Getting Started

This tutorial will guide you step-by-step through your first **ELFINI and Generative Part Structural Analysis** session, allowing you to get acquainted with the product. You just need to follow the instructions as you progress.

These tasks should take about 20 minutes to complete.

Entering the Structural Analysis Workbench

This first task will show you how to load a .CATPart document (and display the corresponding specification tree) by entering the **Generative Structural Analysis** workbench and defining that you will create a Static Analysis case.

Creating a *static analysis case* means that you will analyze the static boundary conditions of the CATAnalysis document one after the other.

Before you begin:

- Note:
  In this example, a material has been previously assigned to the part you will open. In the case no material has been previously assigned to the part, before entering the **Generative Structural Analysis** workbench, you should proceed as follows:

  1. Select the part in the specification tree.
  2. Click the Apply Material icon .
     The Material library appears.
  3. Select a material family, then 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 specification tree and using `Edit -> Properties -> Analysis.`

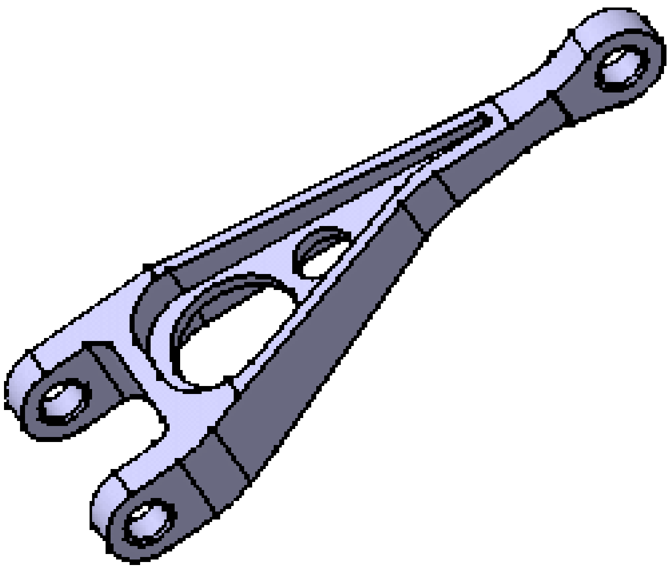
  If you select `Start->Analysis & Simulation -> Generative Structural Analysis` from a CATPart document containing a part without any material assigned, the material library will appear directly for an easy material selection.

- Warning:
  Avoid having CATAnalysis documents automatically saved. For this, go to `Tools->Options->General` (menu bar) and de-activate the `Automatic save every xx minutes` option. Otherwise, on some models, each computation will be followed by a Save, thus making temporary data become persistent data.

Open the CATPart Document

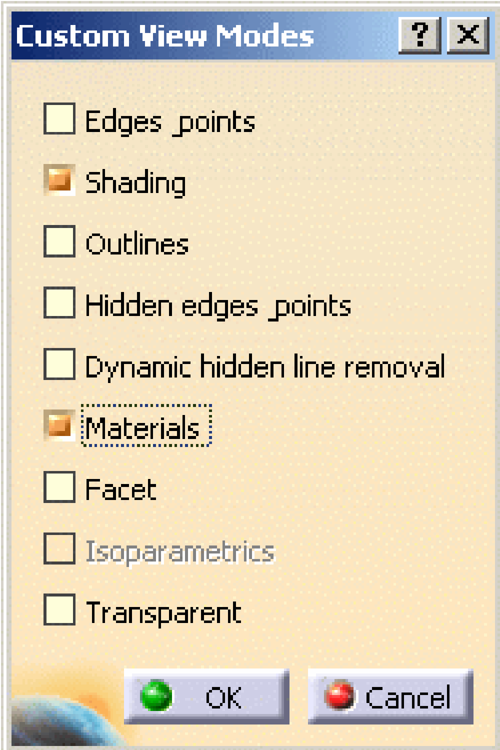
1. Select `File -> Open,` then select the desired .CATPart file. In this tutorial, you will open CATPart named sample01.CATPart.

This opens a **Part Design** document containing the selected part.



Define the View Mode

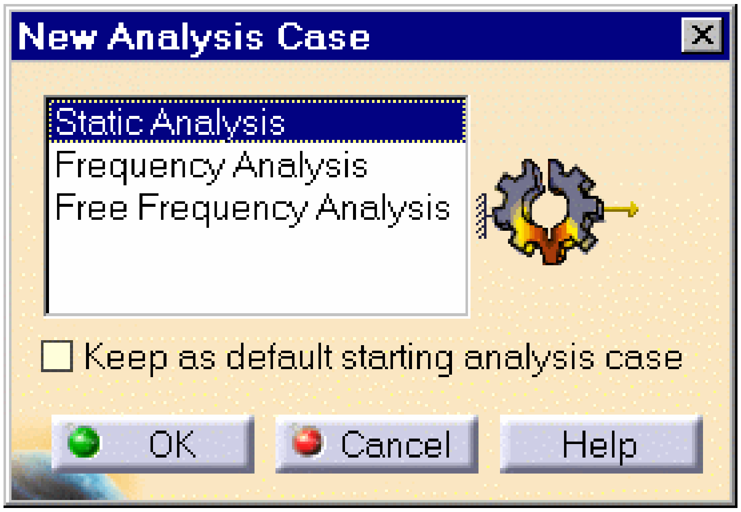
2. Select `View -> Render Style -> Customize View` option from the toolbar and activate the `Materials` option from the displayed Custom View Modes dialog box.



Enter Generative Structural Analysis Workbench

3. Select `Start -> Analysis & Simulation -> Generative Structural Analysis` from the menu bar.

The New Analysis Case dialog box appears with `Static Analysis` as default type.



`Static Analysis` means that you will analyze the static boundary conditions of the CATAnalysis document one after the other.

 `Frequency Analysis` means that you will analyze the dynamic boundary conditions of the CATAnalysis document .

`Free Frequency Analysis` means that you will analyze the buckling dynamic conditions of the CATAnalysis document.

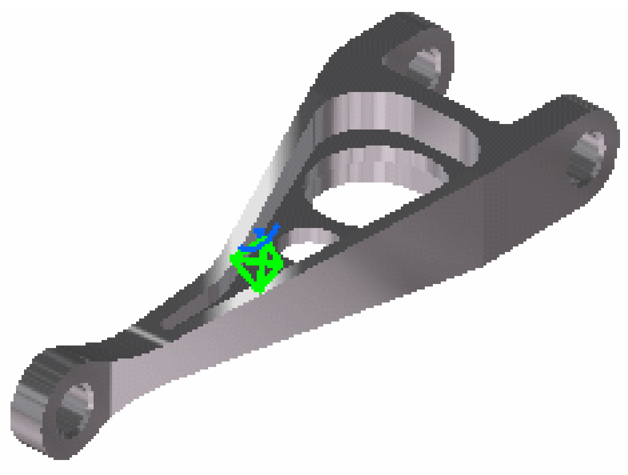
`Keep as default starting analysis case` means that when you next open the `Generative Structural Analysis` workbench from the menu bar, the selected case appears as default.

4. If needed, select an Analysis Case type from the New Analysis Case dialog box. In this particular case, also keep `Static Analysis` type selected.

5. Click `OK` in the New Analysis Case dialog box to enter the workbench.

The **CATAnalysis document** now opens. It is named `Analysis1`. You will now perform different operations in this document.

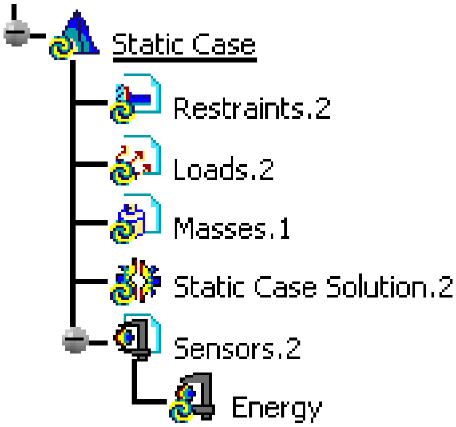
A link exist between the CATPart and the CATAnalysis document.



Double-clicking on the green symbol allows displaying mesh specifications or setting meshing parameters.

The standard structure of the Analysis **specification tree** is displayed.

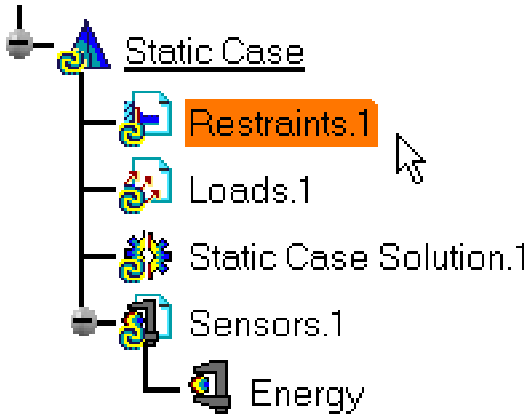
As you can see below, the `Finite Element Model` contains a `Static Case`, which contains empty `Restraints` and `Loads` objects sets, along with an empty `Static Case Solution.1` object set. All along this tutorial, you will assign a Restraint and a Load to the CATAnalysis document and then compute the Static Case Solution.


Static Case
Restraints.2
Loads.2
Masses.1
Static Case Solution.2
Sensors.2
Energy

Creating a Surface Slider Restraint

This task will show you how to restrain several faces of your part in such a way that it can only slide along their tangent planes (geometry supports). You will create a surface slider restraint on a Finite Element Model containing a Static Analysis Case.

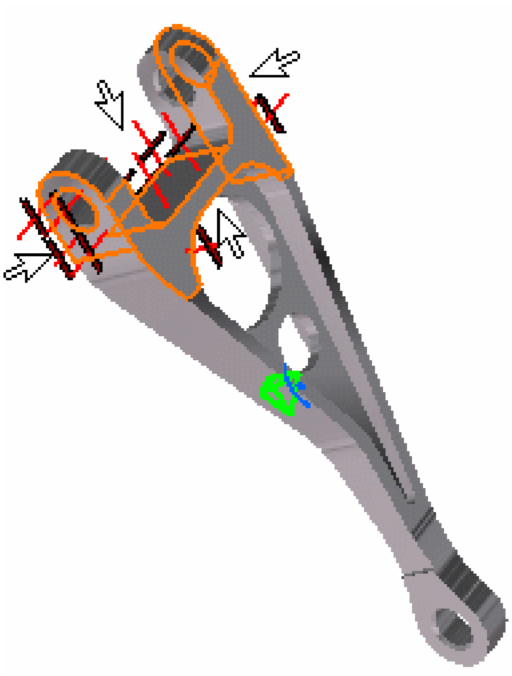
1. Select the `Restraints.1` object in the specification tree to make it active.



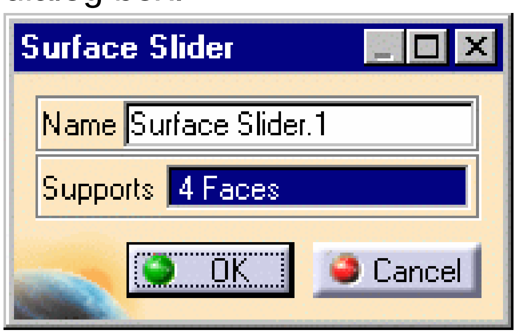
2. Click the Surface Slider icon .

The Surface Slider dialog box appears.

3. Select in sequence the four faces as indicated.

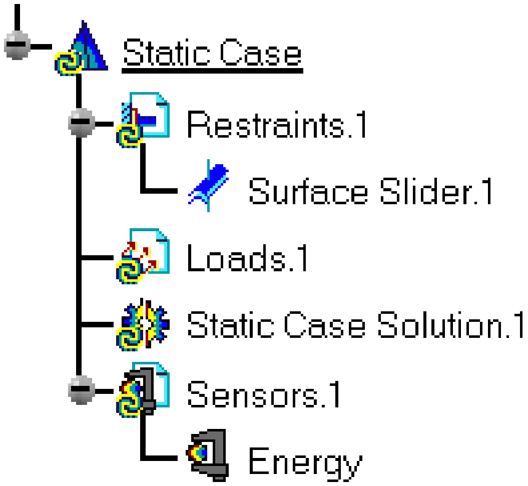


Symbols representing the surface sliders appear as you select the four faces. The elements supporting the surface slider are automatically displayed in the Surface Slider dialog box.



4. Click `OK` in the Surface Slider dialog box to actually create this surface slider.

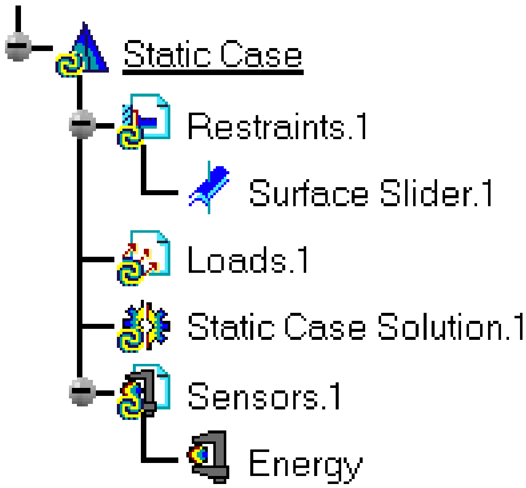
In the specification tree, the `Surface Slider.1` object has been inserted under the `Restraints.1` object.

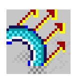

Static Case
Restraints.1
Surface Slider.1
Loads.1
Static Case Solution.1
Sensors.1
Energy

Creating a Distributed Force Load

This task will show you how to distribute on a face of your part a resultant force. You will create a Distributed Force on a Finite Element Model containing a Static Analysis Case.

1. Select the `Loads.1` object in the specification tree to make it active.



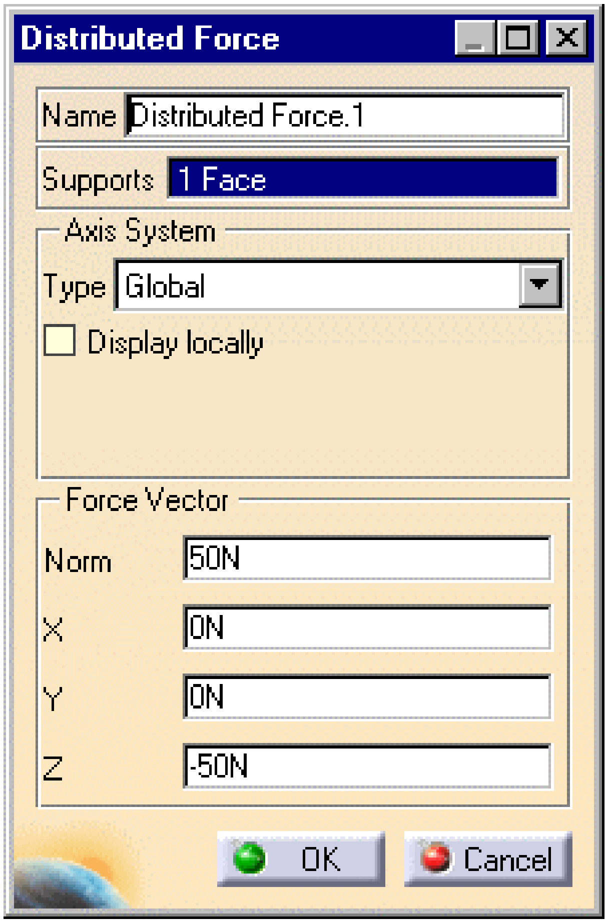
2. Select the Distributed Force icon  .

The Distributed Force dialog box appears.

