GENERATIVE STRUCTURAL ANALYSIS (*GSA*)

FINITE ELEMENTS METHOD TOOLS

WHAT IS THE FINITE ELEMENT METHOD?

EXACT CLOSED-FORM SOLUTION

COMPLEX GEOMETRY → COMPLEX EQUATIONS → hard and long manual computations, but exact solutions

FEM

COMPLEX GEOMETRY → COMPLEX EQUATIONS → easy and fast numeric computations, but approximate solution

The finite element method is a numerical analysis technique for obtaining approximate solutions to a wide variety of engineering problems.

In more engineering tasks today, we find that it is necessary to obtain approximate numerical solutions to problems rather than exact closed-form solutions.

FEM PROCEDURE

1. To define stress or strain state in each point of the structure we obtain infinite number of Degree Of Freedom (DOF) – that leads to infinite number of equation.
2. FEM cuts a structure into several elements (pieces of the structure).
3. Then reconnects elements at „nodes” (nodes = pins or drops of glue that hold elements together).
4. This process results in a set of simultaneous algebraic equations (finite numbers of equations).

Advantages of FEM:

- Can readily handle very complex geometry
- Can handle a wide variety of engineering problems: Solid mechanics - Dynamics - Heat problems - Fluids - Electrostatic problems
- Can handle complex restraints - Indeterminate structures can be solved.
- Can handle complex loading: - Nodal load (point loads)- Element load (pressure, thermal, inertial forces)- Time or frequency dependent loading

FEM PROCEDURE

Typical procedure scheme for FEM:

User → ***PREPROCESS*** → *Build a FE model*

↓

Computer → ***PROCESS*** → *Solving equations, structure analysis*

↓

User → ***POSTROCESS*** → *See the result*

FEM PROCEDURE

PREPROCESS

1. ***Select analysis type:*** Structural Static Analysis, Modal Analysis, Buckling, Contact Analysis, Thermal Analysis…
2. ***Select element type:*** 2d (beam, plane), 3d (solid).
3. ***Material properties:*** Young modulus, Poisson ratio, Yield stress …
4. ***Generate mesh:*** define nodes and elements into geometry.
5. ***Boundary conditions and loads:*** apply restraints and loads.

PROCESS

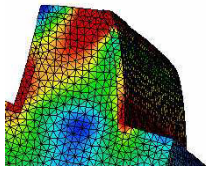
Solve the boundary problems for each elements

POSTPROCESS

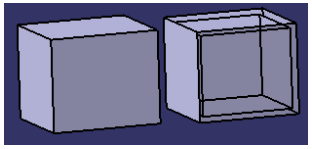
See the result: stress, strain, displacement, natural frequancy temperature

THE USER IS RESPONSIBLE FOR RESULTS

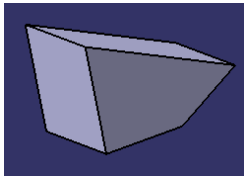
Computer can't be more intelligent than his user.



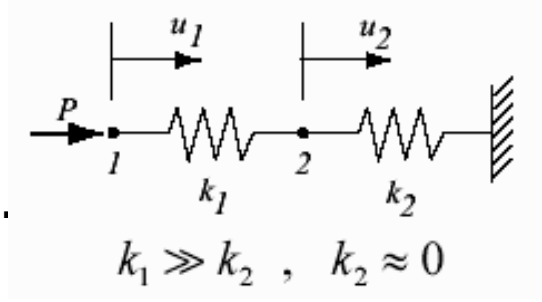
1. Colorful map of result (e.g. stress) can be produce by any software (good or bad). Results must be verify by user.



2. Elements are of the wrong type.



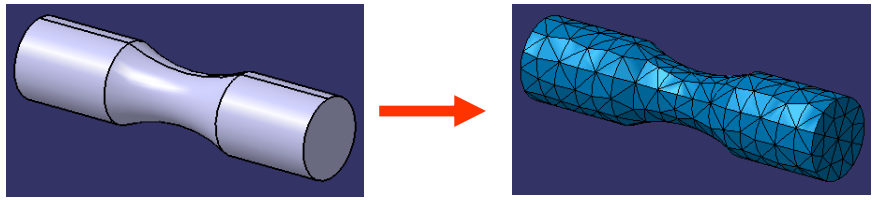
3. Elements can be distorted too much.



4. Computation errors. e.g. very large stiffness difference
5. Supports are insufficient to prevent all rigid-body motions.
6. Incompatible units. E=200 GPa Force = 100 lbs

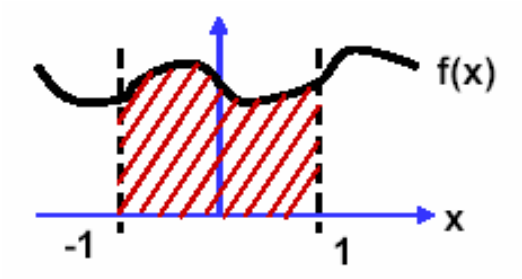
ERRORS OF THE TOOL (FEM RESPONSIBILTY)

1. Simplifying the geometry

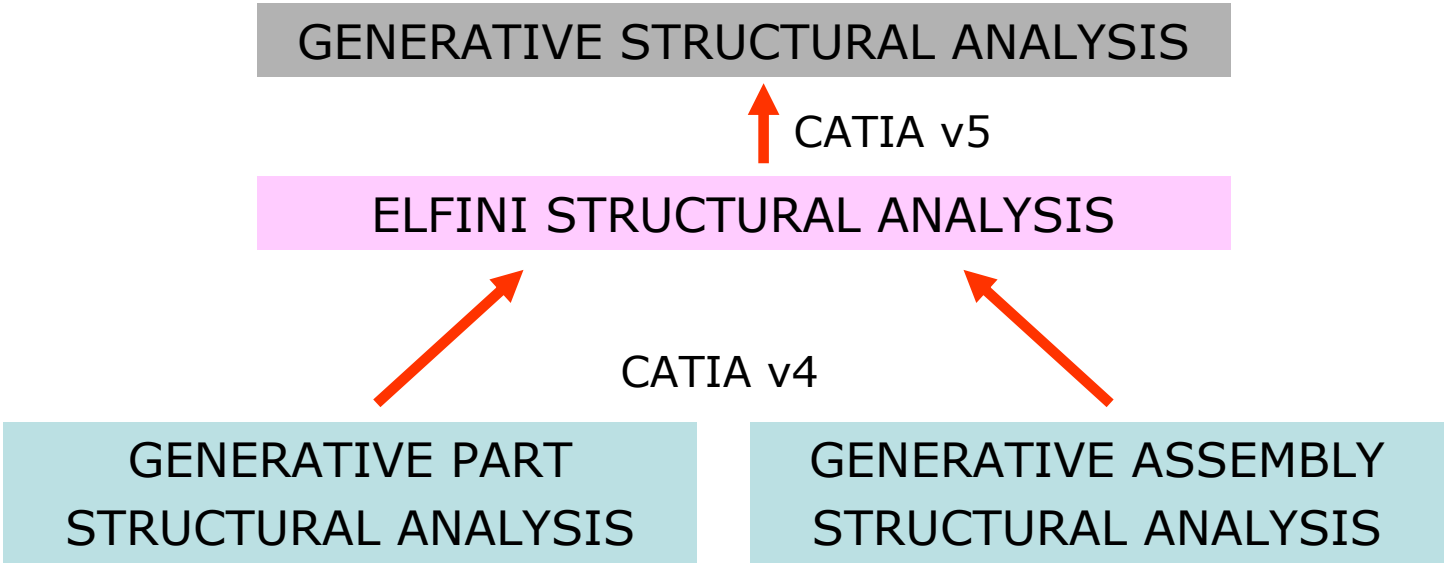


2. Field quantity is assumed to be a polynomial over an element.

3. Simply integration techniques



GENERATIVE STRUCTURAL ANALYSIS

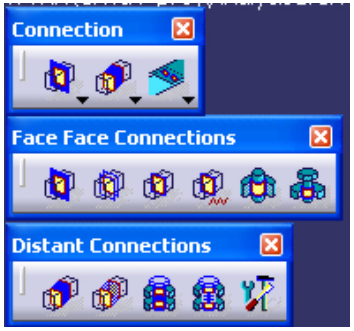


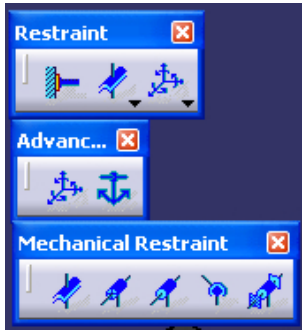
The **ELFINI Structural Analysis** product is a natural extensions of both above mentioned products, fully based on the v5 architecture. It represents the basis of all future mechanical analysis developments.

ELFINI Structural Analysis CATIA v5 products allow you to rapidly perform static mechanical analysis for 3D parts systems.

GSA – Toolbar Groups

Preprocessing

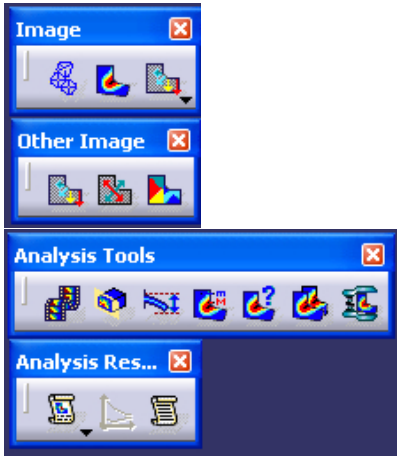




Processing

Compute option

Postprocessing



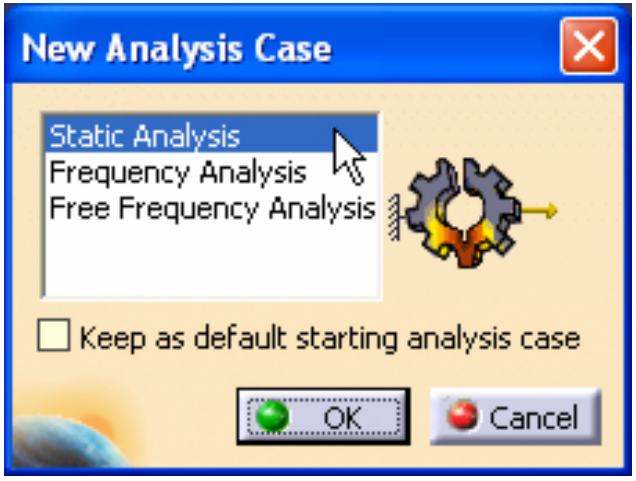
GENERATIVE STRUCTURAL ANALYSIS

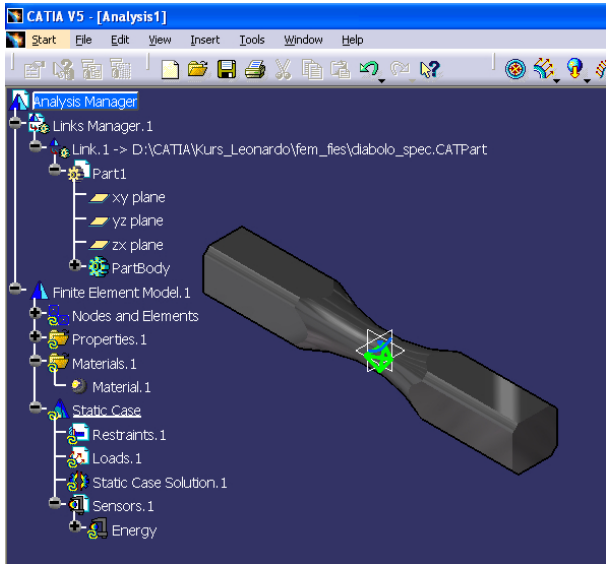
Before you start:

1. create your part for analysis in Part Design module,
2. **apply the material !!!**

Getting started:

1. Open document *diabolo_spec.CATPart.*
2. Go to *Start/Analysis&Simulation/Generative Structural Analysis* option.
3. Select *Static Analysis* in *New Analysis Case* window





GSA – Mesh generation

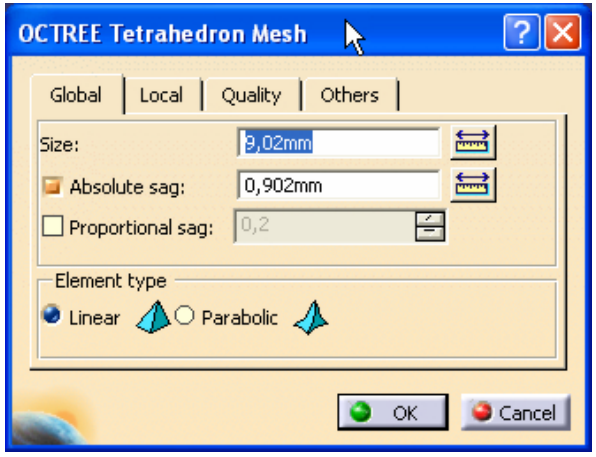
By default system was meshed the geometry. You can see it on the tree.

4. Delete this selection to define your own mesh.
5. Select *Octree Tetrahedron Mesher* icon from *Model Manager Toolbar*.

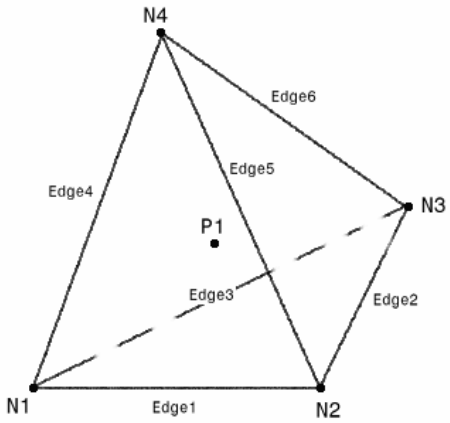
6. Then select the part on the screen the mesh will be applied.

GSA – Mesh generation

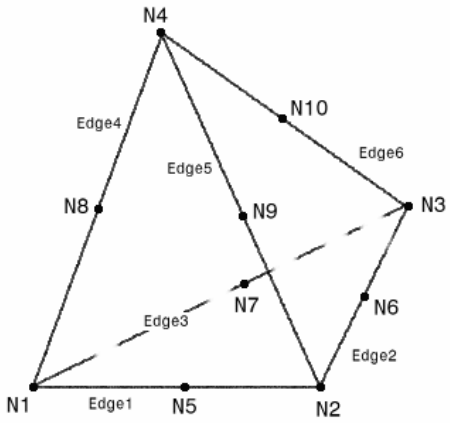
Element type:



Linear



Parabolic



Size - The mesh global size must be bigger than 0,1mm.

Linear Tetrahedron is a four-nodes isoparametric solid element. This element has only one gauss point: the gravity center (P1) of the tetrahedron. There are only three translations per node. Type of behavior – elastic.

