DASSAULT SYSTEMES

CATIA V5 Training
Foils

Generative Part Structural Analysis Fundamentals

Version 5 Release 19
September 2008

EDU_CAT_EN_GPF_FI_V5R19

Instructor Notes:

About this course

Objectives of the course

Upon completion of this course you will be able to:

- Understand the use of Finite Element Analysis

- Mesh a part with different element types and shapes, and define part properties

- Apply clamp, slider, and iso-static restraints; and force, moment and displacement loads

- Compute a static analysis for a single part

- Visualize images of the analysis results, and produce analysis reports

- Refine existing meshes in order to produce more accurate results

Targeted audience

Mechanical Designers

Prerequisites

Students attending this course should have knowledge of CATIA V5 Fundamentals

1 Day

Instructor Notes:

Table of Contents (1/3)

Instructor Notes:

Table of Contents (2/3)

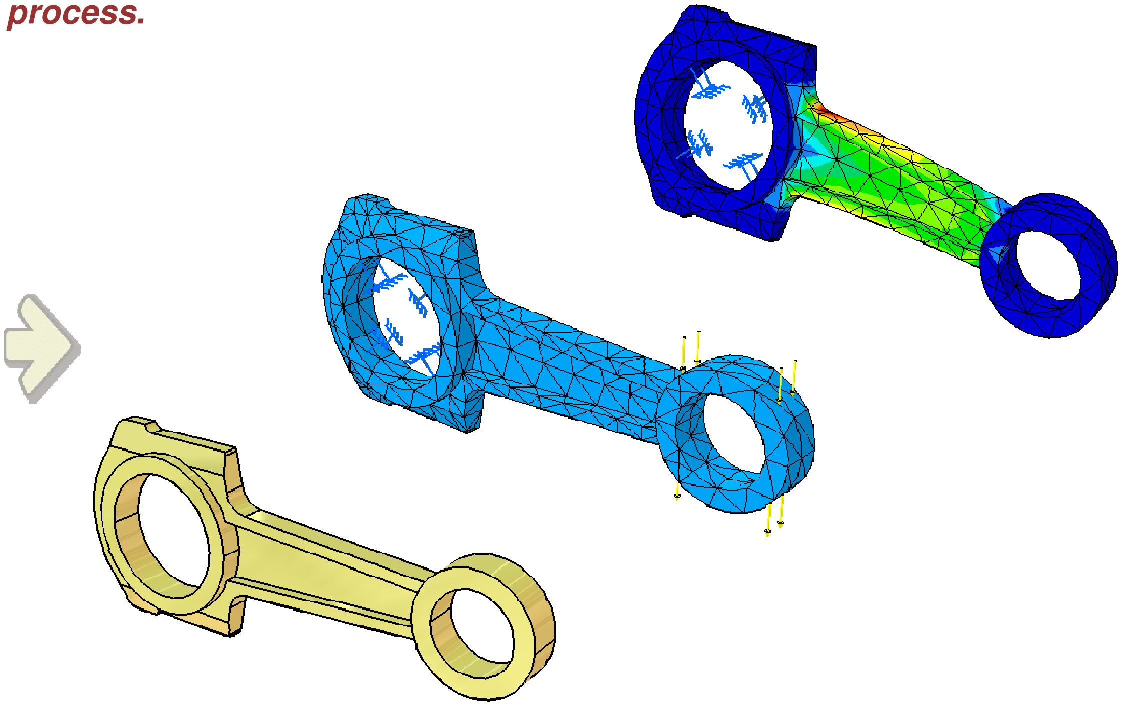
Instructor Notes:

Table of Contents (3/3)

Instructor Notes:

Introduction to Finite Element Analysis

In this chapter, you will learn the basic steps in Finite Element Analysis process.



Instructor Notes:

What is Finite Element Analysis (1/4)

Finite Element Analysis (FEA) is a numerical tool used for solving problems defined by ordinary or partial differential equations.

The most common Finite Element (FE) technique is displacement-based technique. In this approach, displacement is assumed to be an unknown quantity.

The problem is solved using FE methods to find out displacements.

The overall process can be subdivided into smaller steps shown on the pad:

Finite Element Analysis Process

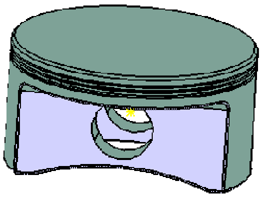
1. Pre-processing
2. Computation
3. Post-processing
4. Mesh Refinement Iterations
5. Report Generation

Instructor Notes:

What is Finite Element Analysis (2/4)

Pre-processing

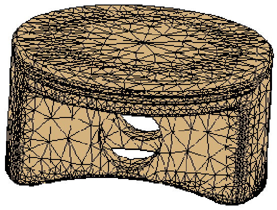
In this step, the actual physical problem is converted into equivalent Finite Element problem.



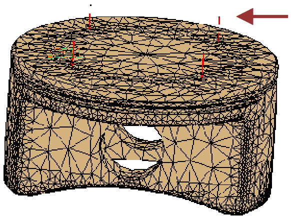
Actual part to be analyzed

- **The complex physical structure is converted into an equivalent Finite Element (FE) model.**

Finite Element Model representing actual part



- **The actual material properties are defined for FE model.**

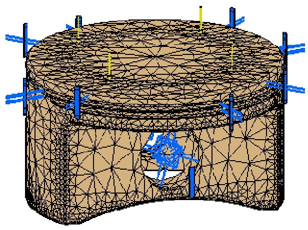


Applying Load on the FE Model

- **Actual physical Forces are converted into equivalent FE Loads.**

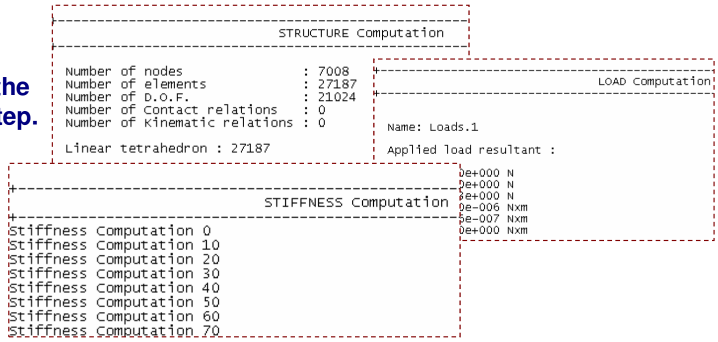
- **The actual physical Boundary Conditions are converted into equivalent FE Boundary Conditions.**

Applying boundary conditions to FE model



Instructor Notes:

What is Finite Element Analysis (3/4)

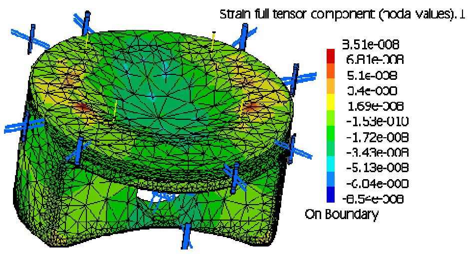


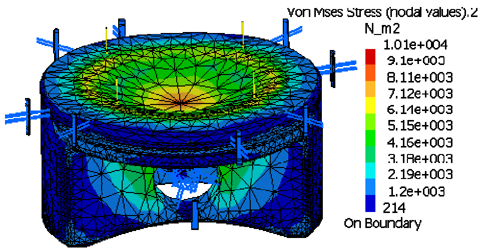
Computation

- **Standard FE solution procedures use the data provided by the pre-processing step.**
- **The FE model is solved to find out the unknown displacement values.**

Post-processing

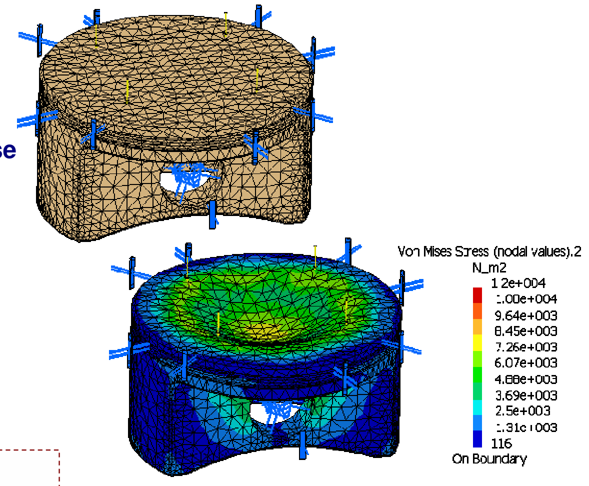
- **Using these displacement values, strains and stresses are calculated for the whole structure.**
- **You can study the deformation of structure, variation of strains and stresses throughout the structure.**





Instructor Notes:

What is Finite Element Analysis (4/4)



Mesh Refinement

- The first solution provides initial estimation of stress / strain values. In order to get a more accurate solution, the mesh needs to be refined and the computation is to be done again. Because when the mesh is refined, the computation is always invalidated.
- A number of mesh refinement and computation iterations are performed till the required solution accuracy is achieved.

Report Generation

- Once the required accuracy level is achieved, various plots such as Displacement, Principal Stress, Von-Mises Stress can be obtained.

MESH:

Entity	Size
Nodes	5584
Elements	20618

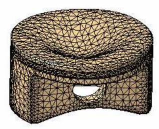
ELEMENT TYPE:

Connectivity	Statistics
TE4	20618 (100.00%)

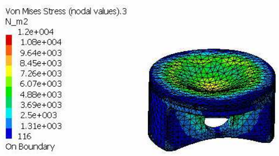
ELEMENT QUALITY:

Criterion	Good	Poor	Bad	Worst	Average
Distortion	13267 (64.35%)	5728 (27.78%)	1623 (7.87%)	57.22(	
Stretch	20				
Length Ratio	20				

Static Case Solution.1 - Deformed Mesh



Static Case Solution.1 - Von Mises Stress (nodal values).3



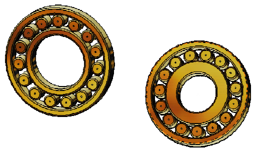
Energy	5.882e-009J
Error in Energy.2	4.795e-009J
Global Error Rate (%).3	53.809516907
Maximum Displacement.4	1.442e-005mm
Maximum Von Mises.5	12020.595N_m2

Instructor Notes:

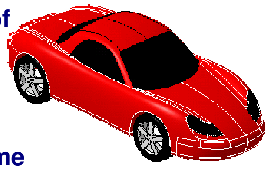
Why to Use Finite Element Analysis

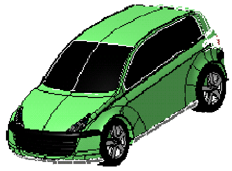
FEA can be applied to practically any problem having arbitrary shape including various boundary and loading conditions. This flexibility is not possible with classical analytical methods. Apart from this you have following advantages:

- You can validate product modifications to meet new conditions.

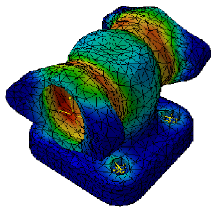


- You can verify a proposed product or structure, which is intended to meet the customer specification prior to manufacturing or construction.
- You can evaluate advantages and effectiveness of various product design alternatives without having any kind of experimental test setup.





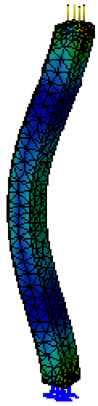
- It helps to implement the product concept first time right with corresponding cost savings thus minimizes the product life cycle time significantly.
- With FEA software tools, you can optimize product for minimum weight, minimum volume with negligible cost to improve product reliability and life.



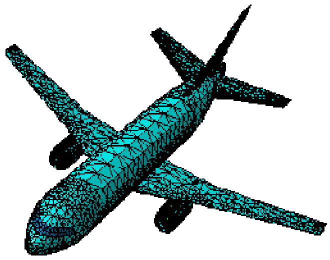
Instructor Notes:

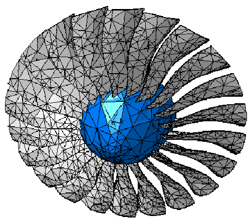
Application of Finite Element Analysis

The FEA is very important tool for engineering design. It is used to solve various complex problems.

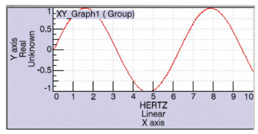


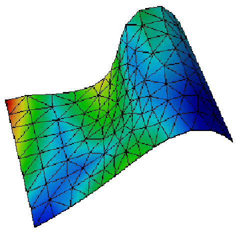
- Structural Analysis
- Dynamic analysis
- Buckling Analysis



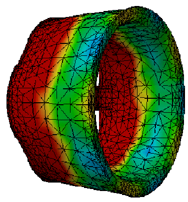


- Vibrations Analysis
- Acoustic Analysis
- Shock Analysis
- Crash Analysis





- Flow Analysis
- Thermal Analysis
- Coupled Analysis
- Mass diffusion
- Metal Forming
- Electrical Analysis
- Electromagnetic evaluations



Instructor Notes:

Introduction to GPS Analysis

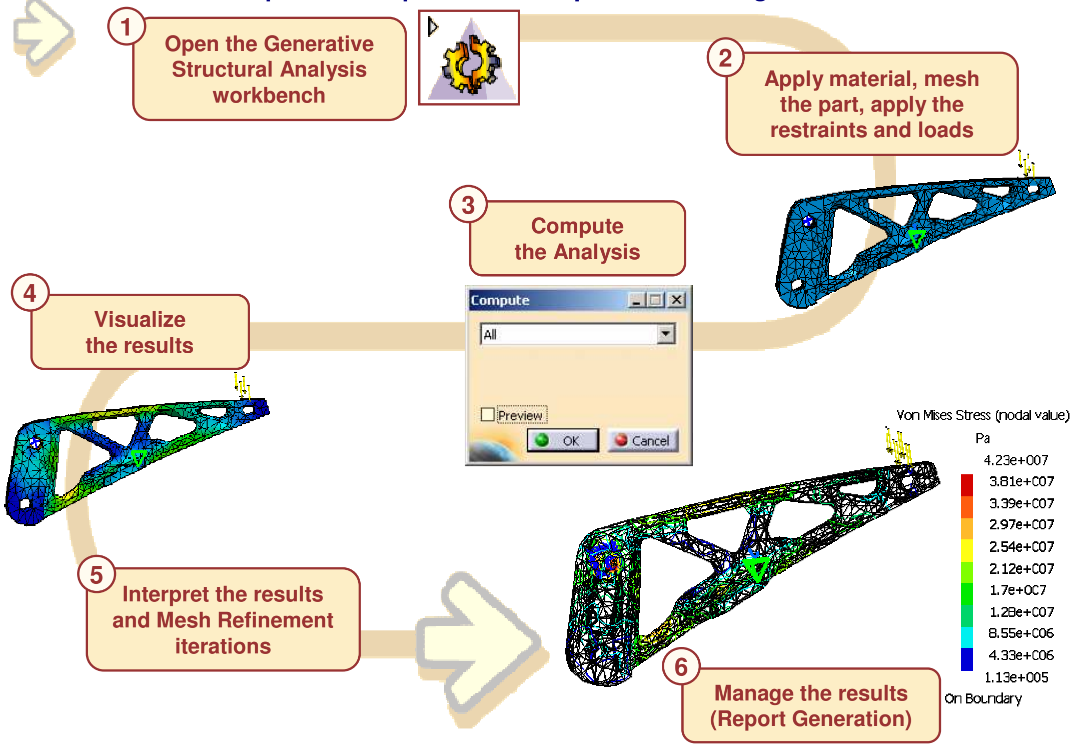
In this lesson, you will learn about the GPS Analysis Workbench and general FEA process in GPS.

- General FEA Process in GPS
- Accessing the GPS Analysis Workbench
- GPS Analysis Tree Structure

Instructor Notes:

General FEA Process in GPS

The GPS workbench provides tools and functionalities to perform FEA in CATIA.
Illustrated below are the FEA process steps that can be performed using GPS workbench.

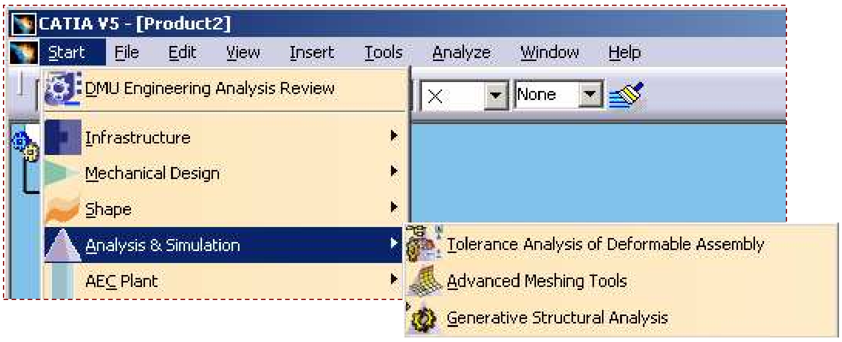


Instructor Notes:

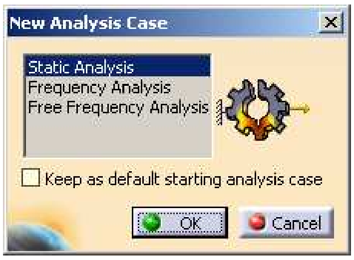
Accessing the GPS Analysis Workbench

1 From the MENU BAR: click on

Start > 'Analysis & Simulation' >Generative Structural Analysis



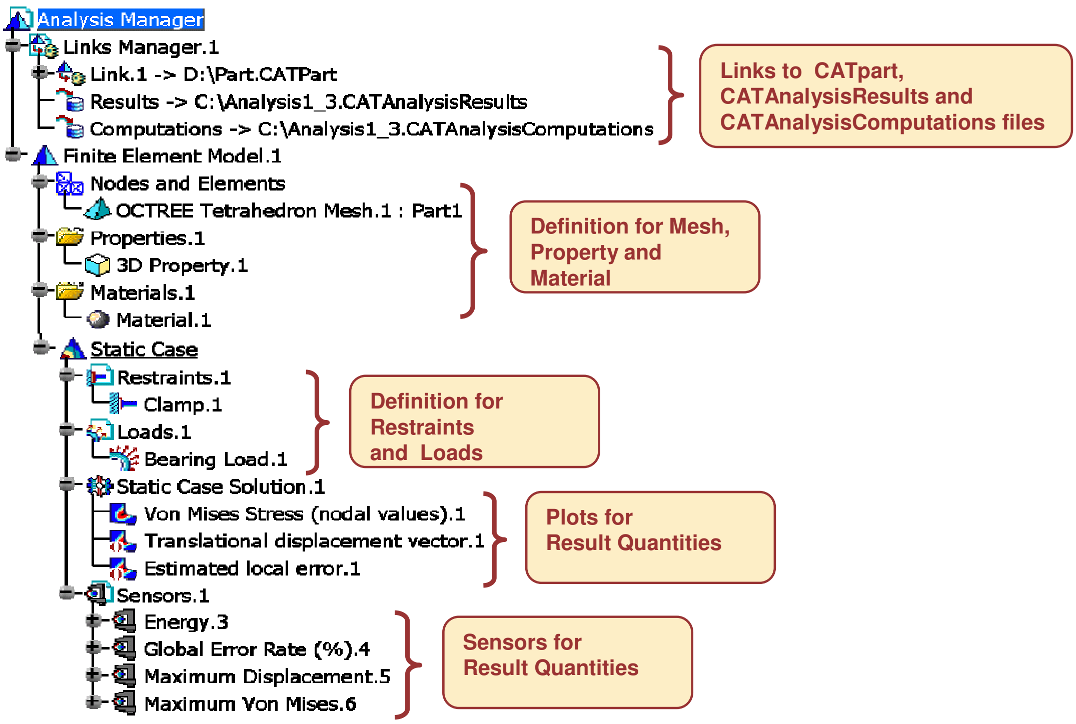
2 Select Static Analysis as a new Analysis case and click OK



Instructor Notes:

GPS Analysis Tree Structure

Entities created during process of FEA get mapped in tree structure as shown.



Instructor Notes:

GPS Pre-Processing

In this lesson, you will learn about the pre-processing steps in Static Analysis.

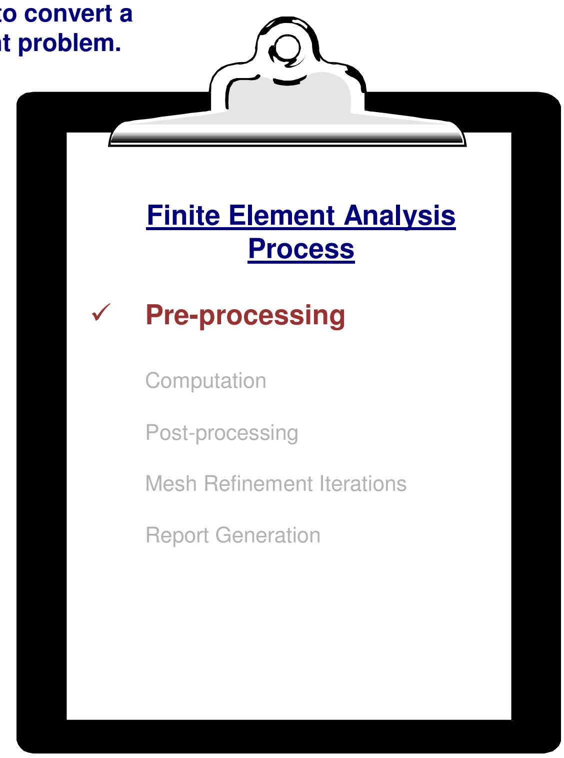
- What is Pre-processing
- GPS Pre-processing Tools
- Applying Material
- Managing Mesh-Part
- Applying Physical Property
- Defining Restraints
- Defining Loads
- Pre-Processing Recap Exercise
- Model Checker

Instructor Notes:

What is Pre-processing

Pre-processing involves performing the steps required to convert a given physical problem into an equivalent Finite Element problem.

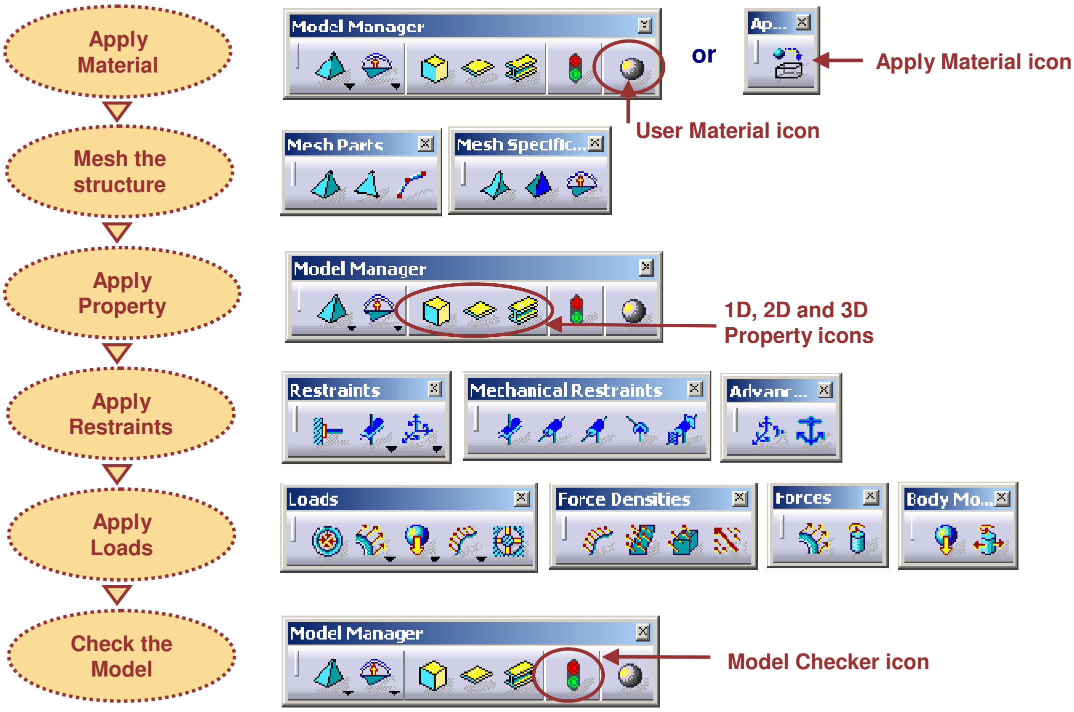
1. **Applying Material Structural Property, which is required to calculate deformation, strains and stresses**
2. **Meshing to create Finite Element model using Elements and Nodes**
3. **Applying Physical Property to associate the physical properties and material to the Finite Element Model**
4. **Applying Restraints on the FE Model, which represents the actual Physical Boundary Conditions**
5. **Applying Loads on the FE model, which represents the actual Physical Forces acting on structure**
6. **Model Checking to validate if all the pre-processing steps have been performed correctly and nothing is missed out**



Instructor Notes:

GPS Pre-processing Tools

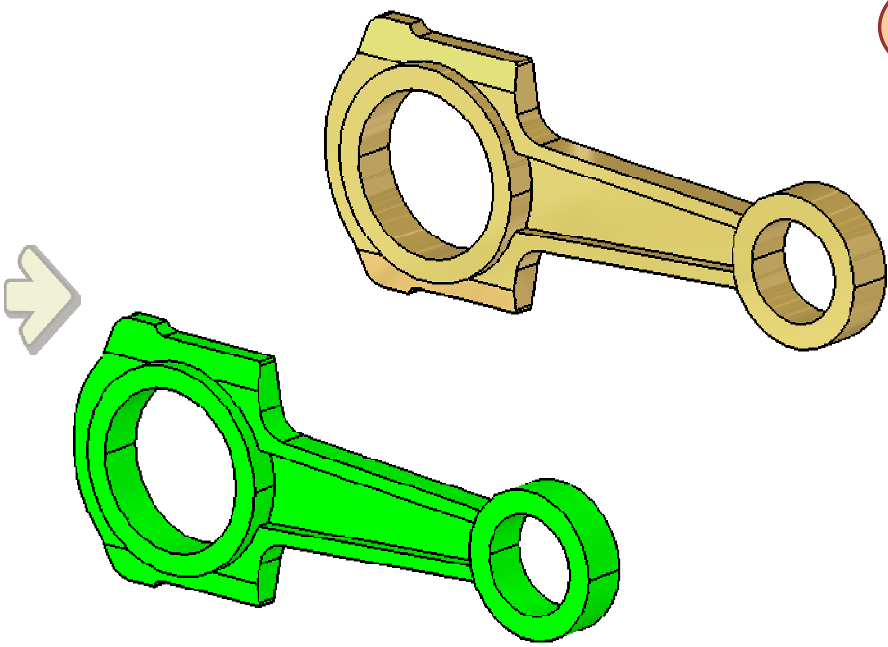
Following is the list of tools available in the 'Generative Structural Analysis' workbench. These tools are used for performing the pre-processing steps as illustrated below:

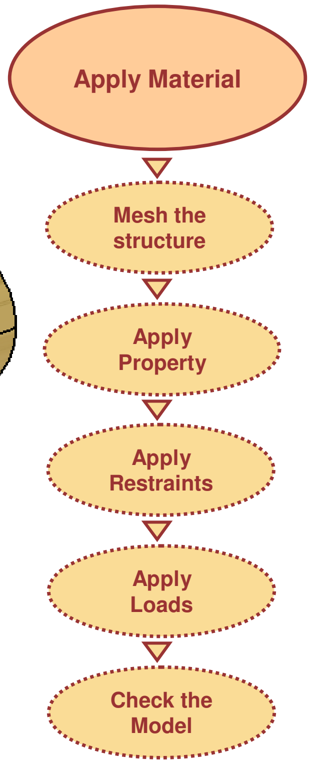


Instructor Notes:

Applying Material

You will learn how to apply material properties to components.



