CATIA V5 Training

Foils

Generative Part Structural Analysis Expert

Version 5 Release 19
September 2008

EDU_CAT_EN_GPE_FI_V5R19

Copyright DASSAULT SYSTEMES

Instructor Notes:

About this course

Objectives of the course

Upon completion of this course you will be able to:

- Define and customize material properties

- Apply pressure, acceleration and force density loads; and define virtual parts

- Apply pivot, ball-joint, and user-defined restraints

- Compute a frequency analysis for a single part

- Create planar sections with which to visualize internal result values

- Compute and refine a mesh using adaptive meshing in order to achieve a pre-defined accuracy

Targeted audience

Mechanical Designers

1 Day

Prerequisites

Students attending this course should have knowledge of CATIA V5 Fundamentals, Generative Part Structural Analysis Fundamentals

Copyright DASSAULT SYSTEMES

Instructor Notes:

Table of Contents (1/2)

Copyright DASSAULT SYSTEMES

Instructor Notes:

Table of Contents (2/2)

Copyright DASSAULT SYSTEMES

Instructor Notes:

GPS Extended Pre-Processing

In this lesson you will see the pre-processing tools used for advanced analysis

- Advanced Pre-Processing Tools
- Advanced Pre-Processing Recap Exercise
- Frequency Analysis
- To Sum Up

Copyright DASSAULT SYSTEMES

Instructor Notes:

Advanced Pre-Processing Tools

You will see following Advanced Pre-Preocessing Tools

- Defining Loads
- Defining Restraints
- With Which Mesh to Work
- Defining Virtual Parts
- Defining User Material

Copyright DASSAULT SYSTEMES

Instructor Notes:

Defining Loads

You will see different types of loads

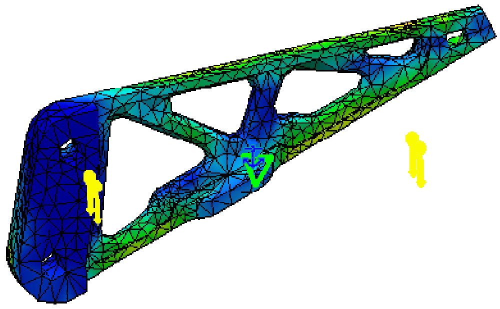
- Acceleration
- Pressure Loads
- Force Density

Copyright DASSAULT SYSTEMES

Instructor Notes:

Acceleration

You will learn how to define an acceleration.

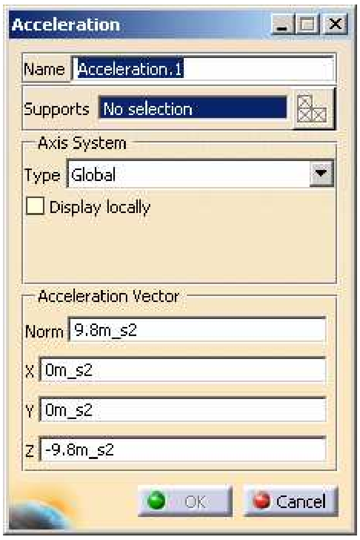


Copyright DASSAULT SYSTEMES

Instructor Notes:

About Acceleration

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



Acceleration: Units are mass body force (or acceleration) units (typically N/kg, or m/s2 in SI).

Supports: Accelerations can be applied to Volumes or Parts

Axis System:

Global: if you select the Global axis-system, the components of the sliding direction will be interpreted as relative to the fixed global rectangular coordinate system.

User-defined: if you select a User-defined axis-system, the components of the sliding direction will be interpreted as relative to the specified rectangular coordinate system.

Note:To select a User-defined axis-system, you must activate an existing axis by clicking it in the feature tree. Its name will then be automatically displayed in the Current Axis field.

Acceleration Vector:

You need to specify three components for the direction of the field, along with a magnitude information.

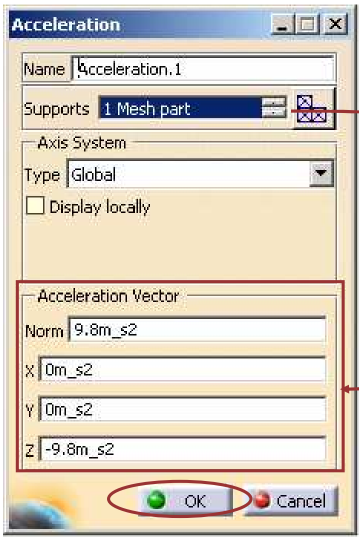
Copyright DASSAULT SYSTEMES

Instructor Notes:

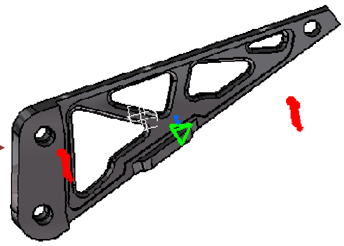
Defining an Acceleration

Before You Begin:
Go to View -> Render Style -> Customize View and make sure the Shading, Outlines and Materials options are active in the Custom View Modes dialog box

1 Switch to 'Generative Structural Analysis' workbench and click on the "Acceleration" Icon



2 Select the geometry support(s): Volumes or Parts



3 Choose the type of Axis System

4 Define the acceleration Vector

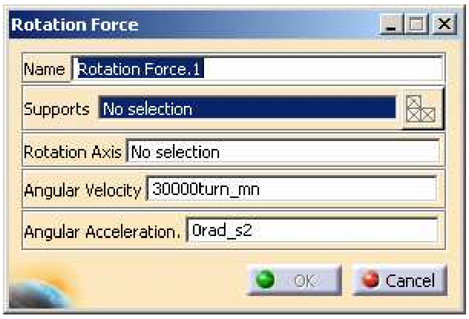
5 Click on Ok

Copyright DASSAULT SYSTEMES

Instructor Notes:

About Rotation force

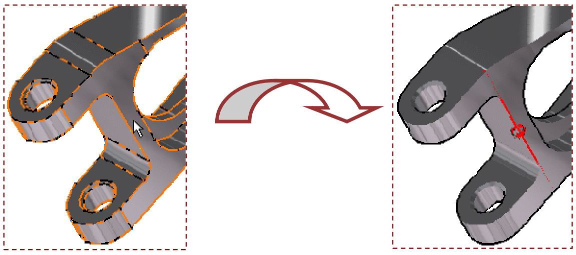
Rotation Forces are intensive loads representing mass body force (acceleration) fields induced by rotational motion applied to parts.

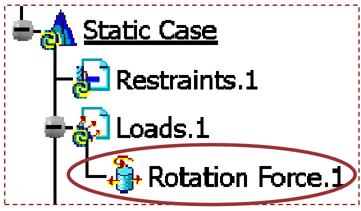


Rotation Force: Units are angular velocity and angular acceleration units (typically rad/sec and rad/sec2 in SI).

Supports: Accelerations can be applied on Volumes or Parts

Rotation Axis: The user specifies a rotation axis and values for the angular velocity and angular acceleration magnitudes, and the program automatically evaluates the linearly varying acceleration field distribution.





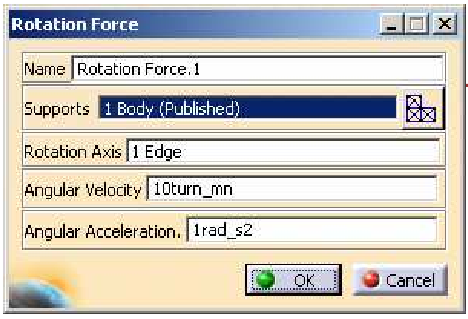
Copyright DASSAULT SYSTEMES

Instructor Notes:

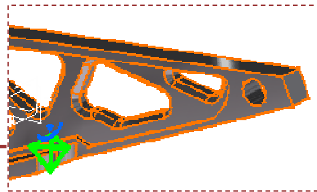
Defining a Rotation Force 

Before You Begin:
Go to View -> Render Style -> Customize View and make sure the Shading, Outlines and Materials options are active in the Custom View Modes dialog box.

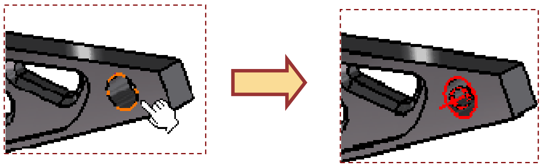
1 Switch to ‘Generative Structural Analysis’ workbench and click on the Rotation Force Icon 



2 Select the geometry support(s): Volumes or Parts



3 Select a hole edge for Rotation Axis



4 Enter Angular Velocity and Angular Acceleration values

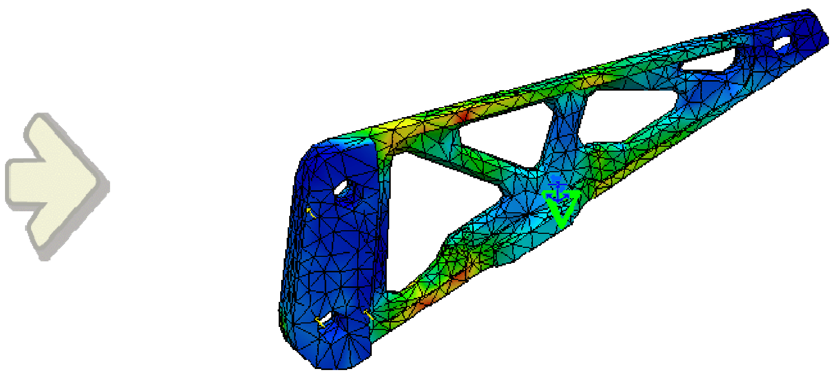
5 Click on OK

Copyright DASSAULT SYSTEMES

Instructor Notes:

Pressure Loads

You will learn how to apply a pressure.



Copyright DASSAULT SYSTEMES

Instructor Notes:

Copyright DASSAULT SYSTEMES

About Pressure

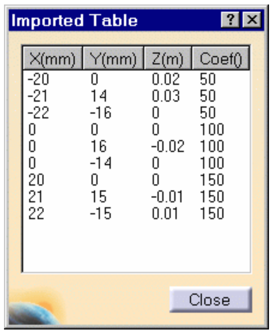
Pressures are intensive loads representing uniform scalar pressure fields applied to surface geometries; consequently the force direction is everywhere normal to the surface.

You can define as many Pressure Loads as desired with the same dialog box.

Supports: Pressure can be applied on Surfaces or Faces

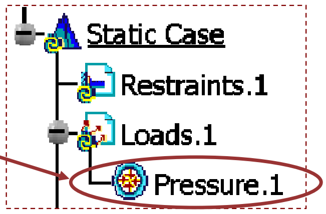
Pressure: Units are pressure units : N/m2 (in SI) but can be MPa (1MPa=1 N/mm² or 1Pa=1N/m²)

You can import external data files. They can be either a .txt file (columns separated using the Tab key) or an .xls file with a pre-defined format (four columns, the first three columns specifying the X, Y and Z points coordinates in the global axis and the last one containing the coefficient).



X(mm)	Y(mm)	Z(m)	Coef()
-20	0	0.02	50
-21	14	0.03	50
-22	-16	0	50
0	0	0	100
0	16	-0.02	100
0	-14	0	100
20	0	0	150
21	15	-0.01	150
22	-15	0.01	150

Pressure objects can be edited by a double-click on the corresponding object or icon in the specification tree



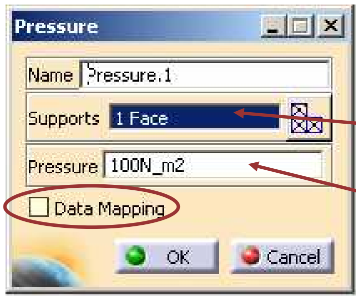
Copyright DASSAULT SYSTEMES

Instructor Notes:

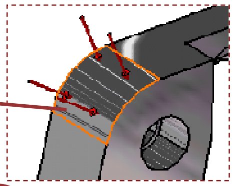
Applying Pressure 

Before You Begin:
Go to View -> Render Style -> Customize View and make sure the Shading, Outlines and Materials options are active in the Custom View Modes dialog box

1. Switch to "Generative Structural Analysis" workbench and click on the "Pressure" Icon 



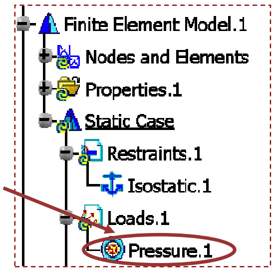
2. Select the geometry support(s) (Surfaces). Any selectable geometry is highlighted when you drag the cursor over it



3. Specify a pressure value or open a Data File for mapping, by selecting 'Data Mapping' option.

4. Click on OK

A Loads object appears in the feature tree under the active Loads objects set.

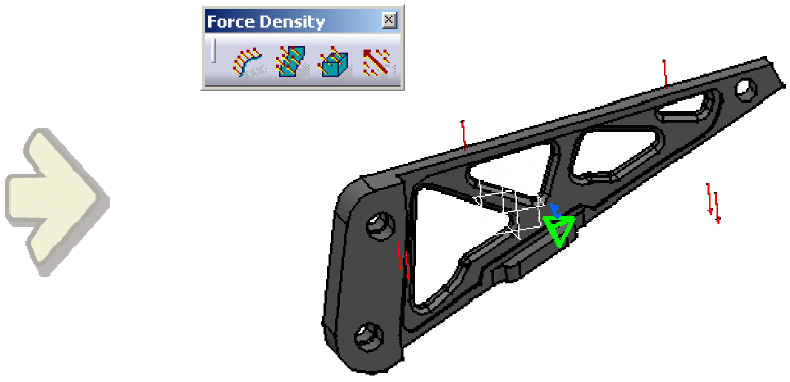


Copyright DASSAULT SYSTEMES

Instructor Notes:

Force Density

You will learn how to apply a Force Vector



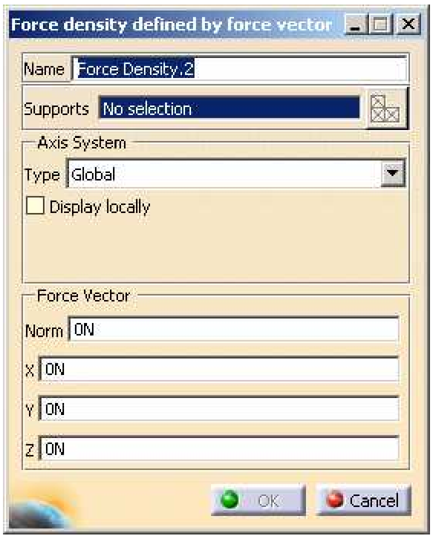
Copyright DASSAULT SYSTEMES

Instructor Notes:

What is a Force Vector

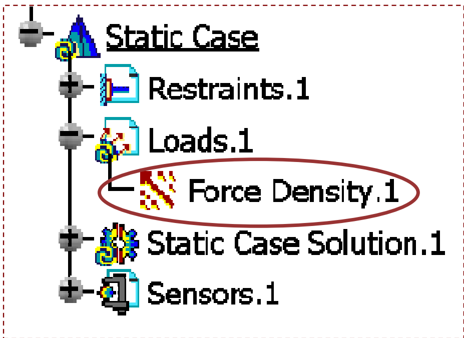
Force Vector allows you to define the equivalent of the existing line/Surface/body force density by giving as input only a force in Newton and not a force density

You can select several geometries of the same type and apply a vector force on them



The supports can be: Edges, Surfaces, 3D bodies

Catia computes automatically the volume/surface/length on which the loads are applied and compute the equivalent force density.

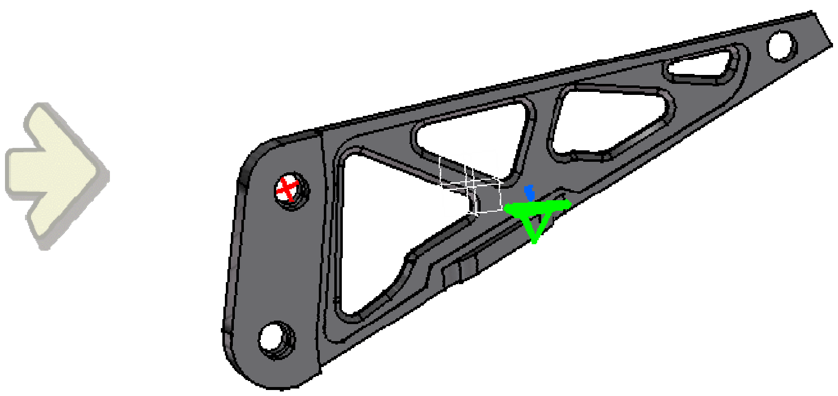


Copyright DASSAULT SYSTEMES

Instructor Notes:

Defining Restraints

You will learn how to apply restraints to a part.



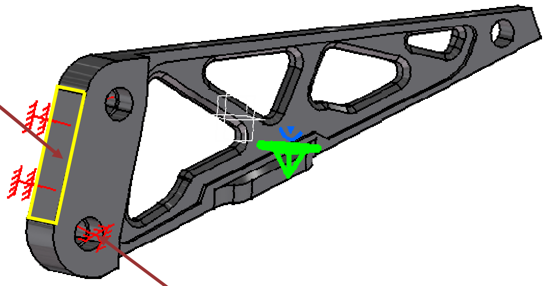
Copyright DASSAULT SYSTEMES

Instructor Notes:

Introduction

Work with the Geometry: A faster way to apply restraints/loads

The restraints and the loads will be applied directly onto the geometry (surfaces, lines, points, groups) as shown on the example below:



This restraint is applied onto the yellow surface

This restraint is applied onto the circle

Then the computation will automatically apply the restraints/loads to the mesh.

Even if you work with the geometry, the part must be meshed to take into account the restraints/Loads.