Parabolic Tetrahedron is a ten-nodes isoparametric solid element. This element has four gauss point (0,138 ; 0,138 ; 0,138), P2 (0,138 ; 0,138 ; 0,585), P3 (0,138 ; 0,585 ; 0,138), P4 (0,585 ; 0,138 ; 0,138).

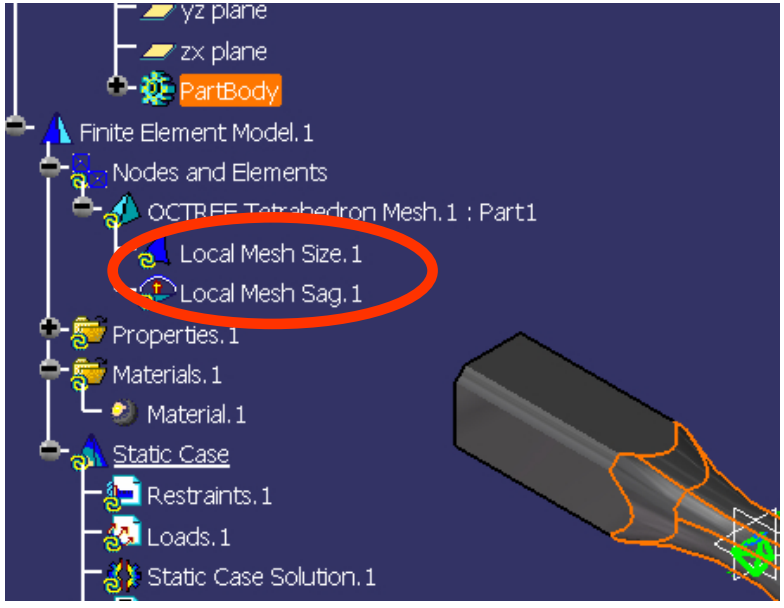
There are only three translations per node. Type of behavior – elastic.

GSA – Mesh generation

Absolute sag – is a minimum distance between nodes and boundary of a part. That leads to deformation of the mesh. Sometimes it is necessary to make mesh size smaller.

The user can change those parameters locally.

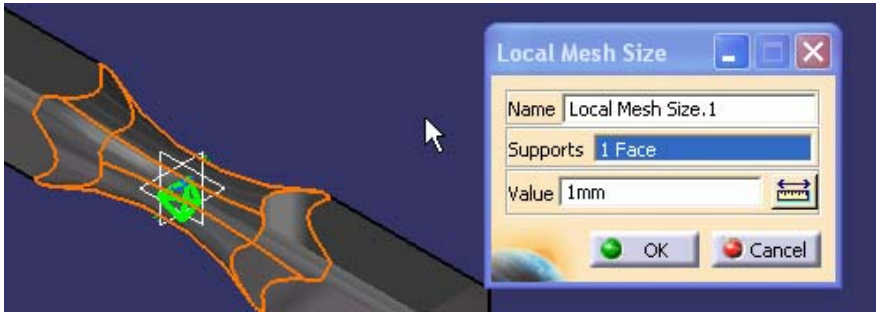
7. Select *Local tab* in window,



choose *Local size* on the list and press *Add* button …

GSA – Mesh generation

… then select *Support* you want to change the parameter. Set the value equal 1mm and confirm.



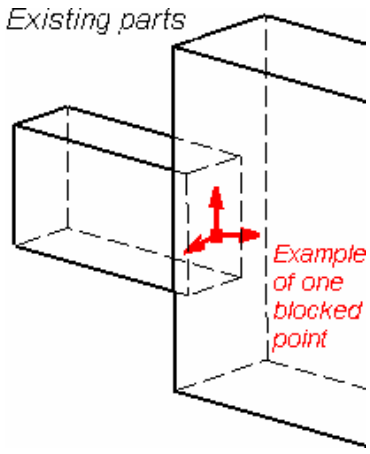
8. Use the same procedure to change *Local Sag* parameter and set its value to 0,3mm.

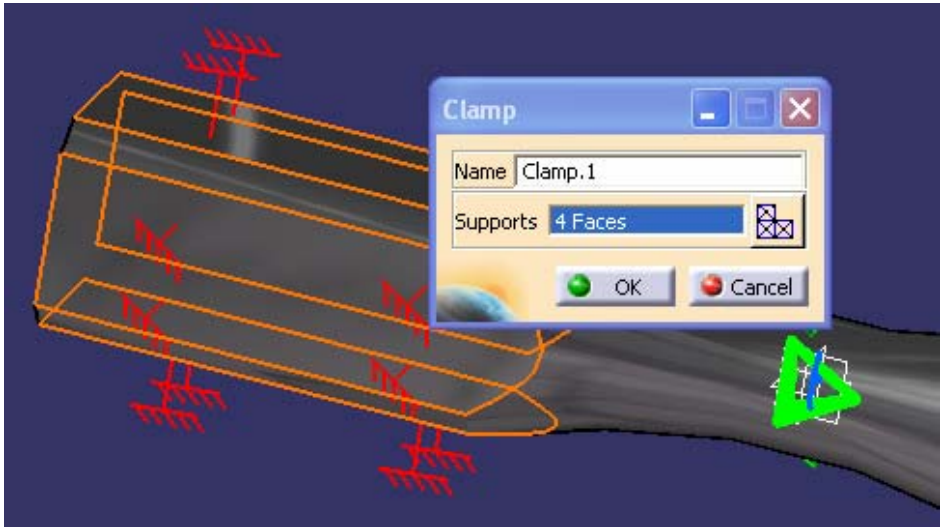
GSA – Clamp Restraint

Clamps are restraints applied to surface or curve geometries, for which all points are to be blocked in the subsequent analysis.

Select the geometry support (a surface, an edge or a virtual part). Any selectable geometry is highlighted when you pass the cursor over it.

You can select several supports in sequence, to apply the Clamp to all supports simultaneously.
Symbols representing a fixed translation in all directions of the selected geometry are visualized.





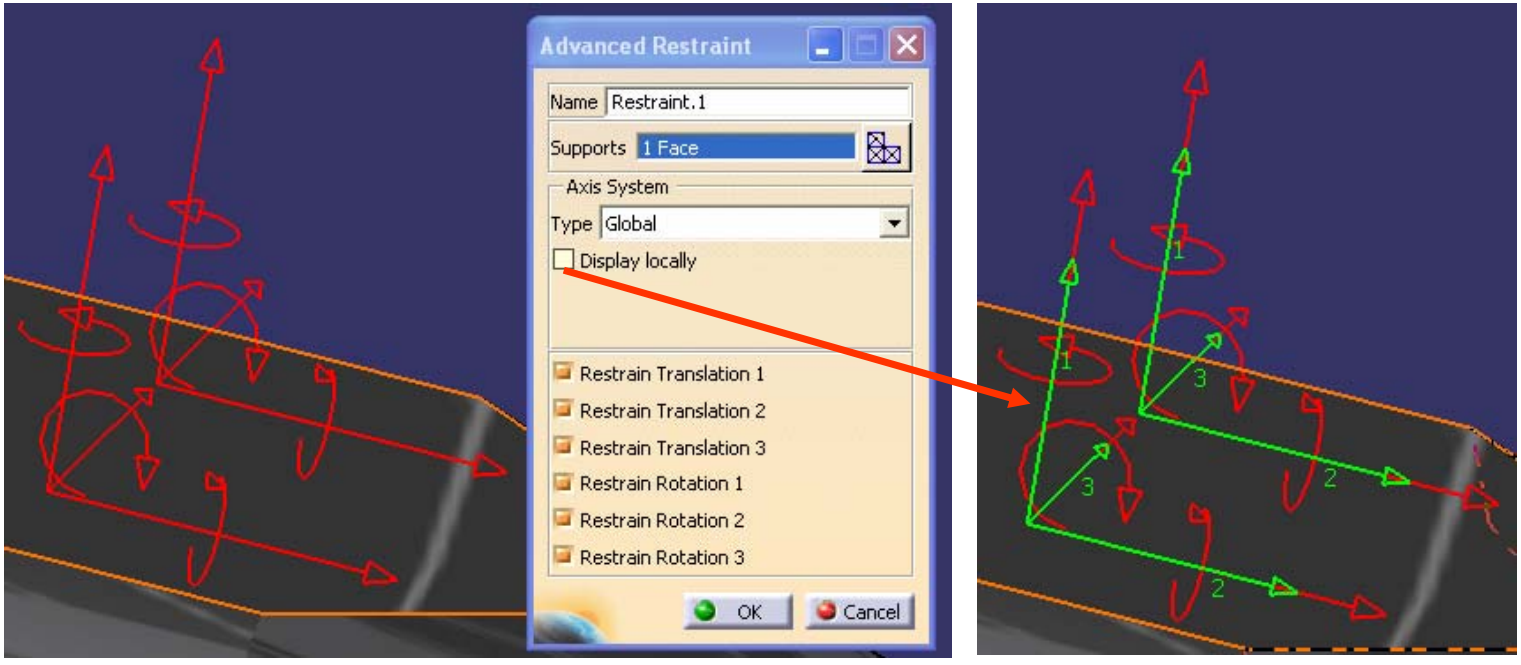
→ means that there is no translation degree of freedom left in that direction.

GSA – Advanced Restraint

Advanced Restraints are generic restraints allowing you to **fix any combination of available nodal degrees of freedom** on arbitrary geometries.

Select *Display locally* to show local axis.

You can select more surfaces to fix during one restraint operation.

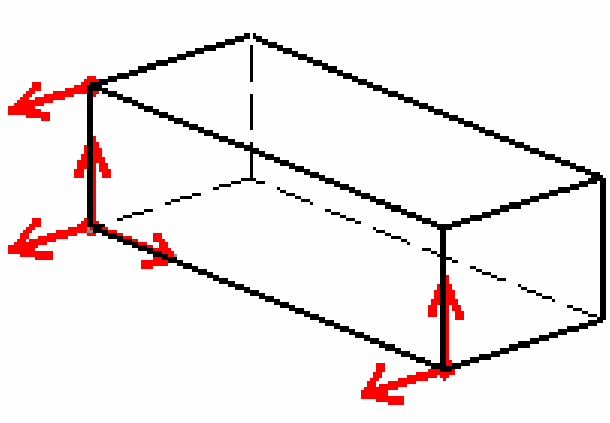


GSA – Iso-static Restraint

Iso-static Restraints are statically definite restraints allowing you to simply support a body.

The program automatically chooses three points and restrains some of their degrees of freedom according to the 3-2-1 rule. The resulting boundary condition prevents the body from rigid-body translations and rotations, without over-constraining it.

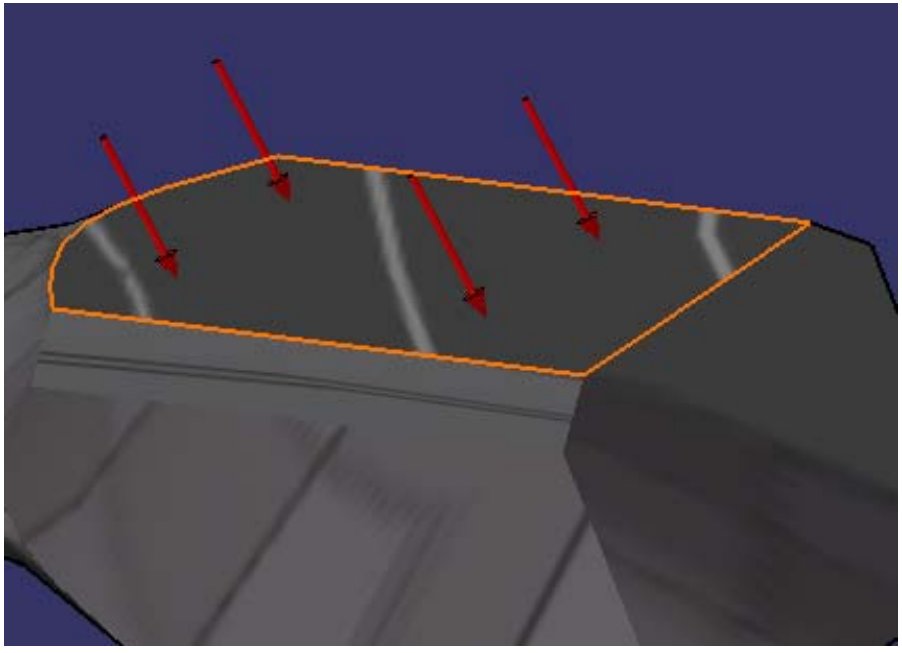
Iso-static restraint is represented as anchor icon and it is connect to whole part.

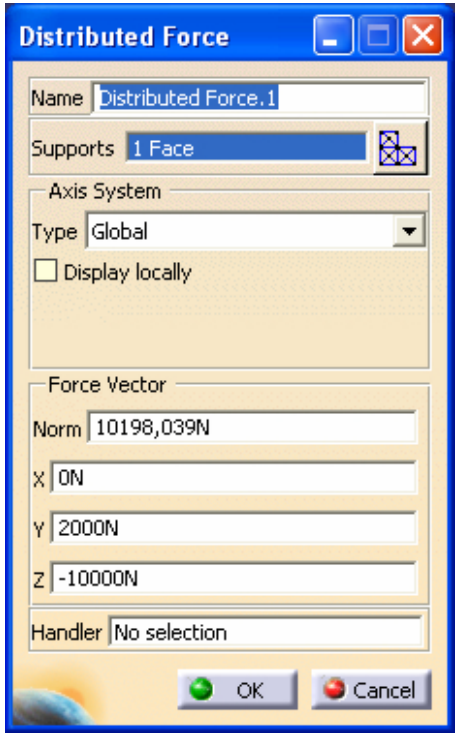


GSA – Distributed Force Load

Distributed Forces are force systems **statically equivalent to a given pure force resultant at a given point**, distributed on a virtual part or on a geometric selection.

The user specifies three components for the direction of the resultant force, along with a magnitude information. Upon modification of any of these four values, the resultant force vector components and magnitude are updated based on the last data entry.