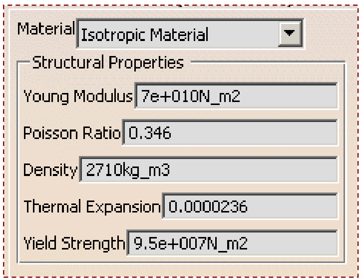


Instructor Notes:

Material Property

Structural properties of material are required to calculate deformation, strains and stresses. These include:

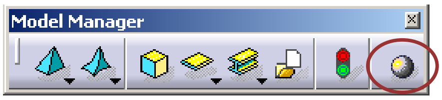
- **Young Modulus**
- **Poisson Ratio**
- **Density**
- **Thermal Expansion**
- **Yield Strength**



You can apply material in following two ways:

A. Select the material from the default Material catalog.

B. Create your own material with required properties.



Instructor Notes:

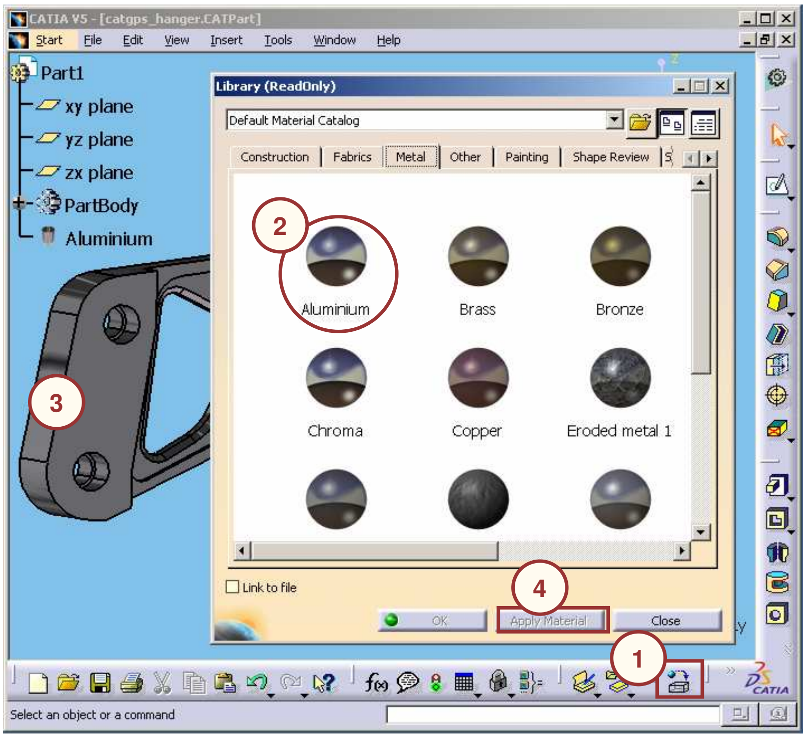
Assigning Material

1 Click on the “Apply Material” icon

2 Select a Material

3 Select the Part

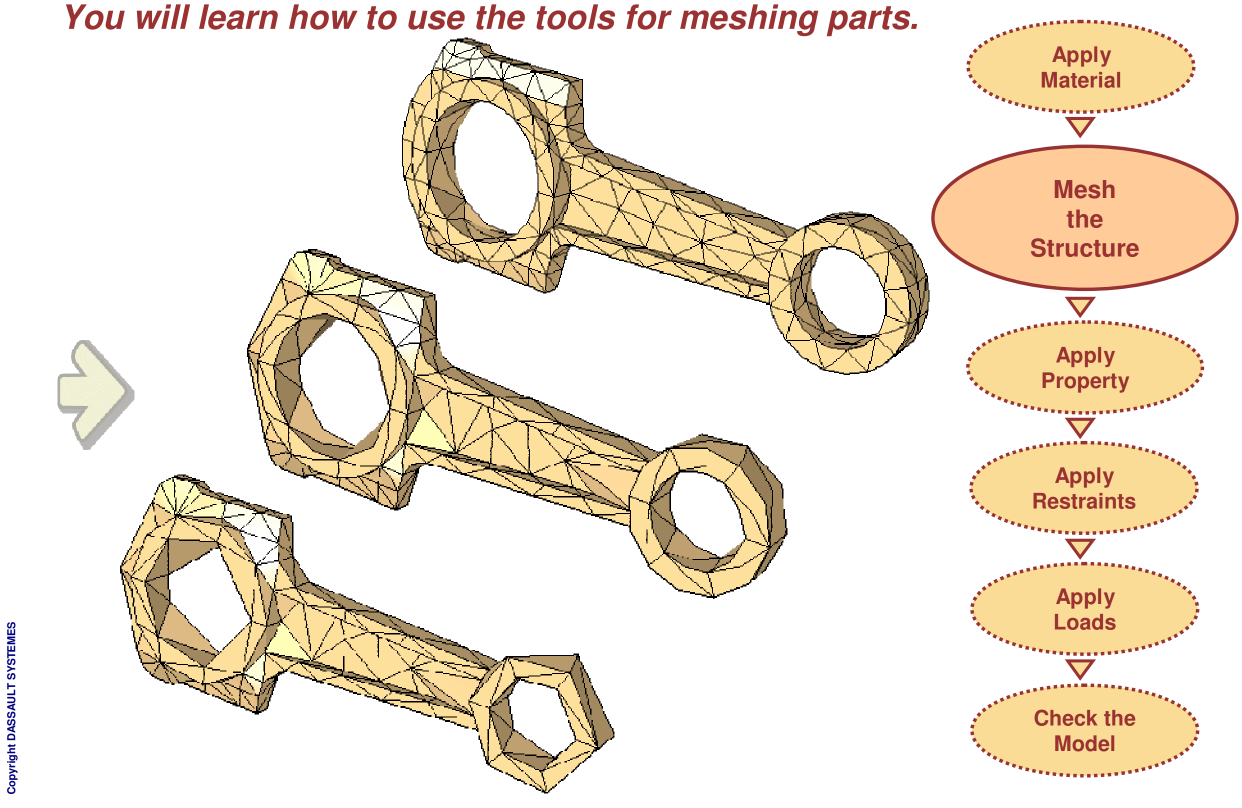
4 Click ‘Apply Material’ and then click OK to confirm the operation



Instructor Notes:

Managing Mesh Parts

You will learn how to use the tools for meshing parts.



Instructor Notes:

Meshing Part (1/2)

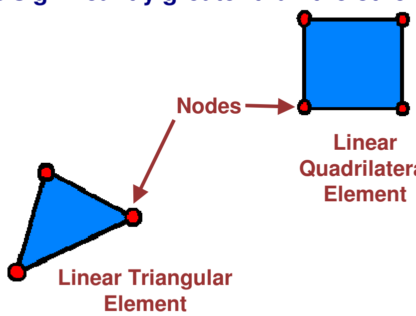
Meshing involves approximating the actual physical structure using several simple geometric shapes called Elements. These elements are interconnected to each other at points called Nodes. The mesh is a representation of the mathematical idealization of the structure.

On the basis of dimensionality, the elements can be classified as :

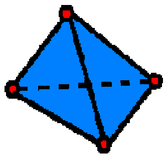
- **1D Elements: These elements must be used to represent structures where one of the dimensions is significantly greater than the other two.**

Linear Beam Element

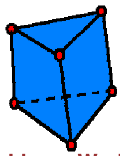
- **2D Elements : These elements must be used to represent structures where two dimensions are significantly greater than the third dimension.**



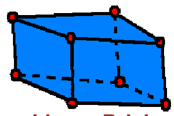
- **3D Elements : These elements must be used to represents structures where all the three dimensions are approximately of the same order of length.**



Linear Tetrahedron Element



Linear Wedge Element



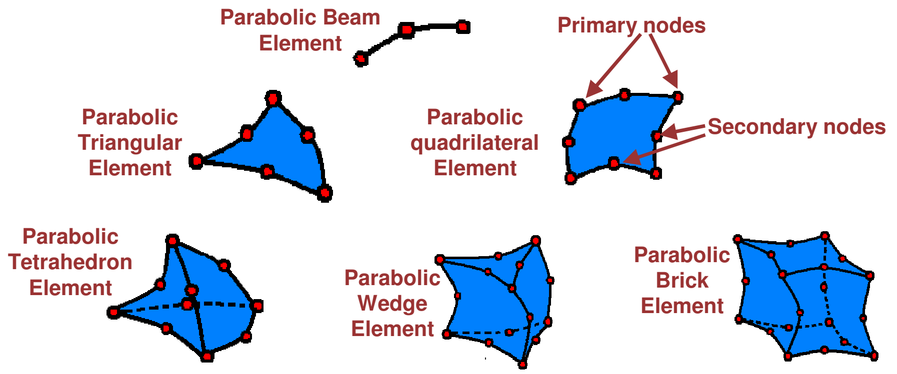
Linear Brick Element

Instructor Notes:

Meshing Part (2/2)

Within each element, displacement of nodes is determined by a polynomial equation called Displacement Equation. Elements can also be classified according to the order of the Displacement Equation as:

- **Linear Elements: Linear Elements have linear displacement interpolation between the nodes. Thus, when linear elements are subjected to loads their shapes follow linear deformation between the nodes. All elements described on the previous page are linear elements.**
- **Higher Order Elements: They use displacement interpolation such as parabolic, cubic or higher order between the nodes. Thus when parabolic elements are subjected to loads their shapes follow parabolic deformation equation. These elements have additional nodes on the edges joining the primary nodes. They are used to improve accuracy of the solution, however they increase the computation time. Higher order elements can be further classified according to dimensionality.**
- **1D Element:**
- **2D Elements:**
- **3D Elements:**

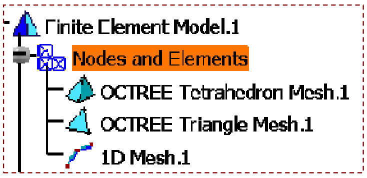


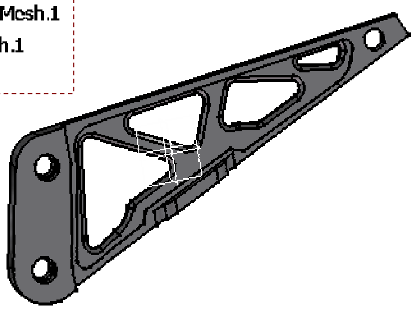
Element parameters such as element shape and displacement equation control the accuracy of the solution.

Instructor Notes:

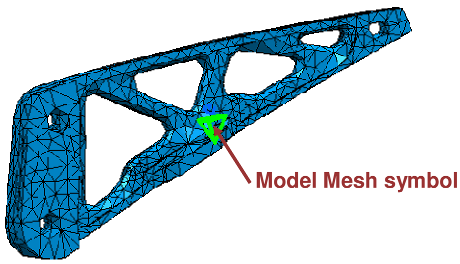
Mesh Definition (1/3)

Once the material is applied, you have to mesh the part. When you select the GPS workbench, some model meshes and model properties are automatically defined. The number of meshes and properties created will be equal to the number of bodies in the model. The meshes and properties are displayed in the Specification Tree along with their corresponding symbols.





Part with only material applied

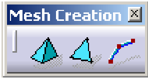


Meshed part

A double-click on the Mesh symbol will allow you to edit them for possible customization.

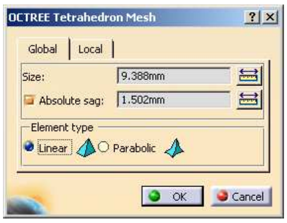
Instructor Notes:

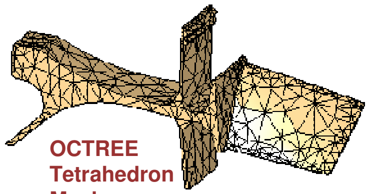
Mesh Definition (2/3)



According to the part geometry, you will define different kinds of meshes.

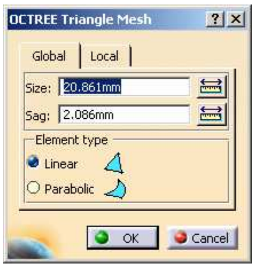
You can use OCTREE Tetrahedron mesh for a solid geometry.

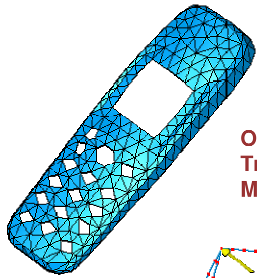




OCTREE Tetrahedron Mesh

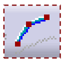
You can use OCTREE Triangle mesh for a surface geometry. You can apply a Surface mesh on a surface or on a solid if you want to take into account its shell only

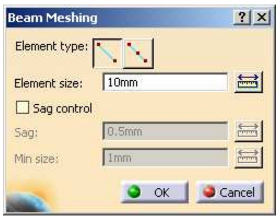




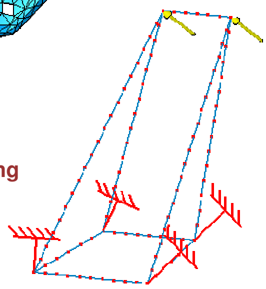
OCTREE Triangle Mesh

You can use the Beam Meshing tool for a Wireframe geometry. You can define the Element size.



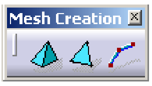


Beam Meshing



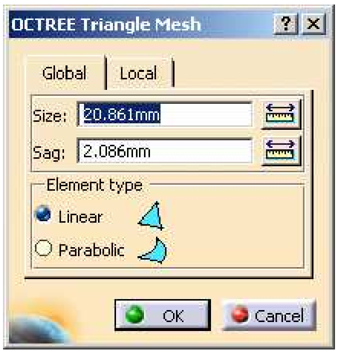
Instructor Notes:

Mesh Definition (3/3)



Parameters:

Size: The average size of each element (tetrahedron or triangle). The smaller the size, more accurate will be the analysis.



Sag: The maximum distance allowed between the mesh and the geometry. This parameter is useful when you are meshing curved shapes but not simple geometry like square, etc. The smaller is the SAG, the better is the match between the mesh and the geometry.

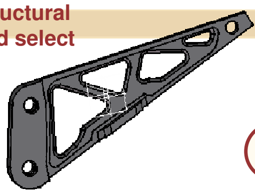
Element Type: Parabolic elements will provide you a more accurate analysis than Linear elements however computation time will be more.

Instructor Notes:

How to Mesh a Part

According to the geometry, choose the mesh type you need.

1. Switch to 'Generative Structural Analysis' workbench and select the geometry to mesh

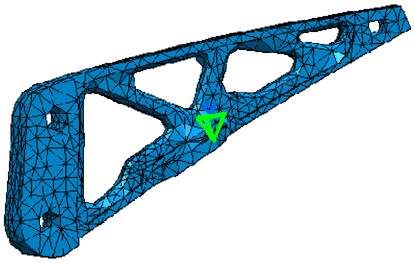


2. Choose the most appropriate mesh for your analysis.

For 3D Parts use OCTREE Tetrahedron Mesher. For 2D parts use OCTREE Triangle Mesher and for 1D use Beam Mesher

3. Specify the mesh parameters: Size, Sag, Element type

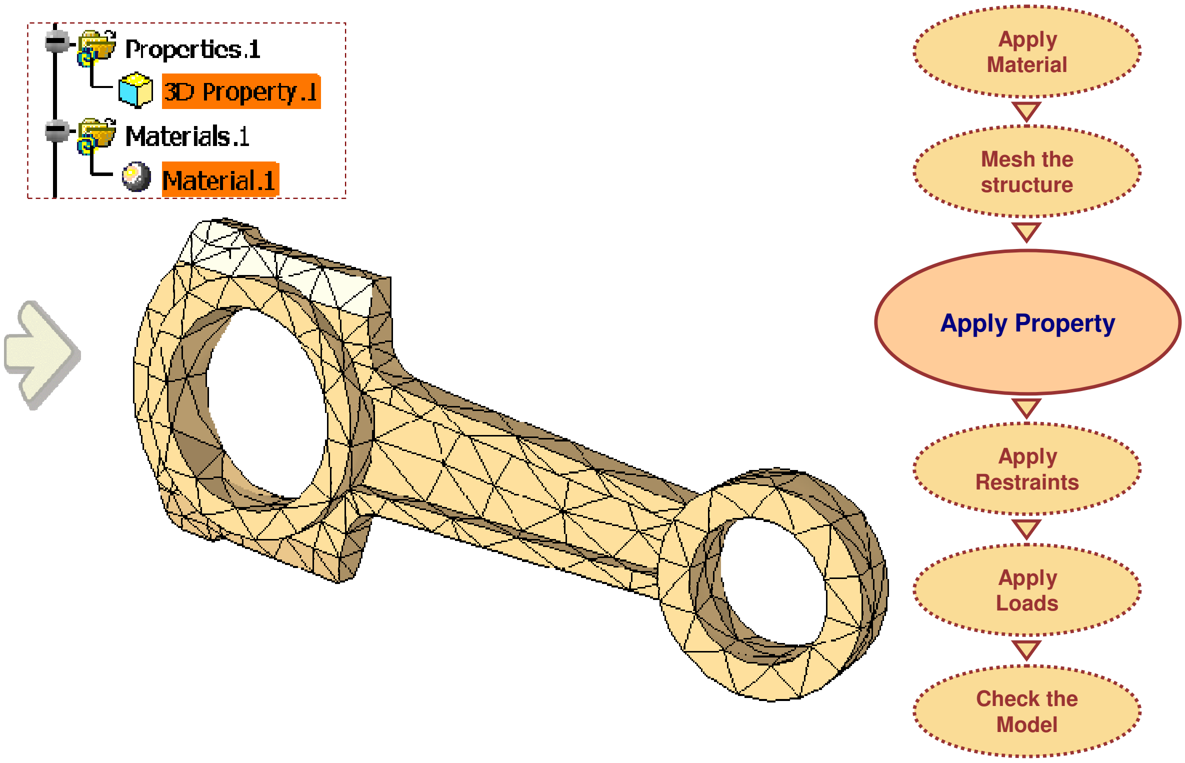
4. Click OK to confirm



Instructor Notes:

Applying Physical Property

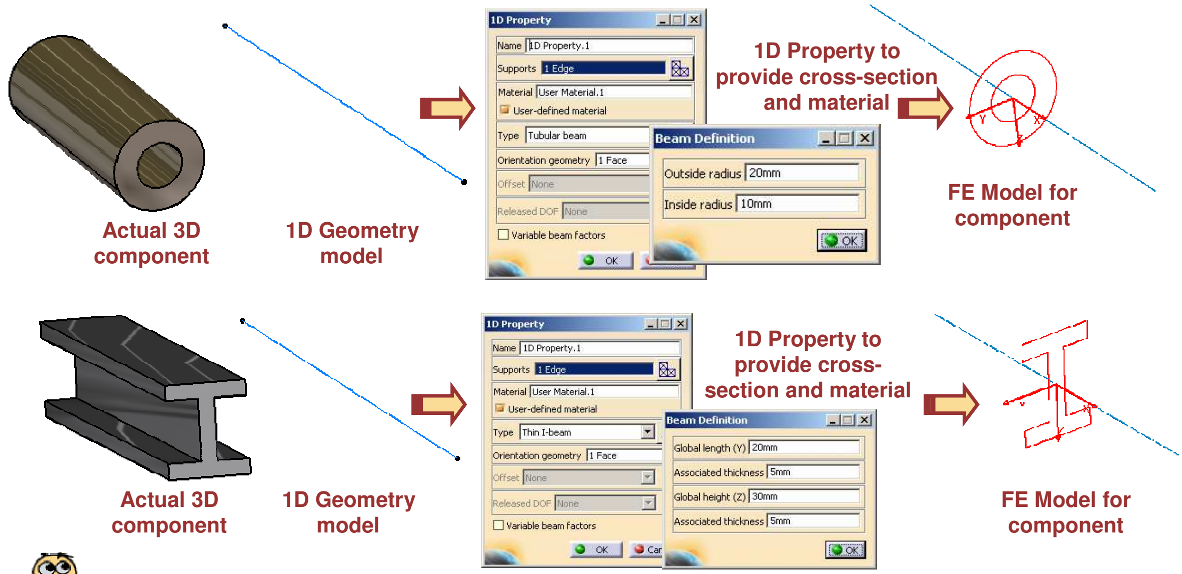
You will learn how to associate physical properties to mesh parts.



Instructor Notes:

What is Physical Property (1/2)

Physical Property associates various geometrical properties along with material to the generated mesh. It uses geometry as support. The type of physical property that needs to be attached depends on the dimensions of the geometry that is meshed.



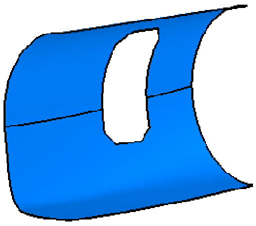
In CATIA physical property does not mean only the mechanical properties of structure. Material properties for the structure are defined separately. The physical property lets you define geometrical properties for the mesh such as different cross-section parameters, thickness, etc. and associate already defined material to the mesh.

Instructor Notes:

What is Physical Property (2/2)

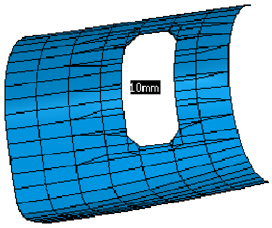
Using 2D property, you can provide thickness and material to the surface mesh parts.

Actual 3D component having thickness



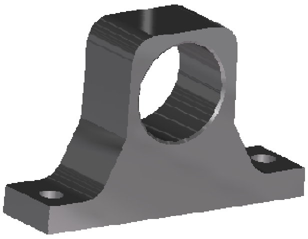
2D surface Geometry model

2D Property provides thickness and material

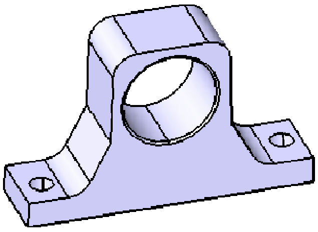


FE Model for component

3D Property assigns only material to the 3D mesh parts.

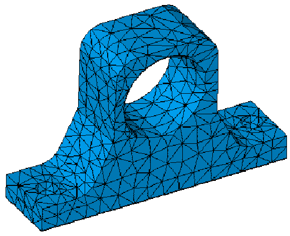


Actual 3D component



3D Geometry model

3D Property provides material



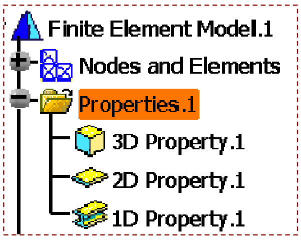
FE Model for component

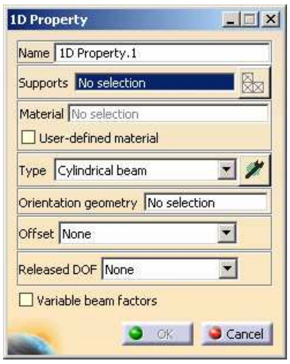
Instructor Notes:

Defining Property

For each Mesh you must specify a physical property. According to the geometry and the type of mesh you have defined, you can assign three different types of physical properties:

- **3D Property**
- **2D Property**
- **1D Property**

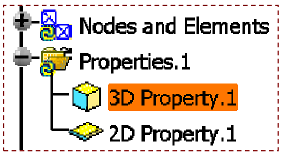




Instructor Notes:

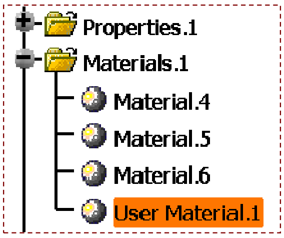
Defining 3D Property

A 3D Property is a physical property assigned to a 3D part and is associative to the geometry. When defining a tetrahedron mesh, 3D property is assigned.



Mesh Part Filter

You can select User-defined material option for material assigned using 'User Material' icon.



Instructor Notes:

Defining 2D Property

A 2D Property is a physical property assigned to a surface part. It references a material assigned to the surface Part and describes its thickness.

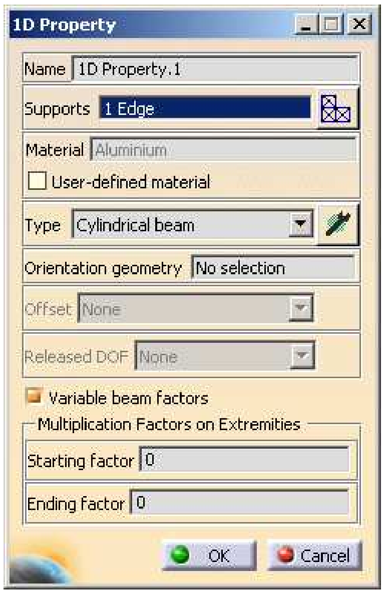
Please note that the thickness you specify should correspond to the thickness that was previously defined in Generative Shape Design workbench.

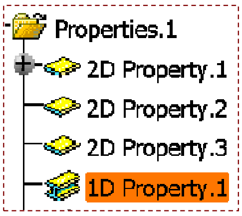
Associativity exists between the thickness of the part and the corresponding CATAnalysis 2D property in case thickness is defined to surface in GSD workbench using Tools>Thin Parts Attribute

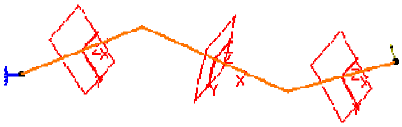
Instructor Notes:

Defining 1D Property

A 1D property is a physical property assigned to a section of a part. Before assigning the property ensure that a material was applied to the geometry and a 1D mesh was assigned to the beam.





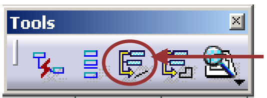


Type option allows you to select cross-section areas for the beam.

Variable beam factors checkbox allows you to compute a linear approximation of variable cross-section beams.

Multiplication Factors on Extremities allows you to specify scaling factor on each side of the section. Then, the beam will be modeled as a sequence of constant section beams with linearly decreasing dimensions.

To apply 1D properties and beam mesh on geometry included in a sketch, you need to declare the geometry as output geometry using Tools > Output feature in sketcher



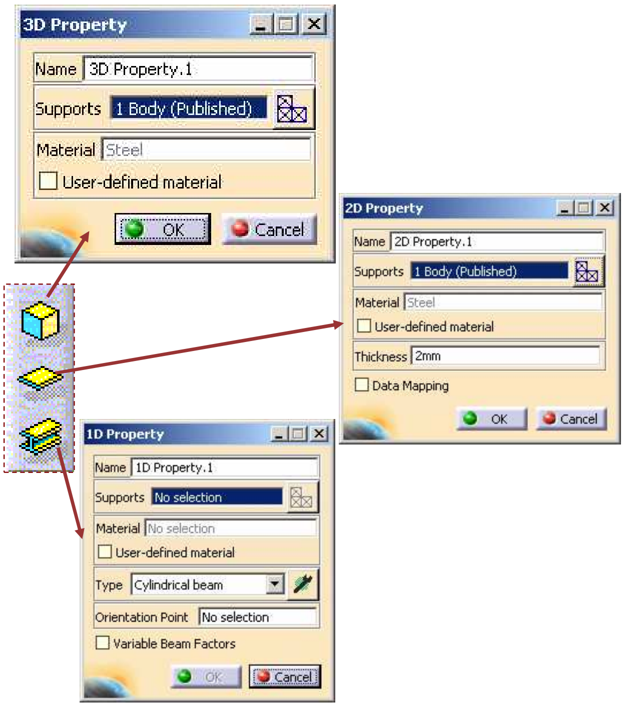
Output feature icon

Instructor Notes:

How to Define a Part Property

Once you have meshed a Part, you have to define its property. According to the mesh part, you will apply either a Solid, Shell or a Beam property.

