











## Generative Part Stress Analysis

-  [Preface](#)
-  [What's new ?](#)
-  [Getting Started](#)
-  [Basic Tasks](#)
-  [Workbench](#)
-  [Description](#)
-  [Glossary](#)
-  [Index](#)



P1



P2



© Dassault Systèmes 1994-99. All rights reserved.

# Preface

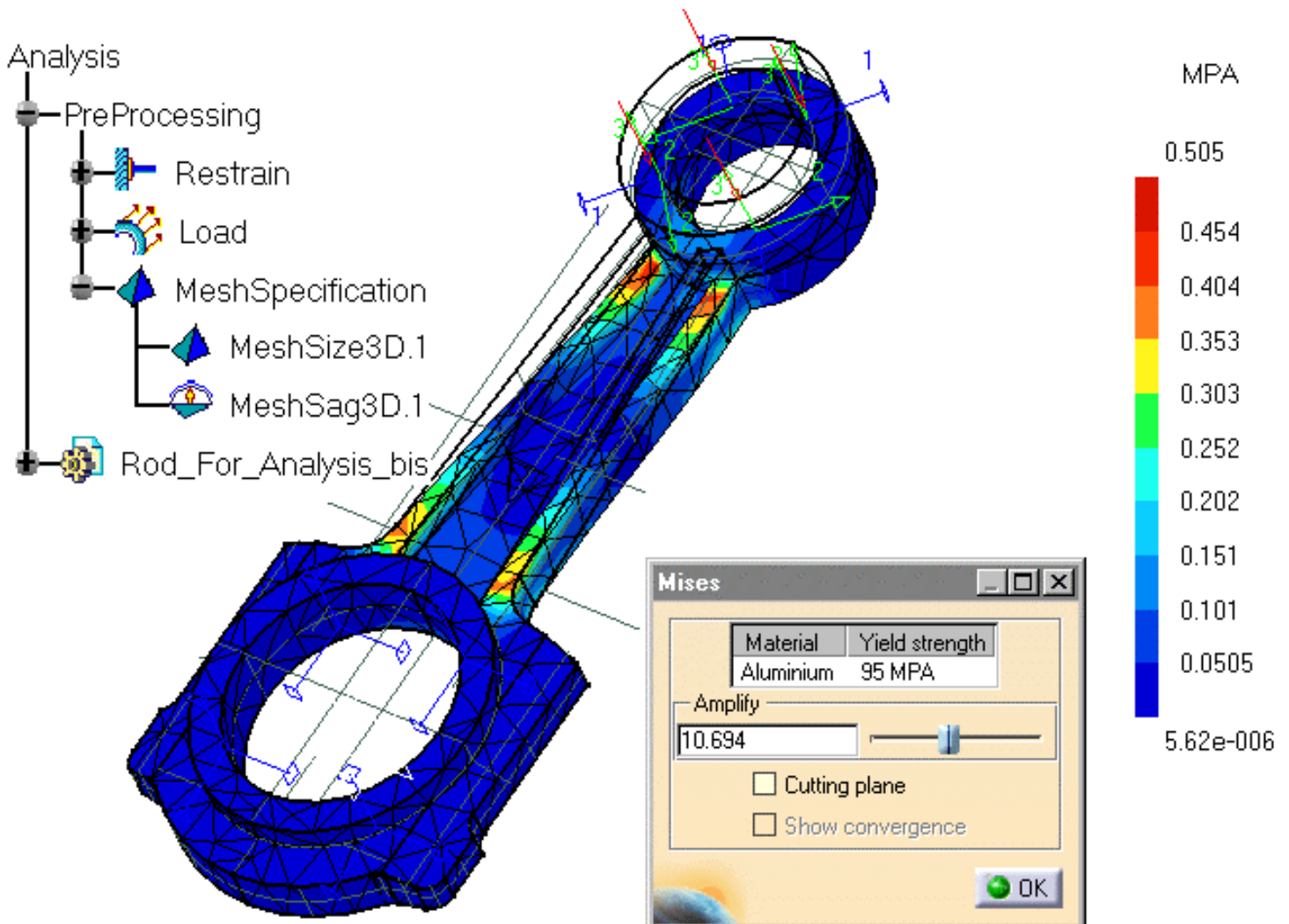
CATIA - Generative Part Structural Analysis and Generative Assembly Structural Analysis Version 5 Release 3 allows you to quickly perform first order mechanical analysis for 3D parts.

This application is intended for the casual user. Indeed, its intuitive interface offers the possibility to obtain mechanical behavior information with very few interactions. The dialog boxes are self explanatory and require practically no methodology, all defining steps being commutative.

As a scalable product, CATIA - Generative Part Structural Analysis Version 5 Release 3 can be cooperatively used with other current or future companion CATIA Version 5 products such as CATIA - Part Design, CATIA - Assembly Design and CATIA - Generative Drafting. The widest application portfolio in the industry is also accessible through interoperability with CATIA Solutions Version 4 to enable support of the full product development process from initial concept to product in operation.

The product *CATIA - Generative Assembly Structural Analysis* has been designed as a very useful extension to *CATIA - Generative Part Structural Analysis* enabling the mechanical behavior study of a whole assembly. This product is fully integrated in *Generative Part Structural Analysis* product and has been conceived with the same "easy to learn" and "fun to use" ergonomy creating then a very powerful tool.

The *CATIA - Generative Part Structural Analysis* and the *CATIA - Generative Assembly Structural Analysis User's Guides* have been designed to show you how to analyze a part or an assembly operating within a specified environment. There are several ways of subjecting a part to external actions. There are also several types of useful output information. This book aims at illustrating these various possibilities.

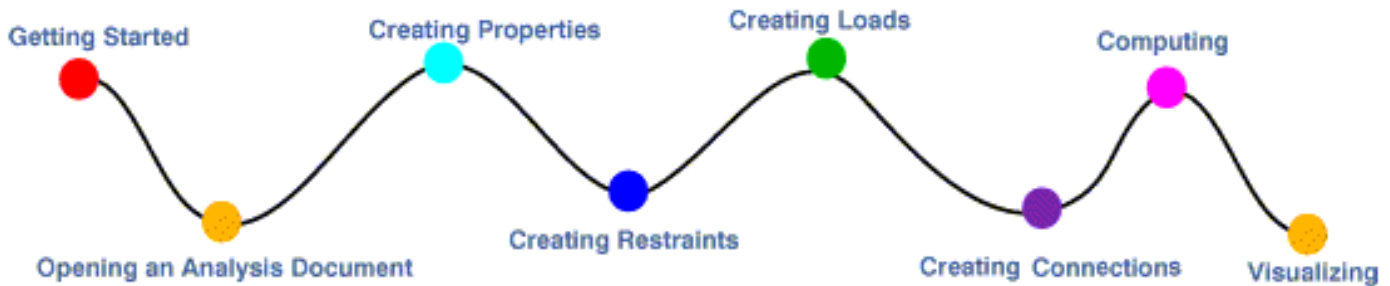




# Using This Guide

This book is intended for the user who needs to quickly become familiar with CATIA - Generative Part Structural Analysis and *CATIA - Generative Assembly Structural Analysis* Version 5 Release 3. The user should be familiar with basic CATIA Version 5 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 next sections present the main capabilities, in the form of basic user's tasks. It may be a good idea to take a look at the section describing the menus and toolbars.



# Where to Find More Information

Prior to reading this book, we recommend that you read the *CATIA Version 5 Infrastructure User's Guide*.

*CATIA - Part Design User's Guide* and *CATIA - Assembly Design User's Guide* may also prove useful.



# What's new ?

CATIA Version 5 Release 3 presents new functionalities for CATIA - *Generative Part Structural Analysis* product:

- The [Adaptivity functionality](#) enables local mesh refinements respecting a Precision Goal.
- After two computations, the [Convergence](#) of parameters as the Maximum of von Mises stresses for example can be studied through a Function Editor.
- The [mesh Sag and Size specifications](#) ergonomics has been enhanced.
- A [Dynamic display](#) enables the user to visualize mesh or nodes local values resulting from a computation.
- The [Color Palette](#) customization functionalities have been improved.
- The [Wireframe Visualization Mode](#) for ISO-Values has been developed.

# Getting Started

Before getting into the detailed instructions for using CATIA - Generative Part Structural Analysis and CATIA - Generative Assembly Structural Analysis Version 5 Release 3, the following tutorial aims at giving you a feel of what you can do with the product. It provides a step-by-step scenario showing you how to use key functionalities.

The main tasks described in this section are:

## Tasks

[Entering the Stress Analysis Workbench and Selecting a Part](#)

[Creating Restraints](#)

[Creating a Force](#)

[Creating a Slider Connection](#)

[Computing a Stress Analysis](#)

[Computing a Normal Modes Analysis](#)

[Viewing Displacements](#)



These tasks should take about 20 minutes to complete.

# Entering the Stress Analysis Workbench and Selecting a Part



This first task will show you how to enter the Stress Analysis workbench and load a part.



Before starting this scenario, you should be familiar with the basic commands common to all workbenches. These are described in the *CATIA Version 5 Release 3 Infrastructure User's Guide*.

You can use the Rod\_For\_Analysis.CATPart document from the SAMPLES/gps\_analysis directory for this task.



1. Select File -> Open then select the desired .CATPart file. This opens a Part Design document containing the selected part.
2. Select the part in the feature tree.

3. Click the Apply Material icon  .

- The Material library appears.

4. Select a material family. Select the desired material from the displayed list then. Click OK.

- The material is applied.



You can visualize the material properties and its analysis characteristics by selecting the material in the feature tree and using Edit -> Properties -> Analysis.

5. Select File -> New then select Analysis from the List of Types.

- An empty document and the Analysis toolbar are displayed.



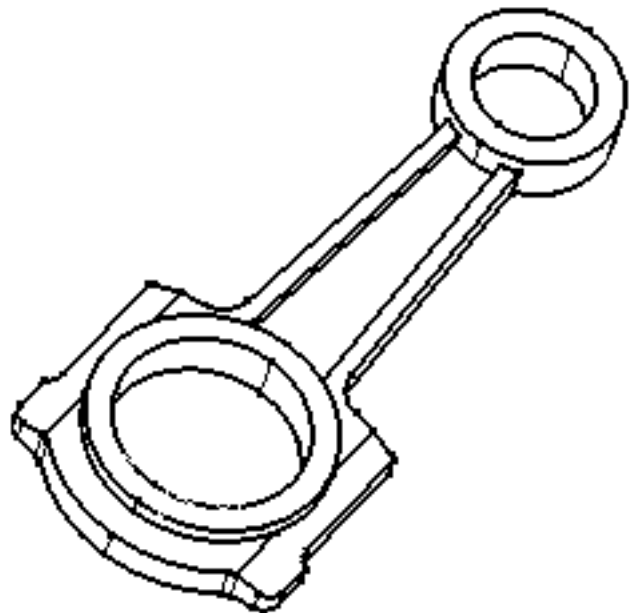
6. Select Window -> Tile Horizontally.

- To have a easy access to the part to load.

7. Click the Select Part icon  .

8. Click the Part in the Part Design document.

- The part is then loaded in the Stress Analysis workbench.

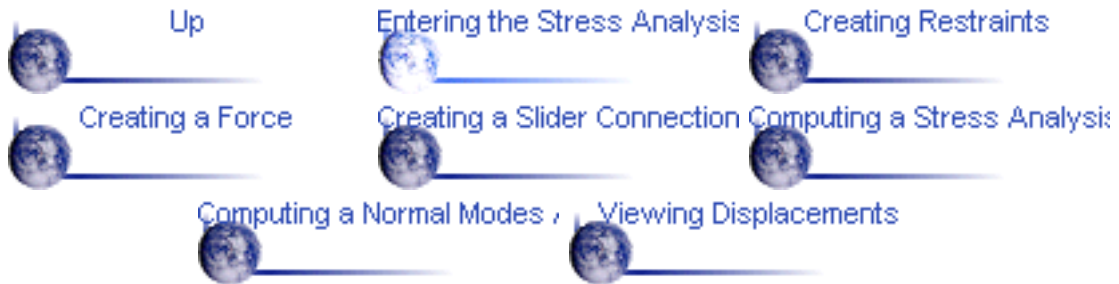


If you select Start->Analysis & Simulation -> Stress Analysis from a CATPart document containing the part without any material, the material library will appear directly for an easy material selection.





You can use command Start->Analysis & Simulation -> Stress Analysis from a CATPart document instead of steps 5 to 8.



# Creating Restraints

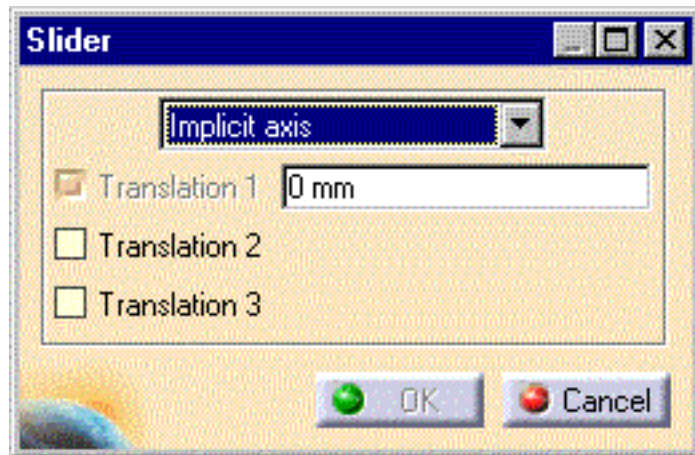


This task will show you how to create slider and clamp restraints on a part.

1. Click the Slider icon

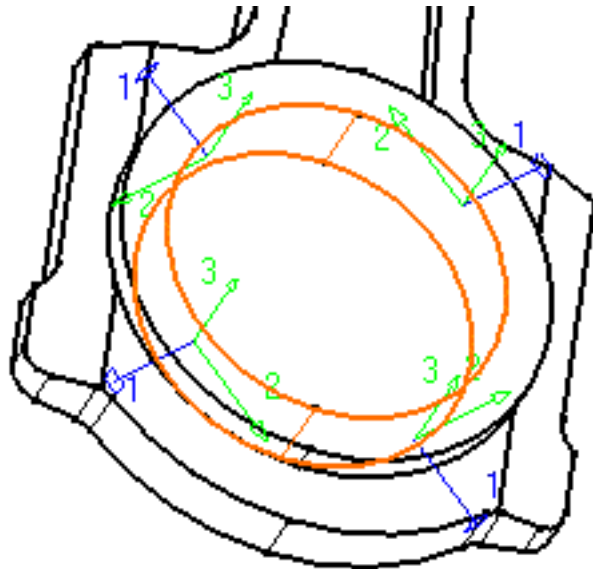


- The Slider dialog box appears.

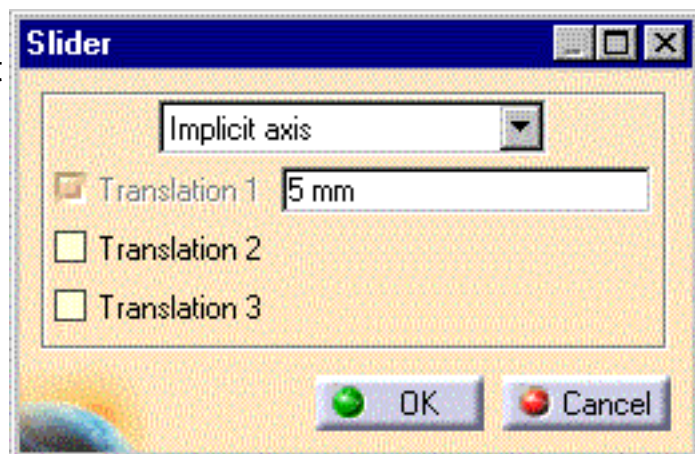


2. Select a geometry (an edge ( Reference axis only ) or a surface : any other geometry you can highlight with the cursor).

- Several axis systems are also displayed on the geometry to symbolize the anticipated result. Green axes represent free translation directions while blue axes represent fixed translation directions.



3. Activate the degrees of freedom on which you want to impose a restraint. Specify the value of each imposed displacement, expressing all of them in local or global coordinates. (The local axis system: cartesian, circular or revolute corresponds to the geometry you have selected).



4. If you need to create more sliders, select other geometrical entities to create the restraint.

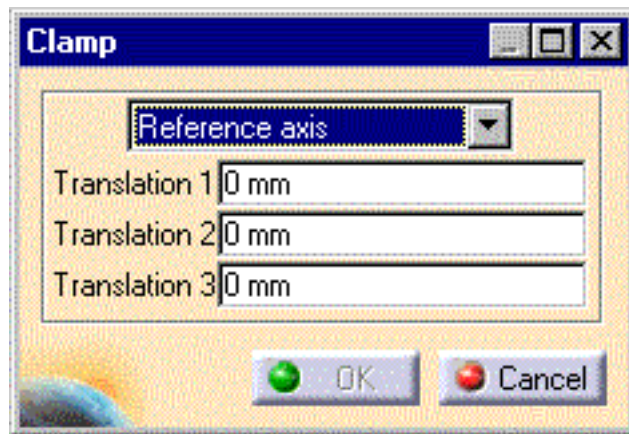
- The dialog box is still displayed and you can create other sliders by repeating the previous steps.

5. Click OK to create the Slider.

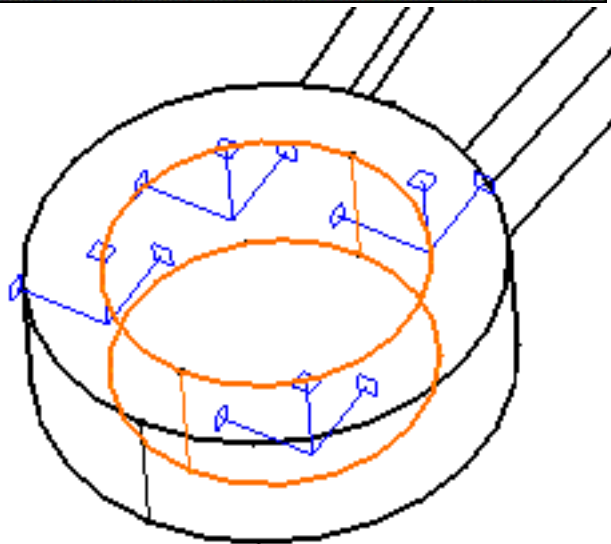
6. Click the Clamp icon



- The Clamp dialog box appears.

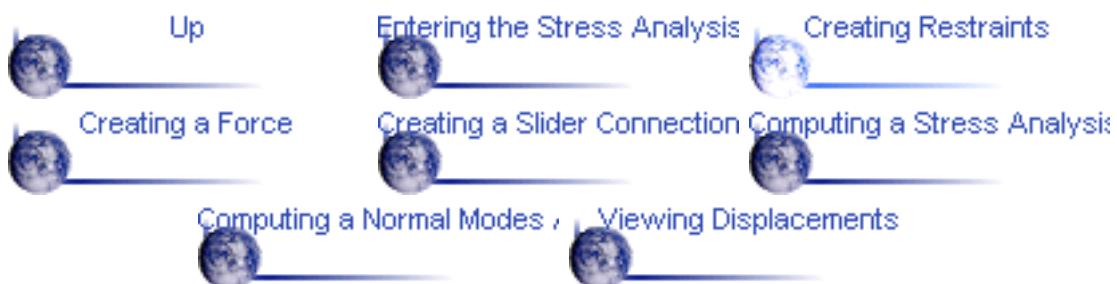


7. Select the cylindrical geometry as shown.



8. Click OK to create the restraint.

The final restraints on your part are as shown.



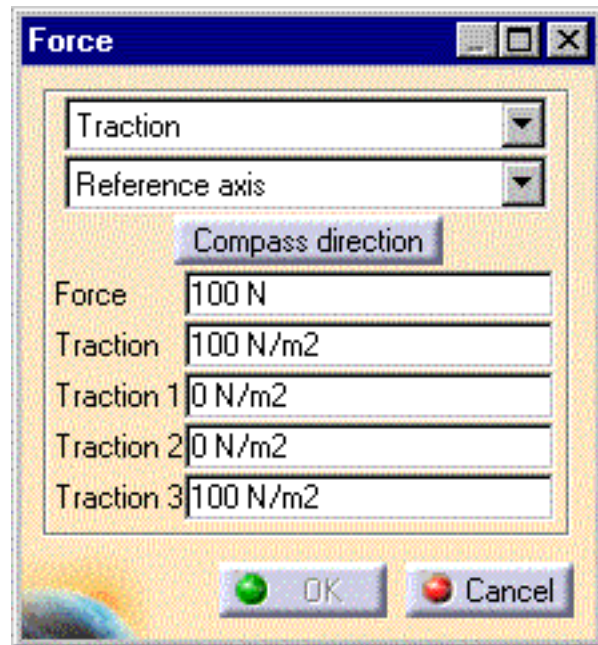
# Creating a Force



This task will show you how to create a Force.

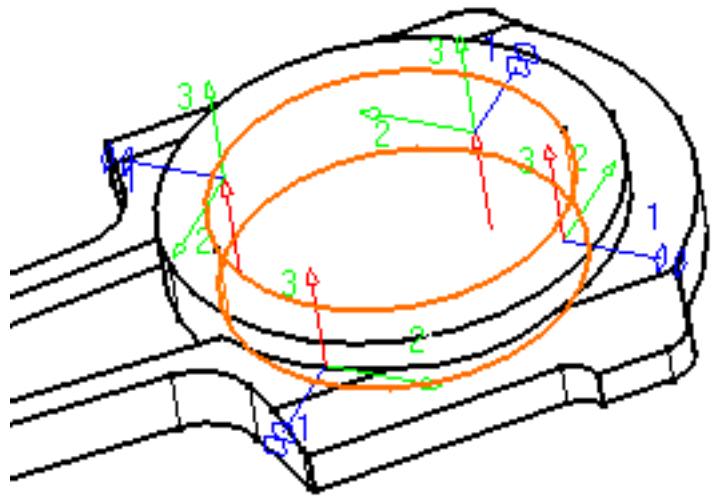
1. Click the Force icon .

- The Force dialog box appears.

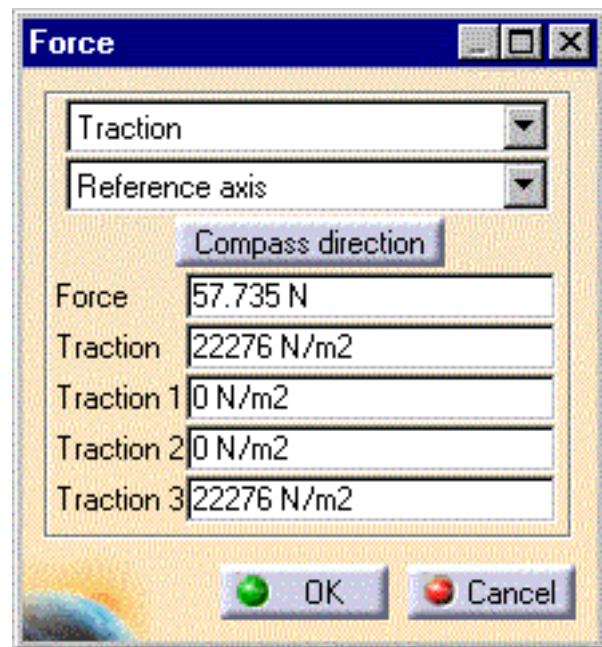


2. Select a geometry.

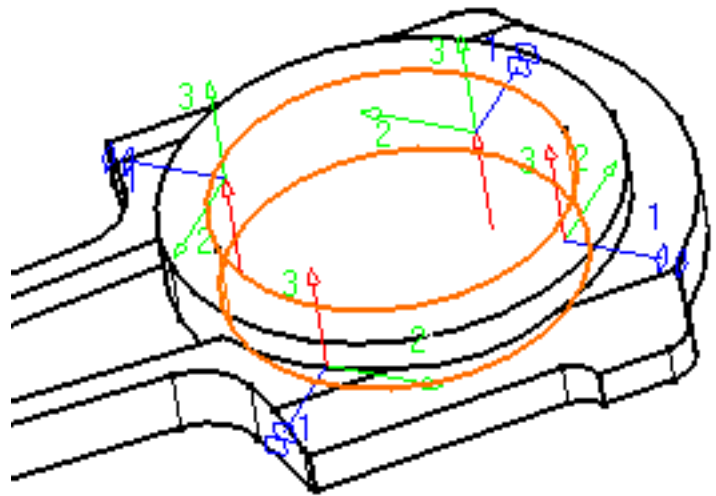
- Several arrows previewing the anticipated result are displayed.



3. Set the values of each force component according to the axis system you have chosen. The arrows are refreshed once new values are entered.



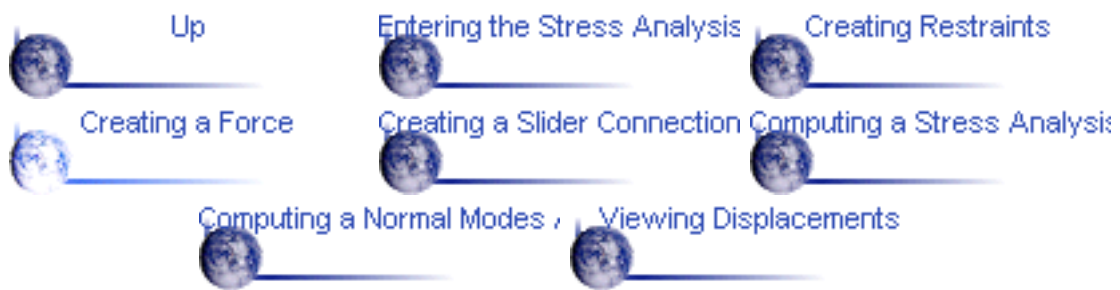
For a CATIA - P2 interface, you can select the compass to impose the direction of the arrows: Drag the compass by handling the red square and drop it on the appropriate surface. The normal (perpendicular) direction to this surface defines the new direction. Then, click on the Compass Direction button to take this new direction into account. You can now invert the direction if desired, editing the values of the three components. Please refer to the *CATIA Infrastructure User's Guide* for further information.



4. Click OK.

- The Force is created.







# Creating a Slider Connection



This task will show you how to create a Slider Connection between two parts.

This task is independent from the previous ones.



You must own the Generative Assembly Structural Analysis product to follow this scenario.

You can use the Rod\_For\_Analysis.CATPart and the Rod\_Axis\_For\_Analysis.CATPart documents from the SAMPLES/gps\_analysis directory for this task.



1. Select "Product 1" ( in the feature tree of the product document ) then select Insert -> Existing Component.

( If a product document is not yet opened, execute File -> New and select Product in the type list ).

2. Select the first Part :  
Rod\_For\_Analysis.CATPart.

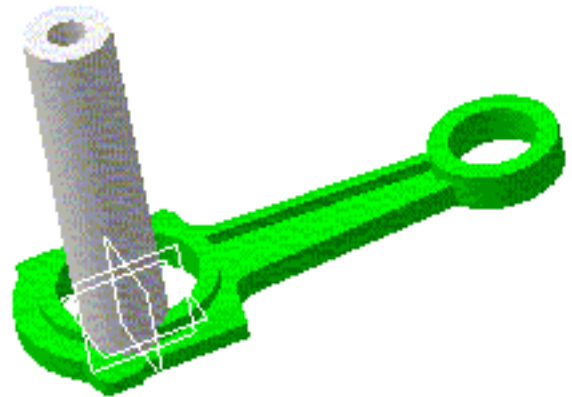
3. Repeat Steps 1 and 2 for the second Part : Rod\_Axis\_For\_Analysis.CATPart  
.



You can use the Copy and Paste or Copy and Paste-Special method to insert a part from a Part Design document to a Product one. For further information please refer to the *CATIA Assembly Design User's Guide*.


4. You must now apply a material on each Part:

-Select Product1 then select Edit -> Design Mode.



5. Expand the feature tree to access to the part's body of the first part.

-Double-click on it to edit it. The Part Design Toolbar appears.

-Select the "Apply Material"  Icon and choose one of the presented materials.



If you use the Functionality Start->Modal or Stress Analysis from a document including a part without any material, the Material Library appears and you can select one of them to apply it on the part before entering the Analysis workbench. Therefore you do not have to enter the Part Design workshop for such an operation.

6. Repeat Step 5 to apply a material on the second part .

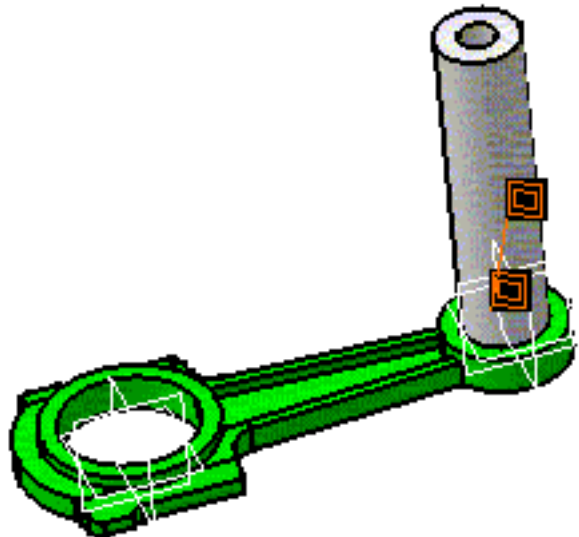
7. Double-click on "Product1" in the feature tree to come back to the Assembly Design workshop.



8. Select the "Contact Constraint" icon.

9. Select the outer surface of the cylinder and the corresponding inner surface of the rod.

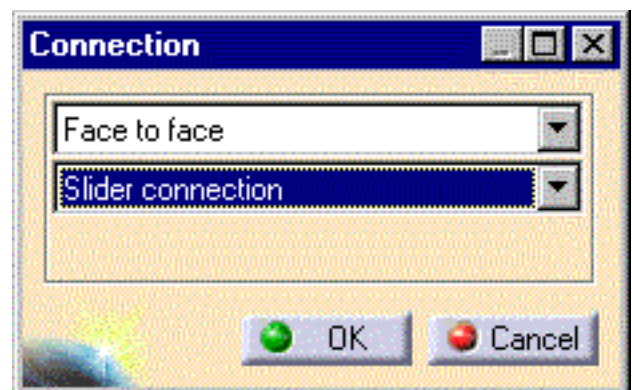
10. Click OK to validate.



11. Select Start->Analysis Simulation ->Stress Analysis to enter the Stress Analysis workbench.

12. Select the "Parts Connection" icon.

-The Connection Panel appears.



13. Select Face to face and Slider connection options in the combo boxes.

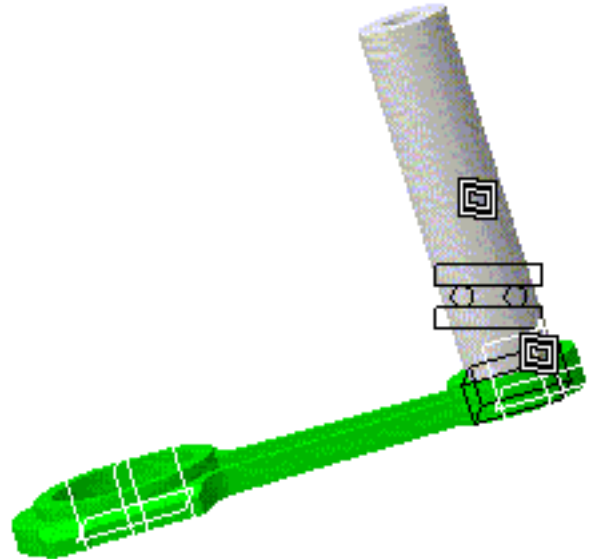
14. Select the constraint symbol which links the parts.

15. Click OK.



16. The Slider Connection is created.

-A symbol representing this particular connection is created. The Assembly is then designed.

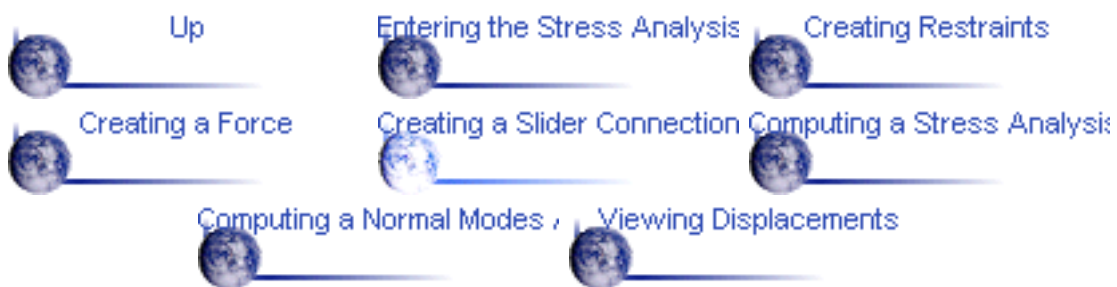


Be sure to fix all the global degrees of freedom of your assembly otherwise a global singularity will be raised at computation (such a model is insolvable). To correct easily the model, the induced motion of the assembly will be simulated and visualized after computation.



The CATProduct and the CATAnalysis files are now associated: If you modify the CATProduct and you save the CATAnalysis file, the CATProduct file will be saved as well.

Once all restraints and loads are set on the Assembly, you can launch a computation as defined in the previous tasks : [Computing a Stress Analysis](#) or [Computing a Normal Modes Analysis](#).



# Computing a Stress Analysis



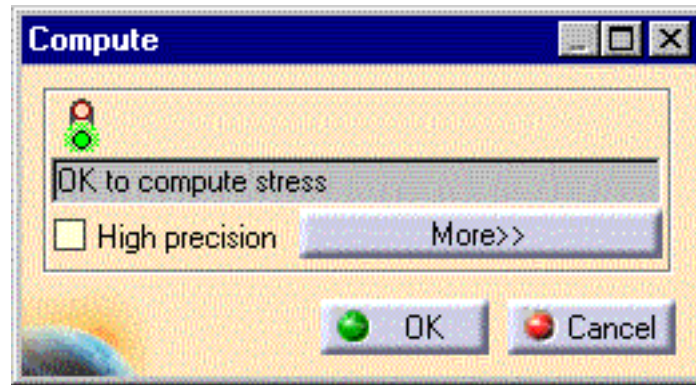
This task will show you how to compute a stress analysis and to visualize von Mises stresses results.

You must have performed the first three tasks prior to beginning this task.



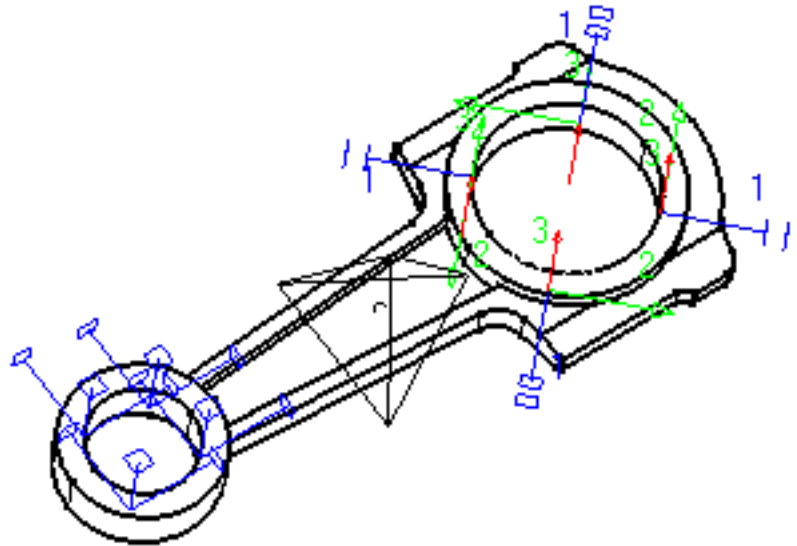
1. Click the Compute icon .

- The Compute dialog box appears.

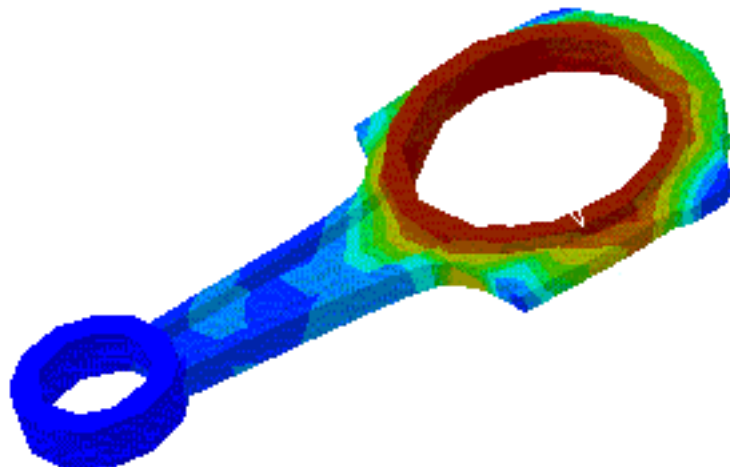


A Status message indicates if the program is ready for computation.

Otherwise a missing element would be declared (at least one restraint, for example).




2. Click OK to compute and display the resulting stresses.

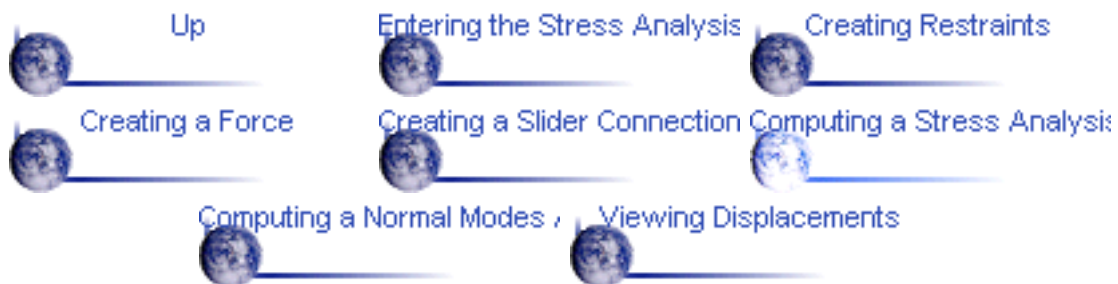
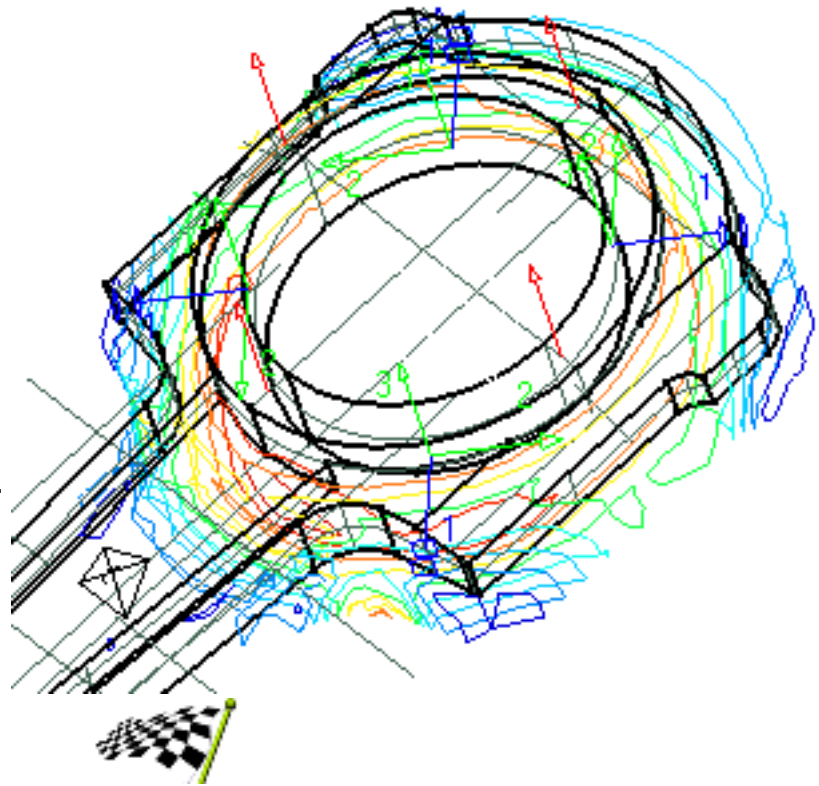


new

3. Click the Wireframe

icon  in the general toolbar.

The von Mises results are visualized with curves which represent the different thresholds.



# Computing a Normal Modes Analysis

A Normal Modes Analysis computation corresponds to a sequence of operations resulting in the computation of dynamic mode shapes and vibration frequencies of the part.



This task shows how to run a normal modes computation on a specified part.



You must use the Dynamics Analysis workbench to perform this task.

The part material, additional masses (if any) and restraints (if any) need to be specified prior to the computation. Note that:

- not specifying a restraint has the effect of generating free body vibration modes.
- not specifying additional mass distributions has the effect of generating vibration modes under the effect of the part's structural mass only.

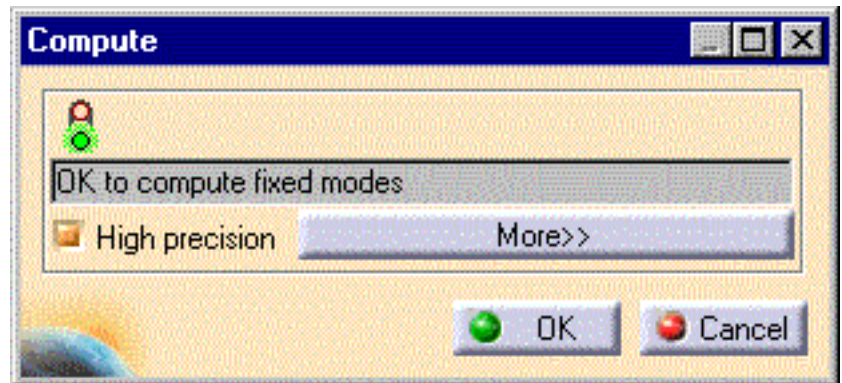
You can use the Rod\_For\_Analysis.CATPart document from the SAMPLES/gps\_analysis directory for this task.



1. Click the Compute icon

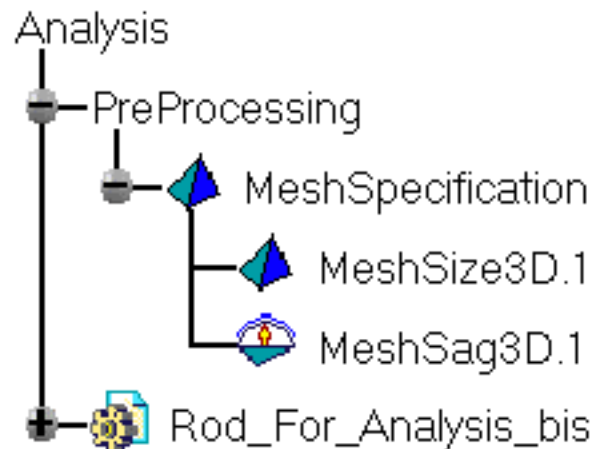


- The Compute dialog box is displayed.



2. The overall Mesh Sag and Mesh Size features are created in the features tree.

3. Click on Cancel in the Compute panel to edit later on those features.



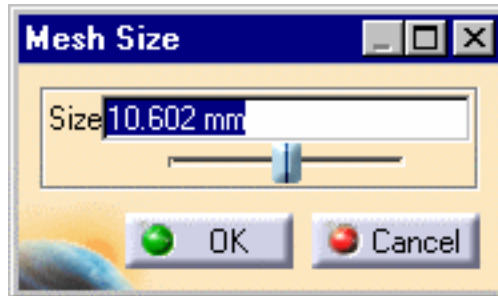
4. By default their values are convenient for a rough computation ( with a linear interpolation). However you can modify them to set more appropriate values: Double click on the Mesh Size symbol



the features tree.

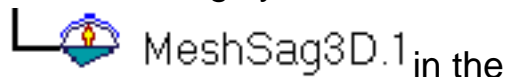
- The Mesh Size dialog box appears

5. Set an overall adequate value.



6. Click Ok.

7. Repeat steps 4 to 6 with the Mesh Sag symbol

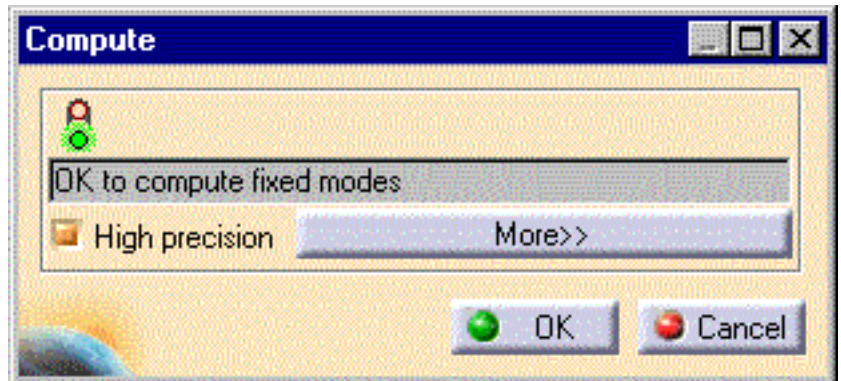


the features tree.

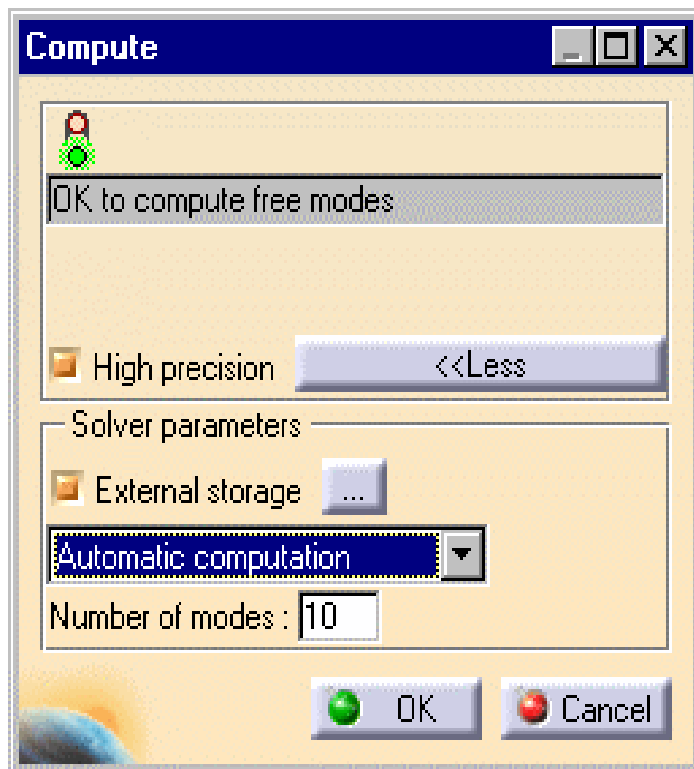
8. Click the Compute icon



- The Compute dialog box is displayed.



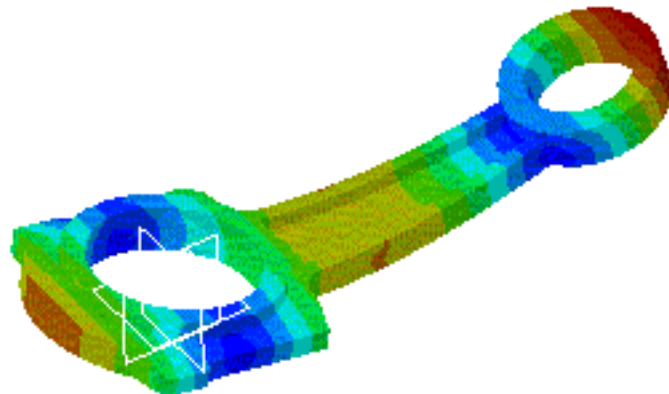
9. Click More to enlarge the panel: you can modify the computation parameters such as the Number of Modes if you wish.



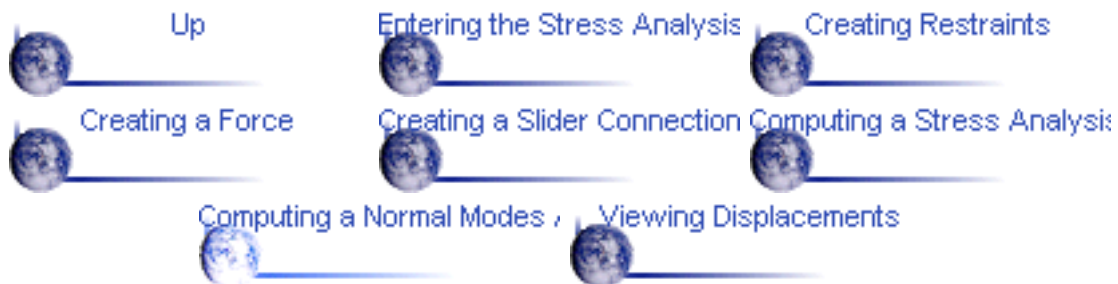
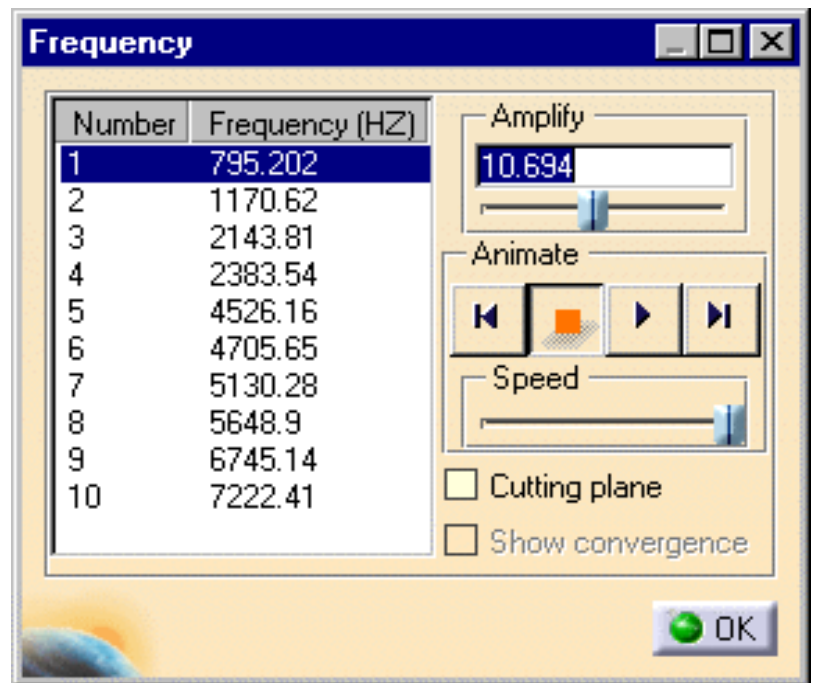
10. Click OK to run the computation.

- The status of the computation is displayed on the Status line

11.The results are displayed.



12.You can select a normal mode in the presented list and animate it.





# Viewing Displacements



This task will show you how to visualize Displacements after a computation.

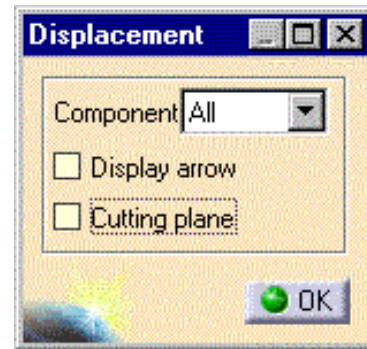


You must have successfully performed a Stress Computation prior to beginning this task.



1. Click the Displacement icon .

- The Displacement dialog box appears.  
Leave the Display Arrow box unchecked.

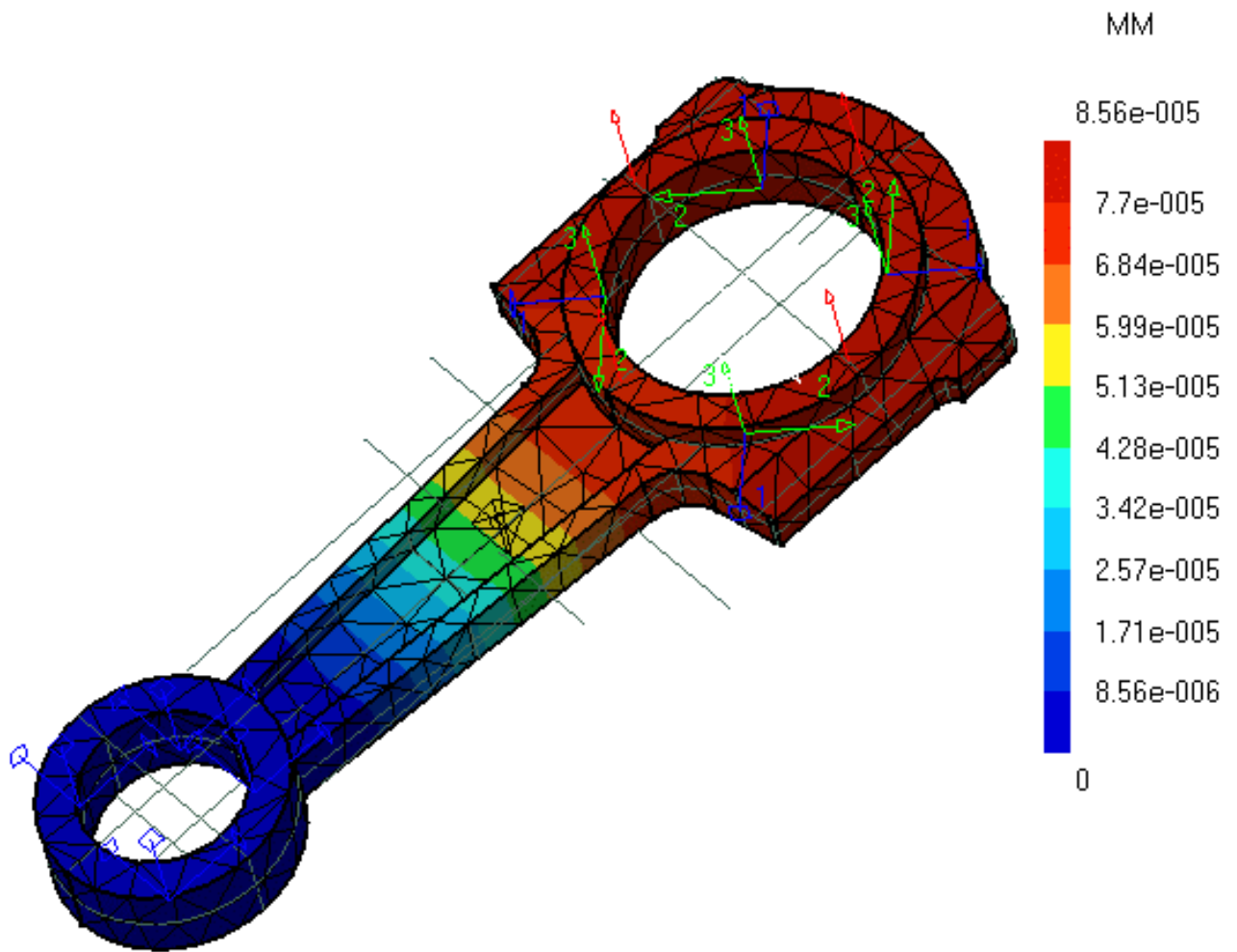


An ISO-value of the part displacement field magnitude is displayed.

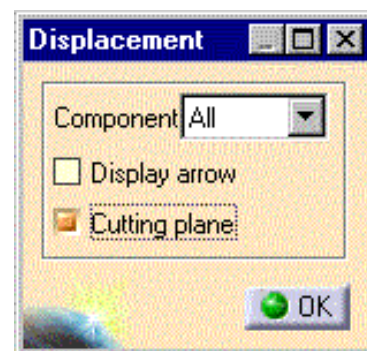
The Palette dialog box enables you to modify the color distribution to focus on specific values.

For this functionality refer to the task [Editing the Color Palette](#),

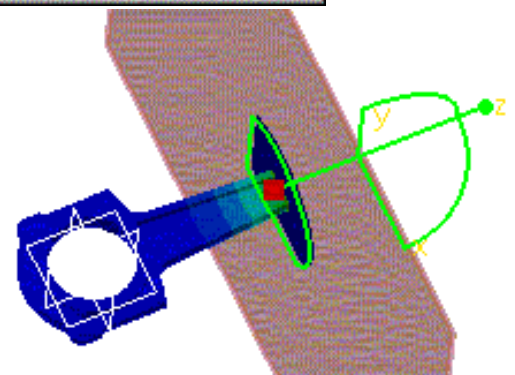




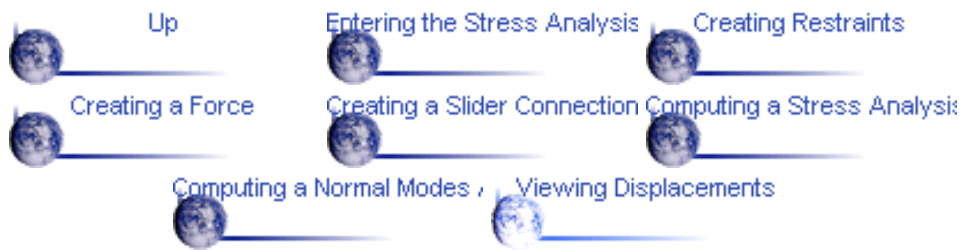
2. You can visualize the part inner displacements using the Cutting Plane: select the corresponding checkbox (CATIA - P2 interface and licence only ).



You can handle the compass with the mouse in order to rotate or translate the Cutting Plane.( To do so, select an edge of the compass and drag the mouse ).



3. Click OK to exit this view.



# Basic Tasks

The basic tasks you will perform in the *CATIA - Generative Part Structural Analysis workbench* are mainly creations of *analysis features* that you will use for the mechanical analysis of your part.

Once the required specifications are defined, you need to compute and visualize the results.

The Basic User Tasks section will explain and illustrate how to create part *physical attributes*, specify computation parameters and visualize results.

You can make extensive use of the CAD-CAE *associativity* concept. Associativity means that any part modifications occurring outside the Analysis workbench are automatically reflected when performing tasks within the *Analysis workbench*. In particular, any parametric changes on the part are automatically accounted for. So, you don't have to worry about updating the part specifications.

The workbench provides *generative* capabilities: you do not have to tell the program explicitly the necessary steps to perform a mechanical analysis. In fact, all you need to enter are the specifications about the way in which the part is subjected to its environment. The program then automatically generates the desired results.

The table below lists the information you will find in the Basic User Tasks section.

| Theme  | Purpose  |
|--|--|
| <a href="#">Managing Material Properties</a> | shows how to apply and edit the material properties of the part                |
| <a href="#">Creating Connections</a>         | shows how to define connections in an assembly model                           |
| <a href="#">Creating Restraints</a>          | shows how to specify restraints (displacement-type boundary conditions)        |
| <a href="#">Creating Loads</a>               | shows how to specify loads (force-type boundary conditions)                    |
| <a href="#">Computing Results</a>            | shows how to define computation parameters and how to submit a computation job |
| <a href="#">Visualizing Results</a>          | shows how to display the results of a computation                              |

# Managing Material Properties

*CATIA - Generative Part Structural Analysis* provides easy methods to apply, analyze and customize material properties that include analysis characteristics.

## Tasks

[Applying Material Properties](#)

[Analyzing Part Material](#)

[Customizing Part Material](#)



# Applying Material Properties

Parts used in this product are *isotropic, mono-material*. For a solid part, Properties represent the mechanical and thermal characteristics of the part's material.

Since the mechanical constitutive equations are based on material characteristics, you must specify the part Properties prior to analyzing its behavior.



This task shows how to apply a property to a part.



You can use the Rod\_For\_Analysis.CATPart document from the SAMPLES/gps\_analysis directory for this task.

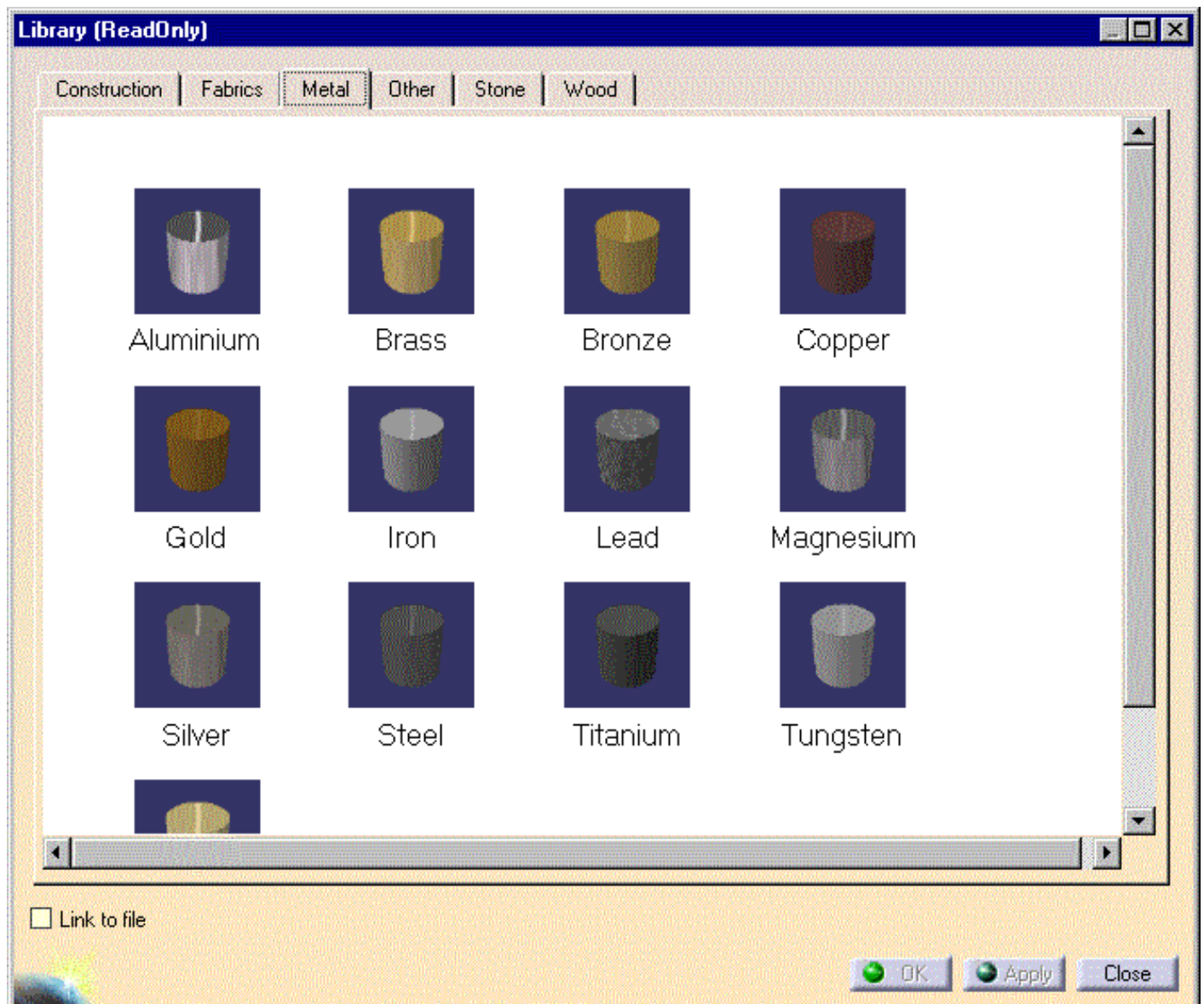


1. Select File -> Open then select the desired .CATPart file. This opens a Part Design document containing the selected part.

2. Select the part in the feature tree.


3. Click the Apply Material icon .


• The Material library appears.



4. Select a material family. Select the desired material from the displayed list then click OK.

• The material is applied.

 You can also use the Copy and Paste method to apply a selected material from the library to the part.

 You can only define one material for a given part. If you need to specify different materials to different sub-parts, you must sub-divide your design into different parts accordingly and analyze them together or separately.





# Analyzing Part Material



This task shows how to analyze the property of a part.



You can use the Rod\_For\_Analysis.CATPart document from the SAMPLES/gps\_analysis directory for this task.



1. Select the material in the feature tree then select Edit -> Properties

- The Properties dialog box appears.

2. Select the Analysis tab to display the Structural Properties of the part's material.

The Properties dialog box is shown with the 'Analysis' tab selected. The 'Current Selection' is 'Aluminium'. The 'Structural Properties' section is expanded, showing the following values:

| Structural Properties |                            |
|-----------------------|----------------------------|
| Comment               | Alloy 1100-H14 ( 99 % Al ) |
| Young Modulus         | 7e+010 N/m2                |
| Poisson Ratio         | 0.346                      |
| Density               | 2710 kg/m3                 |
| Thermal Expansion     | 2.36e-005                  |
| Yield Strength        | 9.5e+007 N/m2              |

At the bottom of the dialog box are three buttons: OK, Apply, and Cancel.



You can modify the analysis characteristics of the selected part's material. Any modifications done only apply to this part.

3. Click OK to save any modifications.



Applying Material Properties



Analyzing Part Material



Customizing Part Material





# Customizing Part Material

There are two ways to customize a part material.



This task shows the first way to customize the material property of a part. This method can only be used by an administrator.



There are no prerequisites for this task.



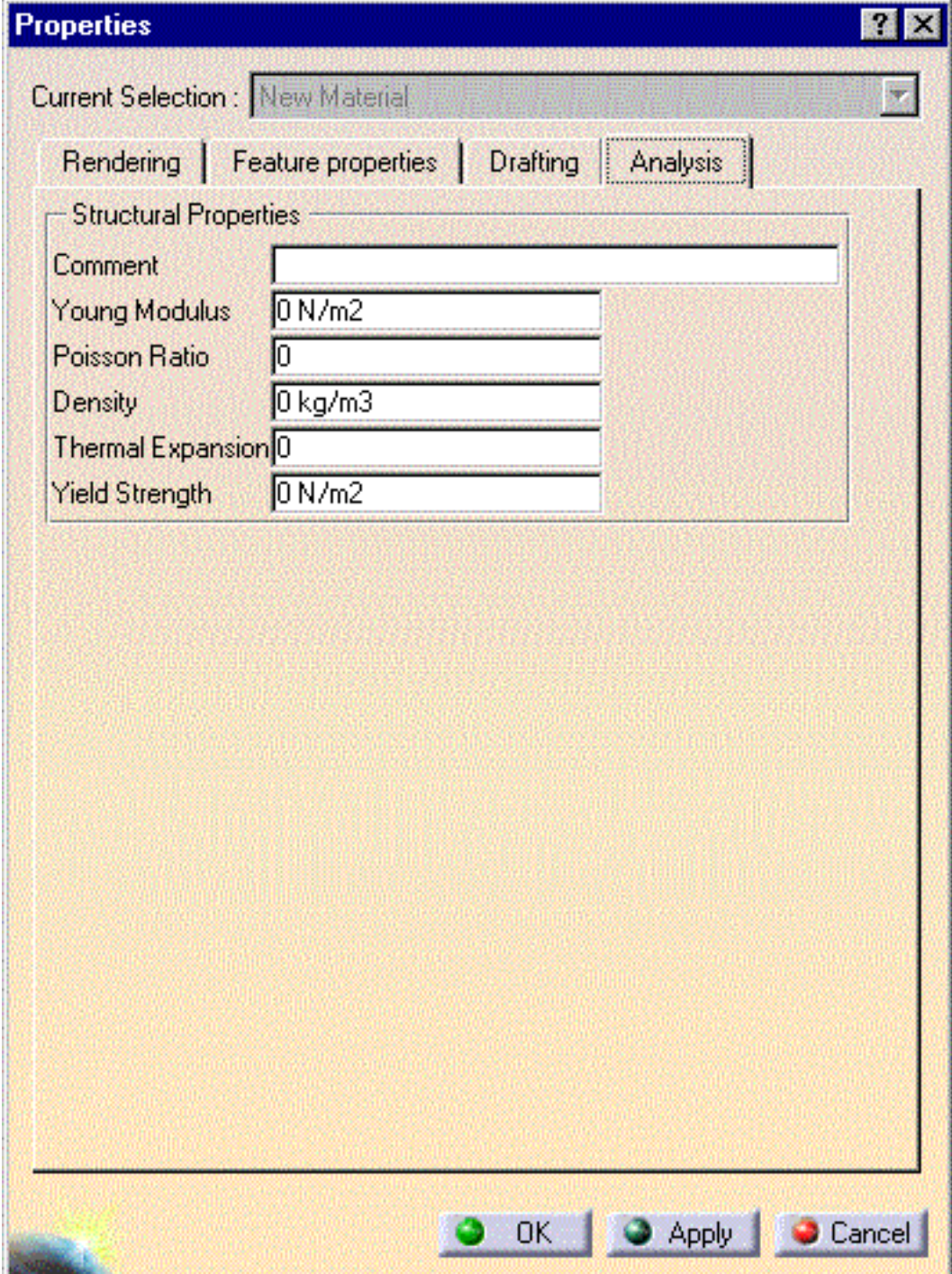
1. Select Start->Infrastructure -> Material Library.

- The material library appears.

2. Double-click the desired material.

- The Properties dialog box appears.

3. Select the Analysis tab to display the Structural Properties of the material.

The Properties dialog box is shown with the 'Analysis' tab selected. It contains a 'Current Selection' dropdown set to 'New Material'. The 'Structural Properties' section is expanded, showing input fields for Comment, Young Modulus (0 N/m2), Poisson Ratio (0), Density (0 kg/m3), Thermal Expansion (0), and Yield Strength (0 N/m2). The dialog has OK, Apply, and Cancel buttons at the bottom.

| Properties   |         |
|--|---------|
| Current Selection : New Material                     |         |
| Rendering   Feature properties   Drafting   Analysis |         |
| Structural Properties                                |         |
| Comment  |         |
| Young Modulus  | 0 N/m2  |
| Poisson Ratio  | 0       |
| Density  | 0 kg/m3 |
| Thermal Expansion                                    | 0       |
| Yield Strength                                       | 0 N/m2  |
| OK Apply Cancel                                      |         |



You can modify the analysis characteristics of the selected material. Any modifications done only apply to this material of this library. For more information, please refer to your *CATIA Version 5 Release 3 Infrastructure User's Guide*.

4. Select File -> Save As to save the library in an appropriate directory named 'material'. The name of this library must be 'Catalog.CATMaterial'.

5. You must concatenate the directory path containing the material directory at the start of the environment variable called 'CATStartupPath'.

The first 'Catalog.CATMaterial' file indicated by the 'CATStartupPath' variable will be taken as the default library accessible by the Apply Material icon of the Part Design workbench. For more information about this functionality, please refer to your *CATIA Version 5 Release 3 Infrastructure User's Guide*:

[Customizing your environment on NT/UNIX Chapter.](#)



This task shows the second way to customize the material's property of a part. This method can be used by both users and administrators.

There are no prerequisites for this task.



1. Select Start -> Infrastructure -> Material Library.

- The material library appears.

2. Double-click the desired material.

- The Properties dialog box appears.

3. Select the Analysis tab to display the Structural Properties of the material.



You can modify the analysis characteristics of the selected material. Any modifications done only apply to this material of this library. For more information, please refer to your *CATIA Version 5 Release 3 Infrastructure User's Guide*.

4. Select File -> Save As to save the library in a personal directory.

5. You can now use this library to apply a material to a part using the Copy and Paste method.

You must use File -> Open to access the library and use your personal material's library.





Up



Applying Material Properties



Analyzing Part Material



Customizing Part Material

# Creating Connections

CATIA - Generative Assembly Structural Analysis provides easy methods to create Kinematic Links between parts.

Using the Stress and Modal Analysis workbenches you can create between parts::

- [Fastened connections.](#)
- [Slider connections.](#)
- [Contact connections](#) ( Stress workbench only )

You can as well create between a part and a virtual body :

- [Rigid connections.](#)
- [Smooth connections.](#)

Using Connections functionalities you can design multi-parts modelizations and study their Static or Modal Behavior.



Be sure to fixe all the global degrees of freedom of your assembly otherwise a global singularity will be raised at computation ( such a model is insolvable).To correct easily the model, the induced motion of the assembly will be simulated and visualized after computation.



# Creating a Fastened/Slider Connection

Fastened/Slider connections create mesh compatible links between the two parts at their common interface. The mesh quality obtained by this method equals almost one block meshing operations. While the Fastened connection merges both parts at their contact surface and makes them behave as a unique body, the Slider connection enables both bodies to translate tangentially to their common interface.

You must not use Fastened/Sliders connections if the two parts are not in contact then, they would not represent the physical reality of your model. If both do not have any contact areas, you should preferably use Virtual connections.



This task will show you how to create a Fastened or a Slider Connection between parts.

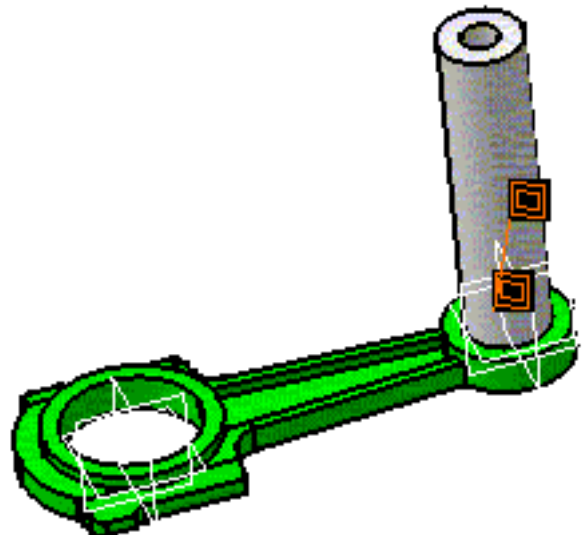


You must own the Generative Assembly Structural Analysis product to follow this scenario.

You can use the Rod\_For\_Analysis.CATPart and the Rod\_Axis\_For\_Analysis.CATPart documents from the SAMPLES/gps\_analysis directory for this task.



1. Repeat steps 1 to 11 of the scenario "Creating a Slider Connection" from the Getting Started section..

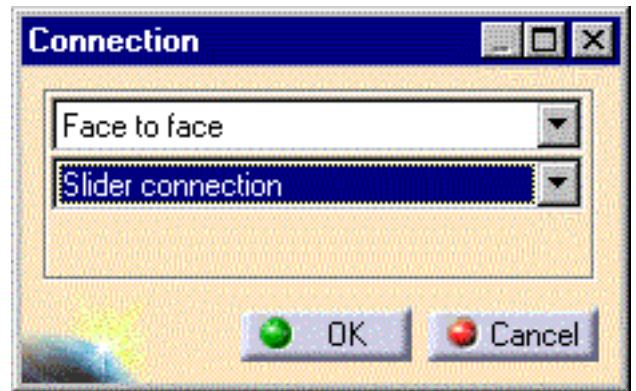


2. Select The "Parts Connection" icon from the Analysis ( Stress or Modal) workshop toolbar.

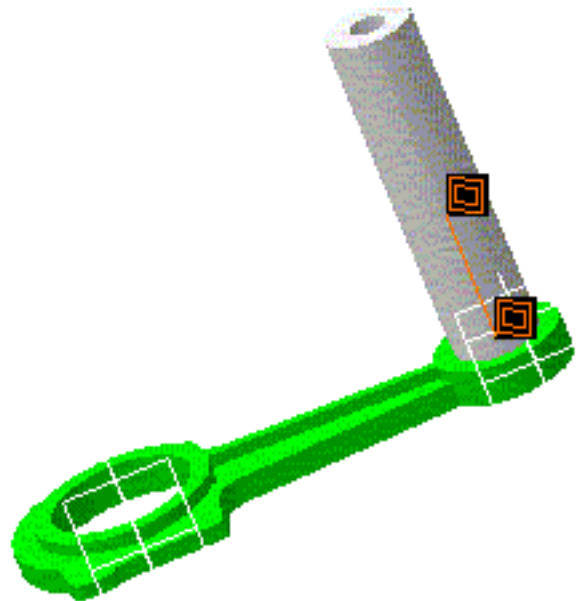
-The Connection panel appears.



3. Select Face to Face and Fastened or Slider connection in the presented combos.



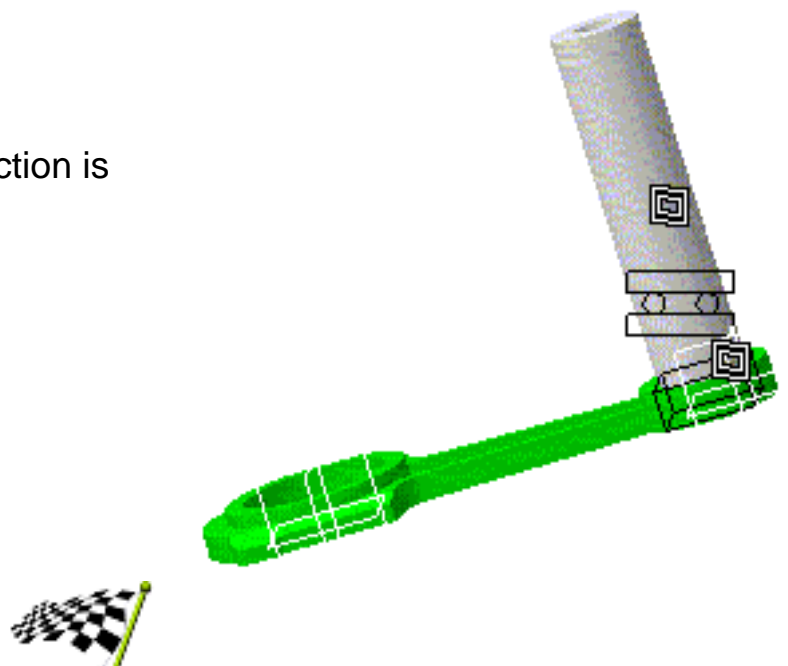
4. Select the Constraint symbol linking the two parts.



5. Click OK.

6. The Fastened or Slider connection is created.

A symbol indicating the type of connection is displayed.





Up



Creating a Fastened



Creating a Contact Connecti

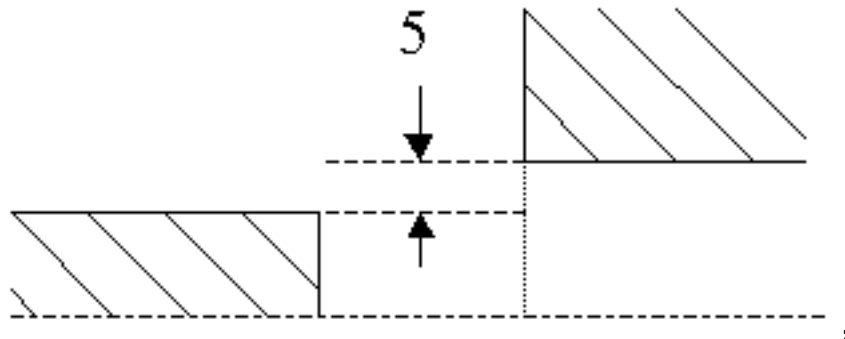


Creating a Virtual Connection

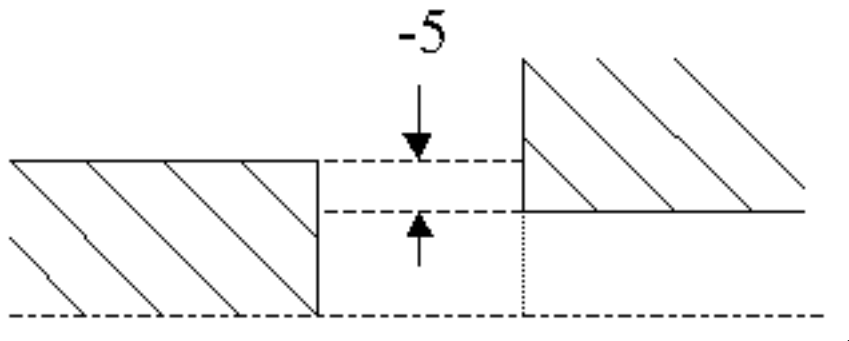
# Creating a Contact Connection

Contact connections create mesh compatible links between the two parts at their common interface. The mesh quality obtained by this method equals almost one block meshing operations. The two parts are free to deform unless the contact condition is reached. If the clearance is negative then a press-fit connection is obtained. The clearance is expressed normally to the contact interface so that a negative clearance value represents the common deformation of both parts.

Positive clearance :



Negative clearance ( before insertion and deformation of both bodies ):



This task will show you how to create a Contact Connection between parts.



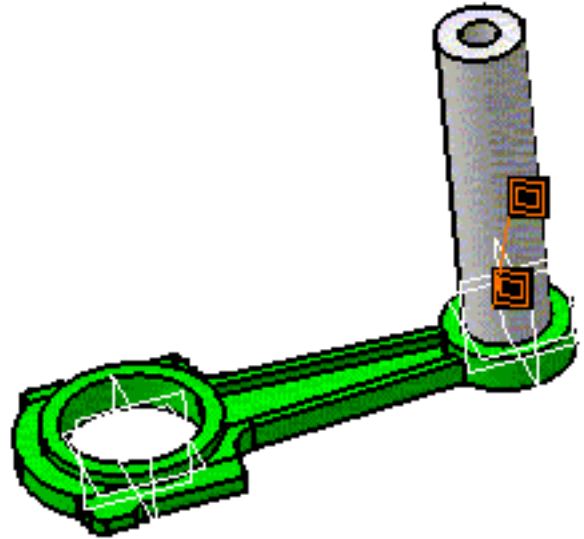
You must own the Generative Assembly Structural Analysis product to follow this scenario.

You can use the Rod\_For\_Analysis.CATPart and the Rod\_Axis\_For\_Analysis.CATPart documents from the SAMPLES/gps\_analysis directory for this task.





1. Repeat steps 1 to 11 of the scenario "Creating a Slider Connection" from the Getting Started section..

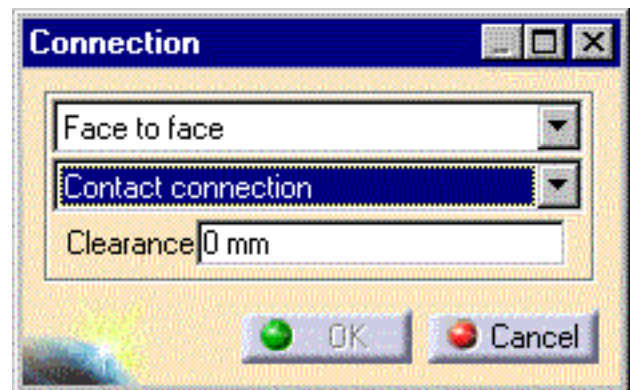


2. Select The "Parts Connection" icon from the Analysis ( Stress or Modal) workshop toolbar

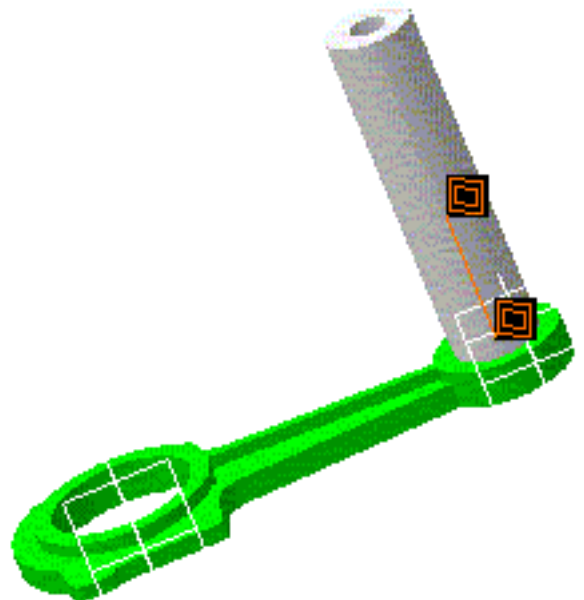
icon from the Analysis ( Stress or Modal) workshop toolbar

-The Connection panel appears.

3. Select Face to Face and Contact connection in the presented combos.



4. Select the Constraint symbol linking the two parts.



5. Set the clearance value and Click OK.

6.The Contact connection is created.

A symbol representing the type of connection is displayed.



# Creating a Virtual Connection

Virtual connections enables the user to link two parts which do not have a common interface with a third body. This third body behaves as a separate mesh transmitting the motion of each part to the other according to two different kinematic laws: the Rigid Connection stiffens each interface and link them rigidly, the Smooth Connection transmits the average motion of each interface to the other leaving them free to deform.



This task will show you how to create a Virtual Connection between parts.

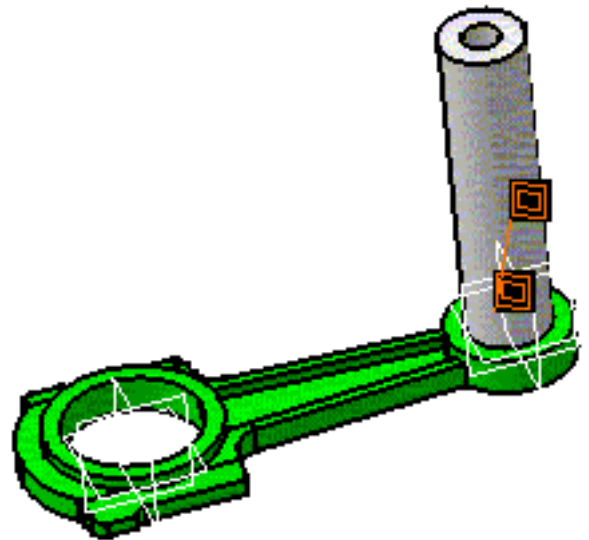


You must own the Generative Assembly Structural Analysis product to follow this scenario.

You can use the Rod\_For\_Analysis.CATPart and the Rod\_Axis\_For\_Analysis.CATPart documents from the SAMPLES/gps\_analysis directory for this task.



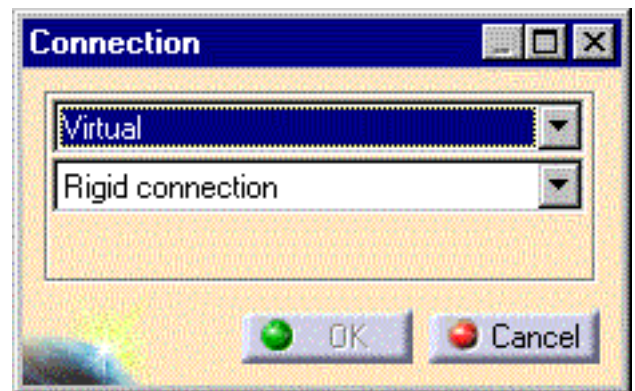
1. Repeat steps 1 to 11 of the scenario "Creating a Slider Connection" from the Getting Started section.



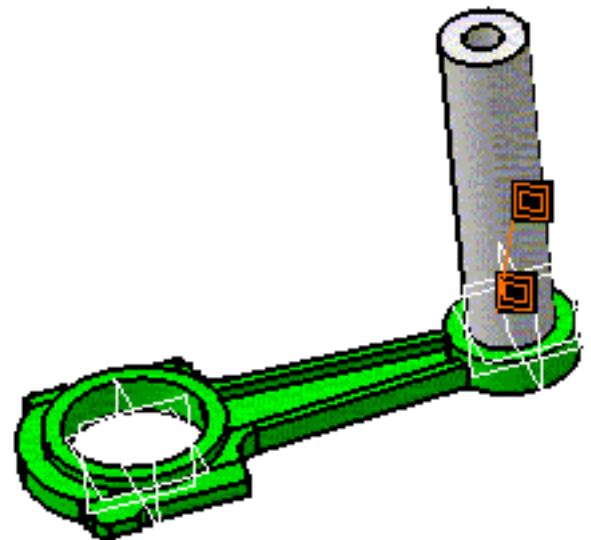
2. Select The "Parts Connection" icon from the Analysis ( Stress or Modal) workshop toolbar

-The Connection panel appears.

3. Select Virtual and Rigid or Smooth connection in the presented combos.



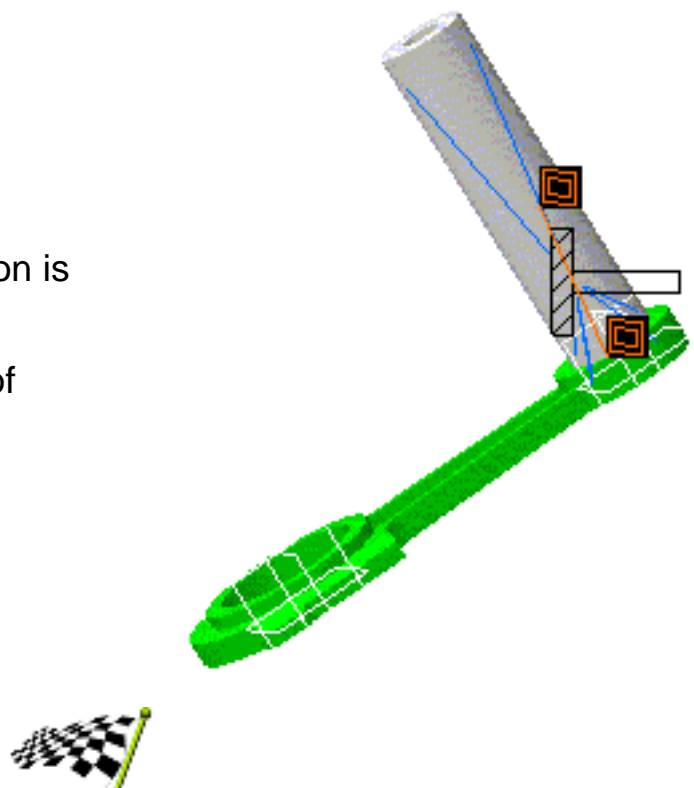
4. Select the Constraint symbol linking the two parts.



5. Click OK.

6. The Rigid or Smooth Connection is created.

A symbol representing the type of connection is displayed.





Up



Creating a Fastened



Creating a Contact Connecti



Creating a Virtual Connection

# Creating Restraints

*CATIA - Generative Part Structural Analysis* provides easy methods to create and edit restraints on any part geometries in both the *Stress Analysis* and *Dynamics Analysis* workbenches.

You can use these Restraint functionalities whenever you want, once you own a Part.

Restraints are required for Stress Analysis computations . They are optional for Dynamics Analysis computations (if not created, the program will compute vibration modes for the free, unrestrained part).

|   |   |   |
|---|---|---|
|  |  |  |
| <a href="#">Creating Clamps</a>   | <a href="#">Creating Sliders</a>  | <a href="#">Creating Virtual Restraints</a>   |



Be sure to fix all the global degrees of freedom of your assembly otherwise a global singularity will be raised at computation ( such a model is insolvable).To correct easily the model ( Stress Analysis Workbench only ), the induced motion of the assembly will be simulated and visualized after computation .



# Creating Clamps

Clamps are restraints applied to surface or line geometries of the part, for which all nodes are to be blocked in the subsequent analysis.



This task shows how to create a clamp on a geometry of a part.

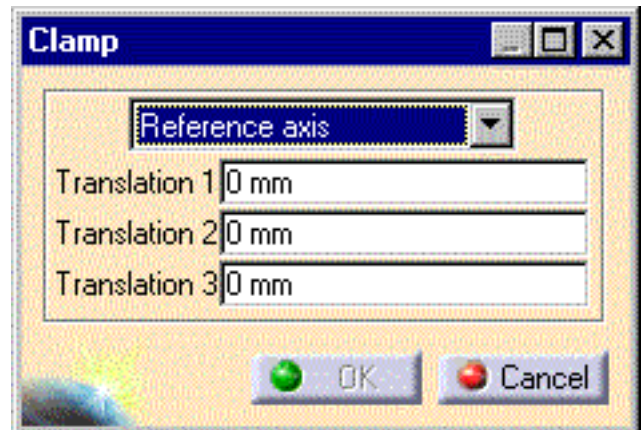


You can use the Rod\_For\_Analysis.CATPart document from the SAMPLES/gps\_analysis directory for this task.

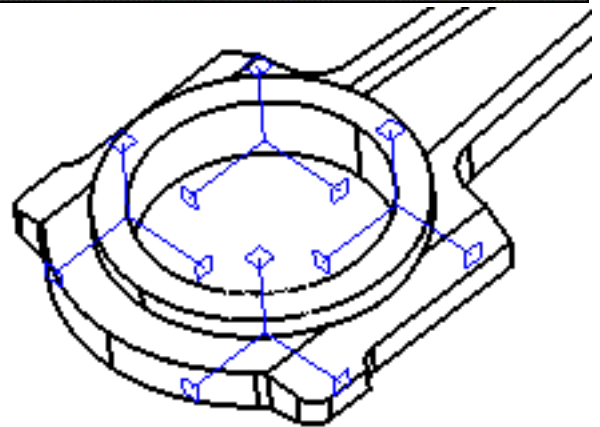


1. Click the Clamp icon .

The Clamp dialog box is displayed.



2. Select a surface ( or an edge ). (Any selectable geometry is highlighted when you pass the cursor over it: Edges can be selected with the reference axis option, all surfaces can be selected with the reference axis option. Spherical surfaces, surfaces of revolution, plane surfaces can be selected with the implicit axis option). Several symbols representing the fixed translation directions of the selected geometry are visualized. The blue vectors represent the restrained directions.



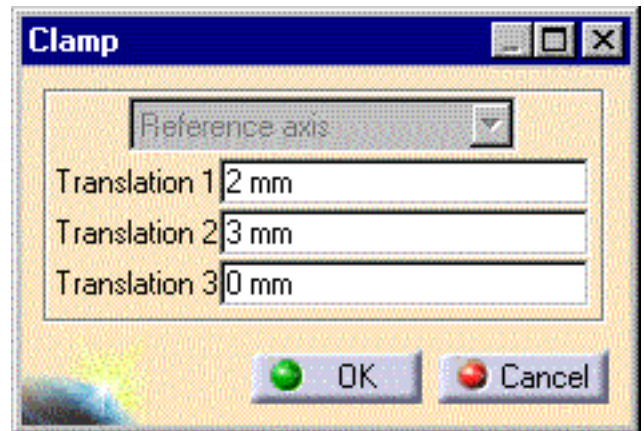
3. For non-zero values of the imposed translation components, you can choose between two coordinate systems:

- Reference Axis: values are relative to the global absolute reference frame.
- Implicit Axis: values are relative to an intrinsic reference frame attached to the geometry.

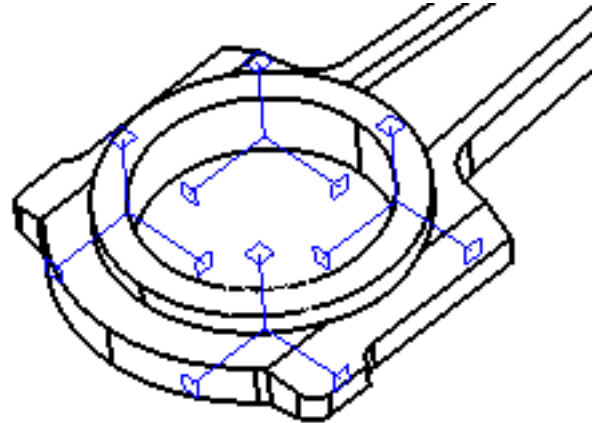
Click the combo box to activate the Implicit Axis option.



4. Set values for the imposed translation displacement components in mm.



5. Click OK to create the clamp.



If you select other surfaces you can create as many clamps as desired with the same dialog box. A series of clamps can therefore be created quickly.

You can either select the surface and then set the clamp value, or set the clamp values and then select the surface.



The arrow symbols orientation change depending on the chosen axis system (if non-coincident).



Up



Creating Clamps



Creating Virtual Restraints



Creating Sliders

# Creating Sliders

Sliders represent a generalization of the clamp restraint in the sense that you can release some of the clamped directions thus allowing the part *to slide* along the released translation directions.



This task shows how to create a slider on a geometry of a part.

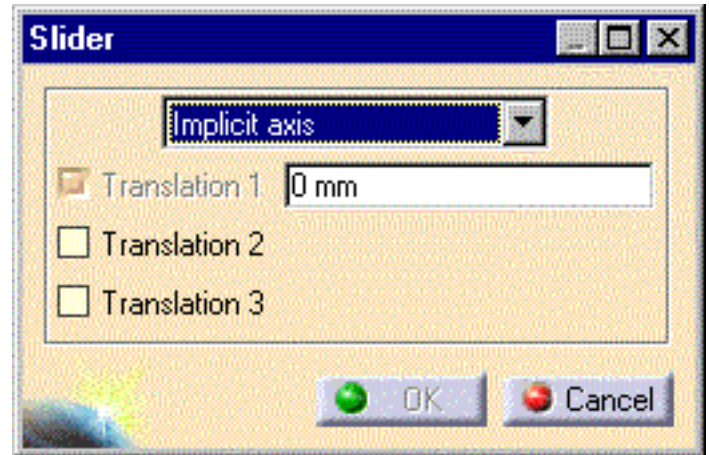


You can use the Rod\_For\_Analysis.CATPart document from the SAMPLES/gps\_analysis directory for this task.

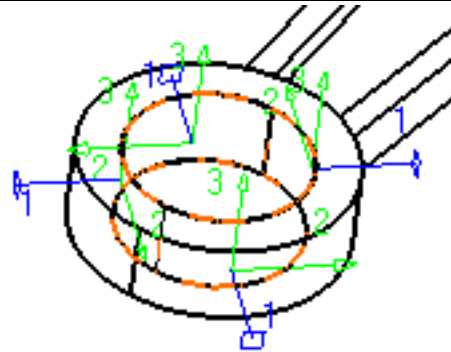


1. Click the Slider icon .

The Slider dialog box is displayed.



2. Select a boundary surface or edge. (Any selectable geometry is highlighted when you pass the cursor over it: Edges can be selected with the reference axis option, all surfaces can be selected with the reference axis option. Spherical surfaces, surfaces of revolution, plane surfaces can be selected with the implicit axis option).



- A few symbols representing the fixed and released translation directions of the selected geometry are visualized: Blue vectors represent fixed directions while green ones represent the free translation directions.

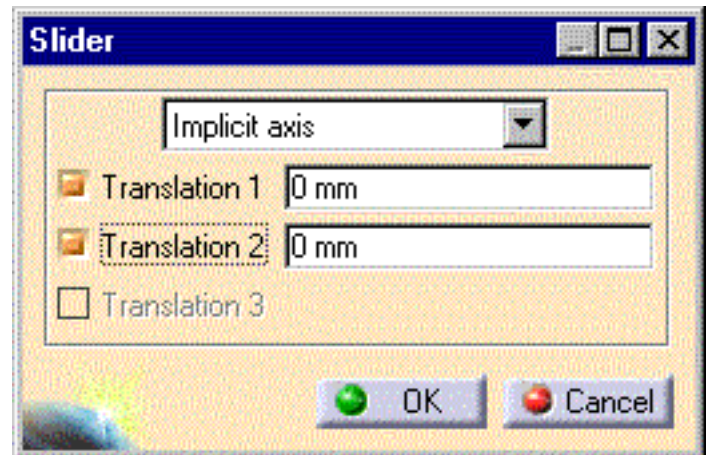
3. You can choose between two coordinate systems:

- Reference Axis: values are relative to the global absolute reference frame.
- Implicit Axis: values are relative to an intrinsic reference frame attached to the geometry

Click the combo box to activate the Implicit Axis option.

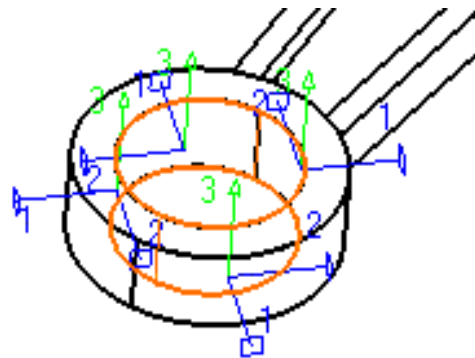
4. Release one or two translation directions by deactivating the corresponding checkbox.

5. Set values for imposing the remaining translation components in mm.



👉 If you select other surfaces or edges you can create as many sliders as desired with the same dialog box. A series of sliders can therefore be created quickly.

6. Click OK to create the slider.



👉 The arrow symbols orientation change depending on the chosen axis system (if non-coincident) and depending on the choice of released directions.



Up



Creating Clamps



Creating Virtual Restraints



Creating Sliders

# Creating Virtual Restraints

Virtual Restraints represent restraints applied indirectly to the part, through the action of a virtual rigid body.

Several cases are considered:

- the rigid body motion, defined via three rotation components and three translation components at a specific point of the rigid body, is transmitted using *rigid kinematic elements*: this has the effect of over-stiffening the part at its interface with the rigid body
- the rigid body motion is transmitted using *non-rigid kinematic elements*: this has the effect of leaving the interface free to deform
- the rigid body motion is transmitted by direct contact with the part, using *linear contact condition elements*: this has the effect of simulating the real distribution of imposed displacements over the true contact area
- The rigid body motion is transmitted using *elastic elements* this has the effect to create an elastic interface over the specified area between the part and the rigid body.

## Creating Rigidly or Smoothly Transmitted Virtual Restraints

When the virtual restraint transmission is *rigid*, the program uses RIG-BEAM kinematic elements to connect the virtual rigid body with the part at their common interface. This means imposing a set of constraints, with the effect of making the interface behave like a rigid-body: the motion will be transmitted to the part by clamping the interface to the virtual rigid body.

When the virtual restraint transmission is *smooth*, the program uses CONSTR-N kinematic elements to connect the virtual rigid body with the part at their common interface. This means imposing a set of constraints with the effect of allowing the interface to deform: the motion will be transmitted to the part without clamping the interface to the virtual rigid body.



This task shows how to create a rigidly transmitted virtual restraint on a geometry of a part.



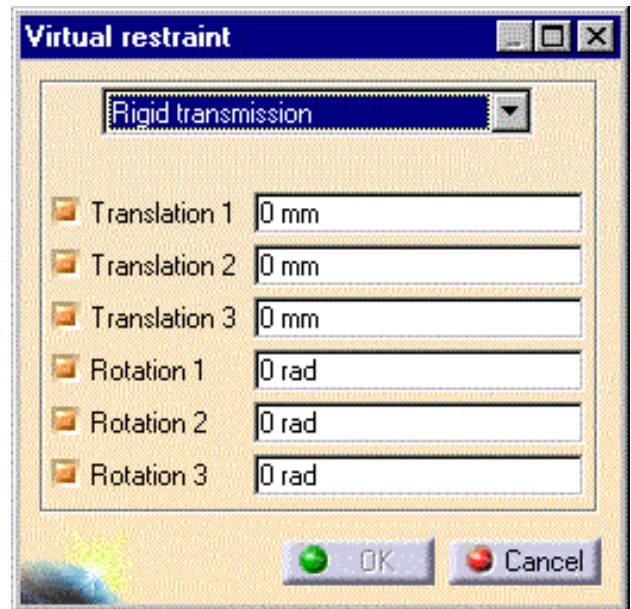
You can use the Rod\_For\_Analysis.CATPart document from the SAMPLES/gps\_analysis directory for this task.



1. Click the Virtual Restraint icon .

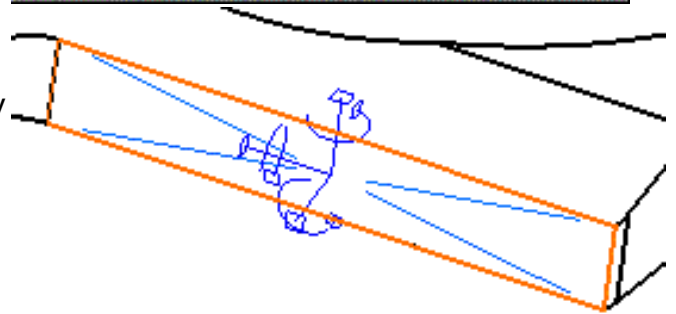


The Virtual Restraint dialog box is displayed.



2. Select a line or a surface.

A few symbols representing the virtual rigid body attached to the selected geometry are displayed. Several symbols representing the imposed translation and rotation directions at the default reference point of the virtual rigid body are also visualized.



3. Click the combo box to activate the Rigid Transmission option.

4. For non-zero rotation components, pick a reference point to locate the instantaneous axis of rotation belonging to the rigid extension of the virtual body. This new rotation axis system will correspond to the reference axis system translated on the selected point. ( To create such a point, you must use the Wireframe and Surface Design's workbench accessible through start -> Mechanical Design. )



This is optional if no rotation is considered (all components equal to zero).

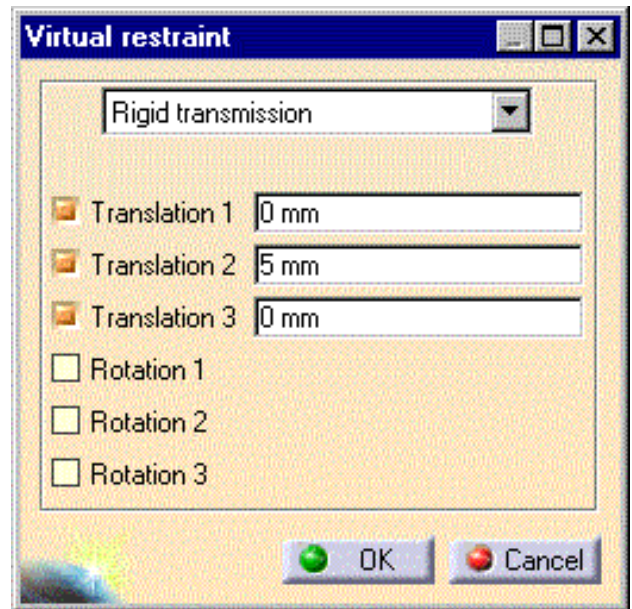
Deactivating the three Rotation switches release them.



If you do not define a reference point, the program will use the center of the selected geometry.

5. Set values for the imposed translation displacements (in mm) and rotation displacements (in radians). Uncheck the check-boxes to release the corresponding degrees of freedom.





6. Click OK to create the restraint.



## Creating Virtual Restraints Transmitted through Contact

When the virtual restraint is transmitted via contact, the program uses CONTACT elements to connect the virtual rigid body with the part at their *common interface*.

This means imposing a set of constraint conditions with the effect of imposing *non-penetration* between the part and the virtual rigid body.

As the Virtual Restraint behaves as a separate mesh, its global directions have to be fixed in order to avoid a singularity.



This task shows how to create a contact virtual restraint on a geometry of a part.

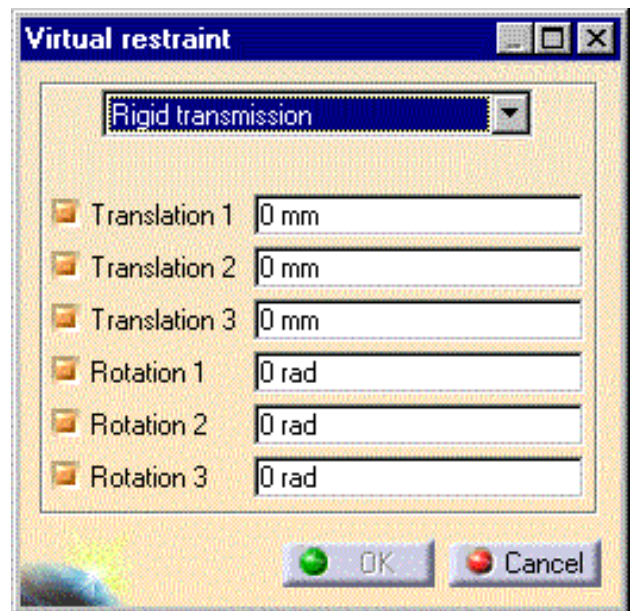


You can use the Rod\_For\_Analysis.CATPart document from the SAMPLES/gps\_analysis directory for this task.



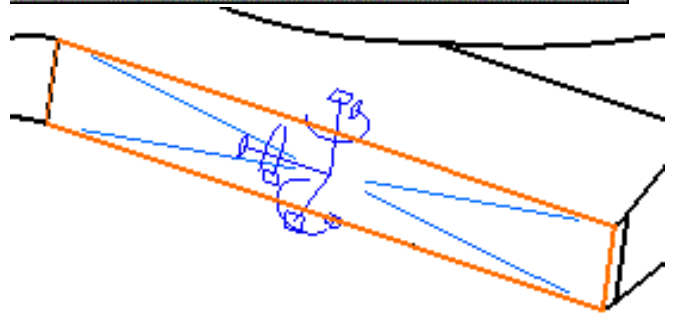
1. Click the Virtual Restraint icon .

The Virtual Restraint dialog box is displayed.



2. Select a surface.

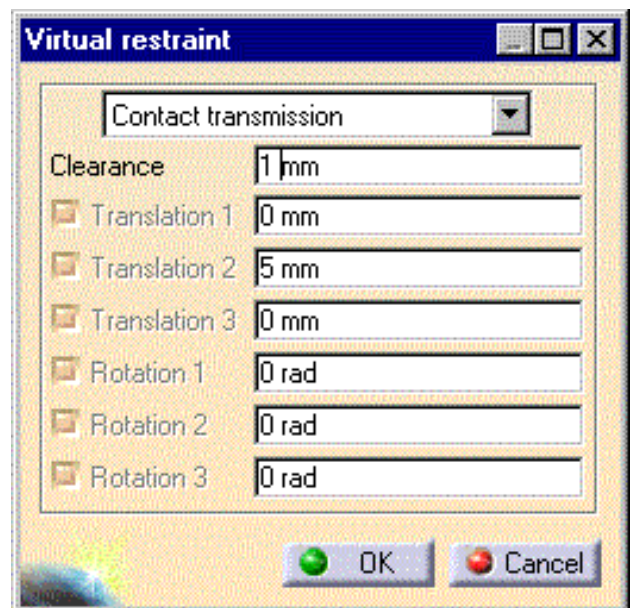
A few symbols representing the virtual rigid body attached to the selected geometry are displayed. Several symbols representing the imposed translations and rotations direction at the default reference point of the virtual rigid body are also visualized.



3. Click the combo box to activate the Contact option.

4. Pick a reference point belonging to the virtual rigid body and set the values of the imposed translation displacements in mm and rotation displacements in radians. ( To create such a point, you must use the Wireframe and Surface Design's workbench accessible through start -> Mechanical Design. )

5. Optionally enter a value for the clearance allowed at the interface.



If you do not define a reference point, the program will use the center of the selected geometry.

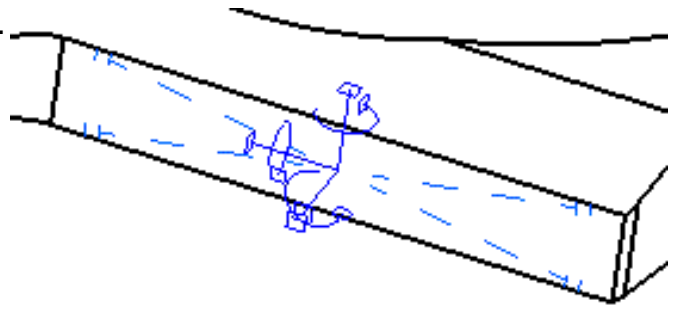


A positive clearance value represents an allowed approach distance between interfaces (prior to contact).

A negative clearance value represents a *press-fit* type condition (interfaces are already too close in the initial configuration). The imposed part deformation is directed to the inside, normal to the selected surface and its norm is equal to the absolute value of the clearance.



6. Click OK to create the contact virtual restraint.



## Creating Virtual Restraints Transmitted through a Spring

When the virtual restraint is transmitted via a spring, the program uses CONSTR-N and SPRING elements to connect the virtual rigid body with the part at their *common interface*.

This means imposing a set of constraint conditions with the effect of leaving the interface *free to deform* while undergoing the elasticity of the connection.

As the Virtual Restraint behaves as a separate mesh, its global directions have to be fixed in order to avoid a singularity.



This task shows how to create a spring virtual restraint on a geometry of a part.



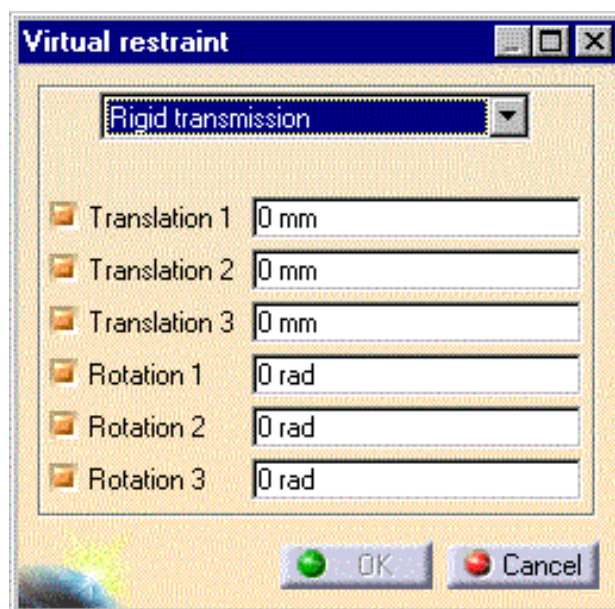
You can use the Rod\_For\_Analysis.CATPart document from the SAMPLES/gps\_analysis directory for this task.



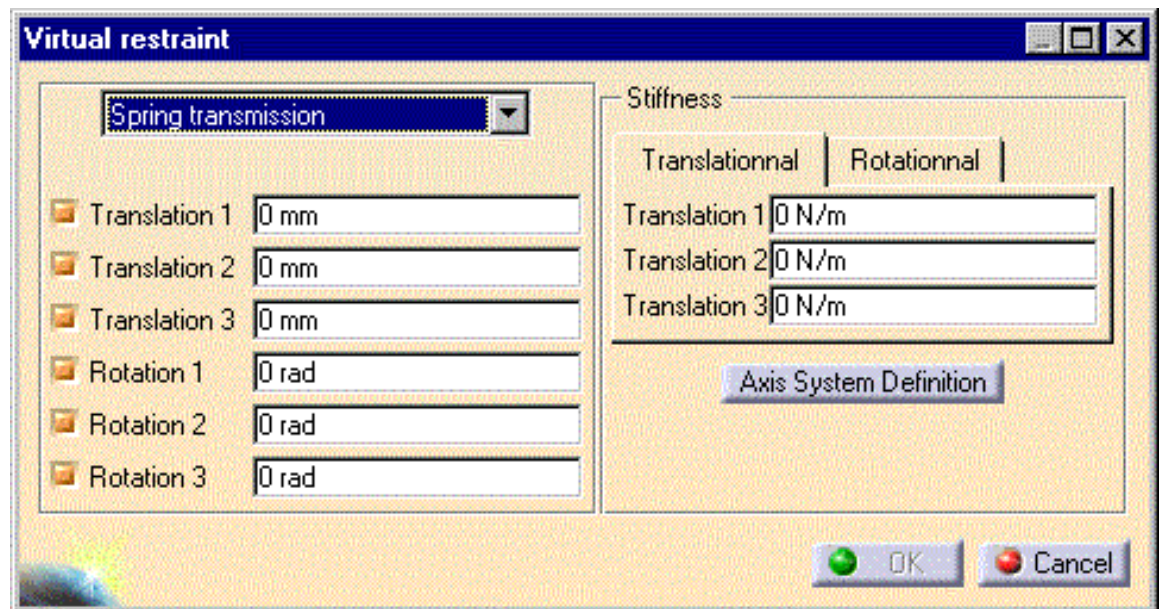
1. Click the Virtual Restraint icon



The Virtual Restraint dialog box is displayed.

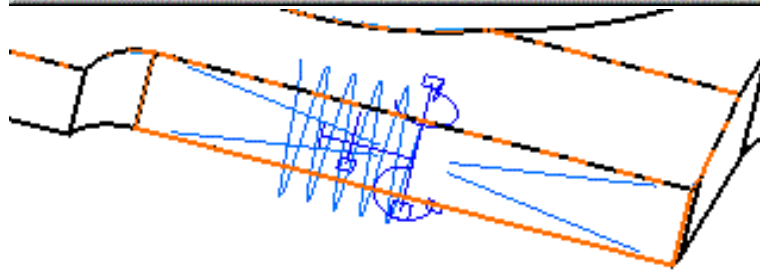


2. Click the combo box to activate the Spring Transmission option.

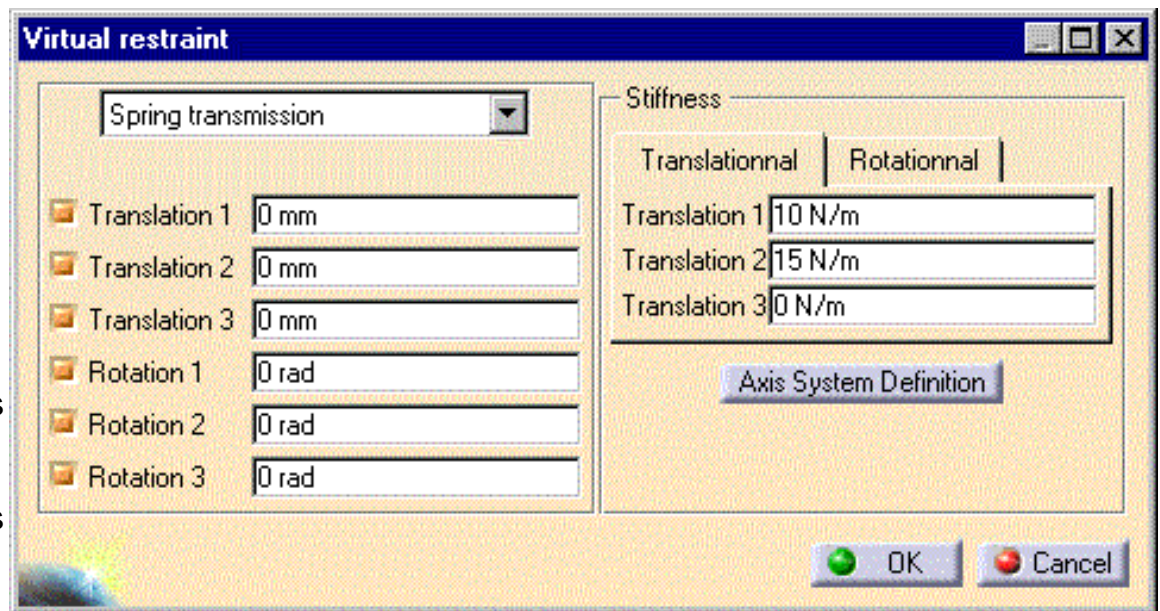


3. Select a line or a surface.

A few symbols representing the elastic connection attached to the selected geometry and the virtual rigid body is displayed. Several symbols representing the imposed translations and rotations direction at the default reference point of the virtual rigid body are also visualized.



4. Pick a reference point belonging to the virtual rigid body and set the values of the imposed translation displacements in mm and rotation displacements in rad. ( To create such a point, you must use the *Wireframe and Surface Design's* workbench accessible through start -> Mechanical Design. )



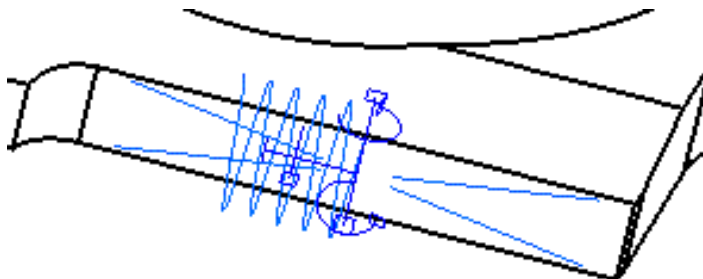
5. Set the values corresponding to the elasticity of the interface along each directions

6. Press on Axis System Definition to define the three directions of the elasticity: Select the first line or plane ( the directing vector or the normal is then selected), select the emptiness ( in the window ) to end the selection if you just use this first direction otherwise select another direction. The third is computed to build a direct axis system.



If you do not define a reference point, the program will use the center of the selected geometry.

7. Click OK to create the spring virtual restraint.





Up



Creating Clamps



Creating Sliders





Creating Virtual Restraints

# Creating Loads

CATIA - Generative Part Structural Analysis provides easy methods to create and edit loads on any part geometries.

Using the Stress Analysis workbench, you can create a load on a geometry or edit a selected load.

Using the Dynamics Analysis workbench, you can create a mass equipment on a geometry or edit a selected mass equipment.

|   |   |   |
|---|---|---|
|  |  |  |
| <a href="#">Creating Pressures</a>  | <a href="#">Creating Forces</a>   | <a href="#">Creating Moments</a>  |
|  |  |  |
| <a href="#">Creating Body Forces</a>  | <a href="#">Creating Rotation Forces</a>  | <a href="#">Creating Mass Equipment</a>   |



# Creating Pressures




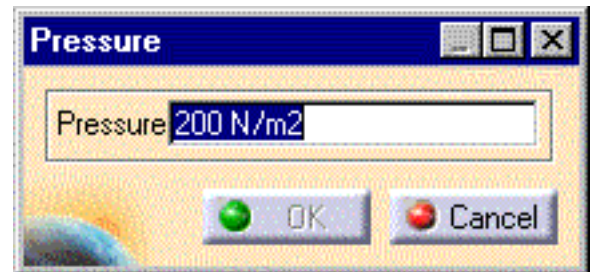
This task shows how to create a pressure applied normally (perpendicularly) to the geometry of a part.



You can use the Rod\_For\_Analysis.CATPart document from the SAMPLES/gps\_analysis directory for this task.

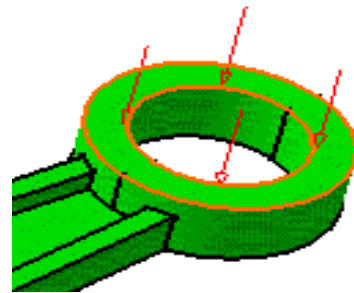


1. Click the Pressure icon .
2. Set the value of the pressure in Pascals (N/m<sup>2</sup>). A positive value describes a pressure whose resultant is directed towards the material side of the selected surface.



3. Select the surface on which you want to apply the load.

- Several arrows symbolizing the pressure are visualized.



You can either select the surface and then set the pressure value, or set the pressure value and then select the surface.



If you select other surfaces, you can create as many pressure loads as desired with the same dialog box. A series of pressures can therefore be created quickly.

4. Click OK to create the pressure.





# Creating Forces

These tasks allow you to create force-type loads, including tractions, distributed forces and forces transmitted through a virtual rigid body.

## Task

[Creating Traction](#)

[Creating Distributed Forces](#)

[Creating Transmitted Forces](#). This includes contact, rigid and smooth transmission types.

## Creating Traction

Traction represents intensive (surface density-type) quantities, as opposed to forces which are extensive (resultants, i.e., integrals over regions) quantities.

The user specifies three components for the direction of the load, along with a magnitude information which can be either a traction vector magnitude or a resultant force magnitude. In each case, the actual traction vector components and the remaining magnitude are updated based on the last data entry.

Therefore units are surface traction units for the traction vector entries (components and magnitude) and force units for the resultant force magnitude entry.



This task shows you how to create a traction on a geometry of a part.



The Functionality Traction ( not Force ) is available with a P1 or P2 licence.

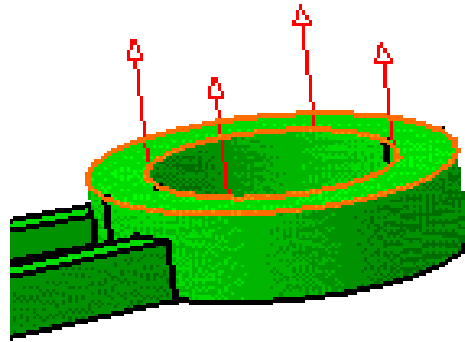
You can use the Rod\_For\_Analysis.CATPart document from the SAMPLES/gps\_analysis directory for this task.



1. Click the Force icon .



2. Select a geometry of the part (surface, curve, edge ) on which the traction is to be applied. (Any selectable geometry is highlighted when you pass the cursor over it.: Edges can be selected with the reference axis option only, all surfaces can be selected with the reference axis option. Spherical surfaces, surfaces of revolution, plane surfaces can be selected with the implicit axis option)



- Several arrows are displayed to visualize the distributed force vectors.



For a CATIA - P2 interface, you can select the compass to impose the direction of the arrows: Drag the compass by handling the red square and drop it on the appropriate surface. The normal (perpendicular) direction to this surface defines the new direction. Then, click on the Compass Direction button to take this new direction into account. You can now invert the direction if desired, editing the values of the three components. Please refer to the *CATIA Infrastructure User's Guide* for further information.

3. Select Traction in the first combo box.

4. Select the (reference or local) axis system in the second combo box.

5. Choose the group of values to be defined, among the following possibilities:

- traction norm
- force norm
- traction components.

6. Enter new values.



The remaining values are automatically evaluated by the program and displayed in the box.

The resulting arrow symbols are also updated.

7. Click OK.

- The traction is created.



## Creating Distributed Forces

Distributed forces correspond to the specification of a force resultant over a region (surface, line, point) of a part.

The program translates this information into the appropriate force distribution over the region. Therefore all units are force units (both for the resultant magnitude and the vector components).




This task shows you how to create a distributed force on a geometry of a part.

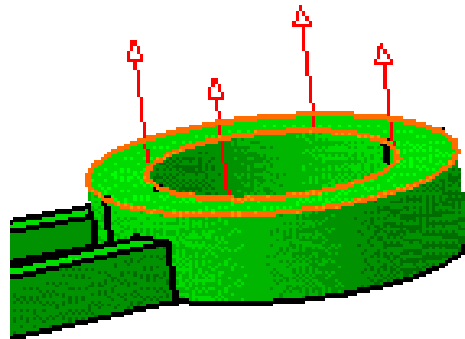


You can use the Rod\_For\_Analysis.CATPart document from the SAMPLES/gps\_analysis directory for this task.



1. Click the Force icon .
2. Select a geometry of the part (surface, curve, edge ).

- Several arrows are displayed to visualize the distributed Force vectors.



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

For a CATIA - P2 interface, you can select the compass to impose the direction of the arrows: Drag the compass by handling the red square and drop it on the appropriate surface. The normal (perpendicular) direction to this surface defines the new direction. Then, click on the Compass Direction button to take this new direction into account. You can now invert the direction if desired, editing the values of the three components. Please refer to the *CATIA Infrastructure User's Guide* for further information.

3. Select Force in the first combo box and Distributed in the second one.



 4. You can:

- either impose the three components of the force in the reference axis system (the program then automatically updates the resultant norm)
- or impose the norm (the program then automatically updates the components).

The resulting arrow symbols are also updated.

5. Click OK .

- The distributed force is created.



## Creating Transmitted Forces

Transmitted forces are forces applied indirectly, through the action of a *virtual rigid body*.

The user specifies the force acting on the virtual body and the program transfers it to the part accordingly.

Several cases are considered:

- the rigid body force, defined via its line of action, passing through a specific point of the rigid extension of the virtual body, is transmitted using *rigid kinematic elements*: this has the effect of over-stiffening the part at its interface with the rigid body.
- the rigid body force is transmitted using *non-rigid kinematic elements*: this has the effect of leaving the interface free to deform.
- the rigid body force is transmitted by direct contact with the part, using *linear contact condition elements*: this has the effect of simulating the real load distribution over the true contact area.

## Creating Rigidly or Smoothly Transmitted Forces

When the force is rigidly transmitted, the program uses RIG-BEAM kinematic elements to connect the virtual rigid body with the part at their common interface. This means imposing a set of loads, with the effect of making the interface behave like a rigid body (the load will be transmitted to the part by *clamping* the interface to the virtual rigid body). The interface of the part undergoing this rigid transmission is then over-Stiffened.

When the force is smoothly transmitted, the program uses CONSTR-N kinematic elements to connect the virtual rigid body with the part at their common interface. This means imposing a set of loads, with the effect of allowing the interface to deform (the load will be transmitted to the part *without clamping* the interface to the virtual rigid body).



This task shows you how to create a rigidly transmitted force on a geometry of a part.



You can use the Rod\_For\_Analysis.CATPart document from the SAMPLES/gps\_analysis directory for this task.

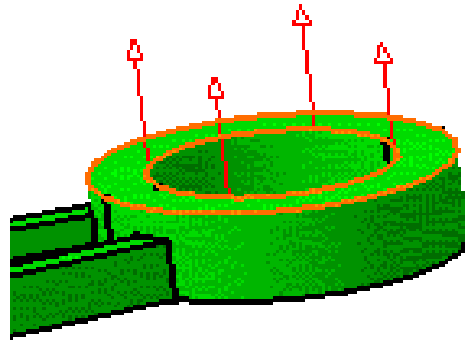


1. Click the Force icon .

2. Select a geometry of the part (surface, curve, edge ).

- A few arrow are displayed to visualize the resulting Force vector.

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



3. Select Force in the first combo box and Rigid Transmission in the second one.



4. You can:

- either impose the three components of the force in the reference axis system (the program then automatically updates the resultant norm)
- or impose the norm (the program then automatically updates the components).



The resulting arrow symbols are also updated.

For a CATIA - P2 interface, you can select the compass to impose the direction of the arrows: Drag the compass by handling the red square and drop it on the appropriate surface. The normal (perpendicular) direction to this surface defines the new direction. Then, click on the Compass Direction button to take this new direction into account. You can now invert the direction if desired, editing the values of the three components. Please refer to the *CATIA Infrastructure User's Guide* for further information.

5. Select the application point among the available geometries to locate the line of action of the force. ( To create such a point, you must use the *Wireframe and Surface Design* workbench accessible through start -> Mechanical Design. ).



If you do not define the force application point, the program will use the point at the center of the selected geometry.

6. Click OK.

- The rigidly transmitted force is created.



# Creating Forces Transmitted Through Contact

When the force is transmitted via contact, the program uses CONTACT elements to connect the virtual rigid body with the part at their common interface.

This means imposing a set of constraint conditions, with the effect of imposing non-penetration between the part and the virtual rigid body. The Contact Transmission capability is also useful to define a load resulting from pressure fitting.



This task shows you how to create a Force Transmitted through Contact on a geometry of a part.



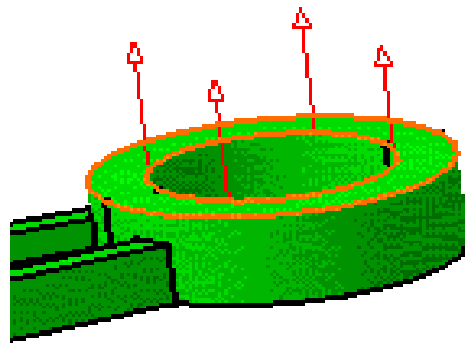
You can use the Rod\_For\_Analysis.CATPart document from the SAMPLES/gps\_analysis directory for this task.



1. Click the Force icon .

2. Select a surface of the part. This surface must be a surface of revolution or spherical or plane.

- A few arrows are displayed to visualize the resulting force vector.



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



3. Select Force in the first combo box, and Contact in the second one.



4. Define the Clearance between the selected part geometry and the virtual body:

- a positive value defines a space between the virtual body and the part before deformation. If the relative approach of the regarding interfaces tends to exceed the clearance, the load previously defined is applied.
- a negative value defines a contact state (contact occurs if the clearance is null or negative). Therefore the load is applied on the selected geometry until its deformation creates a positive clearance.



5. You can:

- either impose the three components of the force in the reference axis system (the program then automatically updates the resultant norm)
- either impose the norm (the program then automatically updates the components).

The resulting arrow symbols are also updated.

For a CATIA - P2 interface, you can select the compass to impose the direction of the arrows: Drag the compass by handling the red square and drop it on the appropriate surface. The normal (perpendicular) direction to this surface defines the new direction. Then, click on the Compass Direction button to take this new direction into account. You can now invert the direction if desired, editing the values of the three components. Please refer to the *CATIA Infrastructure User's Guide* for further information.



The application point is located at the center of the virtual body which is a sphere or a volume of revolution or a limited plane accordingly to the previously selected surface.

6. Click OK.

- The force transmitted through contact is created.





Up



Creating Moments



Creating Pressures



Creating Body Forces



Creating Mass Equipment



Creating Forces



Creating Rotation Forces

# Creating Moments

This task allows you to create transmitted moment-type loads, which include rigid and smooth transmission types.

The two following cases are considered:

- the rigid body moment is transmitted using *rigid kinematic elements*: this has the effect of over-stiffening the part at its interface with the rigid body
- the rigid body moment is transmitted using *non-rigid kinematic elements*: this has the effect of allowing the interface to deform freely

## Creating Rigidly or Smoothly Transmitted Moments

When the moment is rigidly transmitted, the program uses RIG-BEAM kinematic elements to connect the virtual rigid body with the part at their common interface. This means imposing a set of loads, with the effect of making the interface behave like a rigid body (the load will be transmitted to the part by *clamping* the interface to the virtual rigid body). The interface of the part undergoing this rigid transmission is then over-stiffened.

When the moment is smoothly transmitted, the program uses CONSTR-N kinematic elements to connect the virtual rigid body with the part at their common interface. This means imposing a set of loads, with the effect of allowing the interface to deform (the load will be transmitted to the part *without clamping* the interface to the virtual rigid body)



This task shows you how to create a smoothly transmitted moment on a geometry of a part.



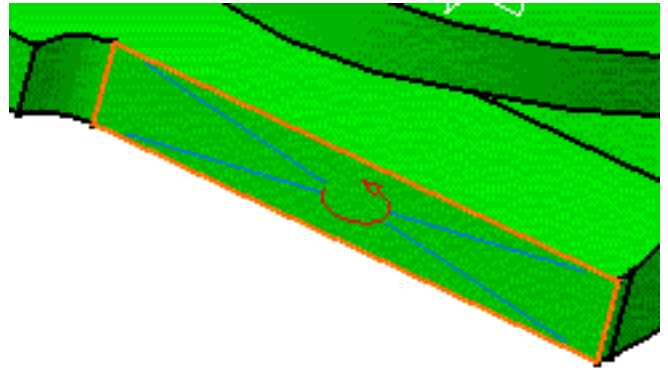
You can use the Rod\_For\_Analysis.CATPart document from the SAMPLES/gps\_analysis directory for this task.



1. Click the Moment icon .

2. Select a geometry of the part (surface, curve, edge ).

- A moment symbol is displayed to visualize the resulting Moment vector.

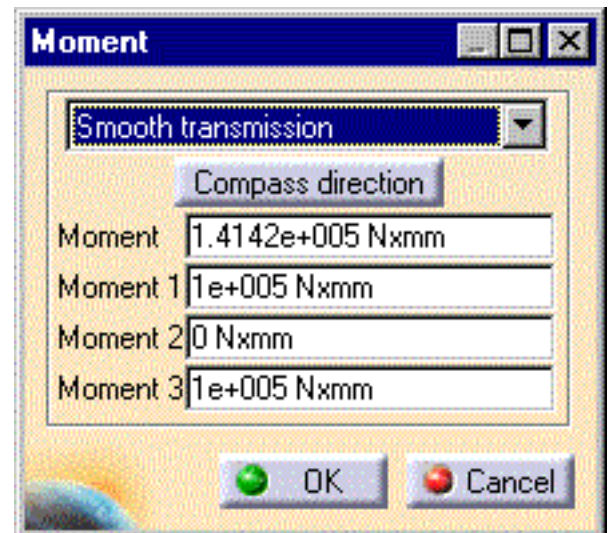


- Any selectable geometry is highlighted when you pass the cursor over it.
- You can select a line in the geometry of the part to define easily another direction. ( To create such a line, you must use the *Wireframe and Surface Design*'s workbench accessible through start -> Mechanical Design. ).

3. Select Smooth Transmission in the combo box.

4. You can:

- either impose the three components of the moment in the reference axis system (the program then automatically updates the moment norm)
- or impose the moment norm (the program then automatically updates the components).



The resulting moment symbol is also updated.



For a CATIA - P2 interface, you can select the compass to impose the direction of the arrows: Drag the compass by handling the red square and drop it on the appropriate surface. The normal (perpendicular) direction to this surface defines the new direction. Then, click on the Compass Direction button to take this new direction into account. You can now invert the direction if desired, editing the values of the three components.

5. Click OK.

- The moment is created.





Up



Creating Moments



Creating Pressures



Creating Body Forces



Creating Mass Equipment



Creating Forces



Creating Rotation Forces

# Creating Body Forces

This task allows you to create body force-type loads, including volume and mass body forces.

## Creating Volume Body Forces

Body forces represent intensive (volume density-type) quantities, as opposed to forces which are extensive (resultants, i.e., integrals over regions) quantities.

The user specifies three components for the direction of the load, along with a magnitude information which can be either a force vector magnitude or a resultant force magnitude. In each case, the actual Body Force vector components and the remaining magnitude are updated based on the last data entry.

Therefore units are volume body force units for the Body Force vector entries (components and magnitude) and force units for the resultant force magnitude entry.

This task shows you how to create a body force on a geometry of a part.



You can use the Rod\_For\_Analysis.CATPart document from the SAMPLES/gps\_analysis directory for this task.



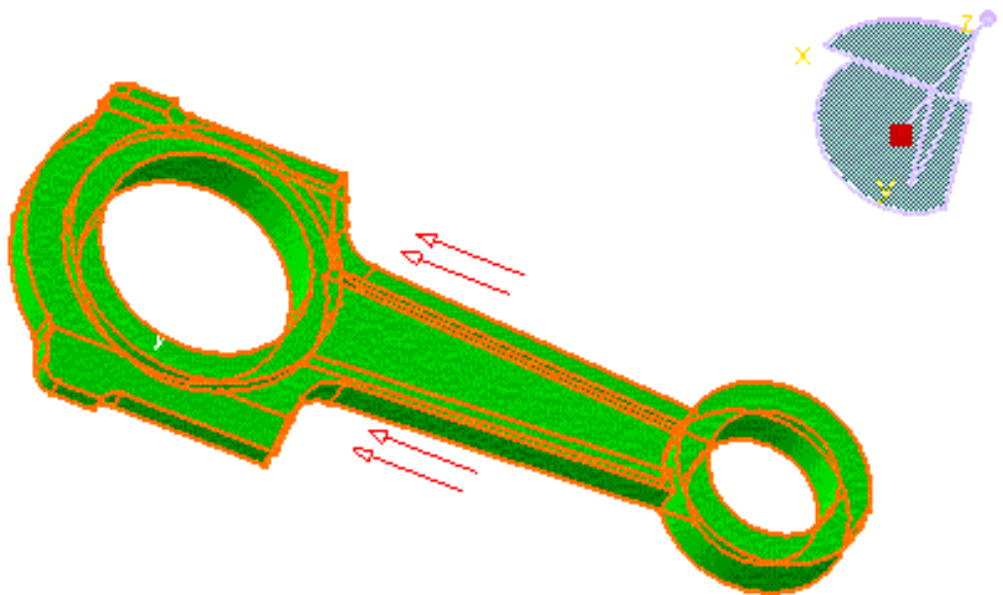
1. Click the Body



Force icon .

2. Select a geometry of the part (volume).

- Several arrows are displayed to visualize the distributed Body Force vector.





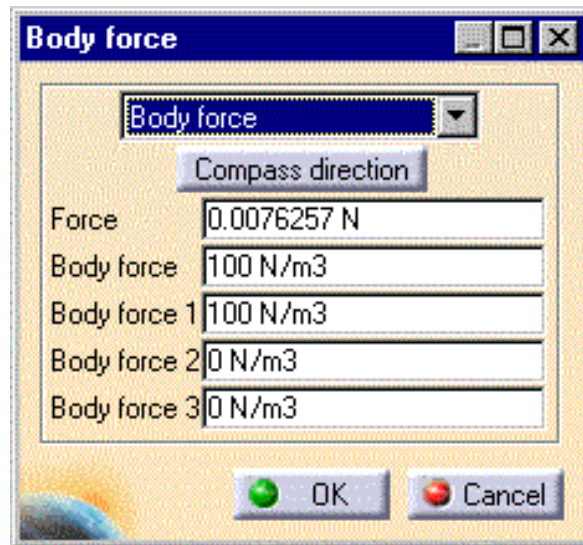
Any selectable geometry is highlighted when you pass the cursor over it.

For a CATIA - P2 interface, you can select the compass to impose the direction of the arrows: Drag the compass by handling the red square and drop it on the appropriate surface. The normal (perpendicular) direction to this surface defines the new direction. Then, click on the Compass Direction button to take this new direction into account. You can now invert the direction if desired, editing the values of the three components.

3. Select Body Force in the combo box.

4. Choose the group of values to be defined, among the following possibilities:

- body force norm and direction
- body force components.



5. Click OK or Apply.

- The volume body force is created.



## Creating Gravity Forces (Mass Body Forces)

Gravity forces correspond to the specification of a mass force density exerted throughout the part.

The user specifies three components for the direction of the load, along with a magnitude information. In each case, the actual mass body force vector components or the remaining magnitude are updated based on the last data entry.

Therefore units are acceleration units for the mass body force vector entries (components and magnitude).



This task shows you how to create a mass body force on a volume of a part.



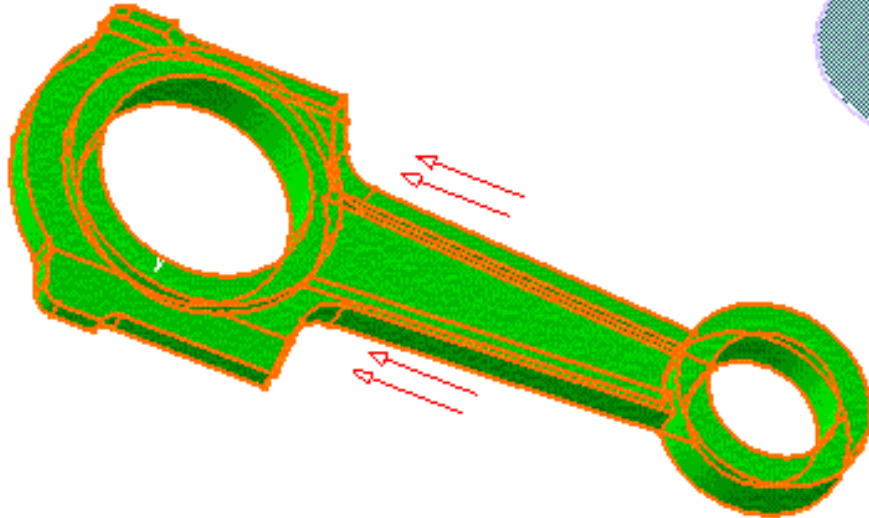
You can use the Rod\_For\_Analysis.CATPart document from the SAMPLES/gps\_analysis directory for this task.

1. Click the Body

Force icon

2. Select a geometry of the part (volume).

- Several arrows are displayed to visualize the distributed Body Force vector.



- Any selectable geometry is highlighted when you pass the cursor over it.
- For a CATIA - P2 interface you can select the compass to define easily another direction : Drag the compass by handling the red square and drop it on the appropriate surface. The normal (perpendicular) direction to this surface defines the new direction. Then, click on the Compass Direction button to take this new direction into account. You can now invert the direction if desired, editing the values of the three components.

3. Select Gravity in the combo box.

4. You can:

- either impose the three components of the acceleration in the reference axis system (the program then automatically updates the resultant norm)





- or impose the norm (the program then automatically updates the components).

The resulting arrow symbols are also updated.

5. Click OK or Apply.

- The mass body force is created.



# Creating Rotation Forces

This task allows you to create centrifugal loads due to the radial acceleration of a body undergoing uniform rotation. The inertia load is of acceleration type (mass body force). The acceleration field distribution is automatically evaluated by the program from user-supplied information about the angular velocity and acceleration magnitudes and the rotation axis.




This task shows you how to create a rotation force on a part.

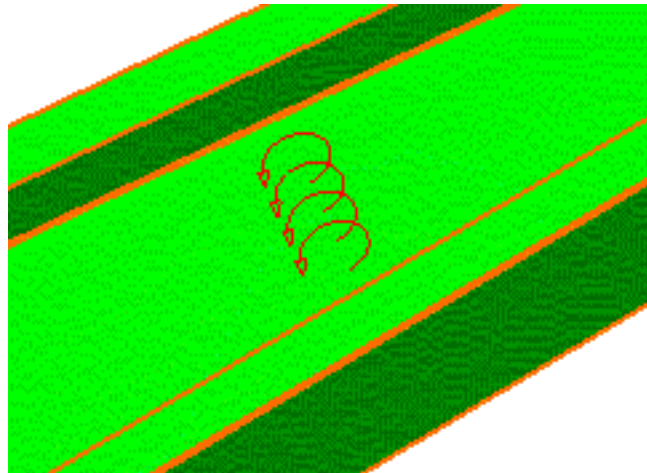


You can use the Rod\_For\_Analysis.CATPart document from the SAMPLES/gps\_analysis directory for this task.



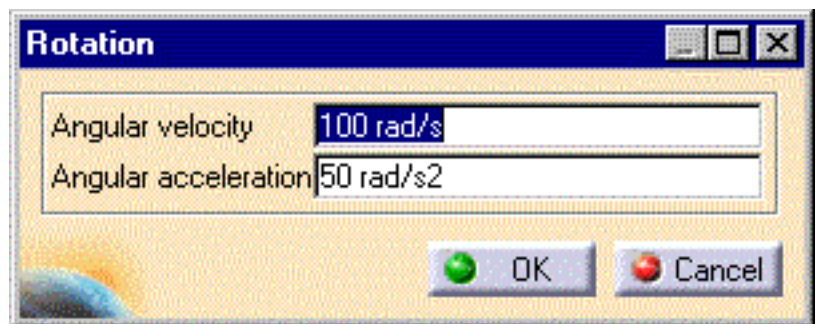
1. Click the Rotation icon .
2. Select the part on which the load is to be applied.

- Several symbols are displayed to visualize the distributed Force vectors.



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

3. Define the Angular Velocity and the Angular Acceleration magnitudes..



4. Select the rotation axis by selecting a line.. ( To create such a line, you must use the *Wireframe and Surface Design's* workbench accessible through start -> Mechanical Design. ).

- The program proposes a direction of rotation by displaying a few rotation symbols.



If you do not define the rotation axis, the program will choose one arbitrarily.

5. Click OK or Apply.

- The rotation is created.



# Creating Mass Equipment

Mass Equipment are additional masses attached to the geometry (point, line or surface) of the part. They represent scalar, purely inertial (non-structural) loads.

Mass Equipment is either distributed (mass distributed evenly on a selected part geometry) or lumped (massive bodies attached to a part geometry).

Mass Equipment can greatly affect the vibration characteristics of the part (dynamic mode shapes and frequencies) but has no effect on the stress results. Masses are generally negligible compared to loads. However, loads due to masses can be easily statically modeled using the Stress Analysis workbench (please refer to [Creating Forces chapter](#)).

Therefore, Mass Equipment is only available in the Dynamics Analysis workbench.

## Creating Distributed Mass Equipment

Geometric elements of a part can support an additional mass distributed on them (paint coating as an example).

Distributed mass equipment are directly applied to part geometries. The specification can be made either intensively, via the notion of mass density, or extensively via the total mass.

## Creating Distributed Mass Equipment via Density

The Mass Equipment can be known through its density and based on a surface, a line or a curve, a point. Therefore the units employed here are mass units as well as mass line and surface density units.

The proposed modelization distributes uniformly the Mass Equipment on the selected region of the part.

This Mass Equipment edition mode allows you to easily create and delete additional masses.



This task shows you how to create a Distributed Mass Equipment on a geometry of a part through its density.



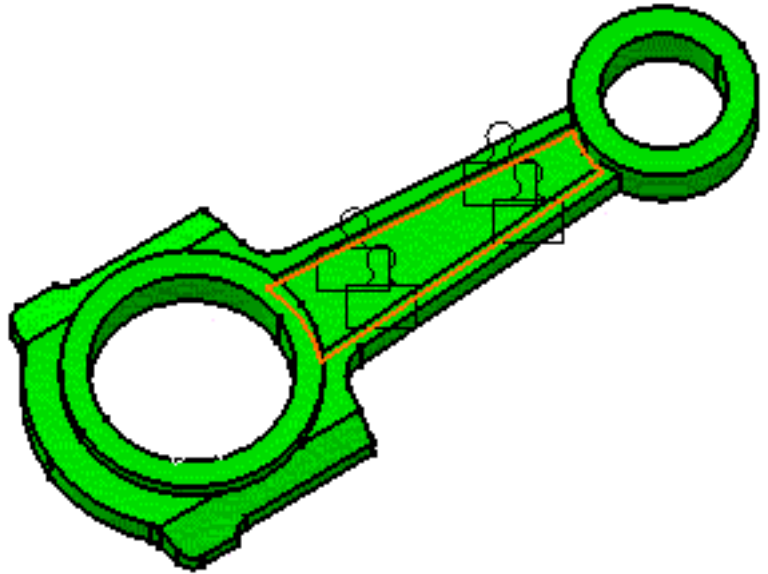
You can use the Rod\_For\_Analysis.CATPart document from the SAMPLES/gps\_analysis directory for this task.



1. Click the Equipment icon 

2. Select a geometry of the part (surface, curve, edge) on which the load is to be applied (as long as you have not clicked OK, you can select another geometry of the part to create a new equipment).

- Several Mass symbols are displayed on the selected region of the part in anticipation of the Equipment location.



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

The unit of density is updated to fit with the selected geometric part (for example, Kg/m<sup>2</sup> for a surface).

3. Click the combo box to select the Density option. Define the value of the mass density or the value of the total mass.

- The remaining magnitude is directly updated.



4. Click OK.

- The distributed mass equipment is created.



## Creating Distributed Mass Equipment via Magnitude

Distributed Mass Equipment can be defined by their resultant magnitude (total mass) and the region on which it is based.

The proposed modelization distributes uniformly the Mass Equipment on the selected region of the part (surface, curve, point and so forth).



This task shows you how to create a Distributed Mass Equipment on a geometry of a part.



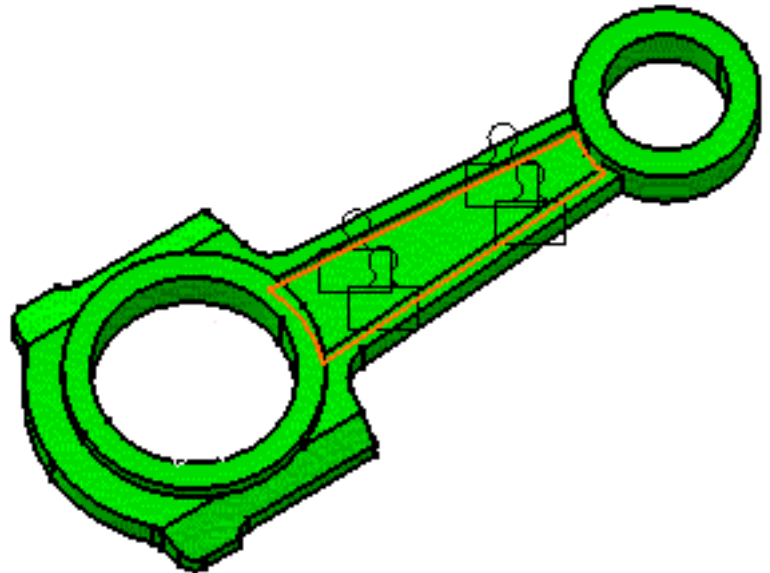
You can use the Rod\_For\_Analysis.CATPart document from the SAMPLES/gps\_analysis directory for this task.



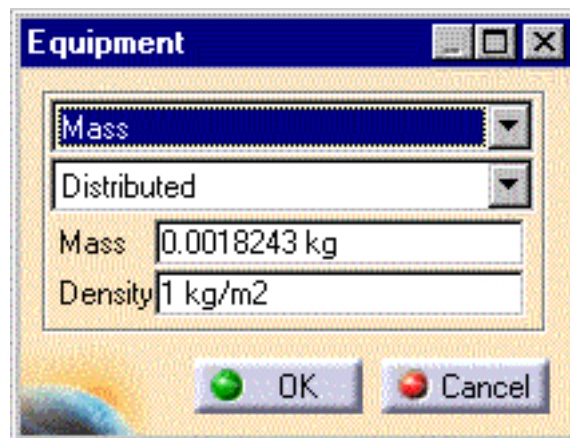
1. Click the Equipment icon 

2. Select a geometry of the part (surface, curve, edge ) on which the Distributed Mass Equipment is to be based.

- Several Mass symbols are displayed to symbolize the future Distributed Mass Equipment location.
- Any selectable geometry is highlighted when you pass the cursor over it.



3. Select Mass in the first combo box and Distributed in the second one.



4. Define the Distributed Mass Equipment resultant magnitude.

5. Click OK or Apply.

- The distributed mass equipment is created.



# Creating Lumped Mass Equipment

Mass Equipment can also represent massive objects, conceptualized as virtual rigid bodies (an engine block for instance). Their action is then transmitted to the part, according to the logic of load transmission.

The two kinds of virtual mass equipment are rigidly and smoothly transmitted equipment.

## Creating Rigidly or Smoothly Transmitted Lumped Mass Equipment

When a lumped mass equipment is rigidly transmitted, the program uses RIG-BEAM kinematic elements to connect the virtual rigid body with the part at their common interface. This means imposing a set of constraints, with the effect of making the interface behave like a rigid body (the inertia load will be transmitted to the part by *clamping* the interface to the virtual rigid body). The interface of the part undergoing this rigid transmission is then over-stiffened.

When the lumped mass is smoothly transmitted, the program uses CONSTR-N kinematic elements to connect the virtual rigid body with the part at their common interface. This means imposing a set of constraints, with the effect of allowing the interface to deform (the load will be transmitted to the part *without clamping* the interface to the virtual rigid body). The interface of the part undergoing this rigid transmission is allowed to deform freely.



This task shows you how to create a rigidly transmitted lumped Mass Equipment on a geometry of a part.



You can use the Rod\_For\_Analysis.CATPart document from the SAMPLES/gps\_analysis directory for this task.



1. Click the Equipment

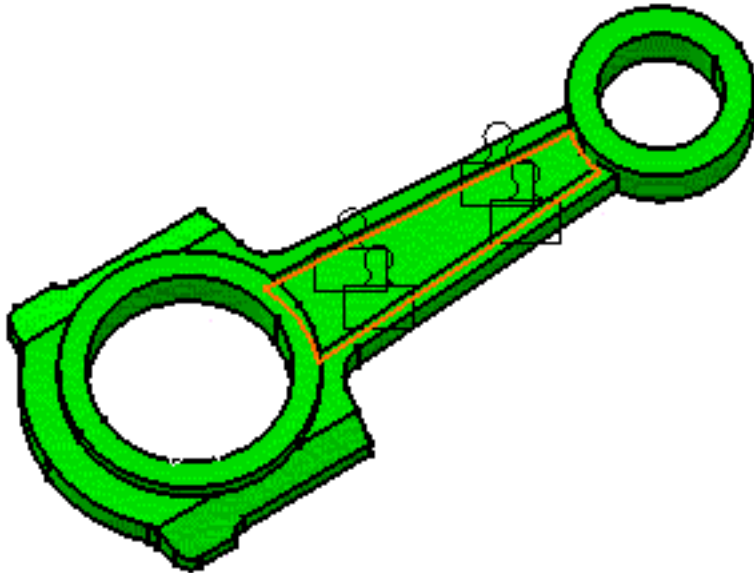


icon .

2. Select a geometry of the part (surface, curve, edge ) with which the Equipment is to be linked.

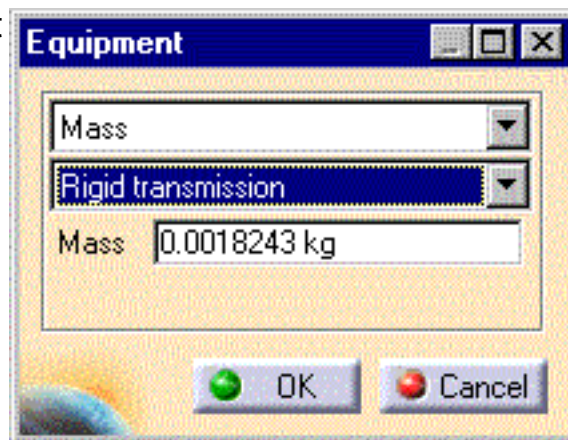


- Several Mass symbols are displayed to symbolize the future Distributed Mass Equipment location.

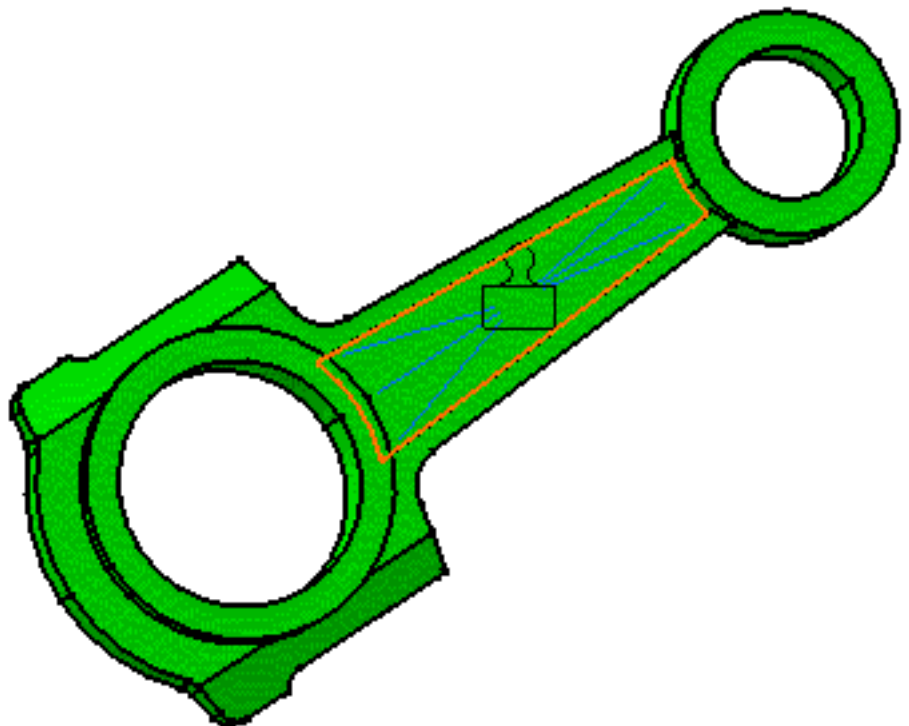


Any selectable geometry is highlighted when you pass the cursor over it.

3. Select Mass in the first combo box and Rigid Transmission in the second one.



4. Select the Mass Equipment location point (this point must not belong to the geometry elements of the part). This point must be created by the means of the *Wireframe and Surface Design* Workbench accessible through Start -> Mechanical Design.



5. Define the Mass  
Equipment magnitude.

6. Click OK or Apply.

- The lumped mass equipment is created.



# Computing Results

CATIA - Generative Part Structural Analysis provides easy computation tools for performing stress analysis and dynamics analysis on parts.

- Using the Stress Analysis workbench, you can compute stress results on a deformed part resulting from the application of restraints and loads
- Using the Dynamics Analysis workbench, you can compute dynamic vibration modes for a part optionally subjected to specified restraints and non-structural masses.
- If you own the CATIA - Generative Assembly Structural Analysis product, you can perform Stress and Dynamic computations on assemblies.

The computations are based on the finite element method (FEM) which requires:

- part discretization (mesh generation)
- the satisfaction of equilibrium conditions (solution computation).

For both static and dynamic computations, you can specify the kind of required accuracy (linear or parabolic formulation) and the size and sag of the mesh used to perform the computation (local and global).

Remember that a smaller (refined) mesh means increased results accuracy.

Two kinds of computation methods are proposed to the user :

- The Direct method,
- The Iterative method.

Both methods obtain the same precision on the results, the difference is based on the computation duration and the memory space management. In one hand the iterative method is the best choice if there is no external storage specified, in another hand, if there is an external storage, it is the best method for very large model computation. The other choice : the direct method is appropriate to compute small or medium models ( <200 000 degrees of freedom ) with an external storage. If you do not know what to choose you can use an automatic mode which will make the decision for you, taking into account the size of your model and the available memory space. You can as well ask how long it will take for each method to compute.



[Computing and Visualizing a High Precision Mesh](#)

[Computing a Rough Stress Analysis](#)

|   |
|---|
| <a href="#">Computing a High Precision Stress Analysis</a>              |
| <a href="#">Refining and Computing a High Precision Stress Analysis</a> |
| <a href="#">Computing a Normal Modes Analysis</a>                       |
| <a href="#">Computing an Assembly Model</a>                             |



# Computing a High Precision Stress Analysis

A Stress Analysis computation corresponds to a sequence of operations including mesh generation, matrix assembly and system resolution for degrees of freedom.

For a High Precision mesh, the element order is parabolic, corresponding to 10-noded elements.

Specifying relatively normal global sizes and sags, and little local conditions, will have the effect of generating a High precision mesh and a large size problem (many unknowns). Such computations serve as accurate calculus approximations following rough approximations.

This task shows how to generate a high precision stress analysis on a part.



You must have defined all physical attributes (material, restraint, and load) prior to this task.

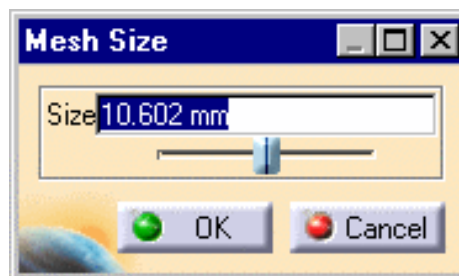


You can use the Rod\_For\_Analysis.CATPart document from the SAMPLES/gps\_analysis directory for this task.

1. Click the Mesh Size icon



The Mesh Size dialog box appears:



2. Set a little value for the local Mesh Size.

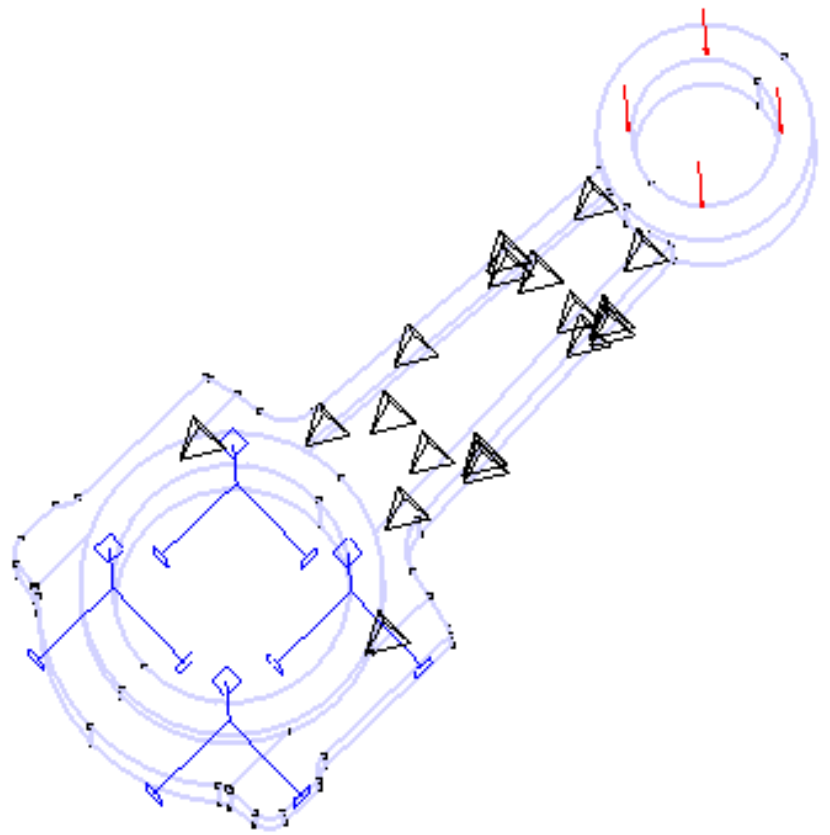
3. Select appropriate surfaces (or lines, curves) on which you want to apply those local conditions.

4. Click Ok to end this local conditions application.



5. Repeat steps 1 to 4 to apply local Mesh Sizes with a different value if necessary.

- Those local conditions if applied where the stresses are concentrated, will increase efficiently (that is to say with maximum accuracy improvement and minimum calculus time loss) and locally the accuracy of the computation.

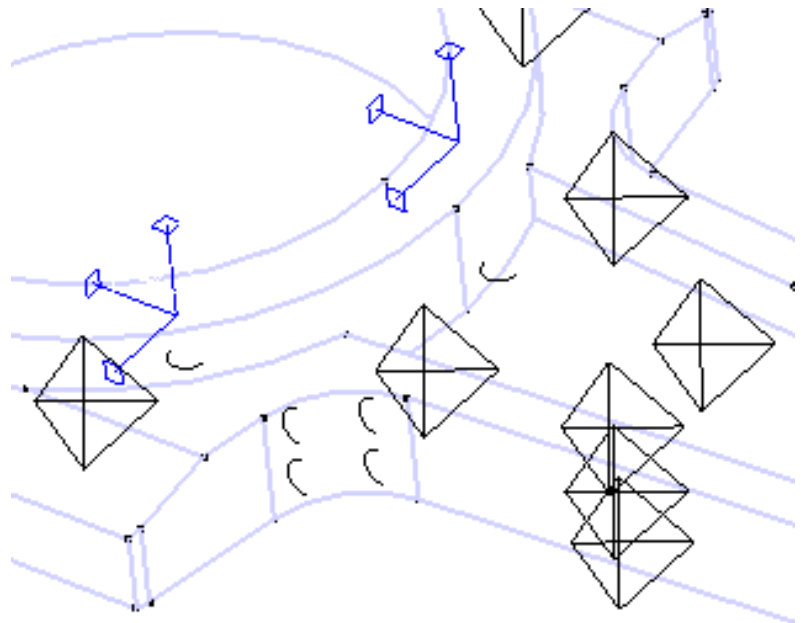


6. Repeat Step 1 to 5 with

the Sag icon . This icon is located in the same icon box than the Mesh Size

icon .

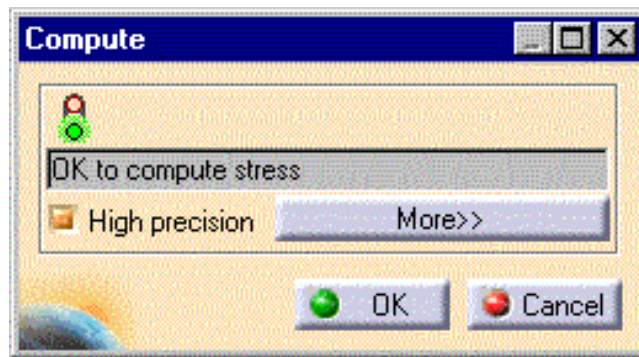
new



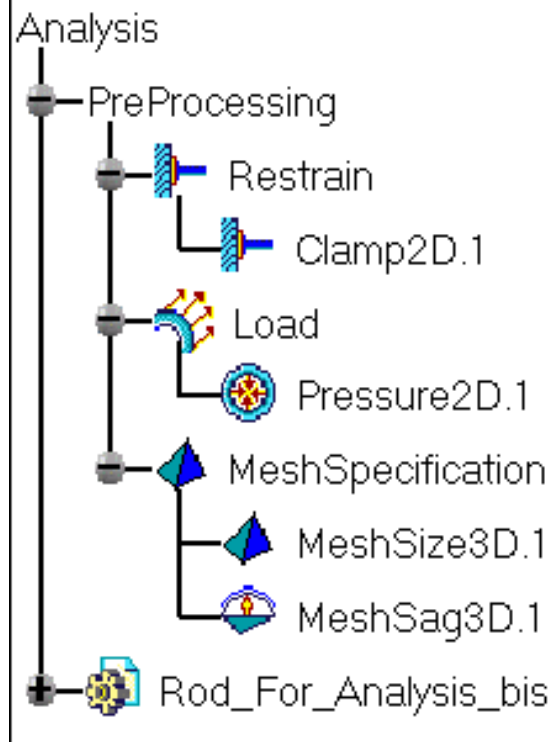
7. Click the Compute icon



The Compute dialog box is displayed.



- The features overall Mesh Sag and overall Mesh Size are created in the features tree. For a better understanding they are here presented alone in the Mesh specification folder.



8. By default their values are convenient for a normal computation (with high precision option). You can modify them to set more appropriate values: Click Cancel in the Computation panel. Double click on the Mesh Size symbol

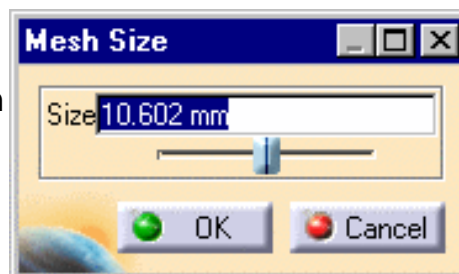


MeshSize3D.1 in the features tree.

- The Mesh Size dialog box appears

9. Set an overall adequate value.

10. Click Ok.





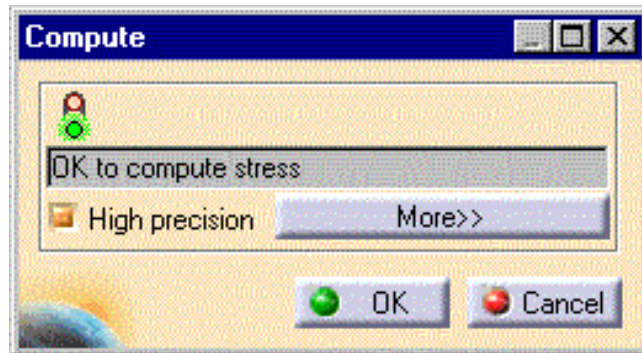
11. Repeat steps 8 to 10 with the Mesh Sag symbol

 MeshSag3D.1 in the features tree.

12. Click the Compute icon



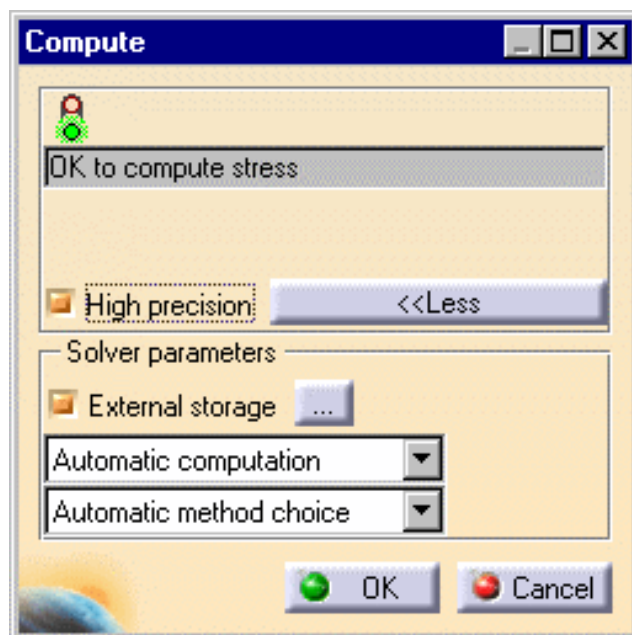
- The Compute dialog box is displayed.



13. Press more to define compute conditions.

- The Compute dialog box is enlarged.

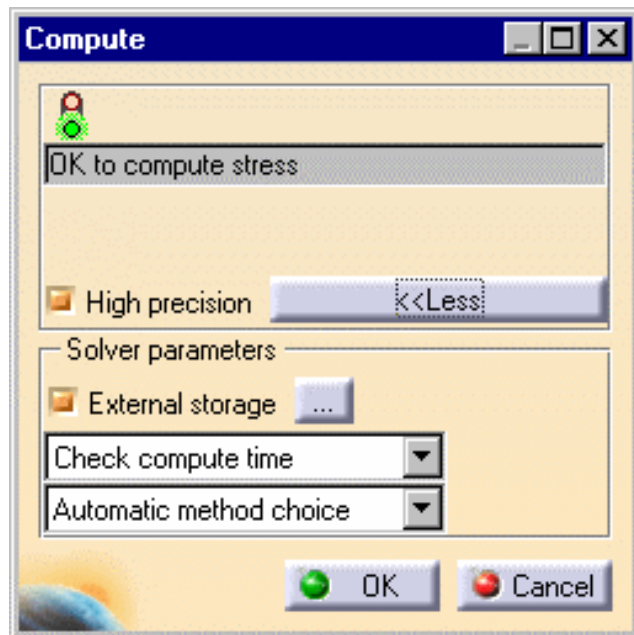
14. Check the External storage checkbox.



To avoid virtual memory problems when computing a large model, you can use the External Storage option. In this case you must set the CATIA Environment variable named CATTemp with an external storage directory path. To use this functionality, please refer to your *CATIA Version 5 Release 3 Infrastructure User's Guide: Customizing your environment on NT/UNIX Chapter*. The file created during computation will be used to store temporary solver data such as numbering, stiffness matrices and so on during computation. After computation, all data is cleared from the file.

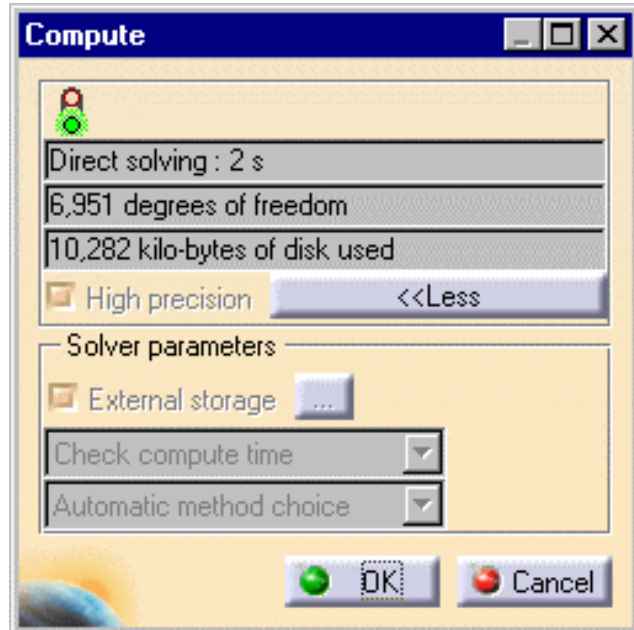
On NT workstations you can customize the CATTemp variable by right-clicking the CATIA icon on your desktop and selecting Edit from the contextual menu. Then you can enter the following line: set CATTemp=C:\Temp (for example) to define the external storage path. You must restart the session to take this modification into account.

15. Select Check compute time and Automatic method choice in the combos.



16. Click OK to evaluate the computation time.

- The computation method type and its execution time evaluation are displayed.

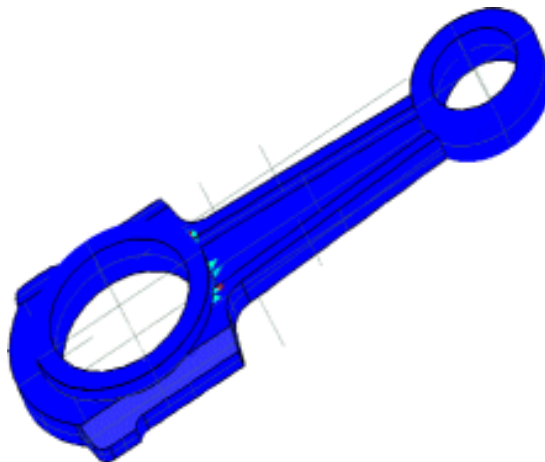


17. Click OK to run the computation and quit the dialog box.

18. Click the Precision icon

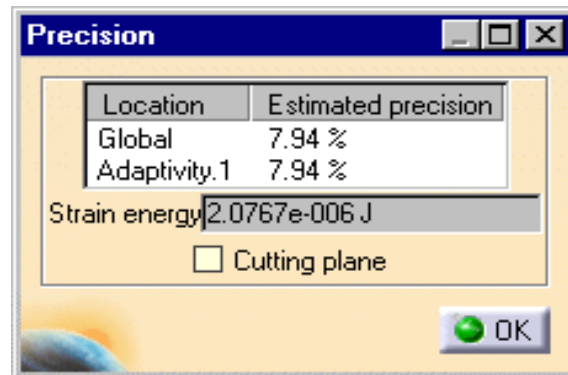


- The precision distribution is displayed on the part as shown.





- Therefore you can check if the locations of the local specifications were relevant enough or if you should improve your choices.
- This functionality enables you to optimize easily the modelization of your part.
- You can add as many different specifications (different values and different location) as desired.
- The Precision box is also displayed. It contains information relative to the estimated global precision of the computation.



19. Click OK to quit the dialog box.



# Computing and Visualizing a High Precision Mesh

A Mesh Only computation generates a high precision solid mesh that is made up of tetrahedron elements. The element order is parabolic, corresponding to 10-noded elements.



This task shows how to generate and visualize a high precision mesh on a part.



You can use the Rod\_For\_Analysis.CATPart document from the SAMPLES/gps\_analysis directory for this task.

You must have defined all physical attributes (material, restraint, and load) prior to this task.

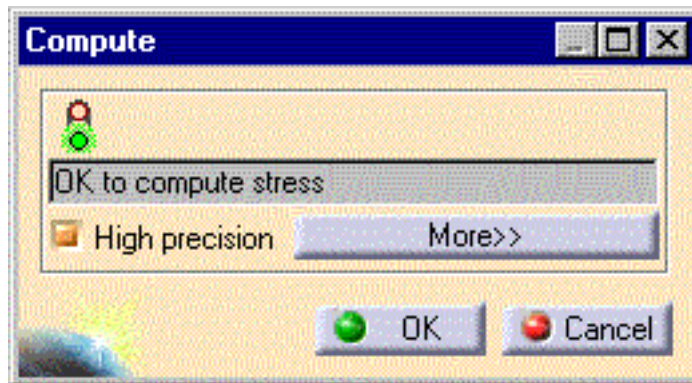


1. Click the Compute



icon .

The Compute dialog box is displayed.



2. Check the High Precision checkbox if not already checked.

3. Press More to enlarge the compute panel and select the Mesh Only option in the combo.





To avoid virtual memory problems when computing a large model, you can use the External Storage option. In this case you must set the CATIA Environment variable named CATTemp with an external storage directory path. To use this functionality, please refer to your *CATIA Version 5 Release 3 Infrastructure User's Guide*: [Customizing your environment on NT/UNIX Chapter](#). The file created during computation will be used to store temporary solver data such as numbering, stiffness matrices and so on during computation. After computation, all data is cleared from the file.

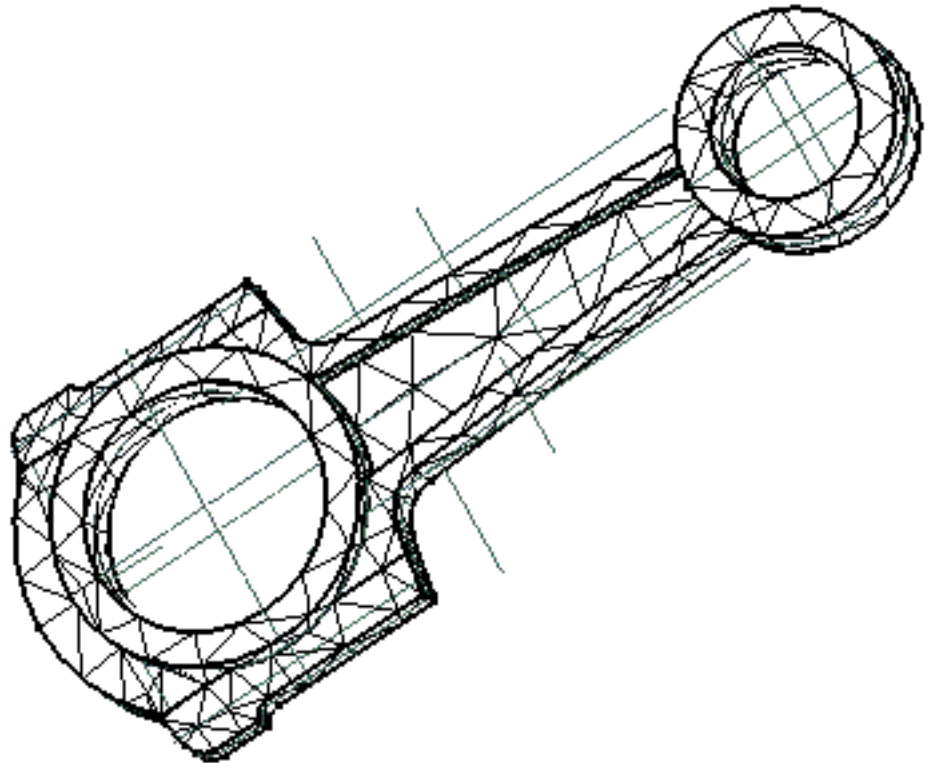
On NT workstations you can customize the CATTemp variable by right-clicking the CATIA icon on your desktop and selecting Edit from the contextual menu. Then you can enter the following line: set CATTemp=C:\Temp (for example) to define the external storage path. You must restart the session to take this modification into account.

3. Click OK to run the computation and quit the dialog box.

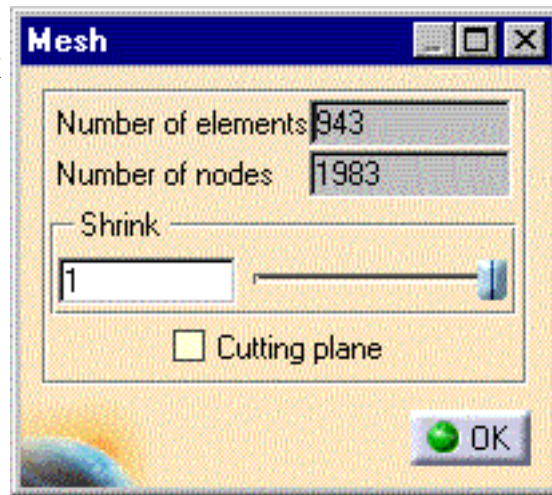
4. Click the Mesh icon



The mesh generated is displayed on the part as shown.



- The Mesh information box is also displayed. It contains information relative to the number of mesh entities.



5. You can use the cursor to modify element shrink.
6. Click OK to quit the dialog box.





# Computing a Rough Stress Analysis

A Stress Analysis computation corresponds to a sequence of operations including mesh generation, matrix assembly and system resolution for degrees of freedom.

For a Low Precision mesh, the element order is linear, corresponding to 4-noded elements.

Specifying relatively large global sizes and sags, and no local conditions, will have the effect of generating a rough mesh and a small size problem (few unknowns). Such computations serve as global approximations and are usually followed by more accurate computations.



This task shows how to run a rough computation on a specified part.



You must use the Stress Analysis workbench to perform this task

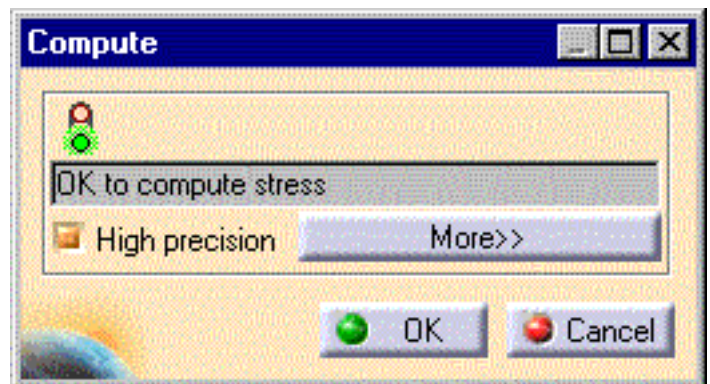
You must have defined all physical attributes (material, restraint, and load) prior to this task.

You can use the Rod\_For\_Analysis.CATPart document from the SAMPLES/gps\_analysis directory for this task.



1. Click the Compute icon .

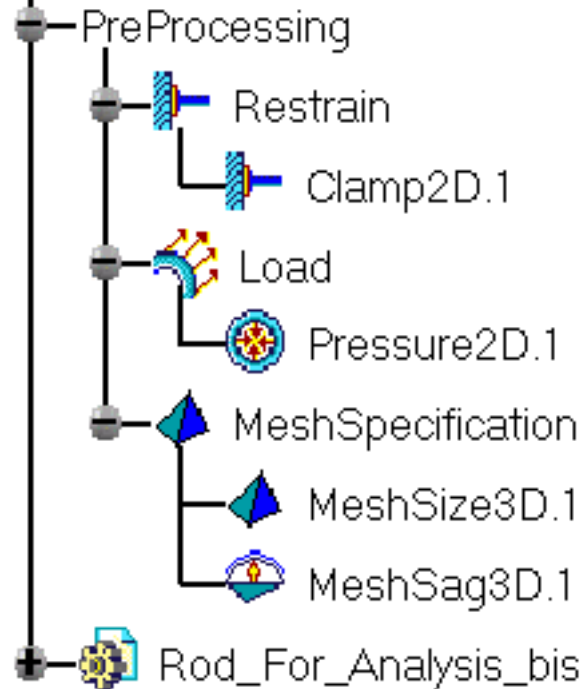
- The Compute dialog box is displayed.



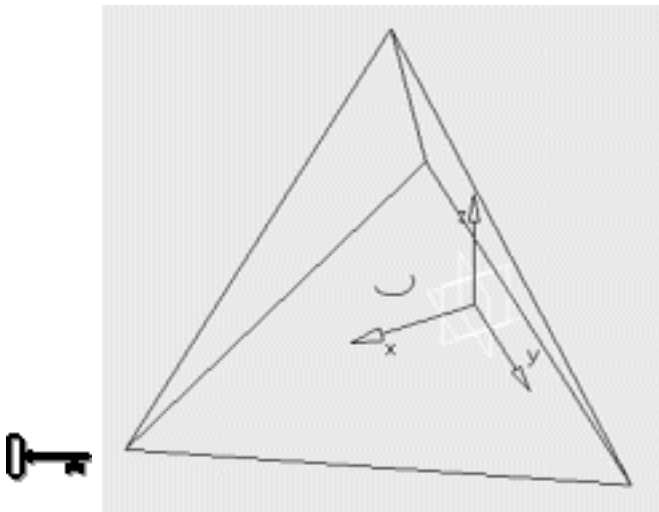
- The features overall Mesh Sag and overall Mesh Size are created in the features tree.



## Analysis




- A tetrahedron symbol of the corresponding size is displayed near the part.



- A circular arc whose radius equals the Sag is also displayed inside the tetrahedron symbol.

- These symbols are displayed either in the middle of the part if they represent the Global Sag and Global Size either in the middle of a geometric entity if they represent Local Sag and Local Size.


2. By default their values are convenient for a rough computation. However you can modify them to set larger values: Click Cancel in the Computation panel. Double click on the Mesh Size symbol

 MeshSize3D.1 in the features tree.

- The Mesh Size dialog box appears

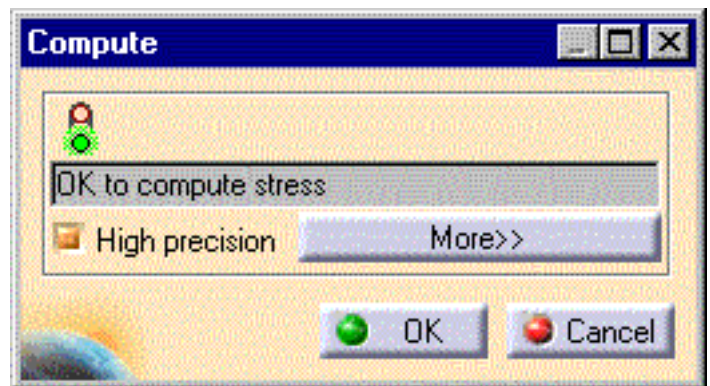
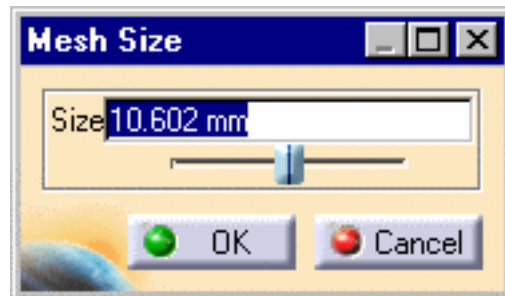
3. Set a relatively large value.

4. Repeat steps 2 and 3 with the Mesh Sag symbol

 MeshSag3D.1 in the features tree.

5. If you have canceled the Computation panel, click again on the Compute icon .

- The Compute dialog box is displayed.



6. Uncheck the high precision option and click on More to display the solver dialog components.

6. Select Automatic computation and Automatic method choice in the combos.



7. Click OK to run the computation.

- The status of the computation is displayed on the Status line.






## Refining and Computing a High Precision Stress Analysis

A Stress Analysis computation corresponds to a sequence of operations including mesh generation, matrix assembly and system resolution for degrees of freedom.

For a High Precision mesh, the element order is parabolic, corresponding to 10-noded elements.

The adaptivity functionality () firstly enables to define error objectives on local extremas areas for a new and refined computation. Secondly, those areas where objectives are then defined can be resized. Furthermore specified objectives are compatible with sag and size specifications : the most restraining conditions will be respected.



This task shows how to Refine and Compute a high precision stress analysis on a part.



You must use the Stress Analysis workbench to perform this task

You must have performed a first computation. Remark: To obtain a quicker and easier High Precision Stress Analysis it is advised to have done an accurate first computation ( with parabolic extrapolation : High Precision option in the compute panel ).