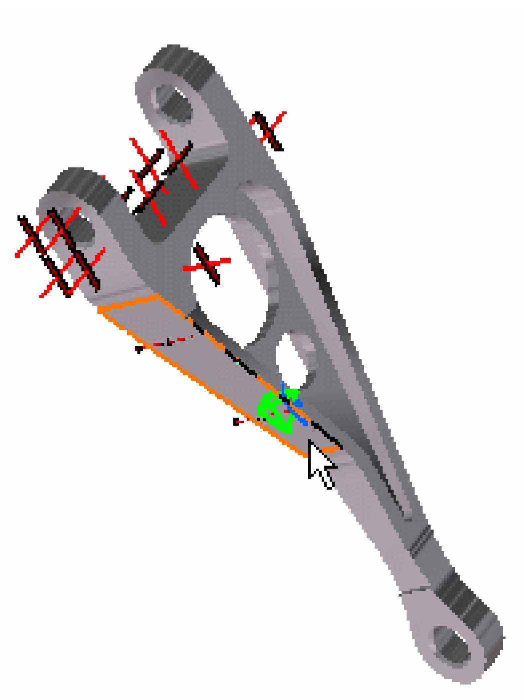
You will distribute on a face of your part a resultant force of 50N parallel to the global z-direction applied at the centroid of the face. For this:

3. Enter `-50N` value in `Z` field (`Force Vector`).

The resultant `Force Vector Norm` field is automatically updated.



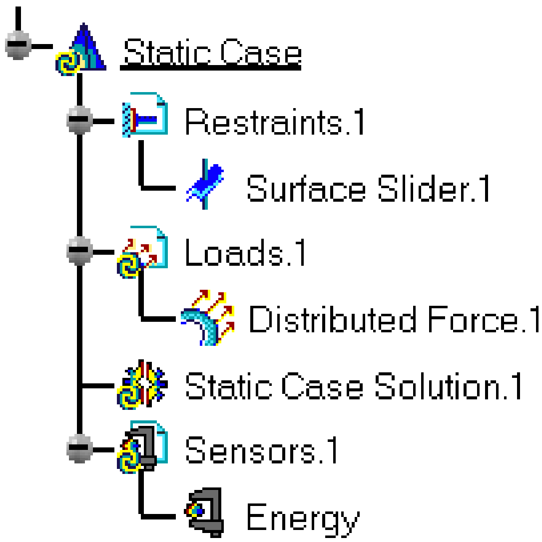
4. Select the part face as indicated below.



A symbol (arrow) representing the Distributed Force is displayed.

5. Press `OK` in the Distributed Force dialog box to create the Distributed Force.

The `Distributed Force.1` object has been inserted under the `Loads.1` objects set in the specification tree.

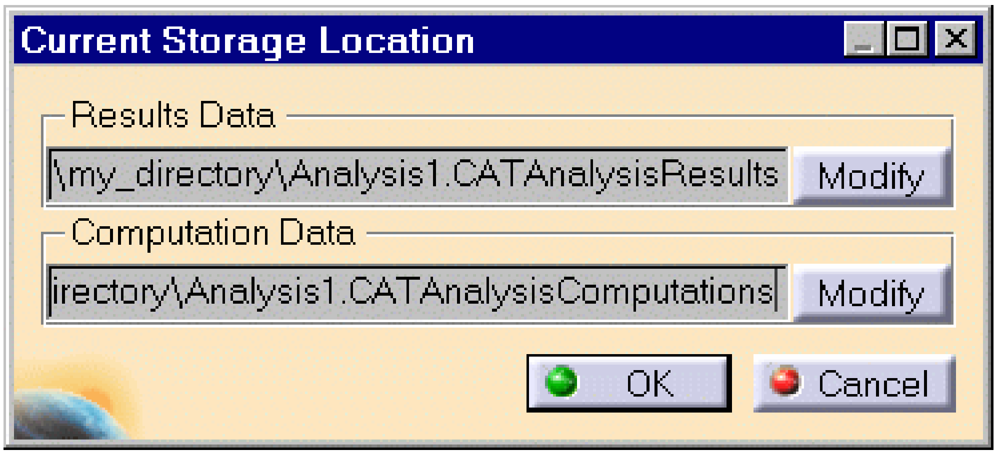


Computing a Static Case Solution

This task will show you how to compute the Static Case Solution of a Finite Element Model on which you previously created a Restraint object and a Load object. You will store the results in a given directory.

1. Select the Storage Location icon .
The Current Storage Location dialog box is displayed.

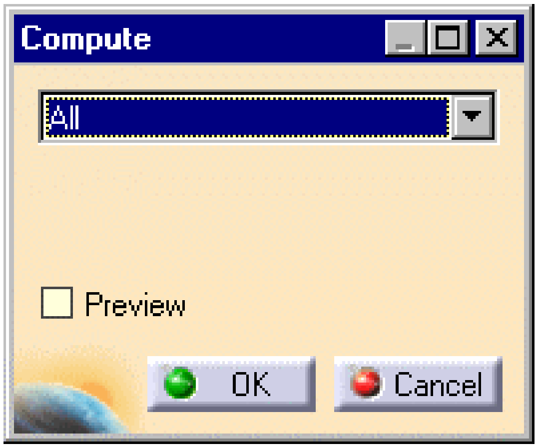
2. If needed, change the path of the `Result Data` and/or `Computation Data` directories and then click `OK` in the Current Storage Location dialog box.



The Results and Computation Data are stored in one single file with given extensions:

- `xxx.CATAnalysisResults`
- `xxx.CATAnalysisComputations`

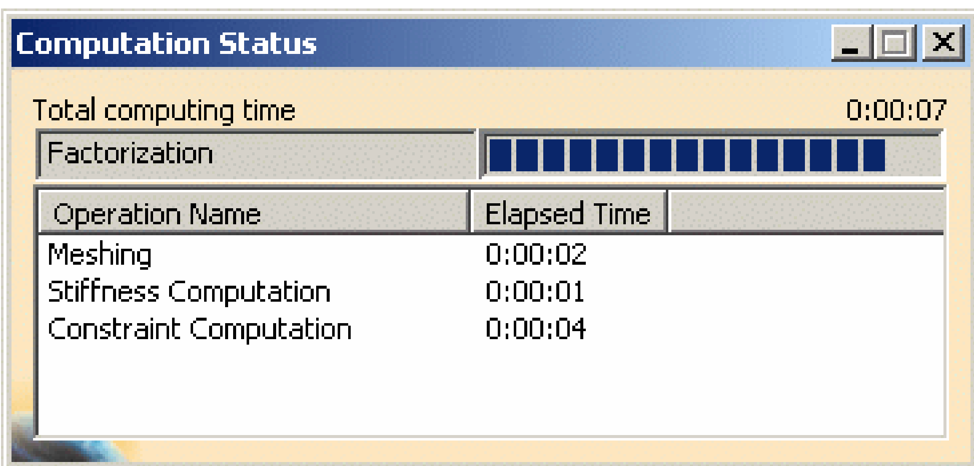
3. Select the Compute icon .
The Compute dialog box appears.
The Current Storage Location dialog box disappears.



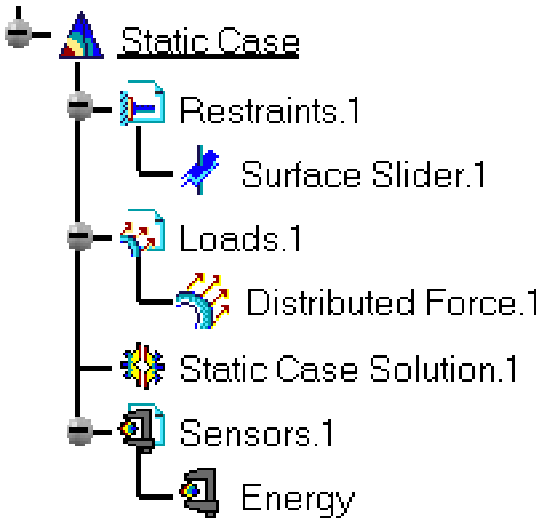
4. Select the `All` default value proposed for defining which are the objects sets to be updated.

5. Click `OK` in the Compute dialog box to launch the computation.

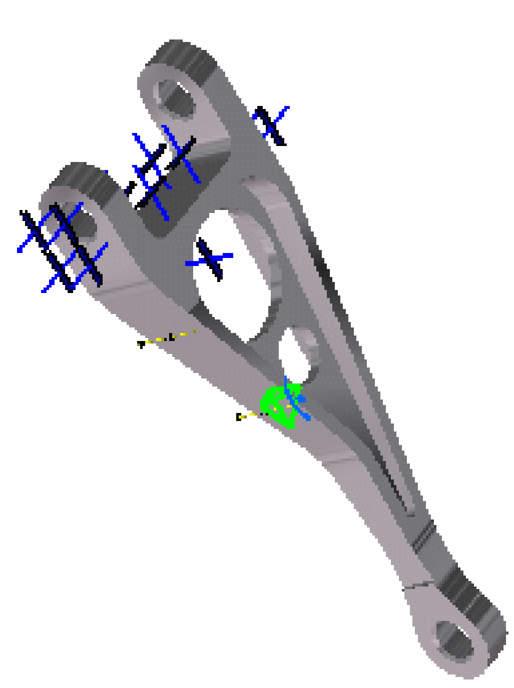
The Progress Bar dialog box provides a series of status messages (`Meshing`, `Factorization`, `Solution`) that inform you of the degree of advancement of the computation process.



- Upon successful completion of the computation, the status of all objects in the analysis specification tree up to the `Static Case Solution.1` objects set is changed to valid. In other words, the symbol appears no more.



The color change of the Restraints and Loads symbols to blue, also reflecting the fact that the Static Case Solution computation was successful.



Viewing Displacements Results

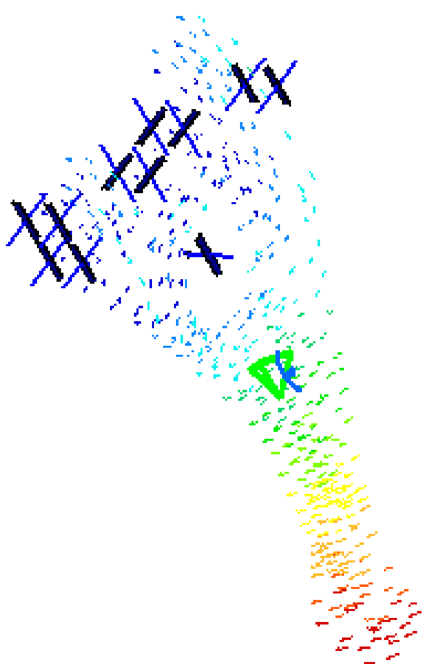
This task will show you how to visualize the displacements of the CATAnalysis according to the restraints and load you assigned to this CATAnalysis. You previously launched the computation of the Static Analysis Case and will now generate a Report with computations of the displacements you are going to perform:

- Displacement
- Stress Von Mises

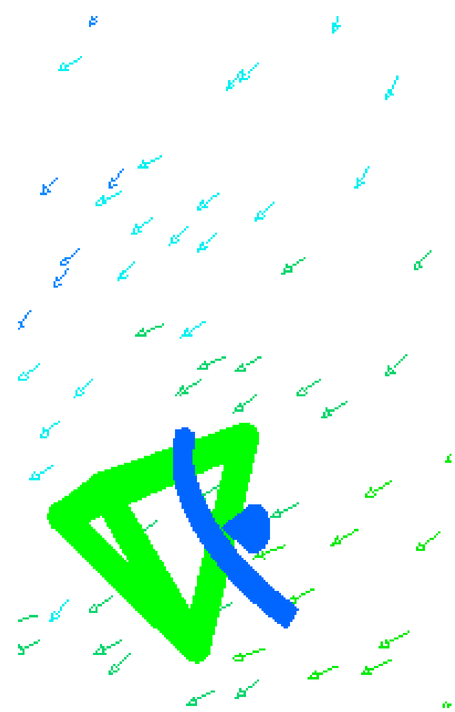
1. Click the Displacement icon .

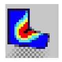
A plot of the displacement field is displayed with arrow symbols. If you go over the plot with the cursor, you can visualize the nodes. The computed displacement field can now be used to compute other results such as strains, stresses, reaction forces and so forth.

Arrows representing the displacement:

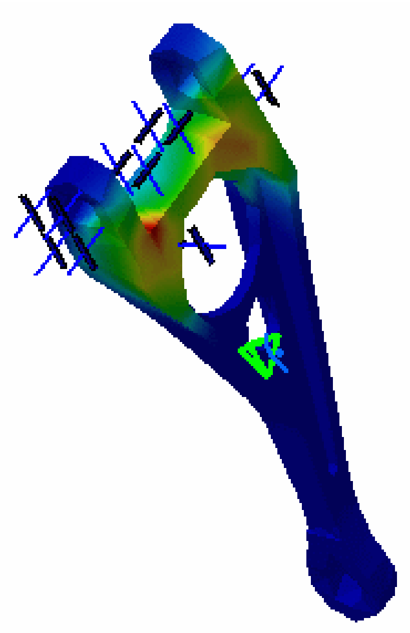


Zoomed arrows:

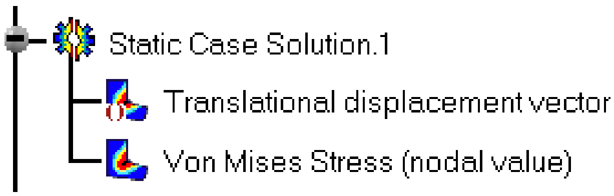


2. Click the Stress Von Mises icon  .

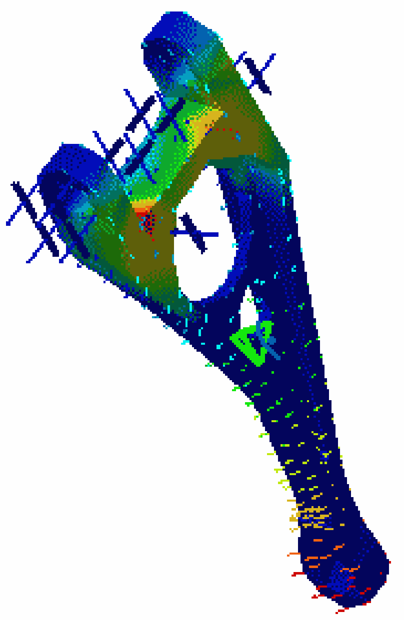
Go to `View -> Render Style -> Customize View` and make sure the `Materials` option is active in the Custom View Modes dialog box.



Both a `Translational displacement vector` image object and a `Von Mises Stress (nodal value)` image object appear in the specification tree under the `Static Case Solution.1` objects set.

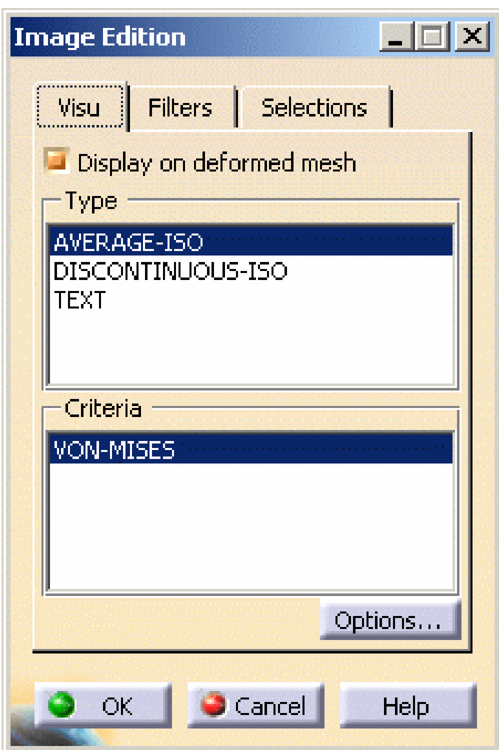


You can choose to have both `Translational displacement vector` and `Von Mises Stress (nodal value)` deformed mesh displayed. For this, right-click on `Translational displacement vector` in the specification tree and select the `Activate/Deactivate` option that is displayed in the contexual menu.

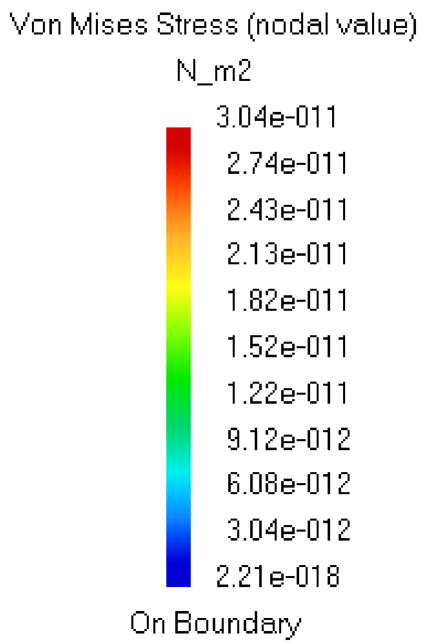


3. Double-click the Von Mises Stress feature in the specification tree to edit the image.

The Image Edition dialog box appears.

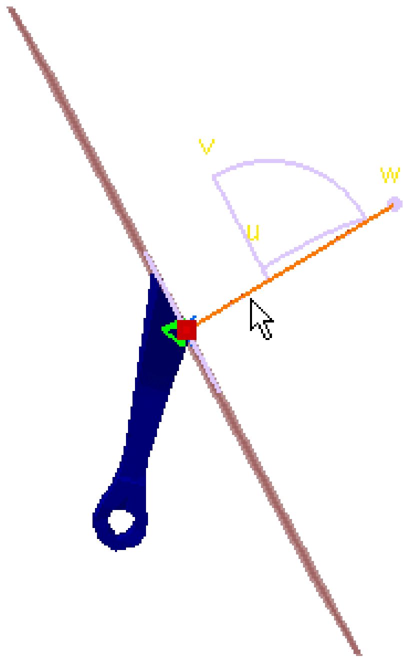


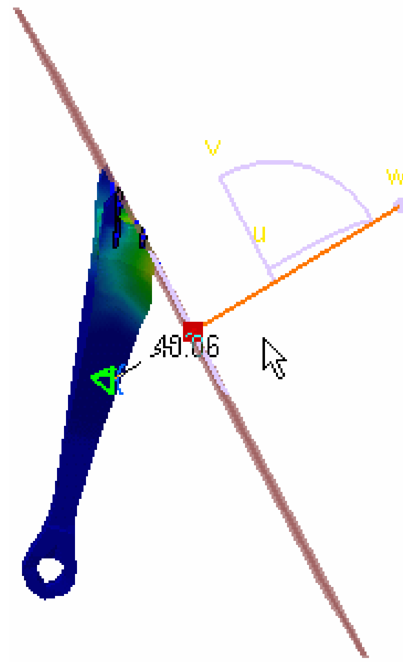
- **Color value:**
  The Color Palette enables you to modify the color distribution and to focus on specific values.

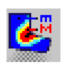


For more details on this functionality, refer to the task Editing the Color Palette.

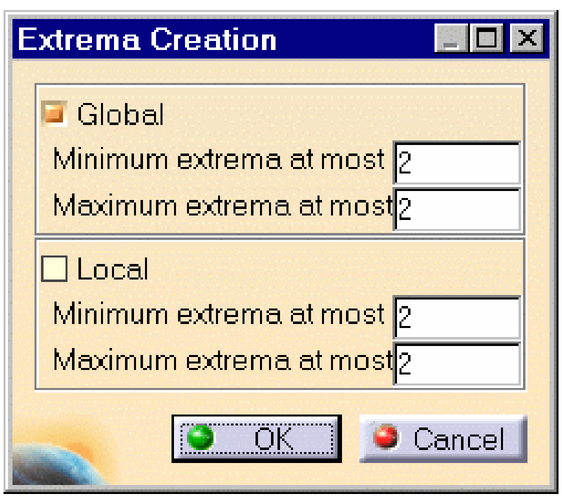
- **Internal von Mises stress field values**
  To visualize internal von Mises stress field values in a plane section through the part, click the Cut Plane Analysis icon  .
  You can handle the compass with the mouse in order to rotate or translate the Cutting Plane (to do so, select an edge of the compass and drag the mouse).
  To exit this view, click `CLOSE` in the Cut Plane Analysis dialog box that appeared.





4. Click the Search Image Extrema icon  to obtain local and global extrema values of the von Mises stress field magnitude.

The Extrema Edition dialog box appears.



Click `OK` once you have defined the number of the extrema you need. In this particular case, you will define that you need two Absolute extrema.

Locations of the global maxima and minima are indicated on the image, and the `Extrema` object appears in the specification tree under the `Static Case Solutions` objects set.

As you can see above, the values are not satisfying for our static case: you need more distributed force so that the Stress Von Mises values might be more significant. You will therefore save the document, modify the values and re-compute the static case in the following task.

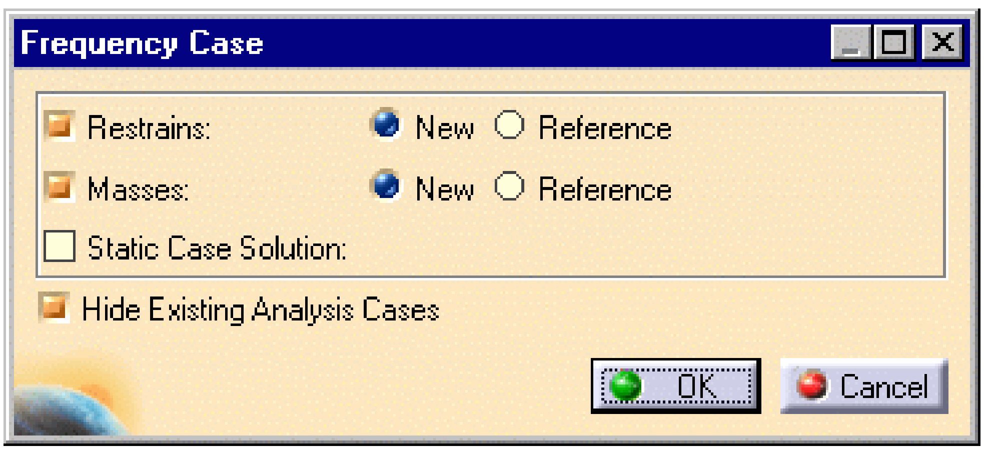
Inserting a Frequency Analysis Case

This task will show you how to insert a Frequency Analysis Case using the CATPart document called sample01.CATPart.

Creating a *frequency analysis case* means that you will analyze the dynamic boundary conditions of the CATAnalysis document.

Before you begin:
Remember that we use the Materials view mode. If needed, go to `View -> Render Style -> Customize View` option from the toolbar and activate the `Materials` option from the displayed Custom View Modes dialog box.

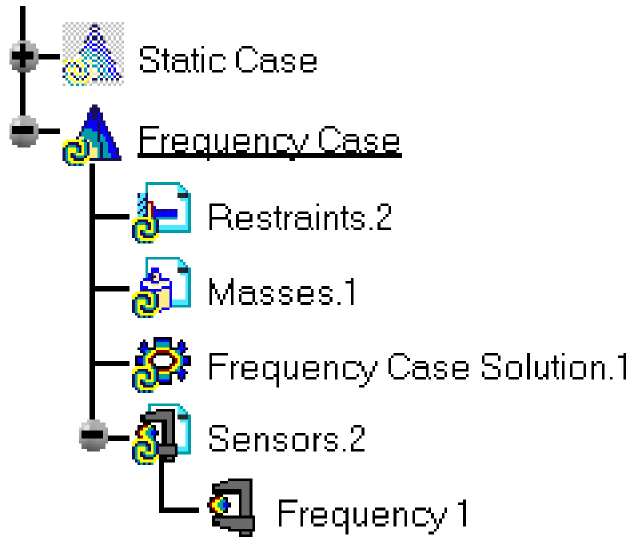
1. Select `Insert -> Frequency Case` from the menu bar.
The Frequency Case dialog box appears with the possibility to either use the existing Analysis case as Reference or create a Frequency case with New feature.



2. Click `OK`.

A new Analysis solution and the standard structure of Analysis specification tree is displayed.

The Finite Element Model contains a Frequency Case, which contains empty Restraints and Masses object sets, along with an empty `Frequency Case Solution.1` object set.

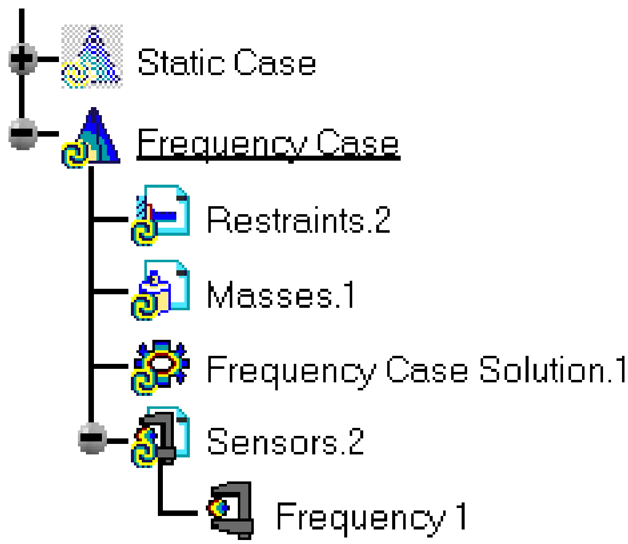


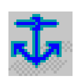
Remember that if you selected `Start->Analysis & Simulation -> New Generative Analysis` from a CATPart document containing the part without any material, the material library will appear directly for an easy material selection.

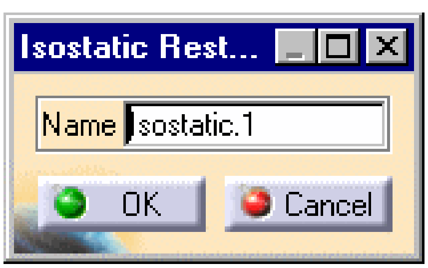
Creating an Iso-static Restraint

This task will show you how to create an Iso-static Restraint on a part. In other words, you will apply statically definite restraints allowing you to simply support a body.

1. Select the `Restraints.2` object in the specification tree to make it active.



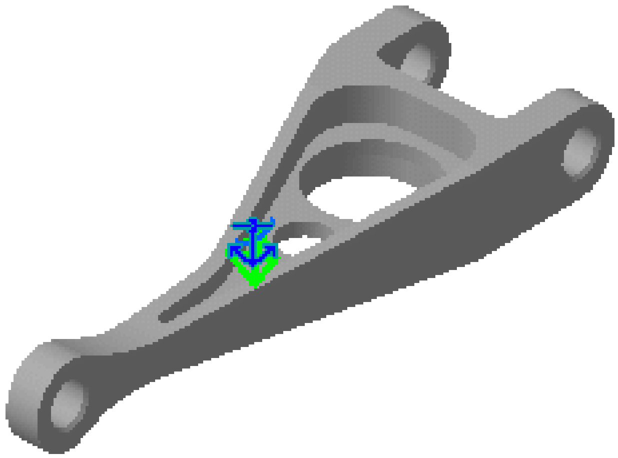
2. Click the Isostatic Restraint icon  .
The Isostatic Restraint dialog box appears.



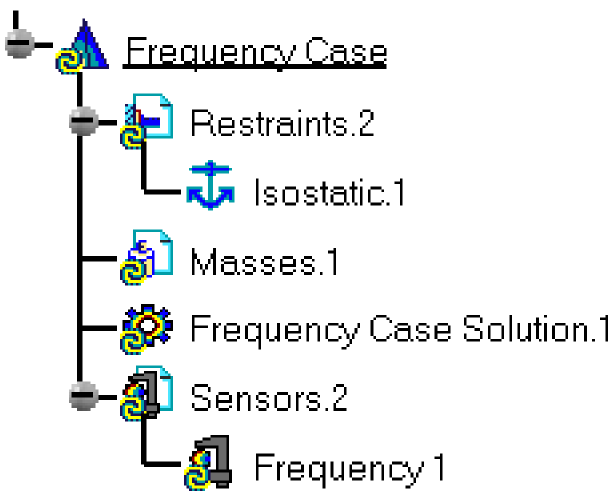
You will restrain your part in such a way that it is statically definite and all rigid-body motion is impossible. The program will automatically determine the restrained points and directions.

3. Click `OK` in the Isostatic Restraint dialog box to create the Iso-static Restraint.

The Isostatic symbol appears on the part.



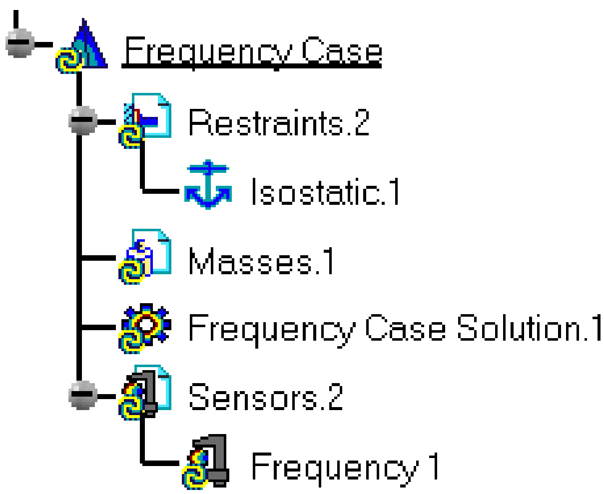
The Iso-static Restraint object `(Isostatic.1)` has been inserted under the `Restraints.1` objects set in the specification tree.



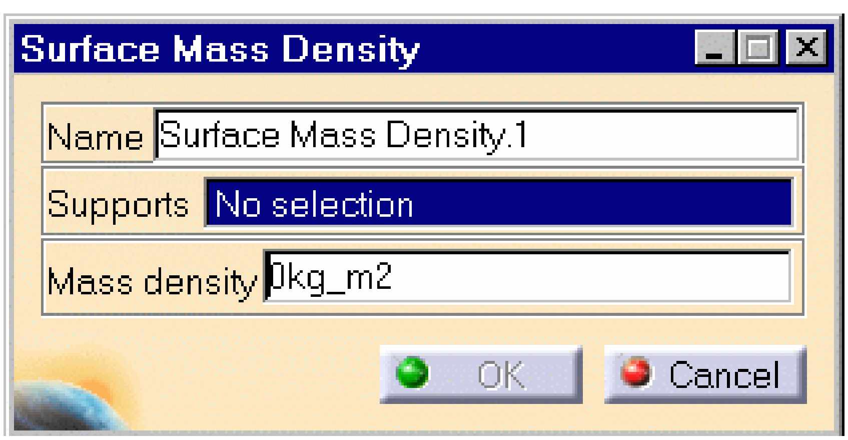
Creating a Non-Structural Mass

This task will show you how to create a Mass Surface Density on the surface geometry supports. In this example, you will distribute a mass density of 50kg/m2 on several faces of your part.

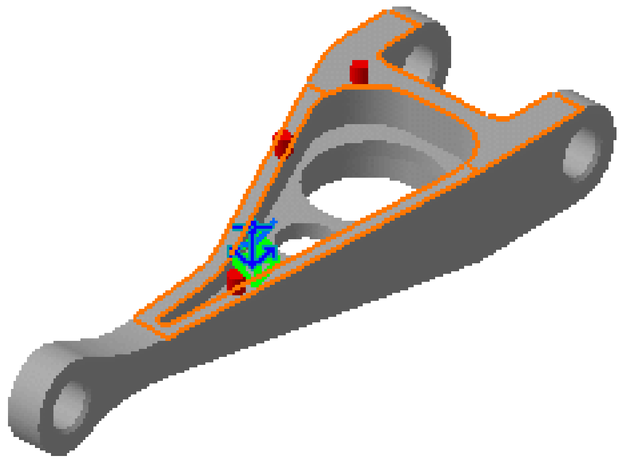
1. Select the `Masses.1` objects set in the specification tree to make it active.



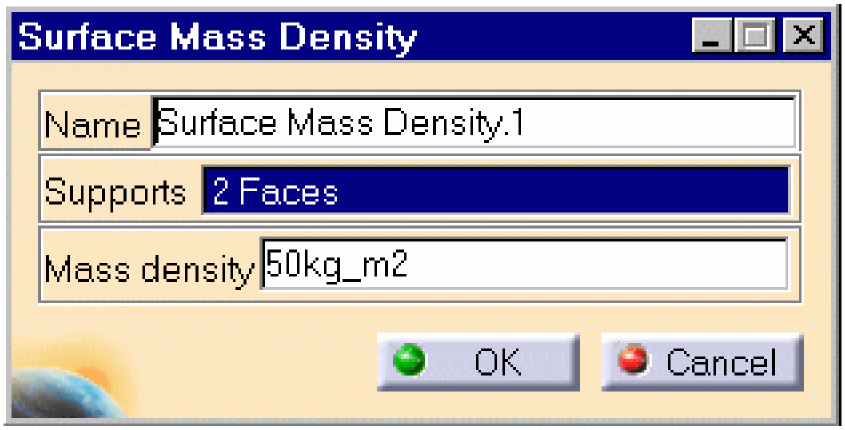
2. Click the Surface Mass Density icon .
The Surface Mass Density dialog box appears.



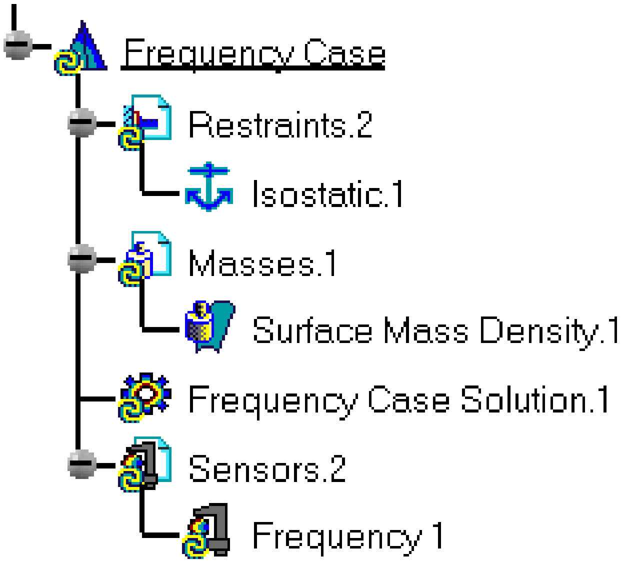
3. Select the faces on which you will distribute a mass density.
Red symbols representing the Mass Surface Density are displayed.



4. Enter a new `Mass Density` in the Surface Mass Density dialog box. In this particular case, enter `50kg_m2.`



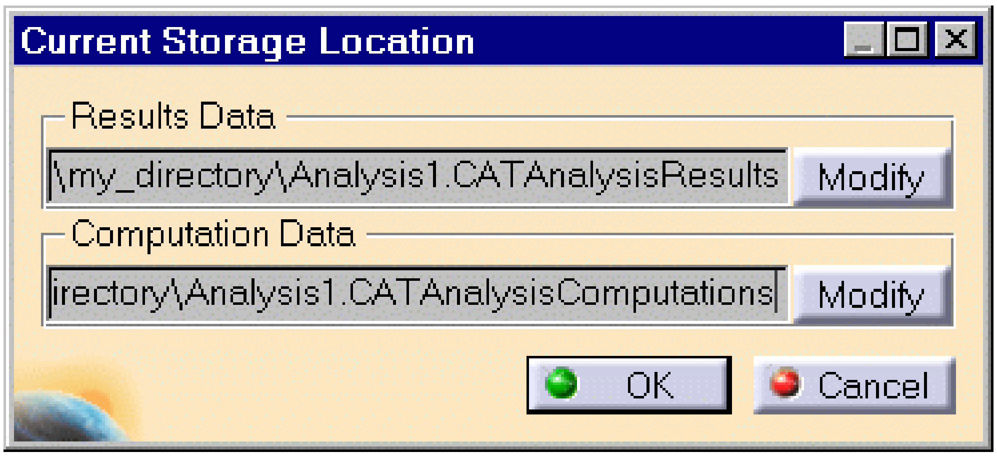
5. Click `OK` in the Surface Mass Density dialog box.
The `Surface Mass Density.1` object is now inserted under the `Masses.1` object set in the specification tree.


Frequency Case
Restraints.2
Isostatic.1
Masses.1
Surface Mass Density.1
Frequency Case Solution.1
Sensors.2
Frequency 1

Computing the Frequency Case Solution

This task will show you how to compute a Frequency Case Solution on which you previously created a Restraint object and optionally a Mass object.

1. Click the Storage Location icon .
The Current Storage Location dialog box is displayed.

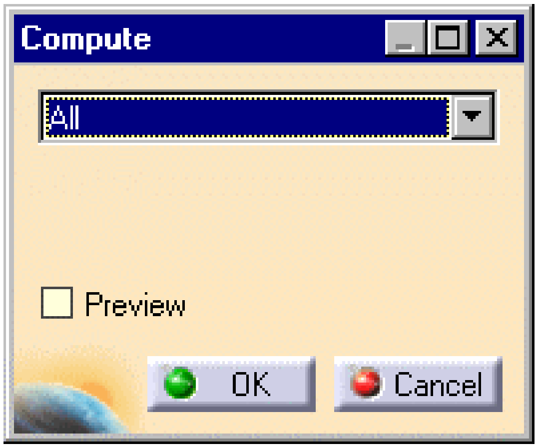


Optionally change the path of the External Storage directory to another directory and then click `OK` in the Current Storage Location dialog box.

The results and computation data are stored in one single file with given extensions:

- `xxx.CATAnalysisResults`
- `xxx.CATAnalysisComputations`

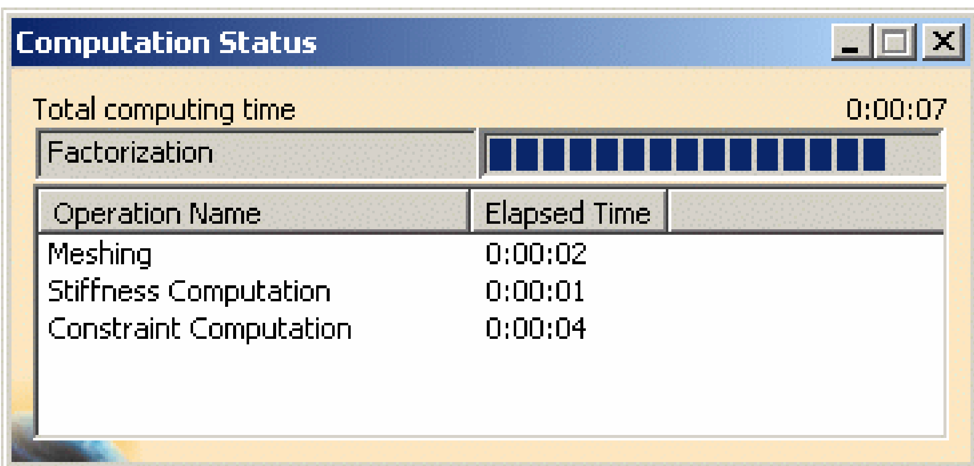
2. Click the Compute icon .
The Compute dialog box appears.



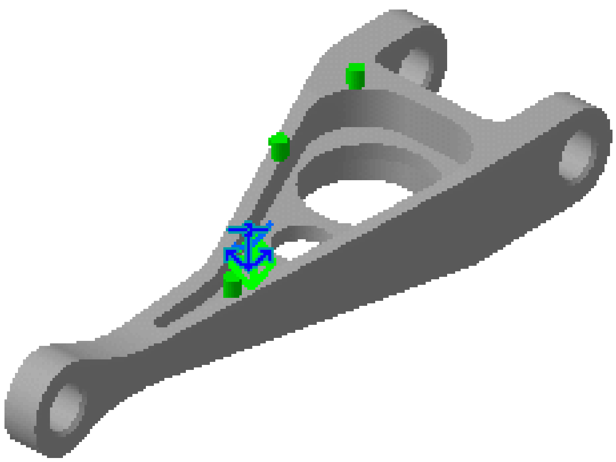
Take the default (`All`) proposed for the objects sets to update.

3. Click `OK` to perform the computation.

The Progress Bar dialog box provides a series of status messages (`Meshing`, `Factorization`, `Solution`) that inform you of the degree of advancement of the computation process.



Upon successful completion of the computation, the status of the Frequency Case Solutions objects set is changed to valid in the specification tree. In other words, the symbol appears no more.



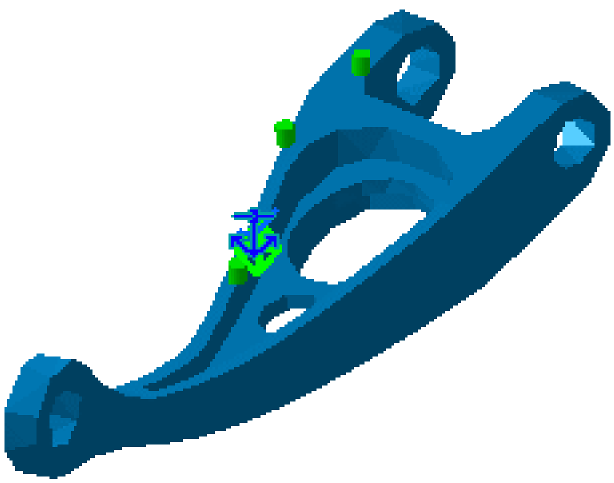
Note the green color change of the Restraints and Masses symbols, reflecting the fact that the Frequency Case Solution computation was successful.

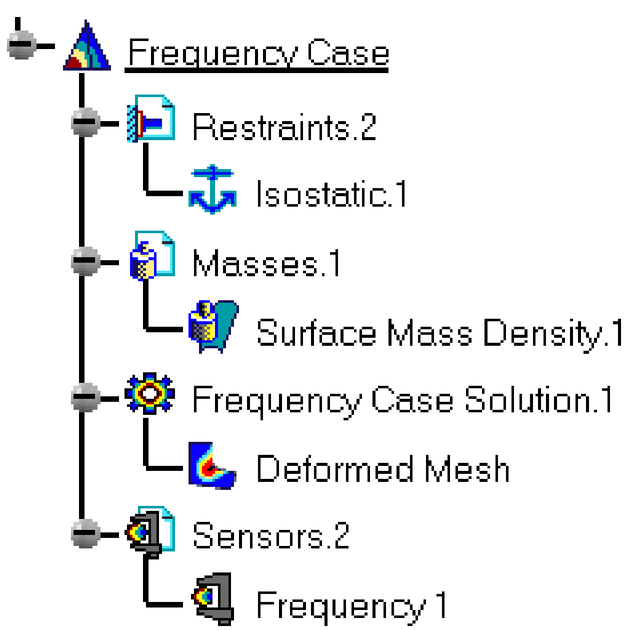
Viewing Frequency Results

This task will show you how to visualize Vibration Modes after computing the Frequency Analysis Case and how to generate a Report.

1. Click the Deformation icon .

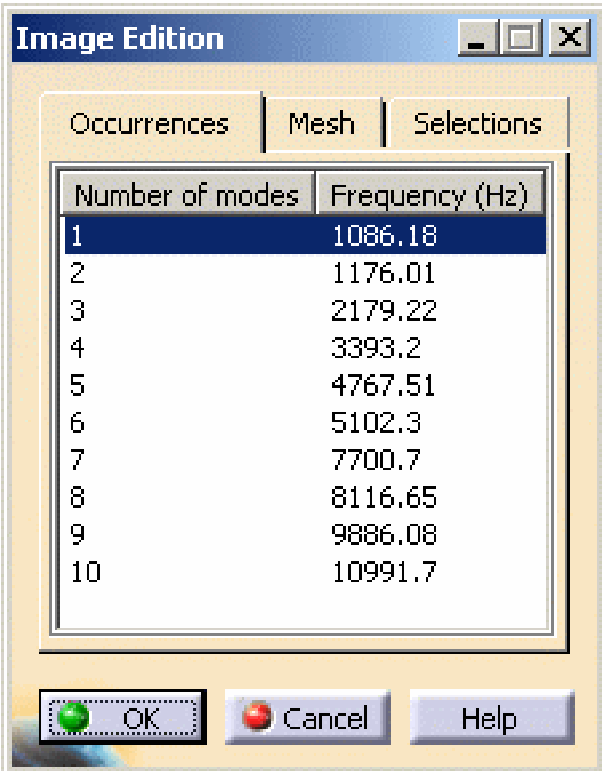
An image of the deformation corresponding to the first vibration mode is displayed, and the `Deformed Mesh` image object appears in the specification tree under the `Frequency Case Solution.1` objects set.



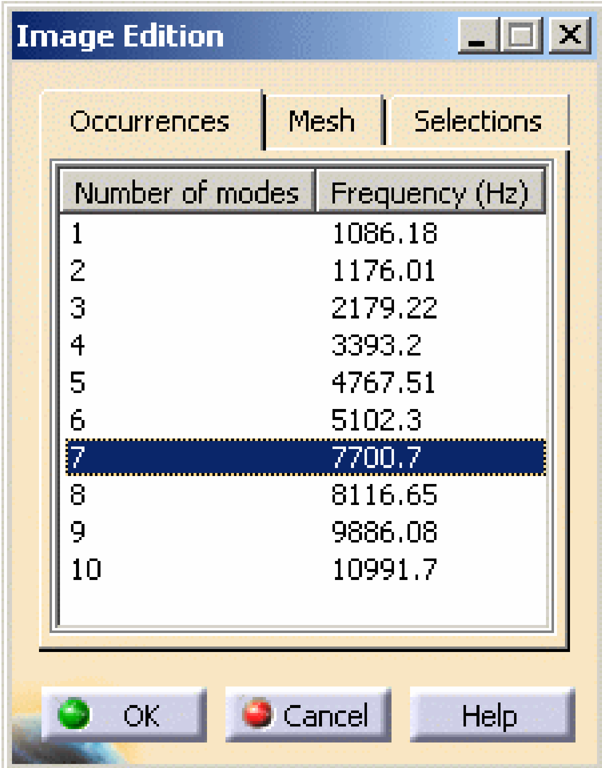


2. Double-click the `Deformed Mesh` object in the specification tree to edit the image.

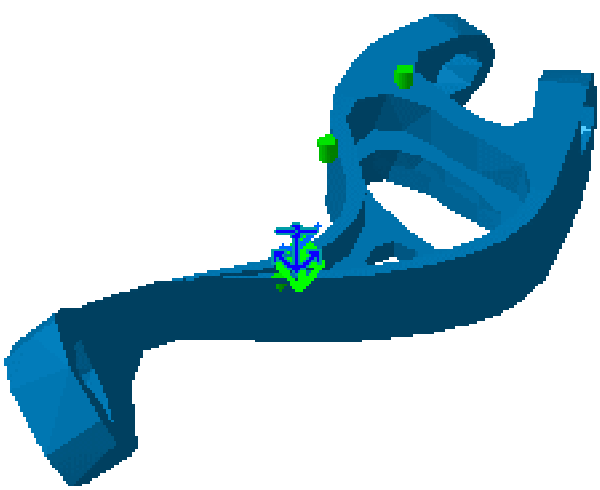
The Image Fem Editor dialog box, containing the list of vibration modes with the corresponding frequency occurrences is displayed. You can visualize any mode by clicking it in this multi-occurrence list.



3. Click the seventh mode in the multi-occurrence list, for example.



The selected mode is displayed.



4. Once you have finished editing images, click `OK` in the dialog box.

- You can further manage your results by using the Results Management action icons on the bottom of your screen.
  For more details, please refer to Results Visualization.
- In addition to standard information, the Report for a Frequency Case Solution contains modal participation factors information, which allows you to evaluate the validity of the modal truncation to the first 10 modes.
- You can modify the number of computed modes by double-clicking the Solution and editing the Solution Definition dialog box.