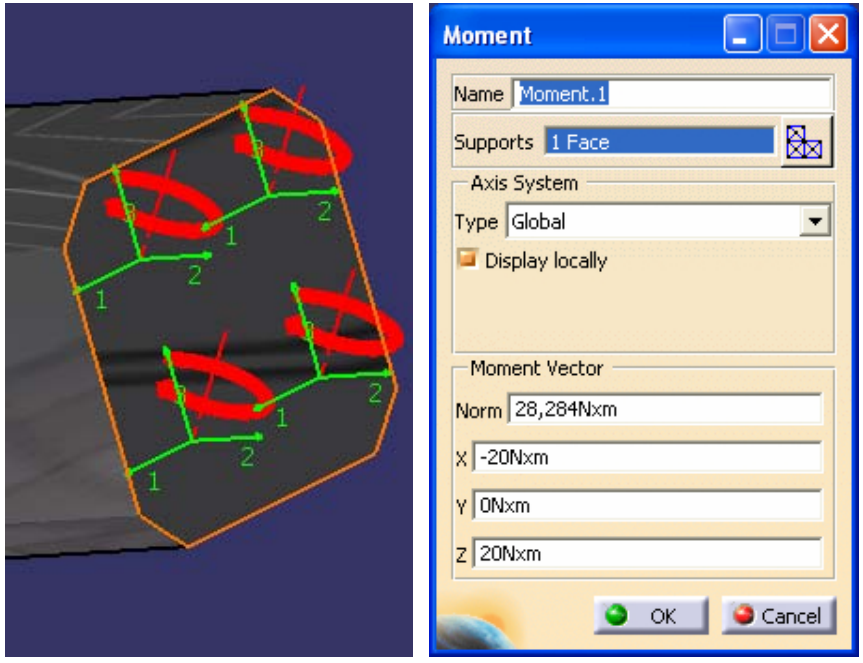




GSA – Moment Load

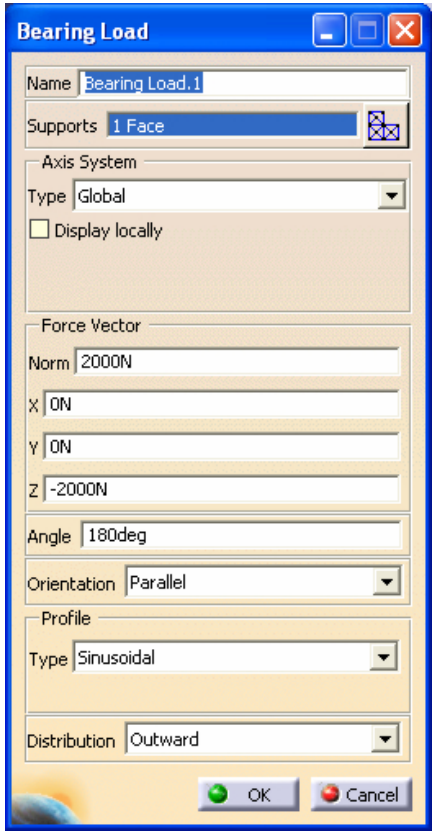
Moments are force systems statically equivalent to a given pure couple *(*single moment resultant*),* distributed on a virtual part or on a geometric selection.

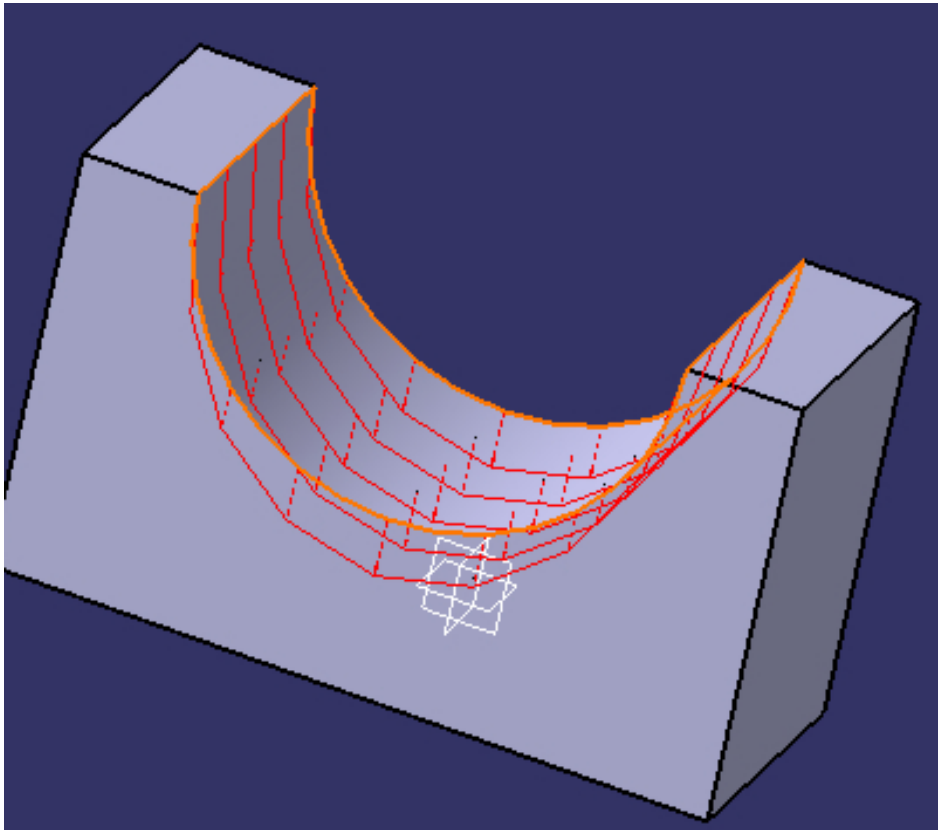
The user specifies three components for the direction of the resultant moment, along with a magnitude information. Upon modification of any of these four values, the resultant moment vector components and magnitude are updated based on the last data entry.



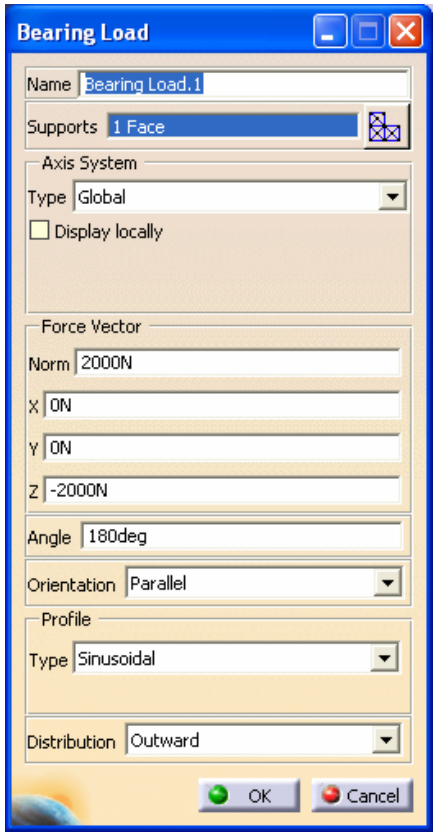
GSA – Bearing Load

Bearing Loads are simulated contact loads applied to cylindrical parts. The user selects a cylindrical boundary of the part. Any type of revolution surface can be selected. In the Bearing Load definition panel, you have to specify the resulting contact force (direction and norm).





GSA – Bearing Load

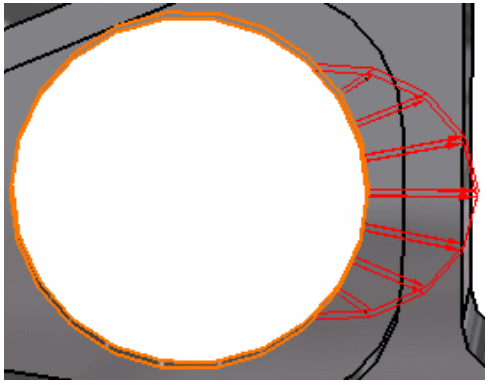


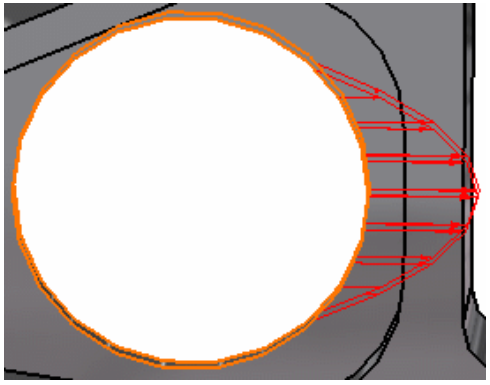
Angle: corresponds to the angle over which the forces can be distributed. When entering an angle value, a highly precise preview automatically appears on the model.

Orientation: provides you with two ways for distributing forces:

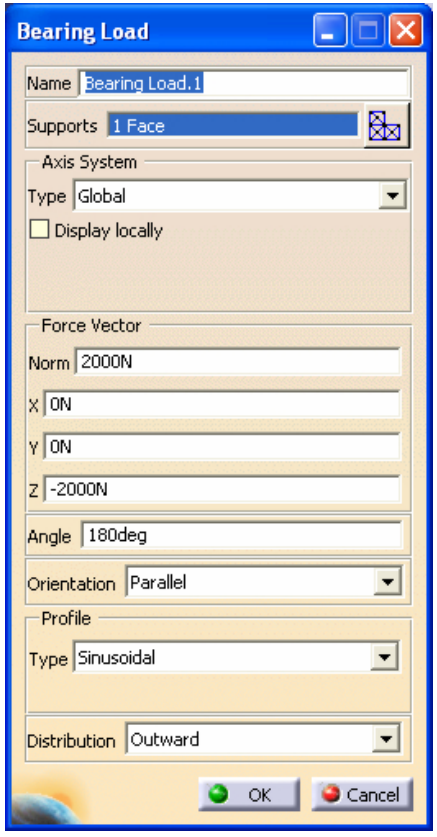
<u>Radial</u>: all the force vectors at the mesh nodes are normal to the surface in all points. This is generally used for force contact.

<u>Parallel</u>: all the force vectors at the mesh nodes are parallel to the resulting force vectors. This can useful in the case of specific loads.





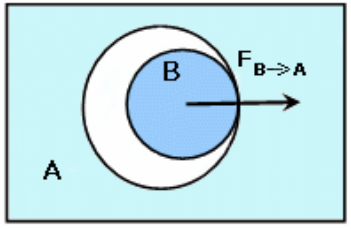
GSA – Bearing Load

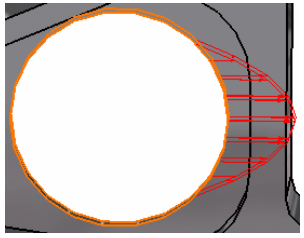


Profile: can be Sinusoidal, Parabolic or Law type, defining how you will vary the Force intensity according to the angle: Sinusoidal, Parabolic or Law.

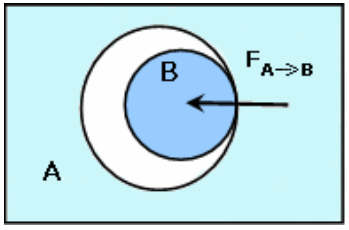
Distribution: lets you specify the force distribution

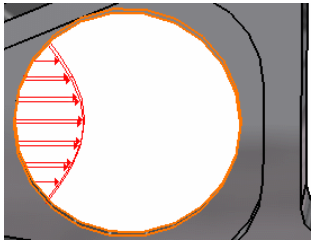
Outward: B pushes on A





Inward: A pushes on B





GSA – Line Force Density

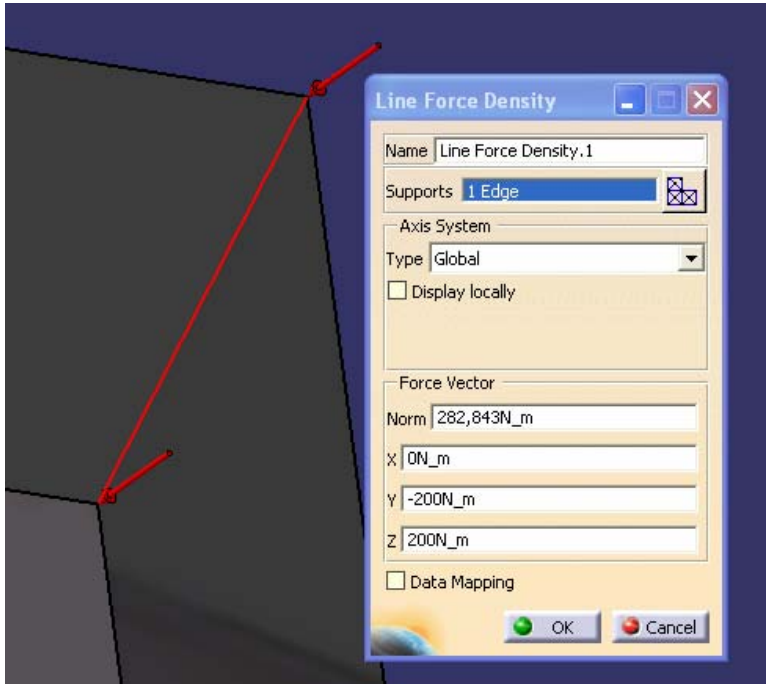
Line Force Densities are intensive loads representing line traction fields of uniform magnitude applied to curve geometries.

The user specifies three components for the direction of the field, along with a magnitude information. Upon modification of any of these four values, the line traction vector components and magnitude are updated based on the last data entry.

Units are line traction units (typically N/m in SI).

Line Force Density can be applied to the edges.

If you select other surfaces, you can create as many Line Force Density loads as desired with the same dialog box.

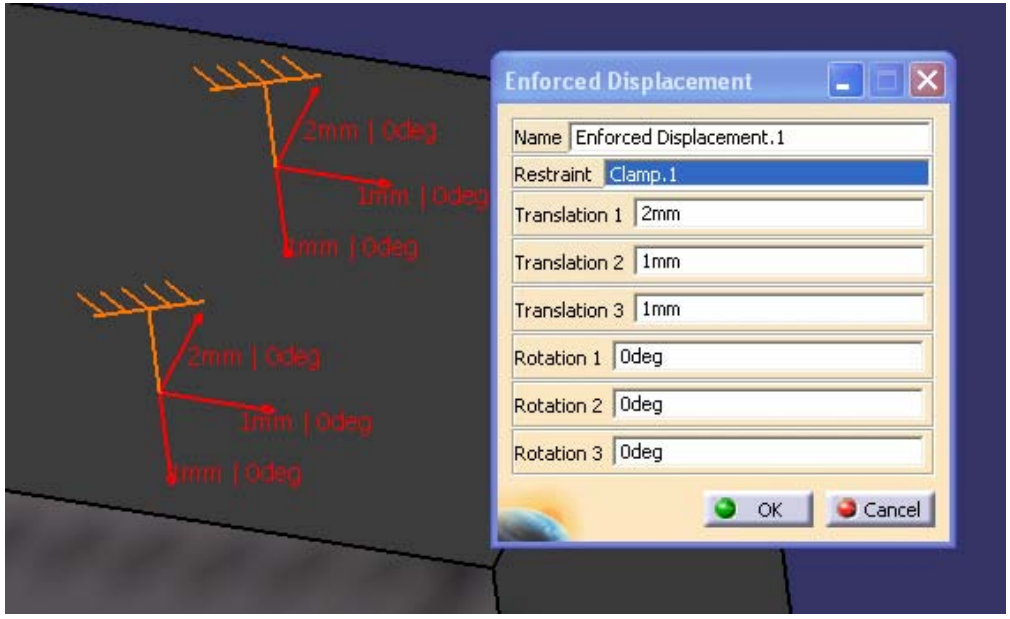


GSA – Enforced Displacement

Enforced Displacements are loads applied to support geometries, resulting for the subsequent analysis in assigning non-zero values to displacements **in previously restrained directions**.