Copyright DASSAULT SYSTEMES

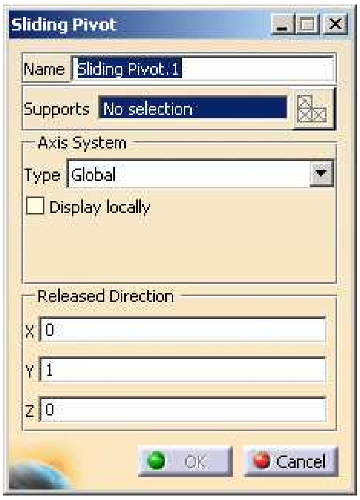
Instructor Notes:

Sliding Pivot  

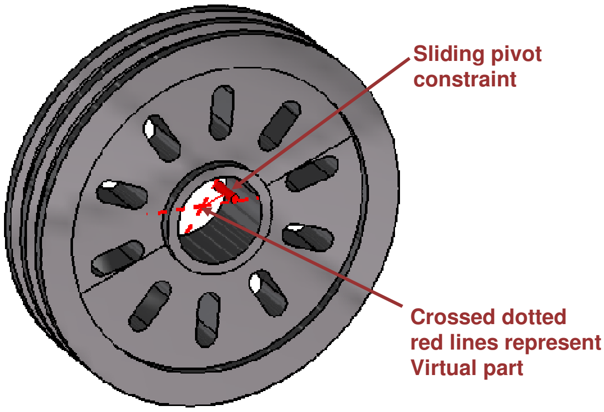
Sliding Pivots are cylindrical join restraints applied to handle points of virtual parts, which results in constraining the point to simultaneously translate along and rotate around a given axis.

They can be viewed as particular cases of general cylindrical joins, which allow a relative combined translation and rotation between two points (in the Sliding Pivot case, one of the two points is fixed, along with the sliding pivot axis)

A "Slider Pivot" restraint has 2 DOF: 1Tr. & 1Rot. **Supports: It needs a "Virtual Part" to be applied.**



For the fixed point, the program automatically picks the handle of the virtual part. The user defines the sliding pivot direction, and as a result the virtual part as a whole is allowed to translate along and to rotate around an axis parallel to the sliding pivot direction and passing through the fixed point.



Copyright DASSAULT SYSTEMES

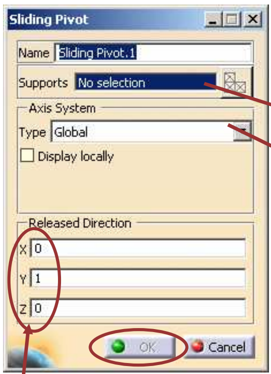
Instructor Notes:

Defining a Sliding Pivot

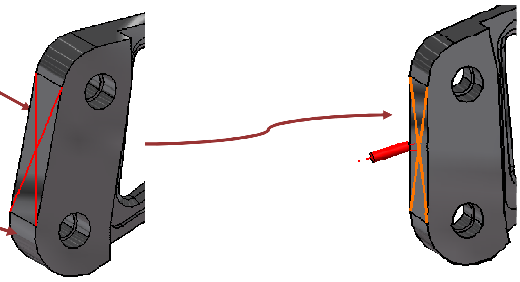
Before You Begin:
Go to View -> Render Style -> Customize View and make sure the Shading, Outlines and Materials options are active in the Custom View Modes dialog box

1 Click on the "Sliding Pivot" Icon in the "Restrain" Toolbar

2 Click on the support: A pre-defined "virtual part"



Virtual part



3 Define the axis-system:

4 Enter the released direction (sliding direction) and click on "OK"

Global: if you select the Global axis-system, the components of the sliding direction will be interpreted as relative to the fixed global rectangular coordinate system.

User-defined: if you select a User-defined axis-system, the components of the sliding direction will be interpreted as relative to the specified rectangular coordinate system.

Note: To select a User-defined axis-system, you must activate an existing axis by clicking it in the feature tree. Its name will then be automatically displayed in the Current Axis field

Copyright DASSAULT SYSTEMES

Instructor Notes:

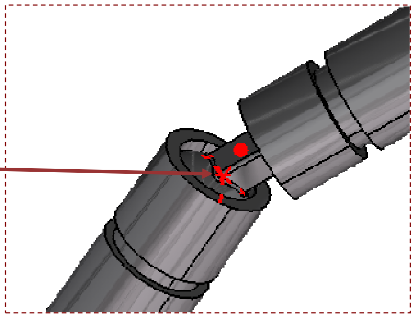
Ball Joint

Ball Joints are spherical join restraints applied to handle points of virtual parts, which result in constraining the point to rotate around a coinciding fixed point.

They can be viewed as particular cases of general spherical joins, which allow a relative rotation between two points (in the Ball Joint case, one of the two points is fixed)

A "Ball Joint" restraints has 3 DOF: 3Rot. Supports: It needs a "Virtual Part" to be applied

For the fixed point, the program automatically picks the handle of the virtual part. The virtual part as a whole is then allowed to rotate around this point



Make sure you fixed all the global degrees of freedom of your assembly, otherwise a global singularity will be detected at the time of the Static Computation (such a model is unsolvable). To allow you to easily correct the model (Static Analysis Cases only), the singular displacement of the assembly will be simulated and visualized after computation

Copyright DASSAULT SYSTEMES

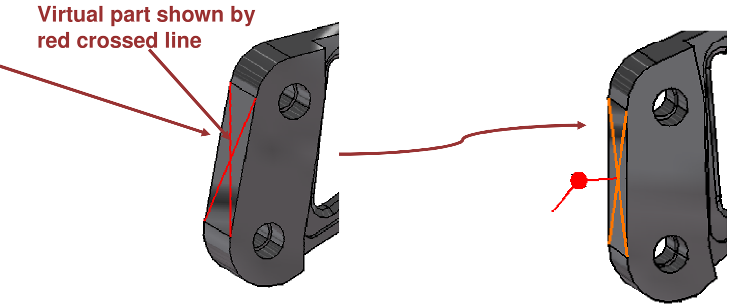
Instructor Notes:

Defining a Ball Joint  

Ball Joint is spherical type restraint applied on virtual parts.

1 Click on the “Ball Joint” Icon in the “Restrain” Toolbar 

2 Click on the support: A pre-defined “virtual part”



3 Click on OK

Copyright DASSAULT SYSTEMES

Before You Begin:
Go to View -> Render Style -> Customize View and make sure the Shading, Outlines and Materials options are active in the Custom View Modes dialog box.

Instructor Notes:

Copyright DASSAULT SYSTEMES

Pivot

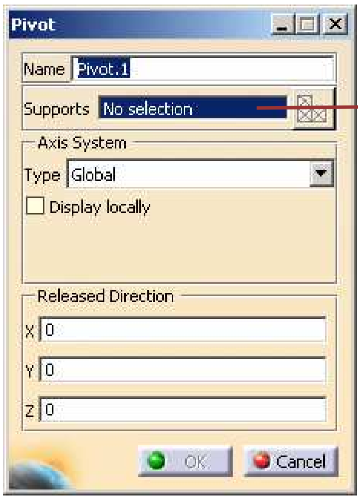
Pivots are hinge (conical join) restraints applied to handle points of virtual parts, which result in constraining the point to rotate around a given axis

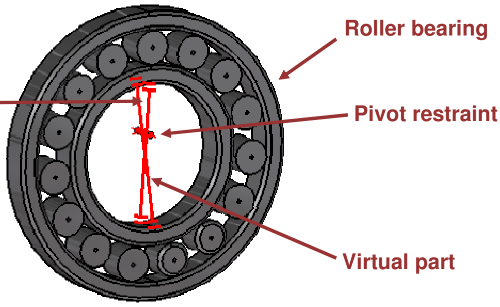
They can be viewed as particular cases of general hinge joins, which allow a relative rotation between two points (in the Pivot case, one of the two points is fixed, along with the pivot axis).

A "Pivot" restraints has 1 DOF: 1 Rot.

Supports: It needs a "Virtual Part" to be applied

For the fixed point, the program automatically picks the handle of the virtual part. The user defines the pivot direction, and as a result the virtual part as a whole is allowed to rotate around an axis parallel to the pivot direction and passing through the fixed point





Make sure you fixed all the global degrees of freedom of your assembly, otherwise a global singularity will be detected at the time of the Static Computation (such a model is unsolvable). To allow you to easily correct the model (Static Analysis Cases only), the singular displacement of the assembly will be simulated and visualized after computation

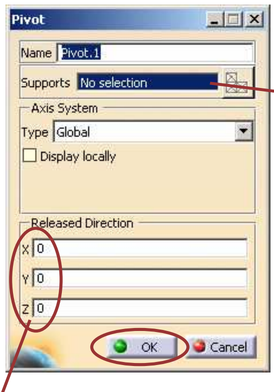
Copyright DASSAULT SYSTEMES

Instructor Notes:

Defining a Pivot

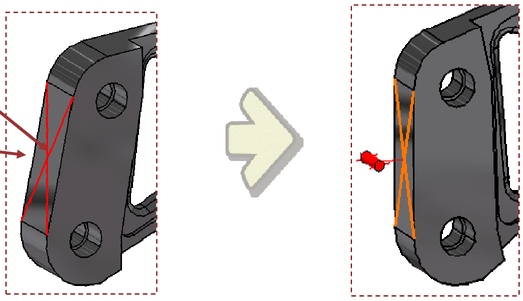
Before You Begin:
Go to View -> Render Style -> Customize View and make sure the Shading, Outlines and Materials options are active in the Custom View Modes dialog box

1 Click on the "Pivot" Icon in the "Restrain" Toolbar



4 Enter the released direction (sliding direction) and click on "OK"

2 Click on the support: A pre-defined "virtual part"



3 Define the axis-system:

Global: if you select the Global axis-system, the components of the sliding direction will be interpreted as relative to the fixed global rectangular coordinate system.

User: if you select a User axis-system, the components of the sliding direction will be interpreted as relative to the specified rectangular coordinate system.

To select a User axis-system, you must activate an existing axis by clicking it in the feature tree. Its name will then be automatically displayed in the Current Axis field.

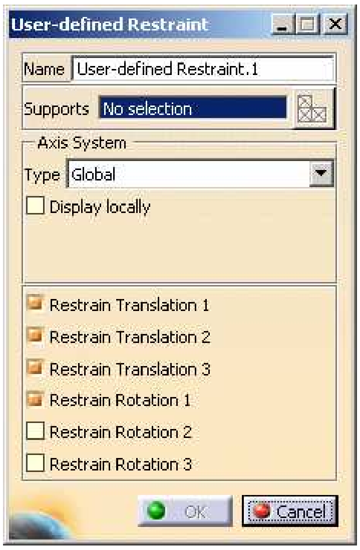
Copyright DASSAULT SYSTEMES

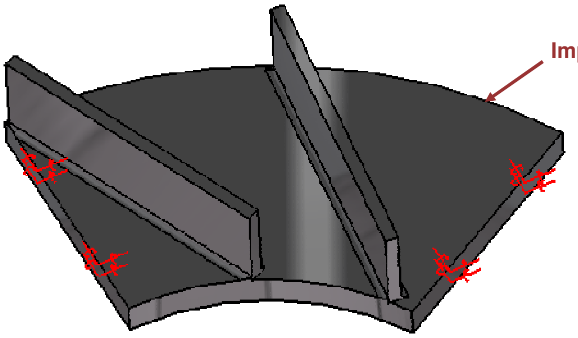
Instructor Notes:

User-defined Restraints

User-defined Restraints are generic restraints allowing you to fix any combination of available nodal DOF on arbitrary geometries. 3Tr. freedom per node for continuum element meshes, and 3Tr. and 3Rot. of freedom per node for structural element meshes

Supports: Points or Vertex, Curves or Edges, Faces or Surfaces,Virtual Parts, groups





means no rotation degree of freedom in the direction

means no translation degree of freedom in that direction

Make sure you fixed all the global degrees of freedom of your assembly, otherwise a global singularity will be detected at the time of the Static Computation (such a model is unsolvable). To allow you to easily correct the model (Static Analysis Cases only), the singular displacement of the assembly will be simulated and visualized after computation

Copyright DASSAULT SYSTEMES

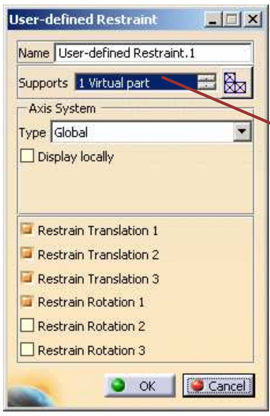
Instructor Notes:

Defining User-defined Restraints 

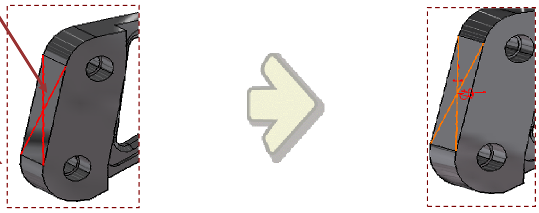
Before You Begin:
Go to View -> Render Style -> Customize View and make sure the Shading, Outlines and Materials options are active in the Custom View Modes dialog box

1. Click on the “User-Defined Restraints” Icon in the “Restrain” Toolbar

2. Click on the support: i.e a “virtual part”. (Any selectable geometry is highlighted when you pass the cursor over it)



Virtual part shown by red crossed lines



3. Define the axis-system:

Global: The DOF components of the sliding direction will be interpreted as relative to the fixed global rectangular coordinate system.

Implicit: The DOF directions will be interpreted as relative to a local variable coordinate system whose type depends on the support geometry.

User: The components of the sliding direction will be interpreted as relative to the specified rectangular coordinate system.

4. Check the DOF you want to restrain

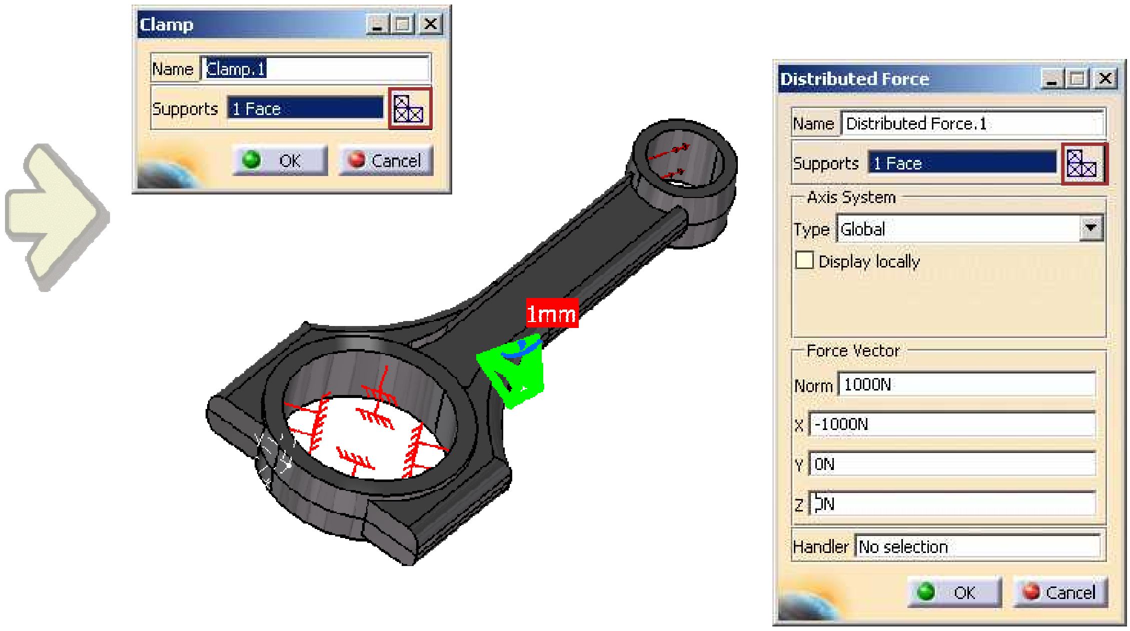
To select a User axis-system, you must activate an existing axis by clicking it in the feature tree. Its name will then be automatically displayed in the Current Axis field.

Copyright DASSAULT SYSTEMES

Instructor Notes:

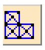
With Which Mesh to Work

You will learn how to define mesh part filter trough preprocessing tools



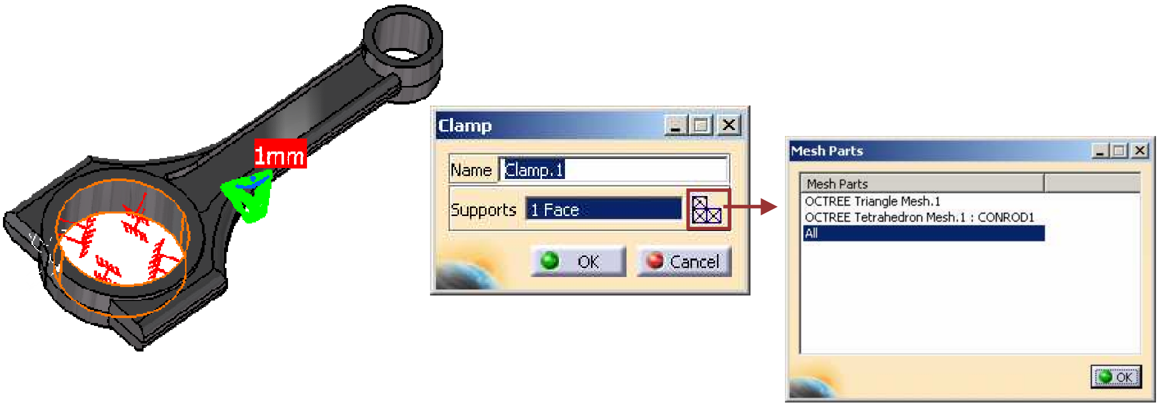
Copyright DASSAULT SYSTEMES

Instructor Notes:

What is a 'Mesh Part' Filter 

This tool is available on every pre-processing tool.

This "Mesh Part" filter allows you to select the mesh parts on which you want to apply the preprocessing feature.



The default is « All », that means all the mesh parts will be taken into account. It means, If you add a new mesh part on the support, the preprocessing feature will be automatically applied on the new mesh part.

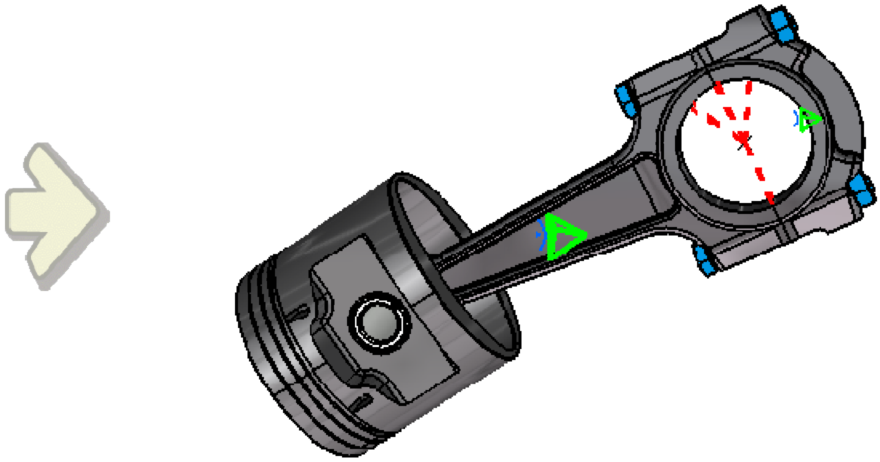
On the other hand, If you select one or many mesh parts, this will not change if you define a new mesh part on the support.

Copyright DASSAULT SYSTEMES

Instructor Notes:

Defining Virtual Parts

You will learn how to define virtual parts.



Copyright DASSAULT SYSTEMES

Instructor Notes:

Introduction to Virtual Parts

Virtual Parts are structures created without a geometric support. They represent bodies for which no geometric model is available, but which play a role in the structural analysis of single parts or assembly systems.

Virtual Parts are used to transmit actions at a distance. Therefore they can be thought of as rigid bodies, except for the case where a lumped flexibility is explicitly introduced by the means of a spring element.

There are 6 kinds of Virtual Parts:

- Rigid Virtual parts
  They stiffly transmit their actions : they locally stiffen the deformable body
- Smooth Virtual parts
  They softly transmit their actions : they don’t stiffen the deformable body
- Contact Virtual parts
  They softly transmit their actions while preventing from body inter-penetration
- Rigid Spring Virtual parts
  They stiffly transmit their actions and behave like a 6 DOF spring
- Smooth Spring Virtual parts
  They softly transmit their actions and behave like a 6 DOF spring
- Periodicity Conditions

Copyright DASSAULT SYSTEMES

Instructor Notes:

Rigid Virtual Parts

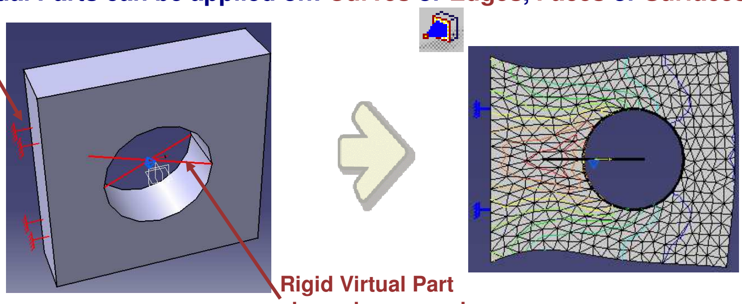
A Rigid Virtual Part is a rigid body connecting a specified point to specified part geometries.

It behaves as a mass-less rigid object which will stiffly transmit actions (masses, restraints and loads) applied at the handle point, while locally stiffening the deformable body or bodies to which it is attached.

The Rigid Virtual Part does not take into account the elastic deformability of the parts to which it is attached. The program proceeds as follows:

- **A node is created in coincidence with the specified handle point**
- **Each node of the specified geometry supports meshes is connected by a kinematical rig-beam element to the handle node**
- **A set of rig-beam relations is generated between the handle node degree of freedom and the connected nodes degree of freedom**

Rigid Virtual Parts can be applied on: Curves or Edges, Faces or Surfaces



Copyright DASSAULT SYSTEMES

Instructor Notes:

Smooth Virtual Parts

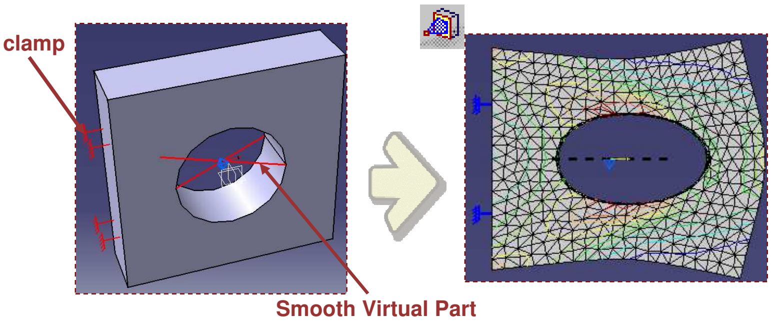
A Smooth Virtual Part is a rigid body connecting a specified point to specified part geometries.

It behaves as a mass-less rigid object which will softly transmit actions (masses, restraints and loads) applied at the handle point, without stiffening the deformable body or bodies to which it is attached

The Smooth Virtual Part does approximately take into account the elastic deformability of the parts to which it is attached. The program proceeds as follows:

- **A node is created in coincidence with the specified handle point**
- **All nodes of the specified geometry supports meshes are connected by a kinematical spider element to the handle node**
- **A set of mean (constr-n) relations is generated between the handle node degree of freedom and the connected nodes degree of freedom**

Smooth Virtual Parts can be applied on: Curves or Edges, Faces or Surfaces



The hole is free to deform.
Forces applied on nodes around the hole have the same intensity (equal to the force value applied on the virtual part divided by the number of nodes on the hole contour).

Copyright DASSAULT SYSTEMES

Instructor Notes:

Contact Virtual Parts

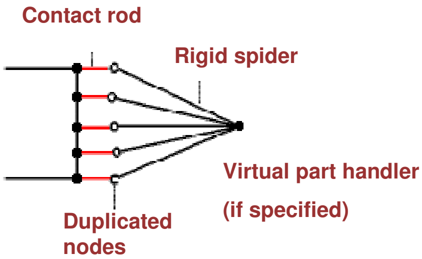
A Contact Virtual Part is a rigid body connecting a specified point to specified part geometries.

It behaves as a mass-less rigid object which will transmit actions (masses, restraints and loads) applied at the handle point, while preventing from body inter-penetration, without stiffening the deformable body or bodies to which it is attached.

The Contact Virtual Part does take into account the elastic deformability of the parts to which it is attached. The program proceeds as follows:

- **A node is created in coincidence with the specified handle point.**
- **Each node of the specified geometry support meshes is offset in the local normal direction by a small amount and a contact element is generated between each pair of offset nodes, generating a set of contact relations with a right-hand side equal to the user-defined clearance.**
- **Each offset node is connected by a kinematical rig-beam element to the handle node.**
- **A set of rig-beam relations is generated between the handle node degree of freedom and the connected offset nodes degree of freedom**

Contact Virtual Parts can be applied on:
Curves or Edges, Faces or Surfaces



Copyright DASSAULT SYSTEMES

Instructor Notes:

Rigid Spring Virtual Parts

A Rigid Spring Virtual Part is an elastic body connecting a specified point to a specified geometry

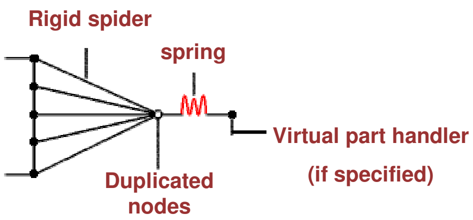
It behaves as a six degree of freedom spring in series with a mass-less rigid body which will stiffly transmit actions (masses, restraints and loads) applied at the handle point, while stiffening the deformable body or bodies to which it is attached.

The Rigid Spring Virtual Part does not take into account the elastic deformability of the parts to which it is attached. The program proceeds as follows:

- A node is created in coincidence with the specified handle point.
- A second node, offset from the first node, is created in a user-specified direction.
- The offset node is connected by a user-specified spring element to the handle node.
- All nodes of the specified geometry supports meshes are connected by rig-beam kinematical elements to the offset node.
- A set of rig-beam relations is generated between the offset node degree of freedom and the connected nodes degree of freedom.

The Rigid Virtual Part is built with Rigid Spider and Spring elements

Rigid Spring Virtual Parts can be applied on:
Curves or Edges, Faces or Surfaces



Copyright DASSAULT SYSTEMES

Instructor Notes:

Smooth Spring Virtual Parts

A Smooth Spring Virtual Part is an elastic body connecting a specified point to a specified geometry.

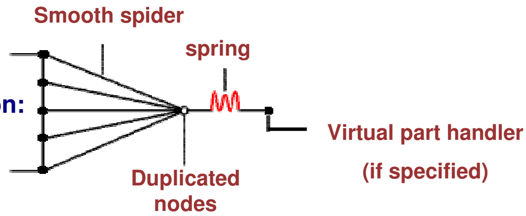
It behaves as a 6-degree of freedom spring in series with a mass-less rigid body which will softly transmit actions (masses, restraints and loads) applied at the handle point, without stiffening the deformable body or bodies to which it is attached.

The Smooth Spring Virtual Part does approximately take into account the elastic deformability of the parts to which it is attached. The program proceeds as follows:

- A node is created in coincidence with the specified handle point.
- A second node, offset from the first node, is created in a user-specified direction.
- The offset node is connected by a user-specified spring element to the handle node.
- All nodes of the specified geometry supports meshes are connected by a kinematical spider element to the offset node.
- A set of mean (constr-n) relations is generated between the offset node degree of freedom and the connected nodes degree of freedom.

The Smooth spring Virtual Part is built with Smooth Spider and Spring elements

Rigid Spring Virtual Parts can be applied on:
Curves or Edges, Faces or Surfaces



Copyright DASSAULT SYSTEMES

Instructor Notes:

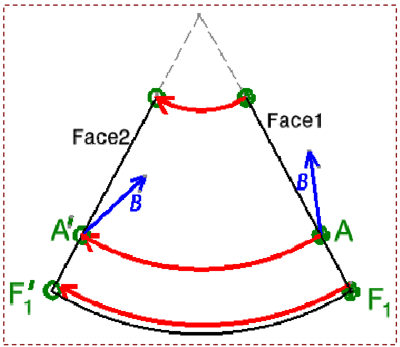
Periodicity Conditions

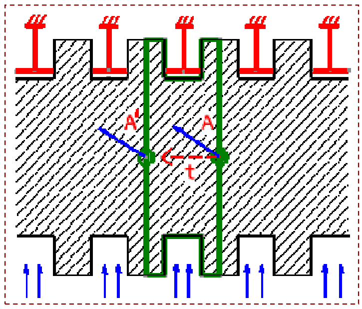
Periodicity conditions enable you to perform an analysis on the solid section of a periodic part.

This solid section should represent a cyclic period of the entire part. Applying periodicity conditions is cost saving: you compute only a section of the part and get a result that is representative of the whole part.

There are 2 kinds of periodicity conditions:

- **Cyclic symmetry: of the geometry as well as both restraints and loads**
- **A Regular symmetry:of the sectioned geometry as well as both restraints and loads**





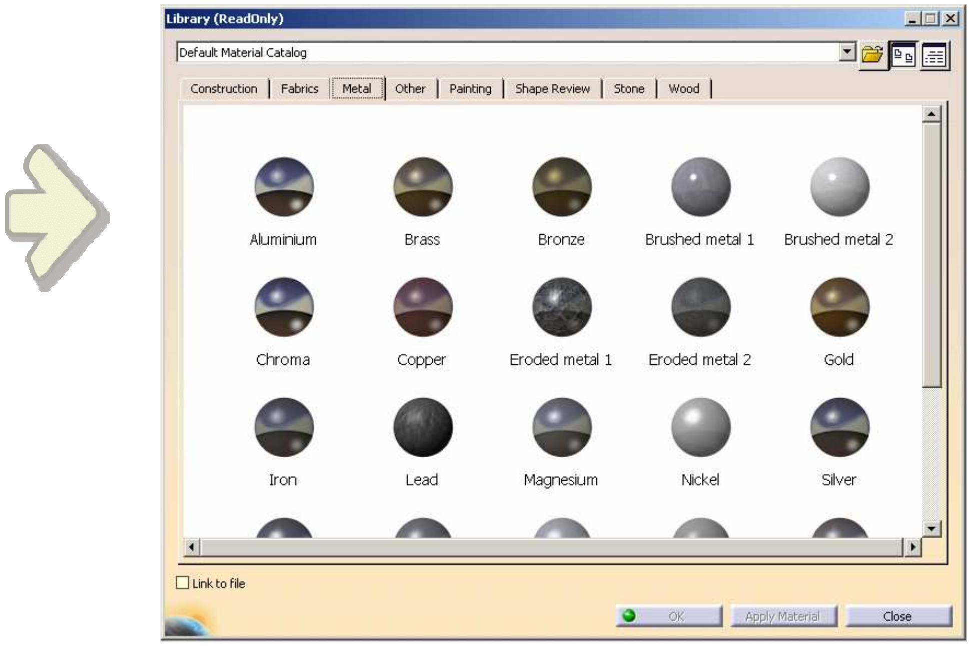
To use periodicity conditions, you need to make sure the geometry as well as the created restraints and loads are periodic. The geometry also needs to be regular at the place the section is cut, discontinuity is not allowed.

Copyright DASSAULT SYSTEMES

Instructor Notes:

Defining User Material

You will see how to define new material using pre-define material properties

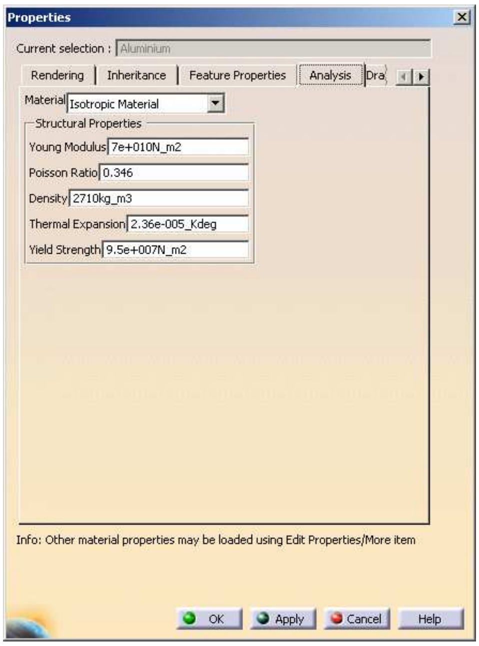


Copyright DASSAULT SYSTEMES

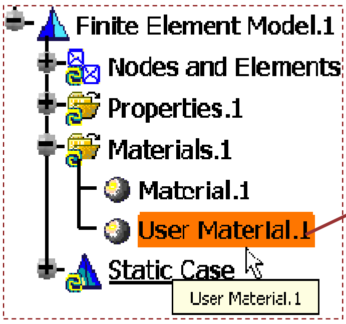
Instructor Notes:

About User Material Tool

The User Material tool allows you to define a new material inside the material set in Generative Structural Analysis. You can apply material properties on your parts/products pre-defined materials or straight on a mesh (coming from I.e “advanced Meshing Tools” workbench)



A “User Material” object appears in the tree. You can edit this object and customize its material analysis properties according to your needs



Copyright DASSAULT SYSTEMES

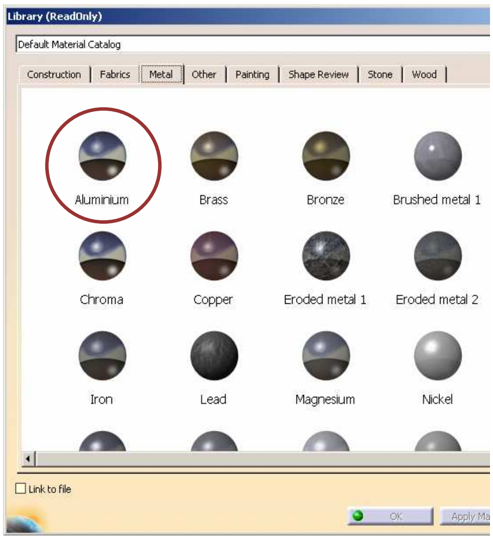
Instructor Notes:

How to Customize Pre-Define Material Properties (1/2)

You can define new material properties by modifying pre-define ones

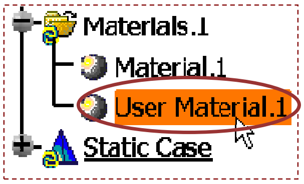
1. Click on the user material Icon

2. Select a material in the Material Library



3. Click on OK to validate your choice

4. A User Material object appears in the tree. Double-click on it to edit it



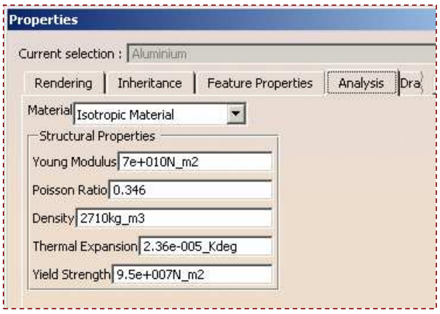
Copyright DASSAULT SYSTEMES

Instructor Notes:

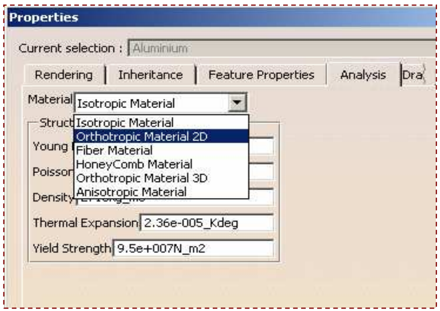
How to Customize Pre-Define Material Properties (2/2)

You can define new material properties by modifying pre-define ones

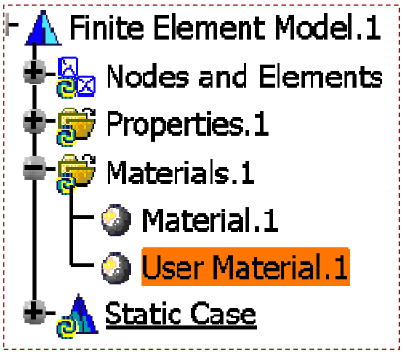
5 Click on the Analysis tab



6 Modify the properties you want



7 Click on OK to validate



Copyright DASSAULT SYSTEMES

Instructor Notes:

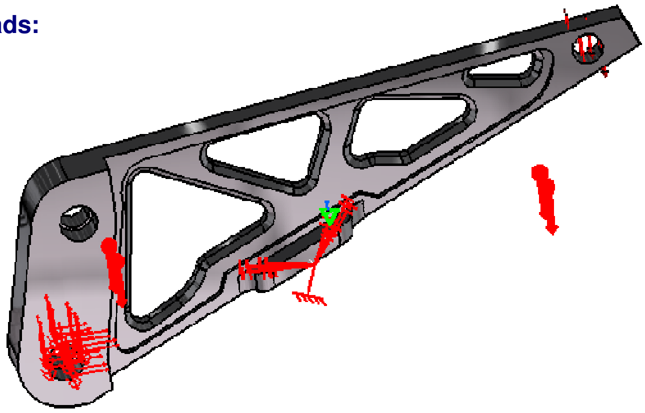
Exercise

'Pre-Processing' Recap Exercise

20 min.

In this exercise you will practice different tools that you have seen in the pre-Processing lesson.
You will :

- Define a Virtual Part
- Define the Restraints
- Apply two different types of Loads:
    - An Acceleration
    - A Force Density

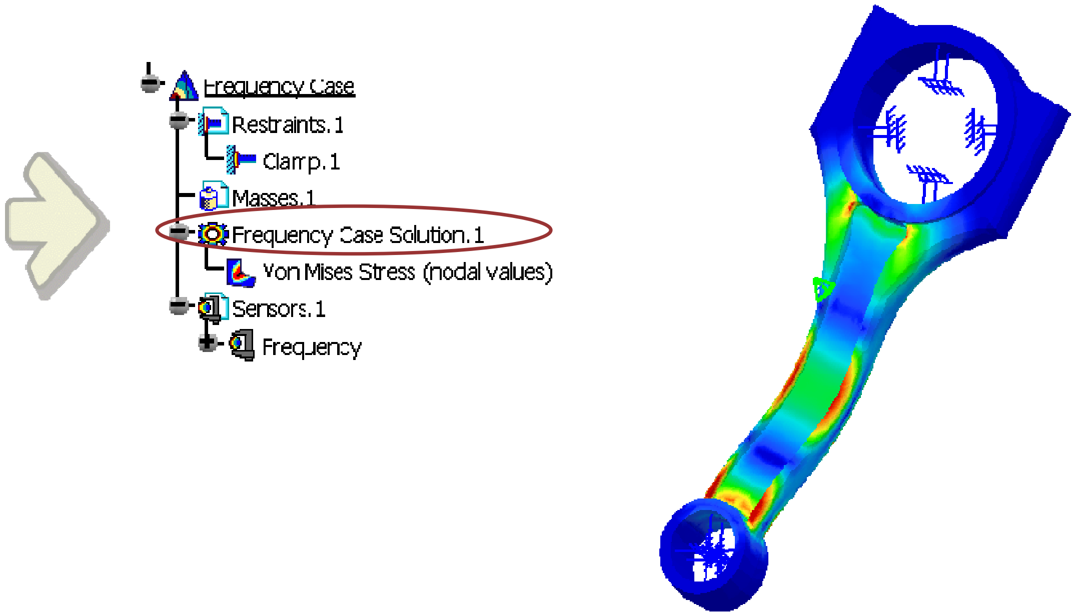


Copyright DASSAULT SYSTEMES

Instructor Notes:

Frequency Analysis

You will see how to define a Frequency analysis



Copyright DASSAULT SYSTEMES

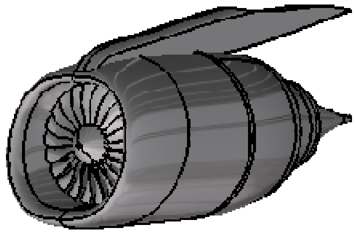
Instructor Notes:

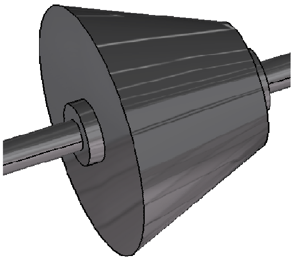
Why Frequency Analysis (1/2)

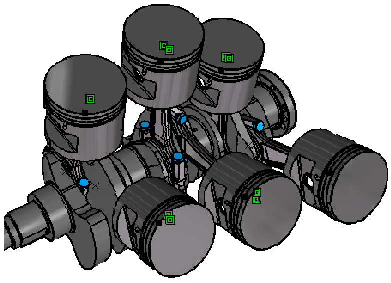
Mechanical structures are also subjected to vibration and time varying loads in addition to static loads.

A structure is subjected to vibrations depending on the source of vibration. There are two ways to excite the structure.

- **Vibrations may be generated within structure itself as in case of rotating turbines, propellers, reciprocating engines.**
- **Structures get excited from other vibration source as in case engine supporting structure vibrates because of combustion engine vibration, airplane wings vibrate due to rotor, turbine casing vibrations**







Copyright DASSAULT SYSTEMES

In actual practice structures are subjected to both types of vibration simultaneously.

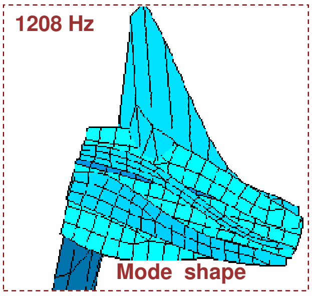
Instructor Notes:

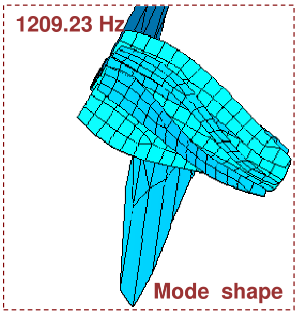
Why Frequency Analysis (2/2)

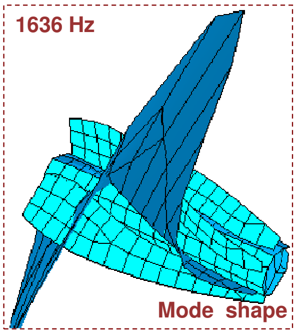
Frequency analysis helps to find out natural frequencies and mode shapes of a structure which are unique characteristics of that structure.

Number of modes	Frequency (Hz)
1	1208
2	1208.02
3	1208.02
4	1208.07
5	1209.23
6	1209.23
7	1636.54
8	1636.54
9	1637.14
10	1637.16
11	1637.6

- **A natural frequency is frequency at which the structure vibrates with very large displacements when excited. Structures have more than one natural frequency. In case, the excitation frequency matches a natural frequency, it leads to structure failure due to resonance. Therefore, it is important to know the natural frequencies of a structure.**
- **A mode shape is specific displacement pattern the structure exhibits at a specific natural frequency. A mode shape helps to visualize how the structure will fail when excited at a given natural frequency.**







These characteristics are important while designing structure subjected to vibrations and time varying loads.

Copyright DASSAULT SYSTEMES

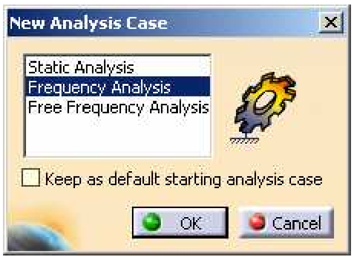
Instructor Notes:

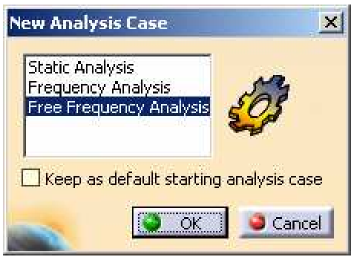
What is a Frequency Analysis

A “Frequency analysis” allows you to create object sets for frequency environmental Specifications. It will implicitly require a normal modes solution procedure for the computation of a vibration case (free frequency) or for a given non-structural mass distribution under given restraints.

When you enter the GPS workbench, you have choice between 3 types of analysis:

- Static Analysis
- Frequency Analysis
- Free Frequency Analysis





There are 2 kinds of Frequency analysis:

- Frequency Analysis: Allows you to compute a modal analysis of a restrained part. Additional mass can be applied
- Free Frequency Analysis: Allows you to compute a vibration case. You can not apply restraints.

Copyright DASSAULT SYSTEMES

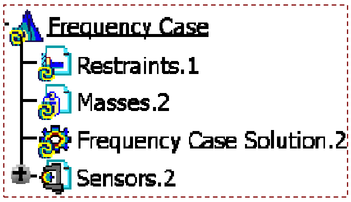
Instructor Notes:

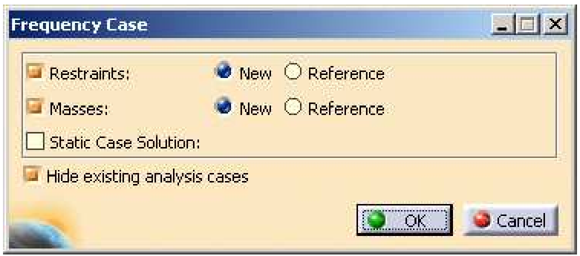
Frequency / Free Frequency Analysis

As you have seen in introduction there are 2 kinds of frequency analysis. You will see their specificities. Whatever the type of frequency analysis, you can not apply loads.

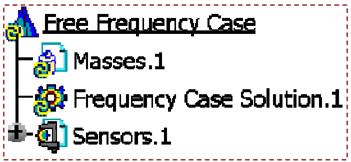
You have the possibility to start from scratch (New) or to use references for defining the restraints and masses. If you choose “Reference” it means you have previously defined in other cases and you just need to select them in the specification tree. You will see how to apply additional Mass in the following slides.

Frequency analysis





Free Frequency analysis



As you can see in the tree, “Free Frequency Analysis” does not allow the creation of restraint. You can only add some masses. Free frequency analysis are used to compute vibration cases.

If the “Restraints” option unchecked in the “Frequency Case”, it becomes a “Free Frequency” Analysis which is equivalent to vibration modes.

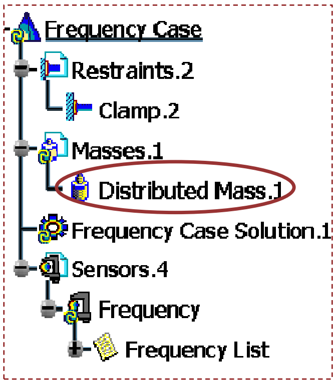
Copyright DASSAULT SYSTEMES

Instructor Notes:

What is Distributed Mass

Distributed Masses are used to model purely inertial (non-structural) system characteristics, such as additional equipment.

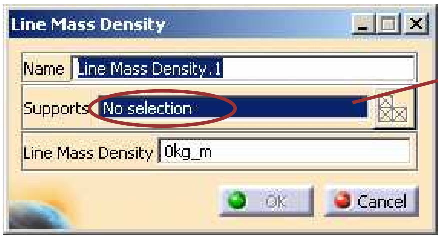
They represent scalar point mass fields and are equivalent to a total mass concentrated at a given point (geometry centroid). They can be distributed on a virtual part or on a geometric selection. Distributed mass is used only in case of Frequency analysis. They are not treated as structural loads.

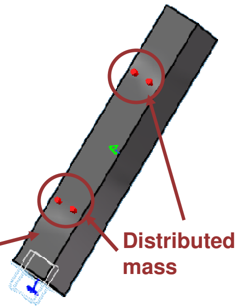


You can use them in frequency case to model additional part in your analysis.

They can be

- Line Mass Density
- Surface Mass Density
- Distributed Mass and Inertia





Structural mass is always taken into account while computing Frequency case.

Copyright DASSAULT SYSTEMES

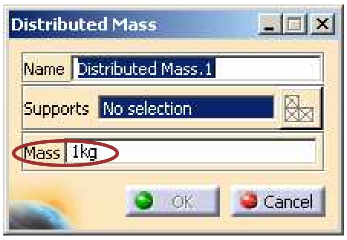
Instructor Notes:

How to Apply a Distributed Mass

Once you specify the distributed mass, the total mass remains constant independent of the geometric support. The point where the total mass is concentrated is the centroid of the selected geometry, or the handler of the virtual part.

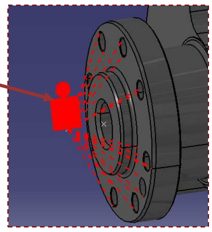
1 Click the Distributed Mass icon. 

The Distributed Mass dialog box is displayed



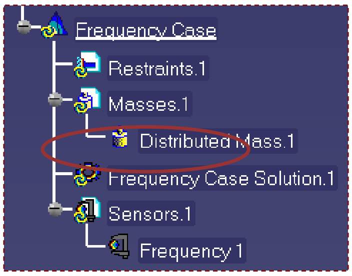
2 Select the support (a virtual part or a geometry).

Distributed mass symbol is visualized (here on a crankshaft)



3 Enter the value of the total mass

4 Click on OK



Copyright DASSAULT SYSTEMES

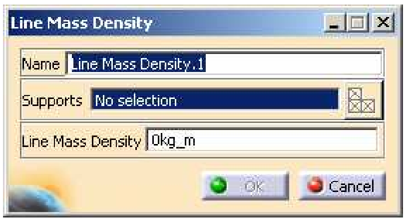
Instructor Notes:

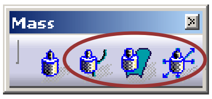
What are Mass Density and Distributed Mass and Inertia

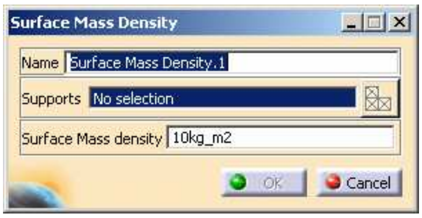
Mass densities are used to model purely inertial (non-structural) system characteristics, such as additional equipment.

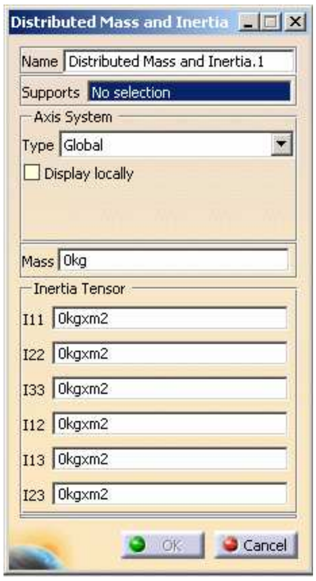
Mass Densities represent scalar mass density fields of given intensity, applied to geometries. They can be distributed on curves/edges, faces/surfaces and groups. This quantity remains constant independently of the geometry selection. Mass sets can be included in static cases: in this case, they are used for loadings based inertia effects

Distributed Mass and Inertia represents application mass and inertia values to virtual parts. Different inertia values for a same mass distribution will give different frequency values.









Copyright DASSAULT SYSTEMES

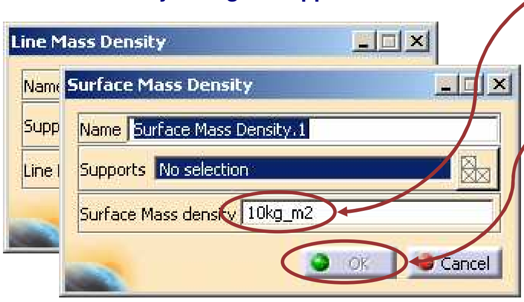
Instructor Notes:

How to Apply a Mass Density

Mass Densities represent scalar mass density fields of given intensity, applied to geometries. The user specifies mass density. The total mass then depends on the geometry selection.

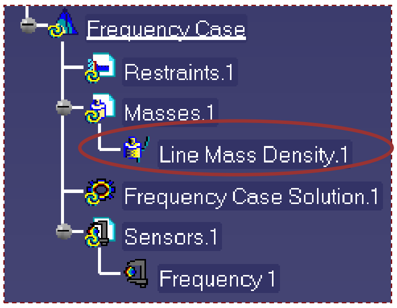
1 Click on a Mass Density icon

A mass density dialog box appears



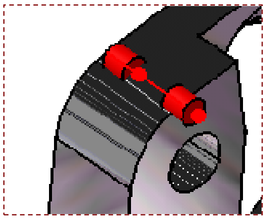
3 Specify the mass density

4 Click on OK



2 Select the support :

- **curves/edges if Line Mass Density**
- **faces/surfaces if Surface Mass Densities**
- **groups**

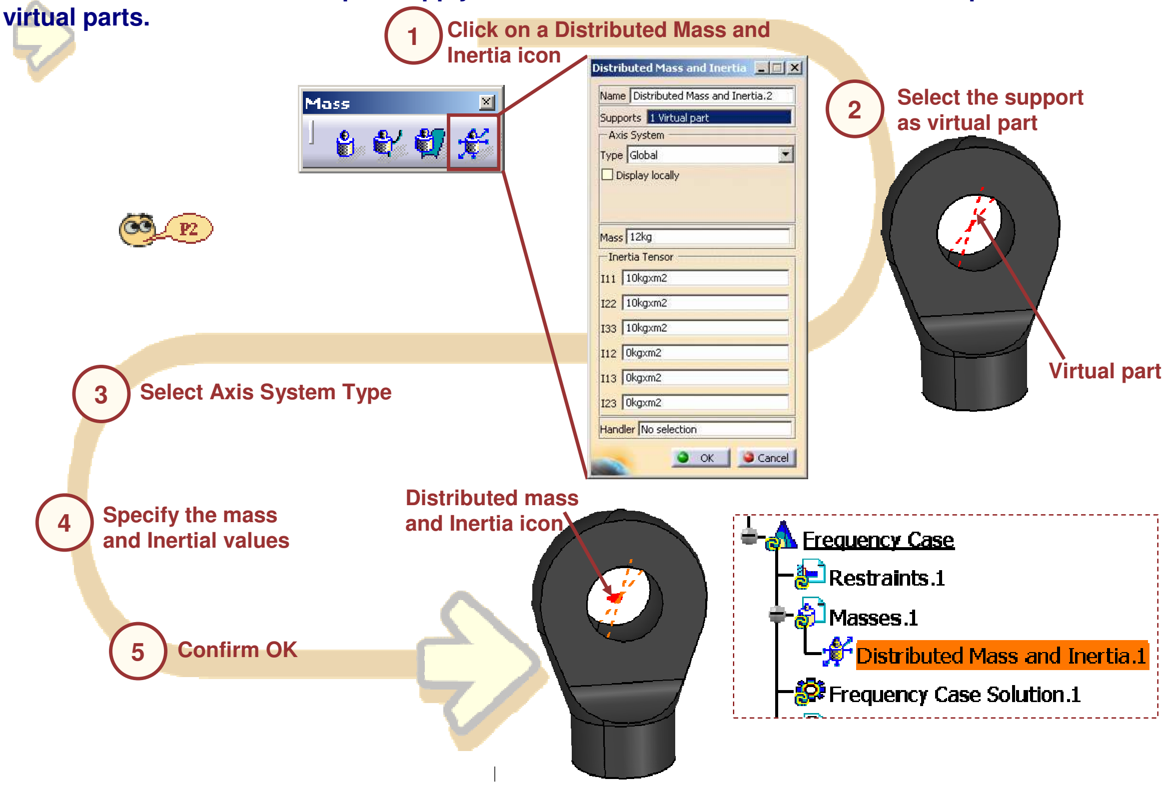


Copyright DASSAULT SYSTEMES

Instructor Notes:

How to Apply Distributed Mass and Inertia

Distributed Mass and Inertia helps to apply different mass and inertia values to mesh parts or virtual parts.



Copyright DASSAULT SYSTEMES

Instructor Notes:

To Sum Up ...

In the Advanced Pre-Processing tools, you have seen how to :

- Define advanced Loads
- Define advanced restraints
- Select a Mesh Part
- Define Virtual Parts
- Define User Material
- Define a Frequency Analysis

Copyright DASSAULT SYSTEMES

Instructor Notes:

Computation

In this lesson you will learn how to compute a frequency analysis and use some advanced computation tools

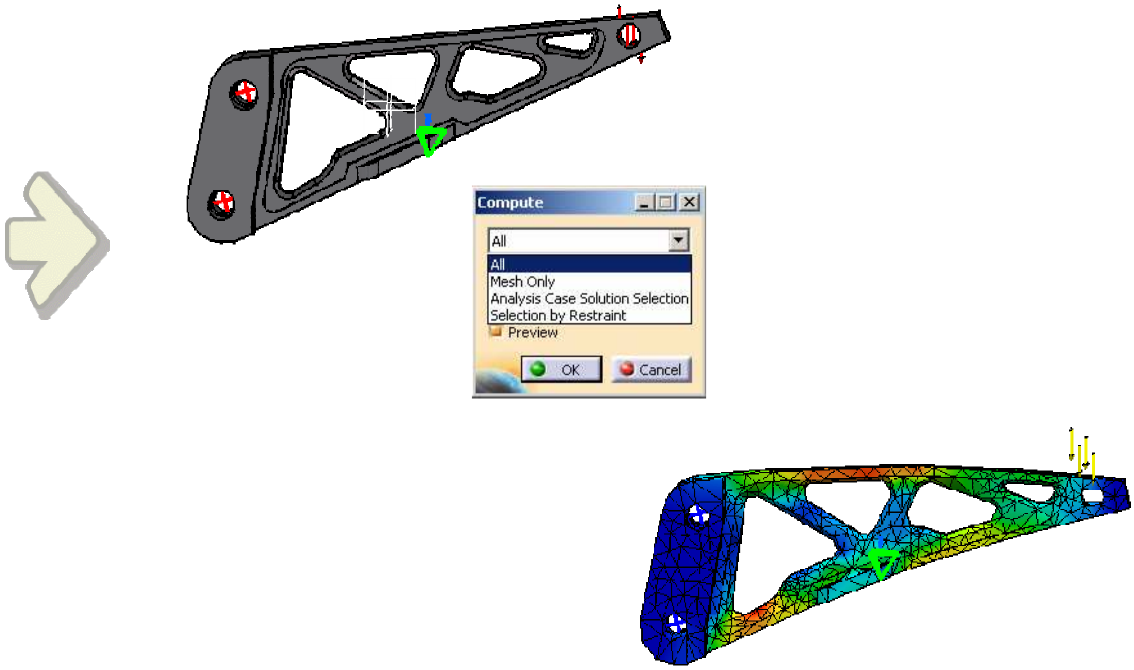
- Computing a Frequency Solution
- Computing with Adaptivity
- Historic of Computation
- To Sum Up

Copyright DASSAULT SYSTEMES

Instructor Notes:

Computing a Frequency Solution

You will learn how to compute a frequency solution.



Copyright DASSAULT SYSTEMES

Instructor Notes:

Introduction

At this step of your work you must make sure that your materials, restraints and loads are successfully defined. The computation will generate the analysis case solution, along with partial results for all objects involved in the definition of the Analysis Case.

The primary Frequency Solution Computation result consists in a set of frequencies and associated modal vibration shape vectors whose components represent the values of the system DOF for various vibration modes.

The program can compute simultaneously several Solution object sets, with optimal parallel computation whenever applicable.

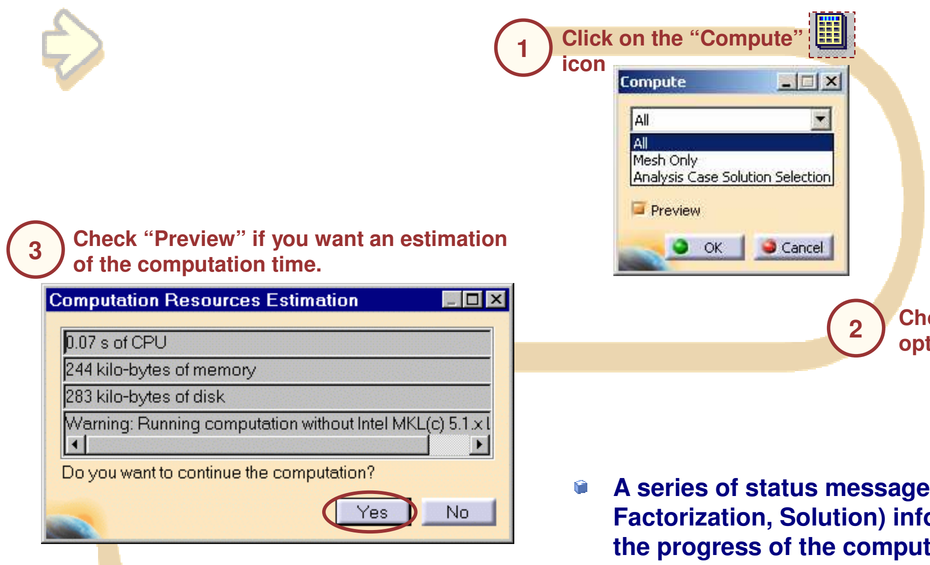
The combo box allows you to choose between several options for the set of objects to update:

- **All** : All the objects defined in the analysis features tree will be computed
- **Mesh only:** the preprocessing parts and connections will be meshed. The preprocessing data (loads, restraints and so forth) will be applied onto the mesh. In case the “Mesh only” option was previously activated, you will then be able to visualize the applied data on the mesh by using the Visualization on Mesh option (contextual menu)
- **Analysis Case Solution Selection:** only a selection of user-specified Analysis Case Solutions will be computed, with an optimal parallel computation strategy
- **Selection by Restraint:** only the selected characteristics will be computed (Properties, Restraints, Loads, Masses).

Copyright DASSAULT SYSTEMES

Instructor Notes:

How to Compute a Frequency Solution



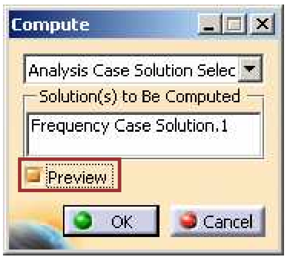
- A series of status messages (Meshing, Factorization, Solution) informs you about the progress of the computing process.
- Upon successful completion of the computation, the FE mesh is visualized on your part, and the status of all objects in the analysis features tree up to the Frequency Case Solution Object set is turned to valid.
- analyze the report of the computation
- visualize images for various results

Copyright DASSAULT SYSTEMES

Instructor Notes:

Additional Information

- If several frequency analysis cases have been defined, you can compute them simultaneously, following the same procedure.
- You can also compute only a selection of cases by selecting Analysis Cases Solution Selection. You can then specify the cases in the Compute dialog box.
- You can compute vibration modes either for the free system or for the system subjected to supports. In the first case there are no restraints so your Analysis Case must contain no restraints objects set.
- To display CPU time and memory requirement estimates prior to launching any computation, check Preview in Compute dialogue box.



Copyright DASSAULT SYSTEMES

Instructor Notes:

Troubleshooting

When computing a frequency analysis, some error messages can appear.

This message will be displayed if you compute a 'frequency' analysis and forget to apply the restraints (to fix the part).

Solution: Apply a Proper restraint on the part and compute again.

Note: To compute the vibration mode of a part (which implies no restraint), a "frequency analysis" is not the adequate case. You must define a "Free Frequency" analysis which does not need any restraint set.

Copyright DASSAULT SYSTEMES

Instructor Notes:

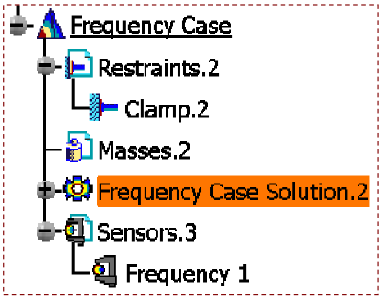
Selecting Frequency Solution Parameters

The definition parameters of an analysis case, (available, in the ELFINI structural analysis product, in the New Case dialog box at the time of a Case Insertion) cannot be modified once the Case has been created. They must not be confused with the computation parameters of a case solution, which are proposed by default at creation, and are editable afterwards.

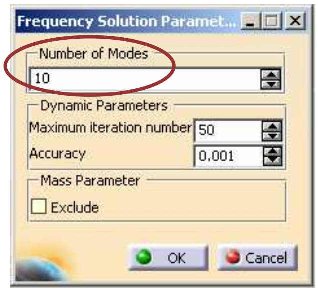
The Frequency Solution Parameters dialog box contains the following parameters:

- Number of modes
- Method (Iterative subspace or Lanczos)
  Iterative subspace is used for complex problems, more accurate however takes more time while Lanczos method is faster and used for smaller problems. (These methods are available only with EST Product.)
- Dynamic parameters (Maximum iteration number and Accuracy)

1 Double-click on the Solution objects set in the analysis feature tree to display the Frequency Solution parameters dialog box.



2 Modify the parameters you want



3 Click on 'OK'

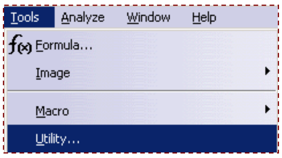
Copyright DASSAULT SYSTEMES

Instructor Notes:

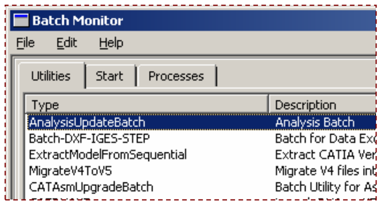
How to Launch Batch Processing

While CATIA computes your analysis, the interactive mode is not available. So, you may launch a batch which performs the computation.

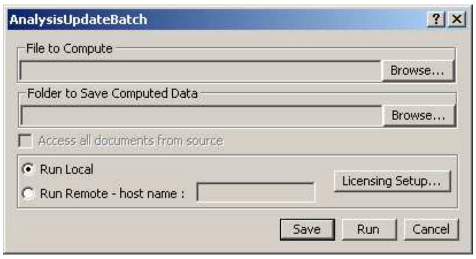
1 Go to Tools > Utility…



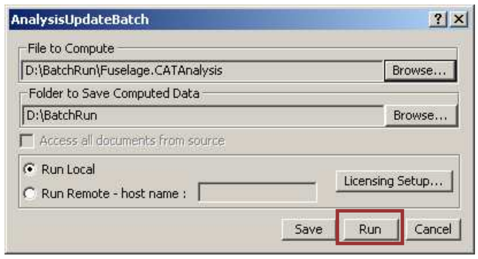
2 Double-click 'AnalysisUpdateBatch'



3 Enter analysis file to compute and folder for saving computation files and select Batch run mode



4 Click on 'Run' to start batch mode computing



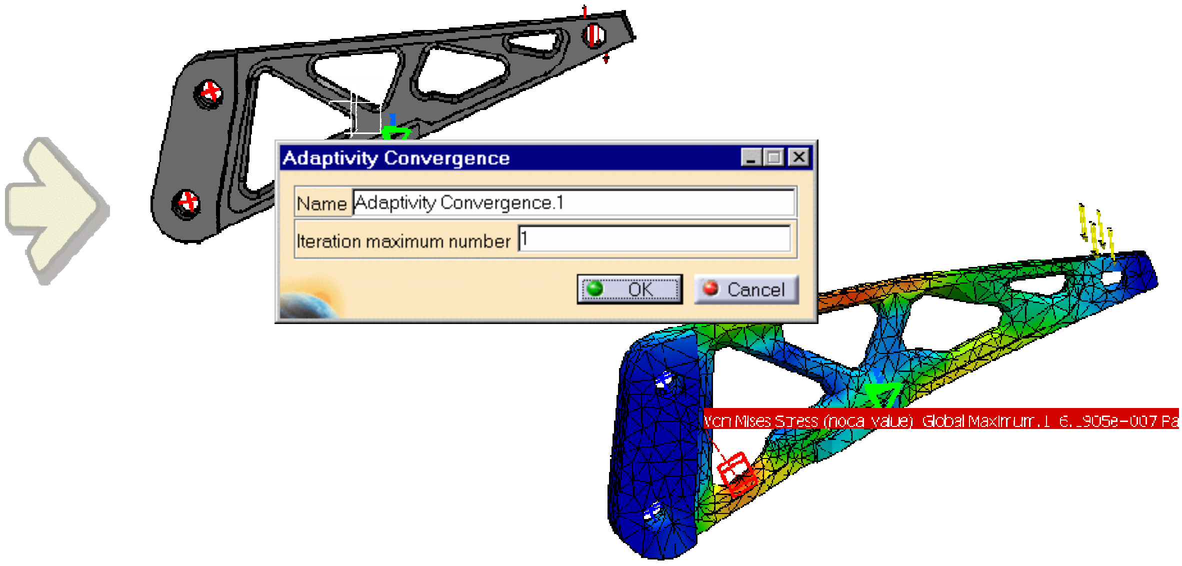
Copyright DASSAULT SYSTEMES

Instructor Notes:

Copyright DASSAULT SYSTEMES

Computing with Adaptivity

You will learn how to compute an analysis taking into account the objectives error defined thanks to the adaptivity tools (Post-Processing)



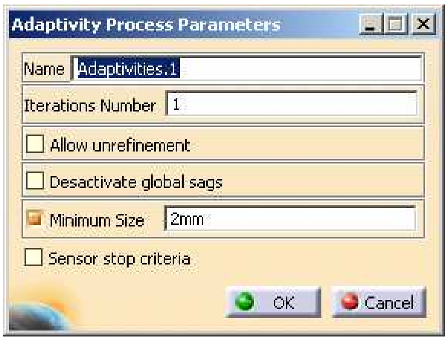
Copyright DASSAULT SYSTEMES

Note: *For all information about "adaptivity", please see the skillet "Mesh Adaptivity" in the post-processing lesson*

Instructor Notes:

About Adaptivity

'Adaptivity' consists in selectively refining the mesh in such a way as to obtain a desired result accuracy in a specified region (see post-Pros. Lesson)



- **The mesh refining criteria are based on a technique called predictive error estimation, which consists in determining the distribution of a local error estimate field for a given Static Analysis Case. "Adaptivity Management" consists in setting global adaptivity specifications and computing adaptive solutions.**
- **The Adaptivity functionalities are only available with static analysis solution or a combined solution that references a static analysis solution.**

- **After you have run the "Adaptive" computation, you can return to the static solution and check that the mesh has been refined according to your specifications in the Adaptivity Entities.**
- **You can create several Adaptivity Entities associated to different Static Solutions and corresponding to different regions of your part, i.e:**

  create several Adaptivity entities associated to the same Static Solution and corresponding to the different regions of your part

  create several Adaptivity Entities associated to different Static Solutions and corresponding to the same region of your part
- **The computation is such as all adaptivity entities specifications are simultaneously respected within the global Maximum Number of Iterations specification.**

Copyright DASSAULT SYSTEMES

Instructor Notes:

Computing with Adaptivity

Once you have defined an "adaptivity" you have to compute the analysis taking the adaptivity into account.

1 Enter the number of iterations

2 Check the different options if needed:

"Allow Unrefinement": Allows the global mesh size to be increased in certain areas

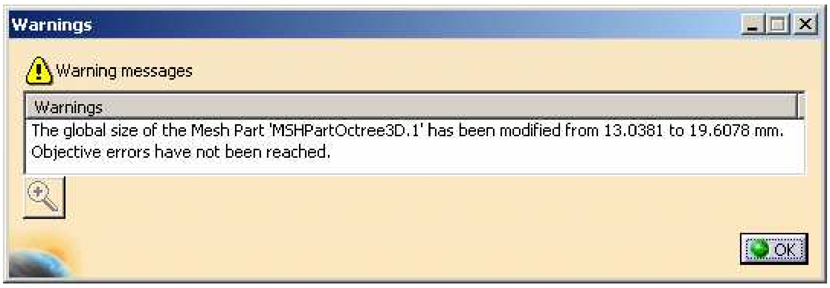
"Deactivate global sags": Allows you to de-activate the global sags defined in the mesh properties

"Minimum Size": allows you to impose a minimum size of element

"Sensor stop criteria": stops the computation when the sensor has converged

3 Click on Ok

At the end of the computation, a Warnings message appears to inform you if the objective error is not reached:

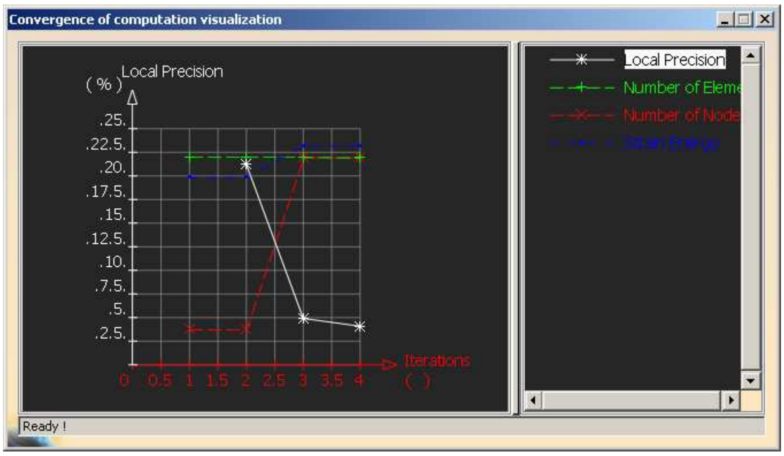


Copyright DASSAULT SYSTEMES

Instructor Notes:

Historic of Computation

You will see how to compare the parameter evolutions of successive computations.

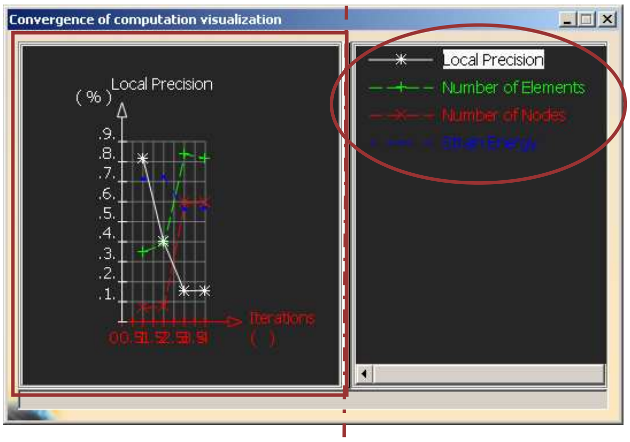


Copyright DASSAULT SYSTEMES

Instructor Notes:

About Historic

A Historic of computation allows you to compare new values possibly assigned to a CATAnalysis. For this you need to perform at least two computation operations. You can select the different options at the right of the dialog box and thus display the convergence information as desired.



On the right you will find:

- By default: Number of Elements and Nodes
- Static Case: energy, Von Mises max, disp. max, global error (results based on created sensors). If "Adaptivity Entities" were previously created, one local error per entity appears on the graph
- Frequency Case: frequency for each mode requested in the computation operation (results based on created sensors).

On the left side of the dialog box you can see the evolution of the parameter selected on the right for each computation (a computation is represented by a cross). The units and the scale are defined automatically according to the selected parameter.

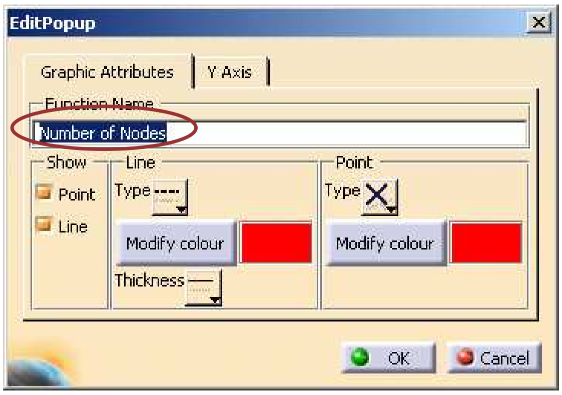
Copyright DASSAULT SYSTEMES

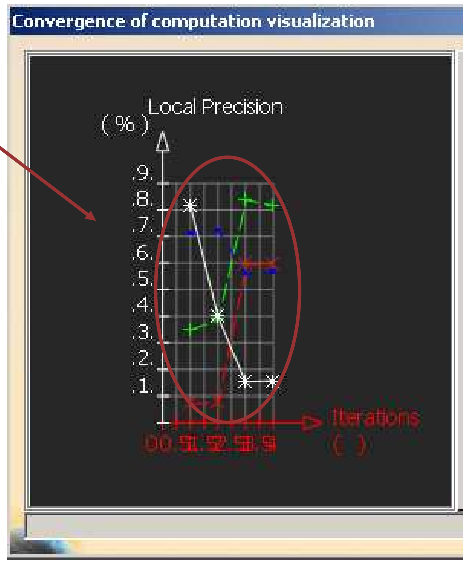
Instructor Notes:

About Lines Edition

You can edit the graph and modify its attributes.

By double-clicking on a line you will display the "EditPopup" dialog box





You can modify the "function name", show or no-show the points and lines, modify their characteristics.

Copyright DASSAULT SYSTEMES

Instructor Notes:

To Sum Up ...

In the Computation lesson, you have seen how to :

- Compute Frequency Analysis
- Compute the Analysis with adaptivity
- See the historic of computation

Copyright DASSAULT SYSTEMES

Instructor Notes:

GPS Advanced Post-Processing Tools

In this lesson you will see the advanced post-processing tools to visualize results and optimize analysis

- Results Visualization
- Results Management
- Refinement
- To Sum Up

Copyright DASSAULT SYSTEMES

Instructor Notes:

Results Visualization

In this lesson, you will see advanced tools to visualize results

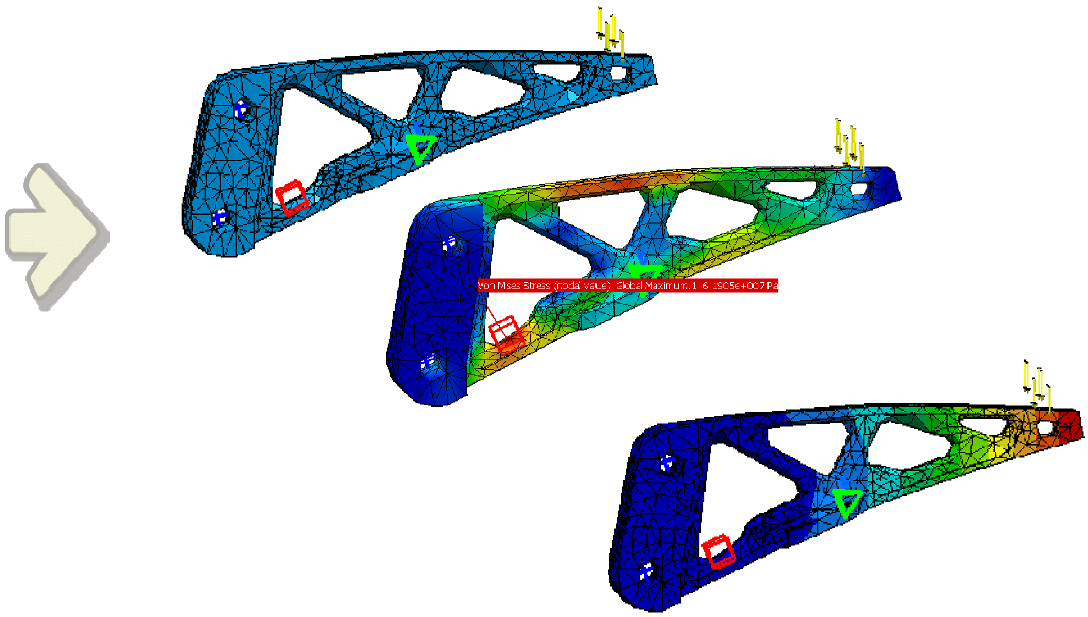
- Image Creation
- Cut Plane Analysis
- Results Visualization Recap Exercise
- To Sum Up

Copyright DASSAULT SYSTEMES

Instructor Notes:

Image Creation

You will see how to generate images corresponding to analysis results.

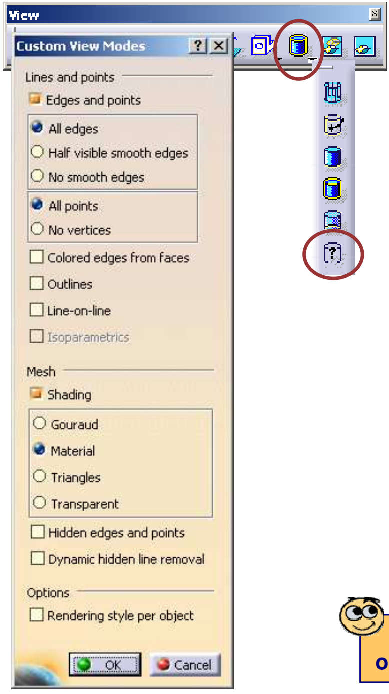


Copyright DASSAULT SYSTEMES

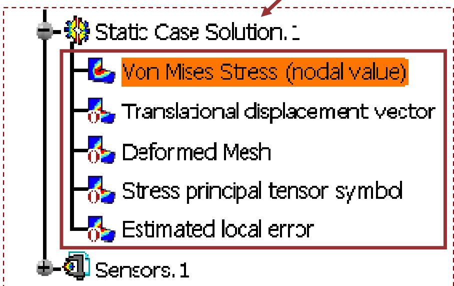
Instructor Notes:

Introduction

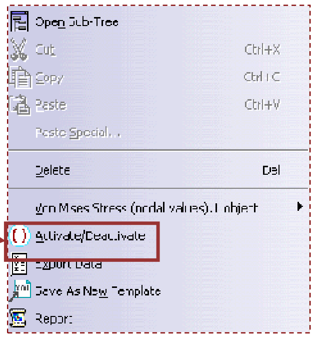
You can only visualize results after you have computed successfully your analysis. Before you begin, make sure the “Edges and points”, “All edges”, “Shading” and “Material” options are activated in the Custom View Modes dialog box:



Results are displayed in the specification tree under “Case Solution”



In the contextual menu, you can activate and de-activate images of the solution.



You will see the different options to visualize results.

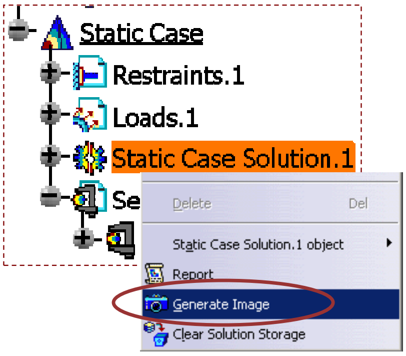
Copyright DASSAULT SYSTEMES

Instructor Notes:

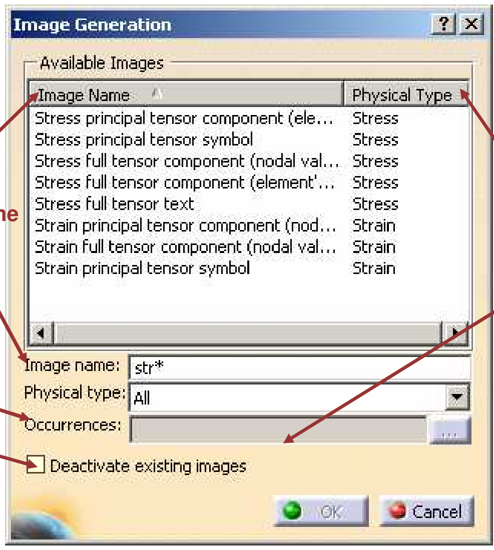
Generating Images

The "Generate Image" tool is available in the contextual menu of each Solution Case

To access this tool, right-click on the solution case in the specification tree. You can create many images at the same time : The multi-selection is allowed (press Ctrl key)



Filter by Image Name



Filter by Physical Type

For non-static cases, you have the possibility to select the current occurrence

If the "Deactivate existing Images" button is checked, it will have the same behavior as the "image generation" using the Image toolbar

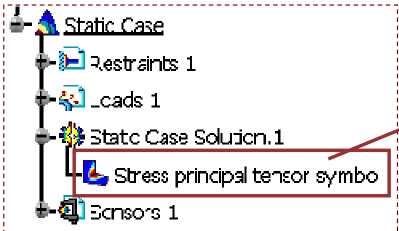
Copyright DASSAULT SYSTEMES

Instructor Notes:

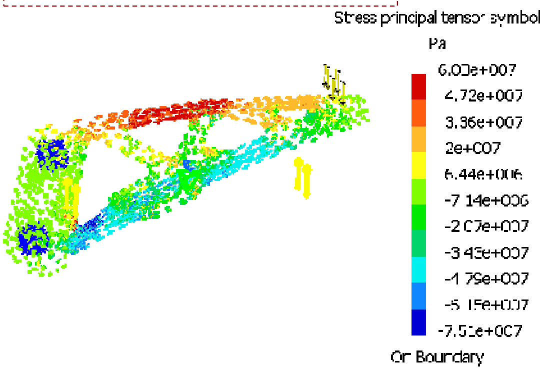
About Principal Stresses

Stress principal tensor symbol images are used to visualize principal stress field patterns, which represent a tensor field quantity used to measure the state of stress and to determine the load path on a loaded part.

At each point, the principal stress tensor gives the directions relative to which the part is in a state of pure tension/compression (zero shear stress components on the corresponding planes) and the values of the corresponding tensile/compressive stresses.



To edit the image using Image Edition dialog box you have to double click on the solution object in the specification tree



The principal values stress tensor distribution on the part is visualized in symbol mode, along with a color palette:

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

Copyright DASSAULT SYSTEMES

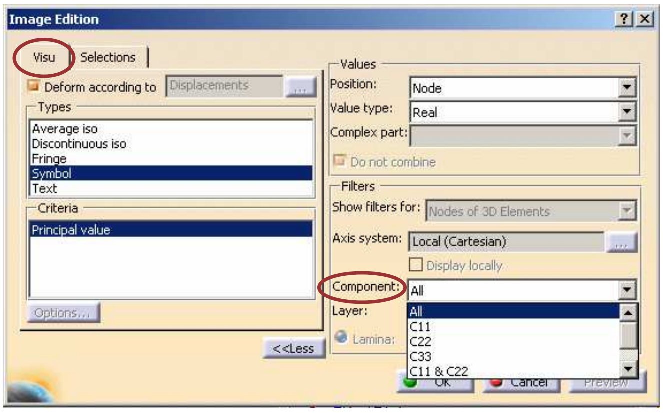
Instructor Notes:

Principal Stresses Image Edition

The “Image Edition” dialog box is composed of 2 tabs:

- **Visu: provides a list with visu types (Average-Iso, Discontinuous-Iso, Text) and a list with criteria (Principal-Value).**
- **Selections: In the case of CATProducts, pre-defined groups of elements belonging to given mesh parts can be multi-selected.**

  More: provides different filters. You can choose to generate images on nodes, elements, nodes of elements, center of elements or Gauss points of elements. You can also choose Value Type options.



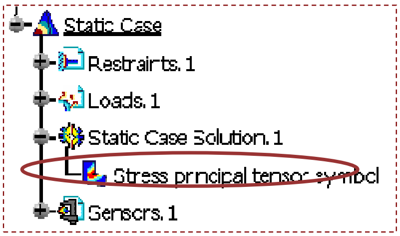
Copyright DASSAULT SYSTEMES

Instructor Notes:

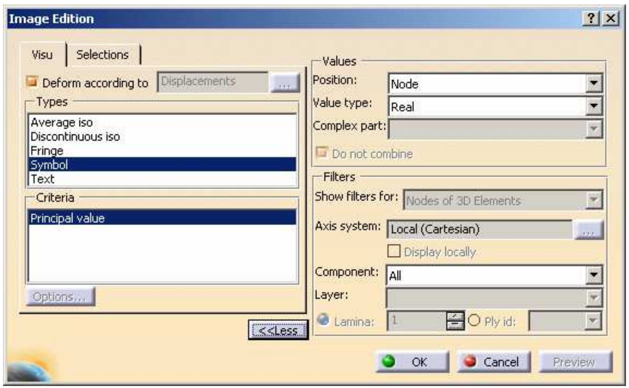
Visualizing Principal Stresses

This task shows how to generate Principal Stresses images on parts.

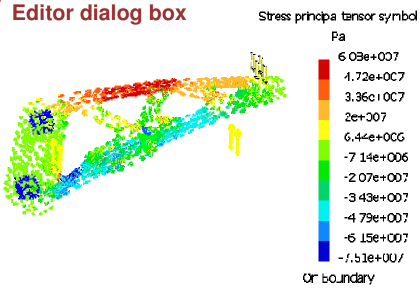
1 Click on the Principal Stresses icon



2 (Optional) Double-click on the generated Principal Stress Image object in the specification tree to edit the image



3 Click OK to quit the Image Editor dialog box



If needed, modify the parameters.

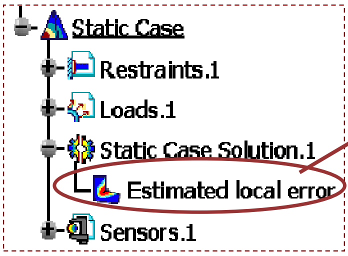
Copyright DASSAULT SYSTEMES

Instructor Notes:

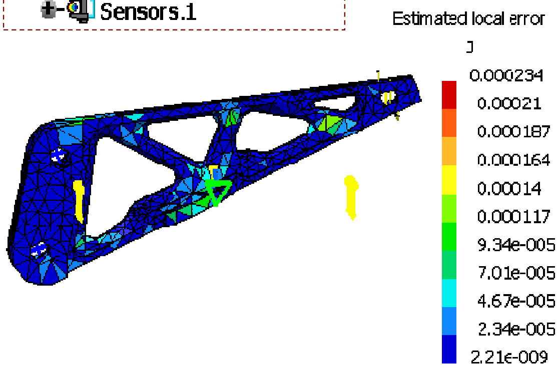
About Precisions

Estimated local error images are used to visualize computation error maps, which represent scalar field quantities defined as the distribution of energy error norm estimates for a given computation.

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.



To edit the image using Image Edition dialog box, double-click on the solution object in the specification tree.



This map provides qualitative information about the way 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 mesh definition step.

Copyright DASSAULT SYSTEMES

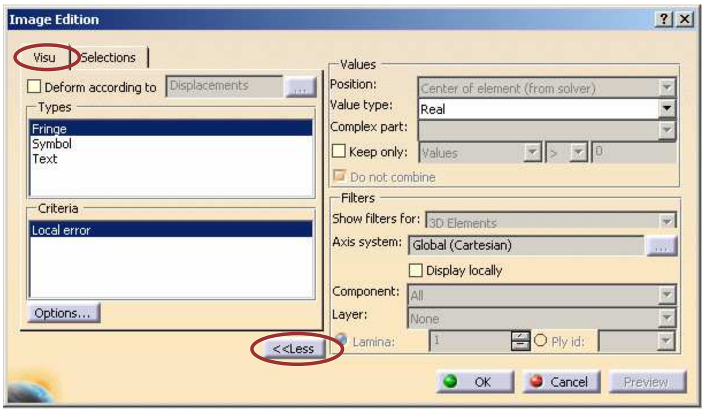
Instructor Notes:

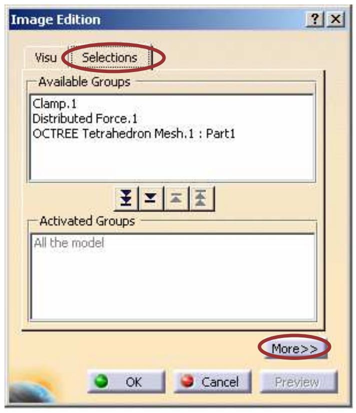
Precisions Image Edition

The "Image Edition" dialog box for Precision is composed of 3 tabs:

- **Visu: provides a list with visu types (Fringe, Symbol and Text) and a list with criteria (Local error is for Fringe visu type and Scalar for Symbol and Text visu types).**
- **Selections: In the case of CATProducts, pre-defined groups of elements belonging to given mesh parts can be multi-selected.**

More: provides different filters. You can choose to generate images on nodes, elements, nodes of elements, center of elements or Gauss points of elements. You can also choose Value Type options.





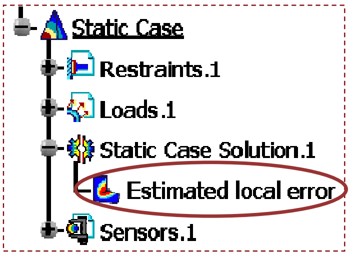
Copyright DASSAULT SYSTEMES

Instructor Notes:

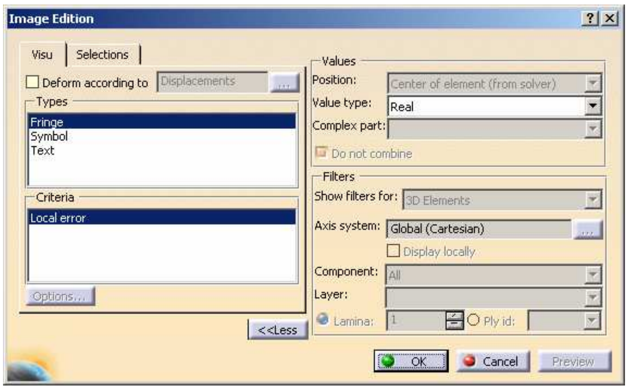
Visualizing Precisions

This task shows how to generate Precisions images on parts.

1 Click on the "Precision" icon

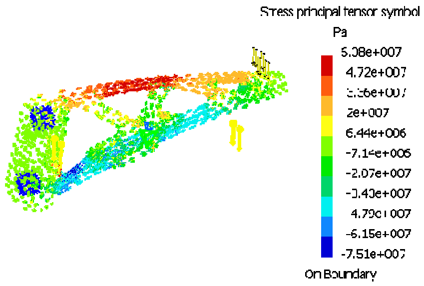


2 (Optional) Double-click on "Estimate Local error" in the specification tree to edit the image



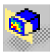
If needed, modify the parameters

3 Click OK to quit the Image Fem Editor dialog box

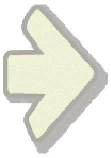


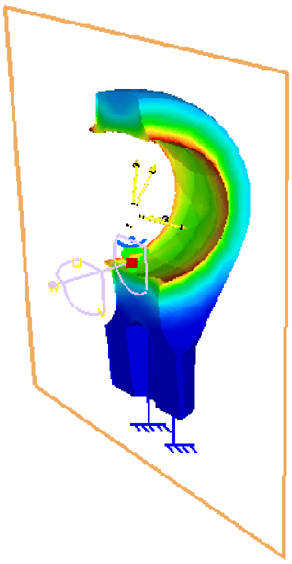
Copyright DASSAULT SYSTEMES

Instructor Notes:

Cut Plane Analysis 

You will see how to create sections on your structure.





Copyright DASSAULT SYSTEMES

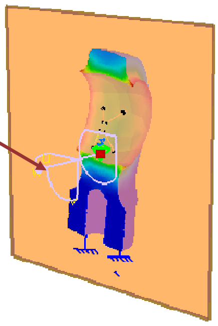
Instructor Notes:

About Cut Plane Analysis (1/2)

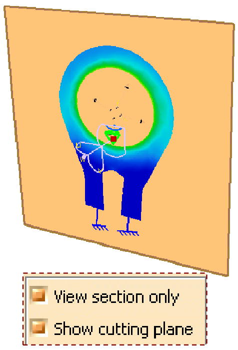
Cut Plane Analysis consists in visualizing sections of the structure to allow you to visualize results within the material.

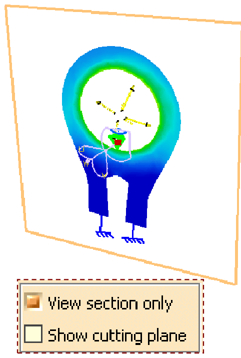
You can move or rotate the section plane using the compass which is automatically positioned on the part, with the cutting plane normal to its privileged direction.

compass



You can decide to display or hide the "cutting plane" to see the section only.





Copyright DASSAULT SYSTEMES

Instructor Notes:

About Cut Plane Analysis (2/2)

When you click on the Cut plane Analysis tool, section plane will take the last location and orientation.

Init Plane parameters: restores the default position and orientation of the cutting plane. The default position and orientation will be one of the following.

- **Before using this tool if you have positioned the compass on the geometry, then these position and orientation of the compass will be taken by the cutting plane.**
- **If the compass is not positioned on the geometry, the cutting plane will pass through the center of gravity of the geometry with the orientation depending on 3D view.**

To move the section plane, you can drag and drop the compass wherever you want on the part, or edit the compass (right click) and use the "Parameters for Compass Manipulation"

Copyright DASSAULT SYSTEMES

Instructor Notes:

Using ‘Cut Plane Analysis’ Tool

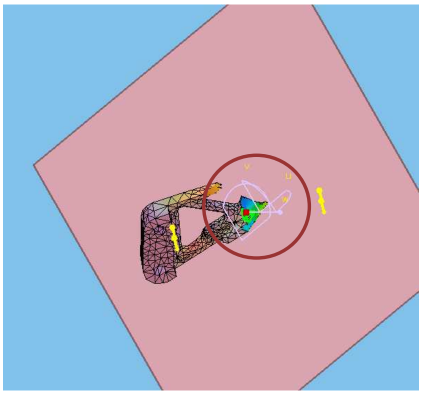
This task shows how to use the “Cut Plane Analysis”.

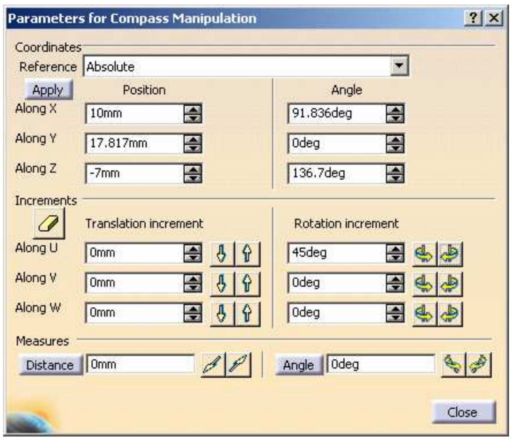
1 Select the image you want to section

Before you click on the “Cut Plane Analysis” icon, you can previously select a face on the structure. It will determine the first position of the section plane

2 Click on the “Cut Plane Analysis”

3 You can edit the compass for more accurate manipulations





Copyright DASSAULT SYSTEMES

Instructor Notes:

Exercise

‘Results Visualization’ Recap Exercise

10 min.

In this exercise you will use different visualization tools to view the analysis results that you have computed in the previously recap exercise. You will:

- **Visualize the Principal Stress**
- **Visualize the Precision Plot**
- **Use the Cut Plane Analysis tool**

Copyright DASSAULT SYSTEMES

Instructor Notes:

To Sum Up ...

You have seen how to :

- Display Images
- Use the Section Plane

Copyright DASSAULT SYSTEMES

Instructor Notes:

Results Management

In this lesson, you will learn how to use some tools for results exploitation

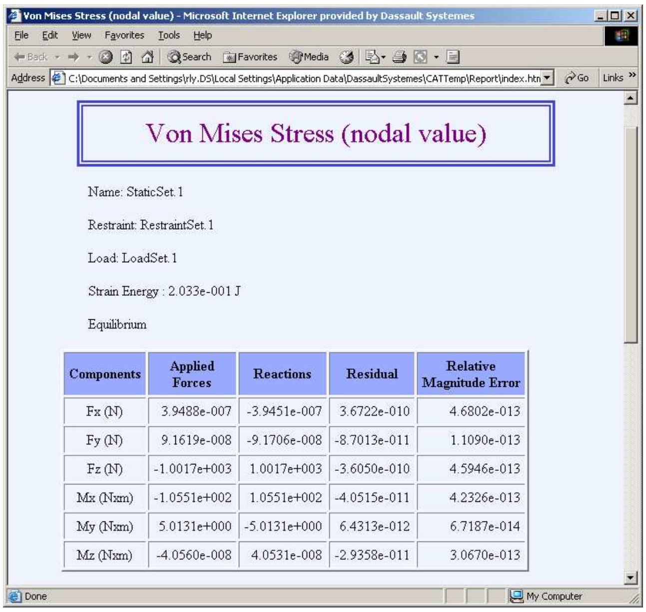
- Publishing Advanced Reports
- To sum up

Copyright DASSAULT SYSTEMES

Instructor Notes:

Publishing Advanced Reports

You will see how to customize and to publish advanced reports.



Von Mises Stress (nodal value)

Name: StaticSet.1

Restraint: RestraintSet.1

Load: LoadSet.1

Strain Energy : 2.033e-001 J

Equilibrium

Components	Applied Forces	Reactions	Residual	Relative Magnitude Error
Fx (N)	3.9488e-007	-3.9451e-007	3.6722e-010	4.6802e-013
Fy (N)	9.1619e-008	-9.1706e-008	-8.7013e-011	1.1090e-013
Fz (N)	-1.0017e+003	1.0017e+003	-3.6050e-010	4.5946e-013
Mx (Nxm)	-1.0551e+002	1.0551e+002	-4.0515e-011	4.2326e-013
My (Nxm)	5.0131e+000	-5.0131e+000	6.4313e-012	6.7187e-014
Mz (Nxm)	-4.0560e-008	4.0531e-008	-2.9358e-011	3.0670e-013

Copyright DASSAULT SYSTEMES

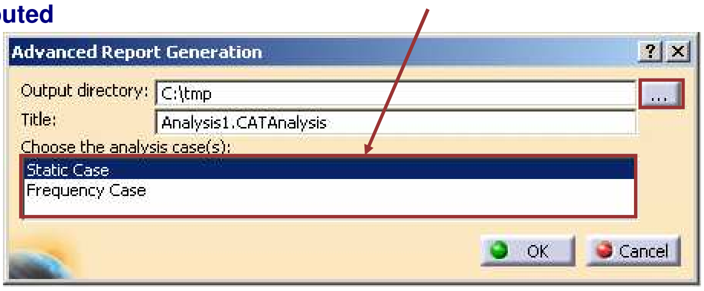
Instructor Notes:

About Advanced Reports

You can fully customize the report that you are going to publish.

Once an object set has been computed (meaning that the “user-defined specifications” are converted into solver commands), all data contained in the object are ready to be used in the “subsequent finite element computation process” and the object can be analyzed.

The “Advanced Report Generation” dialog box gives you the choice between the analysis case you have computed



Output directory: Pressing the button on the right gives you access to your file system for defining a path for the output Report file. You can edit the title of the report.

Title: You can modify the title if desired.

Copyright DASSAULT SYSTEMES

Instructor Notes:

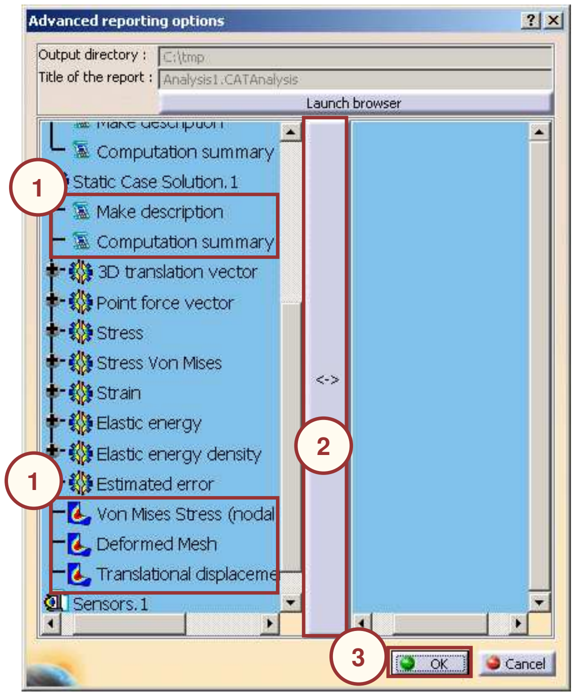
How to Use the Advanced Report Tool

The Generate Advanced Report tool allows you to define which information you need to extract from all the specifications before launching the browser, creating and if needed updating the output Report file.

1 Select the data you want to see in your report

2 Click on the middle arrow to validate the selection

3 Click on OK



Copyright DASSAULT SYSTEMES

Instructor Notes:

To Sum Up ...

You have seen how to:

- **Publish advanced report**

Copyright DASSAULT SYSTEMES

Instructor Notes:

Refinement

In this lesson you will see different ways to improve the precision of your results.

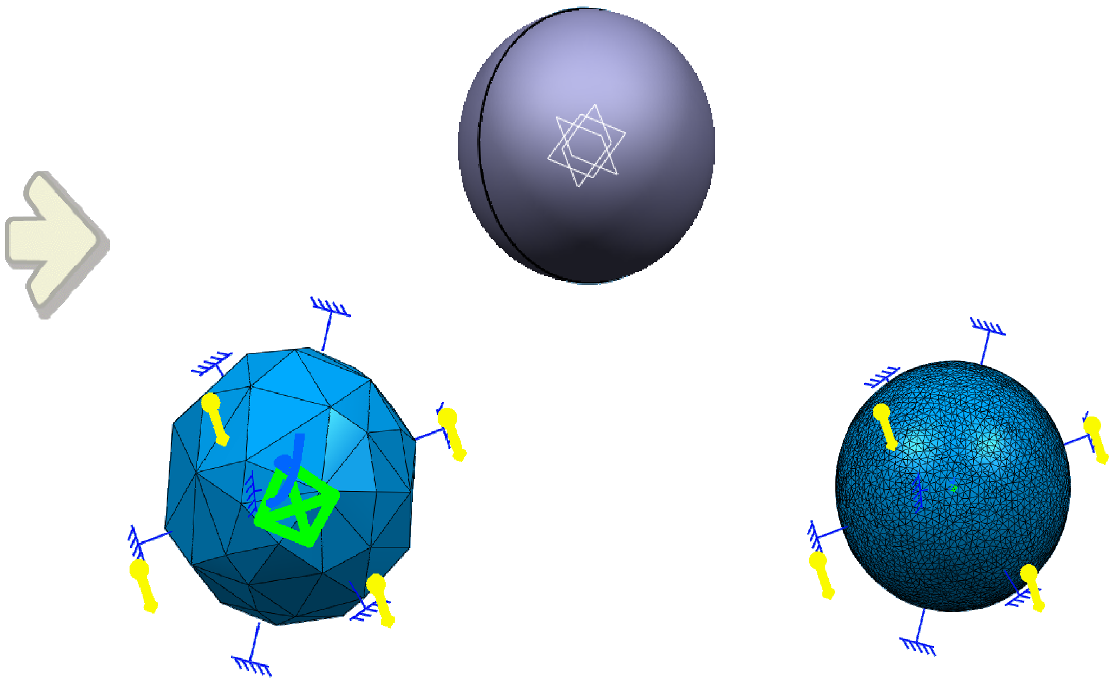
- Refining the Mesh
- Mesh Adaptivity
- To Sum Up

Copyright DASSAULT SYSTEMES

Instructor Notes:

Refining the Mesh

You will see how to refine a mesh for better analysis results.



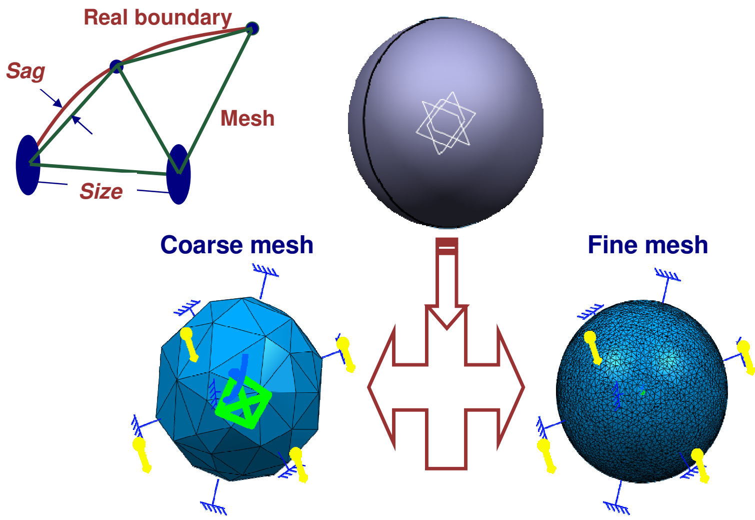
Copyright DASSAULT SYSTEMES

Instructor Notes:

Global and Local Mesh Refinement

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

The mesh "size" is the length of the element edges and the "sag" is the maximum distance allowed by the user between an element edge and the geometry. Consequently, a fine mesh and a small sag provide more accurate results.



Copyright DASSAULT SYSTEMES

Instructor Notes:

How to Refine a Global Mesh

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

2. Apply new values

3. Click on “OK”

You can define a Local size mesh and a local sag:

2’ Click on the Local tab:

3’ Double-Click on “Local size”/”Local sag”

4 Select the local area (support)

5 Enter a new value

6 Click on OK

Copyright DASSAULT SYSTEMES

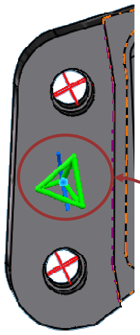
Instructor Notes:

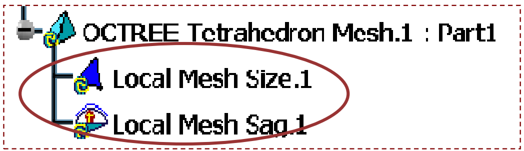
How to Refine a Local Mesh

1. Double-click either on the mesh specifications symbol or on the corresponding feature in the analysis tree.
2. Click on the Local panel:

3. Double-Click on “Local size”/”Local sag”
4. Select the local area (support)

5. Enter a new value
6. Click on OK





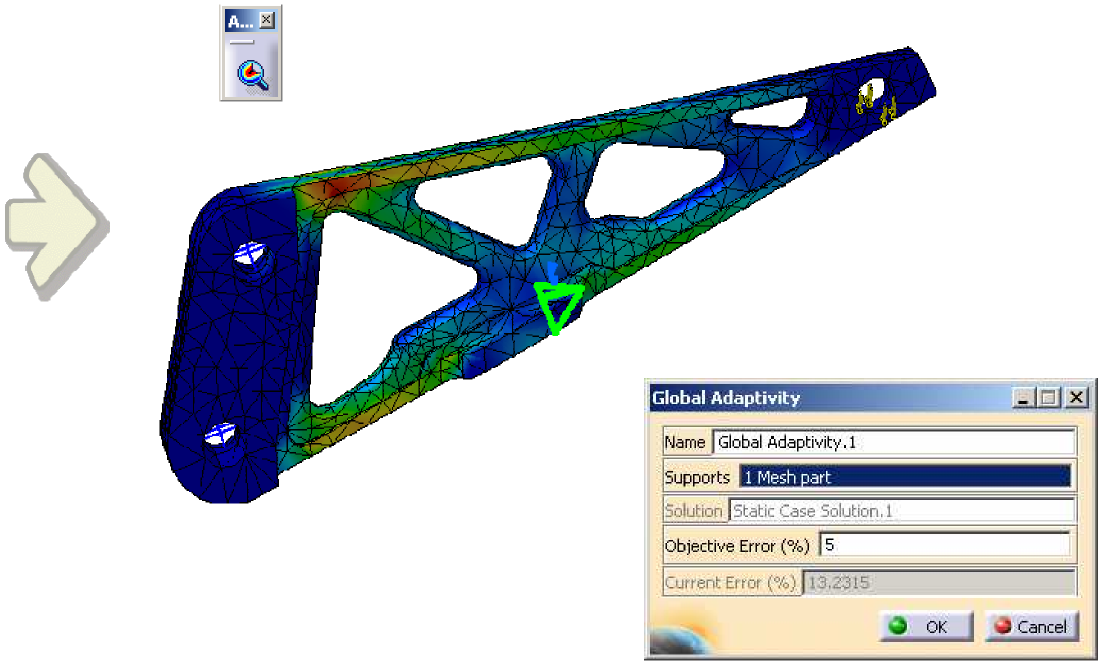
Symbol of Local Mesh and local Sag

Copyright DASSAULT SYSTEMES

Instructor Notes:

Mesh Adaptivity

You will see how to refine a given mesh in areas of interest only.



Copyright DASSAULT SYSTEMES

Instructor Notes:

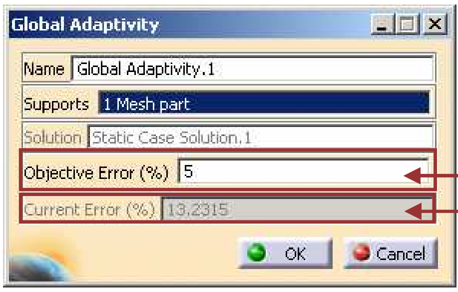
What is a Global Adaptivity

Global Adaptivity is a power tool of GPS which allows you to refine a given mesh to improve the current error.

The Finite Element Mesh is the collection of Nodes and Elements used to represent the system in order to transform the continuous mechanical problem into a discrete numerical problem. A finer mesh is expected to produce better results than a coarse mesh, but at a higher cost (more Memory and Time required to generate the results). This is also true locally: results are more precise in a region where the mesh is refined.

The 'mesh refining criteria' are based on a technique called "predictive error estimation", which consists in determining the distribution of a local error estimate field for a given Static Analysis Case.

'Global Adaptivity' specifications are applied on a body and are relative to the maximum error in the approximate computed solution relative to the exact solution. All the elements of the associated mesh part are taken into account automatically.



Here is your target error

Here is the error at the present time

Copyright DASSAULT SYSTEMES

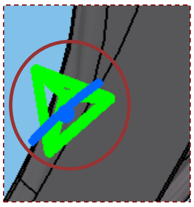
Instructor Notes:

How to Define a Global Adaptivity

Before you define a “ Global Adaptivity ” you must have previously meshed part.

1 Click on the “New Adaptivity Entity” icon

2 You must select a support: Click on the green tetrahedron



3 Enter your required ‘Objective Error’

4 Click on Ok

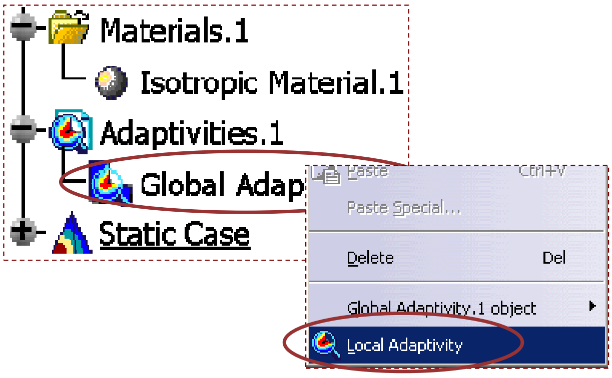
Copyright DASSAULT SYSTEMES

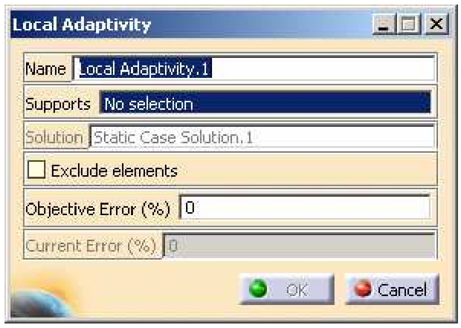
Instructor Notes:

What is a Local Adaptivity

The user can define a local adaptivity specification, to locally overload the global objectives.

It is necessary to define a Global Adaptivity' specification. Local Adaptivity is optional but can be defined using the contextual menu:





Local Adaptivity specifications can be applied on different types of groups :

- **A geometry group (elements connected to an edge, a surface, a vertex)**
- **A box group**

Box groups (cube or sphere) are easier to manipulate: their volume is filled and made transparent to show the intersected part of the geometry. Besides, they can be snapped on extrema, whatever their nature.

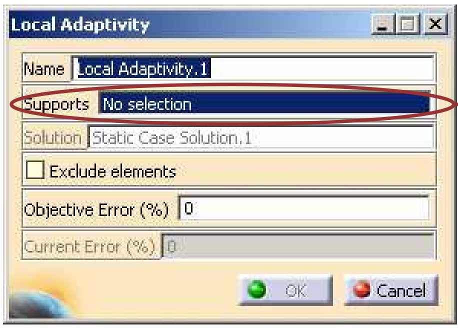
Copyright DASSAULT SYSTEMES

Instructor Notes:

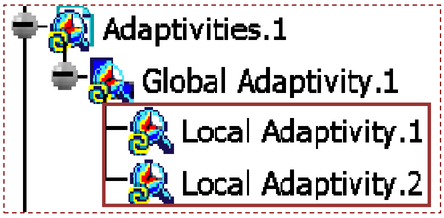
Why Locally Refine the Mesh

Adaptivity consists in selectively refining the mesh in such a way as to obtain a desired result accuracy in a specified region.

Thus, you can apply "Local Adaptivity" where stress, error, displacement… are maximum and consequently optimize the mesh in a very local area.



'Local Adaptivity' specifications can be applied on different types of groups



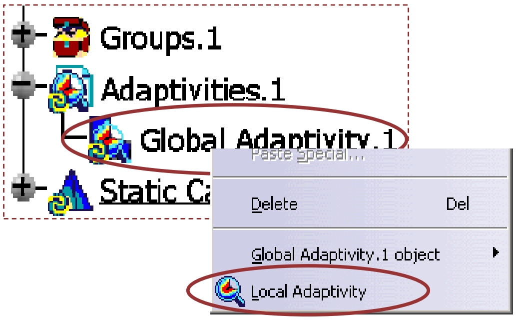
Copyright DASSAULT SYSTEMES

Instructor Notes:

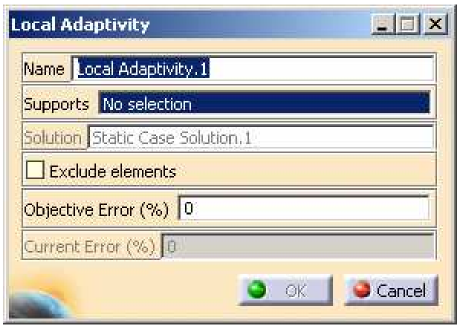
How to Define a Local Adaptivity

If you want to define a “Local”adaptivity you must have previously define a “Global”one

1 **Right click on “Global Adaptivity” in the tree**



The dialog box appears:



2 **Click on “Local Adaptivity”**

3 **Select a support**

4 **Enter an required ‘Objective Error’ value**

5 **Click on Ok**

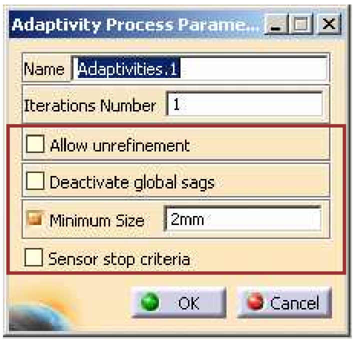
Copyright DASSAULT SYSTEMES

Instructor Notes:

Computing with Adaptivity Process Parameters

Once you have defined an "adaptivity" you have to compute the analysis taking the adaptivity into account.

1 Enter the number of iterations



2 Check the different options if needed:

"Allow Unrefinement": Allows the global mesh size to be increased in certain areas.

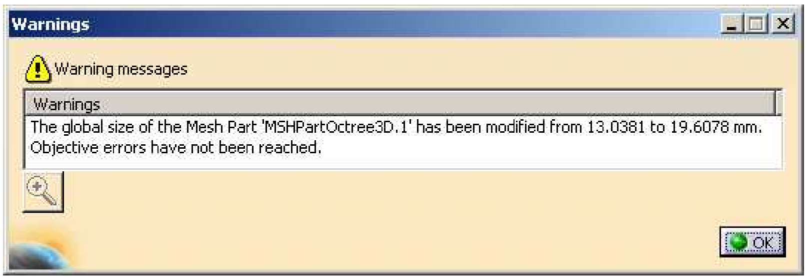
"Deactivate global sags": Allows you to de-activate the global sags defined in the mesh properties

"Minimum Size": allows you to impose a minimum size of element

"Sensor Stop Criteria": allows you to specify a sensor and tolerance(%) value.

3 Click on Ok

At the end of the computation, a Warnings message appears to inform you if the objective error is not reached:



Copyright DASSAULT SYSTEMES

Instructor Notes:

To Sum Up ...

In the “refinement” lesson, you have seen how to improve the analysis by:

- Refining the Mesh either globally or locally
- Using “Adaptivity”

Copyright DASSAULT SYSTEMES

Instructor Notes:

To Sum Up ...

In the Advanced Post-processing lesson, you have seen how to :

- **Visualize Results**
- **Manage Results**
- **Refine the analysis**

Copyright DASSAULT SYSTEMES

Instructor Notes:

Master Exercise: Frequency Analysis

You will practice concepts learned throughout the course by building the master exercise and following the recommended process

- Crank Shaft Frequency Analysis: Presentation
- Frequency Analysis on a Crank Shaft (1): Pre-Processing
- Frequency Analysis on a Crank Shaft (2): Computation
- Frequency Analysis on a Crank Shaft (3): Visualizing the Results

Copyright DASSAULT SYSTEMES

Instructor Notes:

Exercise

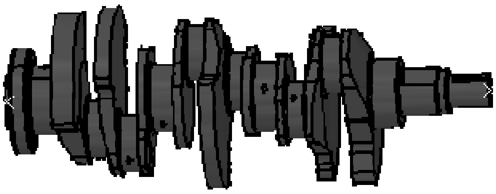
Exercises marked with this callout will work in P2 configuration only

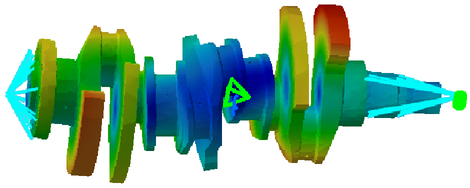
Frequency Analysis: Presentation

30 min

In this exercise you will run a frequency analysis on a Crank Shaft.
You will:

- **Define the Distributed Mass.**
- **Compute the Analysis.**
- **Visualize the modal frequencies and their associated deformation tendencies**





Copyright DASSAULT SYSTEMES

Instructor Notes:

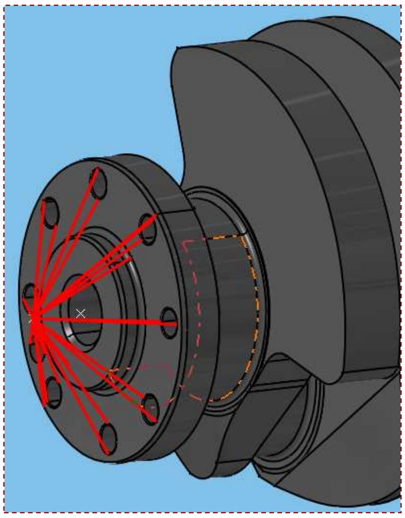
Exercise

Frequency Analysis on a Crank Shaft (Step1): Pre-Processing

15 min

Objectives:

- **Define the distributed mass**



Copyright DASSAULT SYSTEMES

Instructor Notes:

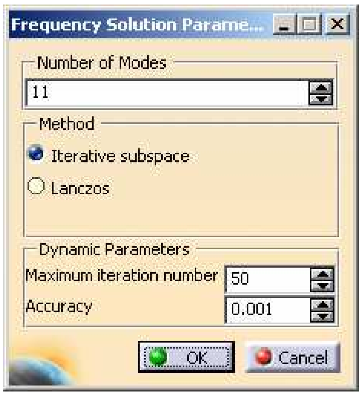
Exercise

Frequency Analysis on a Crank Shaft (Step2): Computation

5 min

Objective:

- **Compute the frequency analysis**



Copyright DASSAULT SYSTEMES

Instructor Notes:

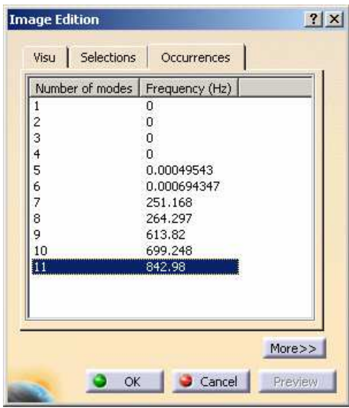
Exercise

Frequency Analysis on a Crank Shaft (Step3): Results

10 min

Objective:

- **Look at the different deformations tendencies for the 5 last modal frequencies**



Copyright DASSAULT SYSTEMES

Instructor Notes:

To Sum Up ...

In the course, you have seen how to :

- Use Advanced Pre-Processing tools
- Compute Frequency Analysis
- Use Advanced Post-Processing tools

Copyright DASSAULT SYSTEMES

Instructor Notes: