression effects.

4. Displacements : We can check if the displacement amplitude at the front area is not too important (overcrowding problems).