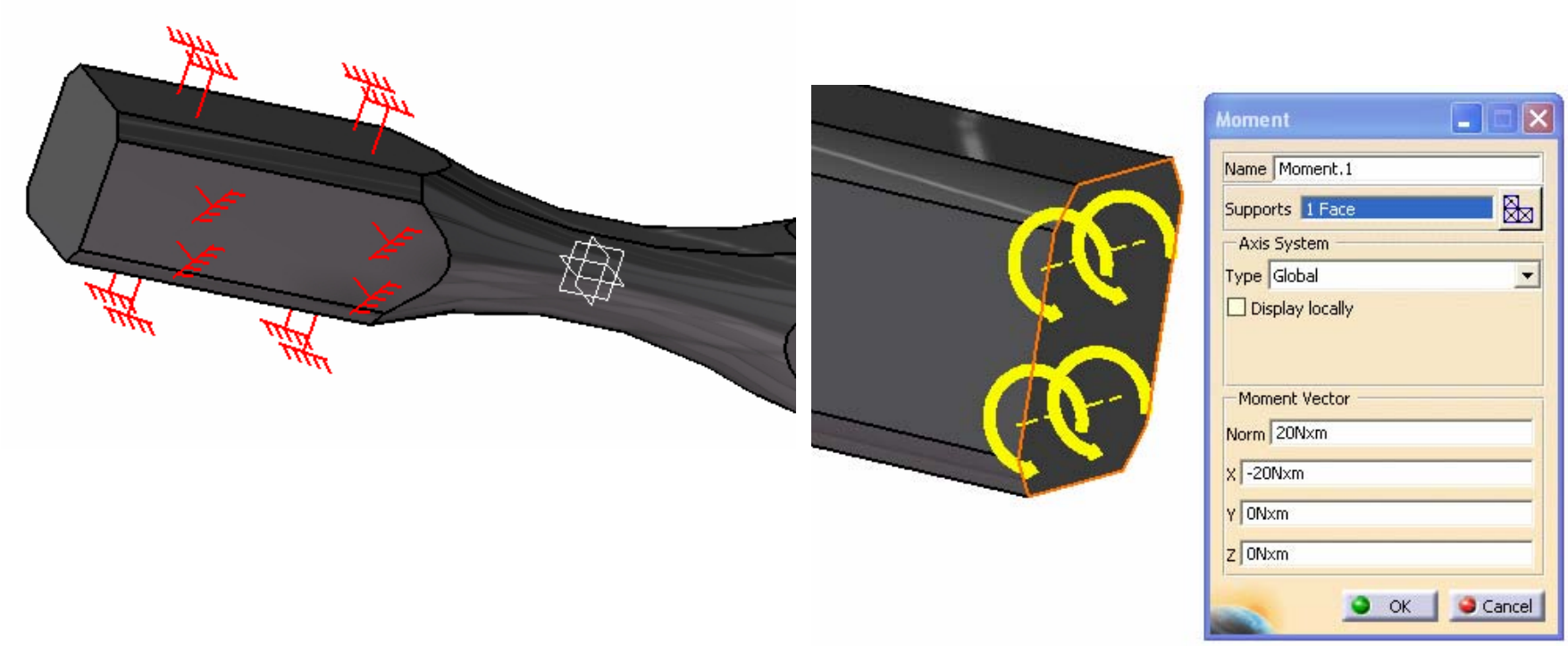
An Enforced Displacement object is by definition associated to a Restraint object.

Make sure you entered non-zero values only for those degrees of freedom which have been fixed by the associated Restraint object. Non-zero values for any other degree of freedom will be ignored by the program.



GSA – Back to the example…

9. Create *Clamp* restraint on four surfaces for one of selected side of the specimen.



10. Apply *Moment Load* to the surface selected o the picture.

Set X-component equal -20Nxm.

GSA – Compute

11. System is ready for computation. Select *Compute* option.

The Compute dialog box appears.

The list allows you to choose between several options for the set of objects to update.

All: all objects defined in the analysis features tree will be computed.

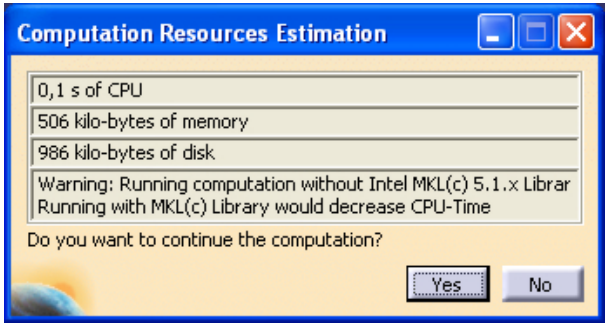
Mesh Only: only the mesh will be computed.

Analysis Case Solution Selection: only a selection of user-specified Analysis Case Solutions will be computed (if specified previously).

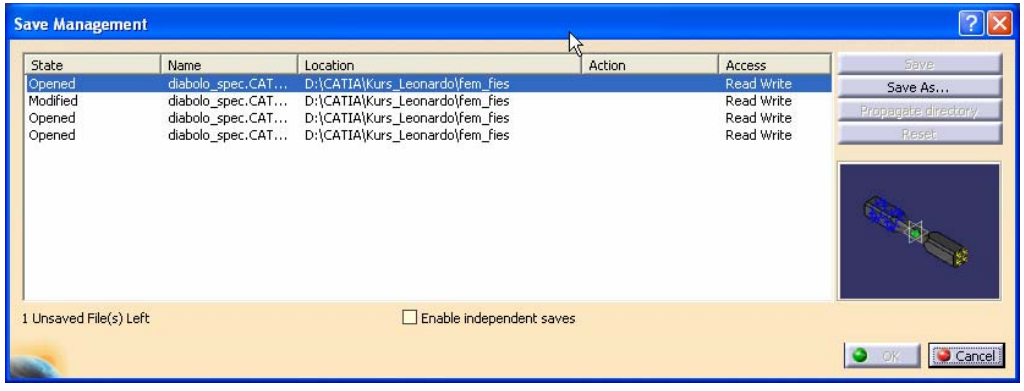
Selection by Restraint: only the selected characteristics will be computed (Properties, Loads, Masses).

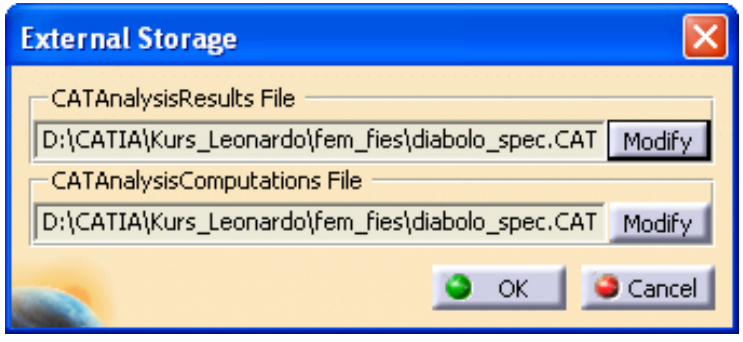
GSA – Compute

System generates an information about calculations:



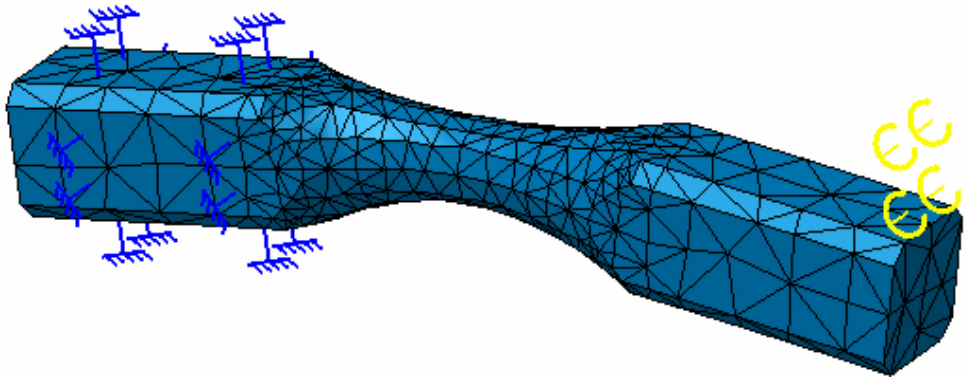
12. Now you can run computations. It can take some time, depending on the number of nodes → mesh size. The results are automaticaly save on disk. You have to use *Save Management* option to select the user path for files save. You can use *External Storage* option.



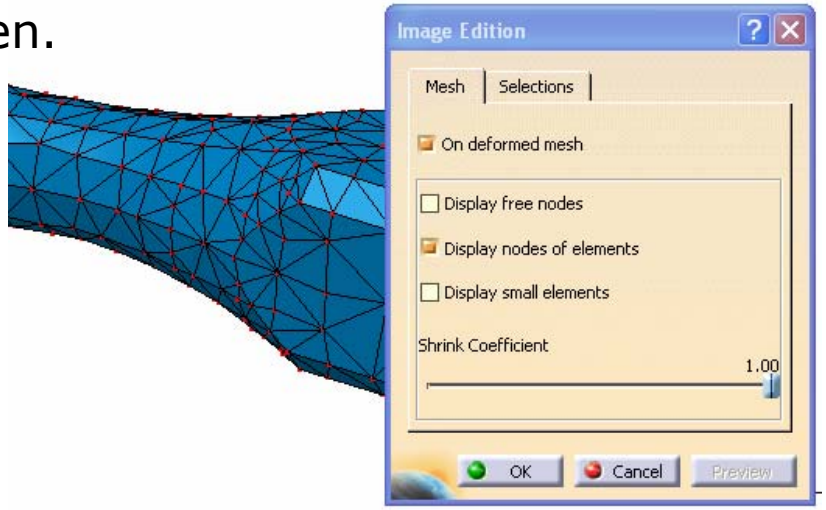


GSA – Results: Displacement

13. Click the *Deformation* icon from *Image Toolbar*. You will see the deformation of the part. The denser mesh is visible in the middle part of the body.

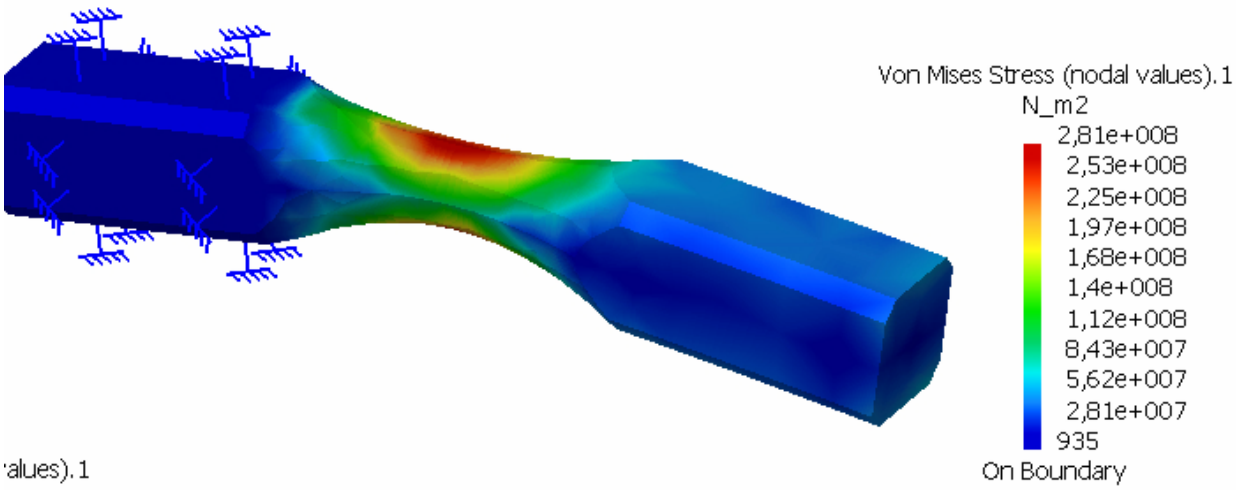


Double-click on themesh on the screen. *Image Edition* window appears. You can select additional information to see (e.g. nodes). You can specify for which part of the element those information have to b visible (*Selection Tab*).

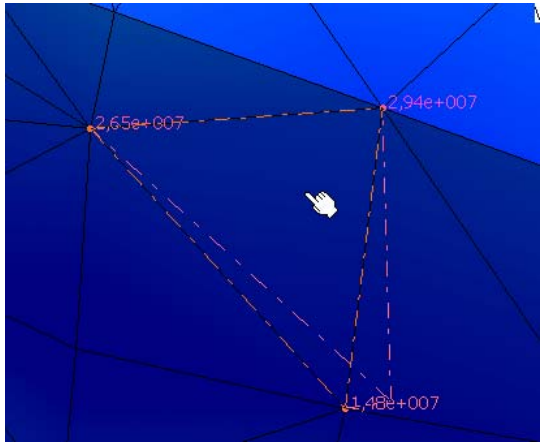


GSA – Results: Von Mises Stress

14. Click the *Von Mises Stress* icon from *Image Toolbar*. You will see the stress map of the part.

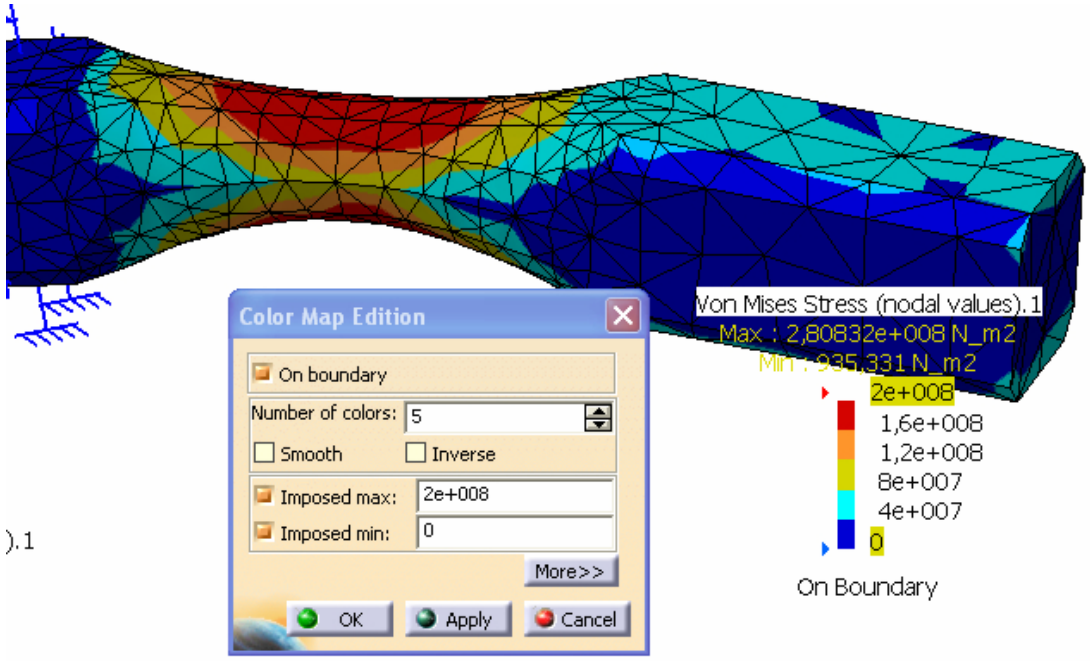


Select specyfied element to see exact result for their nodes.