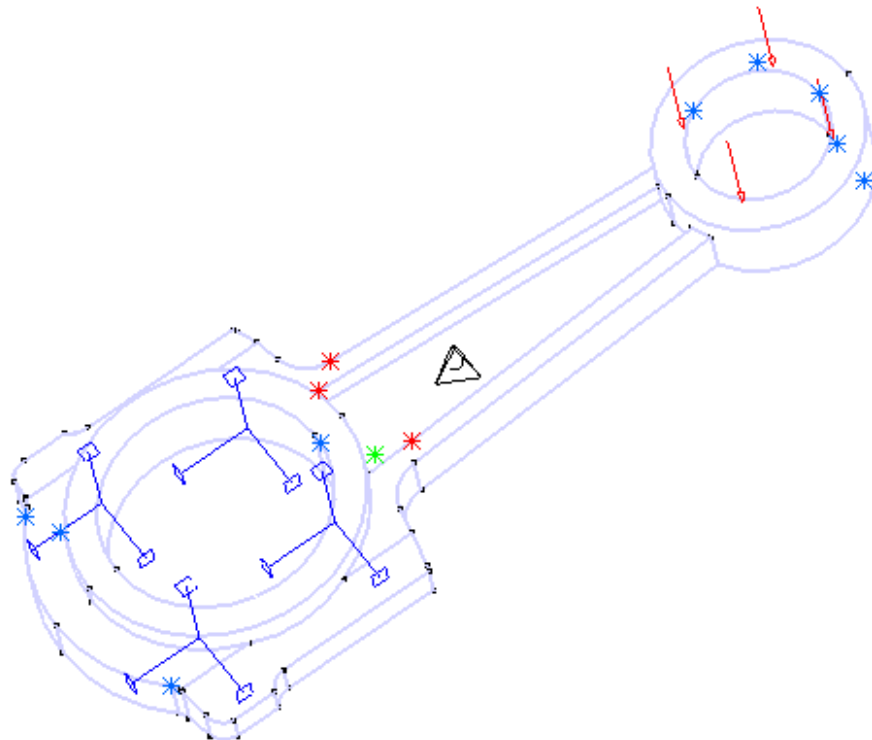
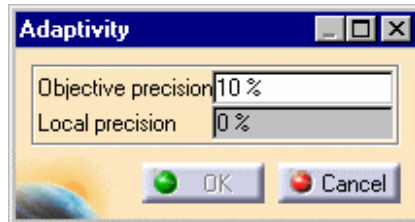
You can use the Rod\_For\_Analysis.CATPart document from the SAMPLES/gps\_analysis directory for this task and make a first computation as explained in the previous chapters.

1. Click on the



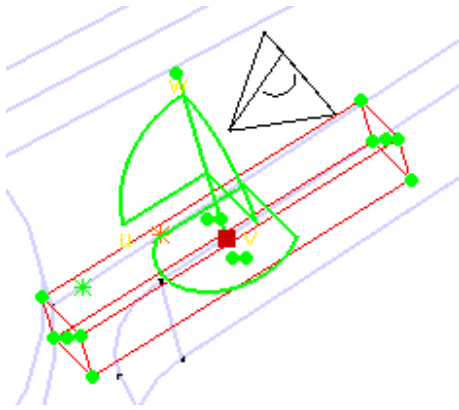
Adaptivity icon :

- The Adaptivity panel is displayed.
- The local extremas of the maximum stress are shown on the part as stars. Their colour depend on the value of the extrema:
- If the local extrema is equal or superior to 80 % of the global maximum of the maximum stress the colour of the star will be red, from 60 to 80% ( 80 not included ) yellow, from 40 to 60 green, from 20 to 40% light blue, below blue.

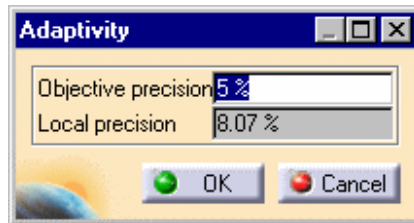


2. Click on a star to select an extrema where you want the computation to be more accurate.

- The Adaptivity Box and a compass are displayed.

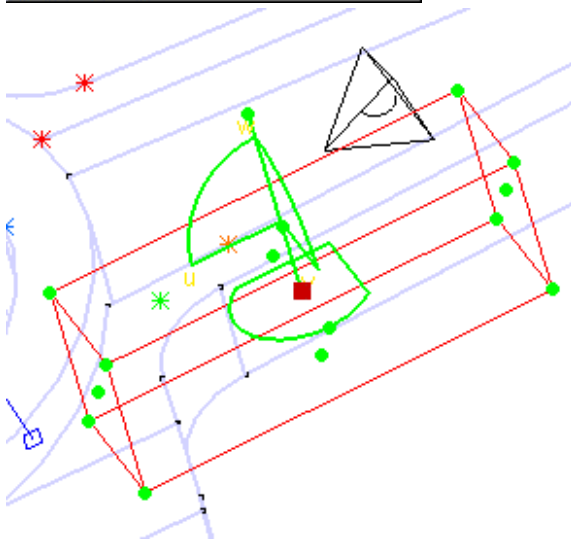


3. Set the required Objective Precision value.



4. You can select the compass and manipulate it to translate or rotate the Adaptivity Box. Green points can be selected and dragged with the mouse to enlarge the Box.


- Doing so enables you to define the volume of the part where the Error Objective will be specified.
- Sag and Size specifications can be added on surfaces, lines or curves separately from this volume selection. Then the Sag, the Size and the Error Objective will be respected wherever specified to refine the mesh.



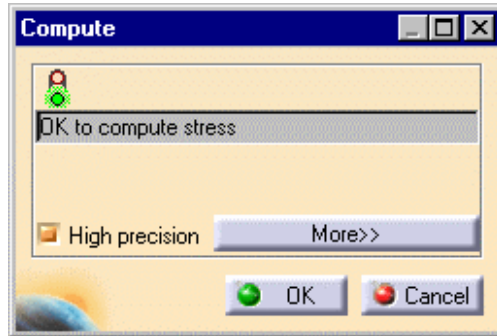
- The Adaptivity Box as other preprocessing features is re-editable.

5. Repeat steps 2, 3 and 4 to treat the different extremas.

6. Click on the

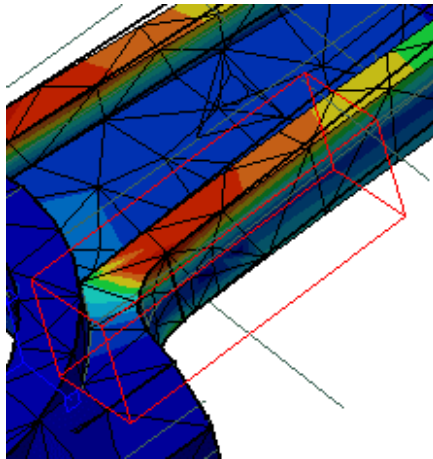
Compute icon  .

- The Computation panel appears.




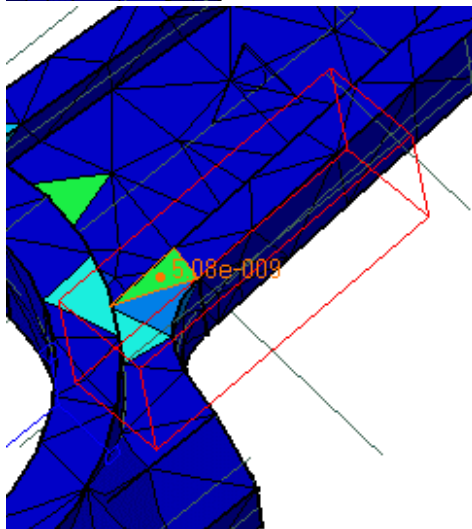
7. Check High precision and press Ok to launch the computation.

- The results are displayed:

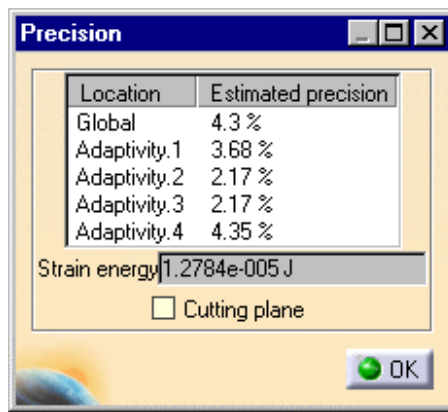


8. Click on the

Precision icon  to examine the Error results.



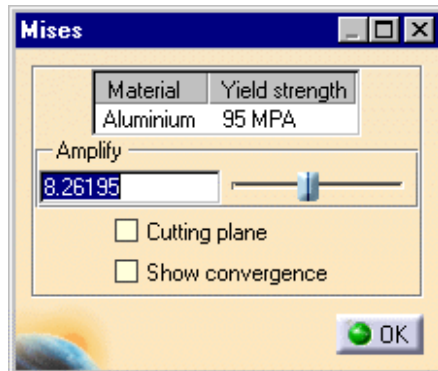
9. The Precision panel displays the estimated precision in each defined Adaptivity Boxes.



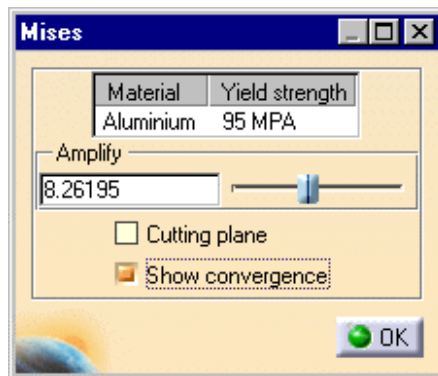
10. Click the von

Mises  icon.

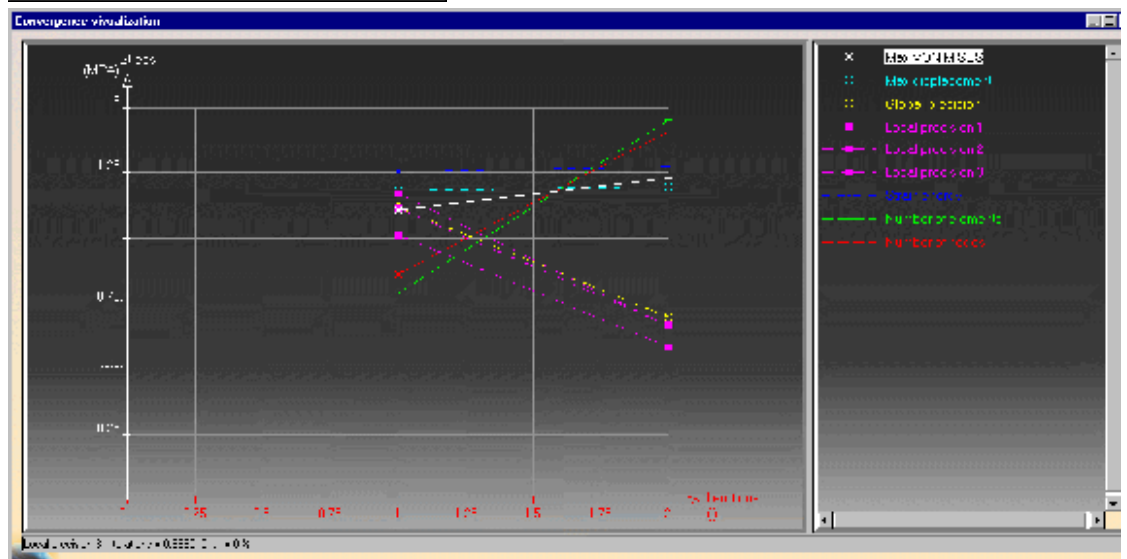
- The von Mises panel appears.



11. Check the Show convergence Checkbox.

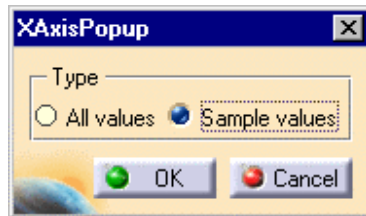


- The Convergence panel is displayed.



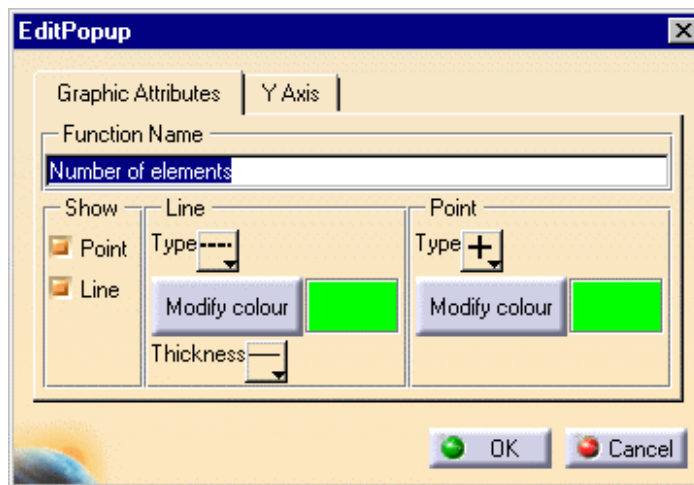


- You can select a curve with the mouse cursor to display its graph.
- A contextual menu available on the curves, points or the background (of the main viewer) is available to hide or show one or all the curves for example. Right click on one of those items with the mouse.
- X Axis display options are available. Double Click on it to edit it. You obtain then the panel shown here. Press Ok when finished.



12 To edit the graphic properties of a curve double click on it or one of its points.

- The Editor dialog box is displayed.



13. Modify the desired parameters and click Ok when ready.

14. To exit from the curve editor right click on the background of the main viewer. An select Exit in the displayed menu.



|  |   |  |
|--|---|--|
|  Up                          |  Computing and Visualizing a  |  Computing a Rough Stress /  |
|  Computing a High Precision |  Refining and Computing a Hi |  Computing a Normal Modes / |
|  |  Computing an Assembly mox   |  |

# Computing a Normal Modes Analysis

A Normal Modes Analysis computation corresponds to a sequence of operations resulting in the computation of dynamic mode shapes and vibration frequencies of the part.



This task shows how to run a normal modes computation on a specified part.



You must use the Dynamics Analysis workbench to perform this task.

The part material, additional masses (if any) and restraints (if any) need to be specified prior to the computation. Note that:

- not specifying a restraint has the effect of generating free body vibration modes.
- not specifying additional mass distributions has the effect of generating vibration modes under the effect of the part's structural mass only.

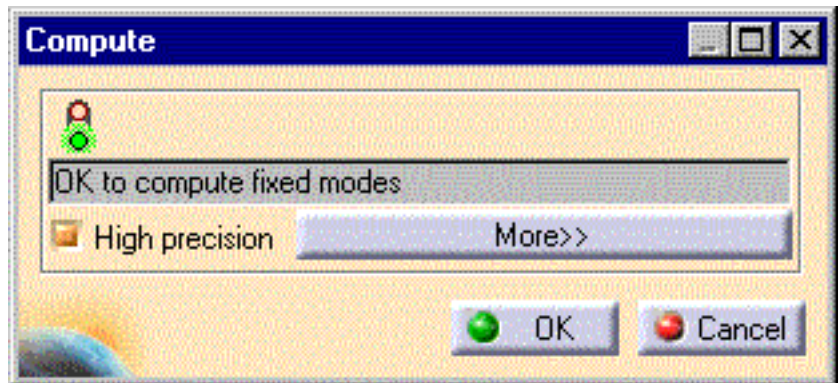
You can use the Rod\_For\_Analysis.CATPart document from the SAMPLES/gps\_analysis directory for this task.



1. Click the Compute icon

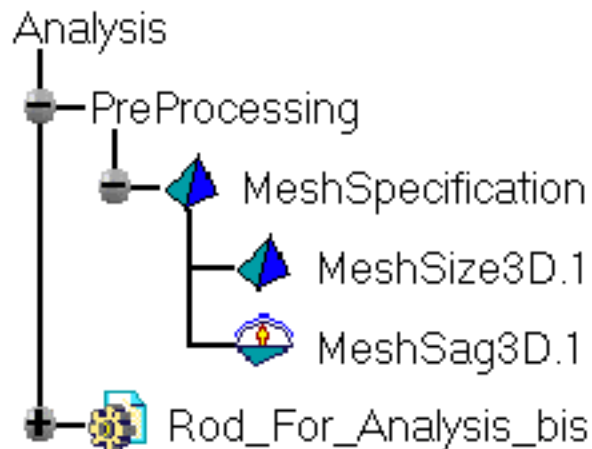


- The Compute dialog box is displayed.




2. The overall Mesh Sag and Mesh Size features are created in the features tree.

3. Click on Cancel in the Compute panel to edit later on those features.

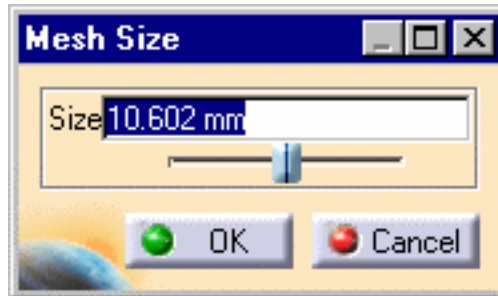


4. By default their values are convenient for a rough computation ( with a linear interpolation). However you can modify them to set more appropriate values: Double click on the Mesh Size symbol

 MeshSize3D.1 in the features tree.


- The Mesh Size dialog box appears

5. Set an overall adequate value.



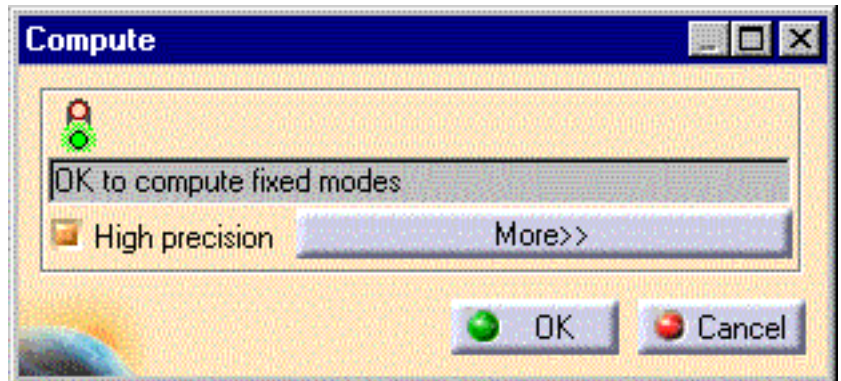
6. Click Ok.

7. Repeat steps 4 to 6 with the Mesh Sag symbol

 MeshSag3D.1 in the features tree.

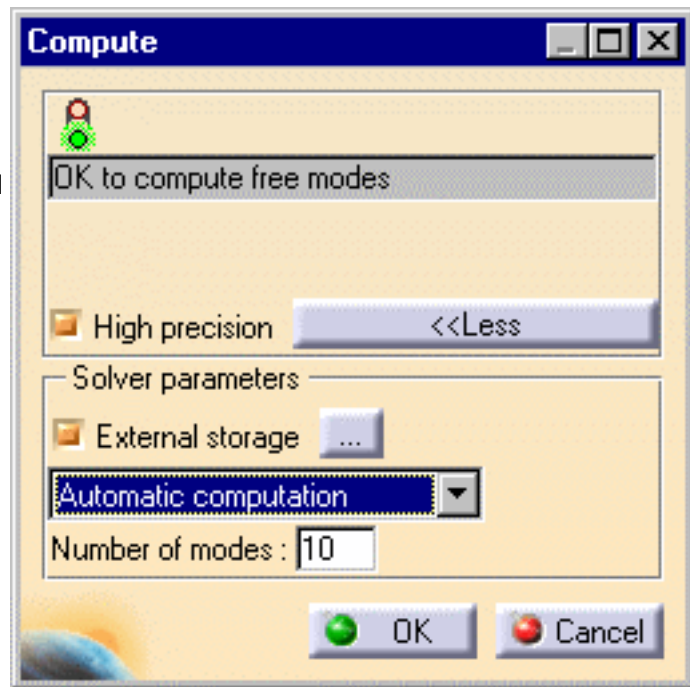
8. Click the Compute icon 

- The Compute dialog box is displayed.



9. Click on More to enlarge the dialog box.

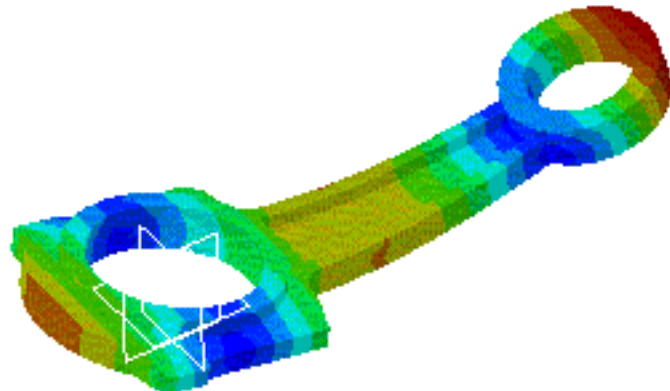
10. you can modify the computation parameters such as the Number of Modes if you wish.



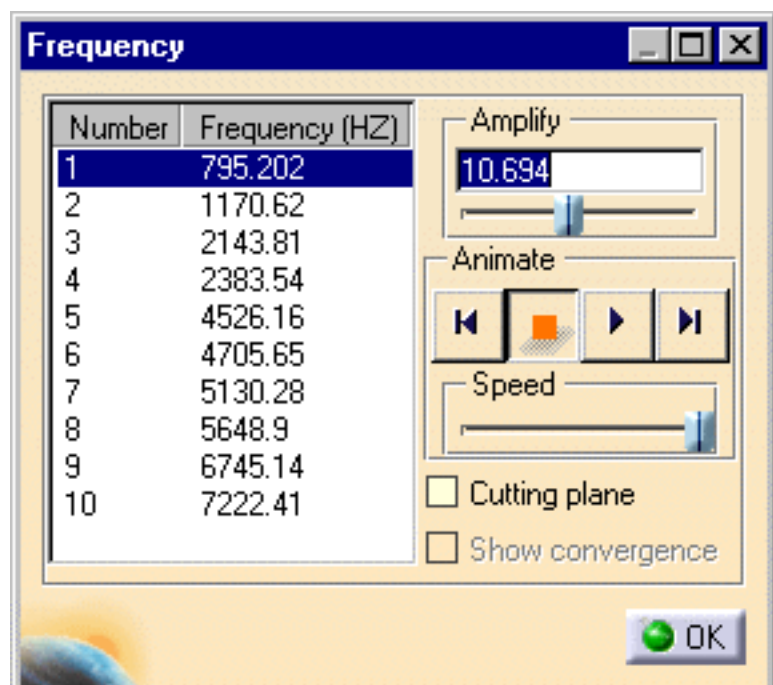
11. Click OK to run the computation.

- The status of the computation is displayed on the Status line

12. The results are displayed.



13. You can select a normal mode in the presented list and animate it.





# Computing an Assembly Model



You must own the Generative Assembly Structural Analysis product to follow this scenario.

You must own a multi-part model specified with a set of constraints and loads.

You can use the Rod\_For\_Analysis.CATPart and the Rod\_Axis\_For\_Analysis.CATPart documents from the SAMPLES/gps\_analysis directory for this task.



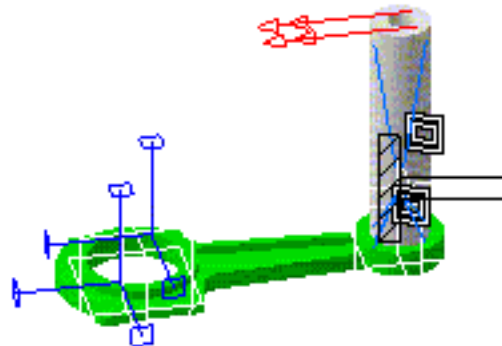
This task will show you how to launch a computation on a multi-part model.



There are several ways to perform a computation on a assembly model:

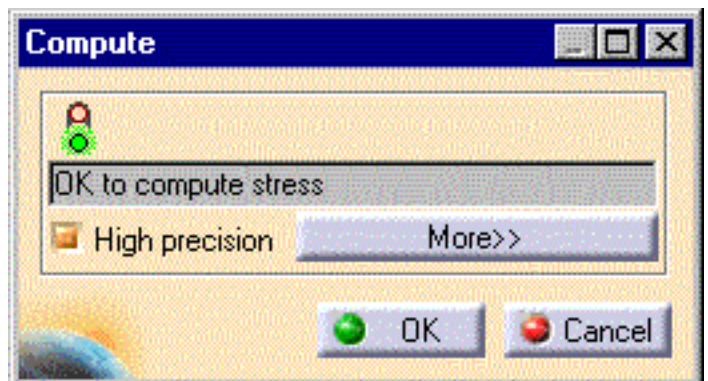
You can treat this model as a whole body specifying different global and local SAG and Elements Sizes for each part, or (for a minor interest ) you can keep the different parts separated provided that you have specified an appropriate set of constraints and loads for each of them. This scenario is valid for a stress and a modal computation (mesh only, rough or accurate computation as presented in the previous chapters).

1. Import your model.( or repeat steps 1 to 11 of the getting started section named [Creating a Slider Connection](#) ).



2. Click the Compute icon .

- The Compute dialog box is displayed.



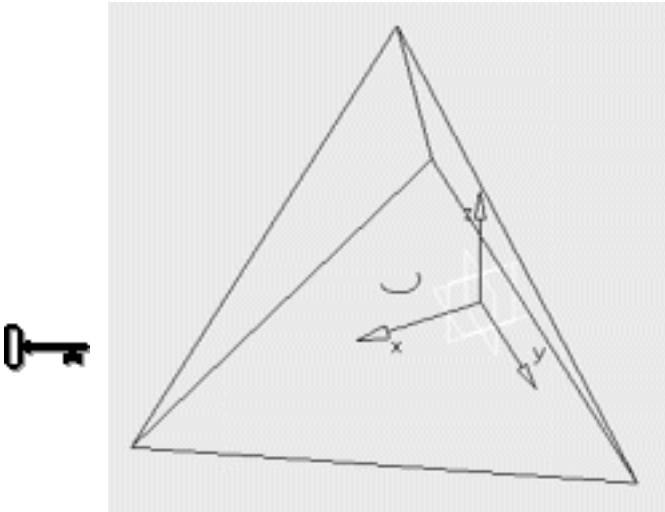
- A couple of overall Mesh Sag and overall Mesh Size are created for each part in the features tree. Here only one couple is presented as example.

 MeshSize3D.1

 MeshSag3D.1




- A tetrahedron symbol of the corresponding size is displayed near each part.



- A circular arc whose radius equals the Sag is also displayed inside the tetrahedron symbol.
- These symbols are displayed either in the middle of the part if they represent the Global Sag and Global Size either in the middle of a geometric entity if they represent Local Sag and Local Size.


3. By default their values are convenient for a rough computation. However you can modify them to set larger values: Click Cancel in the Computation panel. Double click on the Mesh Size symbol

 MeshSize3D.1 in the features tree.

- The Mesh Size dialog box appears

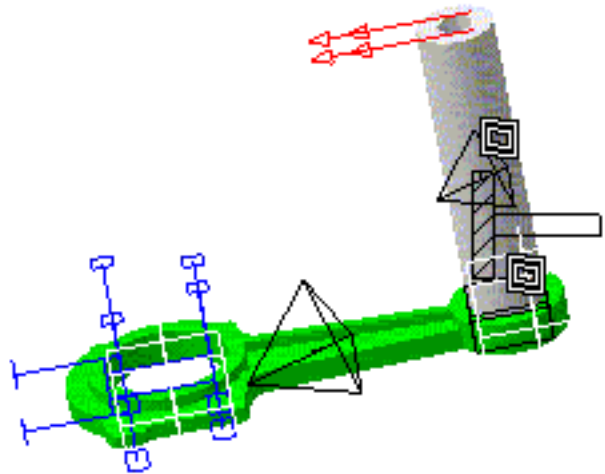



4. Set a relatively large value.  
5. Repeat steps 3 and 4 with the Mesh Sag symbol

 MeshSag3D.1 in the features tree.

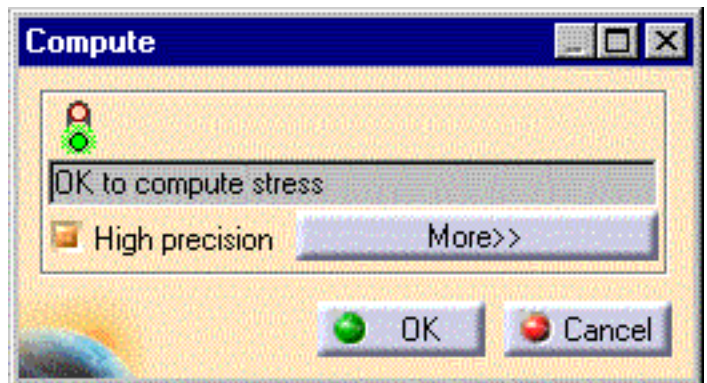
6. Repeat steps 3 to 5 with another couple of Mesh Sag and Size as desired.

7. You can impose as well local SAG and Size on some geometrical areas of each part, see section "[Computing a high Precision Stress Analysis](#)".

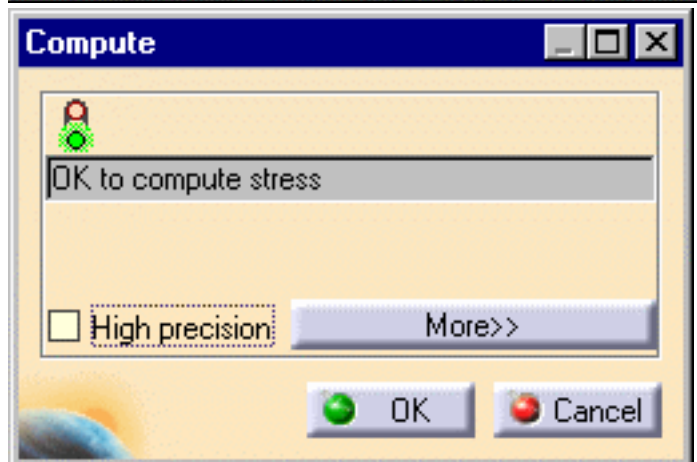


8. Click again the Compute icon  if its panel was canceled.

- The Compute dialog box is displayed.

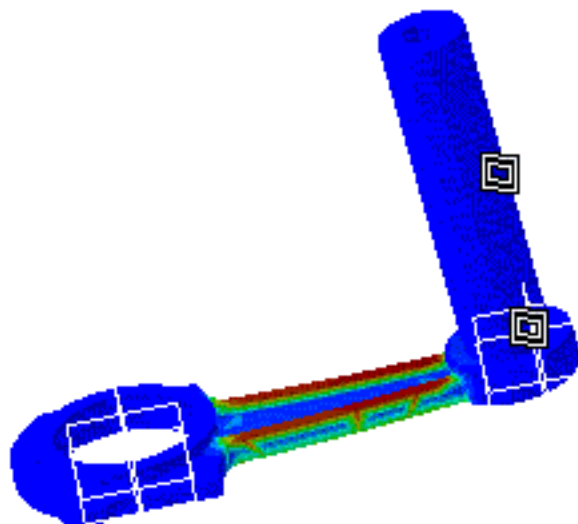


9. Uncheck the High precision checkbox for a rough analysis.



10. Click OK to launch the computation.

The results are displayed:





In case of a lack of global restraint in your modelization of the assembly, the free global degree of freedom ( translation or rotation ) will be stressed by the visualization of the corresponding part or assembly translated or rotated along its direction. As a lack of restraint creates a global singularity, you will have to correct it before launching a new computation.



# Visualizing Results






CATIA - Generative Part Structural Analysis provides easy methods to visualize results of analysis computations.

Using the Stress Analysis workbench, you can visualize:

- Von Mises stresses
- Displacements
- Precision map information
- Principal stress data.

Using the Dynamics Analysis workbench, you can visualize Vibration mode shapes.

Results visualization is made possible by the discretization of your part. For the continuous output fields, the values of interest are displayed on the FE mesh which serves as an evaluation grid for the visualization of result data.

|  |  |  |  |  |
|--|--|--|--|--|
|  |  |  |  |  |
| <a href="#">Visualizing Von Mises Stresses</a>                                     | <a href="#">Visualizing Displacements</a>  | <a href="#">Visualizing Precisions</a>   | <a href="#">Visualizing Principal Stresses</a>                                       | <a href="#">Visualizing Normal Mode Shapes</a>                                       |



For every visualization modes except for the Normal Mode Shapes, you can [customize the Color Palette](#).



# Visualizing von Mises Stresses

The von Mises stress distribution, often used in conjunction with the material yield stress value to check part structural integrity, is directly evaluated from the results of the computation step.

The von Mises stress is a scalar quantity obtained from the principal stress values at a point. This task shows how to visualize von Mises stresses on a part.

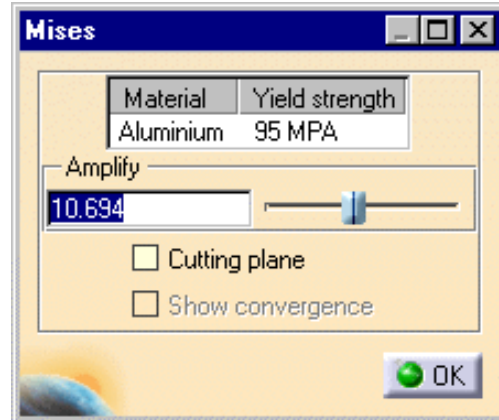


You must have successfully performed the computation using the Stress Analysis workbench.

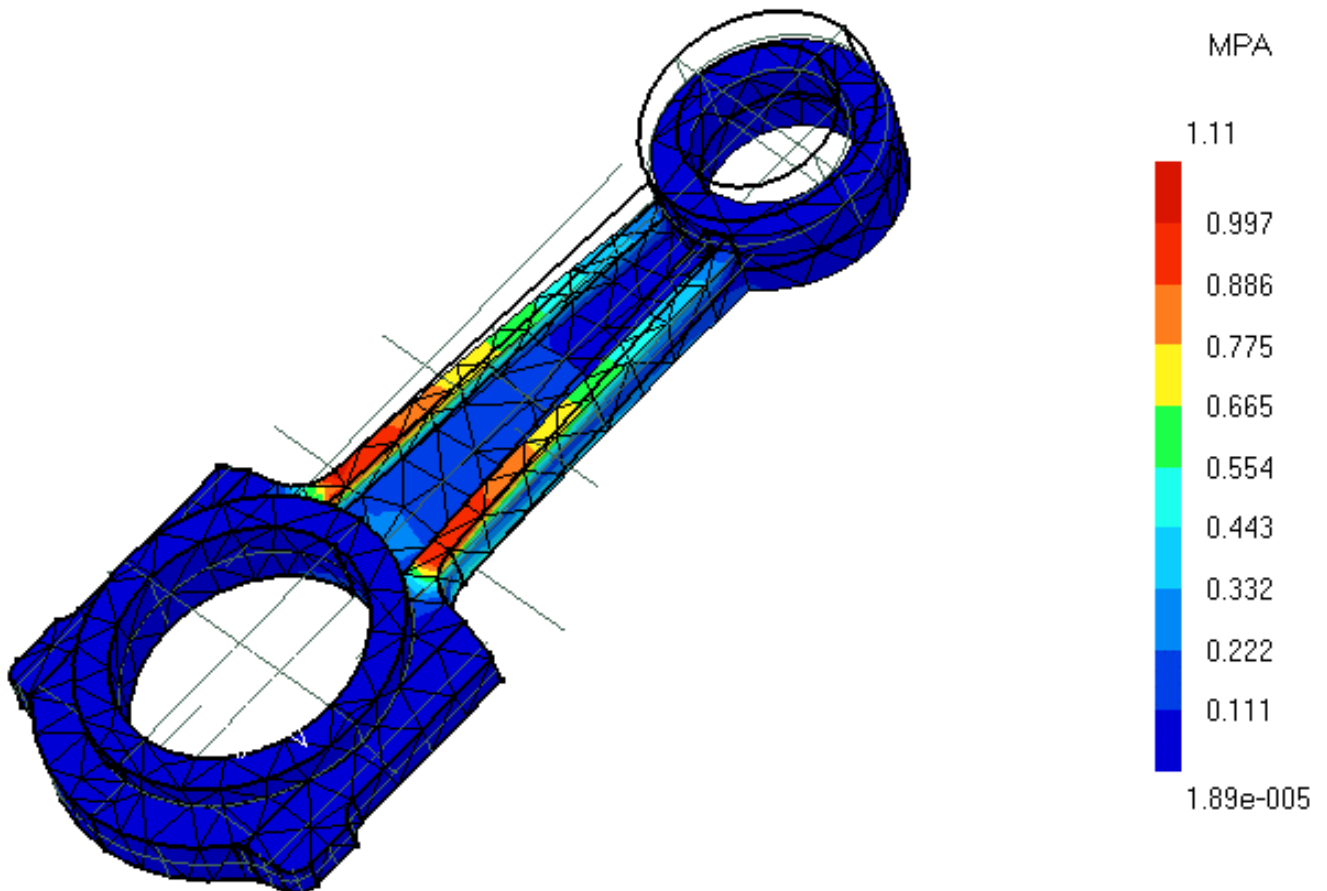
You can use the Rod\_For\_Analysis.CATPart document from the SAMPLES/gps\_analysis directory for this task.

1. Click the Mises icon .

- The Mises dialog box is displayed.



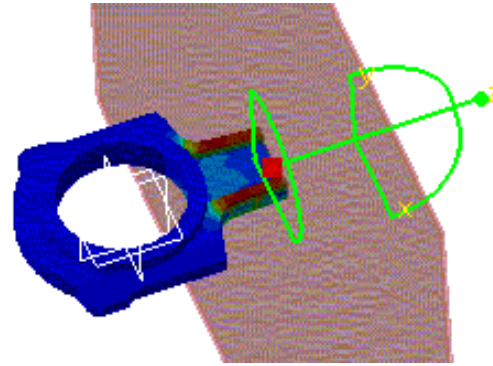
- The Von Mises stress distribution on the part is also displayed in ISO-value mode, along with a color palette.



new

2. You can vary the amplitude of the deformed structure with the cursor.

3. You can visualize the part inner stresses using the Cutting Plane: select the corresponding checkbox. (CATIA - P2 interface and licence only)



You can handle the compass with the mouse in order to rotate or translate the Cutting Plane. ( To do so, select an edge of the compass and drag the mouse).

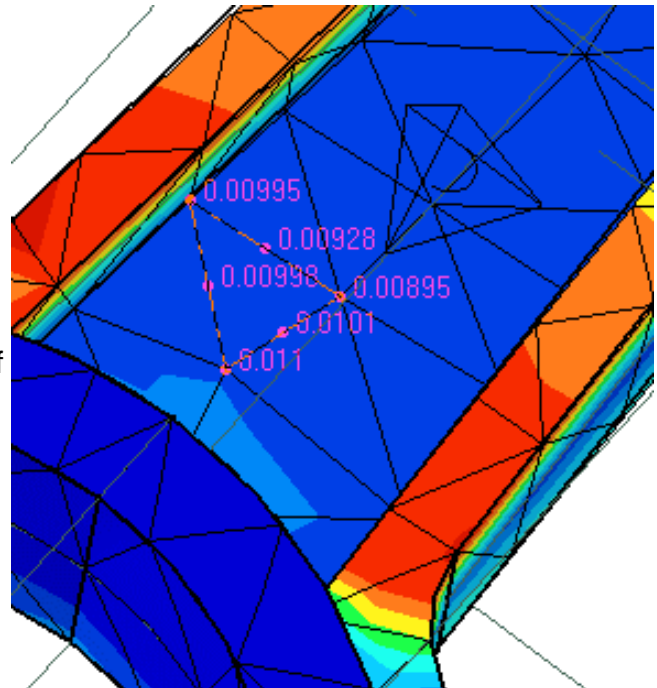


You should note the yield strength value for the material displayed in the dialog box. For a sound structural design, the maximum value of the Von Mises stresses should be less than this yield value.

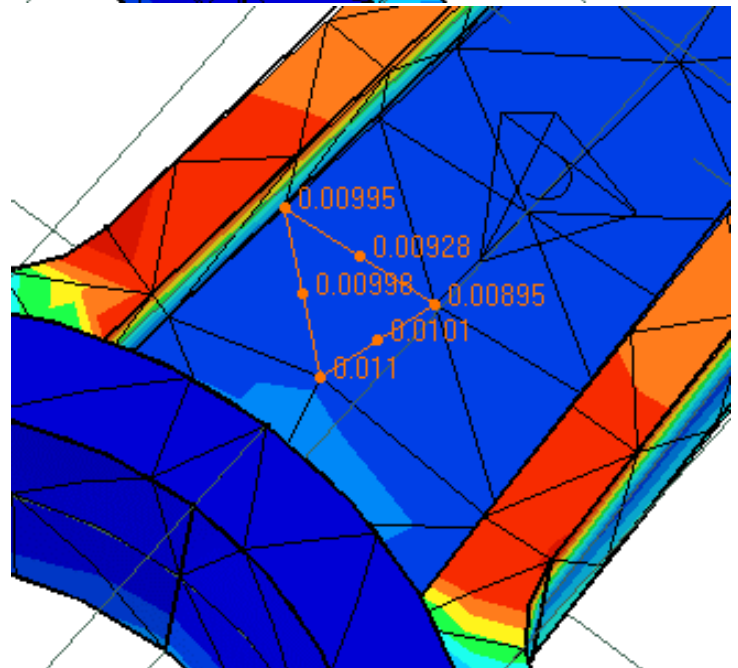
4. Click OK to quit the dialog box.

new

5. When the mouse cursor is passing over a finite element of the mesh, the values of the von Mises stresses are displayed on each of its nodes.



6. Click on one of those finite elements to show those values continuously.





 Up  
 Visualizing Precisions

Visualizing Von Mises Stress  
 Visualizing Displacements  
 Visualizing Principal Stresses  
 Visualizing Normal Mode Shapes  
 Editing the Color Palette



# Visualizing Displacements

The displacement resulting from part loading is important for a correct understanding of the way in which the part behaves.

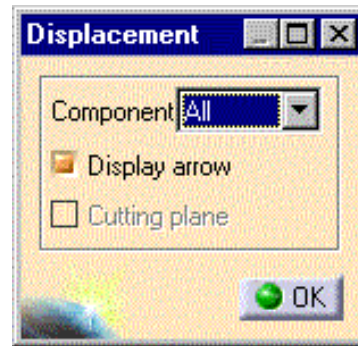
This task shows how to visualize displacements on a part.

You must have successfully performed the computation using the Stress Analysis workbench.

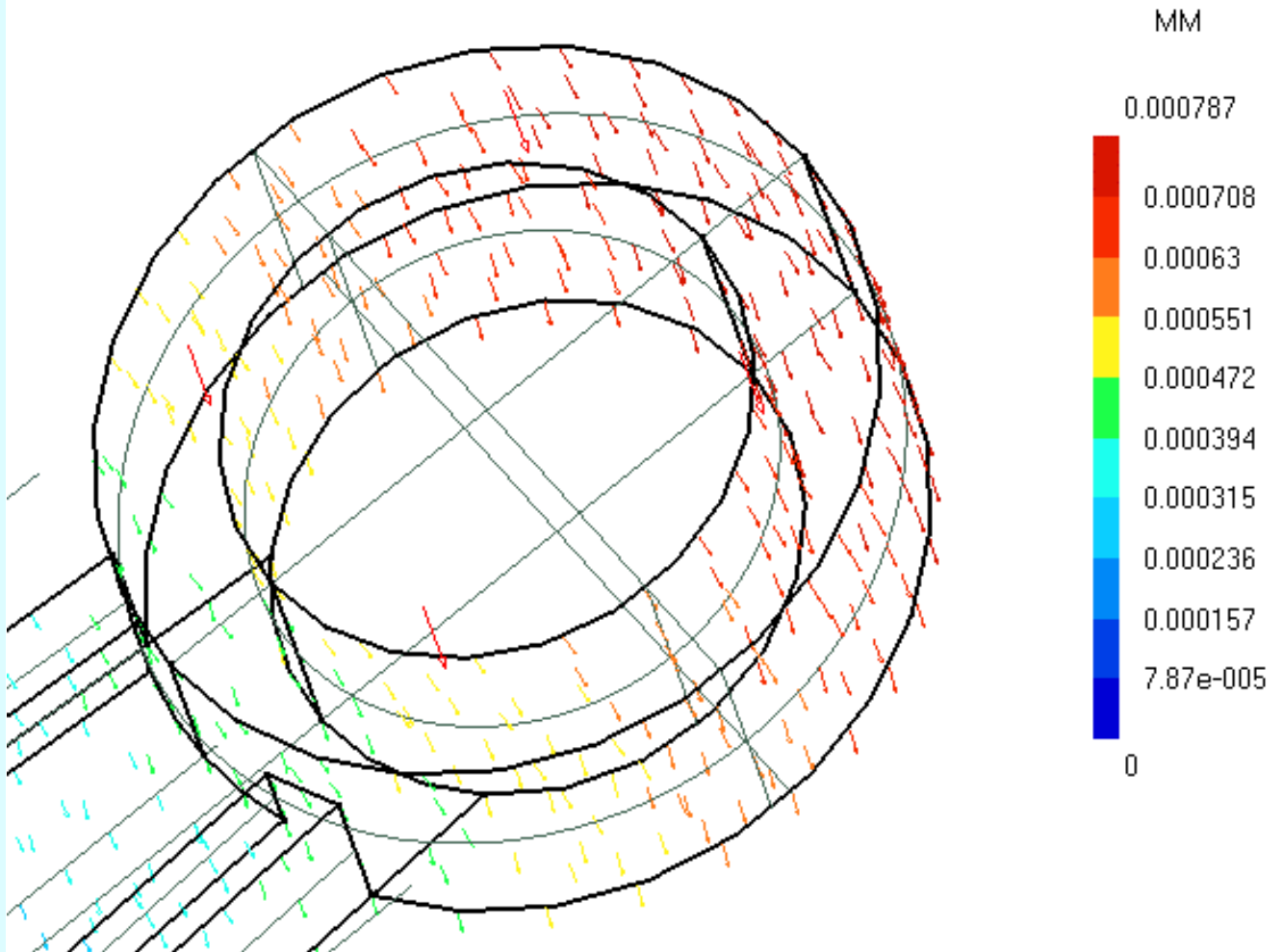
You can use the Rod\_For\_Analysis.CATPart document from the SAMPLES/gps\_analysis directory for this task.

1. Click the Displacement icon .

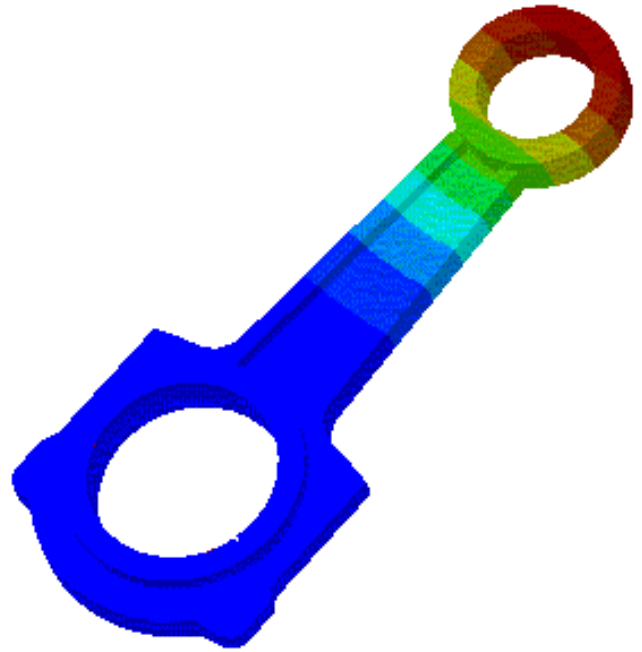
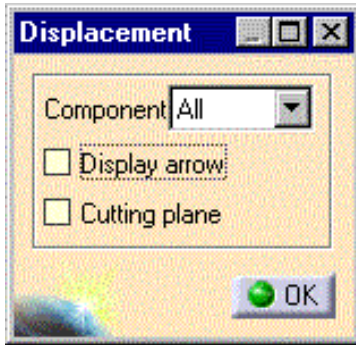
- The Displacement dialog box is displayed.



- The displacement vector field is also displayed in arrow symbol mode, along with a color palette.



2. Reset the Display Arrow checkbox in order to display an ISO-value plot of the displacement magnitude.

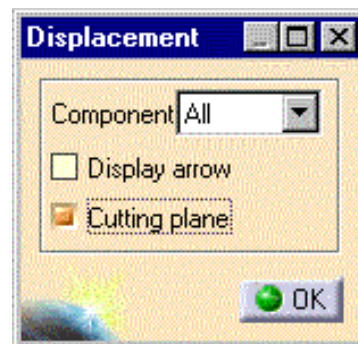


3. Set the Display Arrow checkbox again in order to choose between several modes of component display:

- All components
- X components
- Y components
- Z components
- XY projection
- YZ projection
- ZX projection

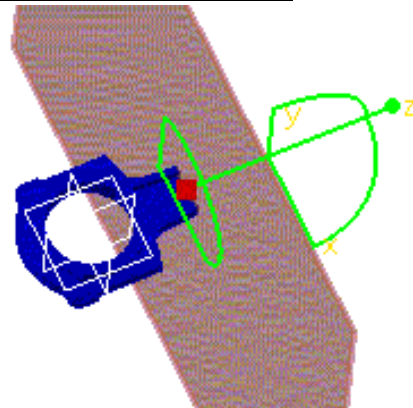
4. Select XY in the Component combo box to display the displacement field projection in the XY plane.

Note the arrow symbols orientation change depending on the chosen components.



5. You can visualize the part inner displacements using the Cutting Plane: select the corresponding checkbox (CATIA - P2 interface and licence only ).

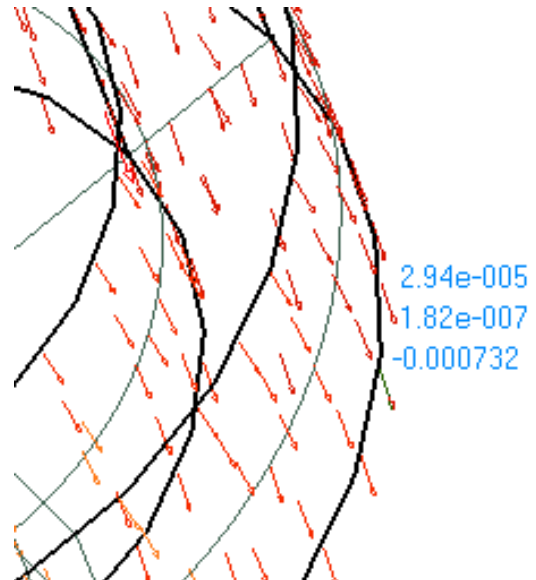
You can handle the compass with the mouse in order to rotate or translate the Cutting Plane.( To do so, select an edge of the compass and drag the mouse ).



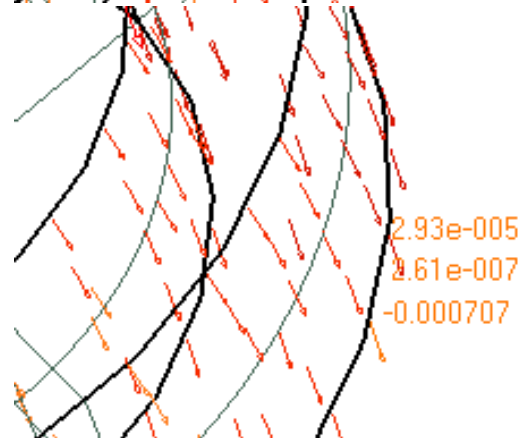
6. Click OK to quit the dialog box.

new

7. When the mouse cursor is passing over an arrow, its coordinates are visualized and expressed in the mesh global reference frame.



8. Click on an arrow to display continuously its coordinates.



Up



Visualizing Precisions



Visualizing Von Mises Stress



Visualizing Principal Stresses



Editing the Color Palette



Visualizing Displacements



Visualizing Normal Mode Shapes

# Visualizing Precisions

All computations are based on certain approximations, hence FEM results are only valid with some uncertainty.

A Computation Precision estimate can be displayed. The program evaluates the validity of the computation and provides a global statement about this validity. It also displays a predicted energy error norm map which gives qualitative insight about the error distribution on the part.

This task shows how to visualize the computation precision information.



You must have successfully performed the computation using the Stress Analysis workbench.

You can use the Rod\_For\_Analysis.CATPart document from the SAMPLES/gps\_analysis directory for this task.

1. Click the

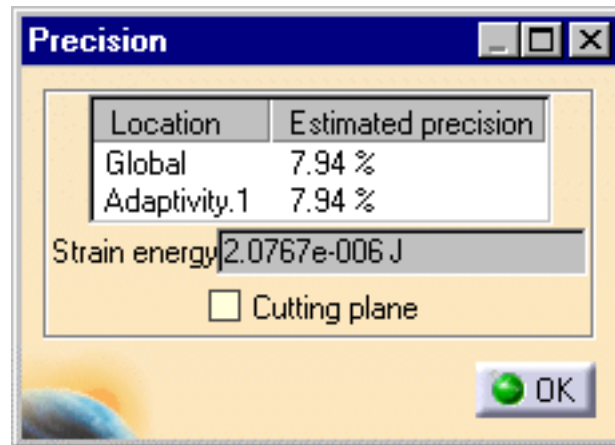


Precision icon



The Precision information box is displayed.

- The Global Precision percentage provides an estimation of the global validity of the computation.
- The total Strain Energy value is also displayed.

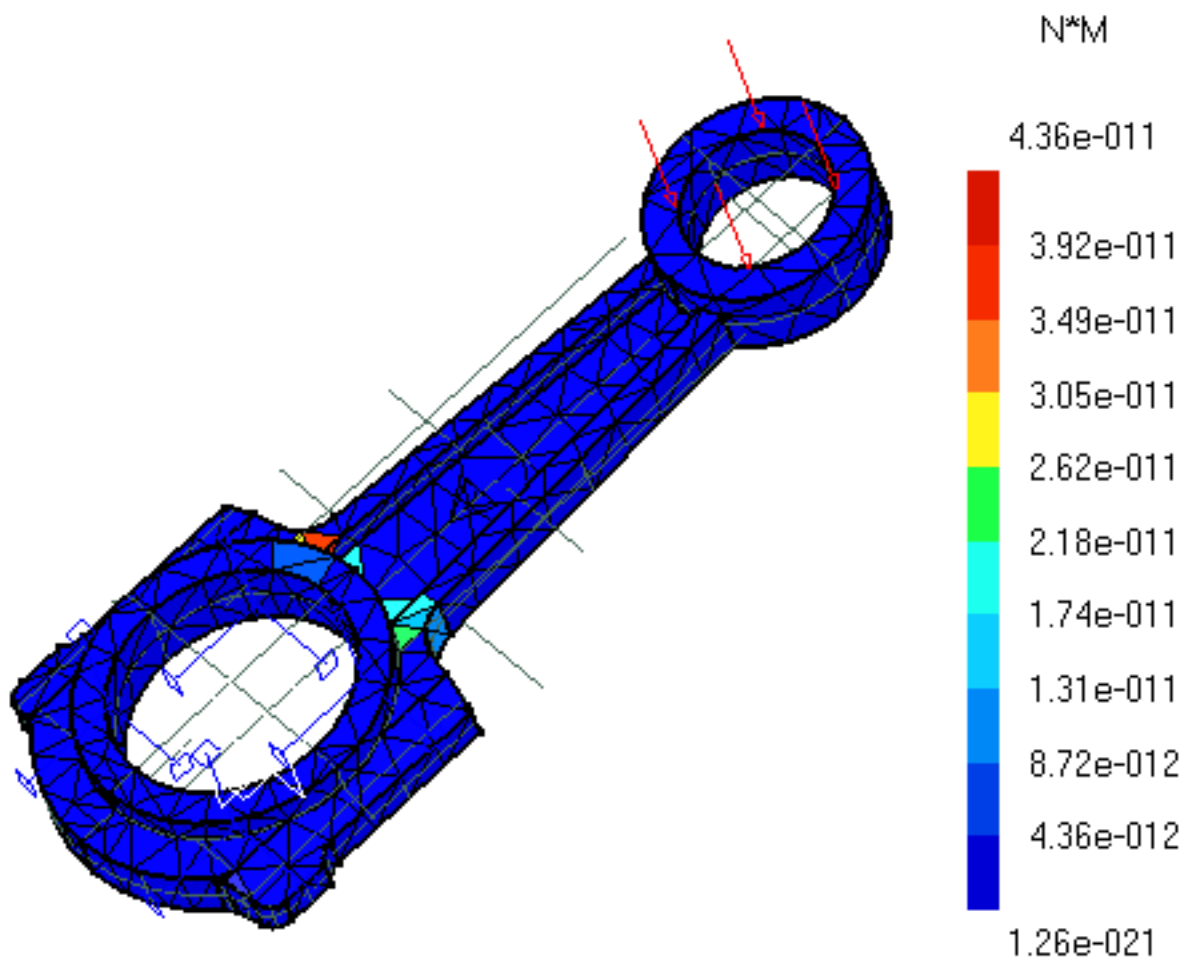




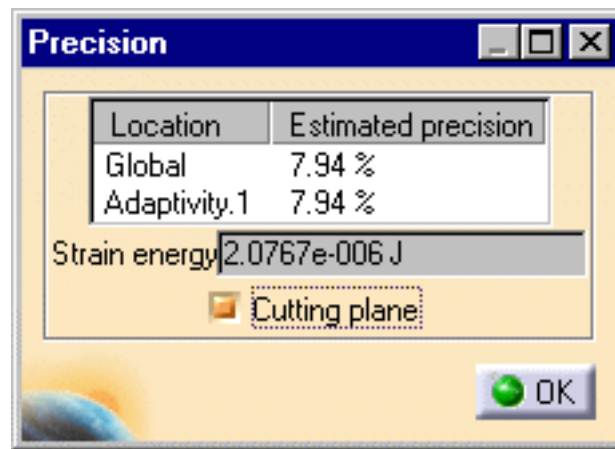
2. The Strain Energy Variation distribution map on the part is also displayed, along with a color palette.

This map provides qualitative information about the way in which estimated computation errors are relatively distributed on the part.

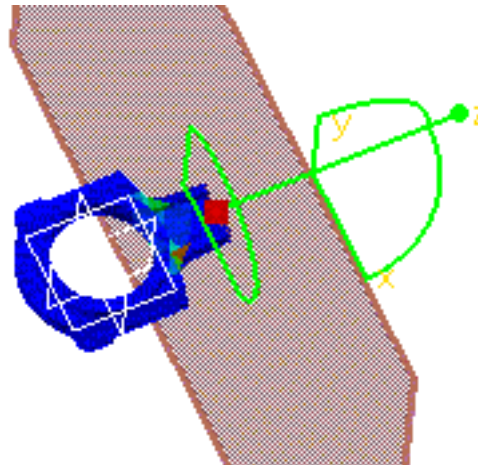
- If the error is relatively large in a particular region of interest, the computation results in that region may not be reliable. A new computation can be performed to obtain better precision.
- To obtain a refined mesh in a region of interest, use smaller Local Size and Sag values in the computation step.



3. You can visualize the part inner precision using the Cutting Plane: select the corresponding checkbox (CATIA - P2 interface and licence only ).

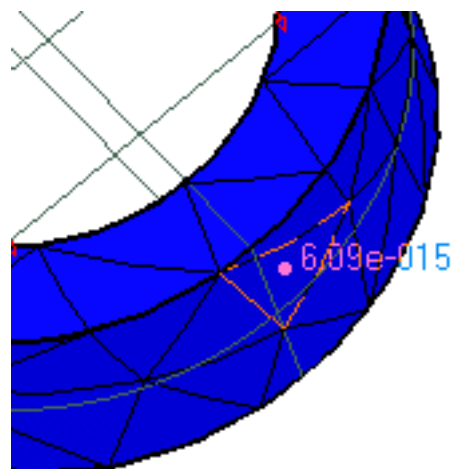


You can handle the compass with the mouse in order to rotate or translate the Cutting Plane. ( To do so, select an edge of the compass and drag the mouse ).



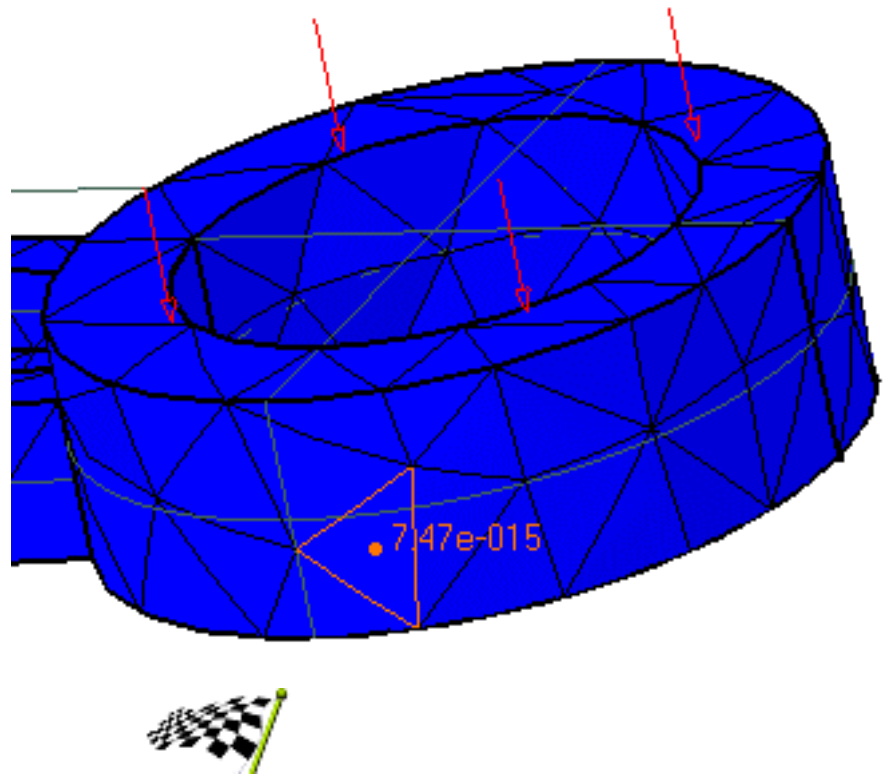
4. Click OK to quit the dialog box.

5. When the mouse cursor is passing over a finite element, its Strain Energy Variation is displayed.



new

6. Click on one element to visualize continuously its Strain Energy Variation.



 Up  
 Visualizing Precisions

 Visualizing Von Mises Stres  
 Visualizing Principal Stresse  
 Editing the Color Palette

 Visualizing Displacements  
 Visualizing Normal Mode Sh



# Visualizing Principal Stresses

Principal Stresses are of great interest for visualizing the load path in a loaded part.

At each point, they represent directions relative to which the part is in a state of pure tension/compression (zero shear stress components on the corresponding planes).

This task shows how to visualize the computation precision information.

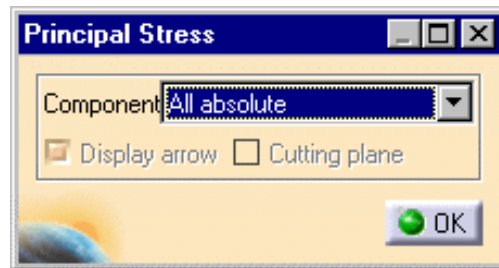
You must have successfully performed the computation using the Stress Analysis workbench.

You can use the Rod\_For\_Analysis.CATPart document from the SAMPLES/gps\_analysis directory for this task.

1. Click the Principal Stresses

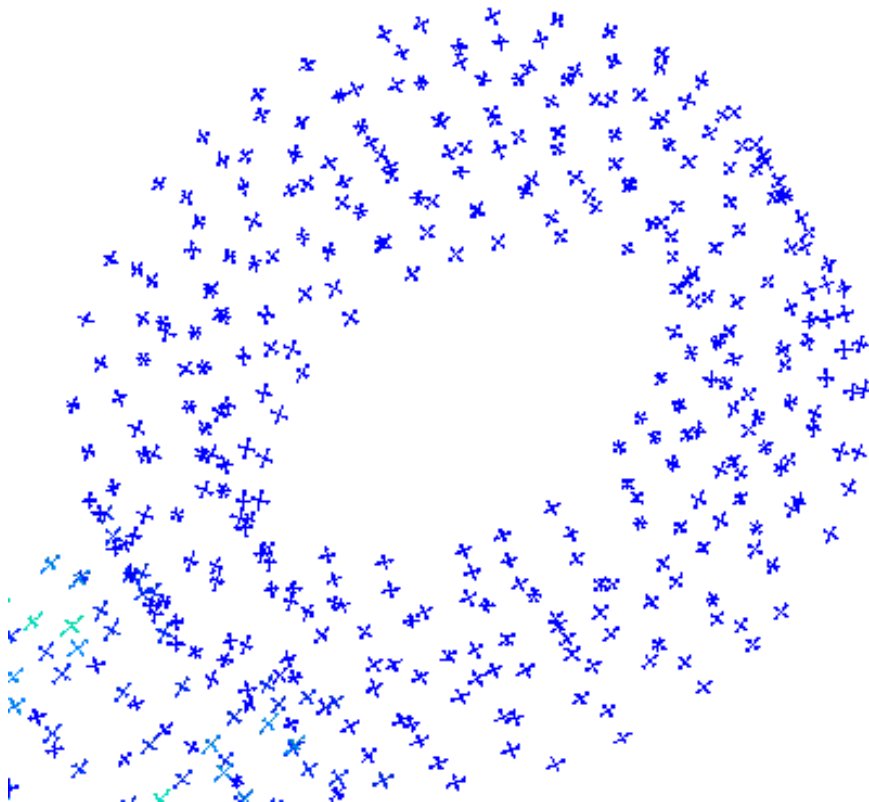
icon 

The Principal Stress dialog box is displayed.



The full principal stress distribution on the part is also displayed in arrow symbol mode, along with a color palette.

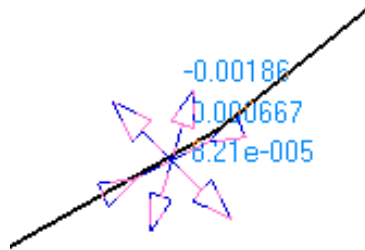
- At each point, a set of three directions is represented by line symbols (principal directions of stress).
- Arrow directions (inwards / outwards) indicate the sign of the principal stress. The color code provides quantitative information.



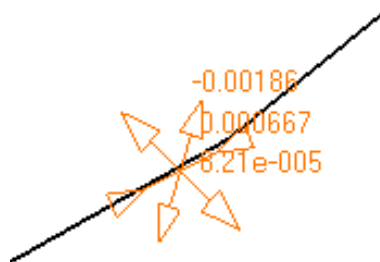
- Here the "All Absolute" option is selected meaning that the absolute value of each principal stress is taken into account for each color distribution.



2. When the mouse cursor is passing over a tensor representation, its principal values are displayed.

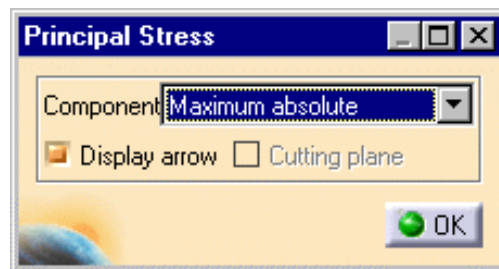


3. You can display them continuously if you click on the tensor symbol.



Several display modes are available:

- All
- Maximum
- Minimum
- Intermediate
- Principal Shearing.

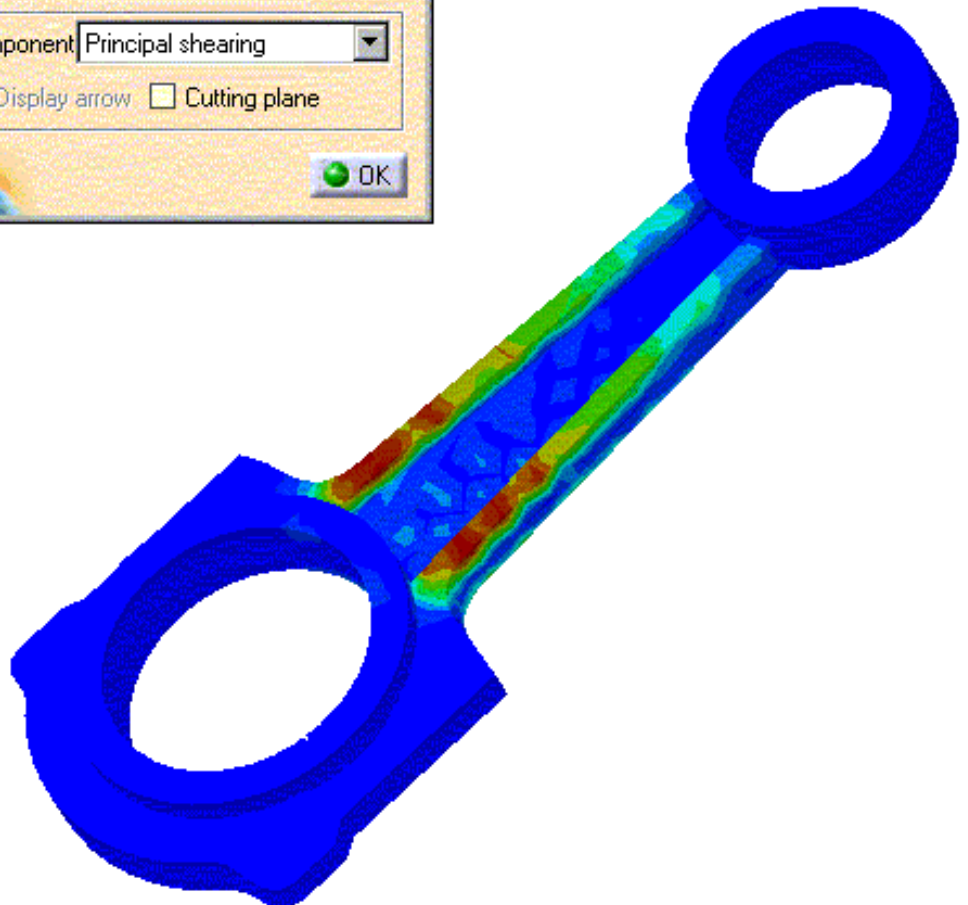
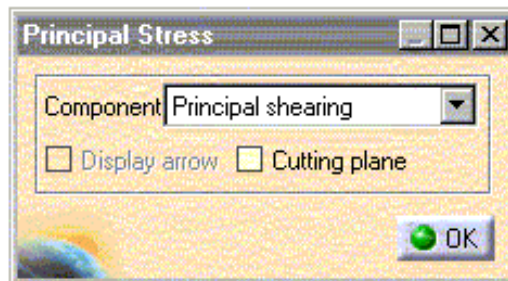


4. Select Maximum Absolute in the Component combo box.

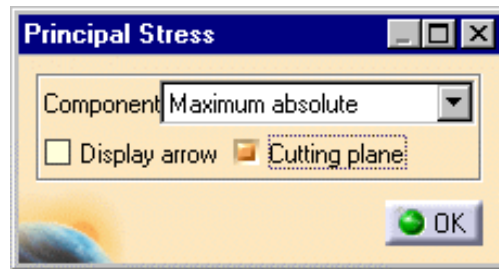
- The display is automatically updated to show the maximum normal stress direction at each point.
- The absolute value of the maximum normal stress is taken into account for the color distribution.

5. In the same way, select Principal Shearing in the Component combo box.

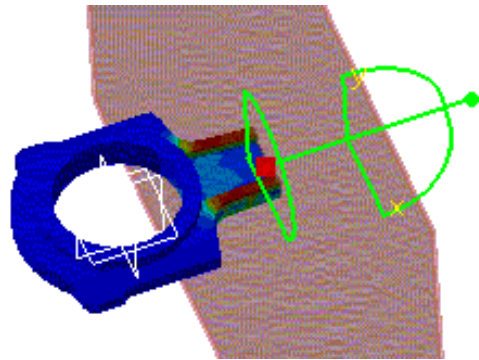
- The display is updated to show the magnitude of maximum shear stress (expressed in a plane bisecting the right angle between the directions of maximum and minimum principal stresses) at each point.



6. You can visualize the part inner principal stresses using the Cutting Plane: select Maximum in the combo and the "Cutting plane" checkbox (CATIA - P2 interface and licence only ).



You can handle the compass with the mouse in order to rotate or translate the Cutting Plane.( To do so, select an edge of the compass and drag the mouse ).



7. Click OK to quit the dialog box.



Up  
Visualizing Precisions

Visualizing Von Mises Stres Visualizing Displacements

Visualizing Principal Stresse Visualizing Normal Mode Sh

Editing the Color Palette

# Visualizing Normal Mode Shapes

Normal Vibration Modes are important in dynamics. Knowing the lowest mode shapes and frequencies of a part is useful for designing parts subjected to transient or harmonic loads (such as rotational machinery) to avoid resonance problems.



This task shows how to visualize the normal vibration modes on a part.



You must have successfully performed a Dynamics Analysis computation prior to this task.

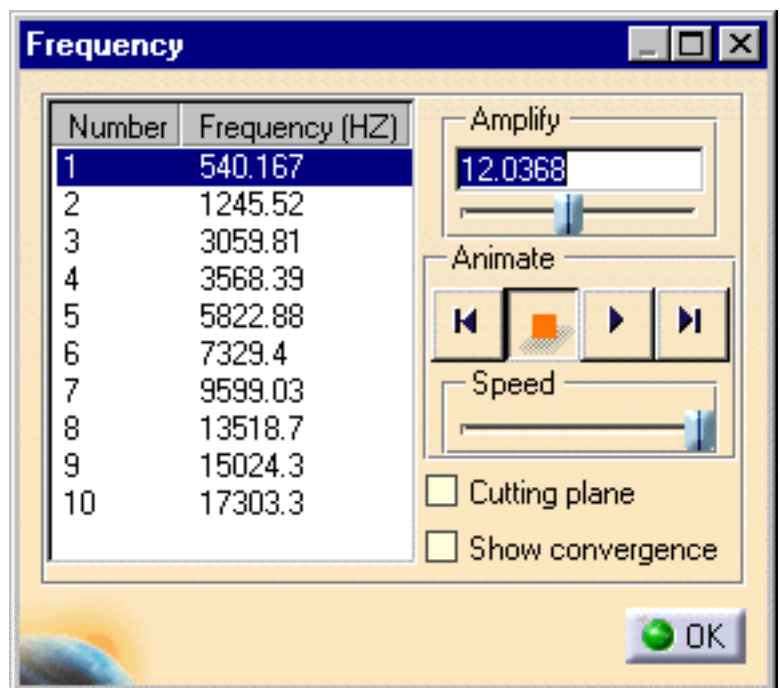
You can use the Rod\_For\_Analysis.CATPart document from the SAMPLES/gps\_analysis directory for this task.



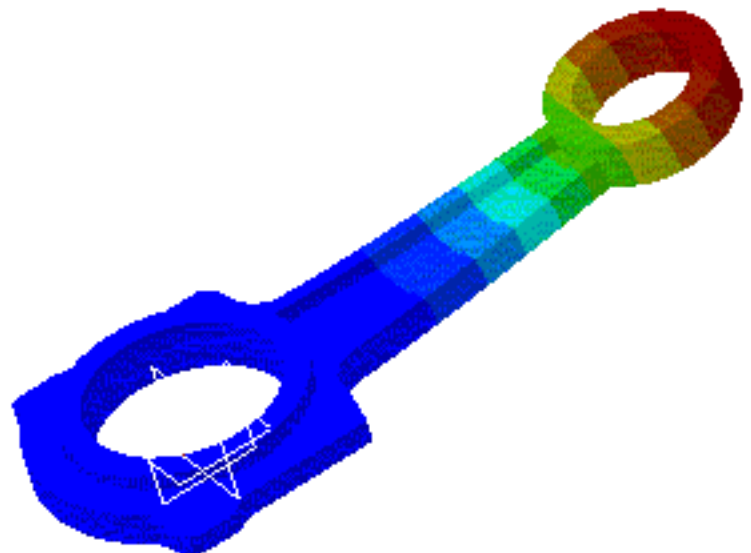
1. Click the Mode Results icon



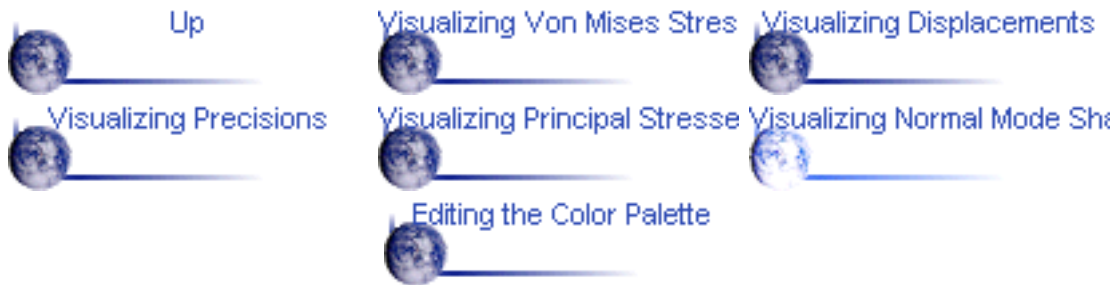
The Frequency dialog box is displayed.



The resulting modes are shown by the corresponding Mode Type displacement ISO-value colors:



2. To change the natural frequency mode solution, you can select the desired value from the list.
3. You can animate the selected mode by checking the Play Animation option. You can also display the part deformation step by step using the Next Step or Previous Step option.





## Editing the Color Palette

The von Mises stresses, the Displacements, the Precision, the Principal Stress distributions are employed along with a Color Palette. Editing the palette enables the user to emphasize on particular values spread on the parts.

This task shows how to edit the Palette on Von Mises display.




You must have successfully performed the computation using the Stress Analysis Workbench.

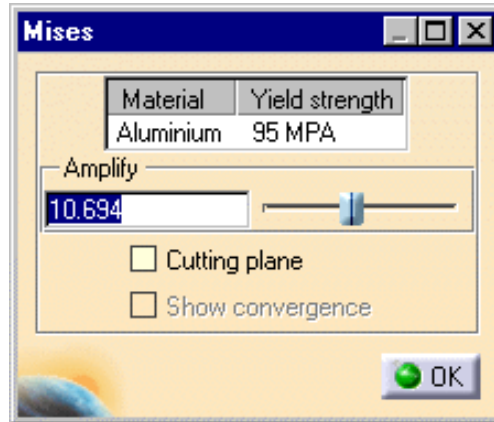
You can use the Rod\_For\_Analysis.CATPart document from the SAMPLES/gps\_analysis directory for this task.

1. Click the von Mises

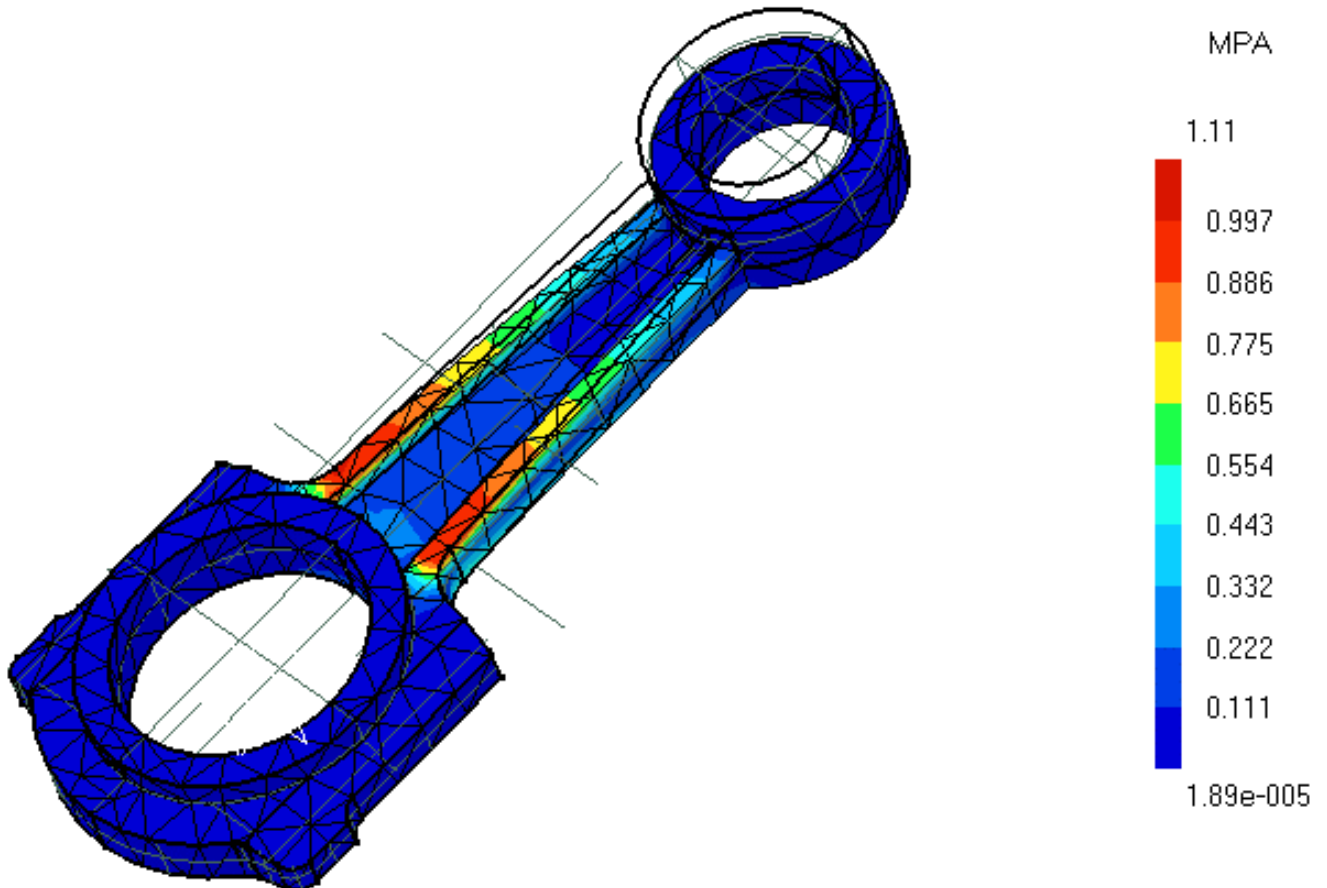


icon .

- The von Mises dialog box is displayed.



- The von Mises stress distribution on the part is also displayed in ISO-value mode, along with a color palette.





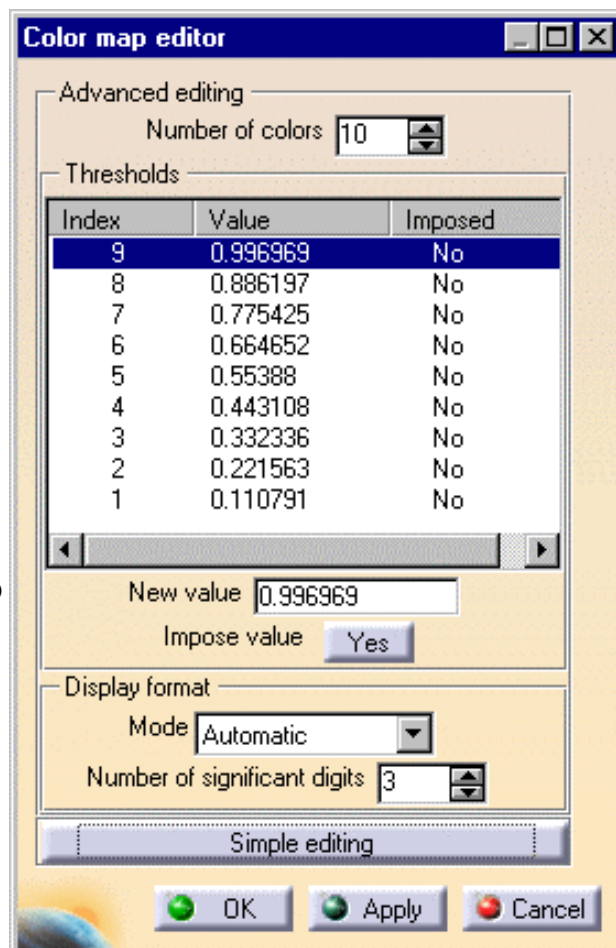
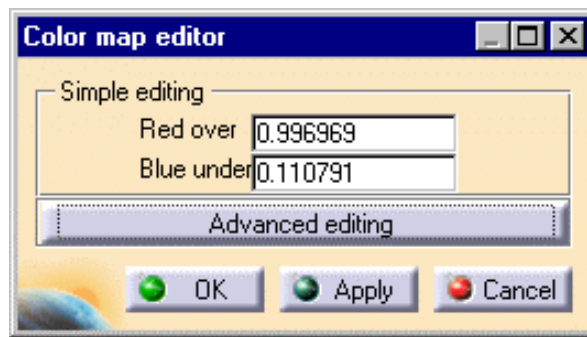
2. Double click on the Palette to edit it.

- The palette dialog box is displayed.

3. You can set the values defining the display of the red and blue colors.

4. Click on Advanced Editing.

- The panel is enlarged.
- A few parameters can be modified such as the Number of Colors, the Values which represent a threshold between two colors, and the display format for a Scientific or a decimal display with the corresponding number of decimal or significant digits.
- You can as well impose a particular value for a threshold in order not to modify it when setting other values.
- After each new value entered for a threshold, the list is computed to take into account the potential interactions between this threshold and the two other thresholds which flank it. If an interaction is detected, the former values are



distributed taking into account if possible the imposed values.

- When you have finished to customize the palette you can click on Apply or Ok to modify the view.

- None of those parameters are saved and are only effective on one display.

5. Modify the desired parameters.

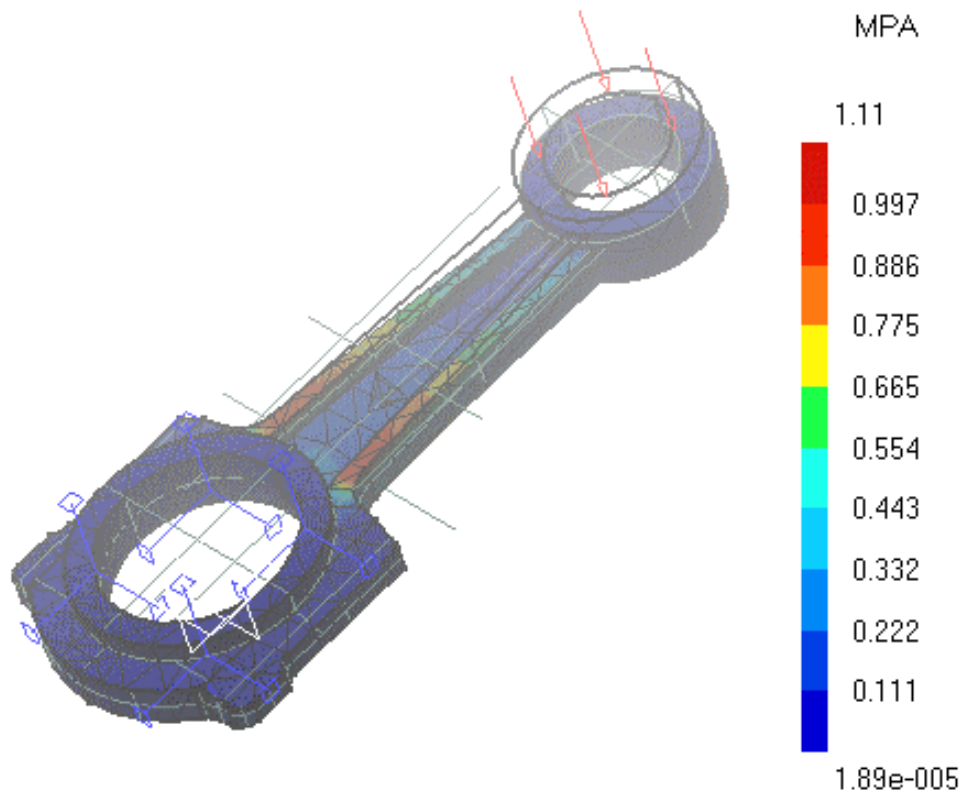
6. Click on Apply to check if the palette fit. If yes Click on Ok

- The palette dialog box disappears and the modifications will be valid only for this display.

7. You can move the Palette in the viewer: To do so, click on the Palette.



- The part's viewer is deactivated and the part is shaded.



8. Move the Color Palette with the middle mouse button to the desired place.

9. Click again on the Color palette to fix it there.



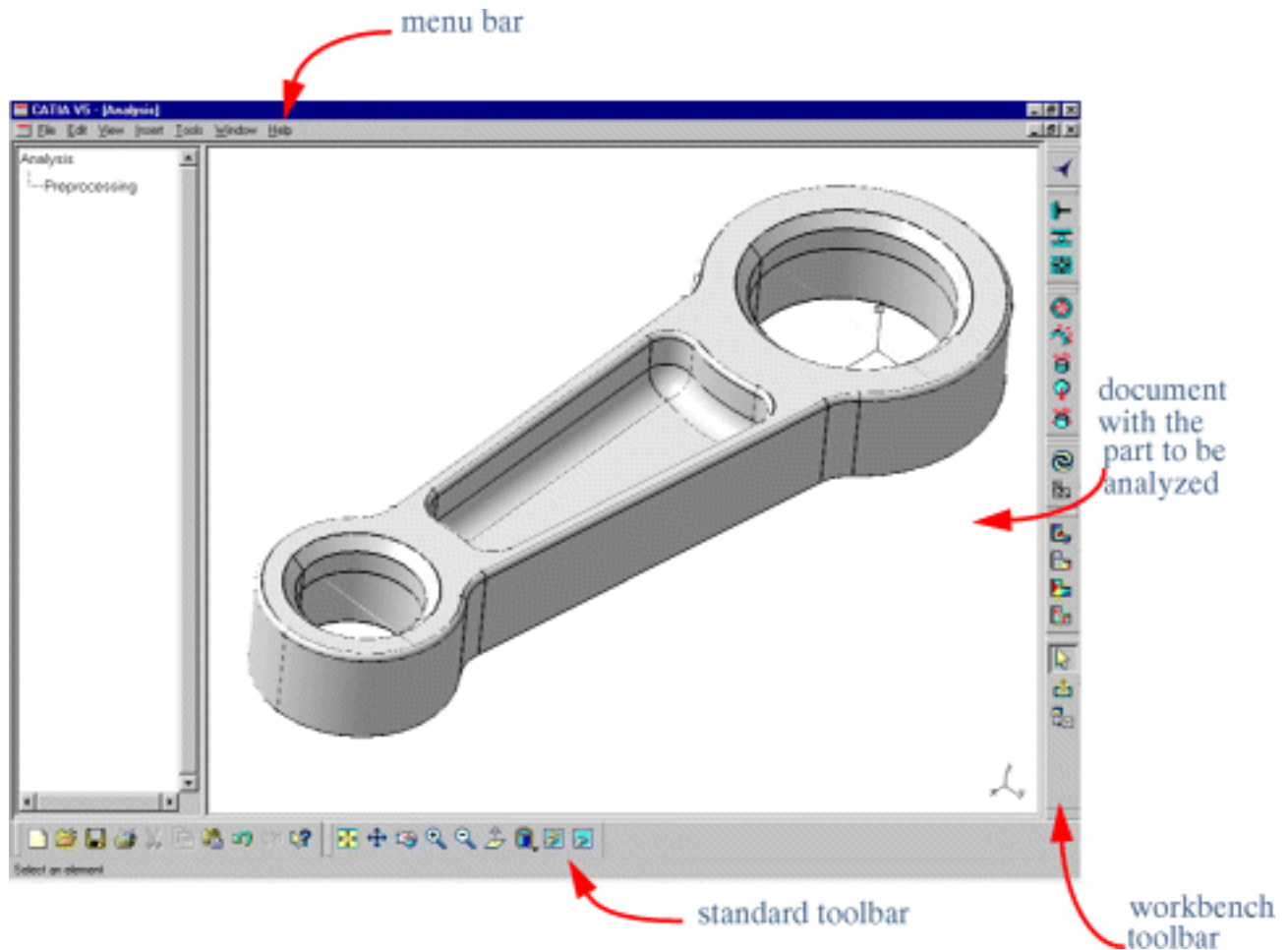
 Up  
 Visualizing Precisions

 Visualizing Von Mises Stress  
 Visualizing Principal Stresses  
 Editing the Color Palette

 Visualizing Displacements  
 Visualizing Normal Mode Shapes

# Workbench Description

This section contains the description of the icons and menus which are specific to the Stress Analysis and Dynamics Analysis workbenches of CATIA - Generative Part Structural Analysis Version 5.



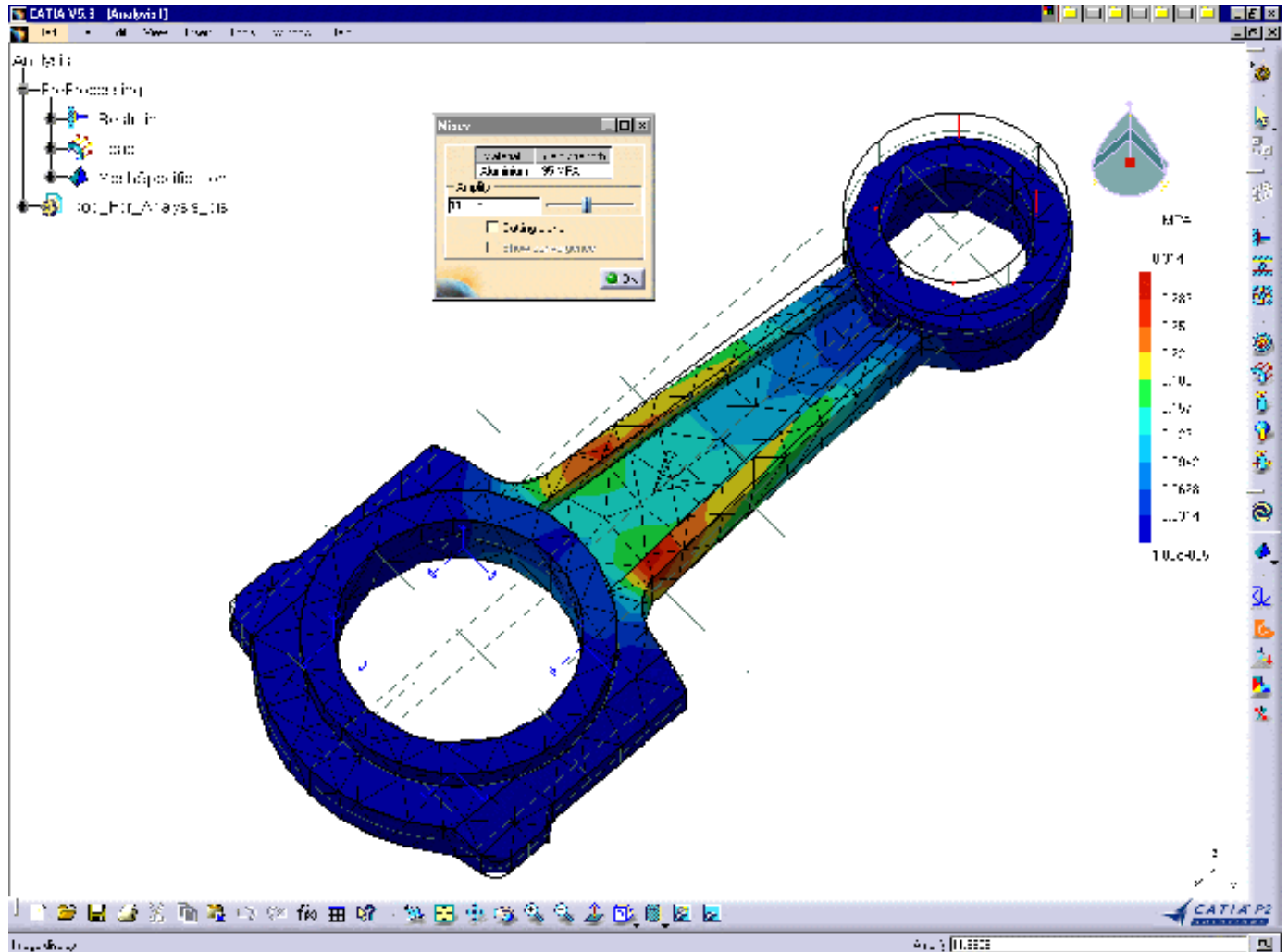
[CATIA - Generative Part Structural Analysis Menu Bar](#) (common to both Stress and Dynamic Workbench)

[CATIA - Generative Part Stress Analysis Workbench](#)

[CATIA - Generative Part Dynamic Analysis Workbench](#)

# CATIA - Generative Part Structural Analysis Menu Bar

In this chapter we will present the various menus and menu commands that are specific to CATIA - Generative Part Structural Analysis Version 5.

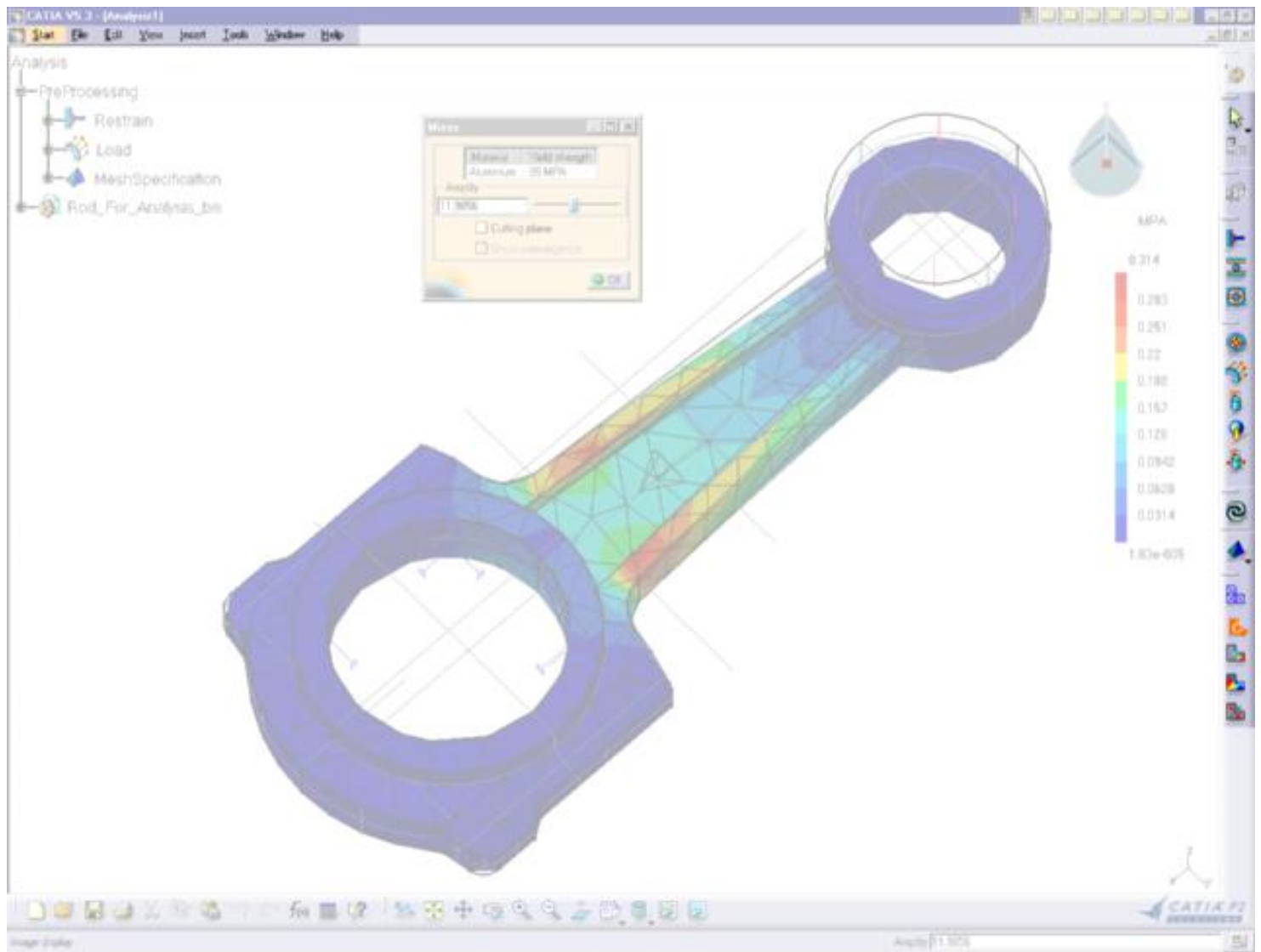


| Menu   | Purpose   |
|--------|---|
| Start  | Enter new workbench.  |
| File   | Perform file creation, opening, saving and printing operations. |
| Edit   | Manipulate selected objects.                                    |
| View   | View document contents.   |
| Insert | Insert objects.   |
| Tools  | Set user preferences.   |
| Window | Arrange document windows.                                       |
| Help   | Get help.   |

Tasks corresponding to the menu commands are described in the *CATIA Version 5 Infrastructure User's Guide*.



# Stress Analysis Workbench



Up



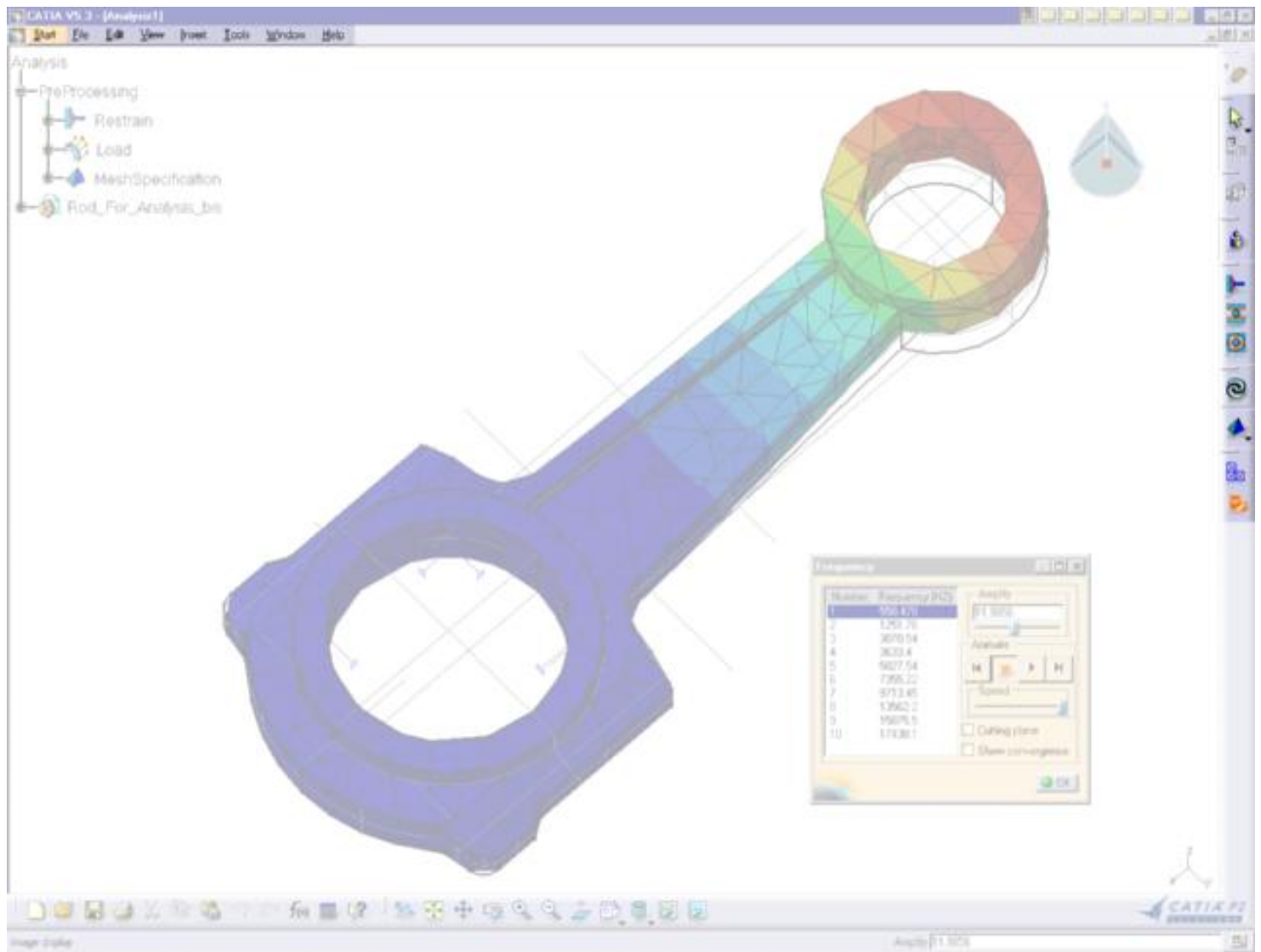
CATIA - Generative Part Str Stress Analysis Workbench



Dynamics Analysis Workber



# Dynamics Analysis Workbench



Up

CATIA - Generative Part Str Stress Analysis Workbench

Dynamics Analysis Workber

# Select Toolbar

The Select toolbar contains the following tools to select workbench, select part and load part.



Select geometry.



See [Entering the Stress Analysis Workbench and Selecting a Part](#)



# Connection Toolbar

The Connection toolbar contains the following tool to assign properties to a part. This toolbar is specific to the *Generative Assembly Structural Analysis* product.



See [Creating Connections](#)



# Equipment Toolbar

In the Dynamics Analysis workbench, the Equipment toolbar contains the following tool to create mass equipment.

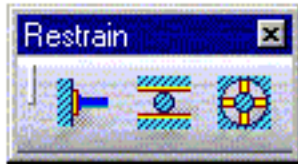


See [Creating Mass Equipment](#)



# Restraint Toolbar

The Restraint toolbar contains the following tools to restrain a part.



See [Creating Clamps](#)



See [Creating Sliders](#)



See [Creating Virtual Restraints](#)



# Load Toolbar

The Load toolbar contains the following tools to assign properties to a part.



See [Creating Pressures](#)



See [Creating Forces](#)



See [Creating Moments](#)



See [Creating Body Forces](#)



See [Creating Rotation Forces](#)



# Compute Toolbars

The Compute toolbars contain the following tools to compute analysis results.



See [Computing and Visualizing a High Precision Mesh](#)

See [Computing a Rough Stress Analysis](#)

See [Computing a High Precision Stress Analysis](#)

See [Computing a Normal Modes Analysis](#)

See [Computing an Assembly Model](#)



See [Computing a High Precision Stress Analysis](#)



See [Computing a High Precision Stress Analysis](#)



See [Refining and Computing a High Precision Stress Analysis](#)





# Image Toolbars

Depending on the workbench, the Image toolbar contains the following tools to visualize analysis results.



See [Computing and Visualizing a High Precision Mesh](#)



See [Visualizing Von Mises Stresses](#)



See [Visualizing Displacements](#)



See [Visualizing Precisions](#)



See [Visualizing Principal Stresses](#)



See [Visualizing Normal Mode Shapes](#)



# Glossary

## A

- axis system** A combination of three normal and unit sized vectors which defines a reference to express geometric entities coordinates. There are two different axis systems called:
- reference axis system (which corresponds to the model Axis System).
  - local axis system, whose vectors are normal or tangent to the selected geometry. The presence of material on one side of the selected geometry does not influence the choice of the vectors directions but the nature of the geometric element determinates whether the local axis system will be Cartesian, circular or revolute.

The reference axis system is symbolized at two locations: the bottom and right side of the workbench (without its origin) and at its real place: the model origin.

- Assembly** A set of parts, each part being associated with a material and linked possibly to another one by the means of a connection. Assemblies can be modeled with the product *CATIA - Generative Assembly Structural Analysis*.

## B

- body force** A load type including volume body force and mass body force. This load type is based on the body of the part (that is, its geometry and possibly its mass density). Therefore, body forces represent intensive (volume density-type) quantities, as opposed to forces which are extensive (resultants, i.e., integrals over regions) quantities.

## C

- clamp** A restraint applied to surface or line geometries of the part, for which all points are to be blocked (by imposing their translation value) in the subsequent analysis.
- connection** A set of constraints between parts at their common interface or a set of constraint modeled by the means of a virtual body between two parts. Using connections, the user can modelize an assembly prior to analyze it.

## CONSTR-N

A Finite Element type enabling points of a geometry that are linked together and free to translate in order to preserve the average behavior.

For example, imposing a translation to such a linked group enables all of the included points, free to translate differently than the imposed translation but the barycenter of the group must correspond to the imposed behavior: the imposed translation. According to the type of the imposed mechanical behavior (kinematic constraint or load) the corresponding kinematic, static or dynamic tensor will be respected at the barycenter of the selected group.

## contact element

This particular element is sufficient to modelize a smooth interface.

A Finite Element type enabling two linked points free to translate prevented that the linear contact condition is respected. Once the linear contact condition is reached, the contact element behaves like a RIG-BEAM element. The linked points are only free to translate along the two normal directions of the beam.

## F

## force

A force-type load, including tractions, distributed forces and forces transmitted through a virtual rigid body. The latter includes contact, rigid and smooth transmission types.

## M

## mass equipment

An additional mass attached to the geometry (point, line or surface) of the part. It represents a scalar, purely inertial (non-structural) load.

## moment

A transmitted moment-type load, which includes rigid and smooth transmission types.

## P

## part

A 3D entity obtained by combining different features in the Part Design workbench. Please see *CATIA - Part Design Users's Guide* for further information.

## pressure fitting

An assembly type which can be modelized with a [virtual restraint](#) or a [force](#), both transmitted through contact. Normal loads only can be applied or transmitted with such a modelization. So moment transmission through this interface cannot be analyzed.

## R

- resultant** For the CATIA - Generative Part Structural Analysis product, the resultant indicates an extensive quantity, an integral over a region, opposed to an intensive quantity which indicates a surface (or volume) density-type quantity.
- RIG-BEAM** A Finite Element type that rigidly links two points.

## S

- sag** Global sag is the general maximum tolerance between discretization and the real part used for the computation.
- Local sag is the maximum tolerance between discretization and the real part applied locally, to a chosen area of the model specified by the user.
- size** Global size is the general size of the longest edge of the finite elements used for the computation.
- Local size is an element size different to the general element size and applied locally, to a chosen area of the model specified by the user.
- SPRING** Finite element type which has an elastic behavior along all its degrees of freedom. This element modelize ideally elastic interfaces between parts.
- slider** A generalization of the clamp restraint in the sense that you can release some of the clamped directions thus allowing the part to slide along the released translation directions.
- storage (external)** An optional computation mode that enables the user to define a directory path where a temporary file will receive solver data during the computation.

## T

- traction** An intensive (surface density-type) quantity, as opposed to forces which are extensive (that is, [resultant](#)) quantities.

## V

- virtual restraint** A restraint applied indirectly to the part, through the action of a *virtual rigid body*. The interface specifications (smooth rigid or contact transmission) are selected by the user.

# Index

## A

Adaptivity ➤  
Apply Material icon ➤  
Assembly ➤ , ➤ , ➤ , ➤  
axis system ➤

## B

Body Force icon ➤ , ➤  
body forces ➤ , ➤

## C

Clamp icon ➤ , ➤  
clamps ➤ , ➤  
Color Palette ➤  
Compute icon ➤ , ➤ , ➤ , ➤ , ➤  
computing  
    high precision mesh ➤  
    normal modes ➤  
    stress analysis ➤  
    assemblies ➤  
CONSTR-N kinematic elements ➤ , ➤ , ➤ , ➤  
connection ➤ , ➤ , ➤  
Connection icon ➤  
contact element ➤  
creating  
    clamps ➤  
    connections ➤  
    distributed forces ➤  
    distributed mass equipment ➤

- forces ➤
- gravity forces ➤
- lumped mass equipment ➤
- moments ➤
- pressures ➤
- properties ➤
- rotation forces ➤
- sliders ➤
- tractions ➤
- transmitted forces ➤
- virtual restraints ➤
- volume body forces ➤

## D

- Displacement icon ➤ , ➤
- displacements ➤
- distributed forces ➤
- distributed mass equipment ➤
- dynamic vibration modes ➤
- Dynamics Analysis workbench ➤

## E

- Equipment icon ➤ , ➤ , ➤

## F

- finite element method ➤
- Force icon ➤ , ➤ , ➤ , ➤
- forces ➤

# G

gravity forces ➤

# L

linear contact condition elements ➤ , ➤

local axis system ➤

lumped mass equipment ➤

# M

mass body force ➤

mass equipment ➤ , ➤

material ➤

Mesh icon ➤

Mises icon ➤

Mode Results icon ➤

Moment icon ➤

moments ➤ , ➤

# N

non-rigid kinematic elements ➤ , ➤ , ➤

normal vibration modes ➤

# P

part ➤

precision ➤

Precision icon ➤

pressure ➤

pressure fitting ➤ , ➤

Pressure icon ➤

principal stresses ➤



Principal Stresses icon ➤

Properties ➤

## R

reference axis system ➤

resultant ➤

RIG-BEAM kinematic elements ➤ , ➤ , ➤ , ➤

rigid kinematic elements ➤ , ➤ , ➤

rotation forces ➤

Rotation icon ➤

## S

SAG ➤

Select Part icon ➤

size ➤

Slider icon ➤ , ➤

sliders ➤ , ➤

SPRING ➤ , ➤

storage ➤

Stress Analysis workbench ➤

## T

tractions ➤ , ➤

transmitted forces ➤

## V

Virtual Restraint icon ➤

virtual restraints ➤ , ➤

virtual rigid body ➤

visualizing

displacements ➤

normal mode shapes ➤

precisions ➤

principal stresses ➤

Von Mises stresses ➤

volume body force ➤

Von Mises stresses ➤