1. **Click on the appropriate property type icon.**
2. **Select the part geometry entity (Part Body or Open Body) in the specification tree.**
3. **Specify the additional information if required.**
3. **Click OK to confirm the property definition.**

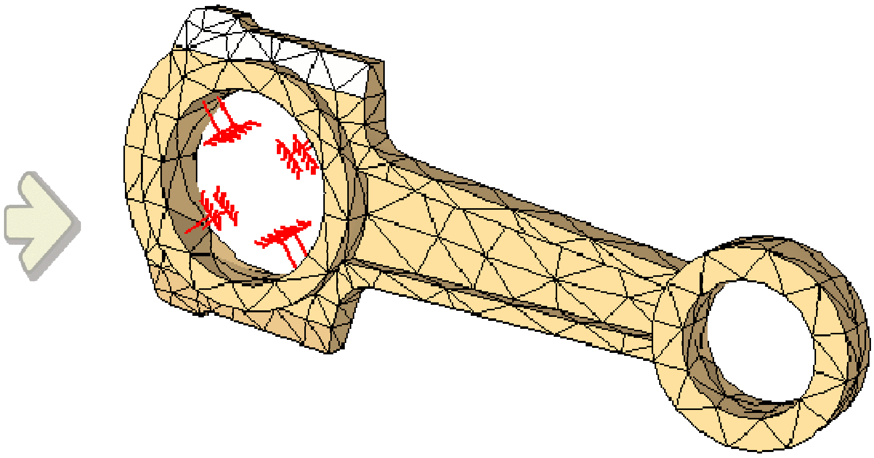


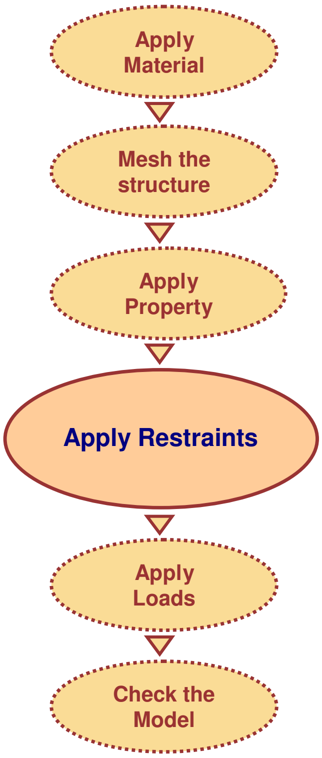
The mesh part must have one single property. As for mesh parts, you can also delete properties.

Instructor Notes:

Defining Restraints

You will learn how to apply restraints to a part.

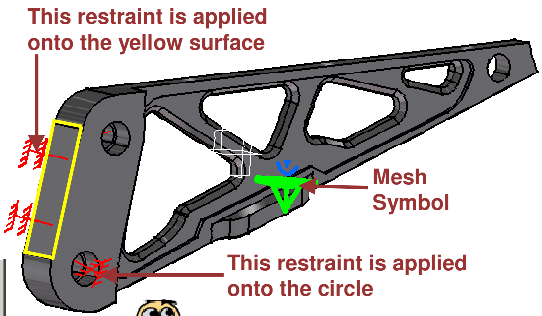




Instructor Notes:

What are Restraints

Restraints are used to specify the support or boundary conditions for the FE model. Restraints restrict the displacement of supports of a structure in the desired direction. This is done by providing zero displacement values for specific Degrees Of Freedom (DOFs) of the nodes in FE model.
The restraints are directly applied onto the geometry (groups, surfaces, lines, points) as shown on the example:



CATIA provides following types of restraints :

- **Clamp**

- **Mechanical Restraints**

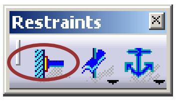
- **Advanced Restraints**

While applying restraints, it is necessary to mesh the part, even if you are working with the geometry. As restraints are finally transferred to mesh through geometry.

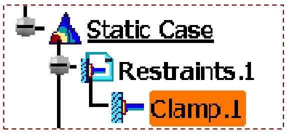
It is necessary to apply a restraint in order to get a unique solution to the FE model computation, otherwise a singularity error will be detected during the computation. In case you get a singularity error, you can generate a displacement image to visualize the singularity displacement. This helps you to correct the FE model.

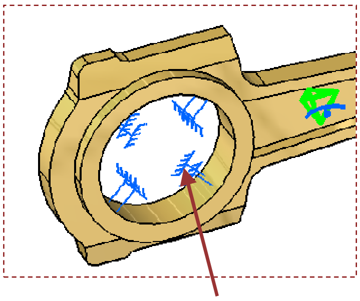
Instructor Notes:

Clamps



Clamps are restraints applied to a surface or curve geometry, for which all DOFs are blocked in the subsequent analysis. Consequently, clamps have zero DOF, which means that no translational or rotational movement is allowed.





Clamp Restraint Symbol

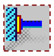
A clamp restraint fixes all DOFs for selected support.

Supports: Clamps can be applied to Points or Vertices, Curves or Edges, Faces or Surfaces, Virtual Parts, Geometrical Groups, Group by Neighborhood and Group by Boundary.

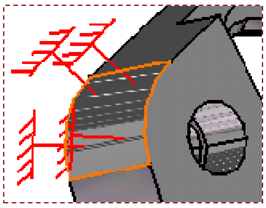
Instructor Notes:

How to Apply a Clamp Restraint

Clamp is used to restraint all DOFs of the mesh nodes corresponding to a selected support.

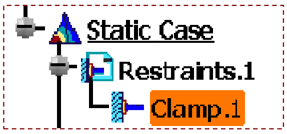
1 Switch to the 'Generative Structural Analysis' workbench and click on the 'Clamp' icon in the 'Restraints' Toolbar. 

2 Select the geometry support. You can select multiple supports.



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

3 Click OK to confirm.



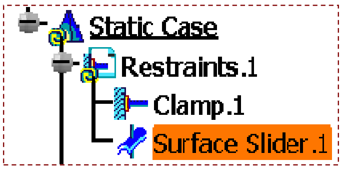
A Clamp object appears in the Specification Tree under the active Restraints set.

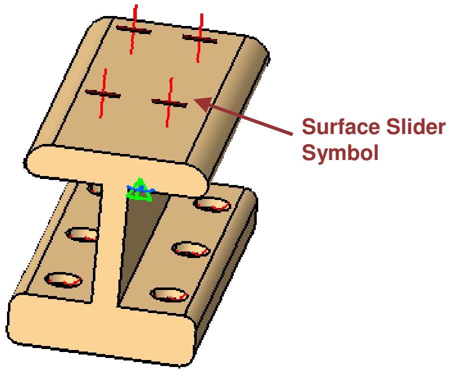
Instructor Notes:

Surface Slider

A Surface Slider is a virtual rigid surface along which the points of a deformable surface can slide. It can be applied to surface geometries.

At each point of the deformable surface, the system automatically generates a constraint which fixes the translational degree of freedom in the direction normal to the surface at that point.





Type of Supports: Surface Slider can be applied to Faces or Surfaces.

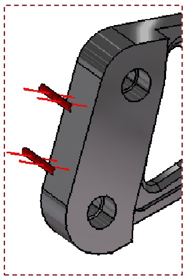
Instructor Notes:

How to Apply a Surface Slider

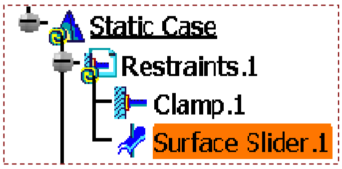
A Surface Slider fixes translational DOF normal to the surface of the mesh nodes corresponding to the selected support.

1 **Switch to the ‘Generative Structural Analysis’ workbench and click on the ‘Surface Slider’ icon in the ‘Restraints’ Toolbar.**

2 **Select the required support.**



3 **Click OK to confirm.**

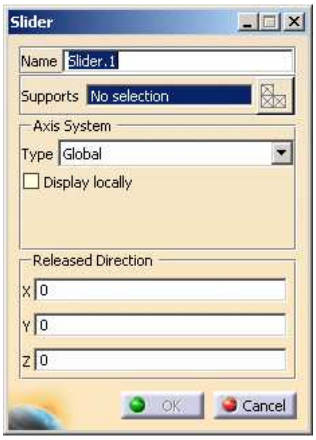


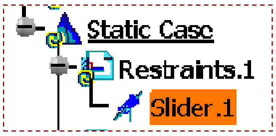
A Surface Slider object appears in the Specification Tree under the active Restraints set.

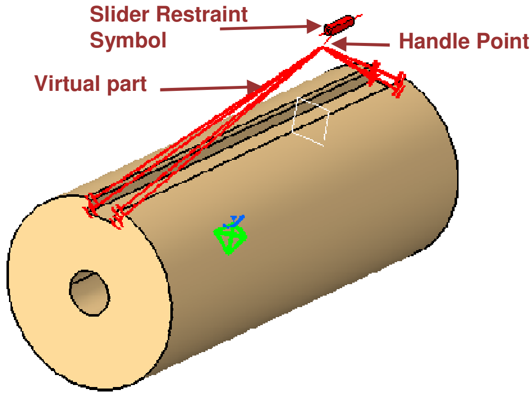
Instructor Notes:

Slider

Sliders are prismatic join restraints applied to handle points of virtual parts, which result in constraining the point to slide along a given axis. For the fixed point, handle of the virtual part is selected as support.
You will define the sliding direction, and as a result the virtual part as a whole is allowed to slide along an axis parallel to the sliding direction and passing through the fixed point.







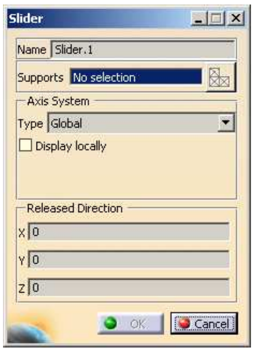
A Slider has 1 Translational DOF.

Supports: It needs a 'Virtual Part' as support.

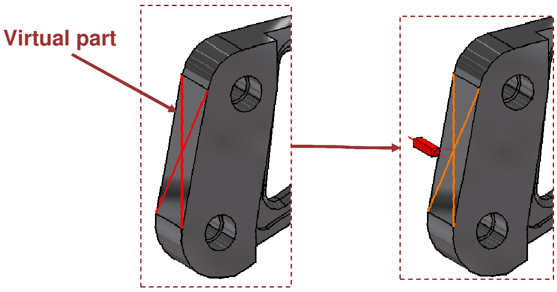
Instructor Notes:

How to Apply a Slider Restraint

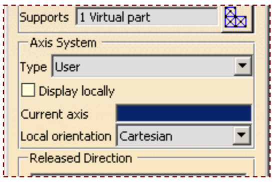
1 Click on the ‘Slider’ icon in the ‘Restrain’ Toolbar.



2 Click the pre-defined ‘virtual part’ as support.



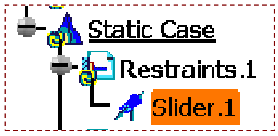
3 Define the axis-system:



Global: if you want the components of the sliding direction to be interpreted relative to the Global Coordinate System.

User: if you want the components of the sliding direction to be interpreted relative to the specified User Coordinate System.

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

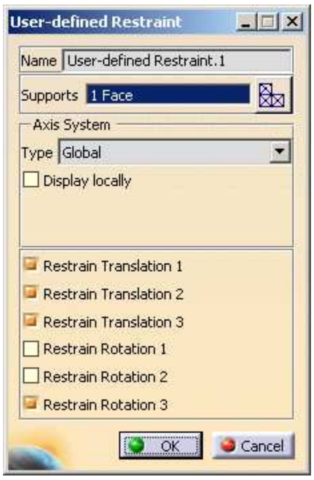


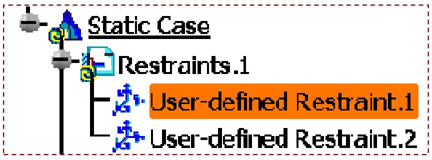
To select a user-defined axis-system, you must activate an existing axis by clicking on it in the feature tree. Its name will then be automatically displayed in the ‘Current axis’ field.

Instructor Notes:

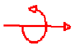
User-defined Restraints

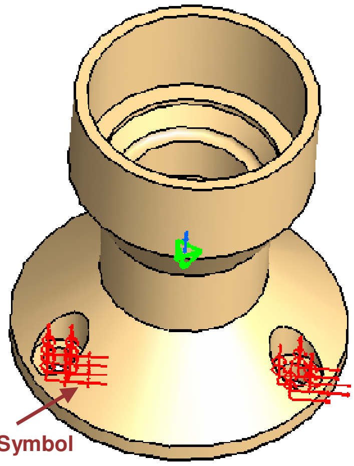
User-defined Restraints are generic restraints allowing you to fix any combination of available nodal DOFs on arbitrary geometries. Three Translational DOFs per node for continuum element meshes, and three Translational and three Rotational DOFs per node for structural element meshes.





means translation DOF restrained in that direction


means Rotation DOF restrained in that direction



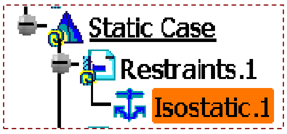
Advanced Restraint Symbol

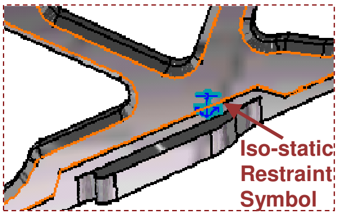
Types of Supports: Points or Vertex, Curves or Edges, Faces or Surfaces, Virtual Parts, Groups, Groups by Neighborhood, Groups by Boundary.

Instructor Notes:

Iso-static Restraint

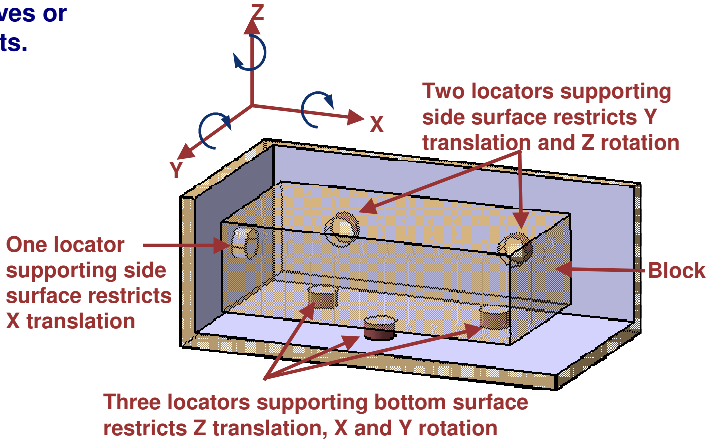
Iso-static Restraints define boundary conditions which represent a simply supported body. The resulting boundary condition prevents the body from rigid-body translations and rotations, without over-constraining it.
It chooses three points and restrains some of their DOFs according to the 3-2-1 principle.





Types of Supports: Points or Vertex, Curves or Edges, Faces or Surfaces and Virtual Parts.

3-2-1 principle is a commonly used principle to locate work piece for restricting its rigid DOF. Consider the block in space having six rigid DOFs, three translational and three rotational. Then, three locators at bottom of block restricts translation in Z-direction and rotation about X and Y axis. Next two locators restrict translation about Y axis and rotation about Z axis. The final locator restricts translation about X axis.



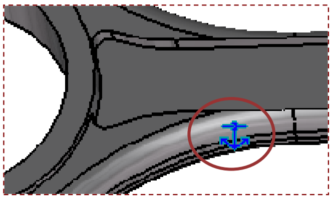
Instructor Notes:

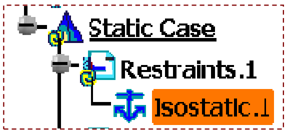
How to Apply an Iso-static Restraint

An Iso-static Restraint simply supports the body by choosing three points and restraining some of their DOFs according to the 3-2-1 rule.

1 Click on the ‘Iso-static Restraints’ icon in the ‘Restraint’ Toolbar.

2 Click OK to confirm.



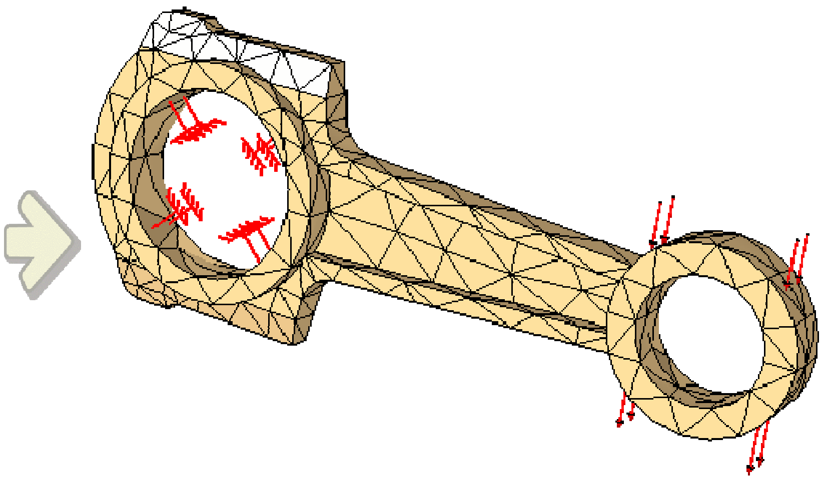


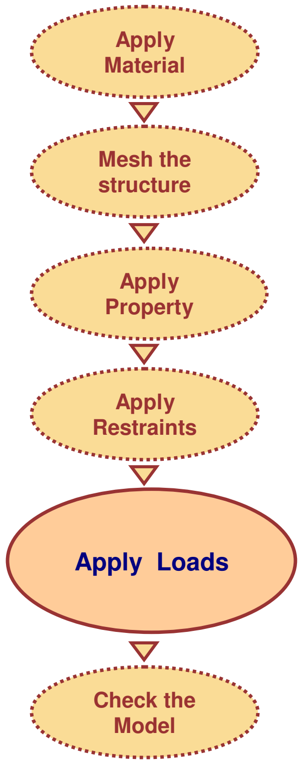
A Isostatic restraint appears in the Specification Tree under the active Restraints set.

Instructor Notes:

Defining Loads

You will learn how to apply loads to a part.





Instructor Notes:

Introduction

Loads are inputs to the FE model. The purpose of FEA is to study the behavior of the structure when a load is applied to it. The loads can be in the forms of forces, moments, pressures, temperature, or accelerations.

You can apply the restraints and loads directly on the geometry as support (surfaces, lines, points) as shown in the example below:

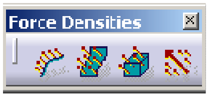
CATIA provides following types of loads which can be applied on structures.

- Pressure

- Body Motion

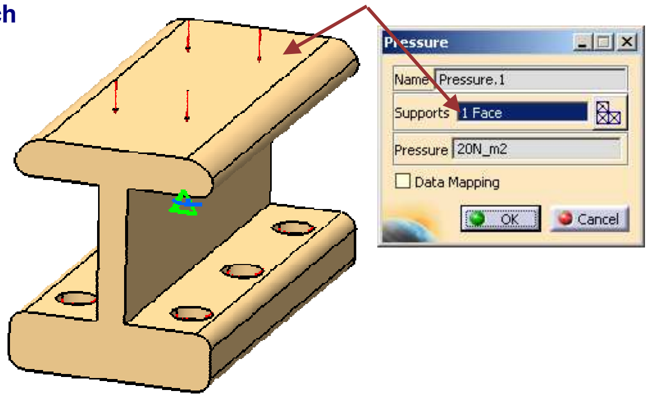
- Forces

- Force Densities



- Enforced Displacements

Face is selected as support for Pressure load shown with down arrows.

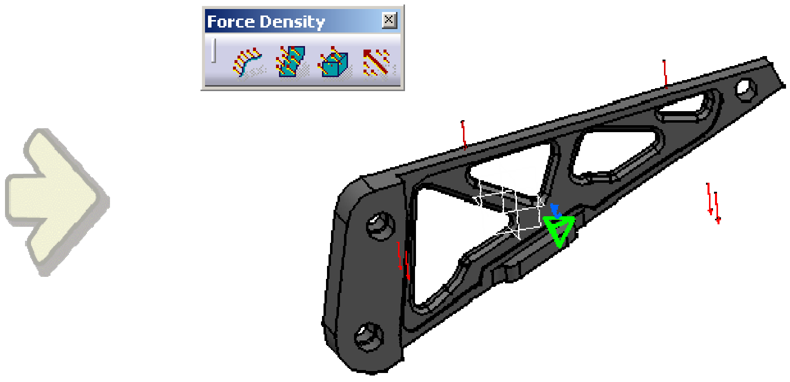


Computation process will transfer the Restraints and Loads applied on Geometry onto corresponding mesh parts. Therefore it is necessary to mesh the parts before applying Restraints and Loads.

Instructor Notes:

Force Density

You will learn how to apply force density to a part .

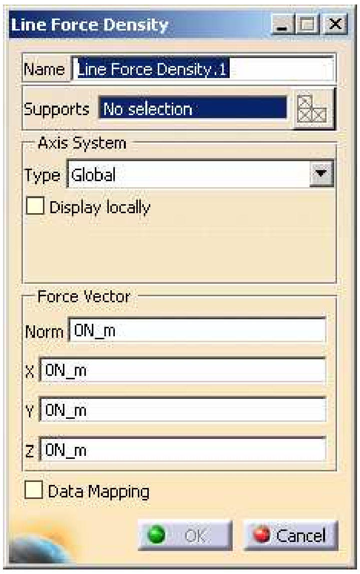


Instructor Notes:

About Line Force Density

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

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



Line Force density: Units are line traction units (typically N/m in SI)

Supports: Line Force density can be applied on Curves and Edges

Axis System:

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

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

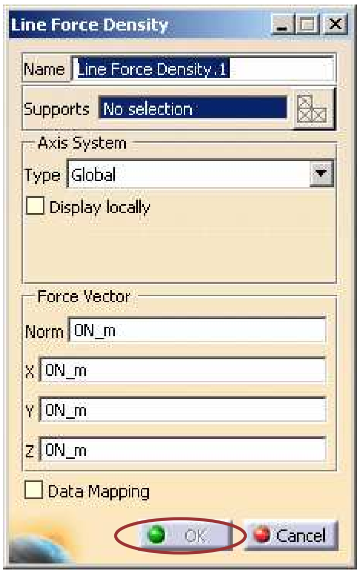
To select a User axis-system, you must activate an existing axis by clicking it in the feature tree. Its name will then be automatically displayed in the Current Axis field.

Instructor Notes:

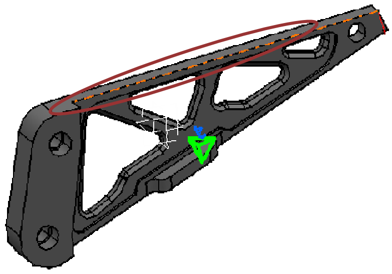
Applying a Line Force Density

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 'Line Force Density' Icon



2. Select the geometry support(s): Curves or Edges



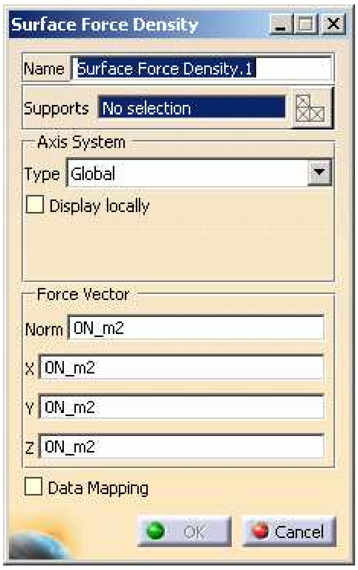
3. Choose the axis-system
4. Enter a Force Vector (X,Y,Z)
5. Click on OK

Instructor Notes:

About Surface Force Density

Surface Force Densities are intensive loads representing surface traction fields of uniform magnitude applied to surface geometries.

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



Surface Force density: Units are surface traction units (typically N/m2 in SI)

Supports: Surface Force Density can be applied on Surfaces and Faces

Axis System: The Axis System Type combo box allows you to choose between Global and User axis-systems, for entering components of the resultant force vector.

Global: The components of the force vector (in a file) will be interpreted as relative to the fixed global rectangular coordinate system.

User: The components of the force vector (in a file) will be interpreted as relative to the specified rectangular coordinate system.

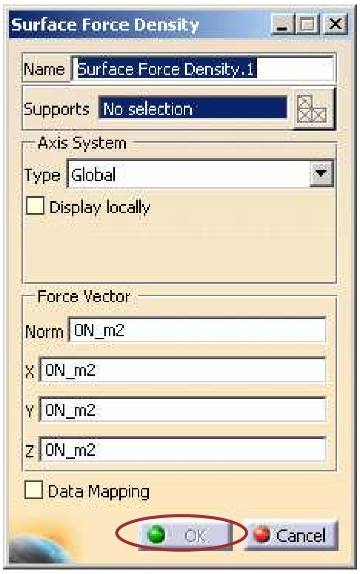
To select a User axis-system, you must activate an existing axis by clicking it in the specification tree. Its name will then be automatically displayed in the Current Axis field.

Instructor Notes:

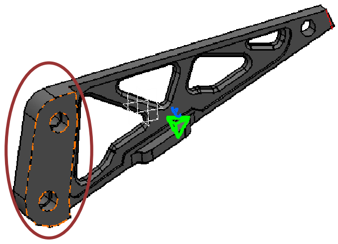
Applying a Surface Force Density

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 'Surface Force Density' Icon



2. Select the geometry support(s): Surfaces or Faces

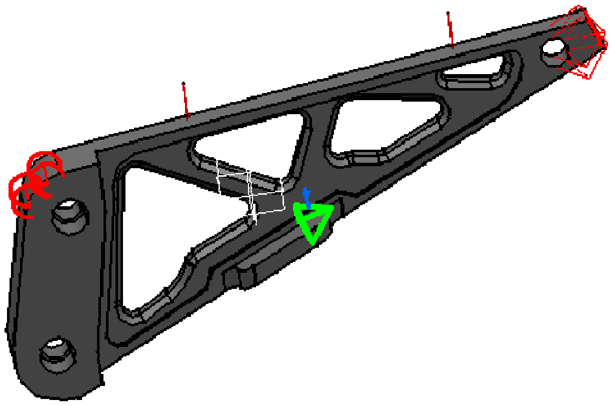


3. Choose the axis-system
4. Enter a Force Vector (X,Y,Z)
5. Click on OK

Instructor Notes:

Forces

You will learn how to apply forces to a part.

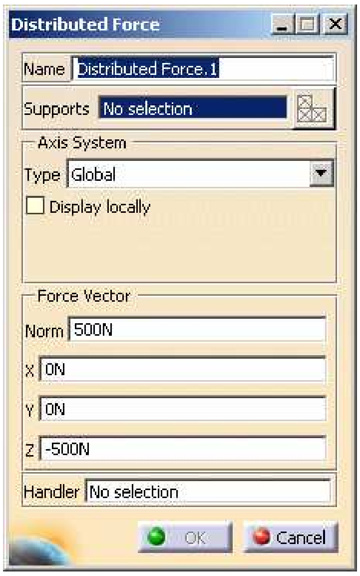


Instructor Notes:

Distributed Force

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

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



Distributed Forces : Units are force units (typically N in SI).

Supports: Distributed Forces can be applied on Points or Vertex, Surfaces or Faces, virtual parts.

Axis System: The Axis System Type combo box allows you to choose between Global and User axis-systems, for entering components of the resultant force vector.

Global: The components of the force vector (in a file) will be interpreted as relative to the fixed global rectangular coordinate system.

User: The components of the force vector (in a file) 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 specification tree. Its name will then be automatically displayed in the Current Axis field.