GSA – Results: Von Mises Stress

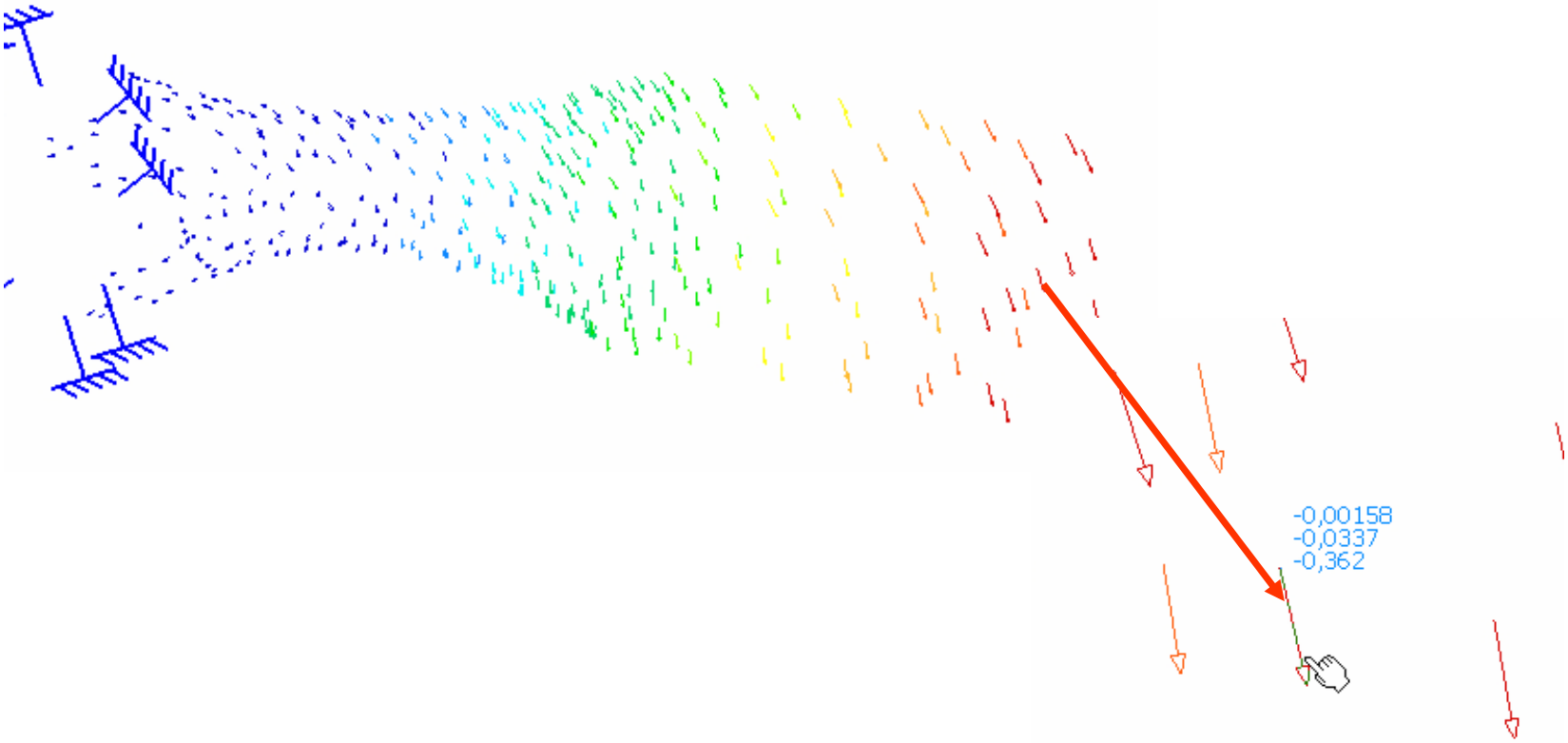
15. Double-click on Color Legend to open *Color Map Edition*.



You can change the number of displayed colors, the edges of the color boundaries can be smoothed or not. You can also set the range of Color Map by using *Imposed max* and *Imposed min* option.

GSA – Results: Displacement

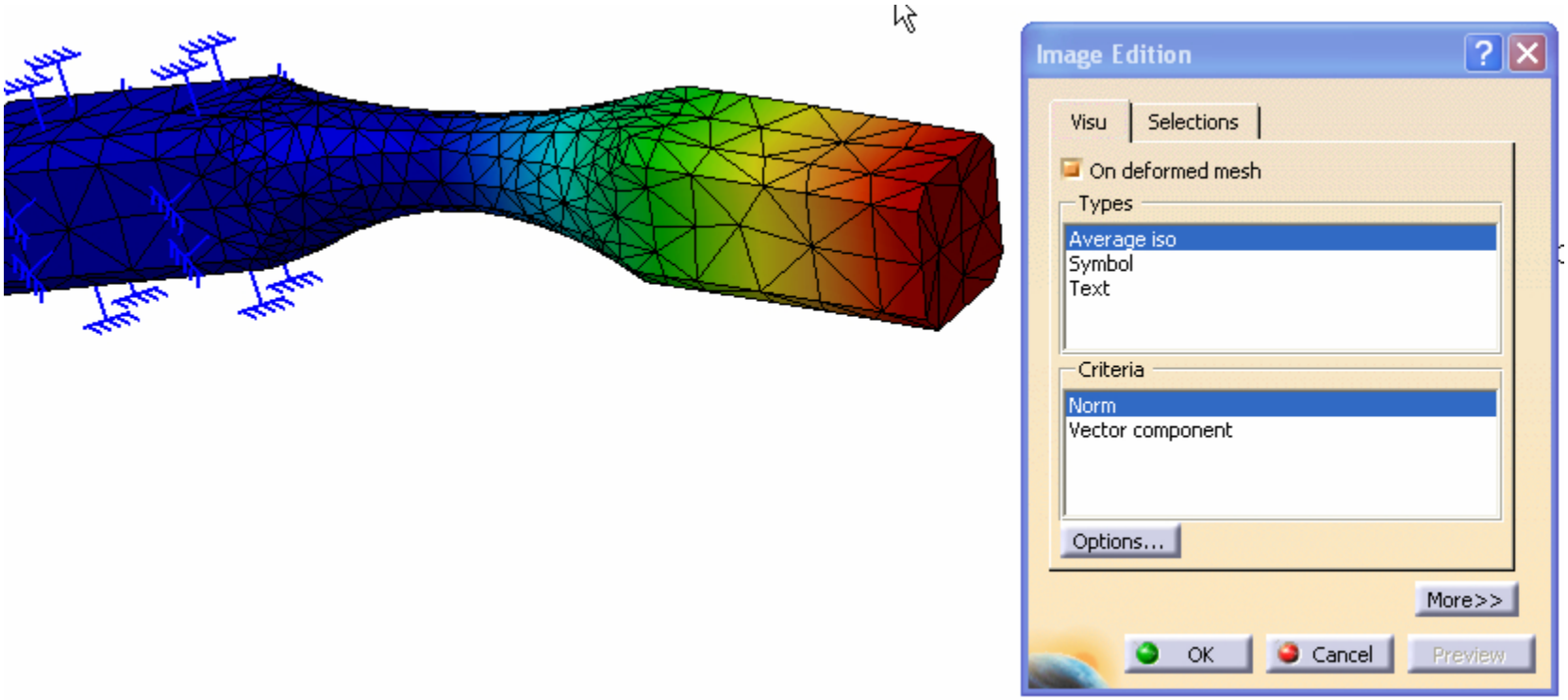
16. Click on *Displacement* icon.



Displacement are displayed as vectors. Select one vector to see exact displacement components for node.

GSA – Results: Displacement

17. Double-click on vector to activate *Image Edition* window. You can change the display method by using *Visu Tab* and *Average iso* option to see average values on nodes.

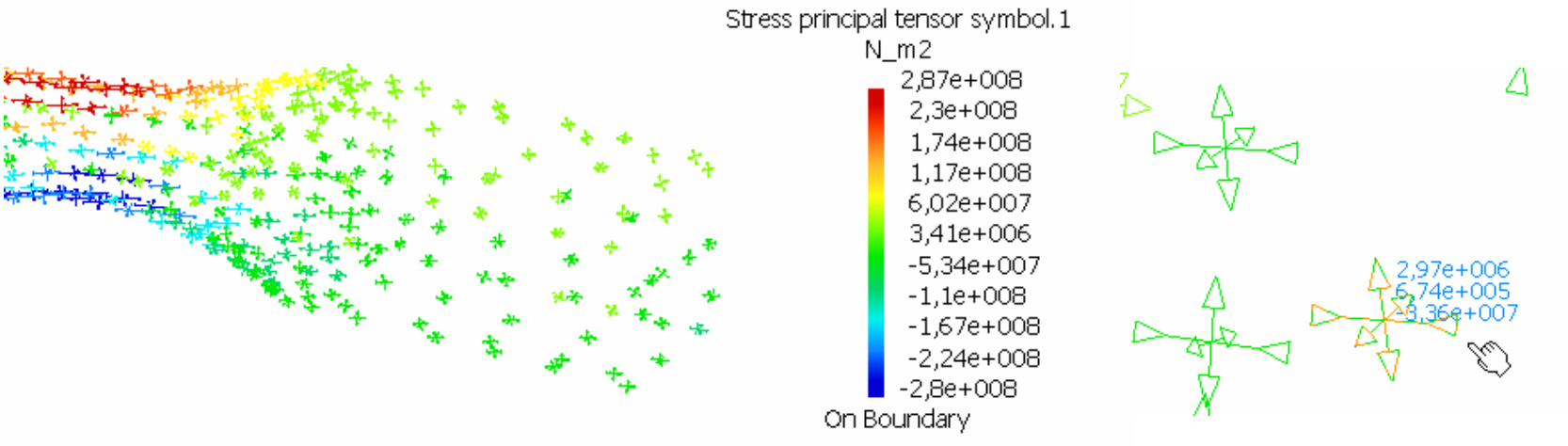


GSA – Results: Principal Stress

18. Click on *Principal Stress* icon.

Principal Stresses are displayed as complex vectors. Select one group to see exact values of all components for node.

It is possible to recognize tension or compression.

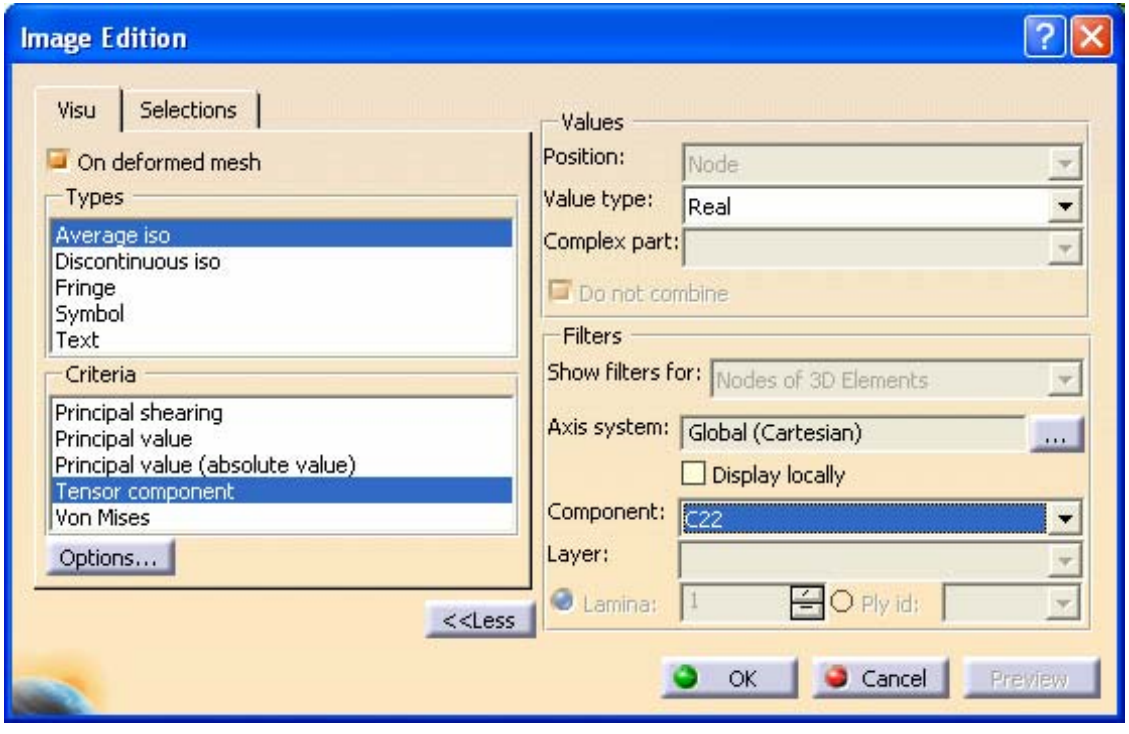


GSA – Results: Principal Stress

19. Double-click on vector to activate *Image Edition* window. You can change the display method by using *Visu Tab* and *Average iso* option to see average values on nodes.

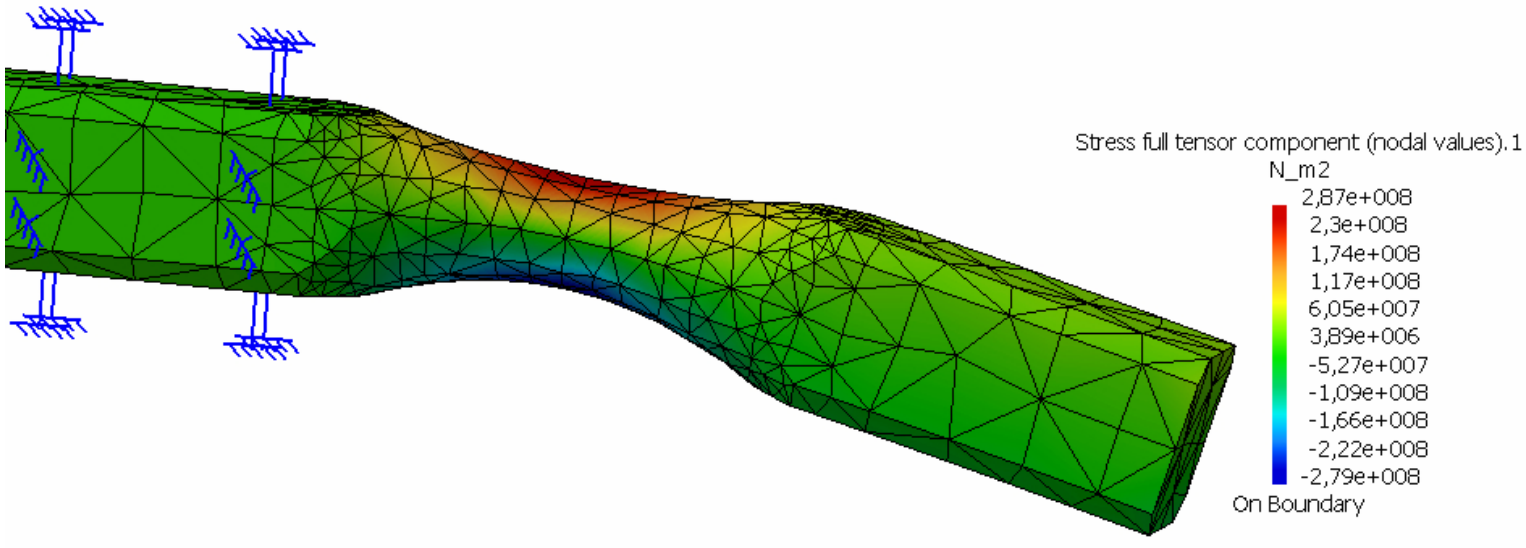
It is possible to acces selected component distribution in whole part.

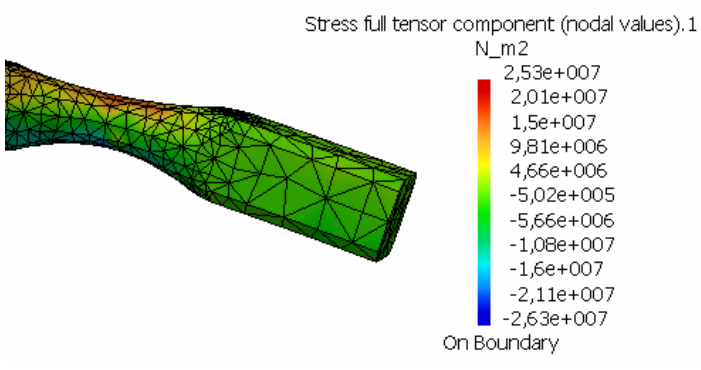
Press *More>>* button and select *Types=Average iso*, *Criteria=Tensor Component* and *Component=C22* to see stresses component parallel to specimen axis.

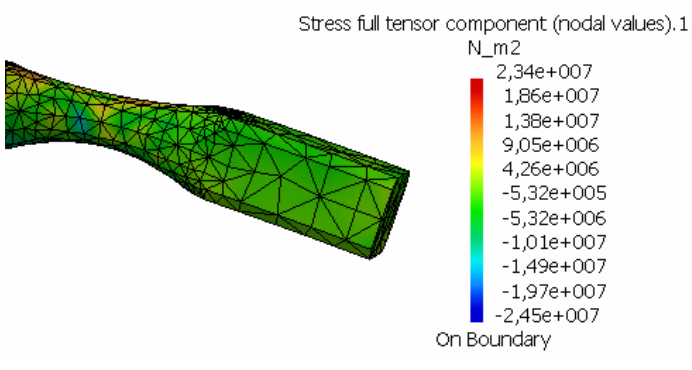


GSA – Results: Principal Stress

The bending stress component. The other components equals 10% less than bending stresses.





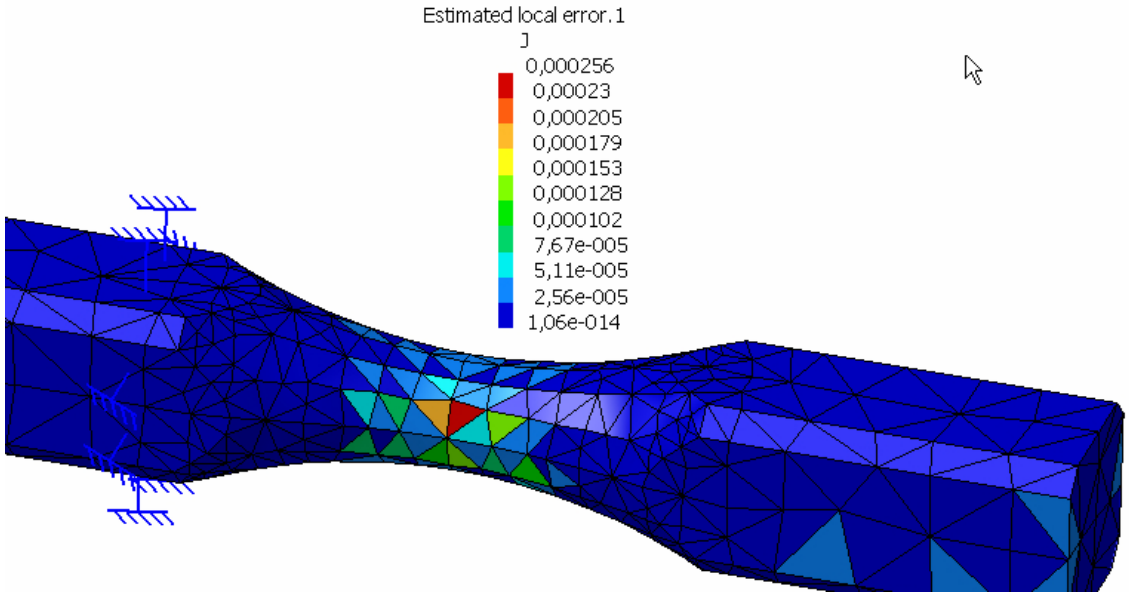


GSA – Results: Precision

20. Select Precision icon from *Other Image Toolbar*. This option allo you to see estimated local errors result. You can recognize the area with the highest value of the calculation error.

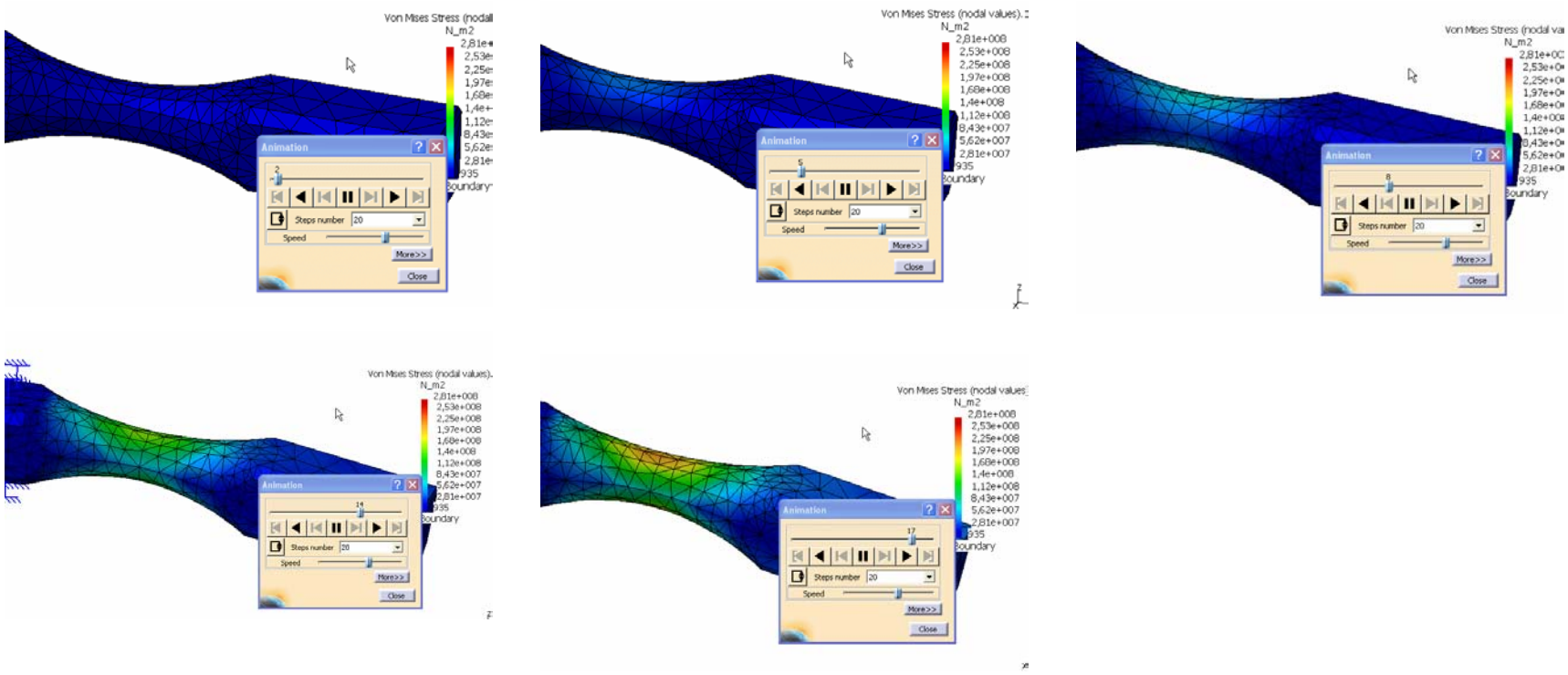
If the error is relatively large in a particular region of interest, the computation results in that region may not be reliable. A new computation can be performed to obtain better precision.

To obtain a refined mesh in a region of interest, use smaller Local Size and Sag values in the mesh definition step.



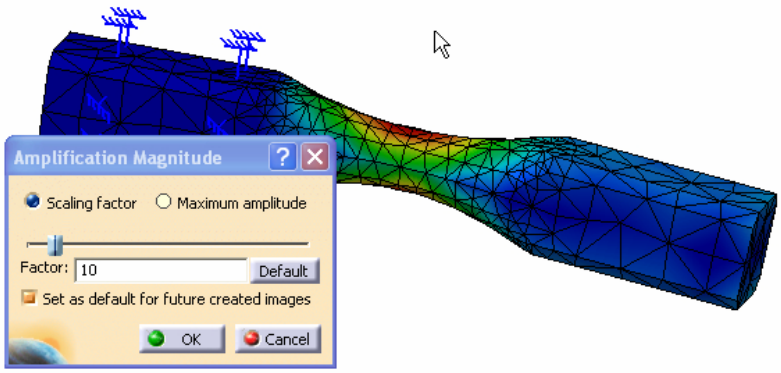
GSA – Aalysis Tools: Animation

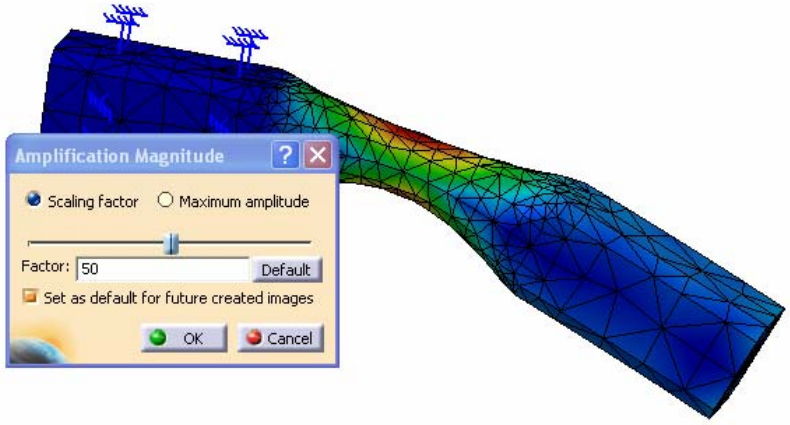
Image *Animation* is a continuous display of a sequence of frames obtained from a given image. Each frame represents the result displayed with a different amplitude. The frames follow each other rapidly giving the feeling of motion.



GSA – Aalysis Tools: Amplification Magnitude

Amplification Magnitude consists in scaling the maximum displacement amplitude for visualizing a deformed image.





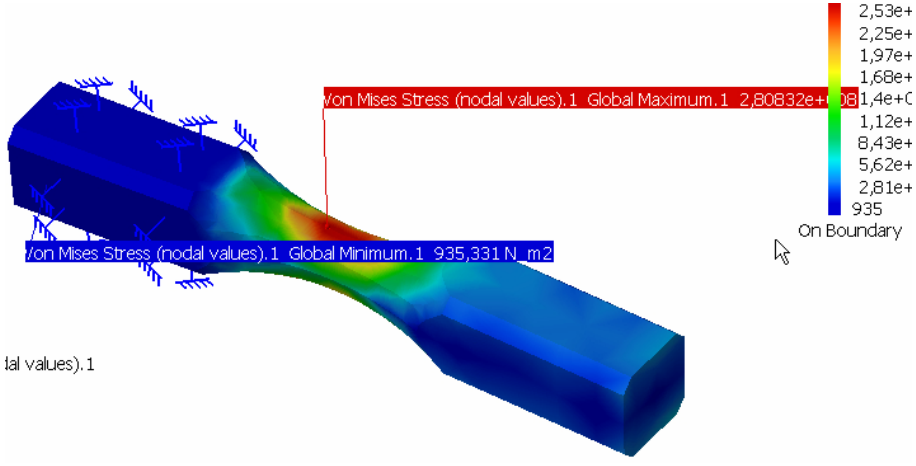
GSA – Aalysis Tools: Image Extrema

Extrema Creation consists in localizing points where a results field is maximum or minimum. You can ask the program to detect either one or both global extrema and an arbitrary number of local extrema for your field.

You can ask the program to detect given numbers of global (on the whole part) and/or local (relatively to neighbor mesh elements) extrema at most, by setting the Global and Local switches.

Global means that the system will detect all the entities which have a value equal to the Minimum or Maximum value.

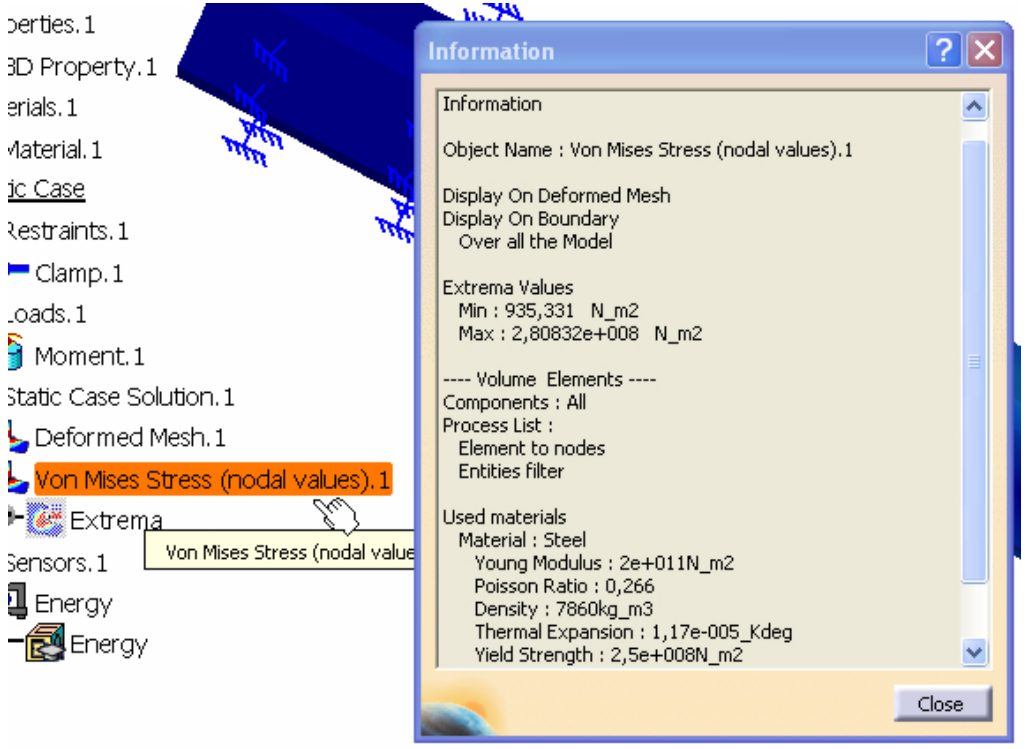
Local means that the system will search all the entities which are related to the Minimum or Maximum value compared to the two-leveled neighboring entities.

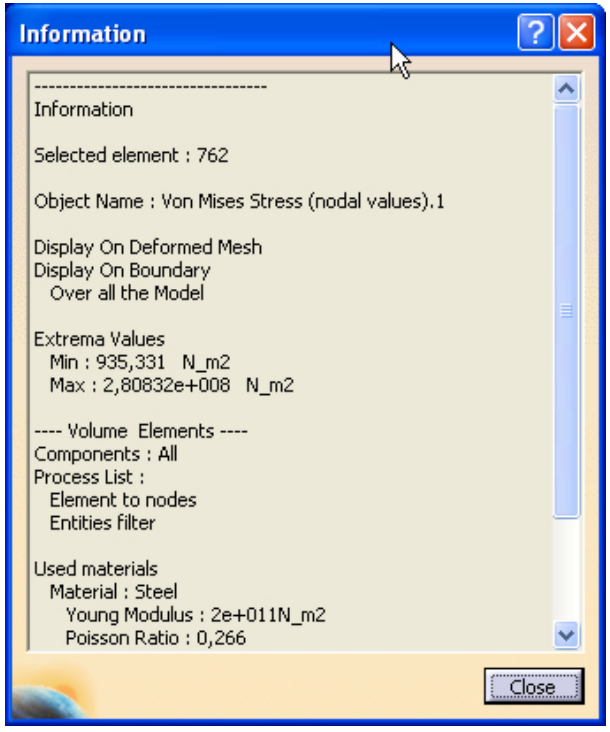


GSA – Aalysis Tools: Information

Information option allow to obtain information about result case (e.g. Von Mises Stress image). To choose element the user can use tree.

To display information about selected element of the mesh simply point that element on the screen.





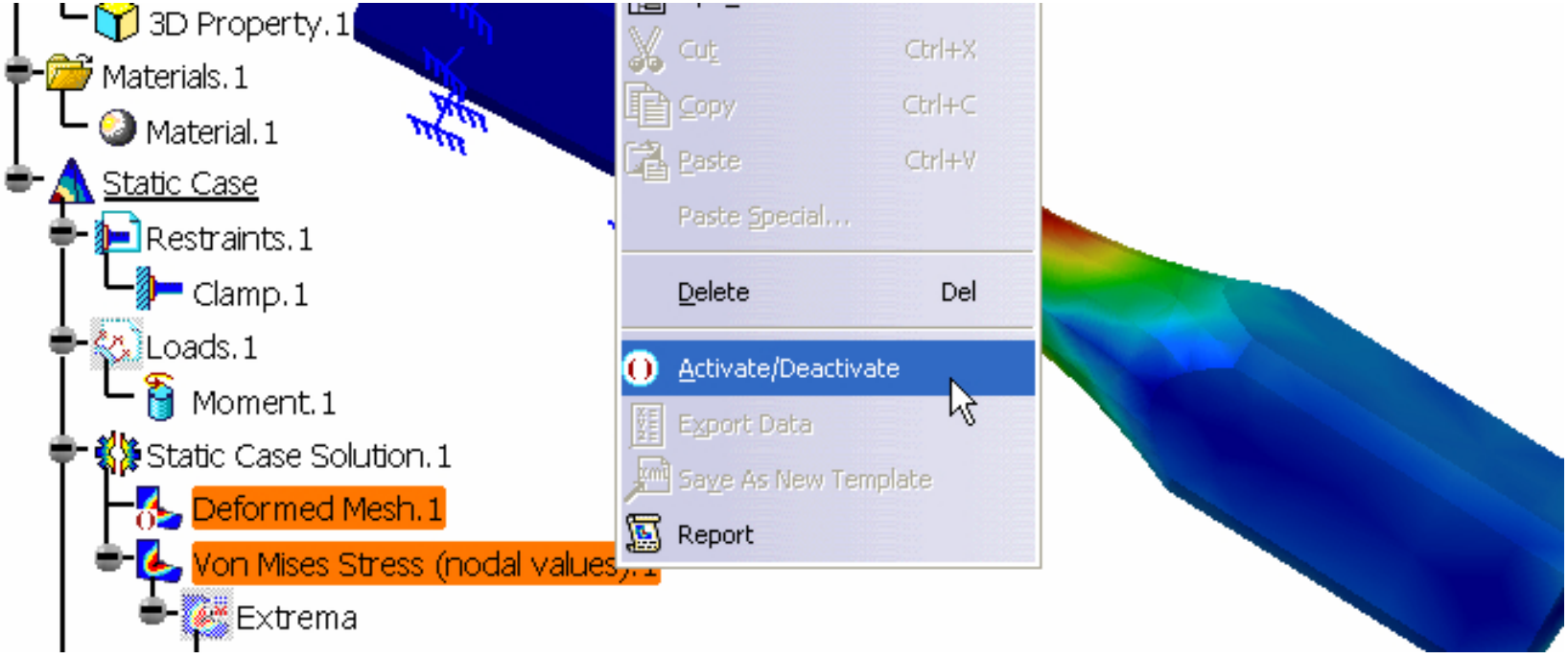
GSA – Aalysis Tools: Images Layout

Generated images corresponding to analysis results are superimposed into one image that cannot be properly visualized. You can tile these superimposed images into as many layout images on the 3D view.

To separate images you have to deactivate selected images.

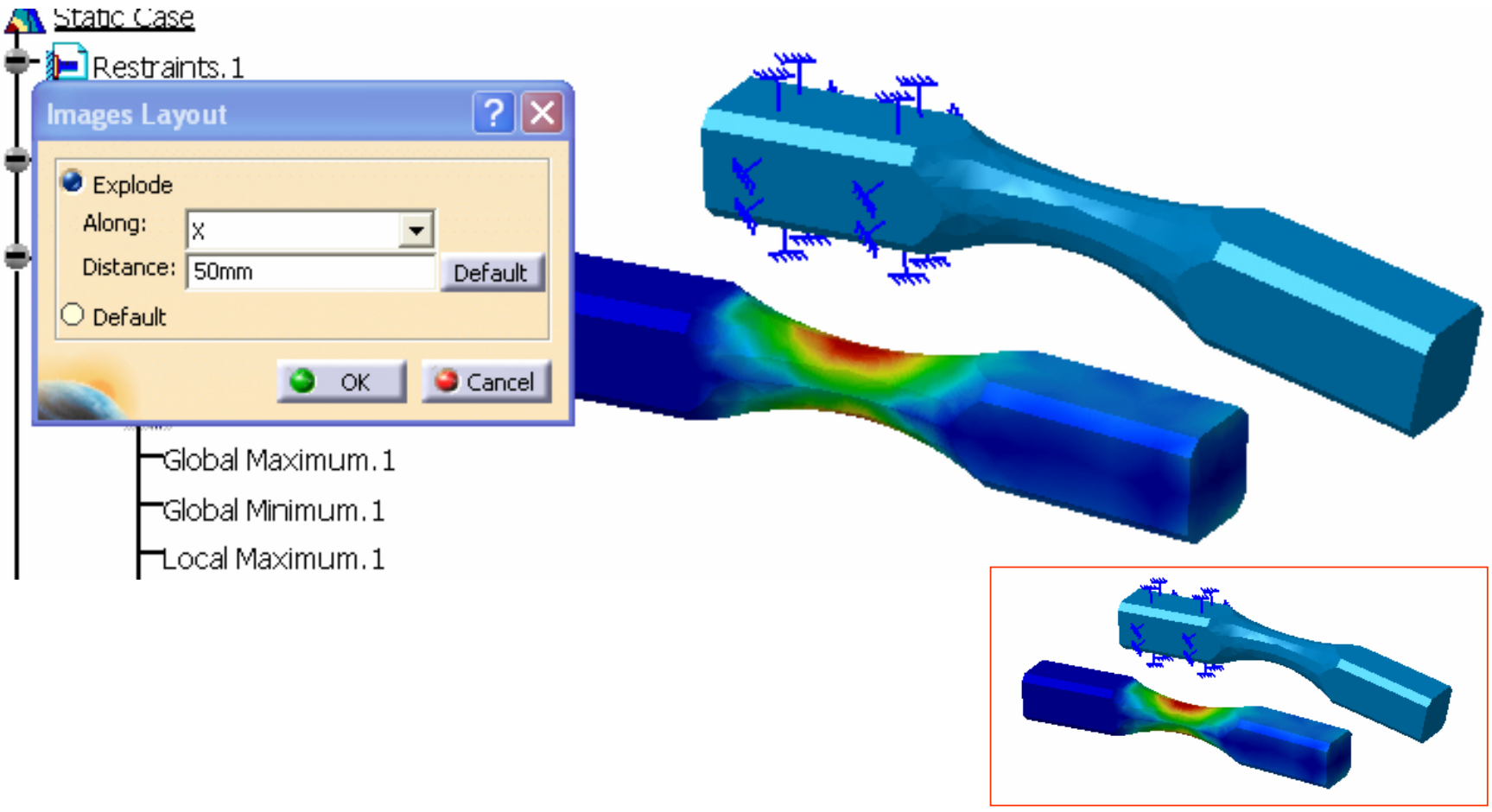
Select *Deformed Mesh* and *Von Mises Stress (nodal mode)* on the tree.

Press Rigth Mouse Button and selec *Activate/Deactivate* option.



GSA – Aalysis Tools: Images Layout

Select *Images Layout* icon. Select object an set offset between images.



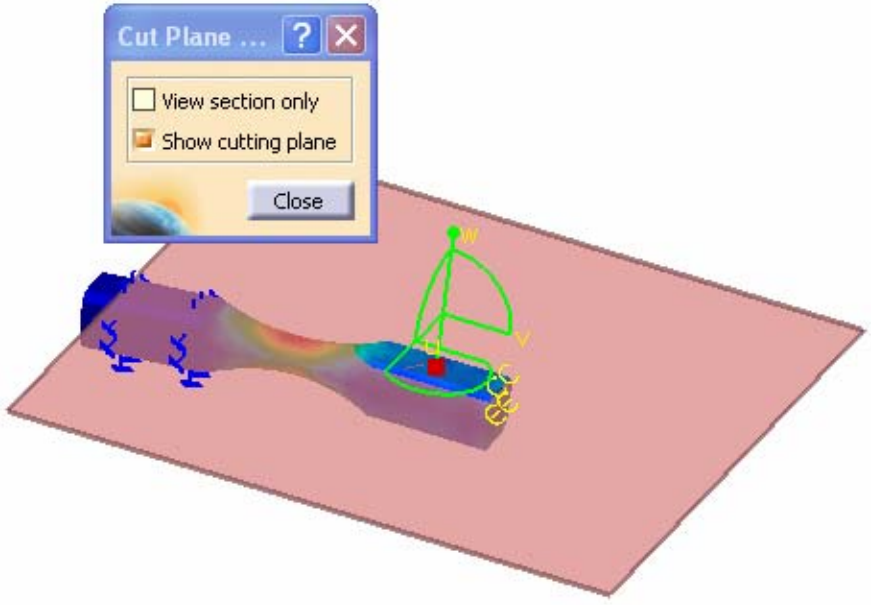
GSA – Aalysis Tools: Cut Plane Analysis

Cut Plane Analysis consists in visualizing results in a plane section through the structure.

By dynamically changing the position and orientation of the cutting plane, you can rapidly analyze the results inside the system.

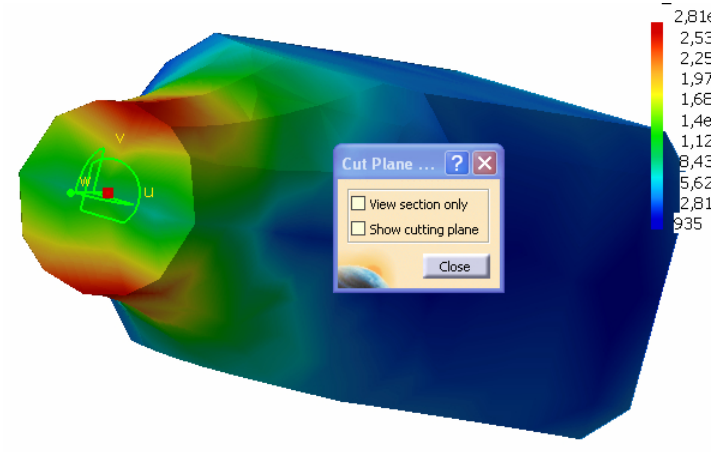
1. Position the compass on the face that will be considered as the reference section.

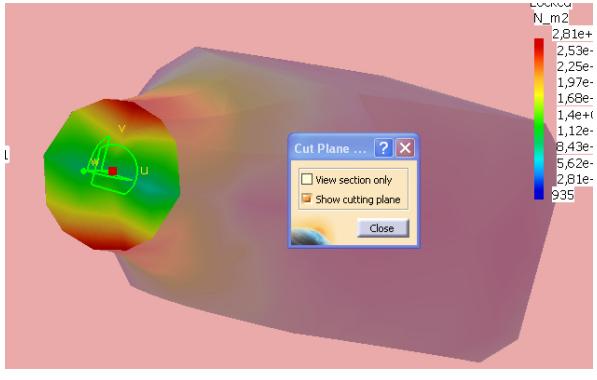
2. Click the Cut Plane Analysis icon. The Cutting Plane appears.

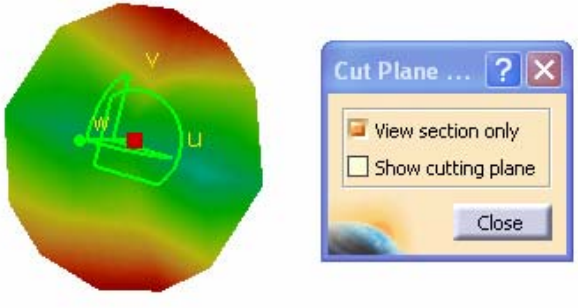


GSA – Aalysis Tools: Cut Plane Analysis

3. You can hide the cutting plane (*Sow cutting plane* check-box)
4. You can see the view section only (*View section only* check-box).
5. Use 3D Compass manipulation to set proper orientation of the cutting plane.

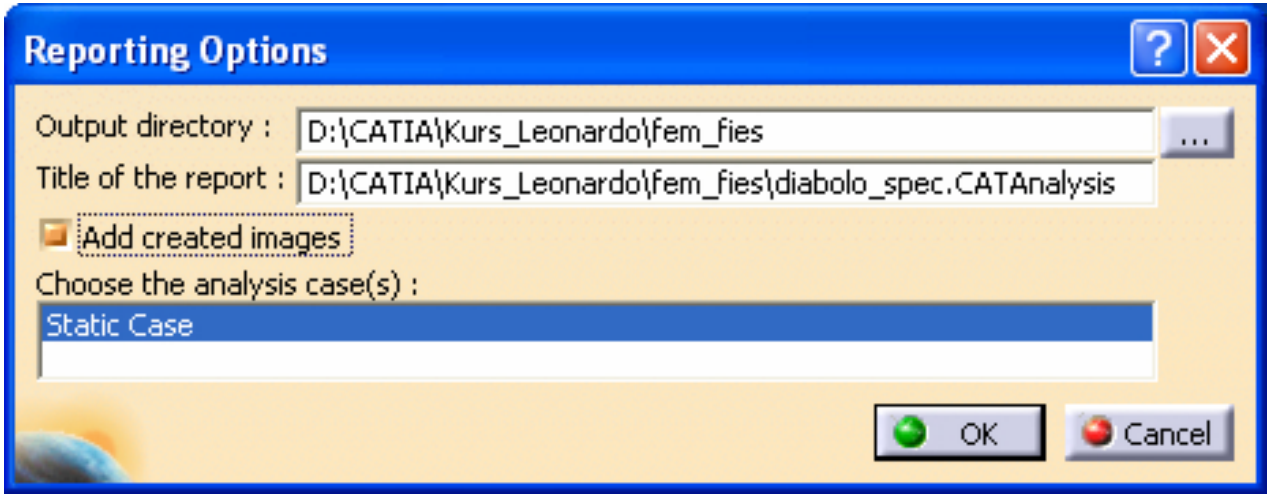






GSA – Aalysis Result: Basic Analysis Report

The *Basic Analysis Report* allows to generate report from analysis.
Select option to start generation. *Reporting Option* window appears.
Specify *Output directory* and *Title of report*.
The user can add all generated images to the report.



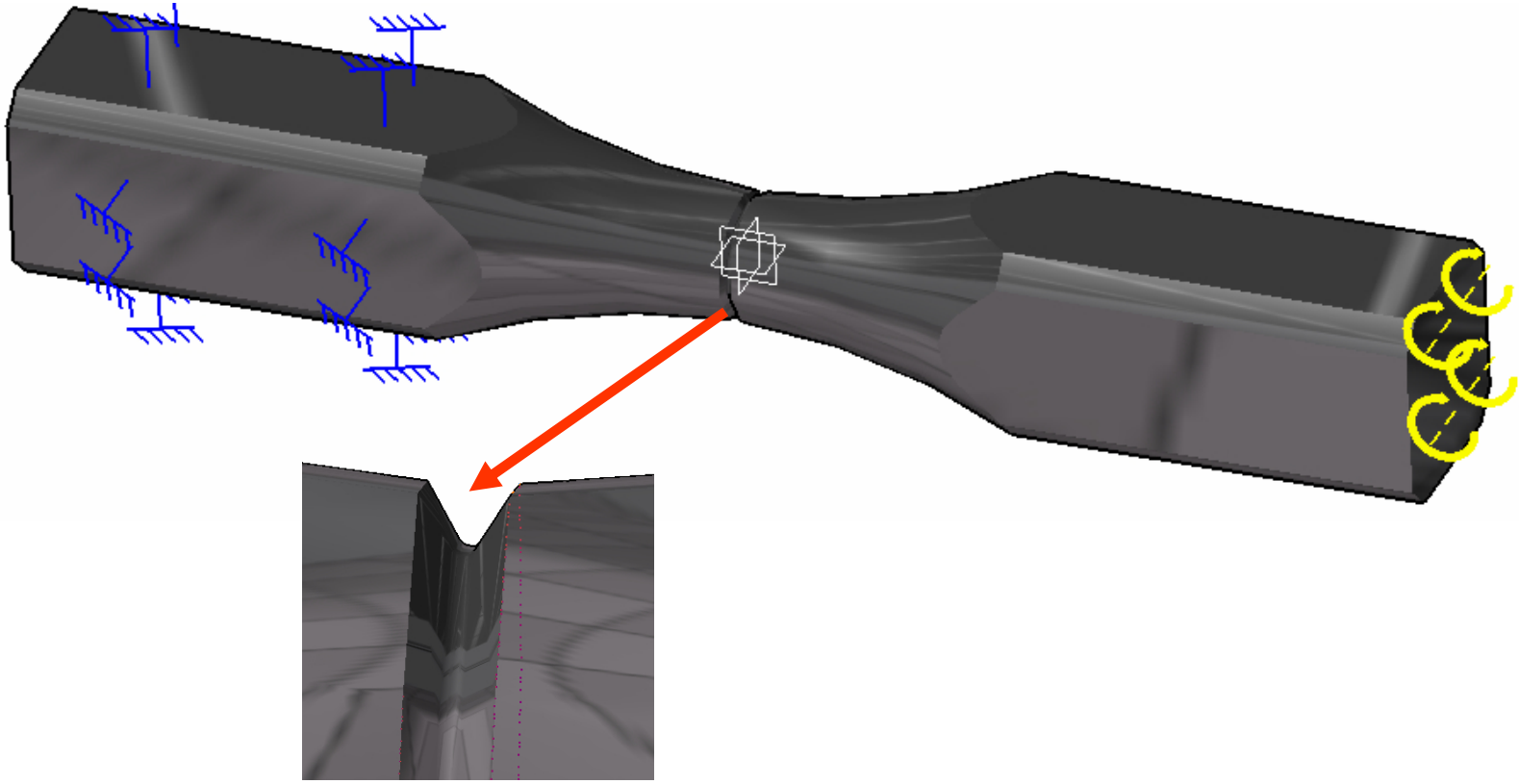
To read the report *Web Browser* is necessary.

GSA – Short task

Open *diabolo_spec_notched.CATPart* file.

Repeat similar analysis for specimen witch notch.

Suggestions: define three areas for which the mesh size is smaller (Local Mesh Size). Set the Local Mesh Size for notch surfaces equal 0,1 or 0,05.



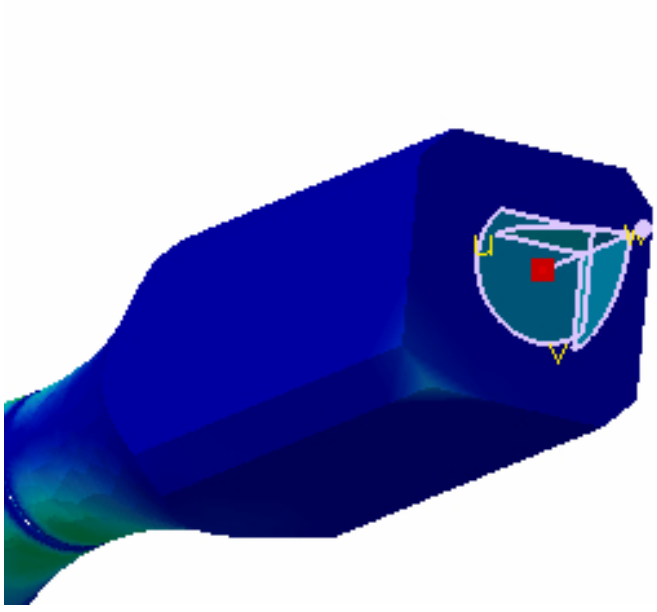
GSA – Short task

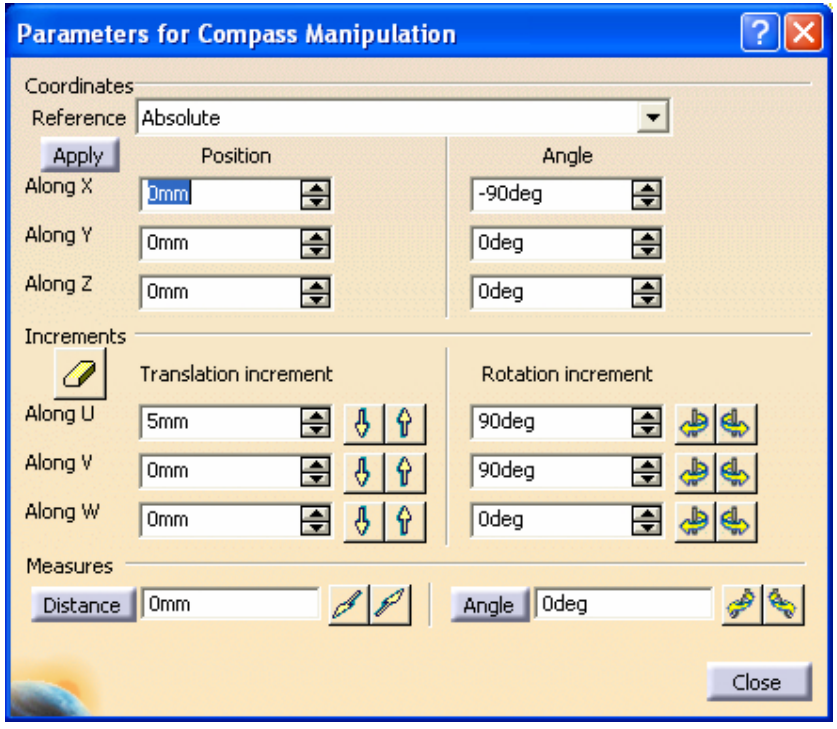
You can define the *Cutting Plane Analysis* more precisely by using exact 3D compass manipulation.

Place 3D compass on one of the surface of the specimen. Double-click on the compass. Set the *Position* value equal 0 for all directions (X, Y, Z).

Set *Angle* for X axis rotation equal -90deg.

Press *Apply* button. The 3D compass is changed his position.

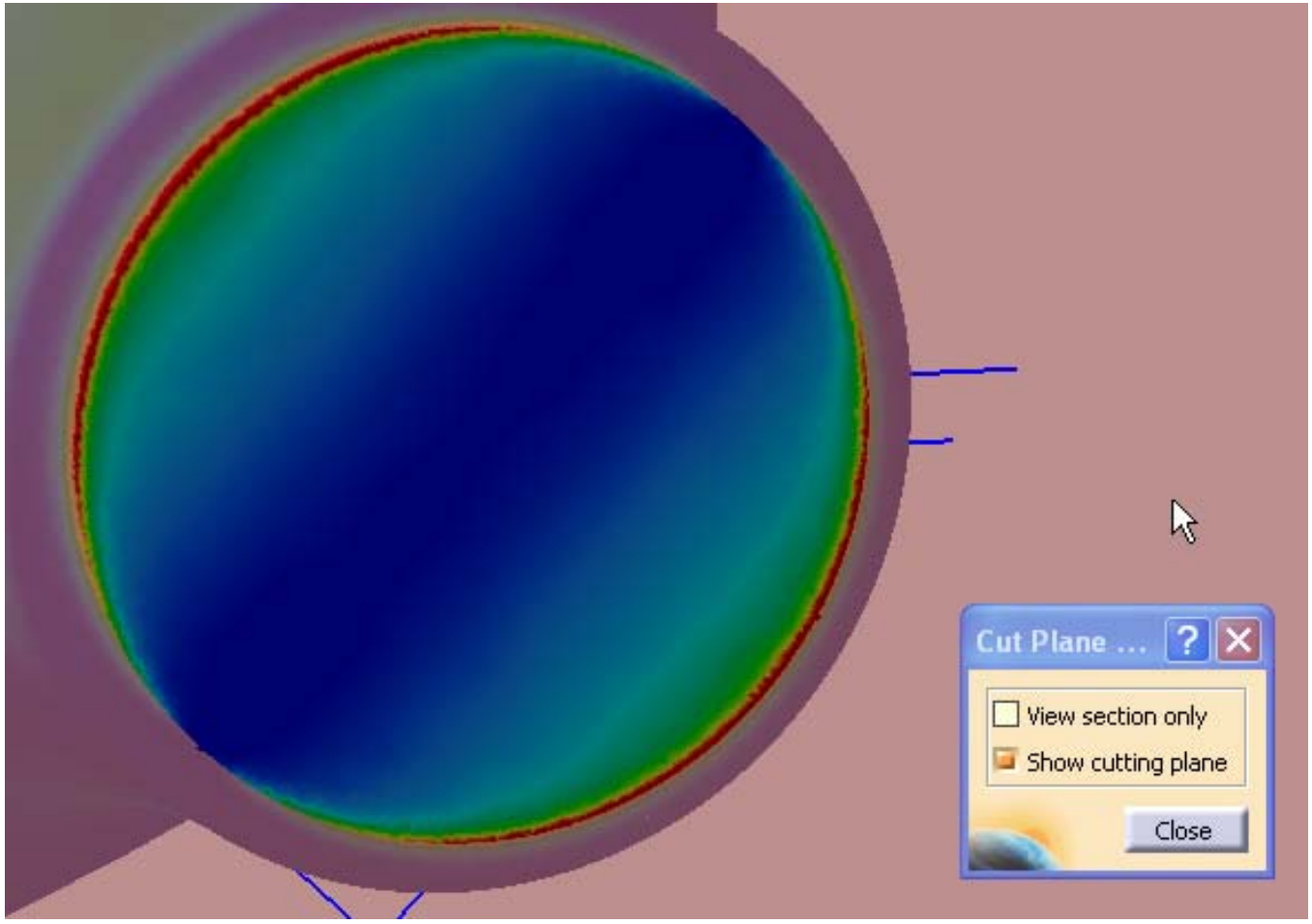




GSA – Short task

Now select *Cut Plane Analysis*  .

You will obtain section view exactly thru notch tip.



GSA – Short task

Double-click on the 3D compass once again. Set *Angle* for Z axis rotation equal -90deg. Press *Apply* button.

Now select *Cut Plane Analysis*  and hide the cut plane.

You will obtain section view along axis of the specimen.