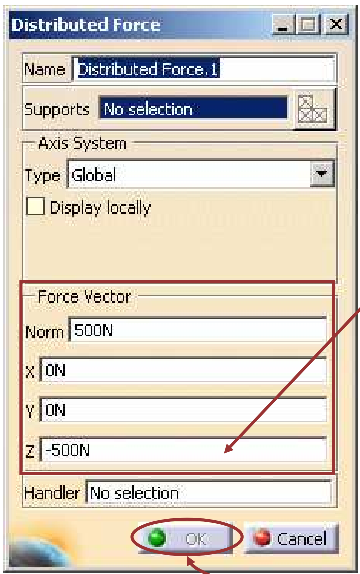
Handler: Point of application of the force resultant for virtual parts, this point is the handler of the virtual part. For extended geometries, this point is the centroid of the geometry.

Instructor Notes:

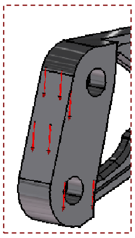
Applying a Distributed Force

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

1 Click on the "Distributed Force" Icon

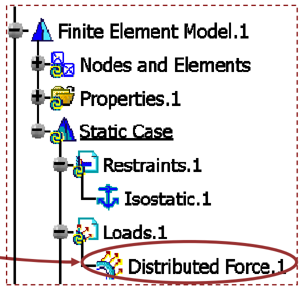


2 Select support(s) :
Surfaces : Forces are extrapolated on the nodes of the closest element.
Points : Forces are directly applied to the associated node



3 Specify force

A Distributed Force feature appears in the features tree under the active Loads objects set.



4 Click on OK

Instructor Notes:

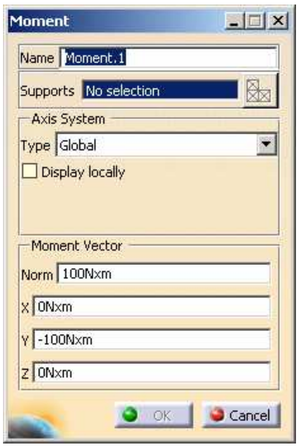
Moment

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

The resultant moment vector remains constant independently of the geometry selection. The point of application of the couple is arbitrary.

The given pure couple system is processed by the program as follows:

- **In the case of extended geometries, it is transformed into an equivalent force system distributed over the selected support**
- **In the case of virtual parts connected to deformable bodies, it is transmitted as a force system collectively to the entire connected geometry**



Moment : Units are moment units (typically Nm in SI).

Supports: Moments can be applied to Points or Vertex, Surfaces or Faces, virtual parts.

Axis System: The Axis System Type combo box allows you to choose between Global and User axis-systems, for entering components of the resultant force vector.

Global: The components of the force vector (in a file) will be interpreted as relative to the fixed global rectangular coordinate system.

User: The components of the force vector (in a file) 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 specification tree. Its name will then be automatically displayed in the Current Axis field

Instructor Notes:

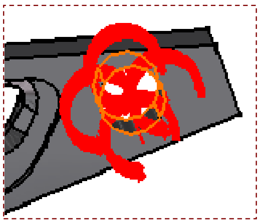
Defining a Moment

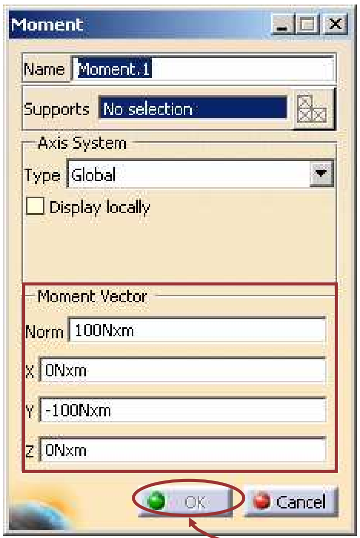
Moments are force systems statically equivalent to a given pure couple (single moment resultant), distributed on a virtual part or on a geometric selection

1 Click on the 'Moment' Icon

2 Select support(s) :
Surfaces : Forces are extrapolated on the nodes of the closest element.

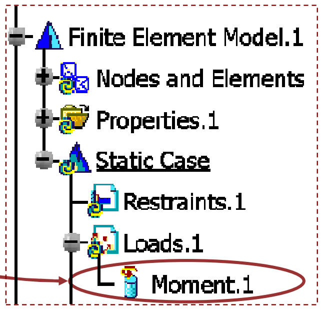
Points : Forces are directly applied to the associated node





3 Specify the moment

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

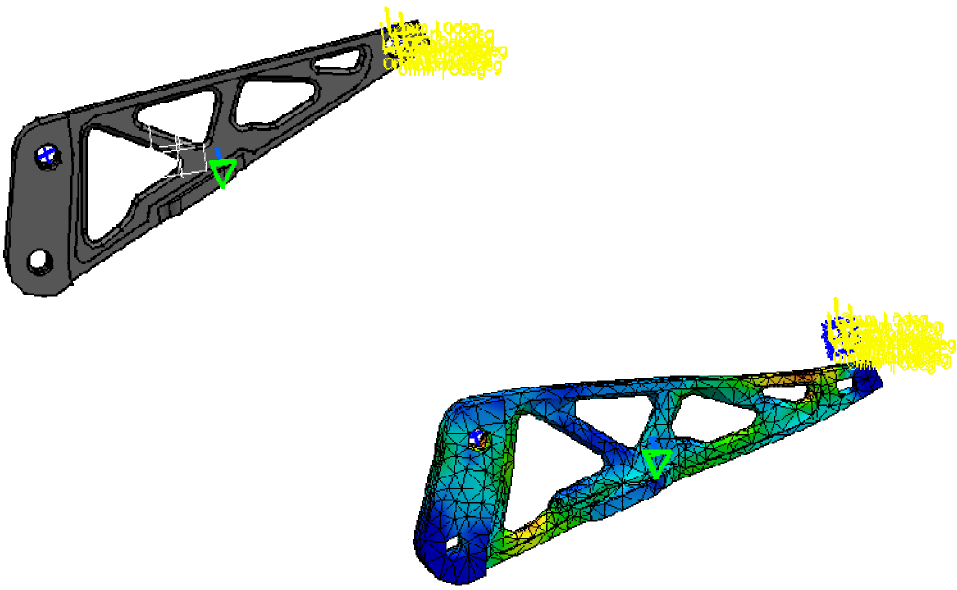


4 Click on OK

Instructor Notes:

Enforced Displacements

You will learn how to apply an Enforced Displacement.

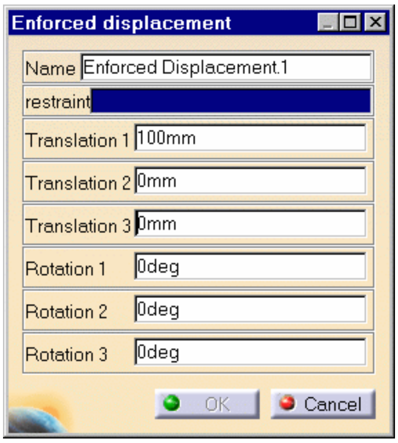


Instructor Notes:

About Enforced Displacements

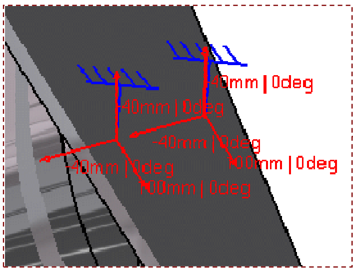
Enforced Displacements are loads applied to support geometries, resulting for the subsequent analysis in assigning non-zero values to displacements in previously restrained directions.

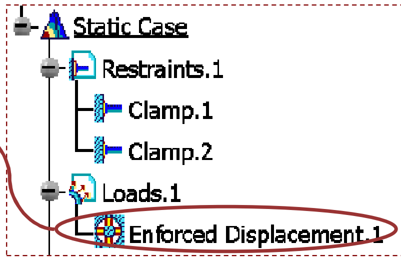
An Enforced Displacement object is by definition associated with a Restraint object. Make sure you entered non-zero values only for those degrees of freedom which have been fixed by the associated Restraint object. Non-zero values for any other degree of freedom will be ignored by the program.



Supports: Enforced Displacements can be applied on restraints (I.e a clamp)

You can enforce translations and rotations



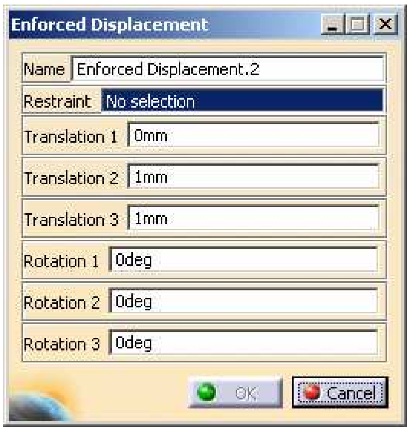


Instructor Notes:

Applying Enforced Displacements

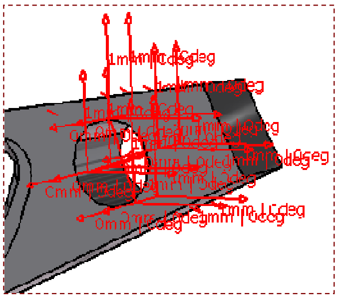
Before You Begin:
Go to View -> Render Style -> Customize View and make sure the Shading, Outlines and Materials options are active in the Custom View Modes dialog box

1. Click on the 'Enforced Displacement' Icon



2. Select the support(s): Restraints

Here we selected a Clamp restraint:



3. Enter your translation and rotation values

4. Click on OK

Instructor Notes:

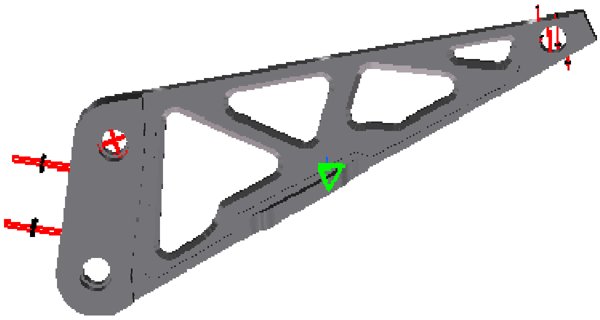
Exercise

‘Pre-Processing’ Recap Exercise

10 min

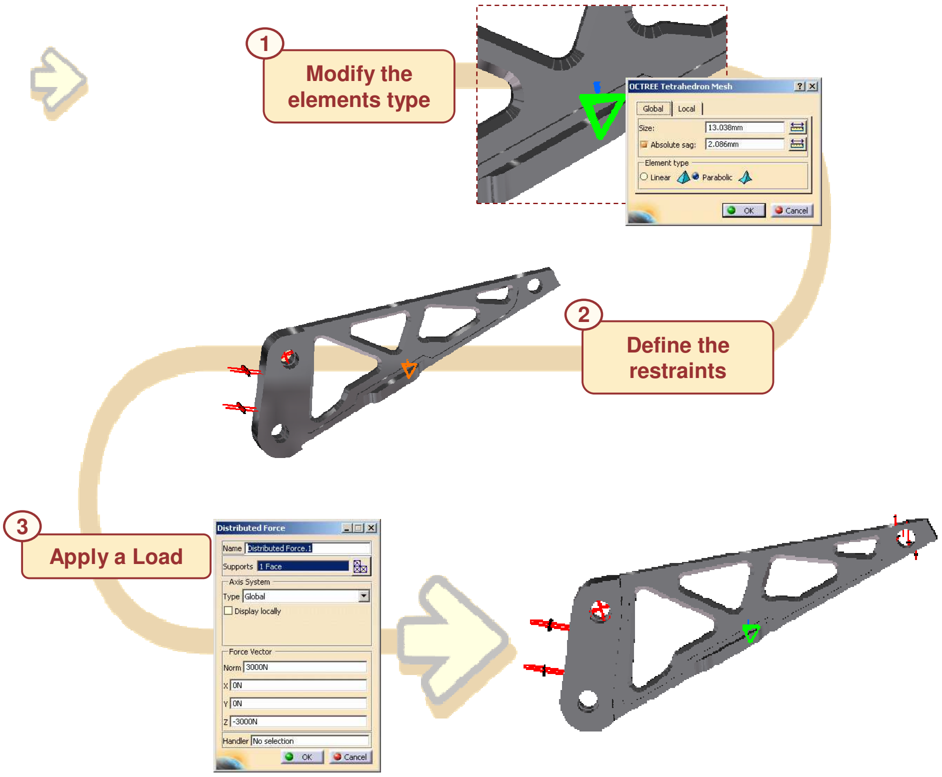
In this exercise you will practice different tools you have seen in the Pre-Processing lesson.
It includes following steps

- **Modify the element characteristics**
- **Define two types of restraints**
- **Apply a load**



Instructor Notes:

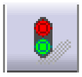
Design Process: Hanger Pre-Processing

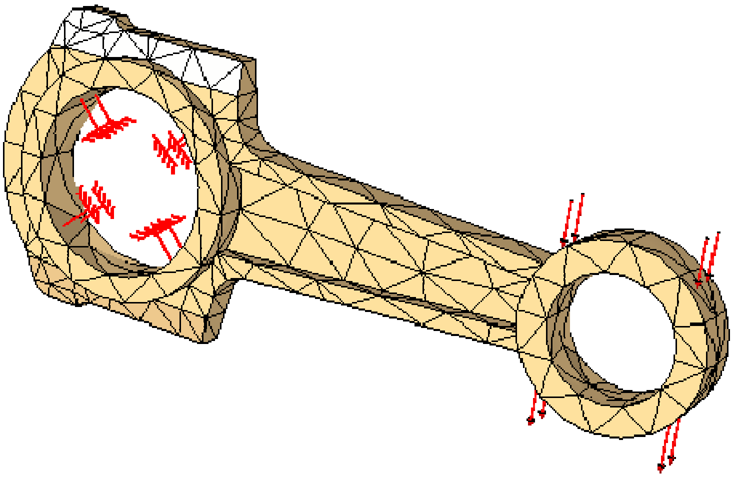


Instructor Notes:

Model Checker

You will learn how to validate the FE model using the Model Checker tool.





Apply Material
Mesh the structure
Apply Property
Apply Restraints
Apply Loads
Check Model

Instructor Notes:

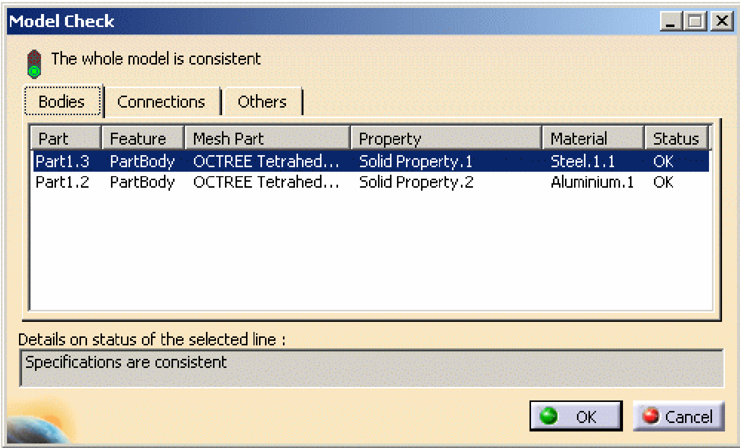
What is Model Checker (1/2)

Model Checker allows you to verify whether all the pre-processing steps are done and if the model is ready for computation. It provides a common platform where you can check all the pre-processed data.
If any information is missing, it shows the status as 'KO' against that row of information and provides a detailed error message at the bottom of the window. Different tabs of the Model Checker provide the following information:

Bodies Tab

- **Missing Mesh**
- **Missing Properties**
- **Missing Material**
- **Missing Support**
- **Diagnosis Problems**

Status :
OK
KO



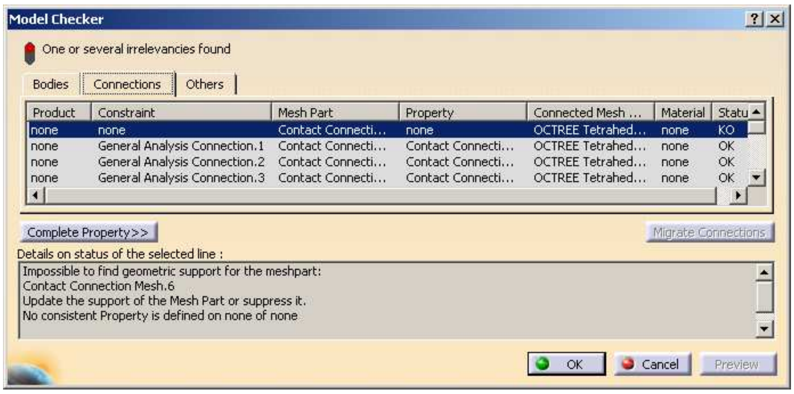
It is recommended to use the Model Checker before computation to make sure that the pre-processing steps are OK.

Instructor Notes:

What is Model Checker (2/2)

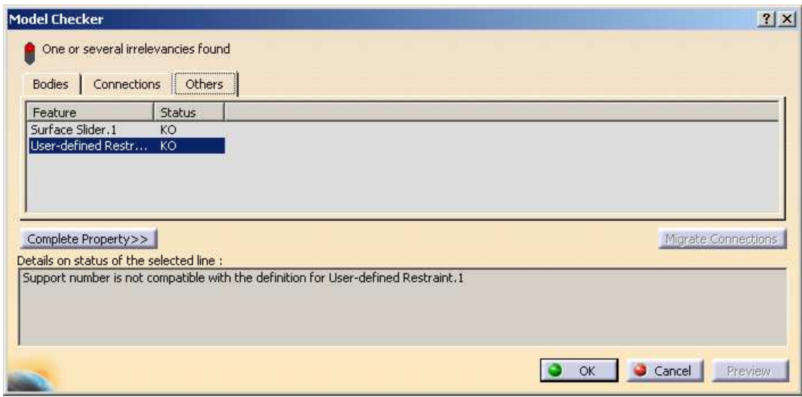
Connections Tab

- Kind of Constraints
- Part involved
- Connection Properties
- Connection Status



Others Tab

- Loads
- Restraints
- Virtual Parts



Instructor Notes:

GPS Computation

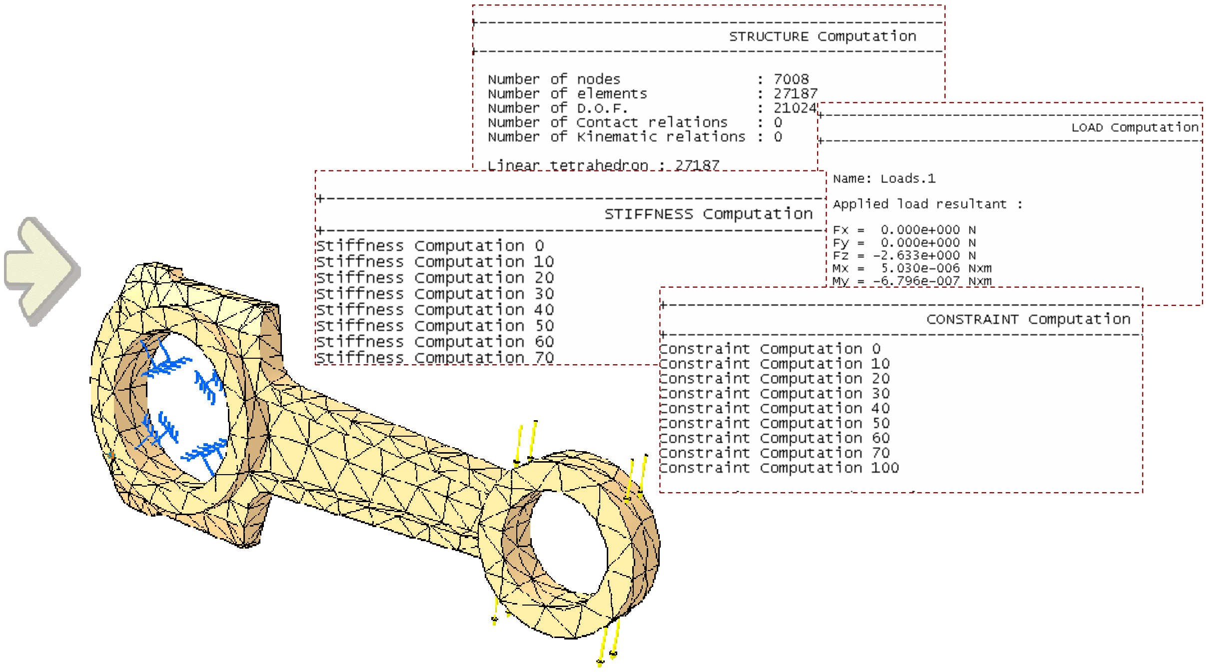
Once you have meshed the part, and applied restraints and loads, you can compute the analysis.

- What is Computation
- Specifying the External Storage
- Computing a Static Case
- Computation Recap Exercise

Instructor Notes:

GPS Computation

In this lesson, you will learn how to use the computation tools for static analysis.

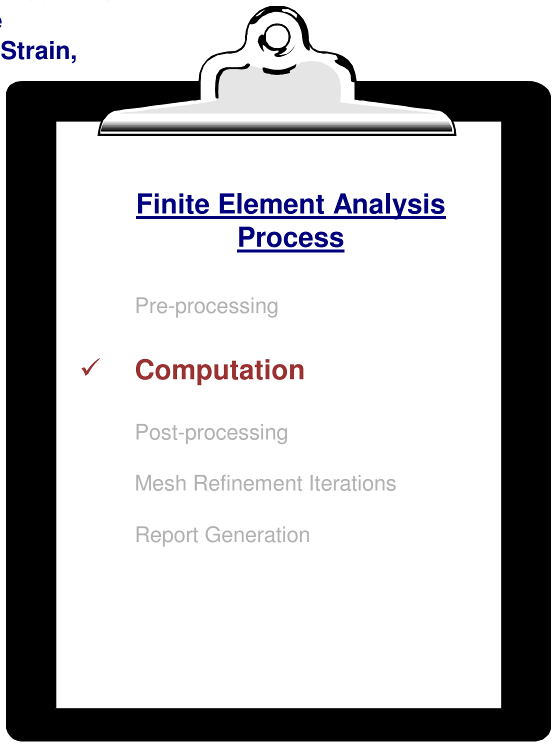


Instructor Notes:

What is Computation (1/2)

Computation is required to calculate the unknown Displacement values at the Nodal Points of the FE model. From these Displacement values other solution quantities such as Strain, Principal Stresses, Von-Mises Stresses are derived.

- **The geometry model is discretized into elements**
- **All properties and applied forces are idealized at the element and nodes level**
- **For each element, nodal forces, stiffness matrices and unknown displacement vectors are computed**



Instructor Notes:

What is Computation (2/2)

- The element connectivity is used to assemble the global stiffness, nodal forces and displacements matrices

- The minimization of the potential energy used to solve global equation using the boundary conditions to suppress the stiffness matrix singularity

$$\{F\} = [K]\{U\}$$

{F} : Nodal Force Matrix

[K] : Global Stiffness Matrix

{U} : Nodal Displacement Matrix

- Nodal displacements are computed

- Strains and stresses are calculated using the Strain-Displacement and Stress-Strain relations

$$\{\varepsilon\} = [B]\{U\}$$

$$\{\sigma\} = [C]\{\varepsilon\}$$

{ε} : Strain Matrix

[B] : Strain-Displacement Matrix

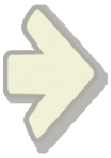
{σ} : Stress Matrix

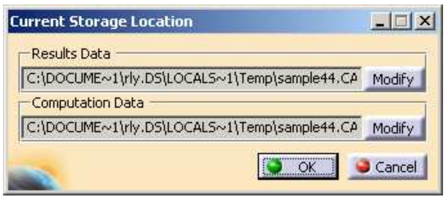
[C] : Stress-Strain Matrix

Instructor Notes:

Specifying the External Storage

You will learn how to use the storage tools.



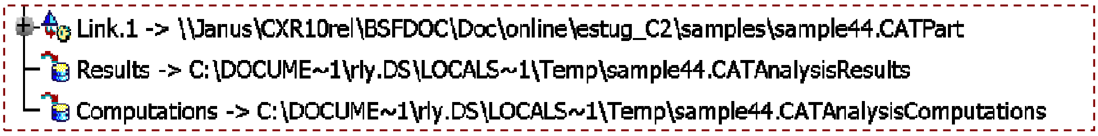


Instructor Notes:

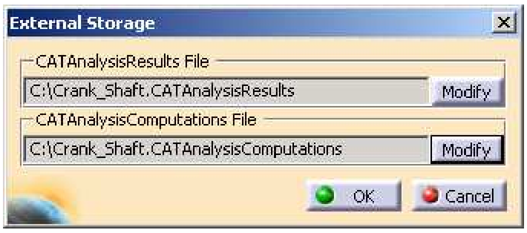
Introduction

All ELFINI Solver computations are systematically stored in a structured way, out of core memory on an external file called External Storage.

- **Stored data resulting from analysis are stored in two files, one for results(CATAnalysisResults) and one for computations (CATAnalysisComputations). After the opening of a new analysis document, the default directories in which they are stored are the last directories chosen by the user.**
- **Result data is the data necessary for generating images: displacements, loads, restraints, singularities, strain energy and so forth. This data is self sufficient if only results are to be saved and no more computations should be performed from them. Result data takes small disk space, any newly performed computation starts from the beginning and therefore may take time.**
- **Computation data corresponds to matrices such as stiffness. This data is needed to perform new computations from the loaded data which will benefit from it. It is time saving, but as a counterpart needs much disk space storage.**



It is recommended that you locate your external storage where there is enough storage space. Analysis files are not automatically saved.



Instructor Notes:

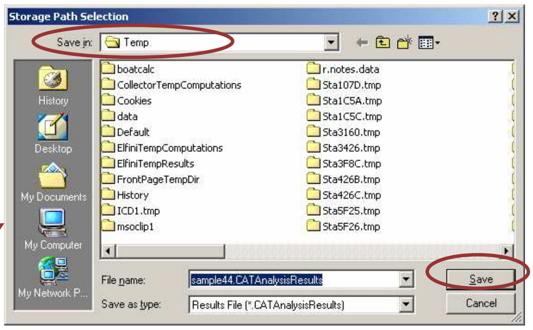
Using the ‘External Storage’ Icon

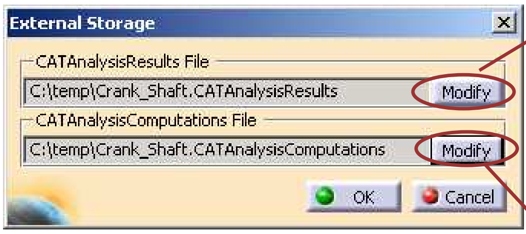
You can modify the storage path using the ‘Storage’ icon

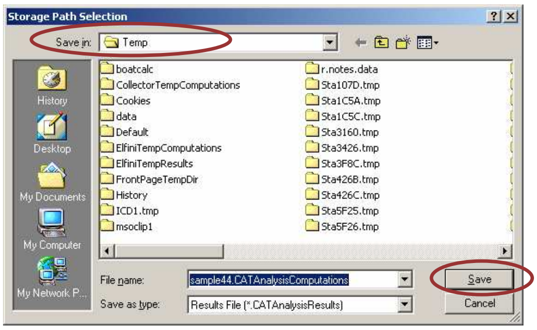
1. Click on the ‘External Storage” icon in the Solver Tools toolbar

The ‘External Storage’ dialog box appears

2. Click the Modify button







3. Select a path for the External Storage directories

4. Click on Save

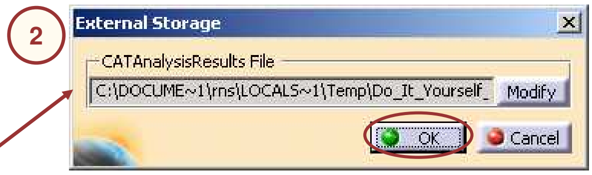
5. Click on OK

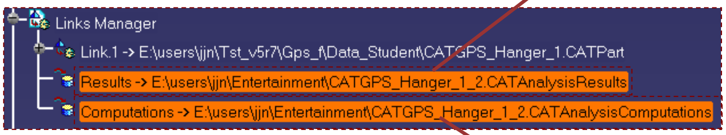
Instructor Notes:

Specifying Storage Path in Specification Tree

You can modify the storage path through the specification tree.

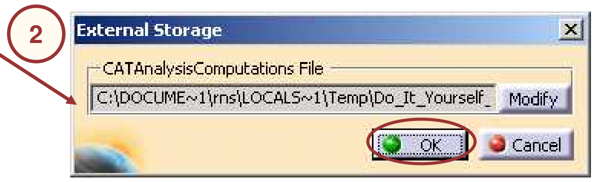
1 Double click on the path you want to modify





2 Select a new path

3 Click on OK



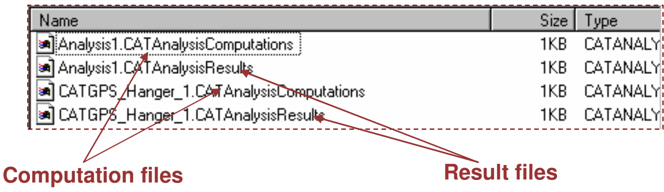
Instructor Notes:

Clearing Old Computations 

Each new computation generates new files. The new files will overwrite the corresponding old ones. Before launching a new computation you may clear the 'Computation Data' and / or the 'Result Data' if you want it to supersede the previous one.

1 Click on the 'External Storage Clean-up' icon 

2 Select the action you want



You can either clear the computation data only or the result data as well:

Instructor Notes:

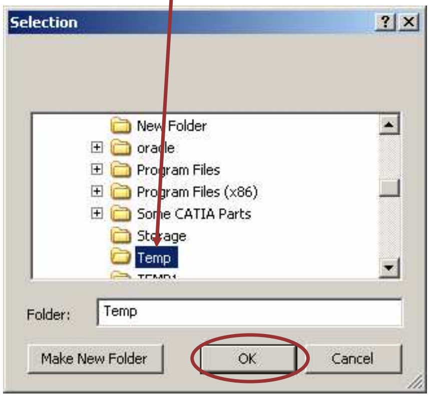
Specifying the Temporary Storage Location 

To calculate, the computer needs a temporary storage location which is cleaned up when the associated analysis session is closed.

1 Click on this icon 

2 Click on "Modify" to select a new directory

3 Select a new path



4 Click on OK

Instructor Notes:

Additional Information

Creating analysis storage

CATAnalysisResults and CATAnalysisComputations files are created :

- The first time you run a computation
- If the user explicitly defines their location

An analysis document which contains only specifications can be stored without links to the Analysis storage.

These files are not seen anymore in partner applications that do not need them.

Reading analysis storage

Data is copied only when it needs to be accessed by computation or post-processing. There is a significant time gain when loading a Computed Analysis Document.

Useless data no longer needs to be read (ex : read a computed document, modify the mesh, and re-computing).

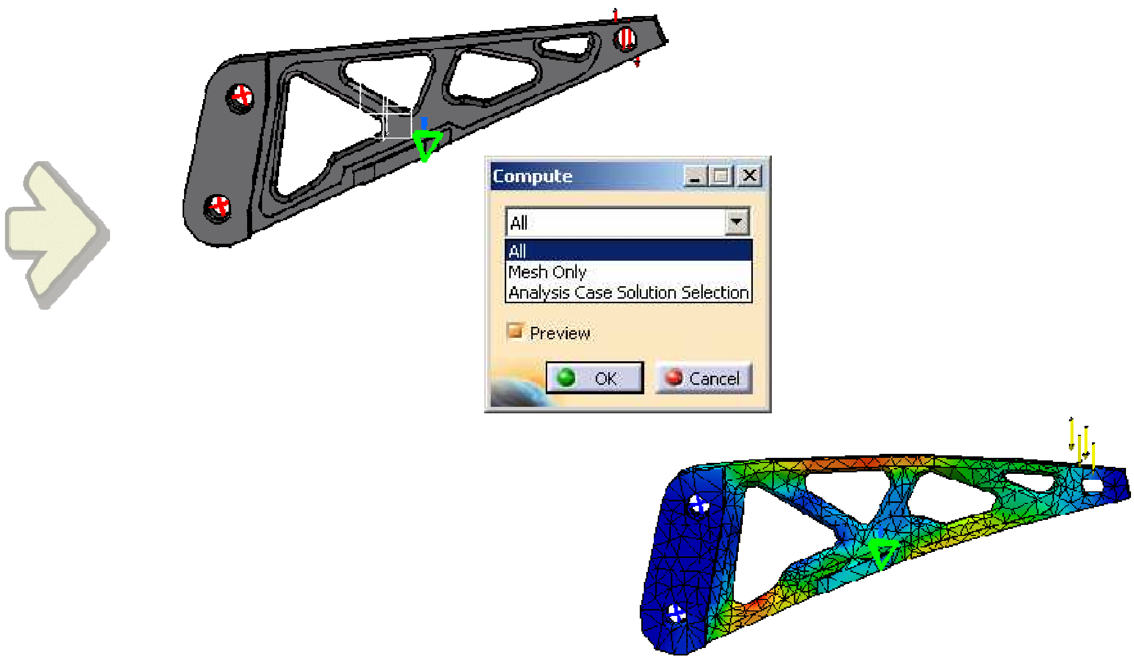
Deleting analysis storage

CATAnalysisResults and CATAnalysisComputations can be deleted manually (equivalent to the Clear capability).

Instructor Notes:

Computing a Static Case

You will learn how to compute a Static Case analysis.



Instructor Notes:

Introduction

At this step of your work you must make sure that your materials, restraints and loads are successfully defined (use Model Checker tool).
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 Static Solution Computation result consists in a displacement vector whose components represent the values of the system's DOF. This result can be further processed to produce other results such as stresses, reaction forces and so forth.

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).**

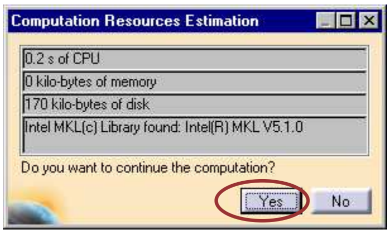
Instructor Notes:

How to Compute a Static Case Analysis

1 Click on the ‘Compute’ icon

2 Choose the compute option you want

3 Check ‘Preview’ if you want an estimation of the computation time.



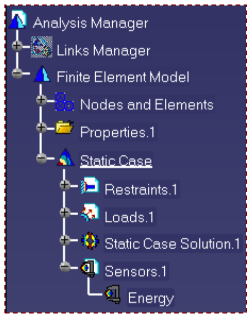
In some cases, if equilibrium is not reached, interactive warning message may inform you that the residual forces are not negligible.

4 Click on Yes to run the computation

A series of status messages (Meshing, Factorization, Solution) informs you about the progress of the computation process. The Static Analysis Solution is computed and can be visualized.

Upon successful completion of the computation the status of all objects in the analysis feature tree is turned to valid. You can now:

- analyze the report of the computation
- visualize images for various results

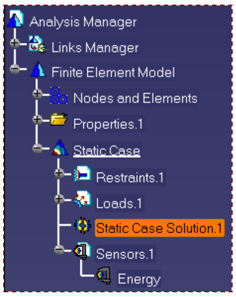


Instructor Notes:

How to Specify Static Solution Parameters

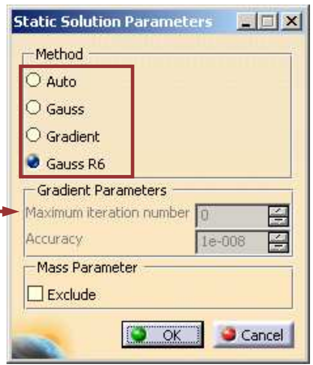
The Definition parameters of an Analysis Case must not be confused with Computation parameters of a Case Solution. In fact, the New Analysis Case dialog box is displayed at the time of a case insertion and the defined parameters cannot be modified once the Case has been created. However, the Computation parameters of a Case Solution are proposed by default at case creation and are editable afterwards.

1 Double-click the Static Case Solution.1 to display the Static Solution Parameters dialog box.



2 Check the method you want to apply and fill the gradient parameters if needed

For the gradient method two additional parameters must be specified



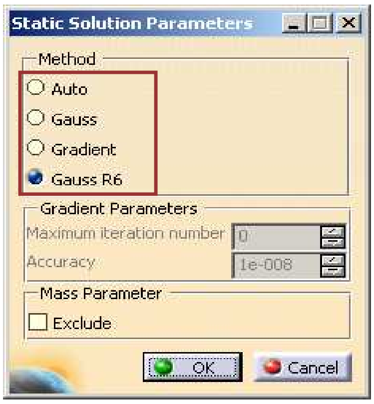
3 You can Check for ‘Exclude’ to exclude structural Mass.

4 Click on ‘OK’

Instructor Notes:

Solving Methods

There are four different solving methods for a static analysis :



1- <u>Auto method</u> : One of the three methods below is automatically computed

Works for any analysis method.

2- <u>Gauss method</u> : Direct method

Recommended for computing small/medium models.

3- <u>Gradient method</u> : Solving iterative method which is memory saving but not CPU time saving

Recommended for computing huge models. Two additional parameters must be specified : maximum iteration number and accuracy factor.

4- <u>Gauss R6 method</u> : Fast Gauss method

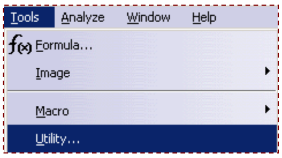
Recommended for computing large size models (<u>default method</u>).

<u>Instructor Notes:</u>

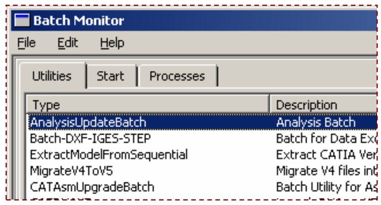
How to Launch Batch Processing

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

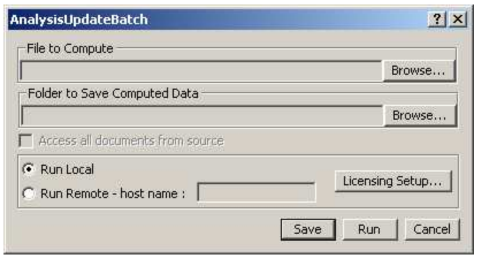
1 Go to Tools > Utility…



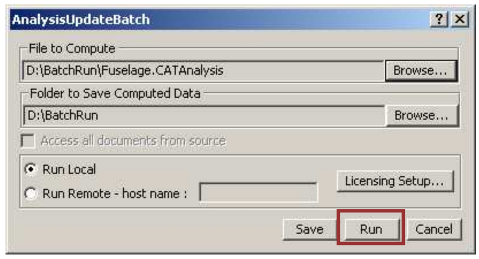
2 Double-click 'AnalysisUpdateBatch'



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



Instructor Notes:

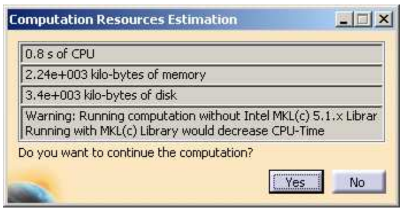
Exercise

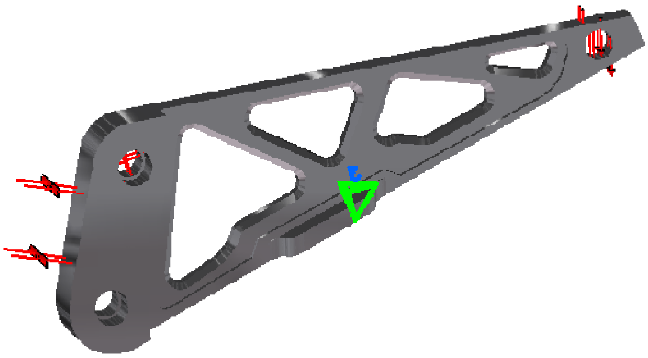
'Computation' Recap Exercise

5 min

In this exercise you will compute the Analysis that you have defined in the first recap exercise.
It includes following steps:

- **Specify the storage folders**
- **Compute the analysis**





Instructor Notes:

GPS Post-processing

In this lesson, you will learn about the main tools used to display and optimize the results.

- Results Visualization
- Mesh Refinement
- Results Management

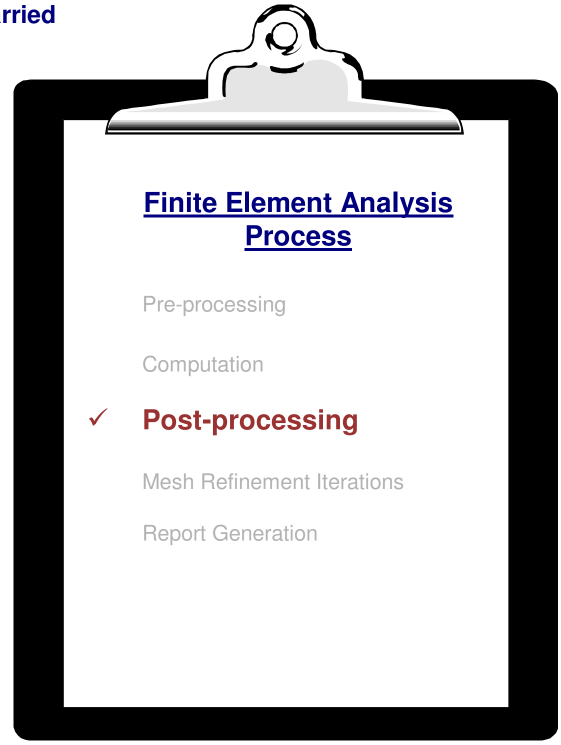
Instructor Notes:

Post-processing

Post-processing involves all those steps which are carried out after the computation of results.

Under post-processing, you will:

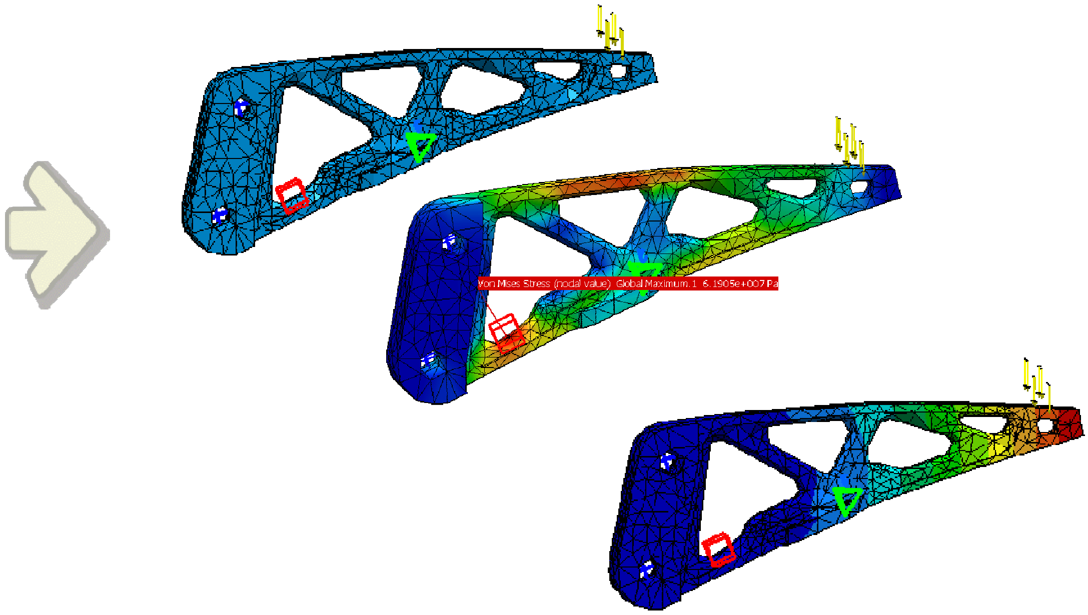
- Create Deformation, Displacement Magnitude, Stress, Reaction Force and other types of images from the computed solution's data.
- Validate the results using different images and study these images to understand and interpret the solution.
- Take decisions for refining the solution further using Mesh Refinement Iterations or other solution types.
- Validate the current design or suggest design changes based on the results.



Instructor Notes:

Result Visualization

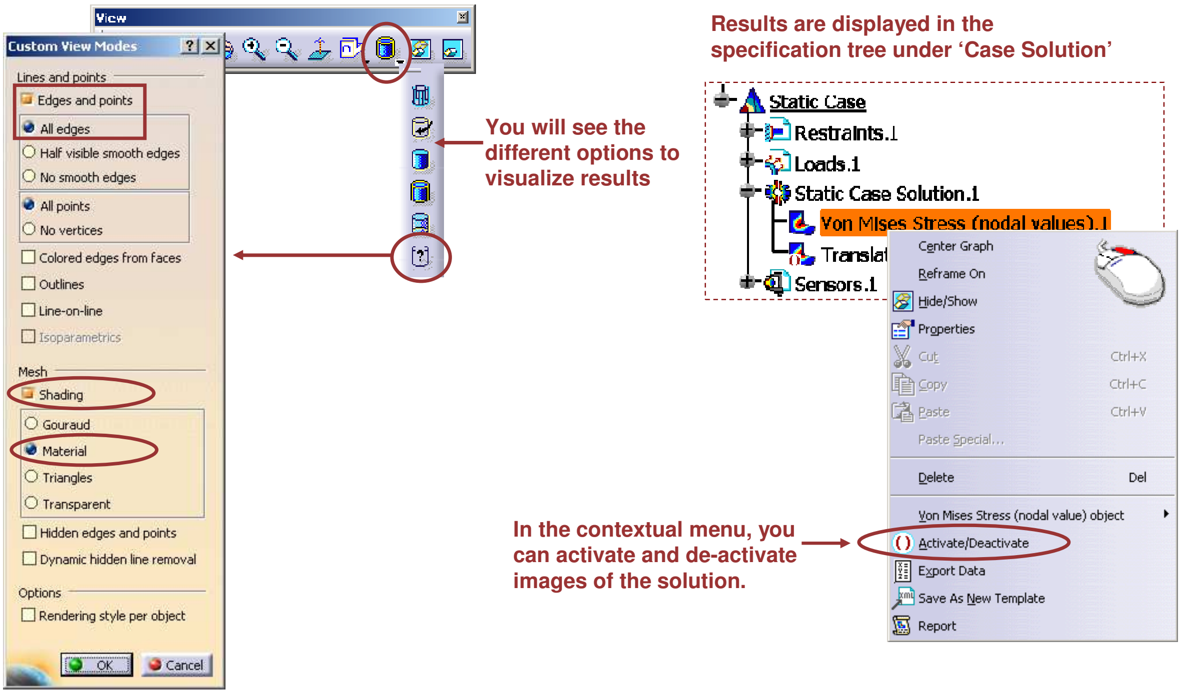
You will learn the functionalities to display the result images and animate the results.



Instructor Notes:

Image Creation

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 'Materials' options are activated in the Custom View Modes dialog box:



Instructor Notes:

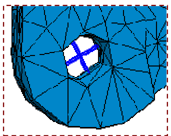
About Deformed Image

Deformed Mesh images are used to visualize the finite element mesh in the deformed configuration of the system, as a result of environmental action. Deformed Mesh Image objects can belong to Static Case Solution object sets or to Frequency Case Solution object sets.

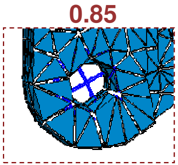
Before you begin: Make sure that the view is correctly customized and you have computed the solution.

You can modify the 'Shrink Coefficient'

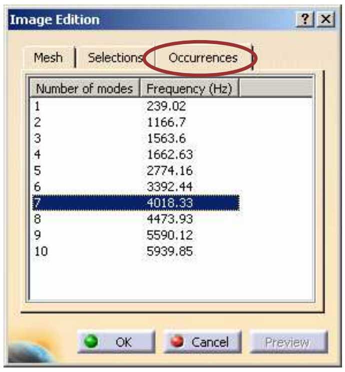
1.00



0.85



These mesh entities are either all the elements included in the mesh part or the elements or nodes associated to the geometry supporting a specification. This is available for all types of specifications: Mesh specifications, Connections, Loads, Restraints and Masses.



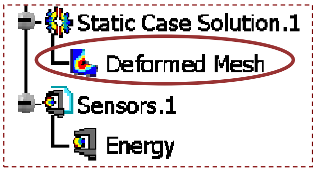
By pressing the Frequencies tab you display the list of modes with the associated frequencies. You can then activate separately each mode of this multi-occurrence solution.

Occurrences Tab occurs only in case of Frequency Analysis.

Instructor Notes:

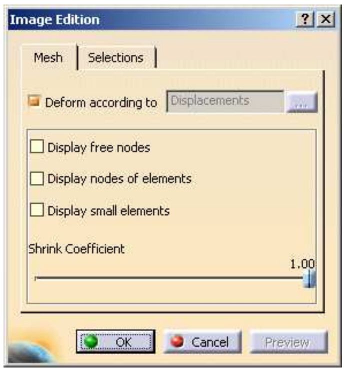
Visualizing Deformations

1 Click on the Deformation icon

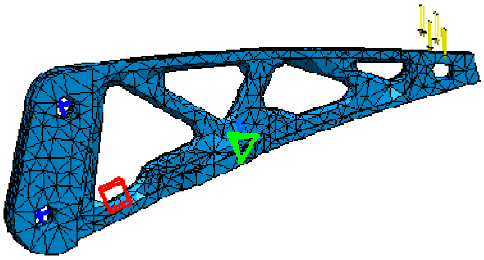


2 (Optional) Double-click on the Deformed Mesh object in the specification tree to edit the image

If needed, modify the parameters



3 Click OK to quit the Image Fem Editor dialog box

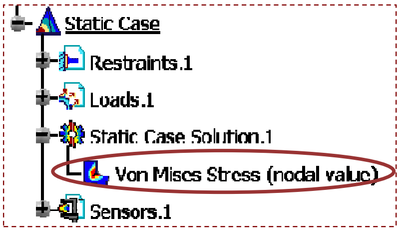


Instructor Notes:

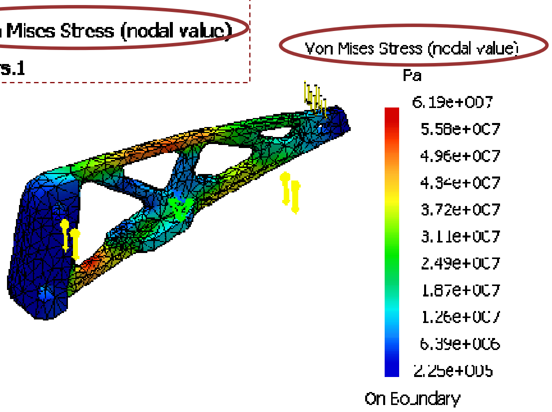
About Von Mises Stresses

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

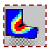
The volume distortion energy density is often used in conjunction with the material yield stress value to check part structural integrity according to the Von Mises criterion. For a sound structural design, the maximum value of the Von Mises stress should be less than this yield value.

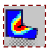


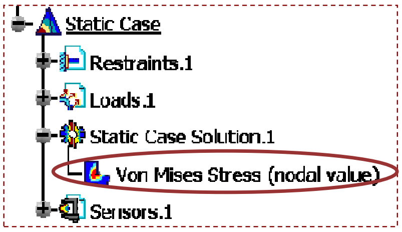
To edit the Image Edition dialog box you have to double click on the 'Von Mises Stress' object in the specification tree.



__Instructor Notes:__

Visualizing Von Mises Stresses 

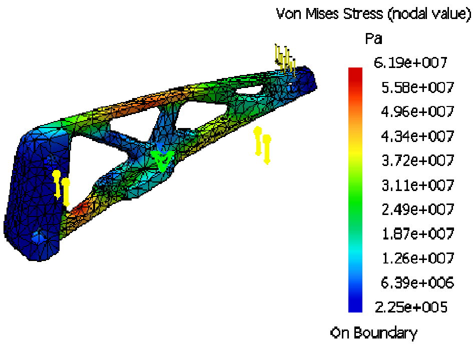
1 Click on the Stress Von Mises icon 



2 (Optional) Double-click on the Von Mises object in the specification tree to edit the image

If needed, modify the parameters

3 Click OK to quit the Image Fem Editor dialog box



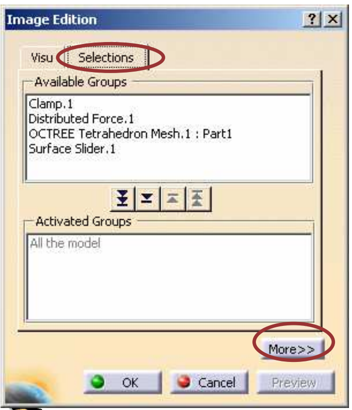
Instructor Notes:

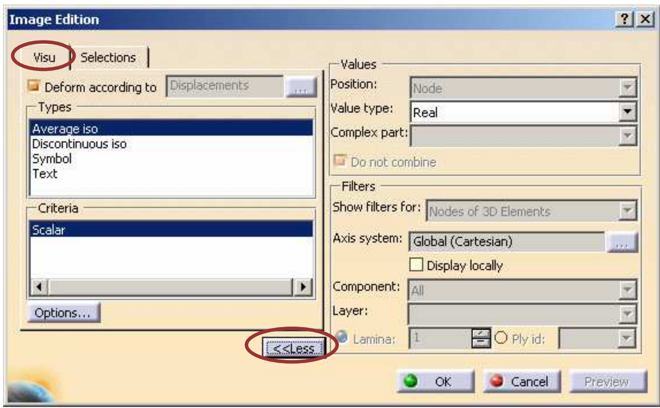
Von Mises Stresses Image Edition

The 'Image Edition' dialog box is composed of 2 tabs:

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

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





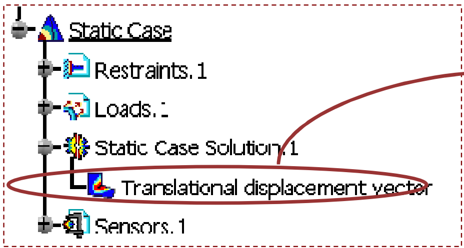
When you click on 'More', some tab can be grayed depending on the the 'type' you choose in the 'Visu' tab.

Instructor Notes:

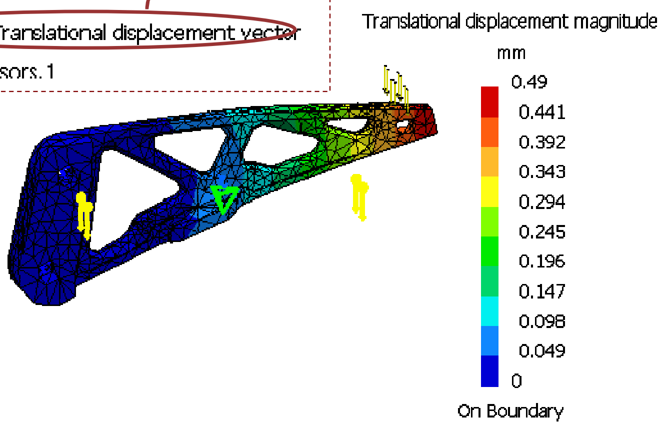
About Displacements

Translational Displacement vector images are used to visualize displacement field patterns, which represent a vector quantity field equal to the variation of the position vectors of mesh entities as a result of environmental action (loading).

Translational Displacement vector Image objects can belong to Static Case Solution objects sets or to Frequency Case Solution objects sets. The displacement resulting from part loading is important for a correct understanding of the way the part behaves.



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

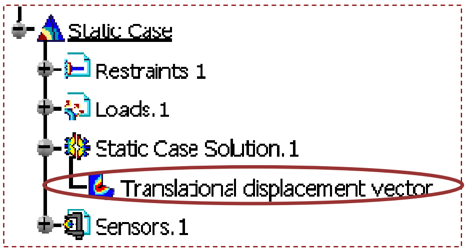


Instructor Notes:

Visualizing Displacements

This task shows how to generate Displacements images on parts.

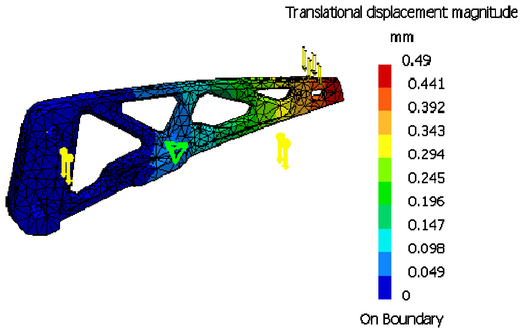
1. Click on the Displacements icon



2. (Optional) Double-click on the *Displacements* object in the specification tree to edit the image

If needed, modify the parameters

3. Click OK to quit the Image Editor dialog box



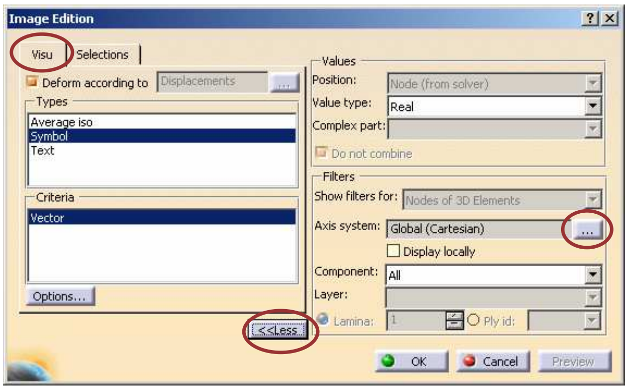
Instructor Notes:

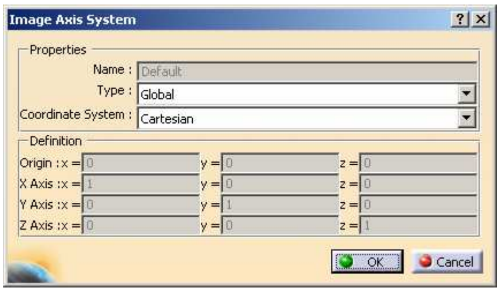
Displacement Image Edition

The 'Image Edition' dialog box is composed of 2 tabs:

- **Visu: provides a list with visu types (Average-Iso, Symbol, Text). Both the option type and visualization are appended to the name. It also provides a list with criteria (VECTOR).**
- **Selections: In the case of CATProducts, pre-defined groups of elements belonging to given mesh parts can be multi-selected.**

More: The only available option to generate images is on nodes. You can also choose Value type options.





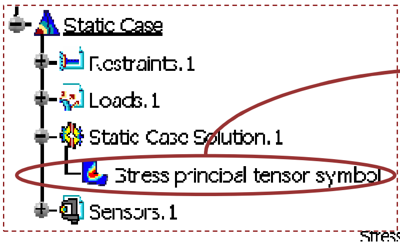
For **angular displacements you can modify the axis-system and choose a cylindrical user axis-system (with x and y in the rotational plane) with tangent displacements components.**

Instructor Notes:

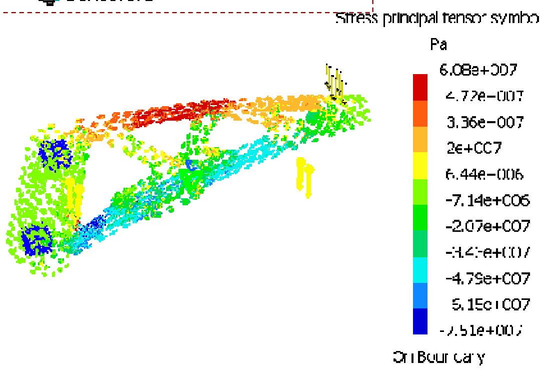
About Principal Stresses

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 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.**

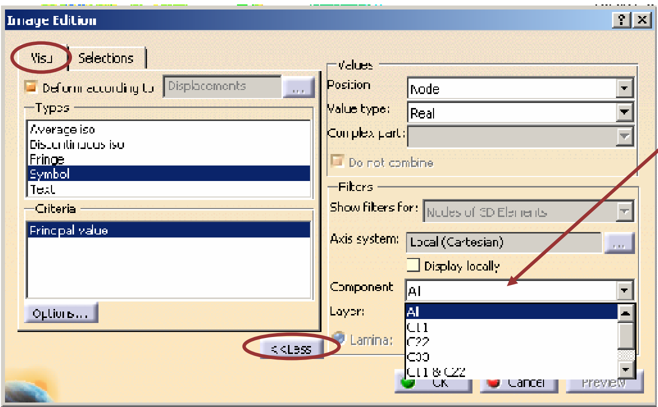
Instructor Notes:

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, Symbol and 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.**

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



Components **:**

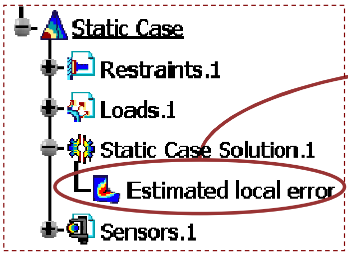
C1 is the max principal stress
C2 the middle principal stress
C3 is the min principal stress
(C1>C2>C3)

Instructor Notes:

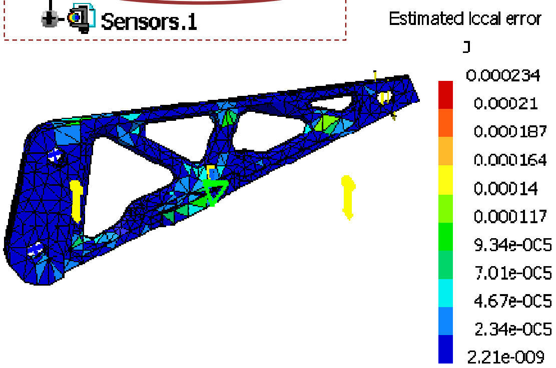
About Precisions

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 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.

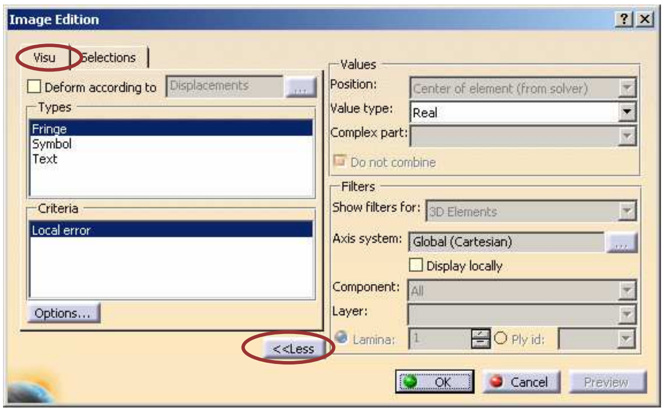
Instructor Notes:

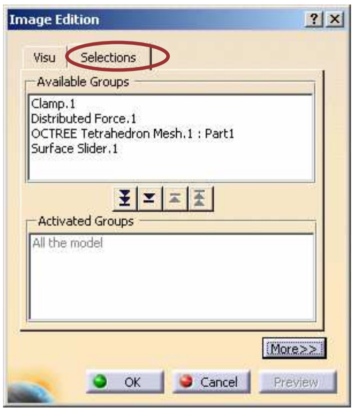
Precisions Image Edition

The 'Image Edition' dialog box is composed of 2 tabs:

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

  More: provides different filters. Only Center of element option is available. You can also choose Value type options.





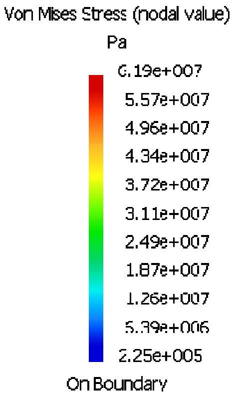
Instructor Notes:

How to Edit the Color Map

When you use the 'image layout' tool, you need to know how to manage the color map for a better results presentation.

When you display a result a color map appears. You can edit it by double-clicking on it.

1 Double-Click



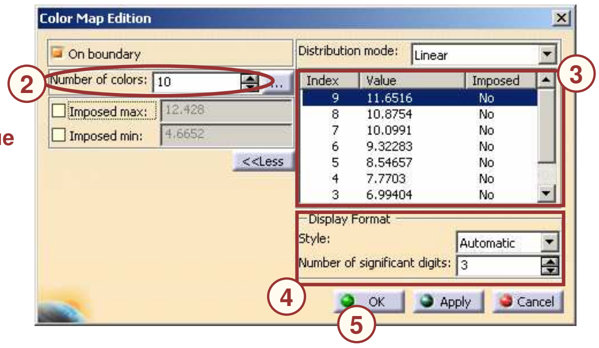
2 You can change the number of colors

3 You can impose value

4 You can change the display format

5 Click on OK when you are done

The color map Editor dialog box appears:

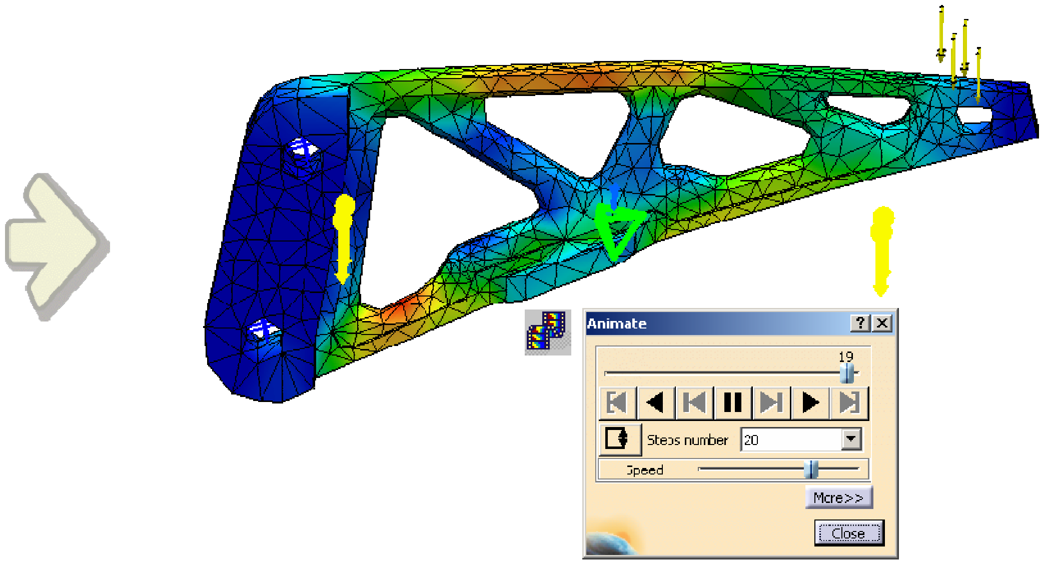


The Color map is like any object: it can be moved. You need to click on it (to activate it) and then you can drop it with the middle mouse button

Instructor Notes:

Image Animation

You will see how to generate an animation of the results.

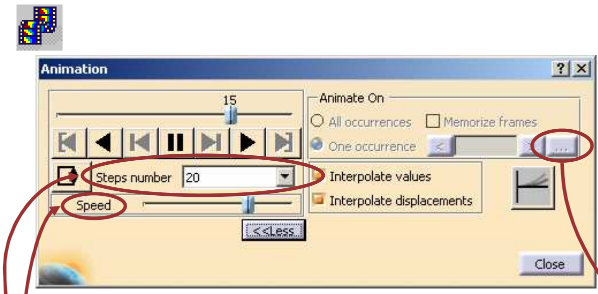


Instructor Notes:

Introduction

Image Animation is a continuous display of a sequence of frames obtained from a given image. Each frame represents the result displayed with a different amplitude. Running the frames sequence gives a feeling of motion.

By animating a deformed geometry or a normal vibration mode, you can get a better insight of the system's behavior.

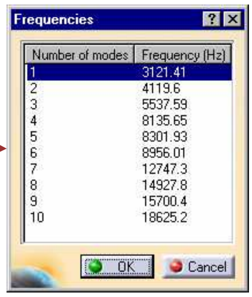


"Interpolate Values": If you check this option, the stress values will be interpolated at each steps of the animation, if not, they will be fixed (Max. values).

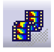
"Interpolate Displacements": If you check this option, the stress values will be animated whereas the deformation will be locked at its maximum.

You can change the number of steps and the speed of the animation

For a dynamic analysis (buckling as well): select the mode to display



Instructor Notes:

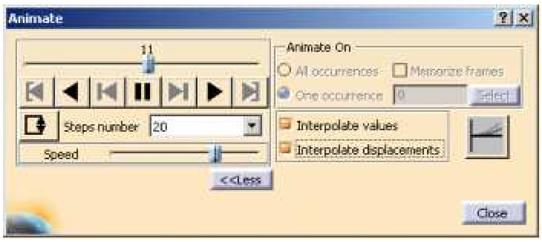
Using the Images Animation Tool 

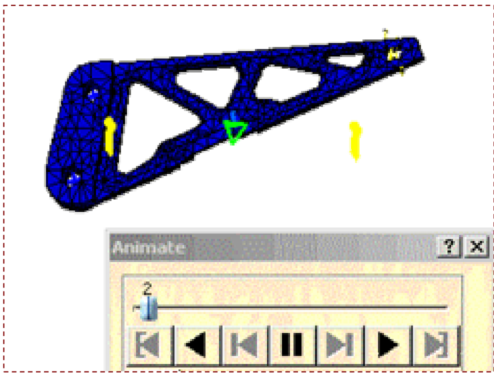
1 Display the result you want with the 'Image' dialog box

Static representation of the results Image appears.

2 Click on the 'Animate' icon

3 Define the options





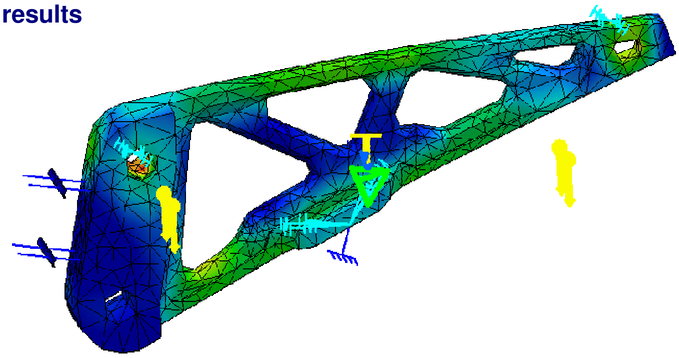
Instructor Notes:

Exercise

'Results Visualization' Recap Exercise

In this exercise you will visualize results of the analysis that you have computed in previous recap exercise. You will:

- Visualize the Von Mises Stress
- Visualize the deformation
- Activate layout to display different results in the same time



Instructor Notes:

Mesh Refinement

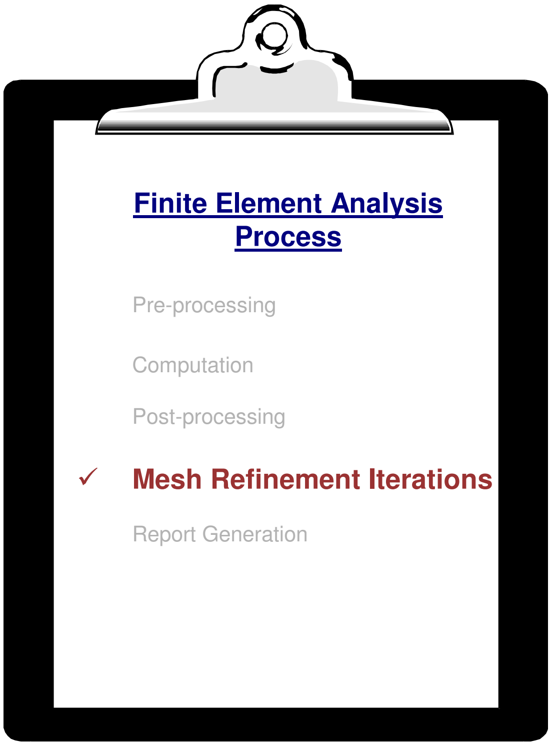
In this lesson, you will learn about the different ways to improve the precision of your results.

- What is Mesh Refinement
- Improving the Element Characteristics
- Refinement Recap Exercise
- Mesh Refinment with Precision: Recap Exercise

Instructor Notes:

What is Mesh Refinement (1/2)

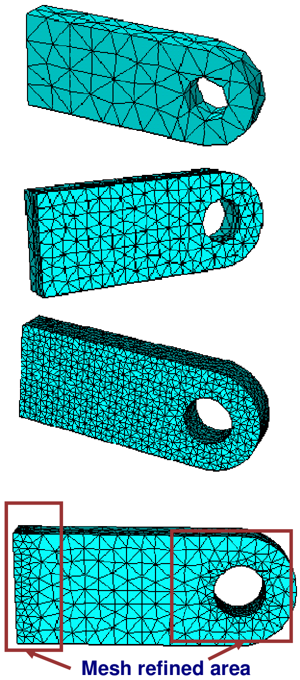
- **Initial FE solution provides you the results that are generally obtained using coarse meshing and simple element types.**
- **The purpose of these initial results is to get a rough idea of the results using simple FE modeling and minimum computation time.**
- **Mesh Refinement is nothing but refining the existing mesh further to get a more accurate representation of the actual physical model.**
- **This will help to reduce the discretization error and the solution will converge towards a more accurate solution.**
- **The mesh can be refined either by increasing the number of finite elements or by using the higher order elements.**
- **There can be several Mesh Refinement Iterations to achieve the exact solution.**



Instructor Notes:

What is Mesh Refinement (2/2)

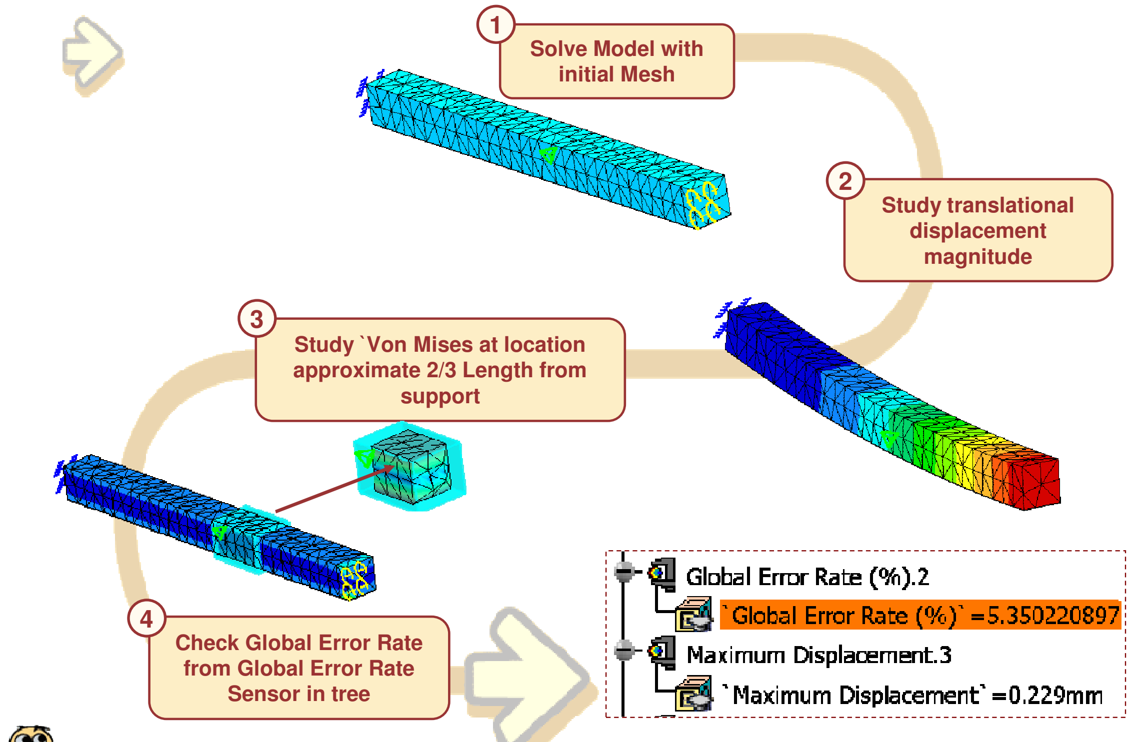
You will need to consider the following while refining the mesh based on precision:



- **First analyze component with a coarse mesh. A coarse mesh is to provide only an initial primary guess of stress values and does not give the clear picture of stress distribution.**
- **Thus you can further refine the mesh globally to get comparatively accurate and continuous picture of stresses. There can be more iterations of global mesh refinement if the stress pattern is going to be more continuous and stress values are changing significantly.**
- **Successive mesh refinement may not lead to significant change in stress values in major areas of the component. It may however increases CPU computation time. You can spot these areas within precision plots where the 'local error' distribution is comparatively low and uniform.**
- **Using precision plot you will also get areas of high local error values. These are generally the areas where loads, constraints are applied on the component and areas of abrupt change in geometry and cross section. If the mesh is further refined in these areas, it will improve the stress continuity leading to accurate values of stresses in these regions.**

Instructor Notes:

Mesh Refinement General Process



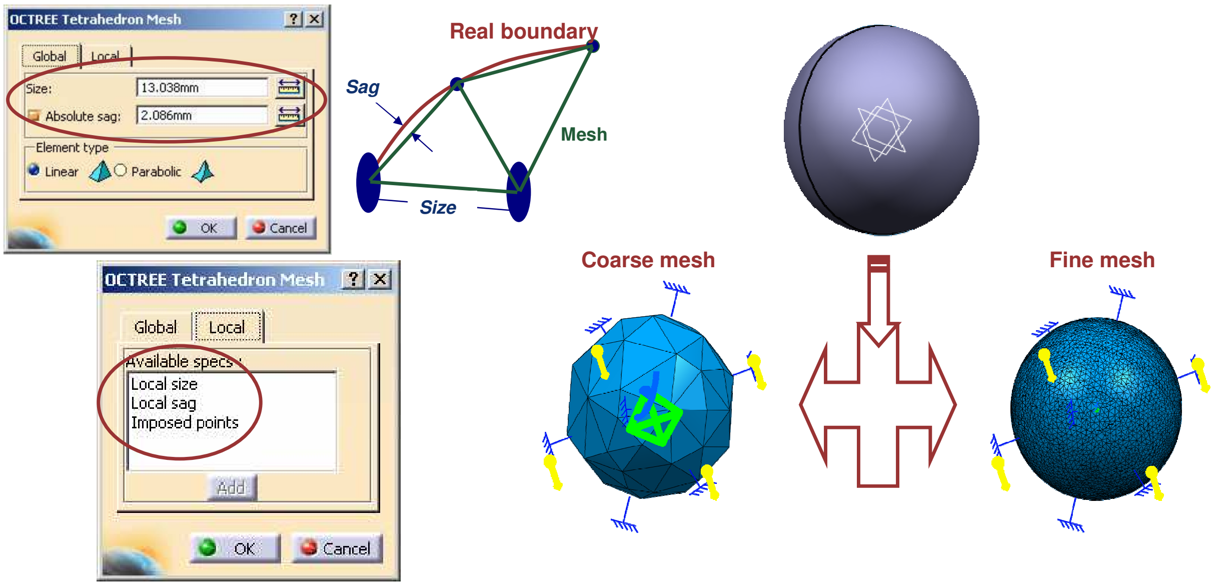
Iterate the process till Global error reduces and error in energy value is in order of 10e-8.

Instructor Notes:

Global and Local Mesh Refinement

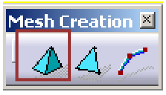
The second step when you want to improve the precision of your analysis results is to refine the mesh of your part. You can refine the Size of a mesh, and the Sag (chord error). This can be performed both globally and locally.

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.



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"

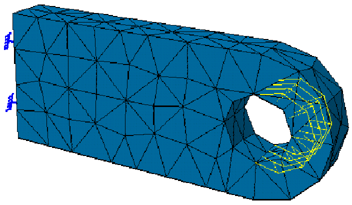
4. Select the local area (support)
5. Enter a new value
6. Click on OK

Instructor Notes:

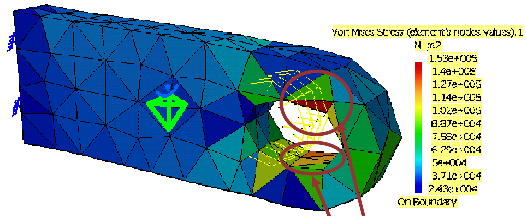
How to Refine Mesh with Precision (1/2)

You will see how to optimize mesh refinement iterations using precision.

1 Solve Model with initial Mesh



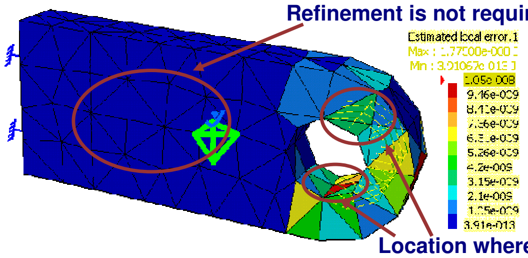
2 Study Von Mises Stresses Discontinuous iso



Location of Significant stress discontinuity

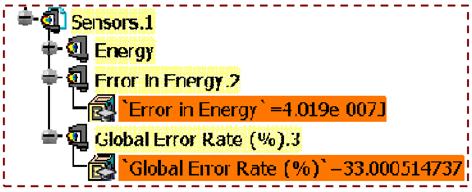
3 Study Local Error Distribution to find areas of High error value

Area where further mesh Refinement is not required



Location where Mesh refinement required.

4 Check Global Error Rate from Global Error Rate Sensor in tree



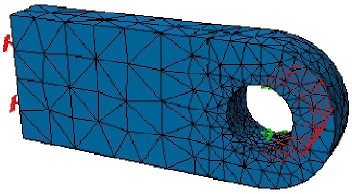
Note that High values of local error are present approximately at location of significant element stress discontinuity, shown in discontinuous iso. This discontinuity is reduced by reducing size of the elements. Therefore, refine mesh in that region and recalculate the solution.

Instructor Notes:

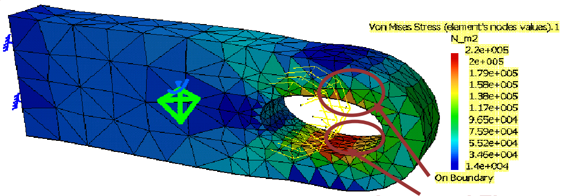
How to Refine Mesh with Precision (2/2)

A high value of Local Error is a measure of stress discontinuity at that location. The more the stress pattern in the 'Discontinuous Iso Image' is uniform, the more accurate the solution will be. In each Mesh Refinement step, the 'Global Error Rate' decreases. The 'Global Error Rate' sensor indicates overall accuracy of the solution. The 'Error in energy' value should be in order of 10e-8. It depends on the judgment of Analyst.

5 Refine Mesh and solve Model

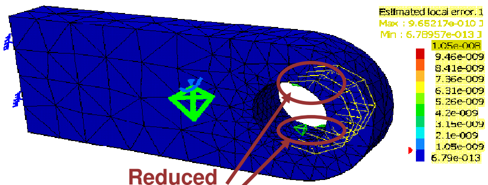


6 Study Von Mises Stress Discontinuous iso



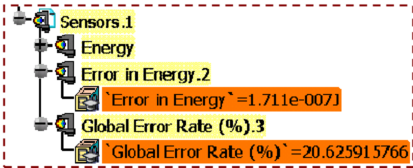
Improved Element Stress Continuity

7 Study Local Error Distribution



Reduced Local error

8 Check Global Error Rate from Global Error Rate Sensor in tree

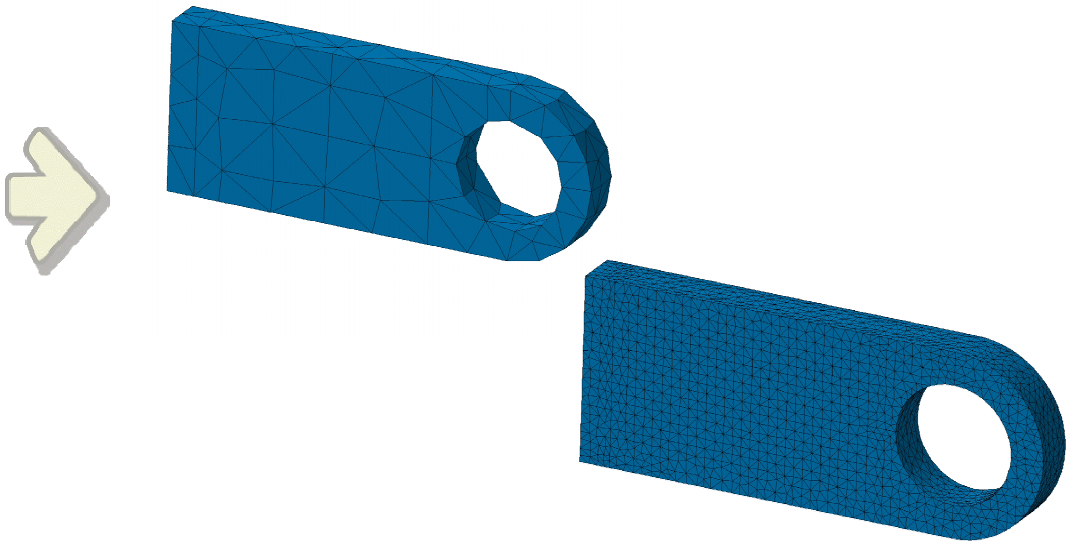


In general, the accuracy of a solution can be improved from an optimum number of global and local mesh refinement iterations with the help of precision plots. Note that Discontinuous iso should only be used for mesh refinement and not to calculate Factor of Safety while designing.

Instructor Notes:

Analysis Result Validation

You will see the main criteria and indicators which allow you to evaluate the quality of a mesh.



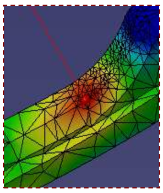
Instructor Notes:

Basic Criteria

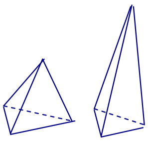
Some criteria allow you to quickly validate or not a mesh :

Local and global error : in practice, they must be respectively around 5% and 10% Max.

Maximum stress value and stress evolution/repartition : Areas which have important stress gradient must be refined with a finer mesh.



Mesh quality : keep in mind that tetrahedrons must keep their ideal shape. Brutal mesh transition can also give erroneous results. Avoid big ratio between the length of your elements (5 Max.) and too big or small angles.



Below are some suggestions to optimize the accuracy of your analysis:

Type of elements : linear or parabolic. The first does not take into account bending effects (too stiff). The second does but costs more in terms of time solving.

Mesh density : finer meshes in important stress gradient areas give good results, but imply much more DOF. Consequently, low stress areas can have a coarse mesh. You can use local size, local sag, or adaptivity.

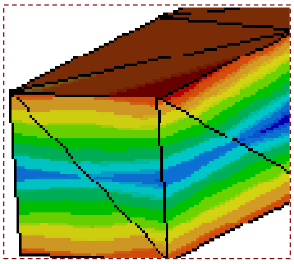
Boundary conditions and connections : BC areas must be chosen to represent reality as much as possible.

Instructor Notes:

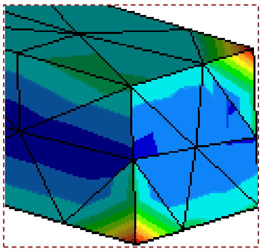
Effect of Mesh Refinement (1/2)

Model solved with various stages of global mesh refinement.

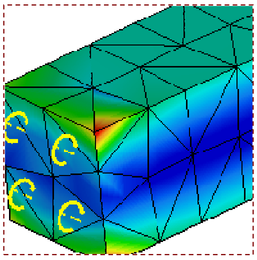
Mesh size = 1000



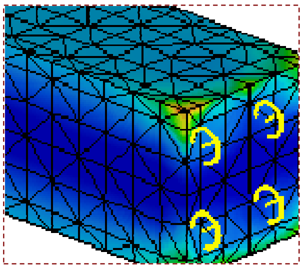
Mesh size = 500



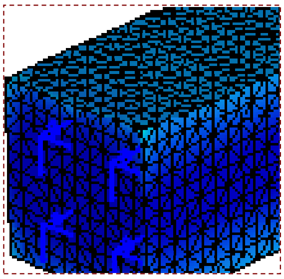
Mesh size = 200



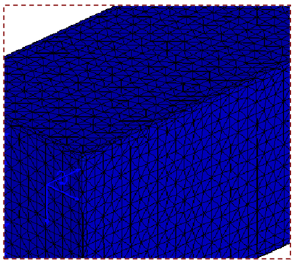
Mesh size = 100



Mesh size = 50



Mesh size = 30



Instructor Notes:

Effect of Mesh Refinement (2/2)

Following are results of cantilever beam global mesh refinement.

Mesh Refienment	No Of Nodes	No Of Elements	DOFs	Max. Displacement (mm)	Max. Von Mises at 2/3 L (N/m2)	Principal Stress at 2/3 L (N/m2)	% Global Error	CPU Time (sec)
Size=1000, sag=4	117	36	351	0.227	6.4e+8	6.4e+8	5.677	0.002
Size=500, sag=4	580	243	1740	0.229	6.4e+8	6.4e+8	5.350	0.06
Size=200, sag=4	2557	1492	7671	0.231	6.4e+8	6.4e+8	4.952	0.8
size=100, sag=4	15935	9382	47805	0.231	6.4e+8	6.4e+8	3.206	2e+1
size=50, sag=4	64714	40449	194142	0.231	6.4e+8	6.4e+8	2.363	5e+2
size=30, sag=4	198075	123953	594225	0.231	6.4e+8	6.4e+8	1.812	6e+3

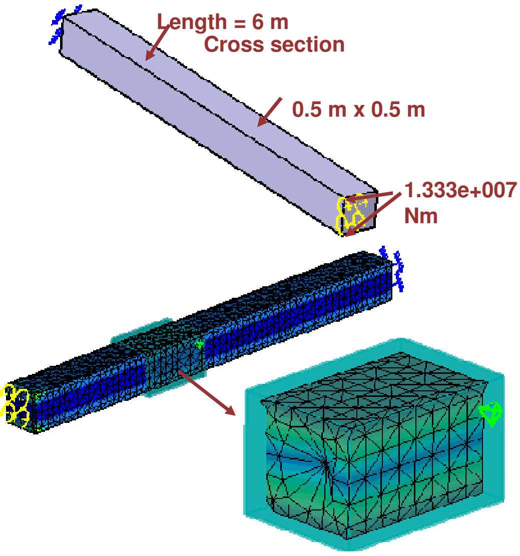
Instructor Notes:

Comparison With Classical Solution

You will compare these results of cantilever beam with classical calculation.

Maximum Bending Stress = M. (Y_{max})/ I

M = Moment Applied
Y_{max} = Maximum distance from neutral axis
I = Moment of Inertia



In this case

M = 1.333e+007 Nm

Y_{max} = 0.25 m

I = $(0.5)(0.5)^4/12$ m^4

Maximum Bending Stress = 6.40 e+ 8 N/m^2

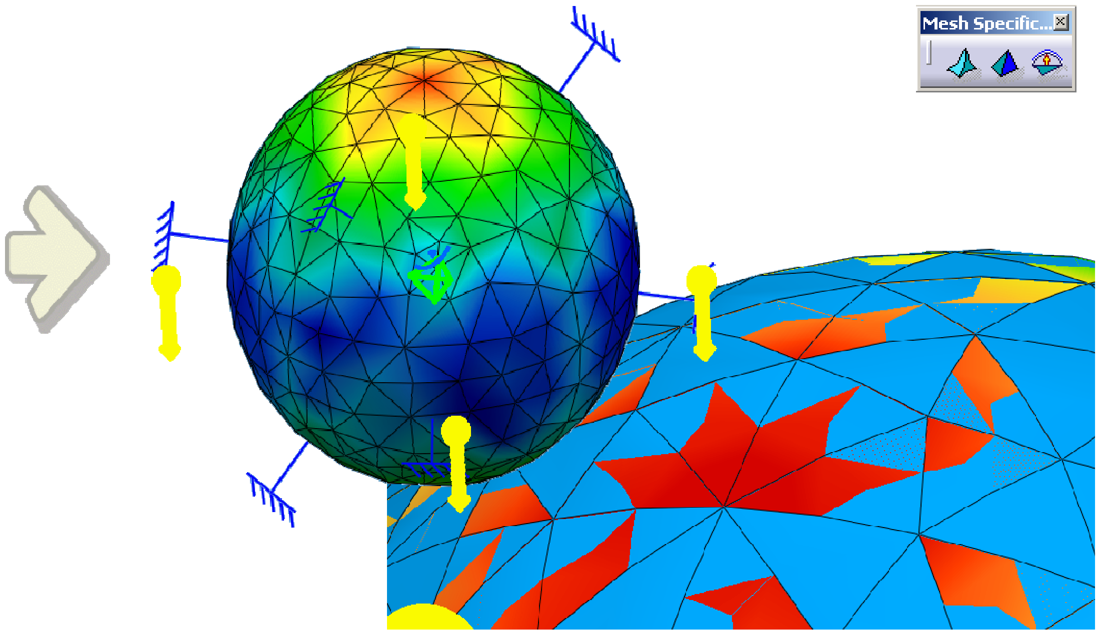
From Analysis Von Mises Stress at 2/3 length from support = 6.40 e+ 8 N/m^2

Please note that results in immediate vicinity of loads, sharp corners, rapid change in cross section etc. give very high stress values and are inaccurate. This is because of local singularity i.e. an infinite local stress concentration at that location. Therefore, more reliable values of stresses are always obtained away from such conditions. These values can be used for comparison with classical calculations.

Instructor Notes:

Improving the Element Characteristics

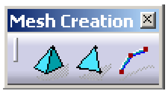
You will see what are the different types of elements and how to improve them.



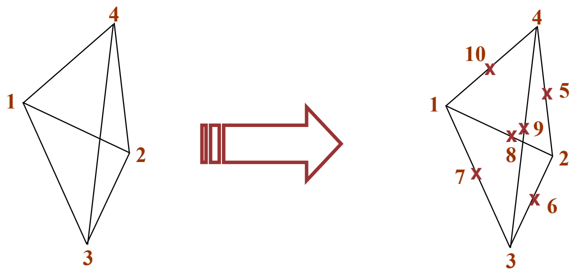
Instructor Notes:

What are the Different Element Types

When you have created your mesh, whether it is a 3D, 2D or 1D mesh, you have the possibility to define its element type. It can be "Linear" (default type) or "Parabolic".



For example, in the case of " Tetrahedron", a linear element has 4 nodes (TE4) whereas a parabolic element has 10 nodes (TE10). When using parabolic elements, the unknown field inside the element is interpolated with 2nd order polynomials.



TE4 TE10

In general, linear TE4 elements have a slower convergence than the Parabolic TE10 elements. For large meshes, TE4 elements can provide erroneous results. Consequently it is preferable to use TE10 elements.

Instructor Notes:

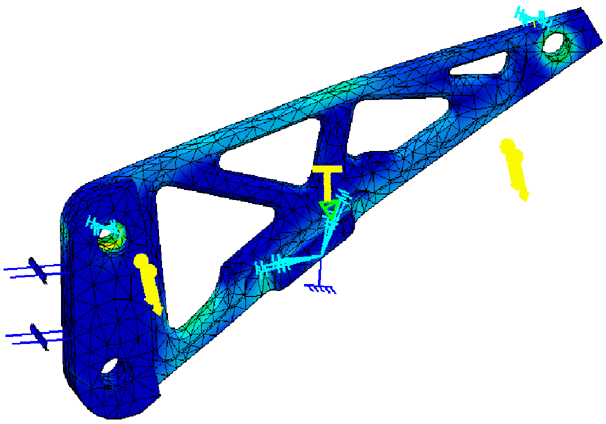
Exercise

‘Refinement’ Recap Exercise

10 min

In this exercise you will improve the mesh part using different methods you have just seen.
You will:

- **Improve the mesh type**
- **Refine the mesh**
- **Refine the SAG**



Instructor Notes:

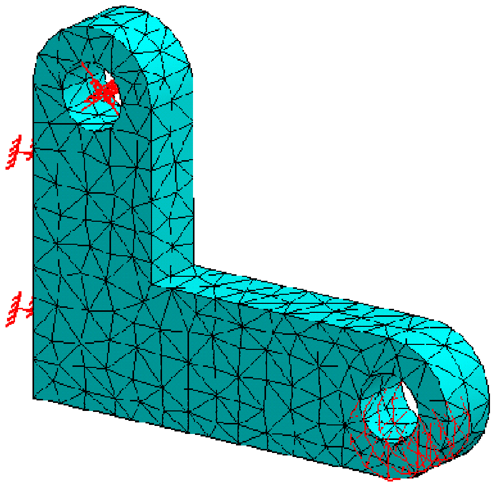
Mesh Refinement with Precision

Recap Exercise

60 min

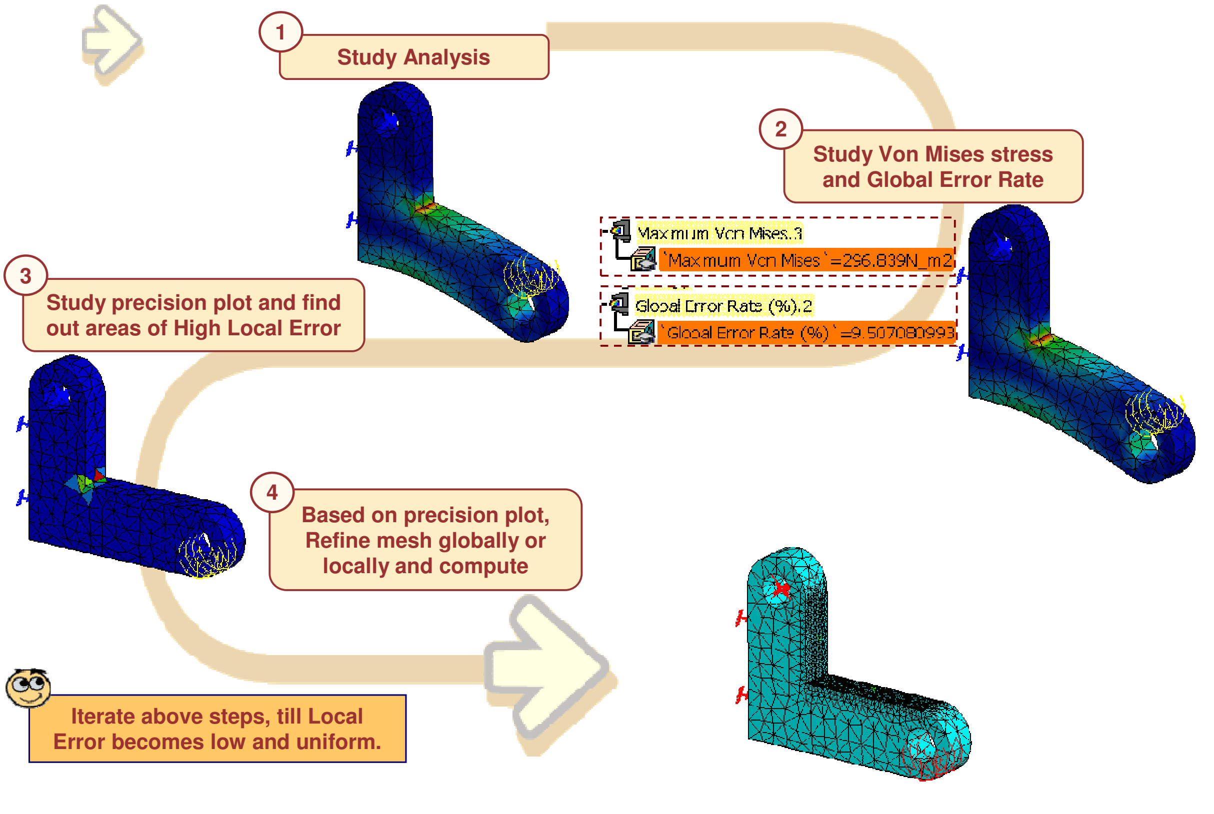
In this step you will :

- Study initial analysis with coarse Mesh
- Refine Mesh globally from Precision Plot study
- Perform Local Mesh Refinement Iteration
- Perform Local Mesh Refinement Iteration to further reduce local error



Instructor Notes:

Mesh Refinement - Design Process



Instructor Notes:

Results Management

You will learn the functionalities to get specific information regarding results and how to generate reports.

- Report Generation
- Extrema Detection
- Images Information
- Publishing Reports
- Results Management Recap Exercise

Instructor Notes:

Report Generation

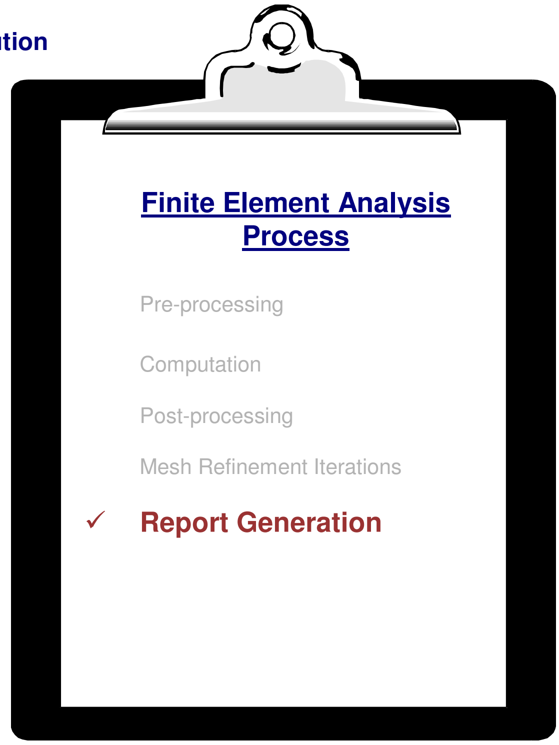
You can generate reports once the mesh refinement iterations are completed and the required level of solution accuracy is achieved.

In this step you will present information generated in following stages of FE Analysis process:

- Pre-processing Information
- Computation Information
- Results Information

Apart from Report Generation, you can analyze the results with the help of Result Management tools. These tools help you to get more detailed information about the results, such as:

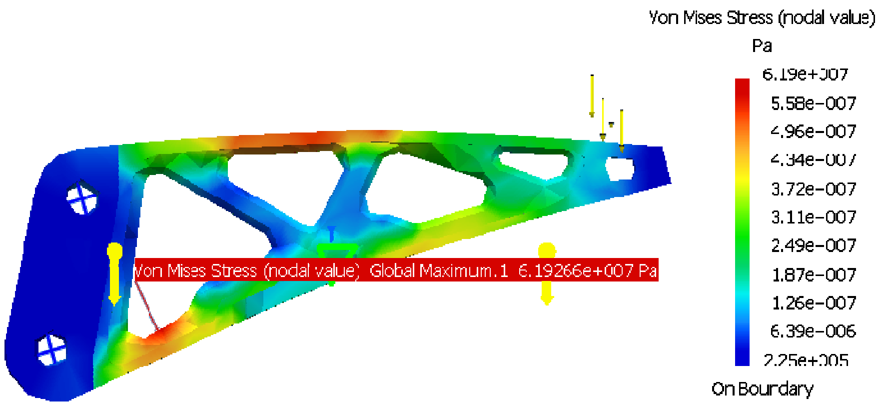
- Maximum and minimum values for displacements, stresses and other result quantities
- Error in energy
- Global error rate
- Image layouts
- Result animations



Instructor Notes:

Extrema Detection

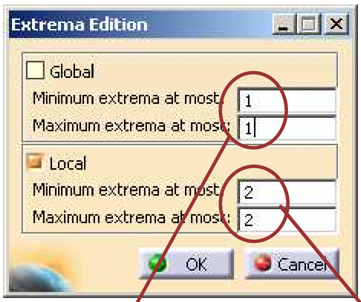
You will see how to detect extrema.



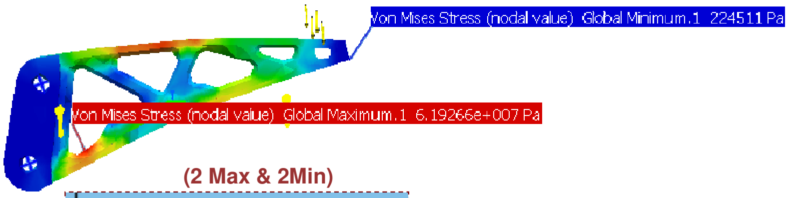
Instructor Notes:

About Extrema Detection

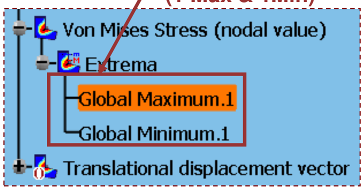
Extrema Detection consists in localizing points where a result field is maximum or minimum.



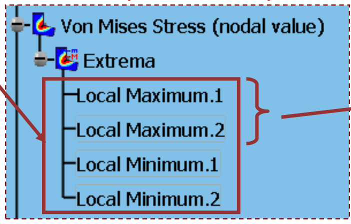
You can ask the program to detect given numbers of global (on the whole part) and/or local (relatively to neighbor mesh elements) extrema at most, by checking the "Global" and/or "Local" options. The example below asks for one minimum and one maximum Global extrema

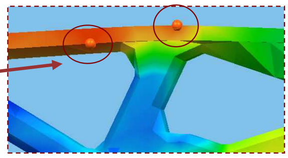


(1 Max & 1Min)

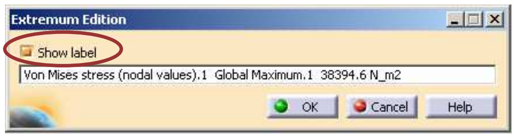


(2 Max & 2Min)





By double-clicking under "Extrema" you can display the extrema you want.

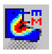


By double-clicking on "Extrema" within the specification tree, the same dialog box appears and you can choose to display the global or local extrema.

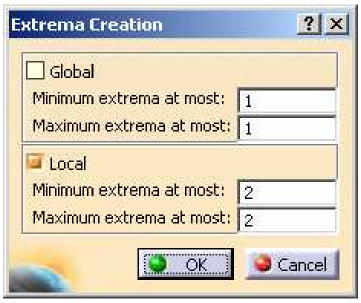
Instructor Notes:

Using the ‘Extrema Detection’ Tool

1 Select the image you want to visualize (typically Von Mises Stress)

2 Click on the “Image Extrema“ icon  in the “Analysis Tools” toolbar

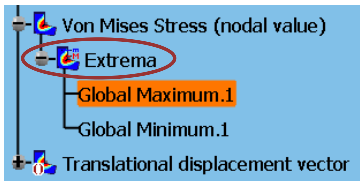
The following dialog box appears:



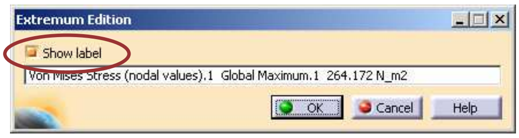
3 Specify if you want to run a “Global” or “Local” detection and enter the number of absolute extrema to detect

4 Optional: Double-click on “Extrema” in the tree and activate the results you want to see

The extrema values are indicated on the image and in the tree



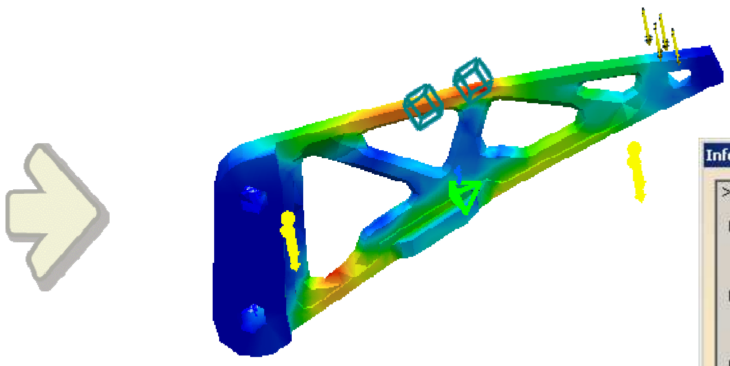
5 Optional: Double-click the results under “Extrema” and check “Show Label” to display the extrema values

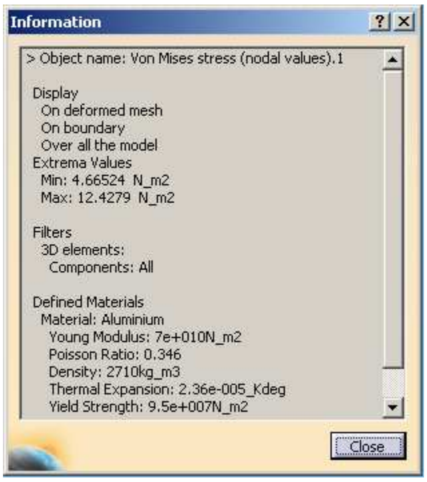


Instructor Notes:

Image Information

You will see how to access to a specific image information.





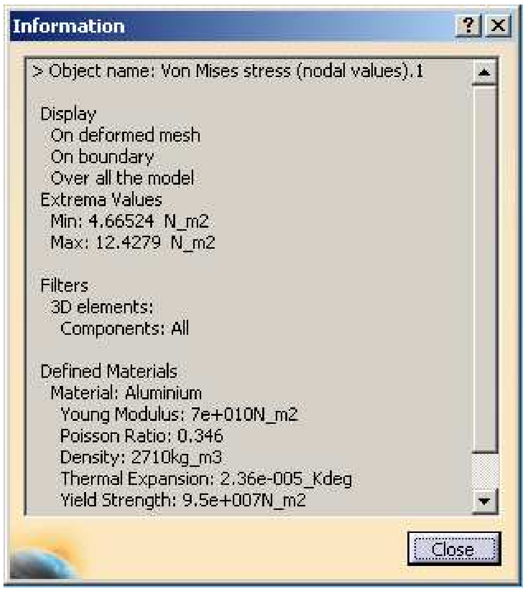
Instructor Notes:

About the Information Tool

For each image that you visualize, you have access to a specific Image Information box.

This tool groups many information about the images you have selected, like:

- **Material and its properties**
- **Extrema values**
- **Estimated Precision**
- **Strain Energy**
- **Estimated Error Rate**



This is particularly useful for the Von Mises and Precision images: it is the only way to know the yield strength of the materials in the part, and the global precision of your analysis.

Instructor Notes:

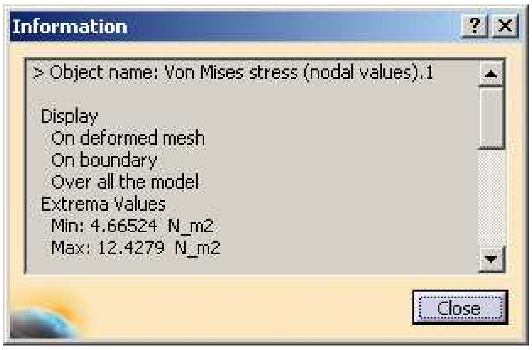
Using the Information Tool

This task shows how to use the Information tool.

1. **Visualize a result image of your analysis solution**

2. **Select an image results in the tree and click on the "information" icon**

Below an example of Von Mises information:



3. **Click "Close"**

Instructor Notes:

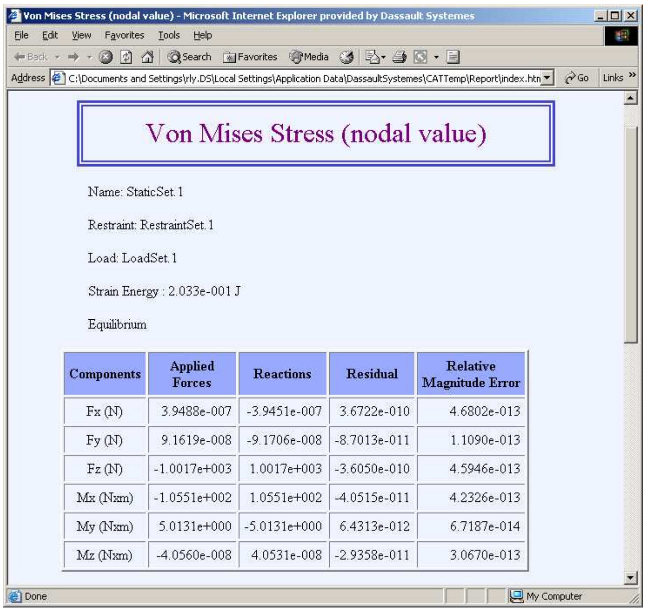
Additional Information

Type of Information	Type of Image		
	Deformed Mesh	Estimated Local Error	Any Other Type of Image
Object Name			
Display (On Boundary or all elements ; Over Local Selections or all the Model)			
Mesh Statistics (nodes and elements)			
Extrema Values (Min and Max)			
Surface elements vs Volume elements			
Process List (component, name, position)			
Used Materials (and Yield Strength)			
Precision Location			
Estimated Precision			
Strain Energy			
Global Estimated Error Rate			
Filters			

Instructor Notes:

Publishing Reports

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



Von Mises Stress (nodal value)

Name: StaticSet.1

Restraint: RestraintSet.1

Load: LoadSet.1

Strain Energy : 2.033e-001 J

Equilibrium

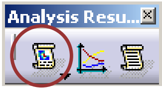
Components	Applied Forces	Reactions	Residual	Relative Magnitude Error
Fx (N)	3.9488e-007	-3.9451e-007	3.6722e-010	4.6802e-013
Fy (N)	9.1619e-008	-9.1706e-008	-8.7013e-011	1.1090e-013
Fz (N)	-1.0017e+003	1.0017e+003	-3.6050e-010	4.5946e-013
Mx (Nxm)	-1.0551e+002	1.0551e+002	-4.0515e-011	4.2326e-013
My (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

Instructor Notes:

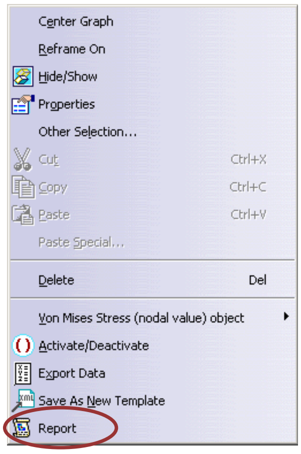
About Reports

A report is a summary of object set computation results and status messages, saved in an editable file.

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.

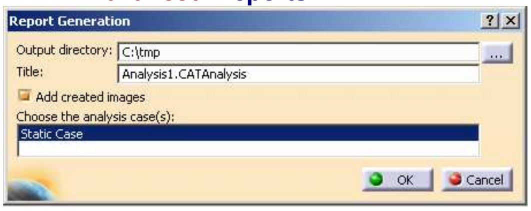
There are two ways to publish a report, either using the “Generate Report” icon 

or using the contextual menu.



There are two kinds of reports:

- **“Basic” reports**
- **“Advanced” reports**



Output directory: Pressing the button on the right gives you access to your file system for defining a path for the output Report file.

Title: You can modify the title if desired.

Add created images: Add automatically in the basic report the images created in the selected case. Choose the analysis case(s).

Instructor Notes:

How to Publish a Basic Report

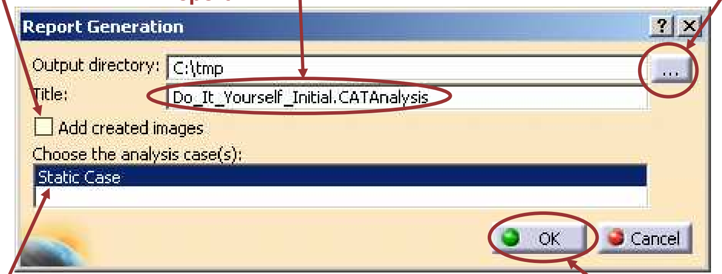
1 Select the result you want to publish

2 Click on the "Generate Report" icon

3 Click on the button to define the saving path

You can add the images created in post-processing

You can edit the title of the report



5 Choose an analysis case from the list

6 Click on OK

Instructor Notes:

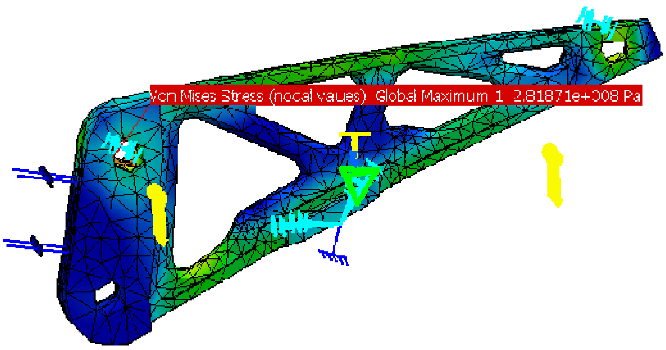
Exercise

'Results Management' Recap Exercise

10 min

In this exercise you will exploit the results you got in the previous recap exercise.
You will :

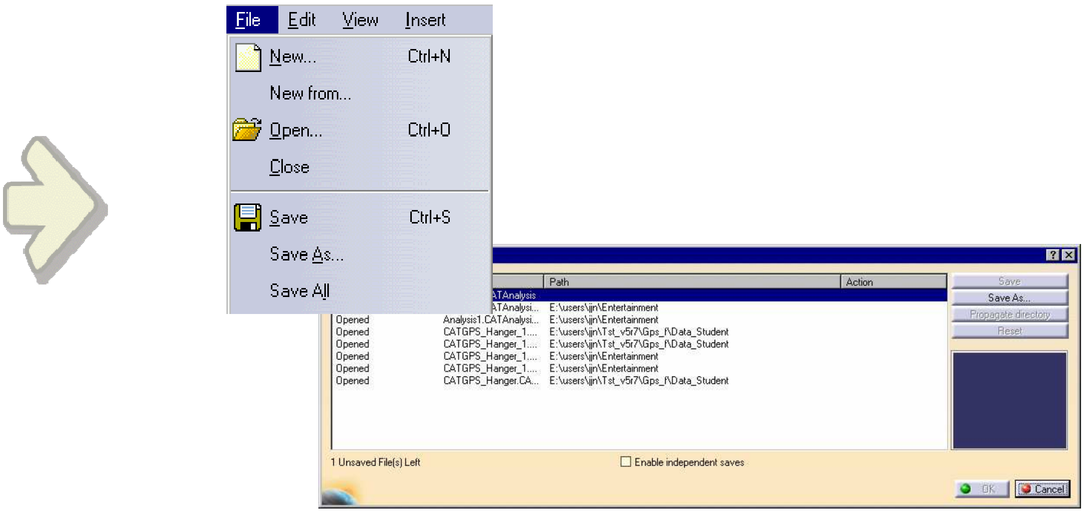
- Find two Stress Extrema
- Find one Displacement Extremum
- Publish a Report



Instructor Notes:

Managing Analysis Document

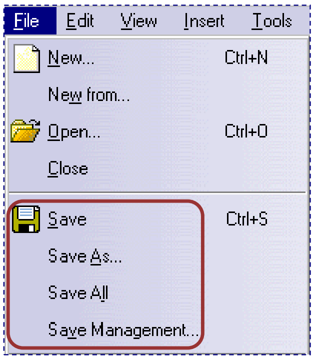
You will learn how to save an Analysis Document



Instructor Notes:

About Saving an Analysis Document

There are various ways to save an Analysis Document and child documents.



Save will save the active component's document and child documents of the active document

Save As... is similar to Save, but it allows you to specify the name and folder for the active document

Save All will save all the open documents that have been modified since last save

Save Management... will propose saving all open documents and children of these documents, but you can control the names and locations of all of them, and you can also preview them

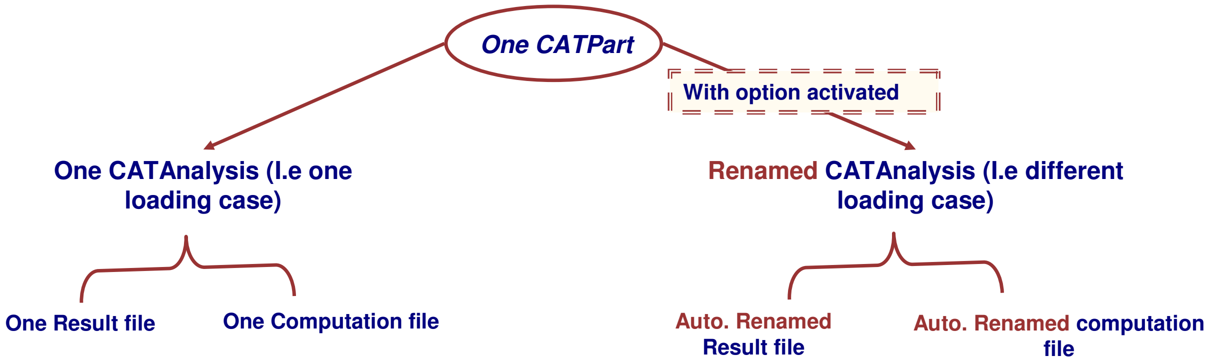
Only the documents that have been modified will be saved or proposed to be saved.

Instructor Notes:

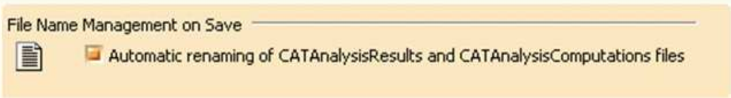
About Save As

Using a specific option, the “Save As” tool allows you to systematically apply the name of the CATAnalysis file on the results and computation files

This tool prevents from having several CATAnalysis documents linked to the same external storage files. Each time you rename the CATAnalysis file and you compute a new analysis, the results and computation files will be automatically renamed.



Option to check:

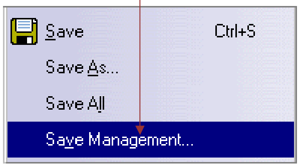


Instructor Notes:

How to Use Save Management

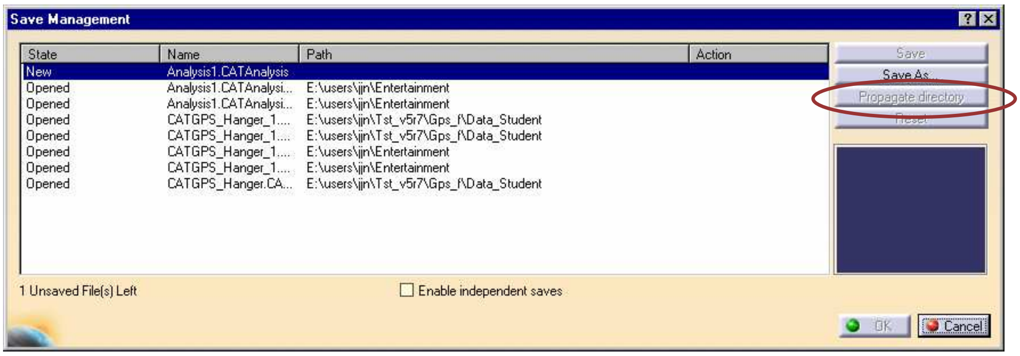
The “Save Management” tool is an easy way to save all modified documents under user-specified names.

1 Click on “Save Management”



All modified open documents will be proposed for saving, regardless of which document is active

2 Specify which documents to be saved



3 You can click on “Propagate directory” to save all the documents linked to the saved document

Instructor Notes:

Saving Document Using 'Send To' Mechanism

'Send To' is an easy way to save all linked documents in a user-specified directory

1. **Select File > 'Send To' > 'Directory'**

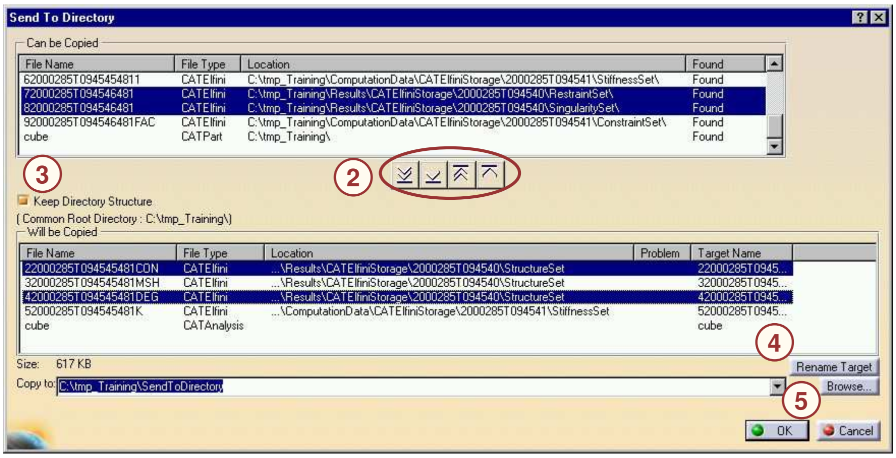
2. **Use these arrows to switch selected files between "Can be copied" and "Will be copied"**

3. **Use 'Keep Directory Structure' option if you want the directory structure of the selected file list to be duplicated.**

   Otherwise the selected files will be copied directly under the target directory.

4. **Click 'Rename Target' to modify the selected target name**

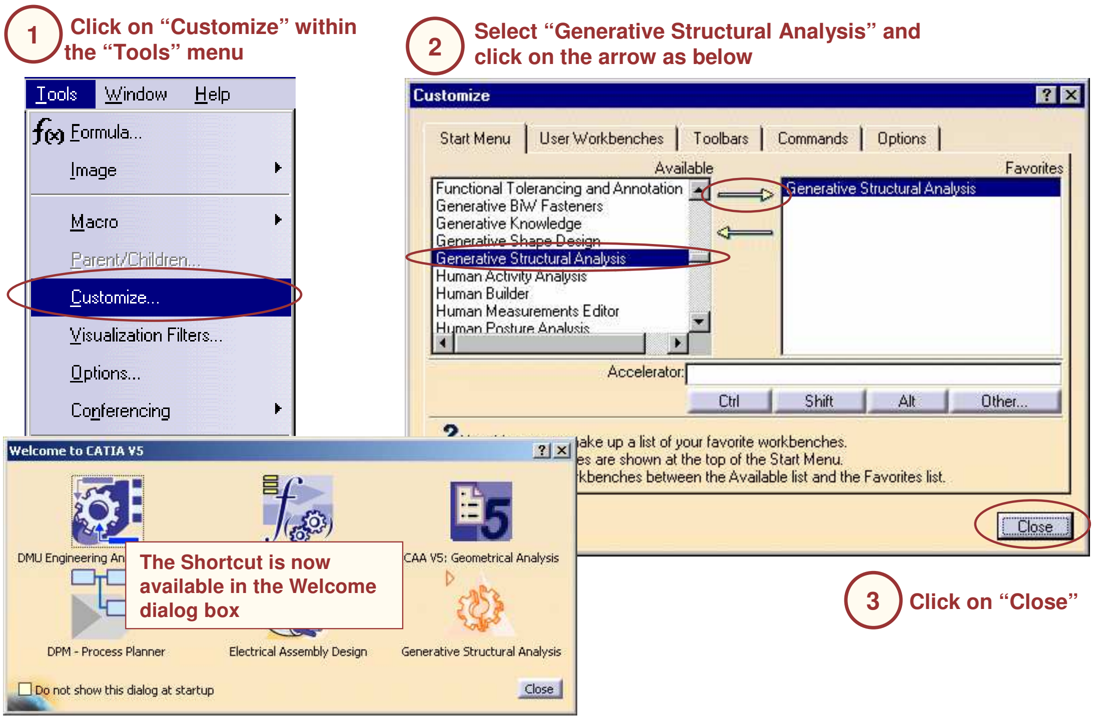
5. **Select the target directory using 'Browse' button**



Instructor Notes:

User Settings – How to Define a GPS Shortcut

This shortcut will give you a faster access to the GPS workbench

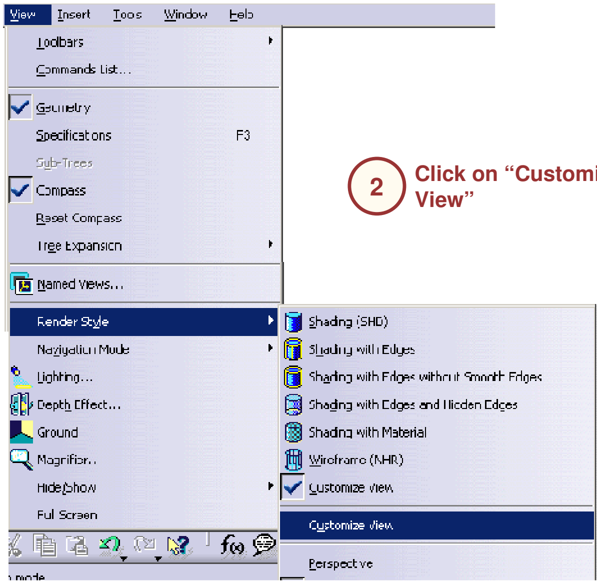


Instructor Notes:

User Settings - Customizing View Modes

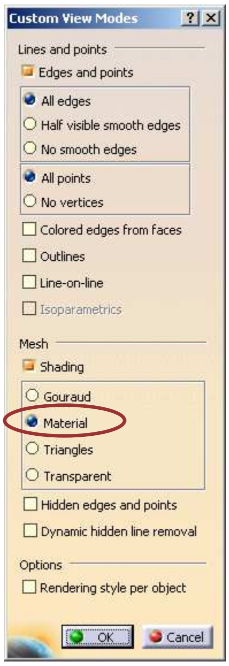
In the Part Design Workbench, customize the "Render Style" by adding materials visualization. This will allow you to view analysis images in average-iso visualization mode.

1 Select Render Style from the View menu



2 Click on "Customize View"

3 Check "Materials" and click on OK



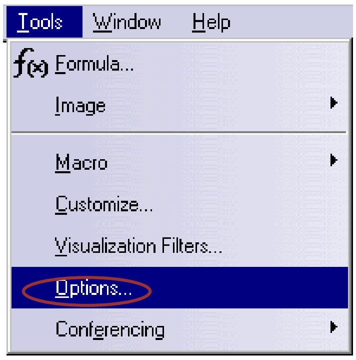
Instructor Notes:

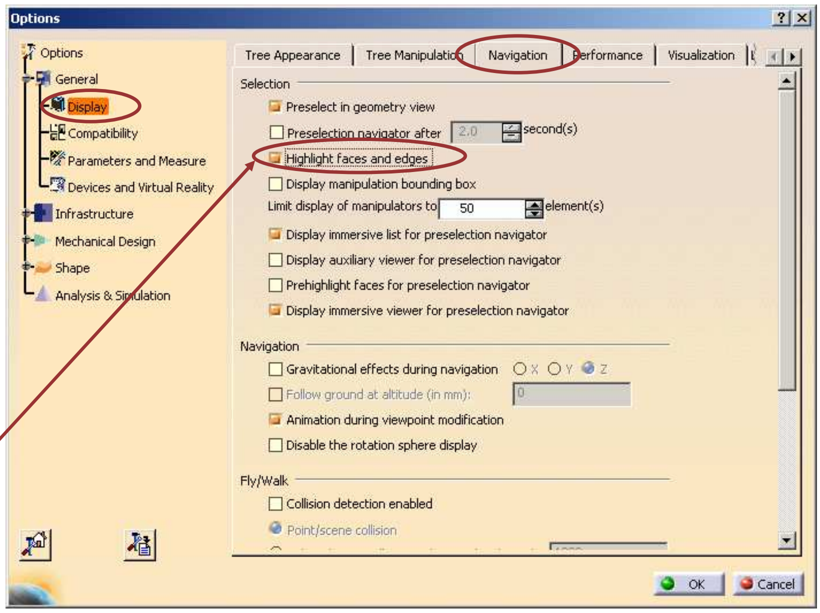
User Settings - Highlighting Faces and Edges

1 Select Tools\Options

2 Click on “Display”

3 Click on “Navigation” tab





4 Check the “Highlight faces and edges” options

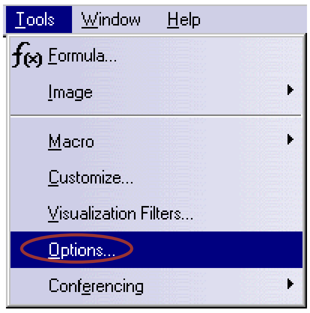
Instructor Notes:

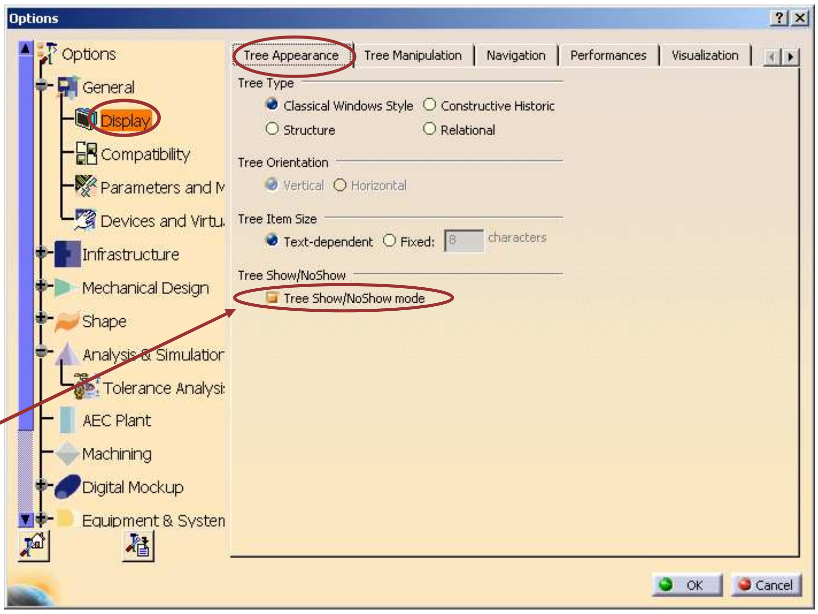
User Settings - Show / No Show Visualization

1 Select Tools\Options

2 Click on “Display”

3 Click on the “Tree Appearance” tab





4 Check Tree Show / No-Show Mode

Instructor Notes:

Master Exercise: Static Analysis

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

- Static Analysis: Presentation
- Static Analysis on a Hanger (1): Pre-Processing
- Static Analysis on a Hanger (2): Computation
- Static Analysis on a Hanger (3): Result Visualization

Instructor Notes:

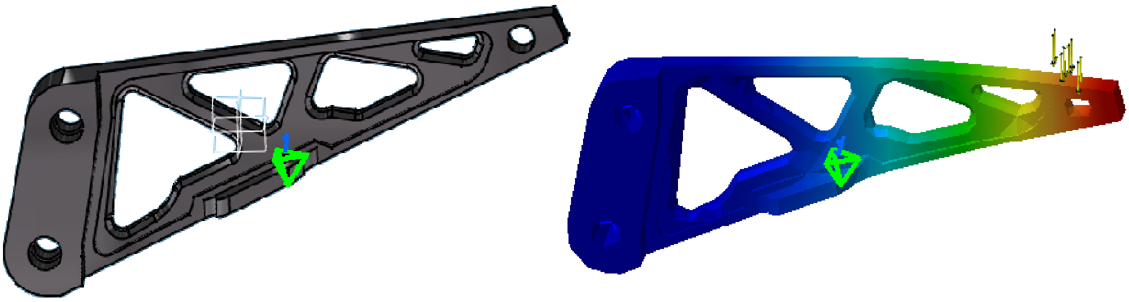
Exercise

Static Analysis: Presentation

50 min

In this exercise you will perform a static analysis of a hanger. It includes following steps:

- Define material, restraints and loads
- Define storage location and Compute the analysis
- Estimate computation error and publish a report.



Instructor Notes:

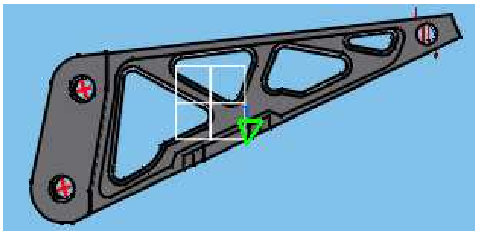
Exercise

Static Analysis on a Hanger (Step1): Pre-Processing

15 min

Objectives:

- Define the material
- Define the restraints
- Define the loads



Instructor Notes:

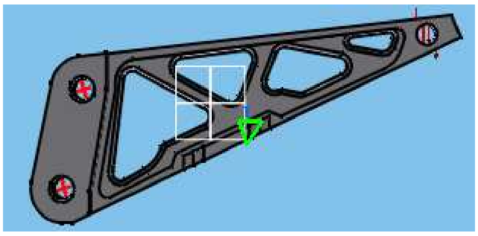
Exercise

Static Analysis on a Hanger (Step2): Computation

5 min

Objectives:

- Define the storage location
- Compute the analysis



Instructor Notes:

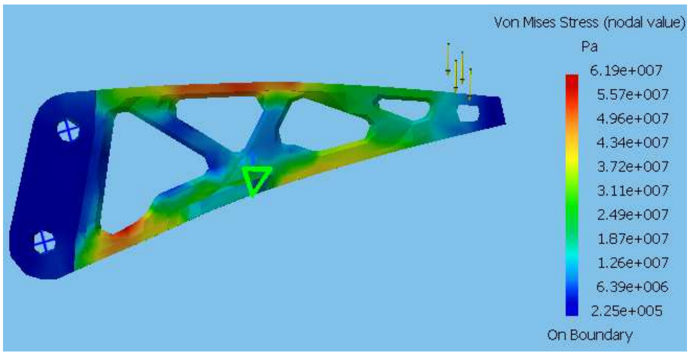
Exercise

Static Analysis on a Hanger (Step3): Result Visualization

30 min

Objectives:

- Estimate the computation error
- Detect extrema
- Define sensors
- Visualize the results
- Publish a report



Instructor Notes: