

# **ELFINI Structural Analysis**

Version 5 Release 16  
November 2006  
EDU-CAT-EN-EST-FI-V5R16

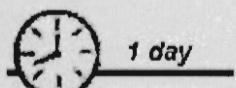
## **Course Presentation**

### **Objectives of the course**

In this course you will discover the advanced functionalities of **ELFINI Structural Analysis**

### **Targeted audience**

CATIA V5 Users



**Prerequisites**  
**Generative Part & Assembly**  
**Structural Analysis, FMS**

# Planning

|           | Day 1   |
|-----------|---|
| MORNING   | <b>Introduction to EST Analysis</b><br><b>Advanced Pre-Processing</b><br><b>Advanced Analysis Case Management</b><br><b>Advanced Post-Processing</b>  |
| Exercises | <b>Advanced Analysis of Landing Gear Assembly</b><br><b>Bearing load Analysis</b>   |
| AFTERNOON |   |
| Exercises | <b>Hanger Combined case and Buckling Analysis</b><br><b>Door Fuselage Buckling Analysis</b><br><b>Pressure Contact between Rod and Axis</b><br><b>Blade Analysis</b><br><b>Truss analysis</b><br><b>Smoby Toy analysis</b><br><b>Analysis using Adaptivity specifications</b><br><b>Analysis Automation</b><br><b>Composite Analysis</b><br><b>Static Constrained Modes Case</b><br><b>Applying Mapping Property</b><br><b>Transfer of Loads</b><br><b>Transfer of Solution</b><br><b>MultiLoads Analysis</b> |

## Table of Contents (1/2)

|  |    |
|--|----|
| • <b>Introduction to EST Analysis</b>      | 6  |
| • <b>Advanced Pre-Processing</b>           | 8  |
| ◆ 2.1. Visualization Transferred onto Mesh | 9  |
| ◆ 2.2. Bearing Loads                       | 12 |
| ◆ 2.3. Data Mapping                        | 14 |
| ◆ 2.4. Self-Balancing on Load set          | 16 |
| ◆ 2.5. Combined Loads/Masses               | 17 |
| ◆ 2.6. Thermo-mechanical loads             | 20 |
| ◆ 2.7. Local 1D Property                   | 22 |
| ◆ 2.8. Variable cross section beams        | 23 |
| ◆ 2.9. Advanced 2D properties              | 26 |
| ◆ 2.10. Periodic conditions                | 31 |
| ◆ 2.11. Grouping for Pre-processing        | 32 |
| ◆ 2.12. Local Adaptivity Specifications    | 34 |
| ◆ 2.13. Mapping Property                   | 35 |
| ◆ 2.14. Import V4                          | 37 |
| • <b>Advanced Analysis Case Management</b> | 39 |
| ◆ 3.1. Multi Analysis Case                 | 40 |
| ◆ 3.2. The Buckling Case                   | 43 |
| ◆ 3.3. Contact Pressures                   | 46 |
| ◆ 3.4. Frequency shifting                  | 48 |
| ◆ 3.5. Combined Case                       | 49 |
| ◆ 3.6. Static constrained modes Case       | 50 |

## Table of Contents (2/2)

|  |           |
|--|-----------|
| • 3.7. Envelop Case                              | 51        |
| • 3.8. Pre-processing Case                       | 55        |
| • 3.9. Multi Loads Case                          | 58        |
| • 3.10. Transfer of Computed Loads/Displacements | 62        |
| ● <b>Advanced Post-Processing</b>                | <b>72</b> |
| • 4.1. Advanced Customisation                    | 73        |
| • 4.2. Advanced Report                           | 77        |
| • 4.3. Advanced images                           | 83        |
| • 4.4. Export of nodal values                    | 85        |
| • 4.5. Grouping                                  | 86        |
| • 4.6. Local sensors                             | 87        |
| • 4.7. Image template                            | 88        |
| • 4.8. FEM model images                          | 89        |
| ● <b>EST Functionalities</b>                     | <b>90</b> |
| ● <b>Frequently Asked Questions</b>              | <b>94</b> |

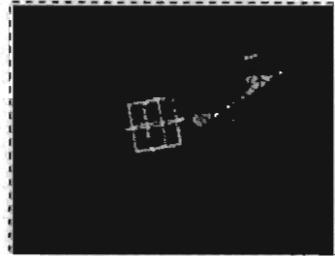
## 1. Introduction to EST Analysis



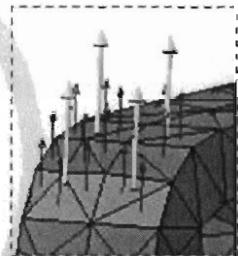
*In this lesson, you will learn about the ELFINI Structural Analysis Workbench. The ELFINI Structural Analysis Workbench is entirely integrated both in GPS and GAS Workbenches.*

## General Process

Starting from a Part or an Assembly, open Structural Analysis Workbench

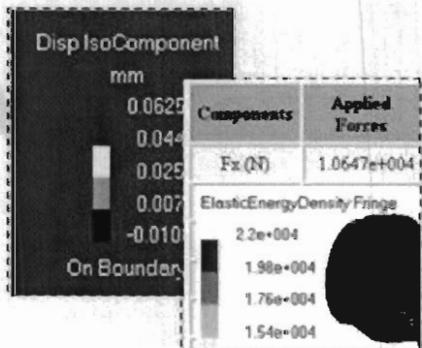


1  
Advanced Pre-Processing  
Visualize Pre-Processing specifications directly on Mesh

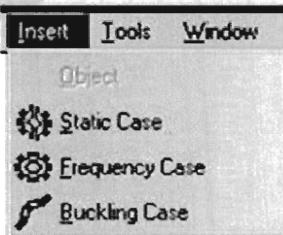


3

Advanced Post-Processing  
Customize image appearance  
Generate Advanced Reports



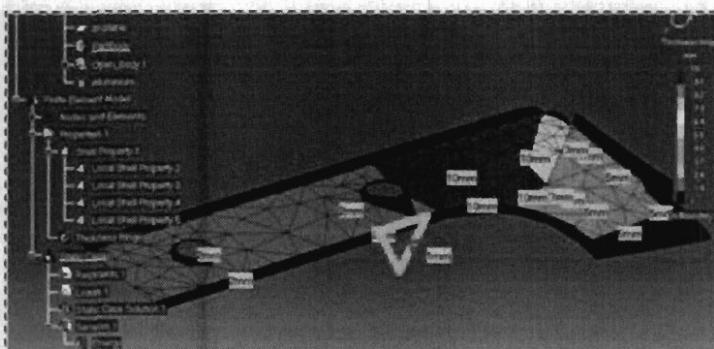
2  
Advanced Computing  
Perform Computation with Multi Analysis Cases, including Buckling



## 2. Advanced Pre-Processing

In this lesson, you have the possibility to visualize the way Pre-Processing Elements are transferred onto the Mesh. Moreover, you will see different type of loads and pre-processing features only available in EST.

- Visualization Transferred onto Mesh
- Bearing Loads
- Data Mapping
- Advanced shell properties
- Periodic conditions
- Thermo-mechanical loads
- Self-Balancing on load set



## 2.1. Visualization Transferred onto Mesh (1/2)



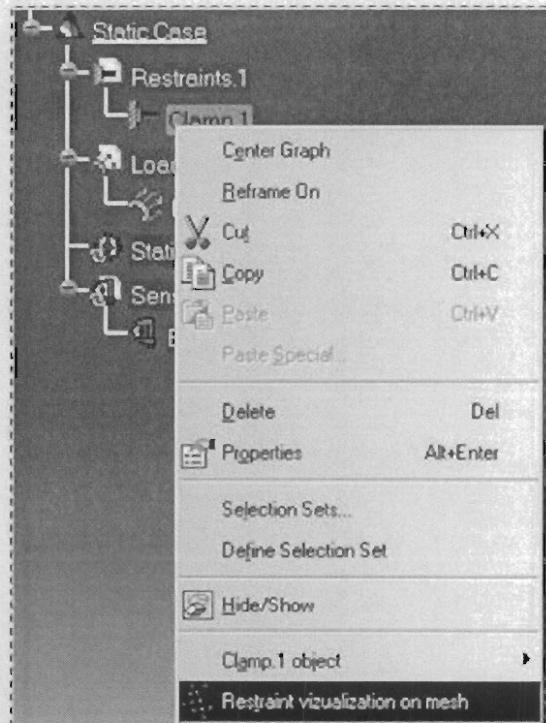
All pre-processing elements (restraints, loads, mass equipment) can be visualized on Mesh.

- 1 Create pre-processing elements.

- 2 Compute Mesh Only, which transfers pre-processing data onto the mesh (loads symbols have to appear yellow).

- 3 Display contextual menu with a right mouse click on an existing pre-processing element (in the tree or directly on the model).

- 4 Click on "Restraint Visualization on Mesh" icon.



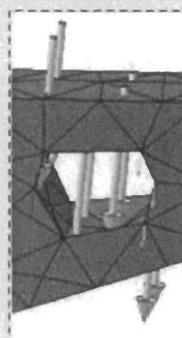
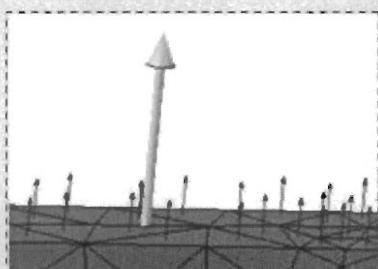
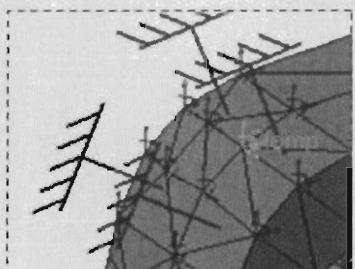
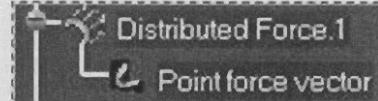
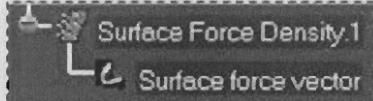
## Visualization Transferred onto Mesh (2/2)



Once you clicked on the visualization on mesh symbol, the mesh is visualized on your model, as well as the way the pre-processing element is applied onto it.



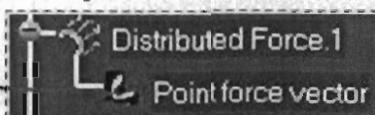
Examples :



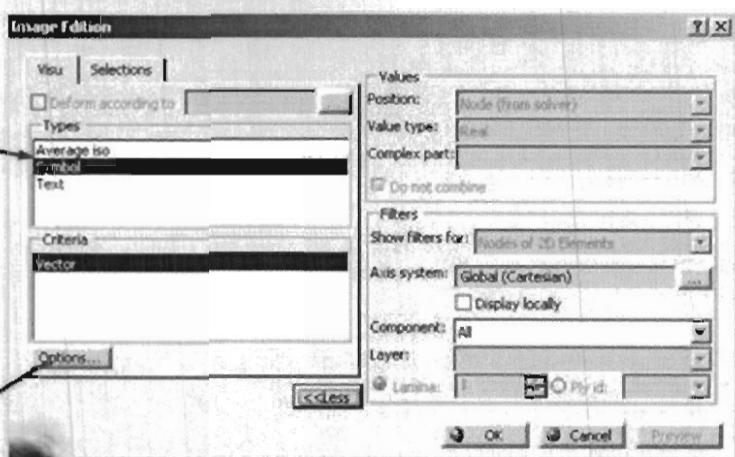
# Visualization Transferred onto Mesh - Customization

Pre-processing visualization on mesh symbols can also be customized.

- 1 Double click on the visualization symbol, either on the tree or directly on the model.



- 2 Customize the visualization (choose the visualization type, customize symbol appearance)



- 3 Click OK.



## 2.2. Bearing Loads (1/2)

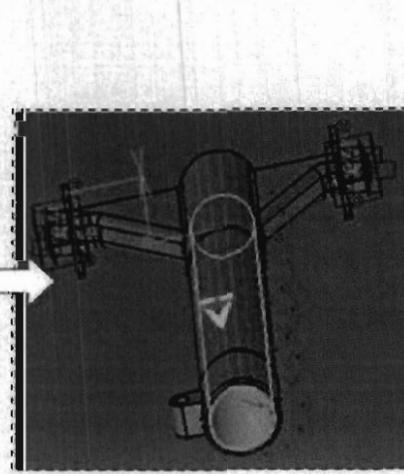
Bearing loads create contact-wise force only on revolution surfaces and simulate the action of a part on another one without modeling it.

Eccentric loads are always perpendicular to the cylinder axis.

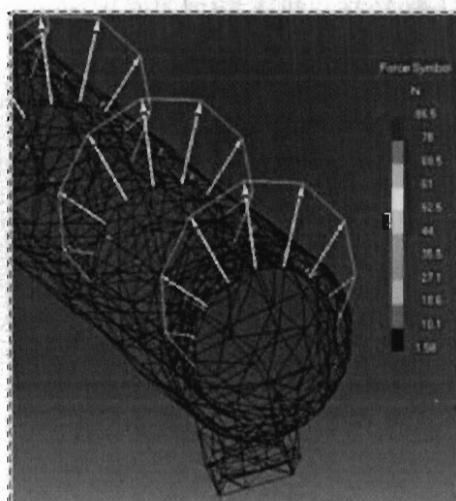
- 1 Click on "Bearing Load" icon.



- 2 Select only a revolution surface-like support and enter a resultant effort.



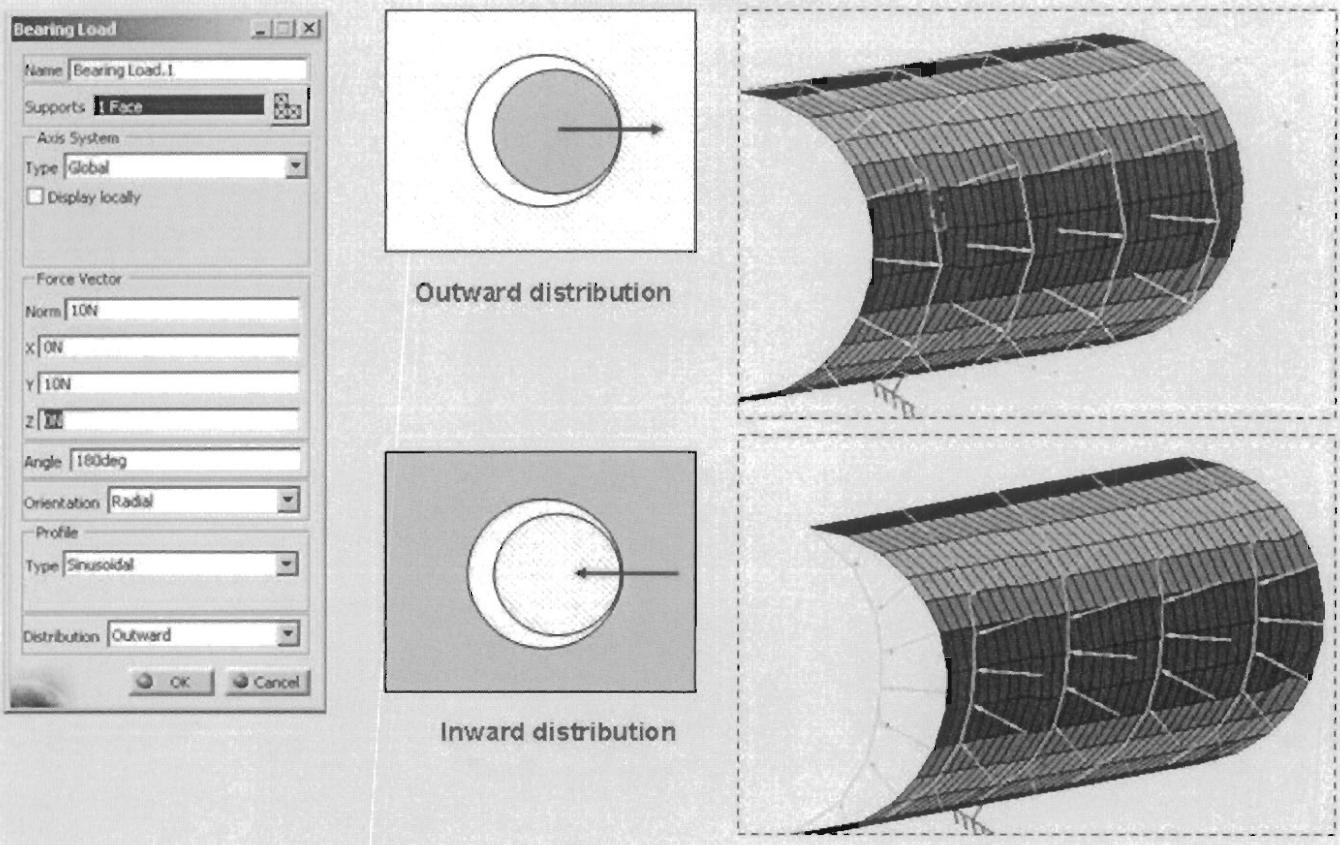
Visualization of force distribution on each node after the computation.



## Bearing Loads (2/2) - Extended bearing load capabilities

Inward distribution of bearing loads.

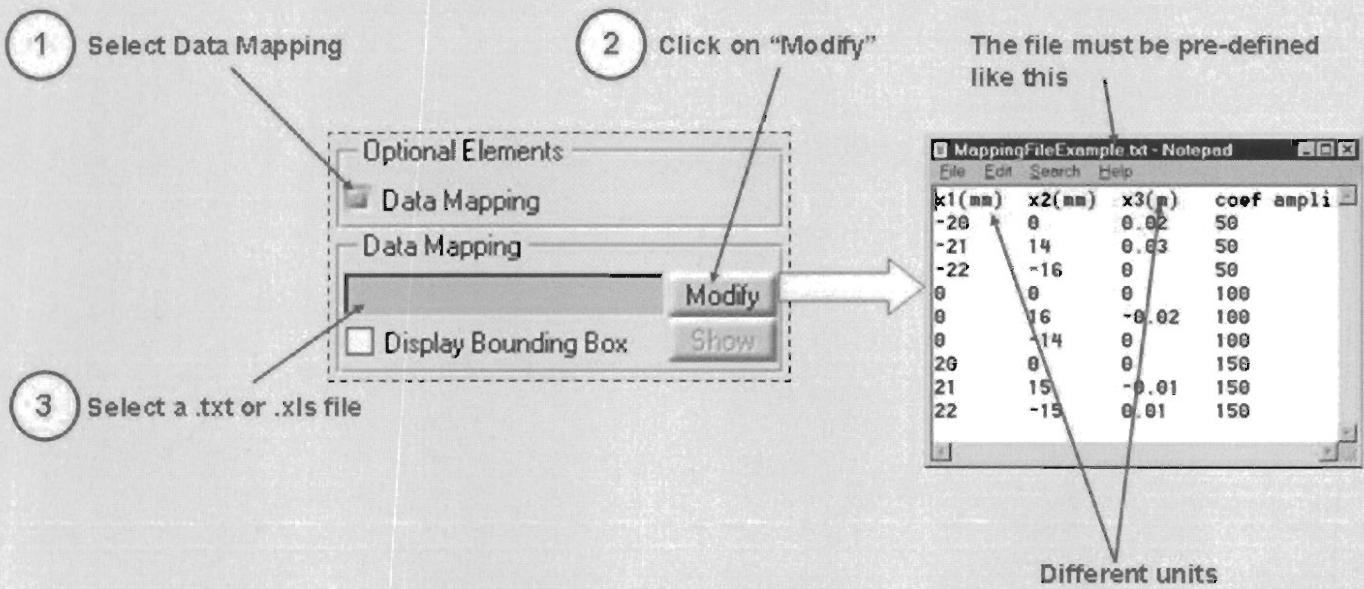
Except for the force orientation, the "inward" bearing load specifications are similar to the "outward" bearing load



### 2.3. Data Mapping : General

You can re-use data that is external from CATIA V5 (experimental data or data coming from in-house codes or procedures).

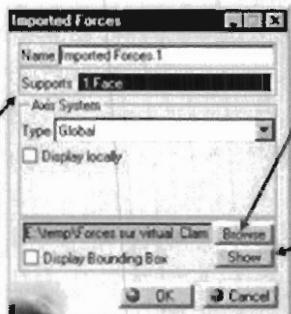
- This data mapping is available for pressure, line loads, surface loads, volume loads, thermo-mechanical loads, thickness property, and forces/moment.
- The selected external data file will be either a .txt file (columns separated using the Tab key) or a .xls file with a pre-defined format (pressure : four columns, the first three columns letting you specify X, Y and Z points coordinates in the global axis and the last one containing the amplification coefficient).



# Data Mapping - Forces and Moments

You can import forces or moments data with the standard data-mapping format (excel or text file). Complete data transfer is also possible thanks to Image export data.

- 1 Click on the Forces or Moments import icon.



- 2 Select support (surface, edge or point).

- 3 Click Browse, and select the import file.

- 4 Click Show, and visualize the import file content

Imported Forces can be applied either on surfaces or virtual parts.

For each point in the data file, the corresponding force is distributed on the three closest nodes of the selected support.

Perform accurate analysis with Export/Import capabilities :

- 1 Analysis of a complete structure with coarse mesh.

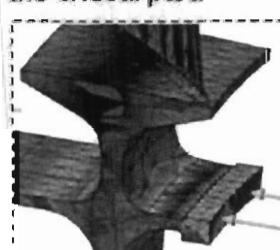


- 2 Export Nodal forces at the connection.

- 3 Import the forces on the connection surface of the single part



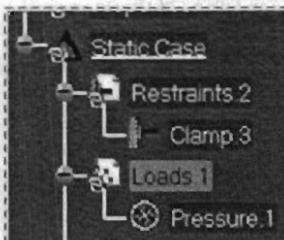
- 4 Perform refined mesh on the critical part.



## 2.4. Self-balancing on Load set

- Self-balancing is an attribute available on a load set meaning that the resultant at the center of inertia is null.

- 1 Double-click on the Loads icon in the tree.



- 2 Select the Self Balancing option, and choose Yes.

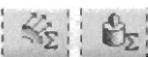


**Result :** load is “self-balancing” so as to have a null resultant at the centre of inertia.

| Components | Applied Forces | Reactions   | Residual     | Relative Magnitude Error |
|------------|----------------|-------------|--------------|--------------------------|
| Fx (N)     | -3.8947e-018   | 1.4566e-014 | -1.4570e-014 | 4.0079e-013              |
| Fy (N)     | 4.6057e-020    | 3.0388e-014 | -3.0388e-014 | 8.3592e-013              |
| Fz (N)     | -1.1693e-016   | 6.4507e-014 | 6.4390e-014  | 1.7713e-012              |
| Mx (Nm)    | 3.3881e-020    | 1.0960e-015 | 1.0960e-013  | 1.7228e-013              |
| My (Nm)    | 3.5865e-017    | 8.8284e-015 | 8.8642e-013  | 1.3934e-012              |
| Mz (Nm)    | 5.248e-020     | 5.0428e-015 | 3.0429e-015  | 7.9270e-013              |

Used with iso-static restraint, you can simulate free-body loading.

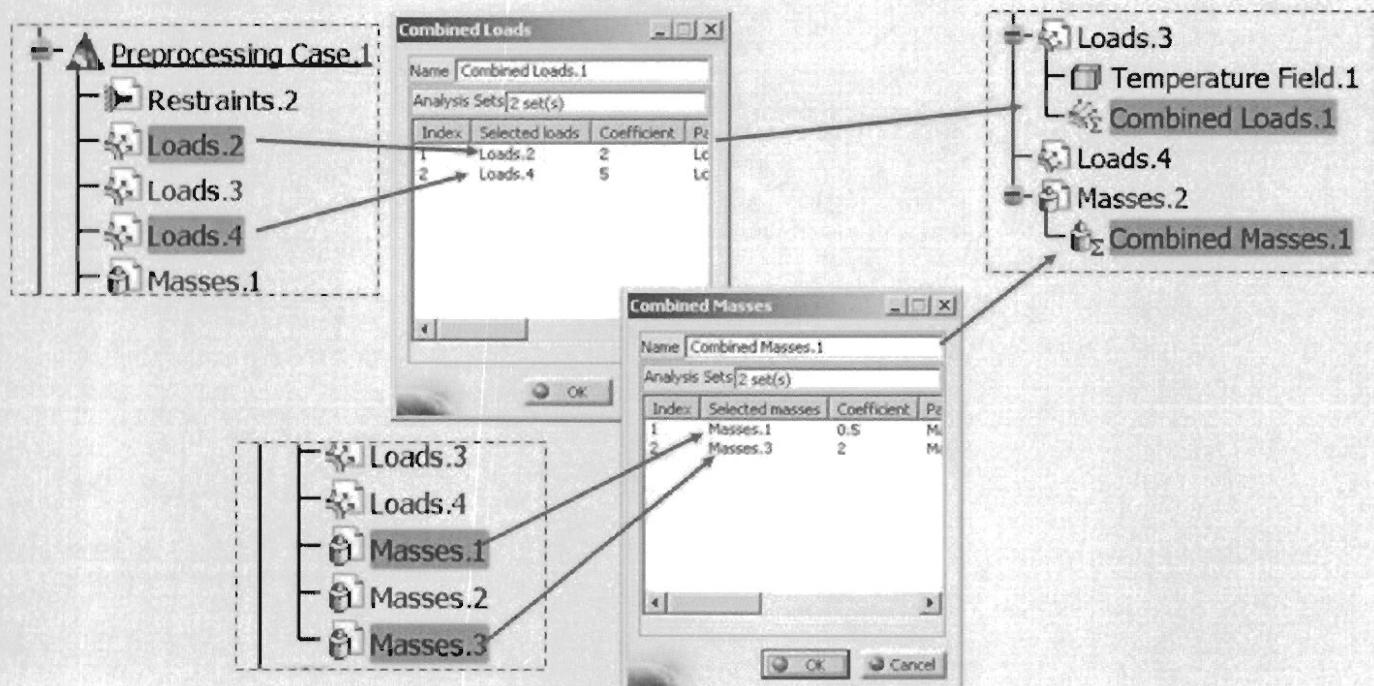
## 2.5. Combined Loads/Masses



V5R16

You can form linear combination of existing loads/masses to reuse combination of load and mass specifications in different computation cases. Loads/Masses can belong to different analysis cases.

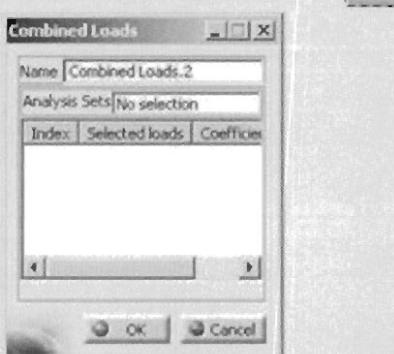
Combination describes any linear combination, ' $\alpha_1 A_1 + \alpha_2 A_2 + \alpha_3 A_3 + \dots$ ' where  $A_i$  are loads or masses and  $\alpha_i$  are the combination coefficients. Unit loads/masses can be defined once and then reused to define a lot of derived load/mass. Combining loads and masses will improve pre-processing tools.



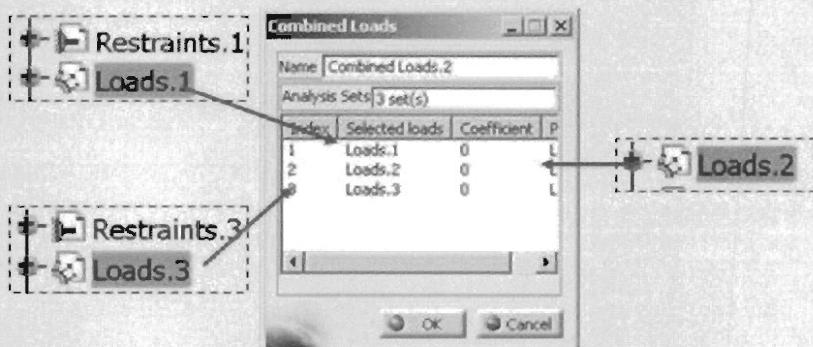
### How to Create Combined Loads/Masses (1/2)

You can form linear combination of existing loads/masses to reuse loads or masses specifications in different computation cases.

- 1 Click on Combined Loads icon in Advanced Loads Toolbar



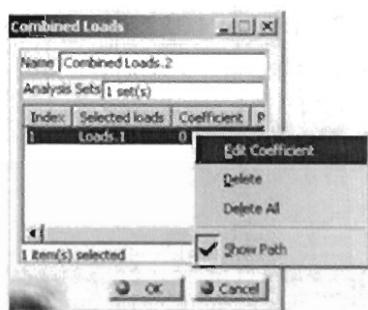
- 2 Select the required Load Set from pre-processing Load sets



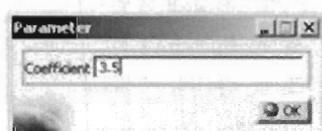
It is possible to select Loads using Search functionality.

## How to Create Combined Loads/Masses (2/2)

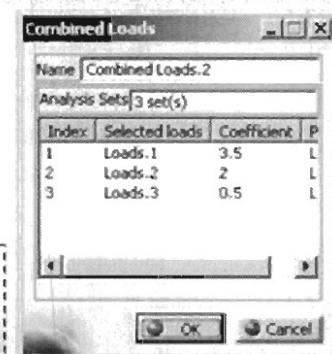
- 3 Select the Loads set in and click on Edit Coefficient in contextual menu



- 4 Enter the required coefficient and click OK



- 5 Similarly Edit Coefficient for other selected Load Sets and click OK



In similar way you can create Combined Mass.

## 2.6. Thermo-mechanical loads

- You can apply a temperature field loading on your model, available on solids and surfaces of bodies. Data mapping is also allowed.

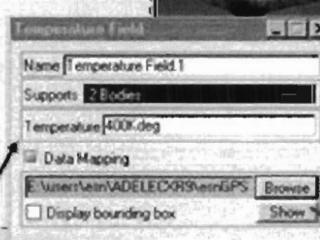
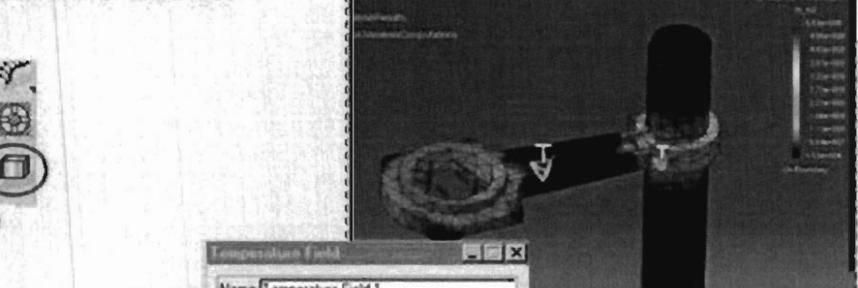
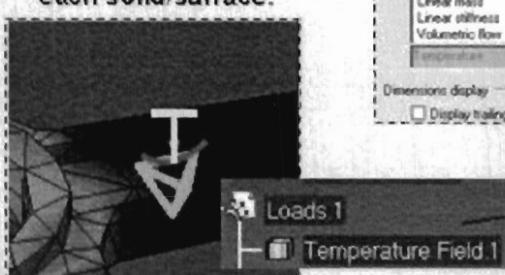
- 1 Click on temperature field icon.



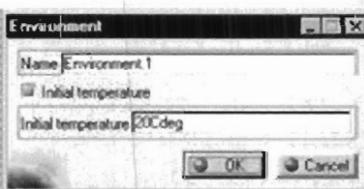
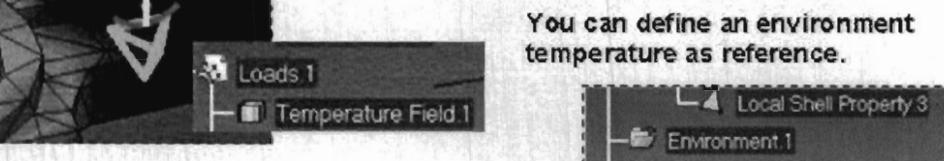
- 2 Select the body (on geometry or in the tree). Enter a temperature value.

NB : You can change temperature unit in Tools/Options/General.

A Temperature field symbol is generated on each solid/surface.



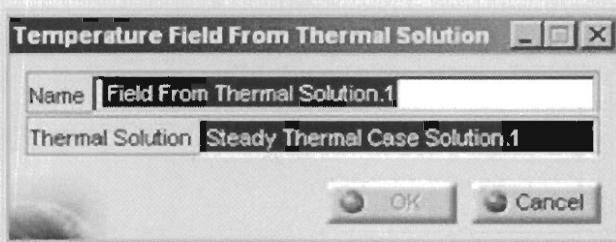
| X(mm)   | Y(mm)     | Z(mm)    | Coeff        |
|---------|-----------|----------|--------------|
| 25.8437 | 4998.71   | -1611.06 | 1.35638e+006 |
| 25.8437 | 4998.71   | -2210.77 | 290588       |
| 22.815  | 4738.73   | -2410.77 | 131087       |
| 11.5168 | 4054.2    | -2410.77 | 221459       |
| 3.81585 | 3546.73   | -2139.55 | 572509       |
| 3950.41 | -2.83764  | -2352.19 | 0            |
| 26.4237 | 1554.01   | -2111.06 | 550868       |
| 4024.5  | 4879.44   | -2352.19 | 0            |
| 35.6839 | 943.778   | -2410.77 | 195921       |
| 45.982  | 199.26    | -2410.77 | 104215       |
| 50.0166 | -0.716432 | -2210.77 | 297330       |
| 50.0166 | -0.716432 | -1611.06 | 1.34609e-006 |



You can define an environment temperature as reference.

# Temperature Field From Thermal Solution

- New Temperature loading option
- Added Value:
  - Enhance interoperability with third party products



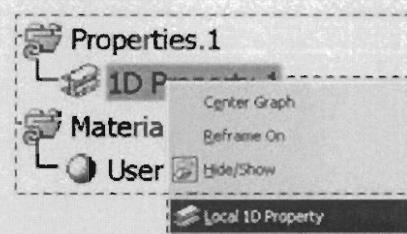
## 2.7. Local 1D Property

- You are now able to overwrite global 1D properties on local geometry.
  - You can have multiple properties management in one single part.
  - Beam modeling is easier.

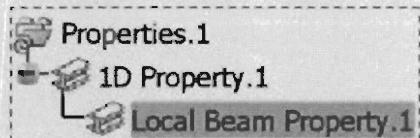
1 Mesh a wire-frame feature, and apply a 1D property.



2 Double-click on the property icon.



3 Select a support (line, curve). Choose a section and an orientation point.

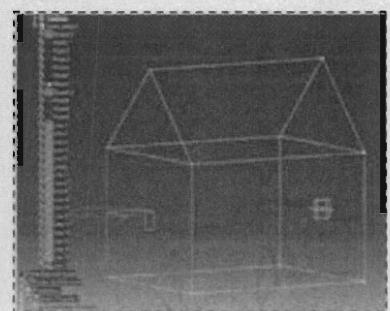


### Beam modelling methodology :

1 Create wire-frame geometry. Use *Join* command to create a single wire-frame feature.

2 Create a beam mesh-part and a global 1D property.

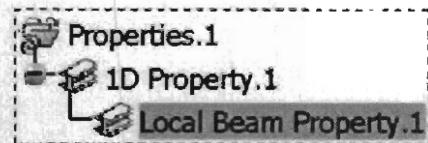
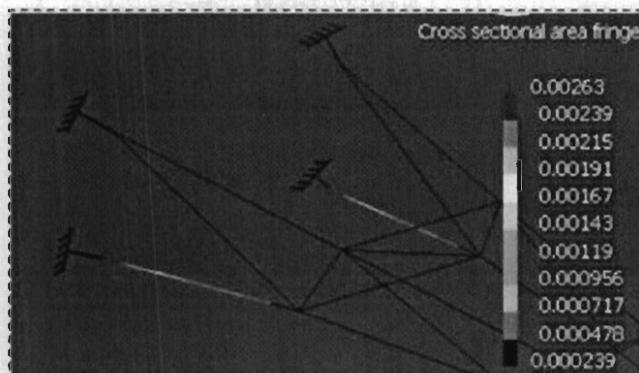
3 Overwrite this default property on local lines or curves.



## 2.8. Variable Cross Section Beams

- Variable beam factors: lets you create a linear approximation of variable cross section beams.

Activate 'variable beam factors' option. Two new fields appear in the Beam Property dialog box. The Multiplication Factors on extremities frame will let you give a scaling factor on each side of the section. The beam will then be modeled as a sequence of constant section beams with linearly decreasing dimensions.



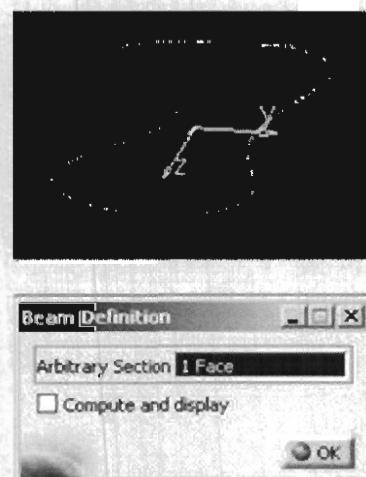
## Beam Properties From Arbitrary Cross-section

- New "Beam From Surface" option
- Added Value:
  - ◆ Provide automatic computation of section parameters
  - ◆ Extends the beam property definition to any profile

1 New "Computed From Surface" item



2 Select the section of the beam

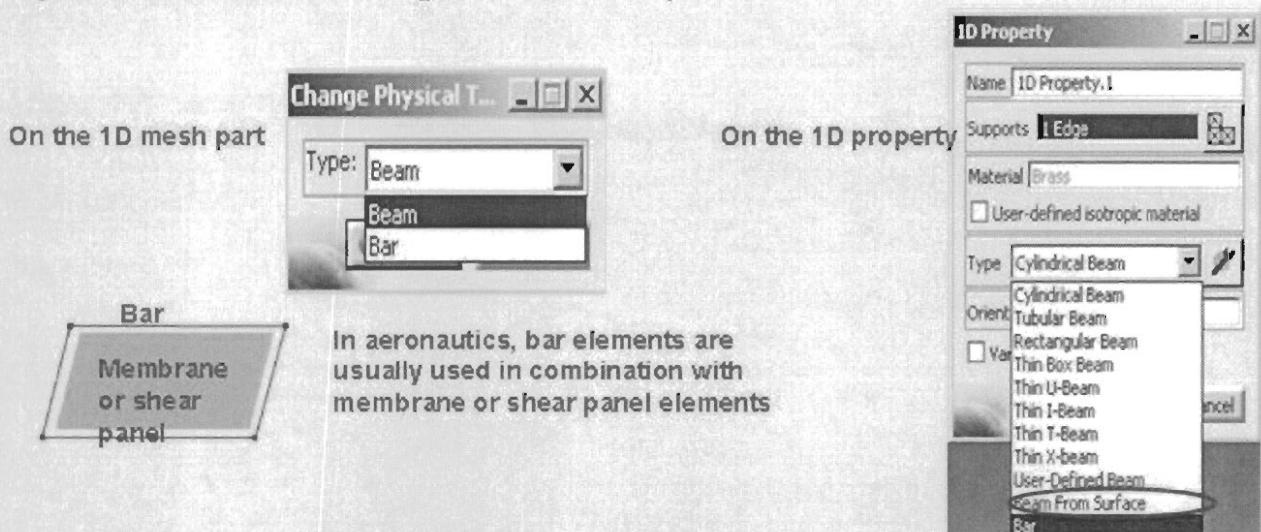


3 Characteristics of the beam

| Computed values   |                          |
|-------------------|--------------------------|
| Beam values       | Axis values              |
| Area              | 10.00m <sup>2</sup>      |
| Shear center (y)  | -7.204mm                 |
| Shear center (z)  | 2.432mm                  |
| Shear factor (xy) | 0.767979603              |
| Shear factor (xz) | 0.760001242              |
| Inertia (x)       | 5.651e-006m <sup>4</sup> |
| Inertia (y)       | 1.048e-005m <sup>4</sup> |
| Inertia (z)       | 2.381e-006m <sup>4</sup> |

# 1D Properties for Vehicle Conceptual Model

- A new type of 1D elements: BAR
  - Bar elements only have stiffness along their axis
  - They have three translation degrees of freedom per node



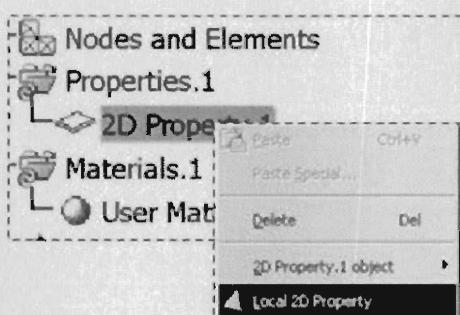
- Dedicated to the analysis of vehicle conceptual model

## 2.9. Advanced 2D Properties (1/5)

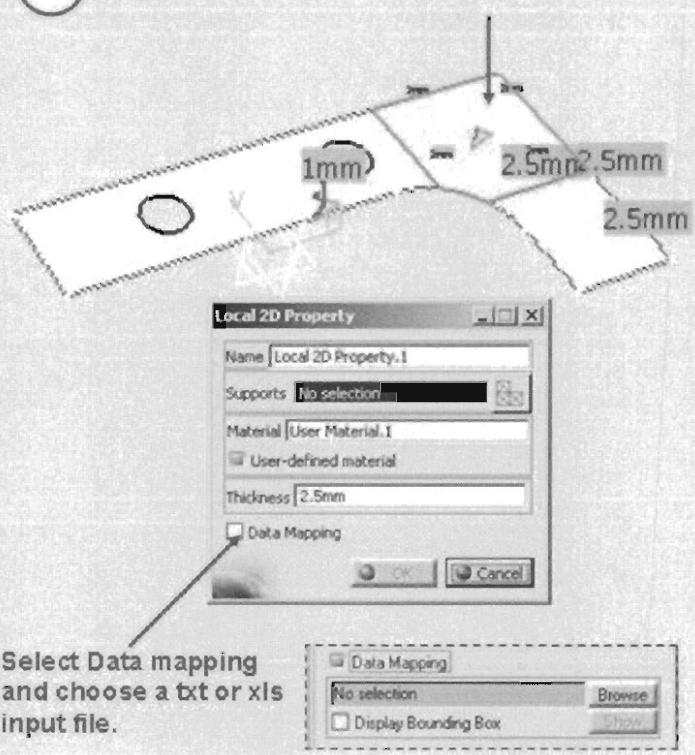
You can edit the 2D property icon and create several Local 2D properties associated to local geometry.

Available through contextual menu (right click) on the 2D property icon

- 1 Contextual menu on the 2D property icon. Select *Create local 2D Property*
- 2 Select a face. Enter a new thickness value

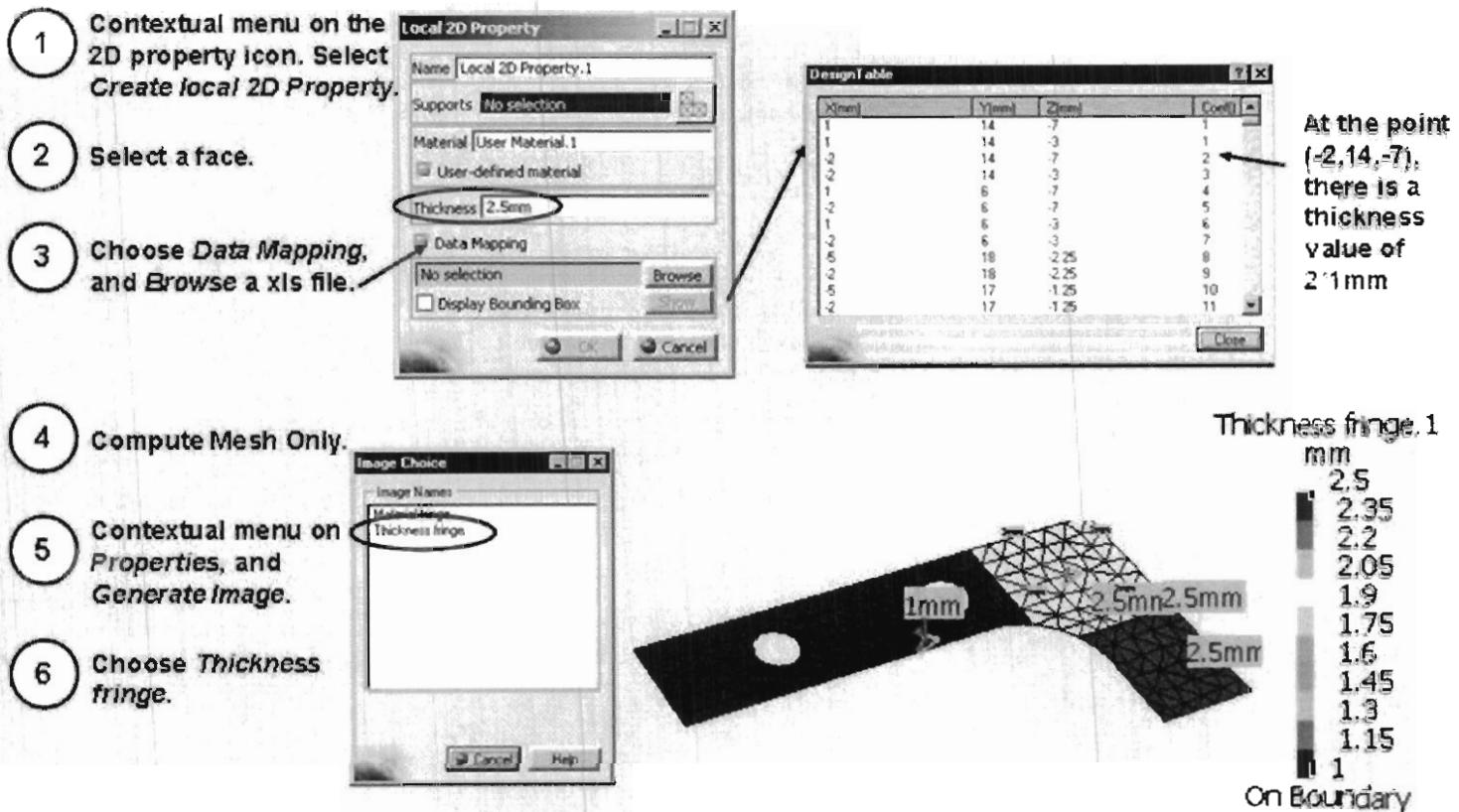


- Create as many local 2D properties as there are different thickness in the geometry.
- The structure can now be analyzed more precisely.



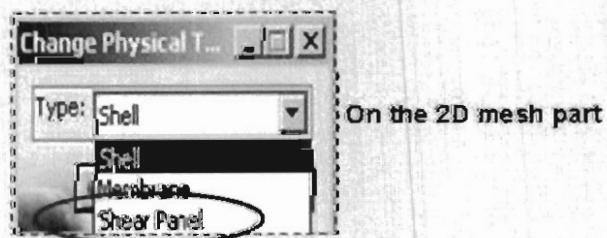
## Advanced 2D properties (2/5)

- You can import 2D properties from an external file, and apply them locally or globally.
- There is no need of compatible grid and node identification.
- specifications.

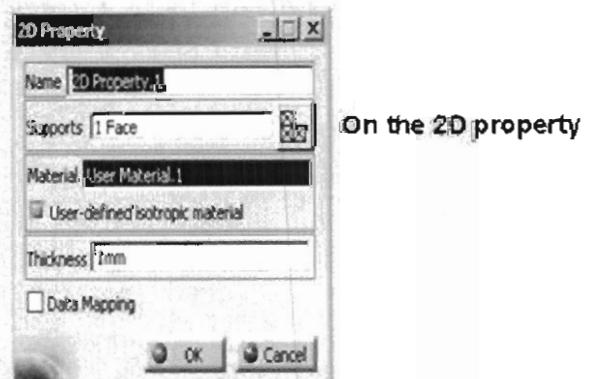


## Advanced 2D properties (3/5)

- 2D properties for vehicle conceptual model
- Two new types of 2D elements:
  - Membrane
  - Shear Panel

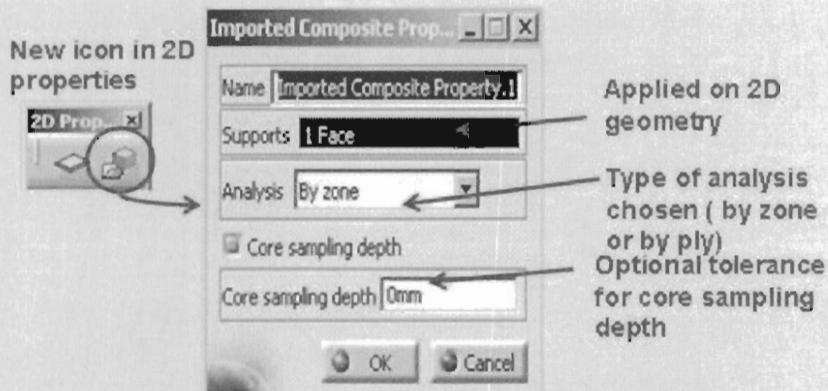


Membrane and Shear panel are 2D elements with no transversal stiffness

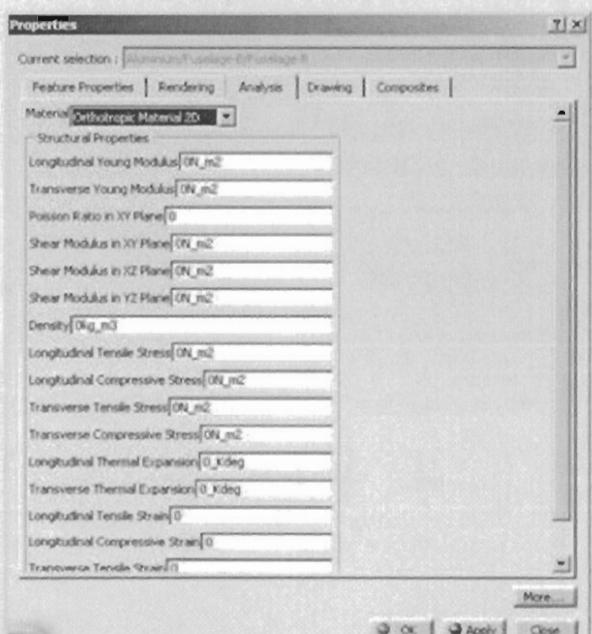


## Advanced 2D properties (4/5)

- Composite FE model generation from the design by Plies/Zones
- Two types of composites analysis :
  - By zone for preliminary design
  - By ply for detailed analysis



The orthotropic material is now available on 2D geometries

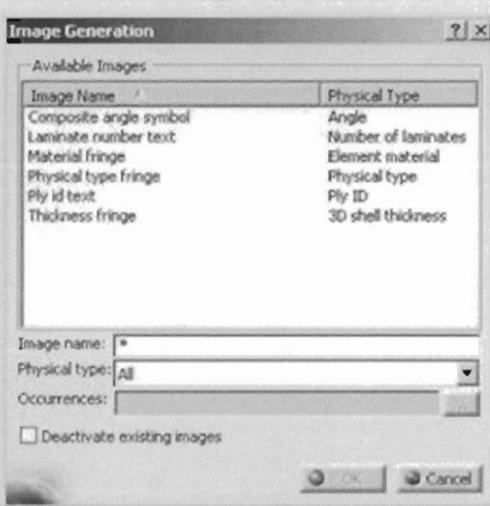


- FE Composite properties automatically generated from Design definition
- Full Support of FE simulation on composite models (pre-processing, solving, post-processing)

## Advanced 2D properties (5/5)

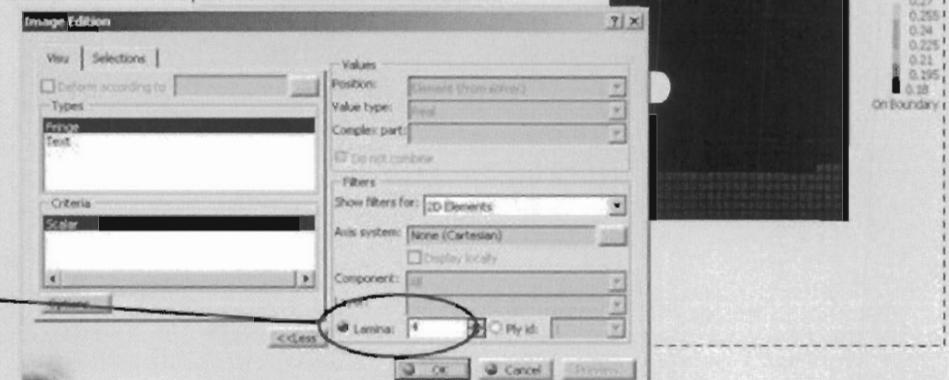
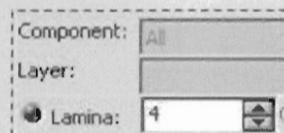
- Composite FE model generation from the design by Plies/Zones

1 "Imported Composite Property" icon, select the part.



2 Launch a "Mesh Only" computation

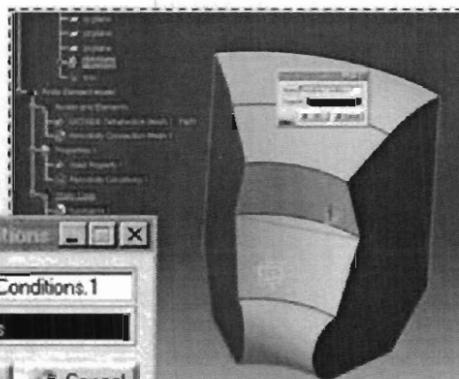
3 Display Composite properties images



## 2.10. Periodic conditions

- With Cyclic Symmetry conditions, you don't have to analyze all your model, but only a symmetric sub-part of it. You can define infinite (translation with vector) or finite (rotation with vector and angle) cyclic symmetry.

1 Click on Periodic Conditions icon.



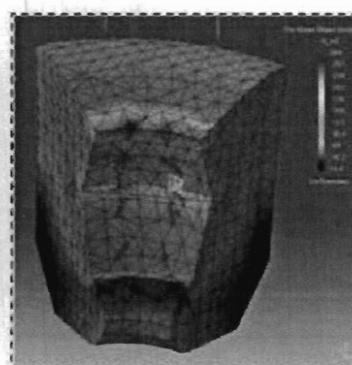
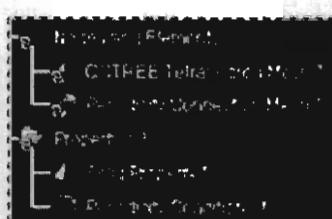
2 Select the two faces of your sub-part.

NB : These faces must be planar and identical, but their meshes can be different (incompatible mesh is taken into account).

3 Compute.

A new mesh-part and a new property are created.

New type of MPC elements (linear combination between the DOF of the nodes faces) are generated between the 2 faces.



## 2.11. Grouping for Pre-processing

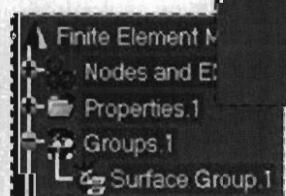
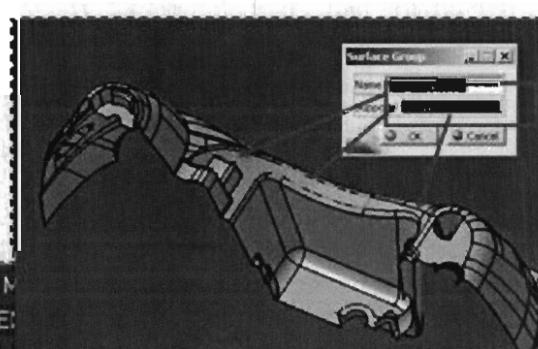
- You can create groups of geometrical entities so as to apply pre-processing conditions only on the associated nodes or elements.

1 Click a Group icon.

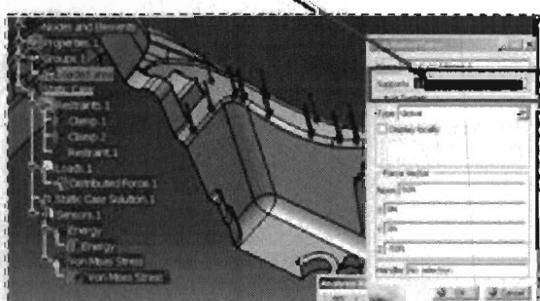


2 Select the support (several surfaces for example).

A new feature set is created in the FEM tree.

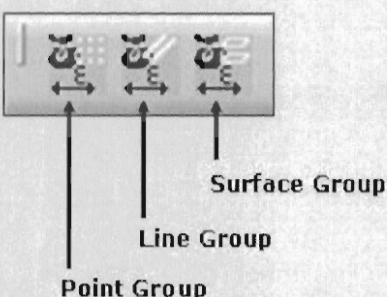


3 Select any pre-processing conditions: restraint, load and choose the group as support.



# FE Groups by Neighborhood

- Provides associative capturing of mesh entities
- Added Value
  - Allows pre-processing on non-associative mesh
    - Extruded meshparts
    - Imported orphan mesh



## 2.12. Local Adaptivity Specifications

- The adaptivity method implemented is the H-method. At constant element order, the mesh is selectively refined (decrease element size) in such a way as to obtain a desired results accuracy. The mesh refining criteria are based on a technique called predictive error estimation, which consists of determining the distribution of a local error estimate field for a given Static Analysis Case. As a result, the use of the adaptivity method makes it possible to reduce the memory costs and the time costs. The Adaptivity functionalities are only available with static analysis solution or a combined solution that references a static analysis solution

1 Global Adaptivity must have been defined

2 Right-click Global Adaptivity.1 object in specification tree, select Local Adaptivity contextual menu

3

**Local Adaptivity**

Local Adaptivity dialog box:

|   |
|---|
| Name: Local Adaptivity.1                  |
| Supports: 2 Faces                         |
| Solution: Static Case Solution.1          |
| <input type="checkbox"/> Exclude elements |
| Objective Error (%): 0                    |
| Current Error (%): 25.8709                |

**Local Adaptivity**

**Name:** lets you change the name of the local adaptivity.

**Supports:** List of selected elements. Multi-selection is available: support can be one or several vertices, edges, faces or group (except body group).

This multi-selection may be non-homogeneous (ie. you can select two edges and three faces, as example).

**Solution:** gives you inform information on the referenced solution.

**Exclude elements:** with this option, selected elements will not have an objective error and then, the Objective error (%) field will disappear. In this case, elements will not be taken into account in the re-meshing algorithm.

**Objective error (%):** objective error of selected mesh part. This option is available only if Exclude elements is deactivated.

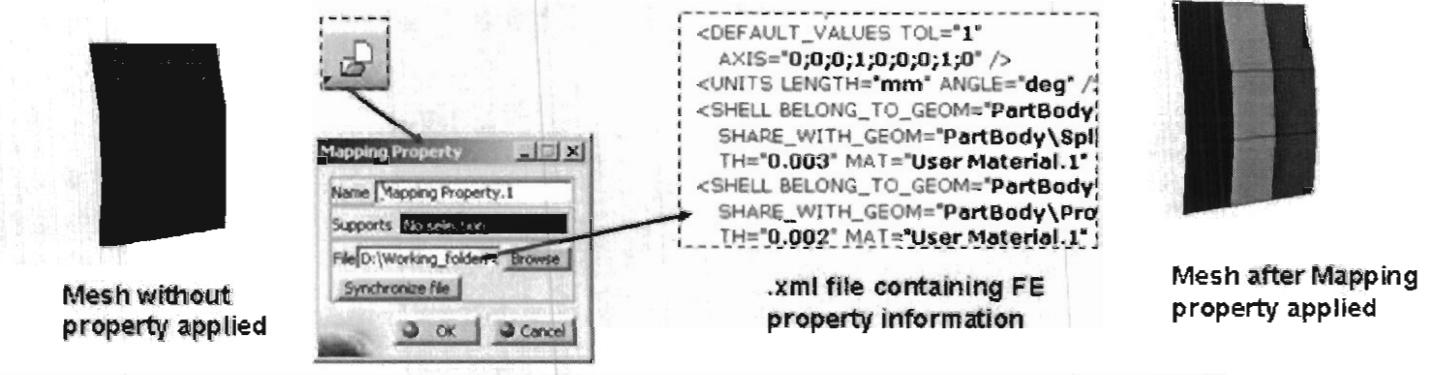
**Current error (%):** gives you information on the current error of the selected support.

## 2.13. Mapping Property (1/2)

'Mapped Property' allows applying in one shot different Finite Element properties on a set of mesh parts which in Part document. It handles 1D, 2D and 3D properties including 2D composite properties.

The feature is based on a file generated by the user. The file contains identification information of element and the associated property values which includes thickness, materials, beam/bar characteristics, tolerance and axes.

This is very useful for structure where mesh properties are variable e.g. aircraft fuselage that have different thickness, material. You can in future modify the property file and reapply the properties again.



Mesh after Mapping property applied



You can now Overload Mapping Property to improve the use of the existing "Mapping Property". Once a "Mapping Property" has been applied on the finite element model, any modification of its characteristics will be applied faster on the model without reapplying the previous unchanged properties specifications.

V5R16

## Mapping Property (2/2)

There are two approaches for the element identification:

- Spatial identification based on x, y, z coordinates of the element gravity center



Here element location is defined using geometric co-ordinates.

```
<BEAM X="-8.10300748817977" Y="0" Z="34.9843281048414" AREA="3.5" INERTIA="2e-8;1e-  
8;1e-8" SHEAR_RATIO_XY="1.2" SHEAR_RATIO_XZ="1.2" SHEAR_CENTER="0;0" MAT="User  
Material.2" />  
<BEAM X="-0.558817890118186" Y="0" Z="9.98824596038694" AREA="4" INERTIA="2e-8;1e-  
8;1e-8" SHEAR_RATIO_XY="1.2" SHEAR_RATIO_XZ="1.2" SHEAR_CENTER="0;0" MAT="User  
Material.1" />
```

- Associative identification based on mesh part/group names and geometrical feature names



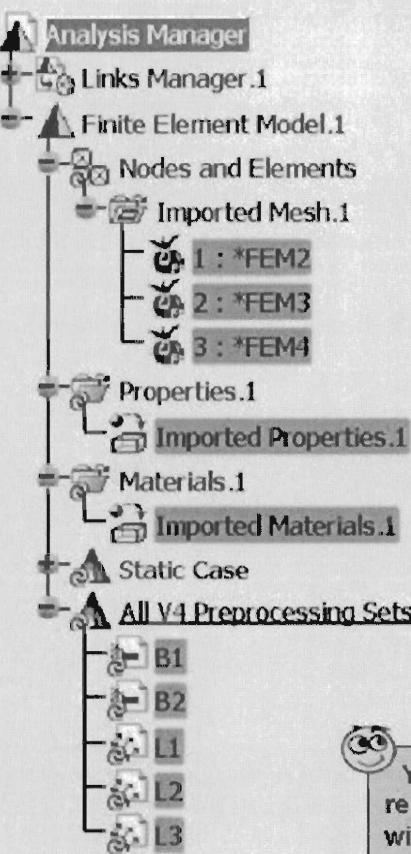
Element location is defined with the help of geometry surrounding the element

```
<SHELL BELONG_TO_GEOM="PartBody\Extrude.1"  
SHARE_WITH_GEOM="Ribs\Project.4;Spars\Intersect.4;Ribs\Intersect.1;Spars\Intersect.3"  
TH="1" MAT="User Material.1" />  
<SHELL BELONG_TO_GEOM="PartBody\Extrude.1"  
SHARE_WITH_GEOM="Ribs\Intersect.1;Spars\Intersect.4;Ribs\Intersect.2;Spars\Intersect.3"  
TH="1" MAT="User Material.1" />  
<SHELL BELONG_TO_GEOM="PartBody\Extrude.1"  
SHARE_WITH_GEOM="Ribs\Intersect.2;Spars\Intersect.4;Ribs\Project.3;Spars\Intersect.3"  
TH="1" MAT="User Material.1" />
```

These two approaches can be mixed however the solution will not wholly associative. Each time the property is updated (standard compute command) the whole file is parsed and properties applied to the elements

## 2.14. Import V4

Import V4 provides access to V4 Finite Element data for V5 analysis and analysis of assembly.  
Import is based on a result view only.



The following V4 entities will be transferred:

- FEM Model  
Mesh, Selections, Elements, Materials, Properties, axis systems, groups
- Preprocessing  
Restraints, Loads, Non Structural-masses

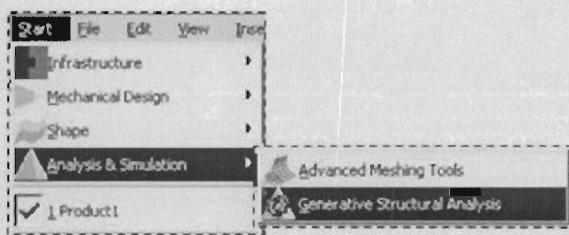
All V4 transferred data will be created and stored on CATAnalysis file.

V5R16

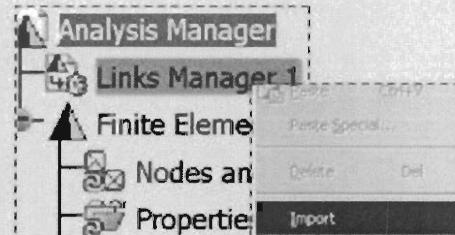
You can now import V4 Displacements as "orphan" results and can perform post-processing on V4 solutions without having to rebuild and compute equivalent cases.

## Import V4

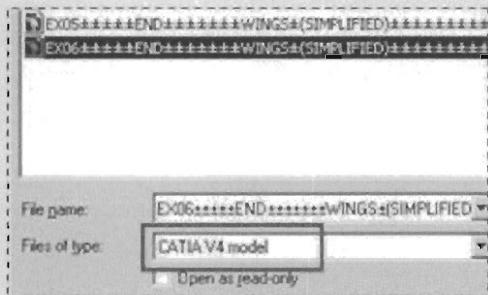
- 1 Open new Analysis document in Analysis & Simulation Workbench



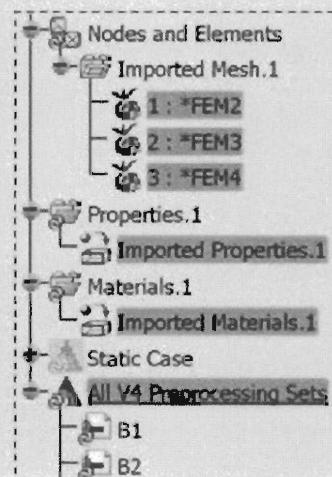
- 2 Go to Links Manager Contextual Menu and click Import



- 3 Select CATIAV4 model file



- 4 Expand the tree.



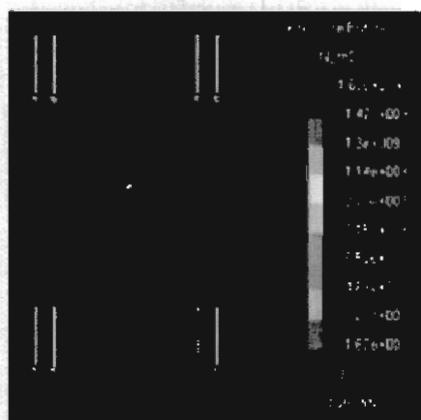
You will get FE entities imported

# 3. Advanced Analysis Case Management

In this lesson, you can see that the Finite Element model can contain an arbitrary number of Analysis Cases.

If you do not have the CATIA – ELFINI Structural Analysis product license, your Finite Element model can simultaneously contain at most one Static Analysis Case and one Frequency Analysis Case.

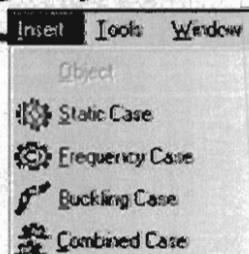
- Multi Analysis Case
- The Buckling Case
- Contact Pressures
- Frequency shifting
- Combined case



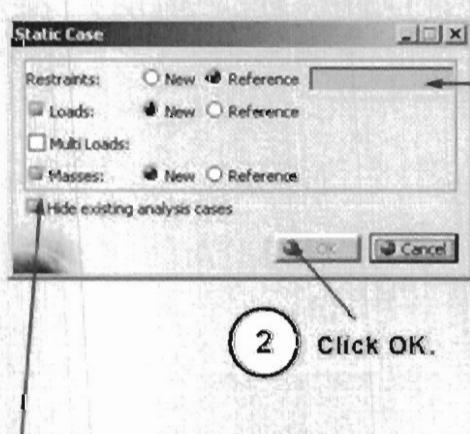
## 3.1. Multi Analysis Case - Creation

- You can insert any type of analysis case to an existing analysis document.
  - Each pre-processing element can be defined either from scratch, or by replicating an pre-processing element of an existing analysis case.

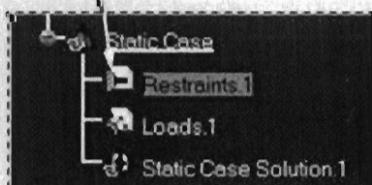
1 In the Insert menu, choose one of the three proposed analysis case types.



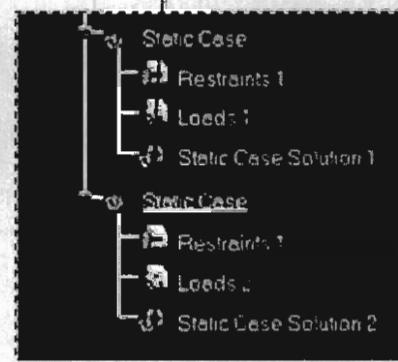
The corresponding case definition box is displayed, containing the available pre-processing element types.



Select an existing pre-processing element if necessary.



Newly created case is defined as current.

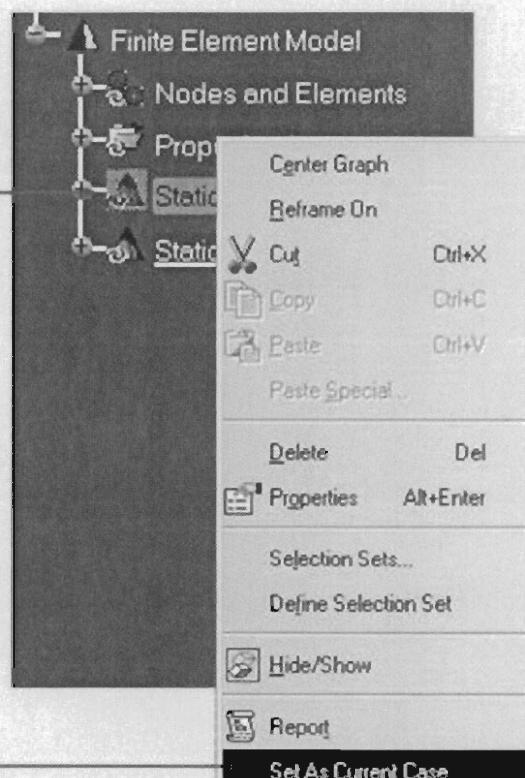


Activate bottom button if you want the symbols of previous analysis cases to disappear from mesh.

## Multi Analysis Case - Switching

- All the analysis cases you have created appear in the analysis tree.
  - Only one case is active at a time.
  - The active case appears underlined in the features tree. To make another case the current one, proceed as shown.

- Right click on an existing (inactive) analysis case, the contextual menu is displayed.



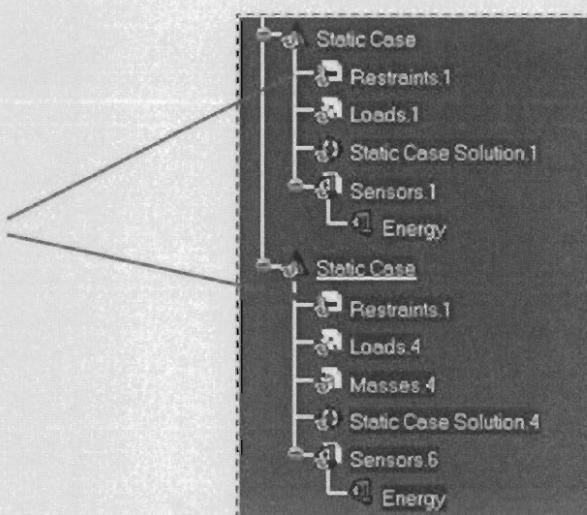
- Select option "Set As Current Case".

## Multi Analysis Case - Smart Update

- If you have several analysis cases in your analysis document that share a common restraint set, then the restraint is factorized only once.

- Define two different analysis cases based on the same restraint set

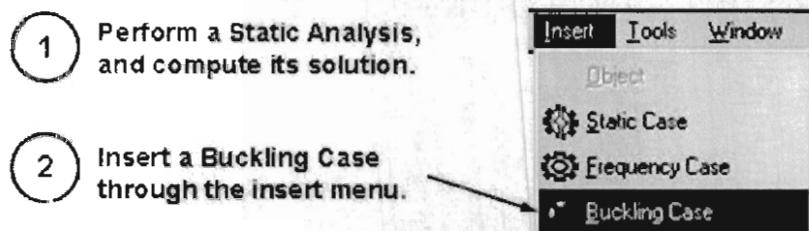
- Compute



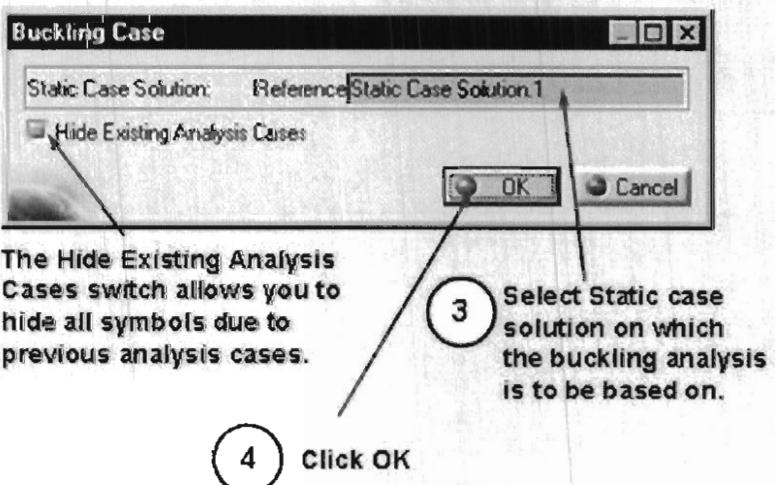
The factorization of the restraint set is performed only once

### 3.2. The Buckling Case - Creation

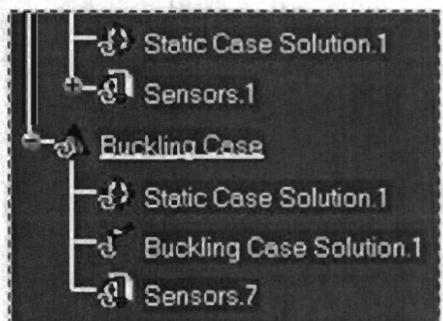
- Inserting a new Buckling Case allows you to perform the computation of buckling modes associated with a given Static Analysis Case



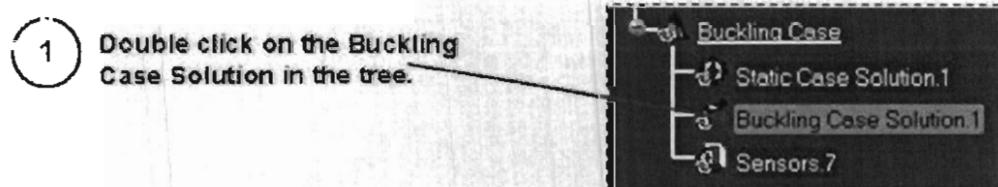
The Buckling Case dialog box is displayed.



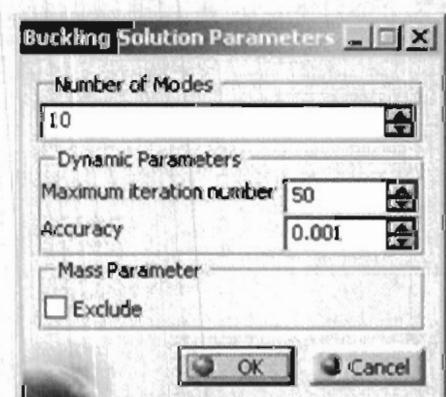
The new Buckling Analysis Case representation in the features tree contains two object sets : a Static Case Solution and a Buckling Case Solution.



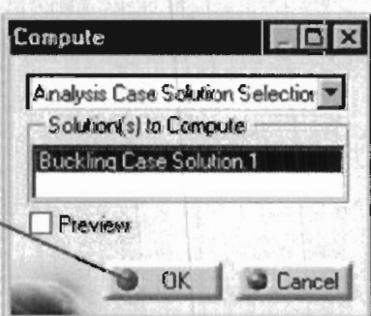
### The Buckling Case - Computation



The Buckling Definition parameters box is displayed.



- 2 Adjust parameters (number of modes, dynamic parameters).



- 3 Compute Buckling Case Solution.

# The Buckling Case - Post-Processing

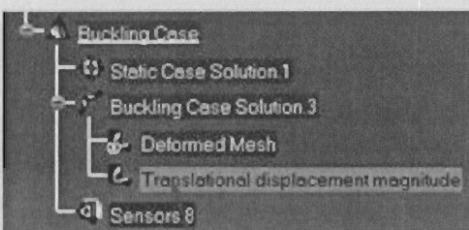
- Post-processing a Buckling Case is similar to post-processing a Frequency Case. In both cases you can create either a Displacements image or a Deformations image, and then select one of a given set of buckling modes.
- Each mode is associated with a coefficient, by which you multiply the loads in order to obtain the given buckling mode.

- Create either Deformations or Displacements image (click on image icon and select buckling case solution).



Image Editor box is displayed.

The created images appear in tree under Buckling Case Solution.



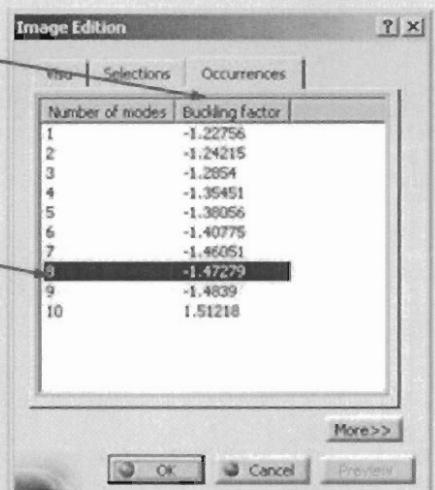
- Double click on image (tree or model).

Buckling factors represent the amplification coefficients to apply to the effort in static case for which the buckling modes appear.

- Choose mode and customize image.

Only the first Buckling factor is generally used to verify if the load is critical.

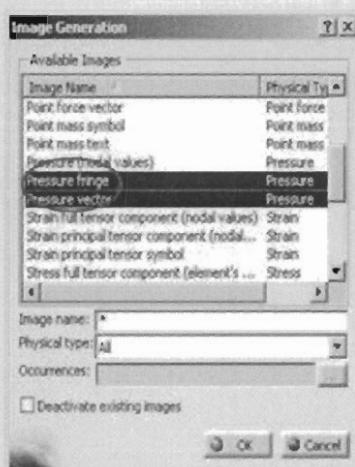
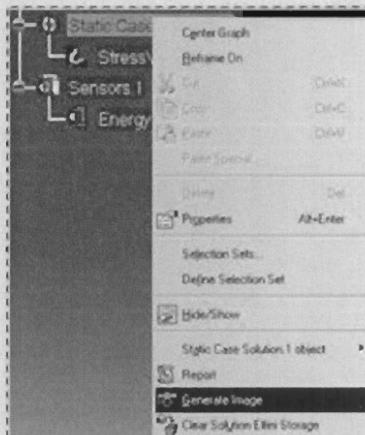
You have to multiply the load value (a unit or the real loading) by the Buckling Factor to have the loading limit.



## 3.3. Contact Pressures

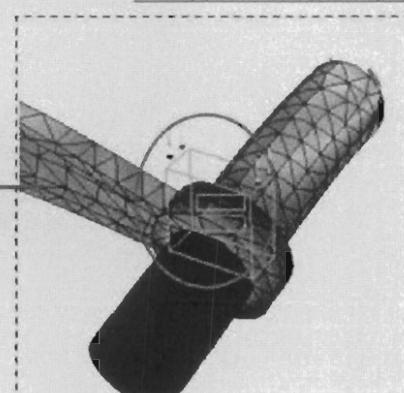
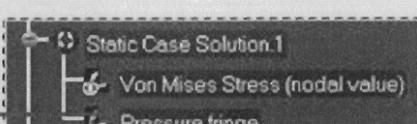
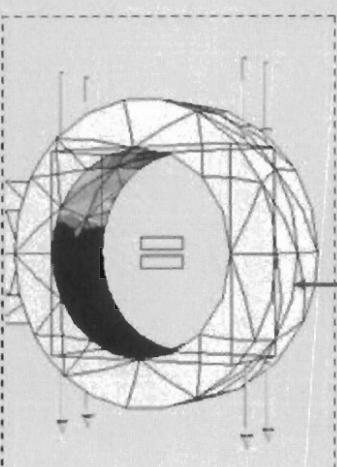
- Contact Pressures are advanced visualization functions which allow the user to visualize on each part the pressure values exercised on contact surface

- Perform an assembly structural analysis with contact connexions and compute it.



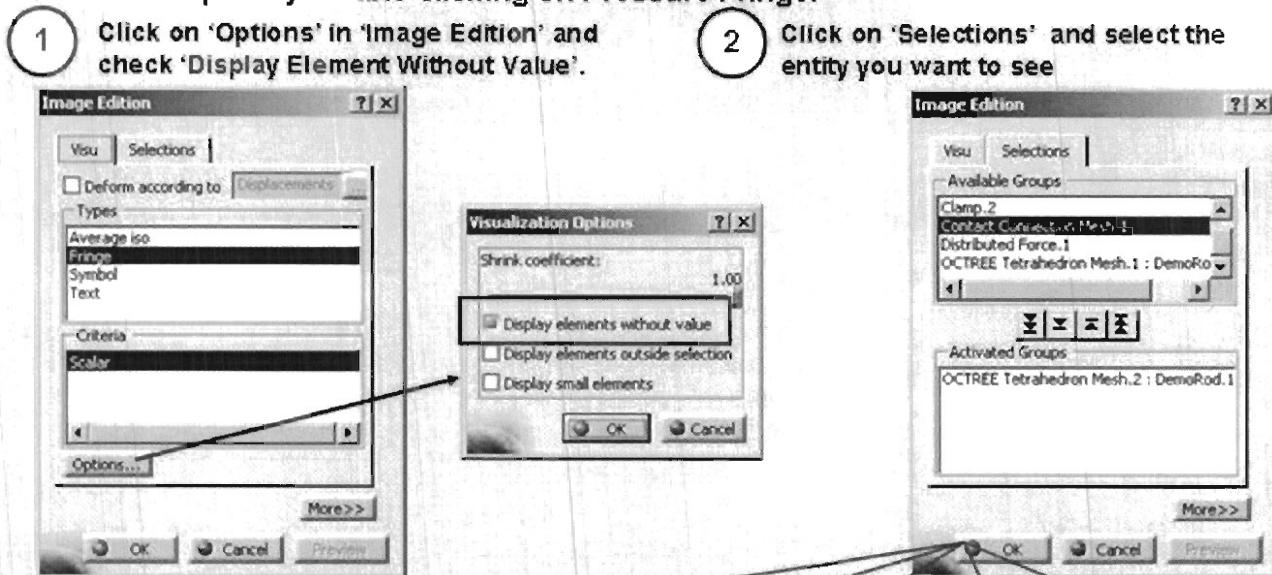
- Right-click on "Static Case Solution.1" and select Generate Image.

- 3- Select Pressure Fringe or Pressure Vector



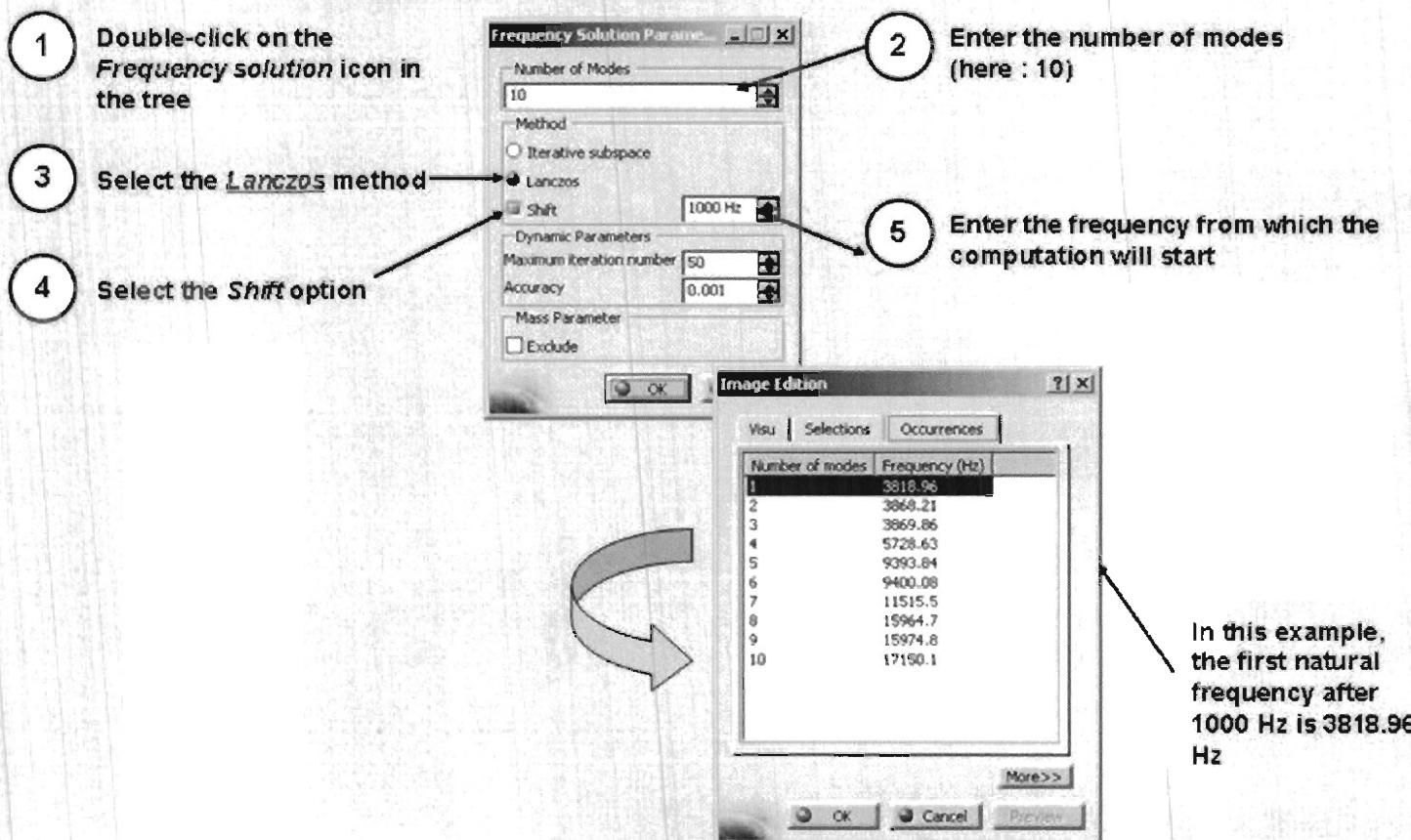
# Contact Pressures - Customization

- Customize the Contact Pressures Visualization to visualize only the contact pressures on each part by double-clicking on Pressure Fringe.



## 3.4. Frequency Shifting

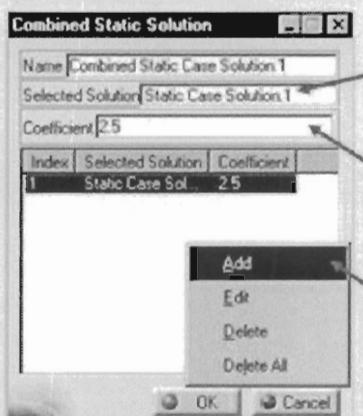
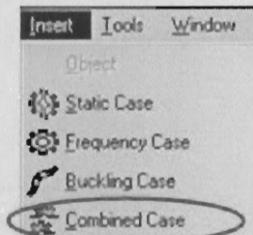
- With Frequency Shifting, you can avoid heavy computation to look at natural frequencies in a given range, and so reduce computation time.



### 3.5. Combined Case

- You have the possibility to create static cases linear combination. Once you have compute several static cases, you can perform solutions combinations in Post-processing analysis.

- Create static cases. Insert a *Combined Static Case*.

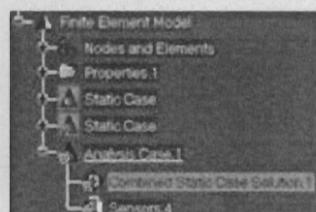


2 Select the first *Static Case Solution* in the tree.

3 Affect a coefficient :

$$\text{CSCS} = A^* \text{SCS.1} + B^* \text{SCS.2}$$

4 Add another Static Case : Contextual menu into the window and click *Add*.

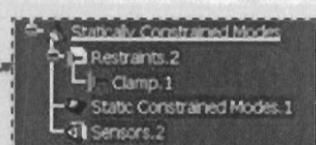
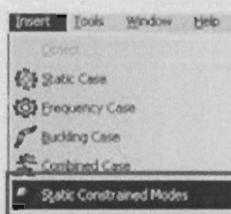


### 3.6. Static Constrained Modes Case

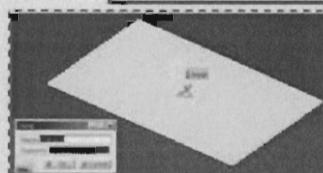
You have Generate a multi-occurrence solution made of static constrained modes cases.

- The output is a multi-occurrence solution made of static constrained modes : unitary imposed displacements is imposed on each DOF constrained by the restraint.
- The solution consists in (nb restrained nodes \* nb restrained dof) structure displacements.

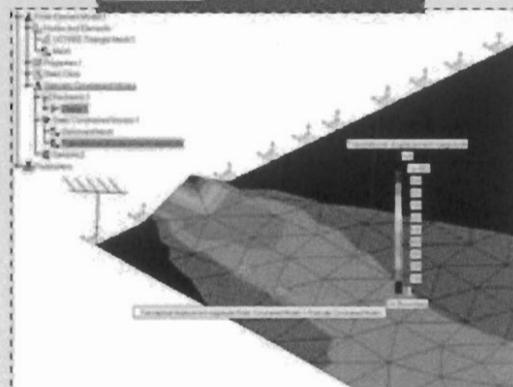
- Insert a *Static Constrained modes Case*.



- Create a Restraint  
For example a clamp on an edge



- Launch computation



- Display the results:  
deformed image/ displacement image

- Animate the deformed shape

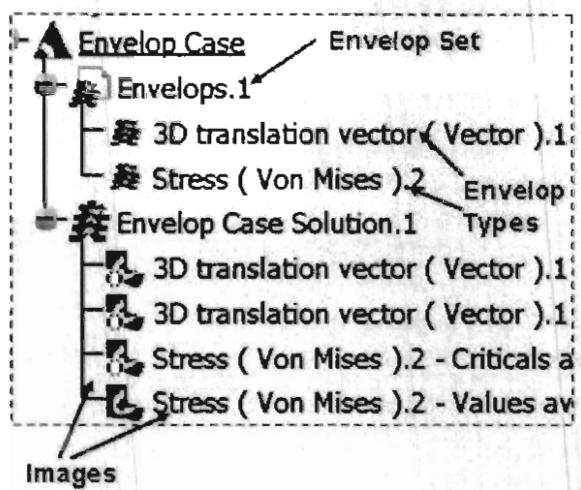
The animation shows each restrained nodes moving along its constrained DOF : X, Y and Z

## 3.7 Envelop Case (1/2)

This functionality enable you to search a selected number of most critical values such as minima, maxima or absolute value among several analysis sets. There are two types of Analysis set

- solutions set
- load set

These extremes may be computed on any kind of values provided by the selected analysis sets.

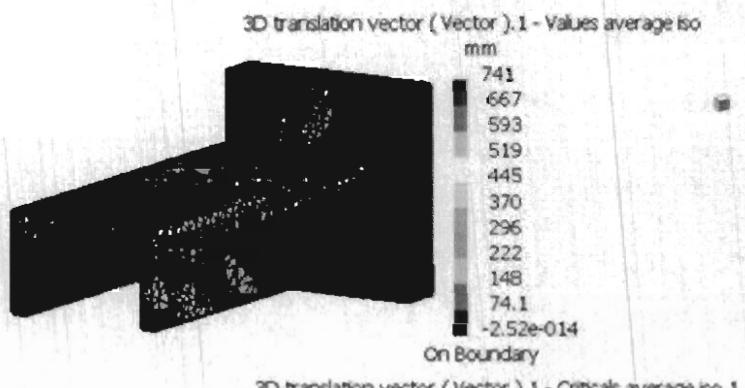


The Envelop Case will contain:

- An *Envelop Set*: It contains definition of the support i.e. the entities on which the envelopes will be computed and the selected analysis sets. Under this *Envelop Set*, you can create the *Envelop Types* which will contains information about what is to be computed like physical type, values type, axis system definition, results definition.
- An *Envelop Solution*: Solution will contain the result of the envelop computation. It will be updated using the standard 'Compute' command.
- The results of an *Envelop Solution* will be available through images.

## Envelop Case (2/2)

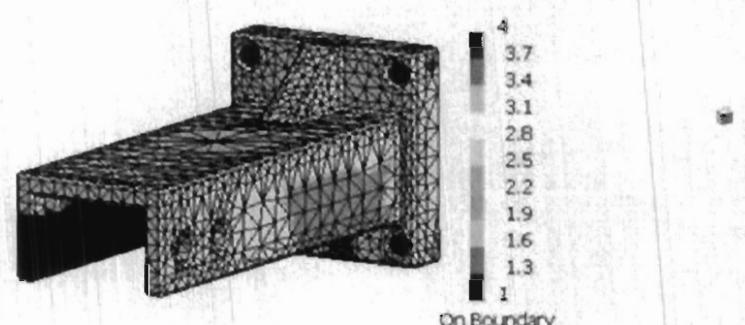
Two types of images will be available per envelop types.



- **Values images**

The solution index corresponds to the specified solution case.

The most critical value will be displayed according to the chosen visualization mode



- **Critical solutions images**

The critical solutions will be displayed, by their index.

For both types of images, it will be possible to change the visualization type.

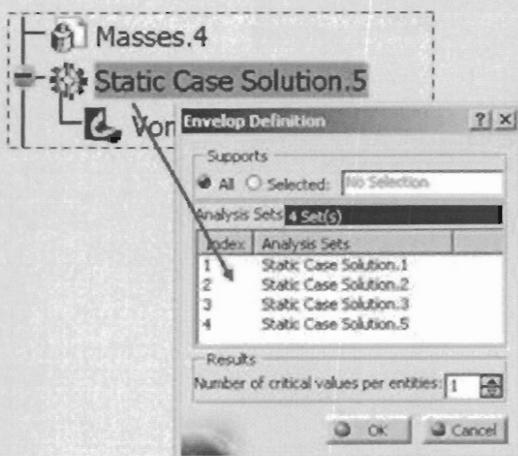
# Envelop Case - Creation

You will learn how to perform envelop case.

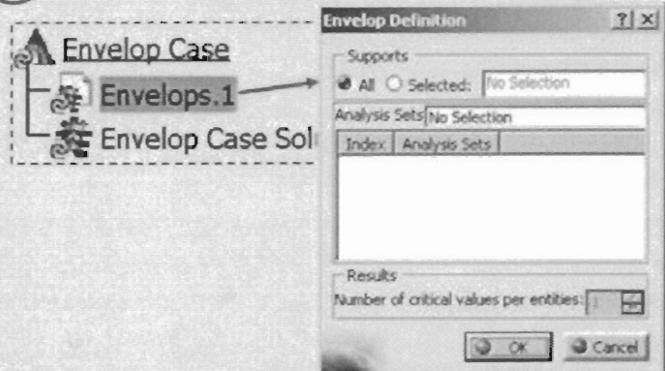
- 1 Go to Insert → Envelop Case



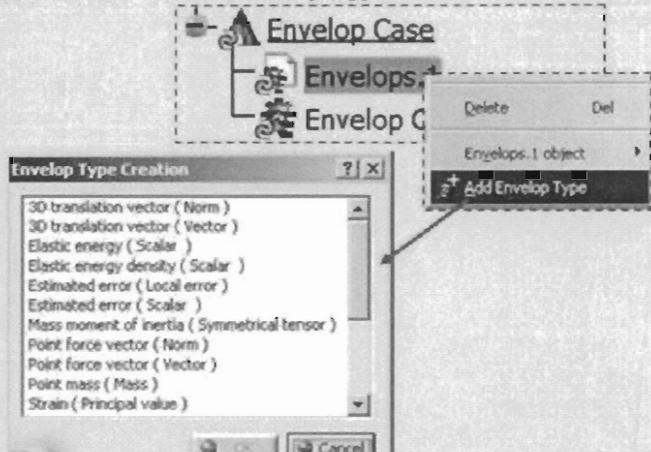
- 3 Check support as 'All' and select the required Solution Sets in 'Analysis Sets'



- 2 Double click on Envelops.1

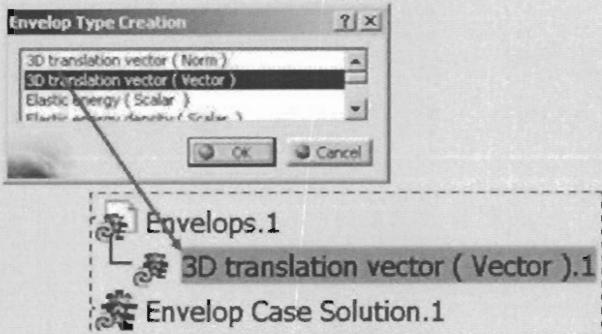


- 4 Go to 'Envelops' contextual menu and click on 'Add Envelop Type'

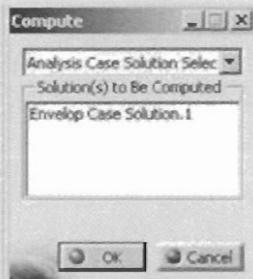


## Envelop Case – Creation

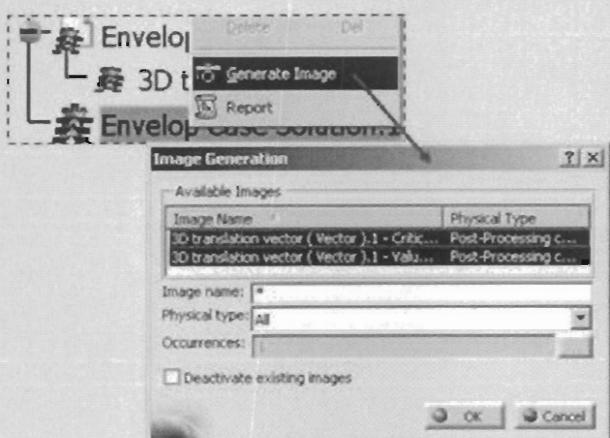
- 5 Select required Envelop Type for which case to be calculated



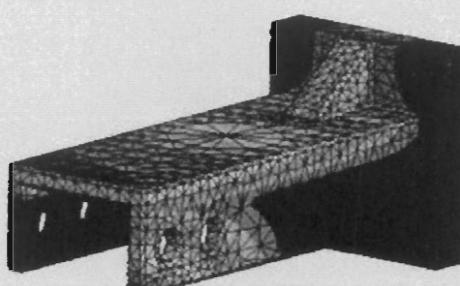
- 6 Compute the Envelop case



- 7 Go to 'Envelop Case solution' contextual menu, click on 'Generate Image'



- 8 Select critical symbol image and values image and click ok to confirm.

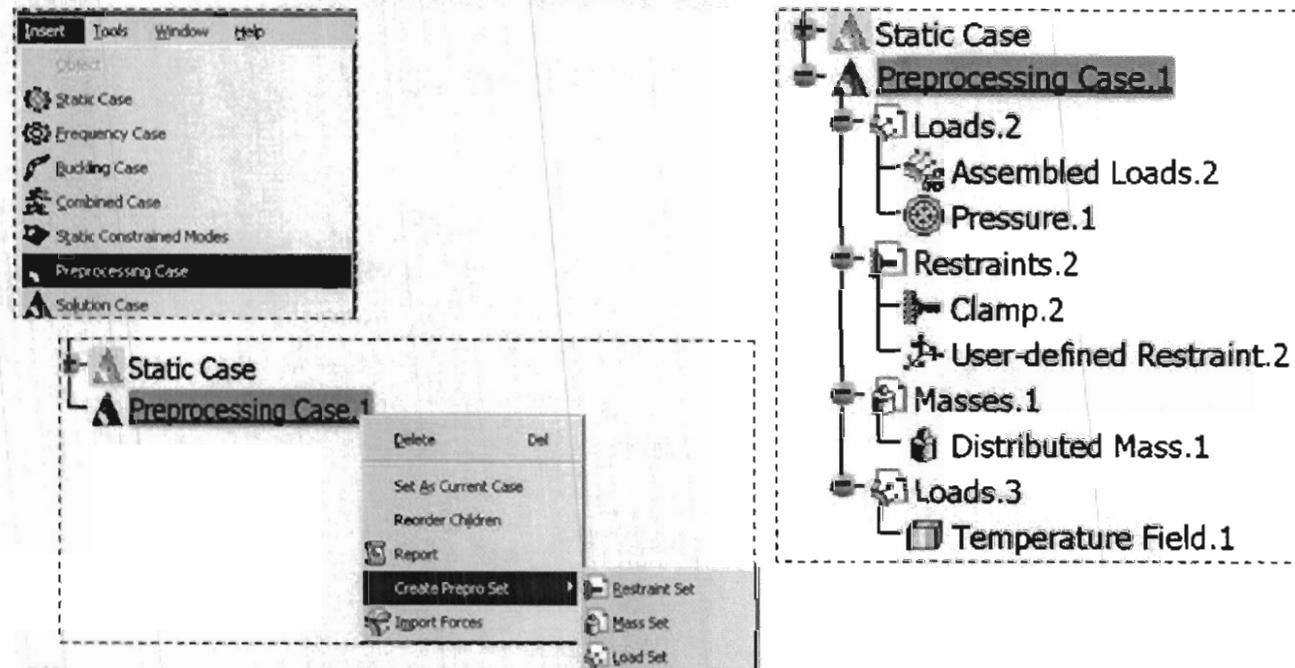


### 3.8. Pre-processing Case

V5R16

Pre-processing case is new kind of analysis case which contains a lot of pre-processing data and no solution sets. It is useful tool to organize, store pre-processing data and reuse it in Solution-based cases.

You can create several pre-processing cases in the same analysis model. You can define any type of Load Sets, Mass Sets or Restraint Sets.



It is possible to insert transferred  
Computed Loads in pre-processing sets.

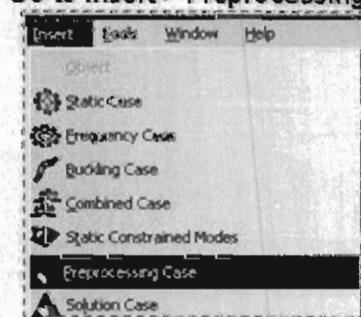
### How to Create Pre-processing Case

Pre-processing case is used to store pre-processing data and reuse it in Solution-based cases.



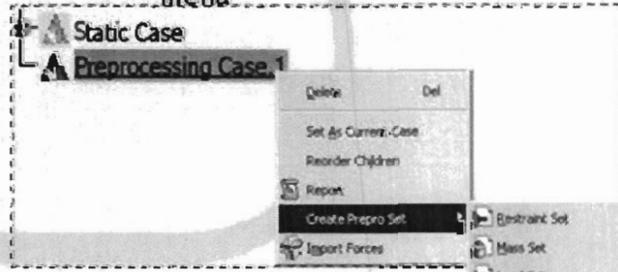
1

Go to Insert > Preprocessing case



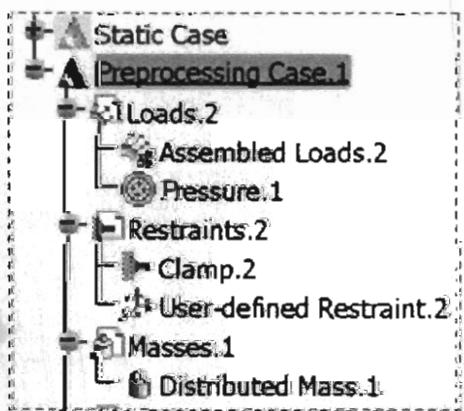
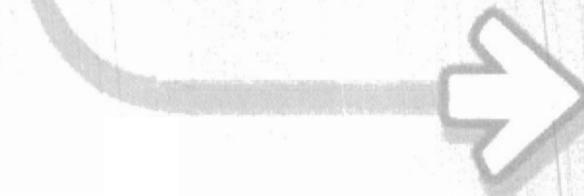
2

Create pre-processing set in  
Preprocessing Case contextual  
menu.



3

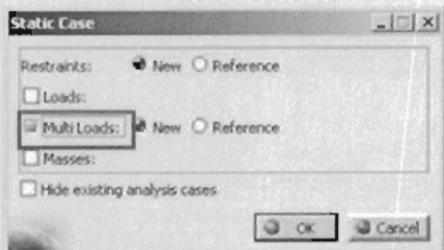
Create required Restraint  
Set, Mass set or Load sets



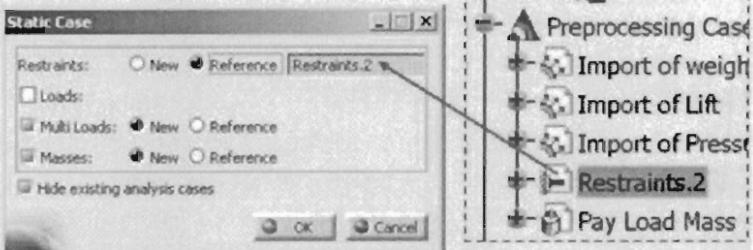
# How to Use Pre-processing Case

You will see how to use pre-processing data created in Pre-processing Case in Static Multi Loads Case.

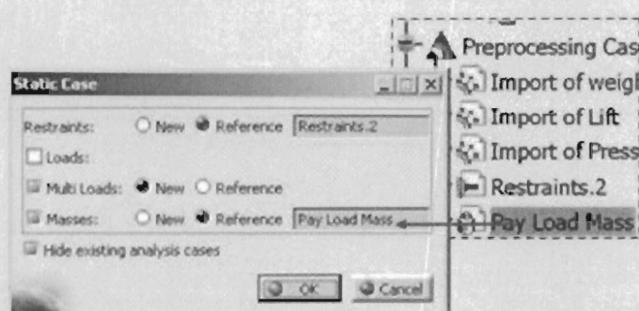
- 1 Go to Insert > 'Static Case' and check 'Multi Loads' option.



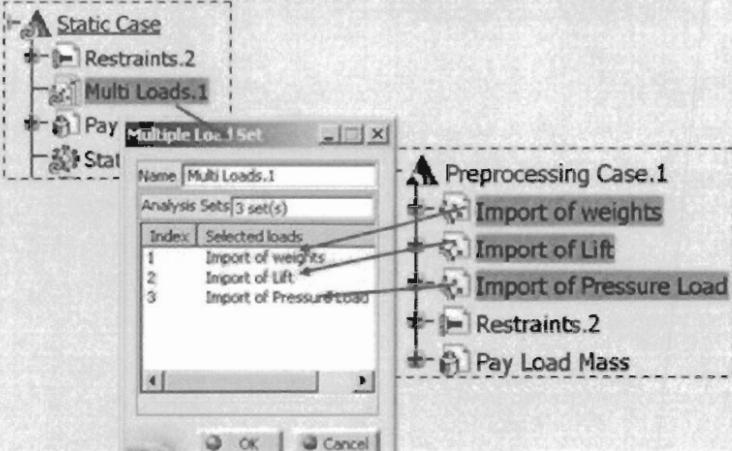
- 2 Check 'Reference' option in Restraints and select required Restraint Set from Pre-processing Case



- 3 Check 'Reference' option in Masses and select required Mass Set from Pre-processing Case and click OK



- 4 Double-click on Multi Loads and select required Loads from Pre-processing Case in Multiple Load Set Panel

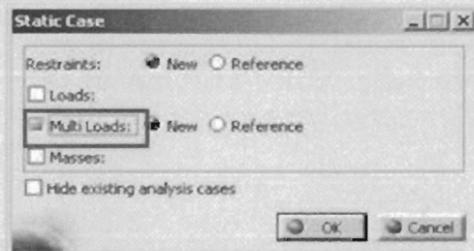


## 3.9. Multi Loads Case (1/2)

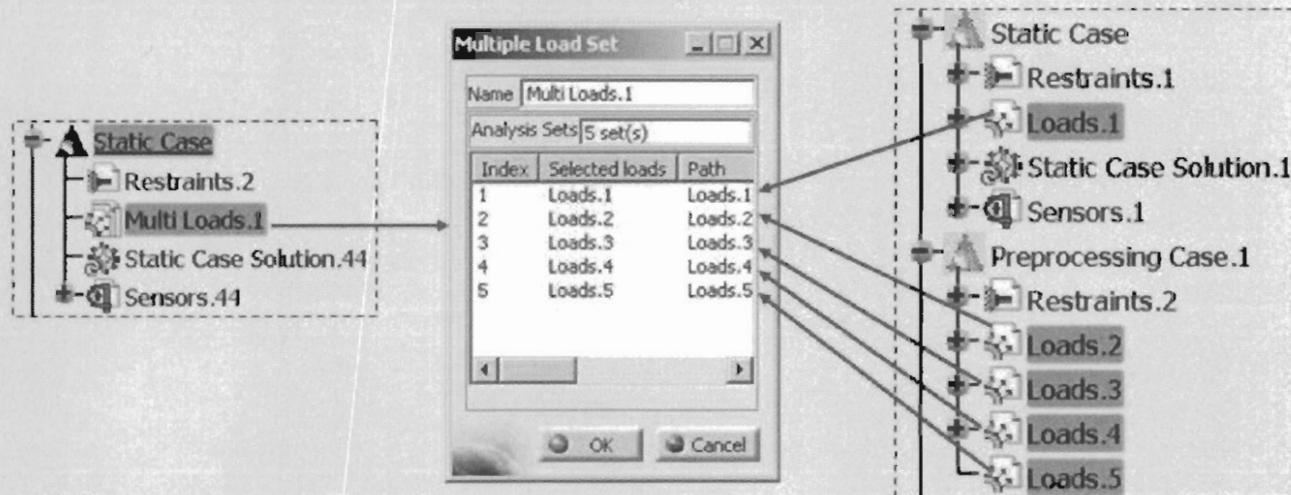
V5R16

Multi Loads Case is special Static Case containing:

- One Restraint Set
- One Multiple-Load Set
- One optional Mass Set



Multi Loads Set is a new type of Load Set. You can not create Load Set under a Multi Loads. You can only reference several Load Sets defined in any other cases. Thus, Multi Load Case Solution is multi-occurrence and each occurrence corresponds to a Load Set. It is necessary that all referenced Loads should belong to same Analysis Document.



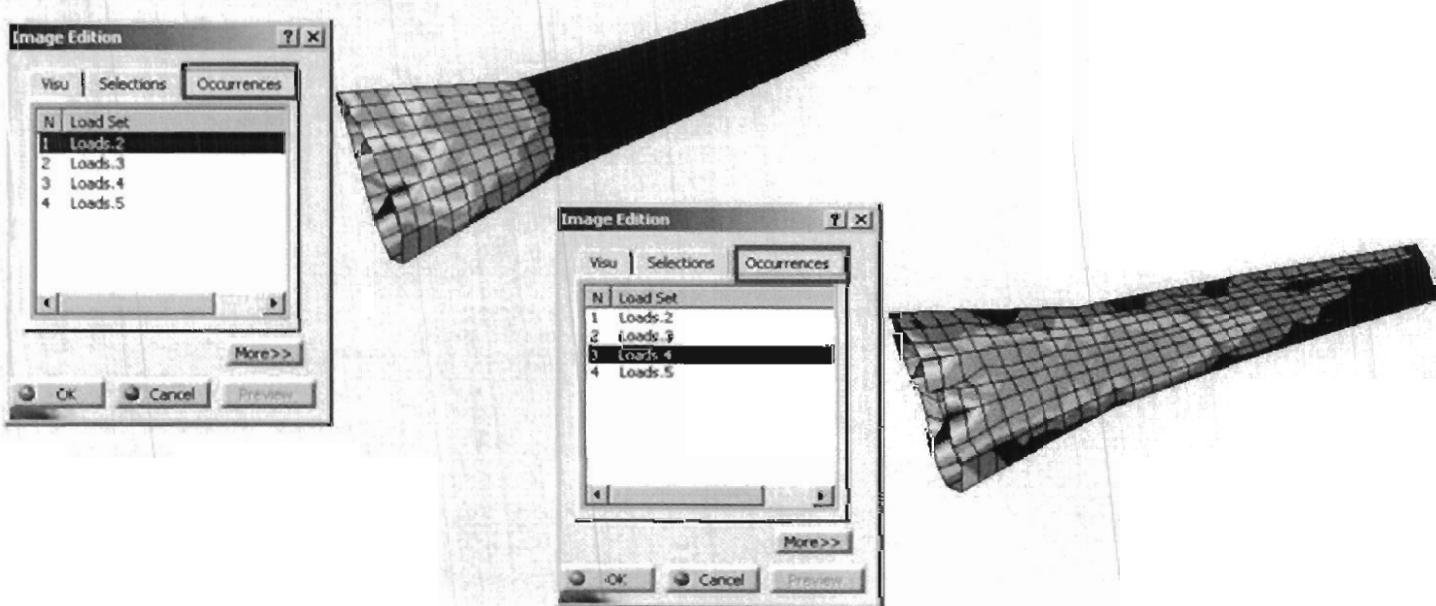
Multi Load Case uses Preprocessing case to store a large number of loads.

## What is Multi Loads Case (2/2)

Multi Loads Set can reference any type of Load Set, including inertia loads and enforced displacements, V4 imported Loads.

Inertia Loads computed within a Multi Loads Case are relative to the Multi Load Case additional Mass Set. Enforced Displacements computed in a Multi Load Case are relative to the Restraint entity specified in Enforced Displacement definition panel (which may not be included in the Multiple-Load Case Restraint Set).

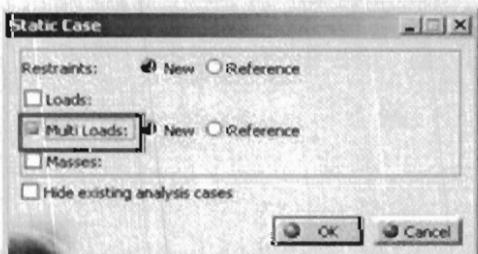
You can create images under the Solution using standard 'Generate Image' command. In image edition panel, the you can choose the Load Set. 'Occurrence' tab page displays the Load Set names.



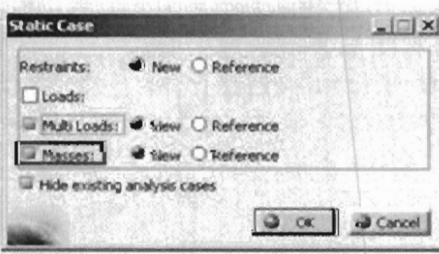
## How to Use Multi Loads Case (1/2)

Multi Loads Case used to solve large number of Load sets on single Finite Element model in one shot and thus avoids creation of separate analysis cases for each Load set. Multi Loads case generally references Loads from preprocessing.

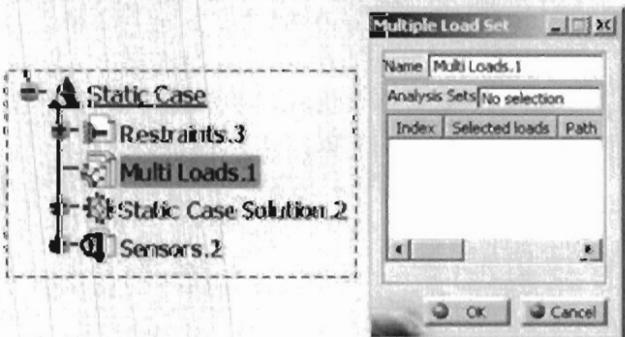
- 1 Go to Insert > 'Static Case' and check 'Multi Loads' option.



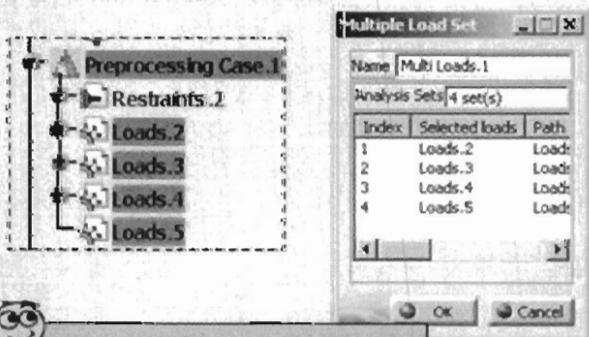
- 2 Check the Masses option, if required (optional) and press OK.



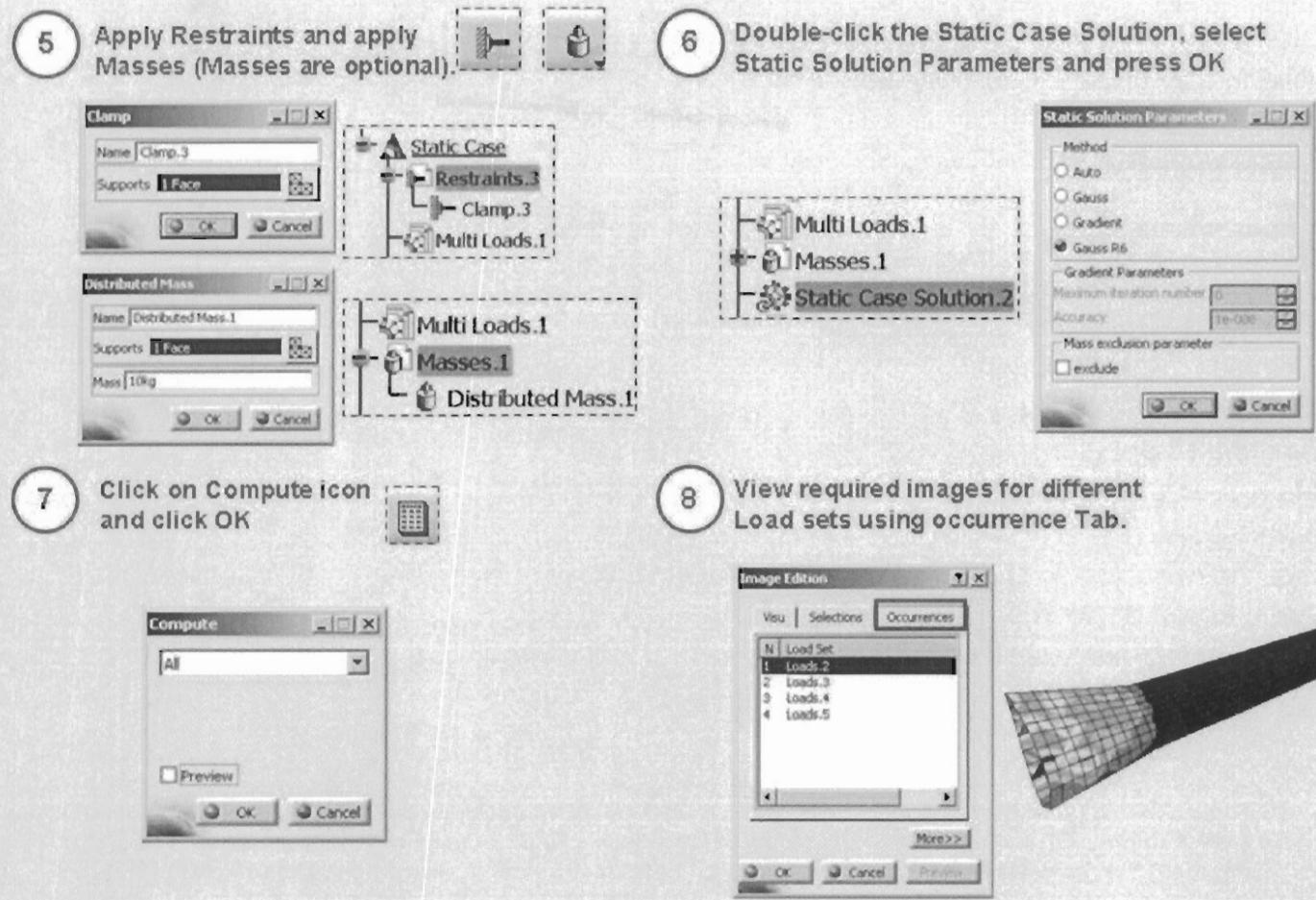
- 3 Double-click the Multi Loads in Static Case just created.



- 4 Select the required Loads from tree in the Multiple Load set.



## How to Use Multi Loads Case (2/2)

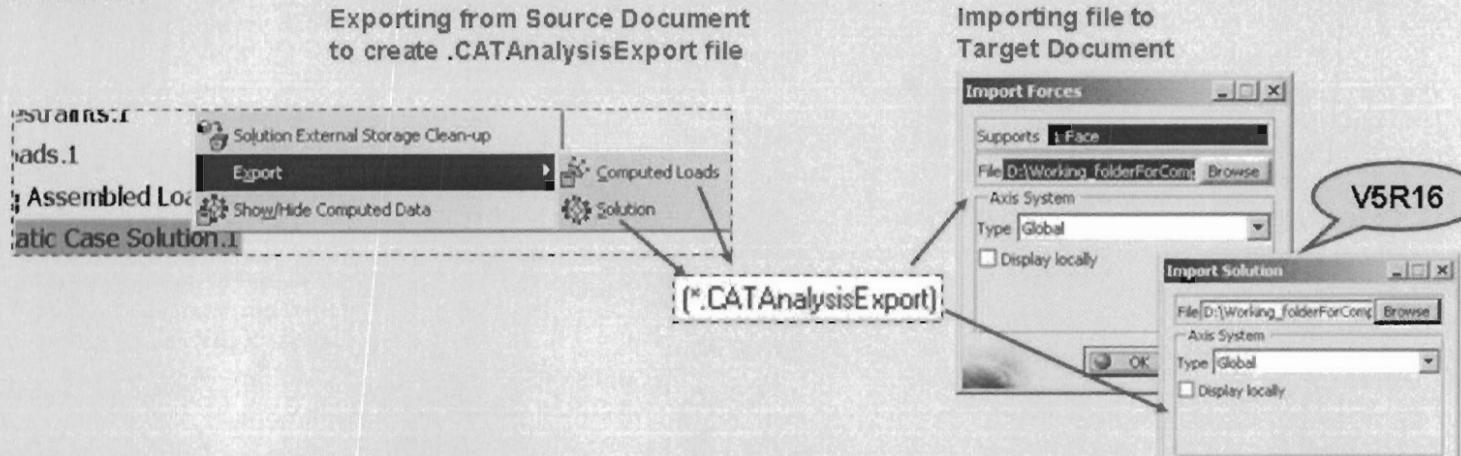


### 3.10. Transfer of Computed Loads/Displacements (1/2)

You can transfer Computed Loads/Displacements from one document to another document.

- Computed Loads (Forces) are applied loads evaluated by solver at the Degrees Of Freedom (DOFs) of mesh nodes.
- Displacements are translations/rotations at DOFs of mesh nodes.

A displacements transfer is nothing but copy solutions, between a source document and a target document. A loads transfer is a copy computed loads, between a source document and a target document.



The Transfer of Computed Loads/Displacements from Source Document to Target Document will be done in two separate steps:

- Export of Computed Loads/Displacements into .CATAnalysisExport file from Source Document.
- Import of .CATAnalysisExport file resulting from Source Document > Export to Target Document.

## What is Transfer of Computed Loads/Displacements (2/2)

To export computed loads, an 'Export Computed Loads' command will be available on the following type of data:

- Static Case
- Multi Loads Case

To export displacements, an 'Export Solution' command will be available on the following type of data:

- Static Case Solutions
- Combined Case Solution
- V4 Imported Solutions
- Assembled Solutions
- Multi-Load Cases Solutions

You can either transfer Computed Loads/Displacements:

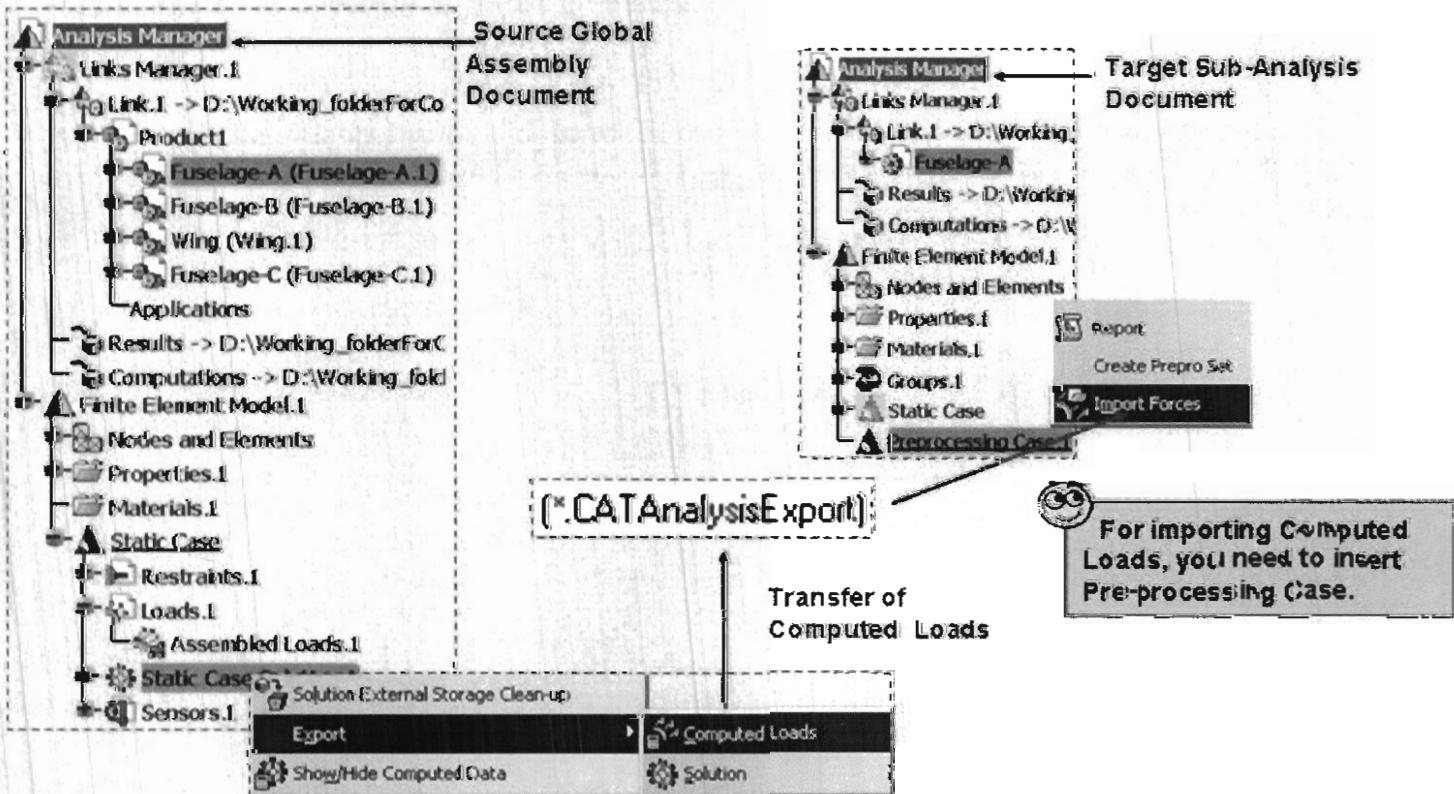
- From Assembly to Sub-Analysis or
- Between two analyses based on identical mesh



The .CATAnalysisExport file will contain the computed loads corresponding to the load sets used by the solution. Whatever the type of the transferred loads (Distributed Forces, Pressures, Force Densities) the file will contain the nodal force/moment vectors corresponding to these loads.

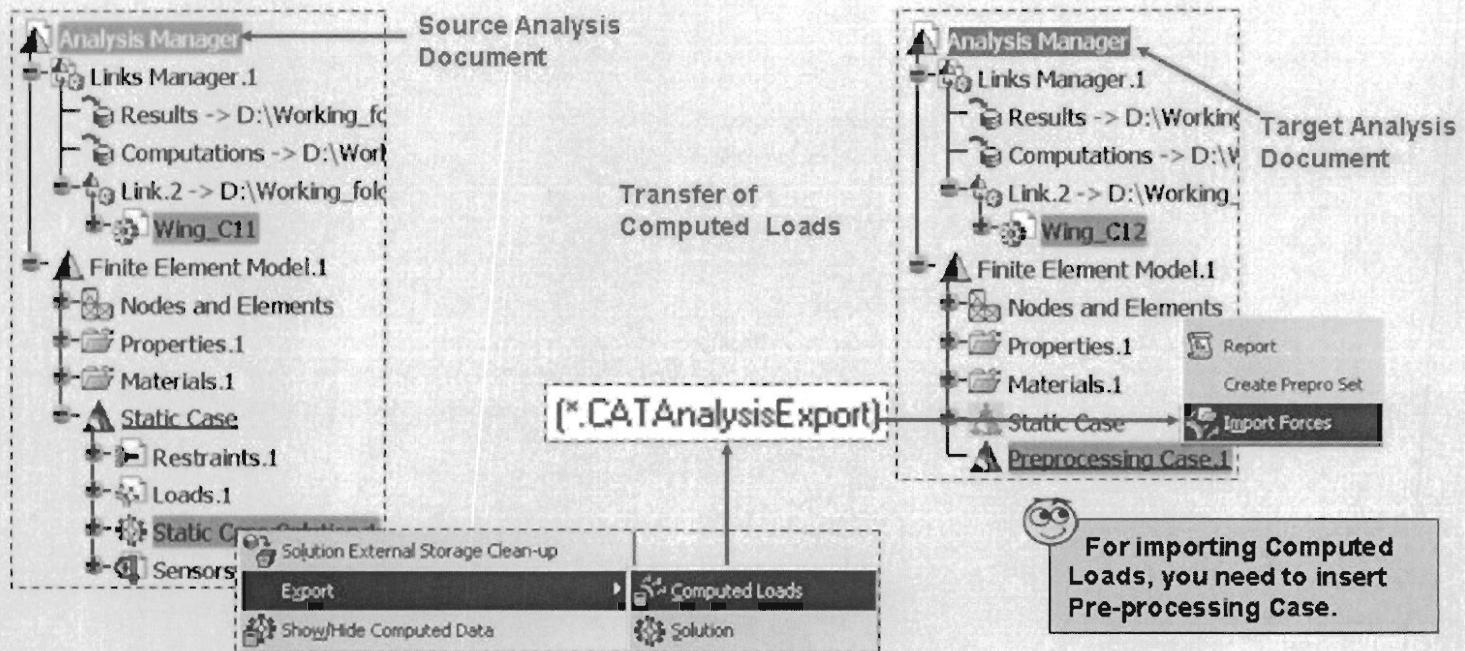
## What is Transfer of Computed Loads (1/2)

- Transfer of Computed Loads from Assembly to Sub-Analysis level



## What is Transfer of Computed Loads (2/2)

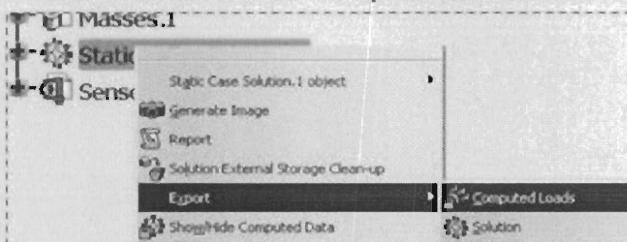
- Transfer of Computed Loads between two analyses document based on identical mesh.



## How to Transfer Computed Loads (1/2)

You can use Transfer of Computed Loads to transfer forces either from Assembly Analysis to Sub-Analysis level or between two Analysis Documents having identical mesh.

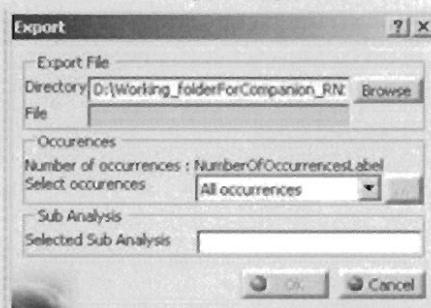
- 1 Go to Export > Solution in Static Case Solution contextual menu in Source Document and click 'Computed Loads'.



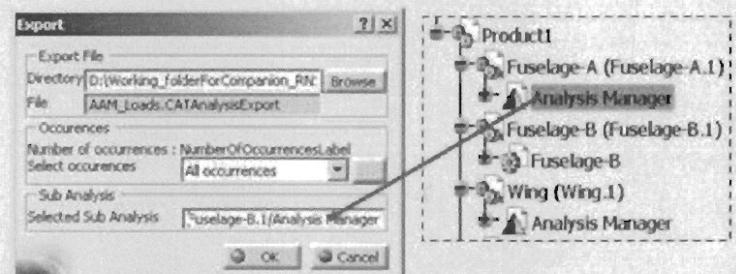
- 3 Enter the File name and click on 'Save'



- 2 Browse the location where you want to store .CATAnalysisExport file

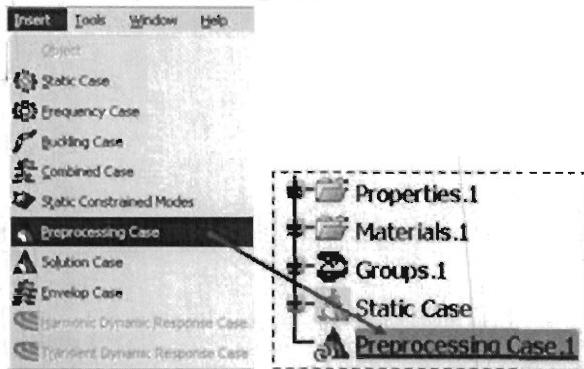
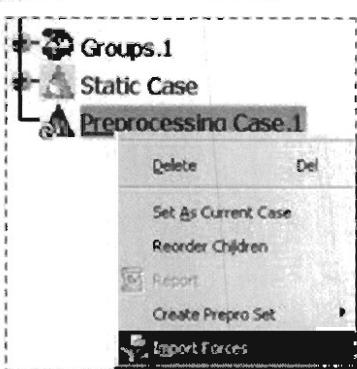
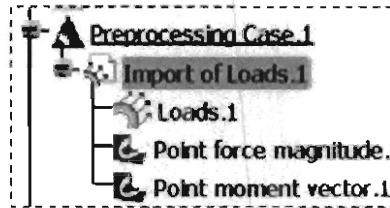


- 4 Select the number of occurrences and sub-analysis, if required and click OK.



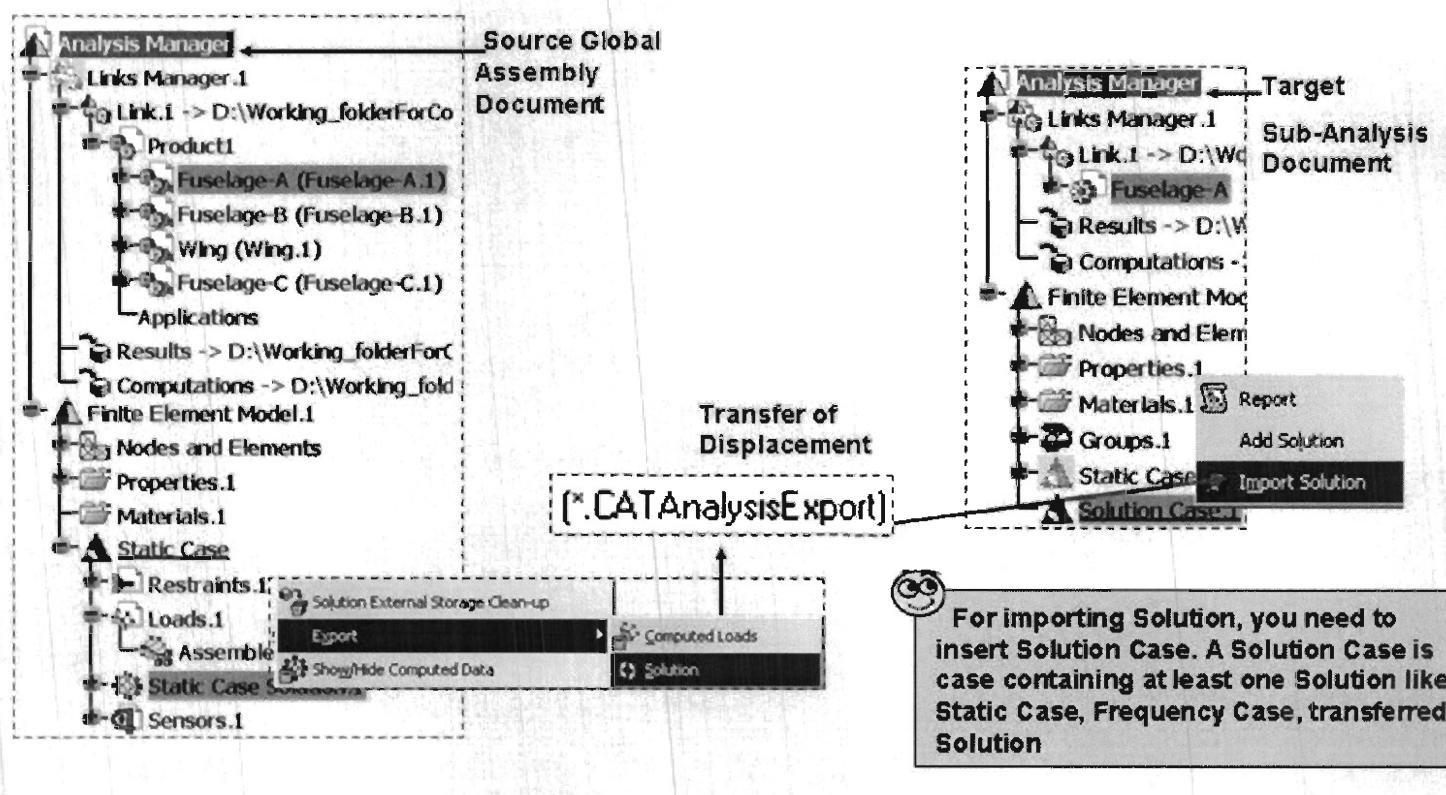
Sub Analysis option is available only In Assembly of Analysis. By default, the entire model will be exported.

## How to Transfer Computed Loads (2/2)

- 5 Open Target Document and add Preprocessing Case.
- 
- 6 Click the 'Import Forces' in Preprocessing Case contextual menu.
- 
- 7 Select the required support and .CATAnalysisExport file created from Source Document and click OK
- 
- 8 View images for Transferred Computed Loads using 'Generate Image' in 'Import of Loads' contextual menu.
- 

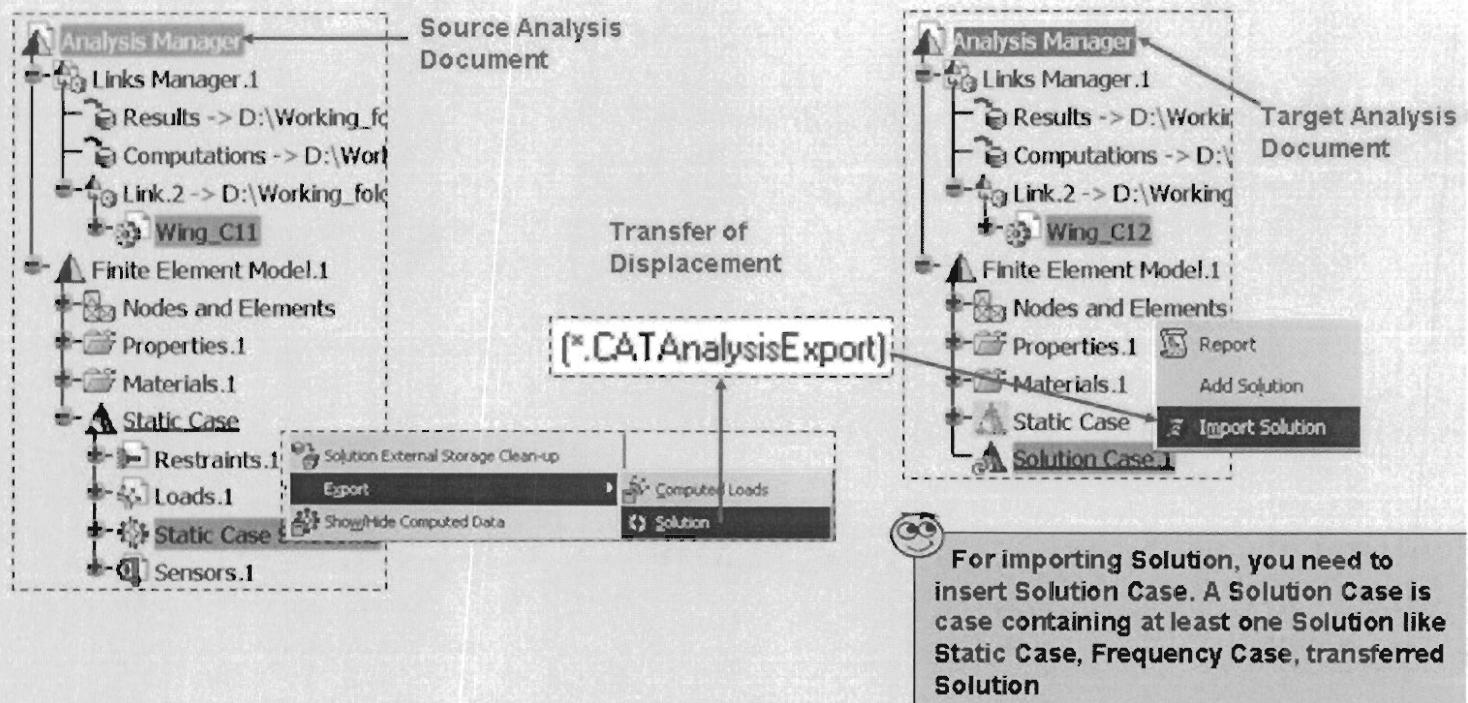
## What is Transfer of Displacements (1/2)

- Transfer of Displacements from Assembly to Sub-Analysis level



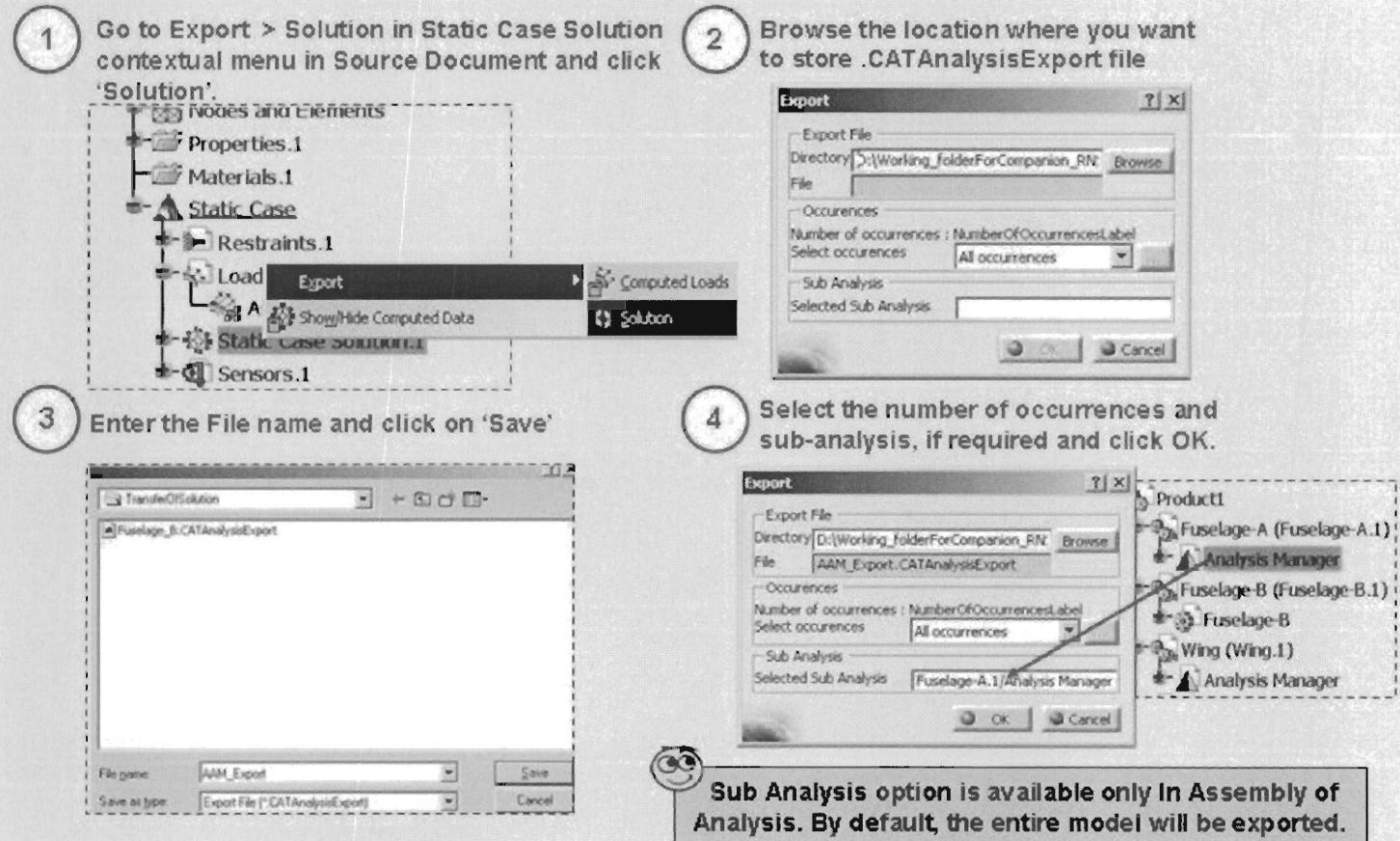
## What is Transfer of Displacements (2/2)

- Transfer of Displacements between two analyses document based on identical mesh.



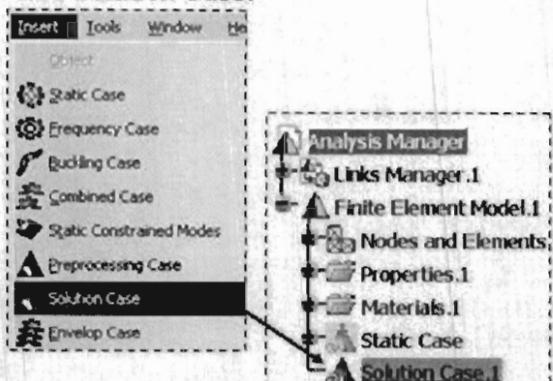
## How to Transfer Displacements (1/2)

You can use Transfer of Displacement to transfer displacements either from Assembly Analysis to Sub-Analysis level or between two Analysis Documents having identical mesh.

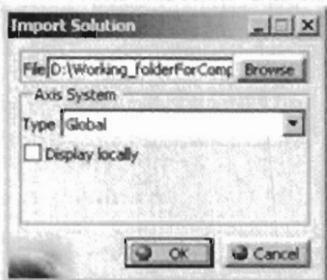


## How to Transfer Displacements (2/2)

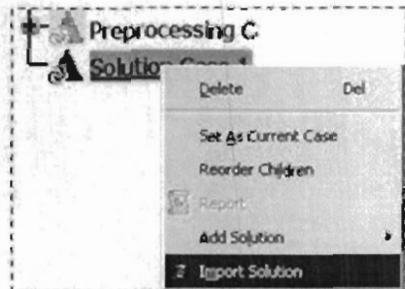
- 5 Open Target document and add Solution Case.



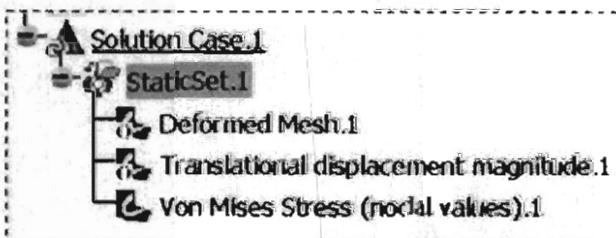
- 7 Browse .CATAnalysisExport file created from Source Document and click OK



- 6 Click the 'Import solution' in Solution Case contextual menu.



- 8 View required images for Transferred Solution using 'Generate Image' in 'StaticSet' contextual menu

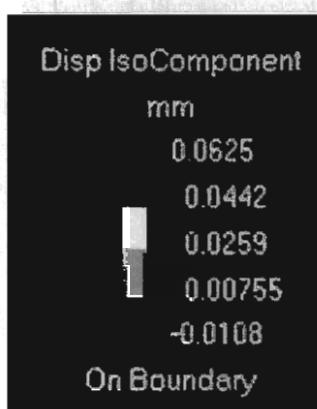


## 4. Advanced Post-Processing

*In this lesson, you have advanced post processing tools on top of the normal GPS and GAS post-processing tools.*

*These tools allow you even more to customize the results of your analysis computations.*

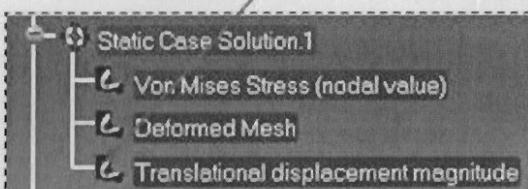
- Advanced Customization
- Advanced Report
- Advanced images
- Export of nodal values



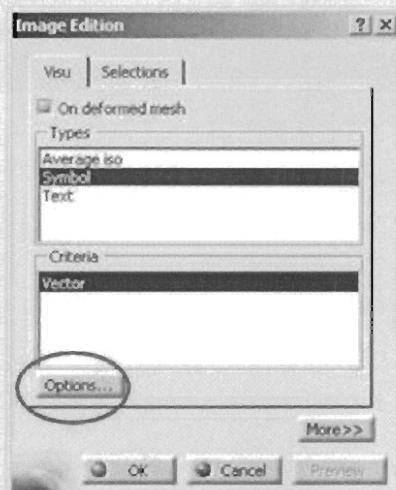
## 4.1. Advanced Customization - Image (1/3)

- With EST you get extra tools to customize your results images.
  - Here you will customize a displacements image.

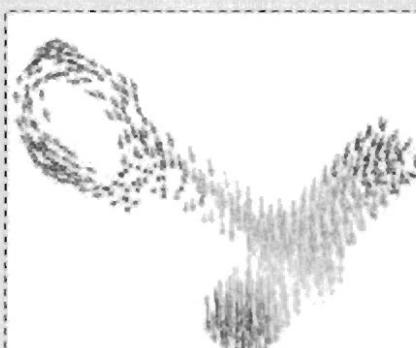
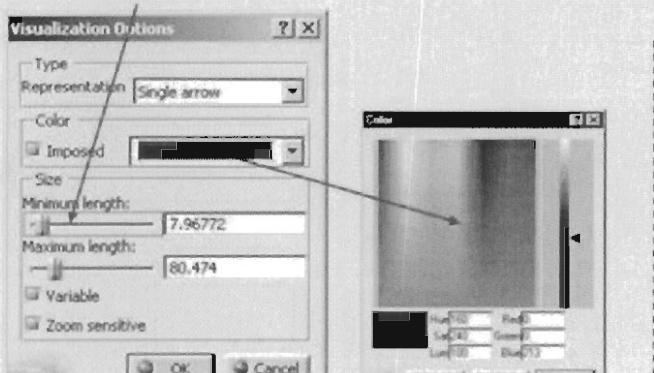
- Double-click either on an image representation in the tree, or on the image itself.



- Select Symbol.

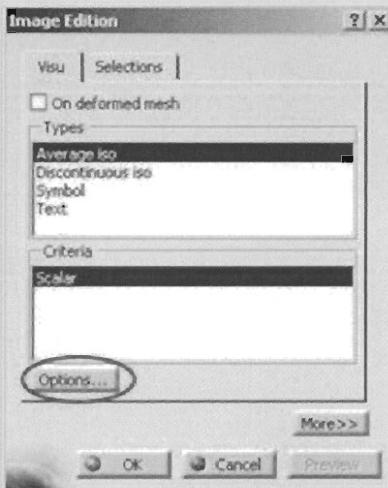


- Click on Options button and Change Arrow Length.



## Advanced Customization - Image (2/3)

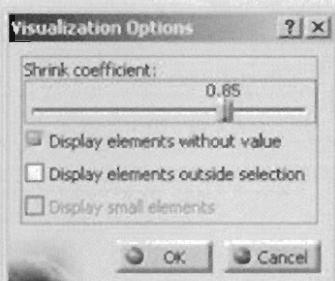
- Select AVERAGE-ISO visualization mode (on Von Mises image).



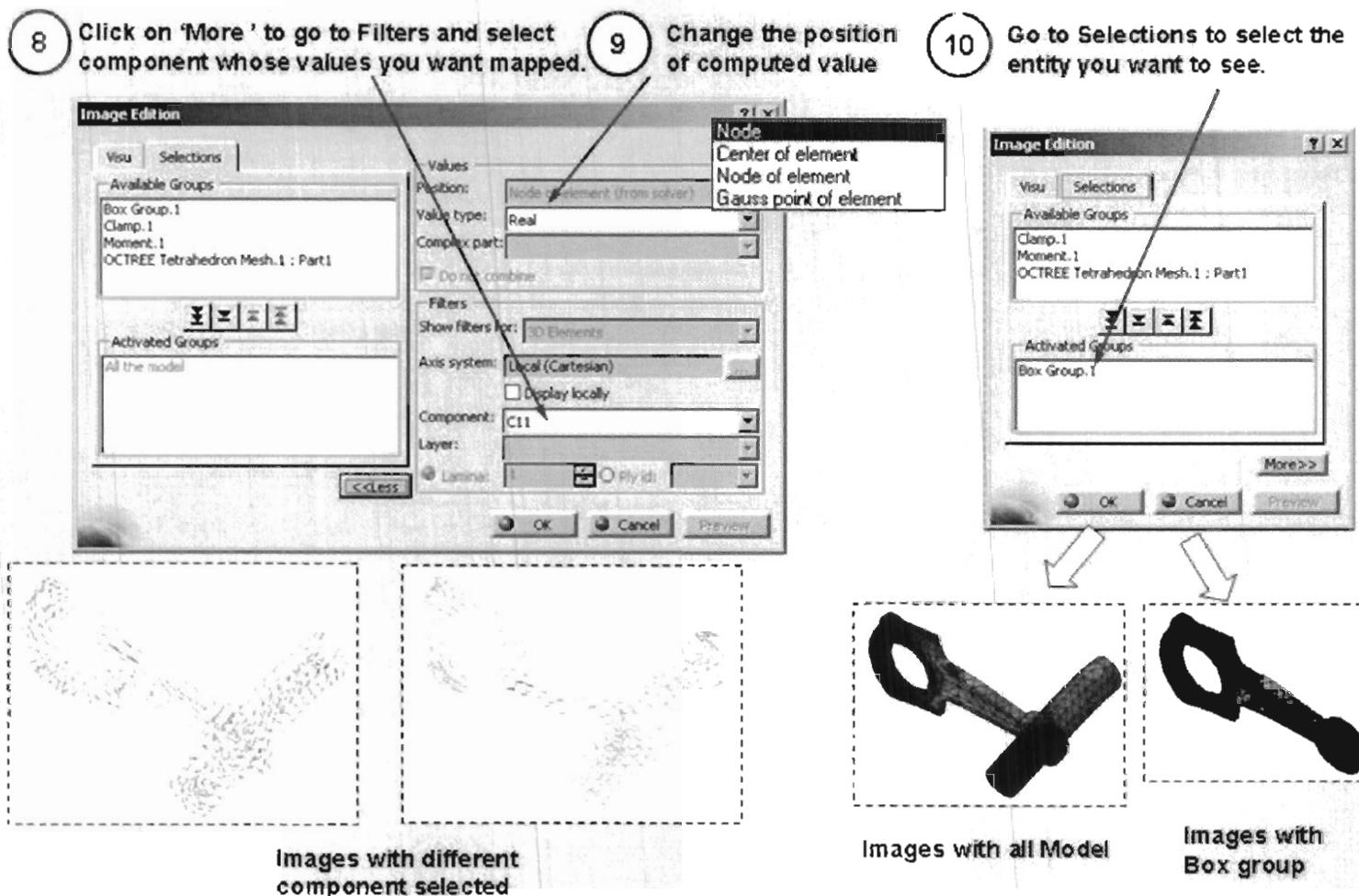
- Click on Options button.

- Change Shrink value

- Activate 'Display Element without value'



## Advanced Customization - Image (3/3)



## Advanced Customization - Color Palette

- With EST you have extra tools for mapping the color axis.

The screenshot shows the 'Color Map Editor' dialog box with several annotations:

- Create an image and then double-click on the color palette.
- Click More.
- Thresholds defined according to an histogram or linear distribution.
- Change the display format for a scientific or decimal display and Set significant digits.

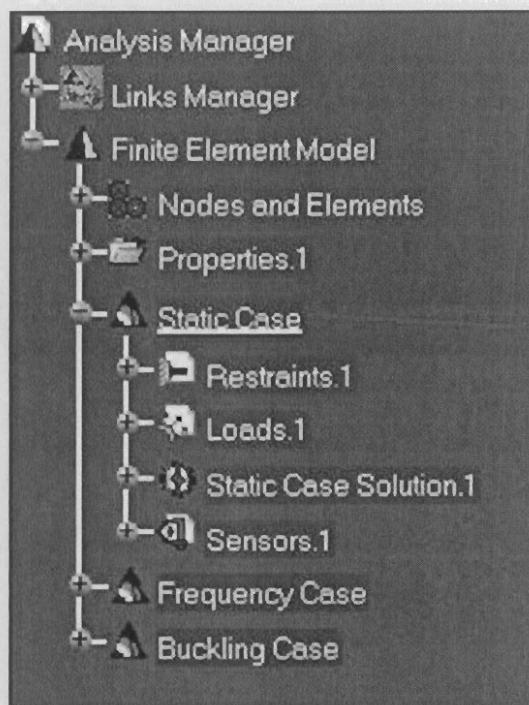
Below the dialog box is a table showing the color map distribution:

| Index | Value         | Imposed |
|-------|---------------|---------|
| 9     | 6.24757e+008  | No      |
| 8     | 6.02815e+008  | No      |
| 7     | 5.80873e+008  | No      |
| 6     | 5.58932e+008  | No      |
| 5     | 5.36996e+008  | No      |
| 4     | -4.24185e+006 | No      |
| 3     | -3.18139e+006 | No      |

An annotation points to the table with the text: "Option Histogram Distribution automatically adjusts color map to physical distribution."

## 4.2. Advanced Report (1/6)

- Advanced Report gives you access to all the data and results of your analysis, and allows you to build a fully customized report.



Imagine you are working on an analysis document containing three cases :

Static Case.

Frequency Case.

Buckling Case.

## Advanced Report (2/6)

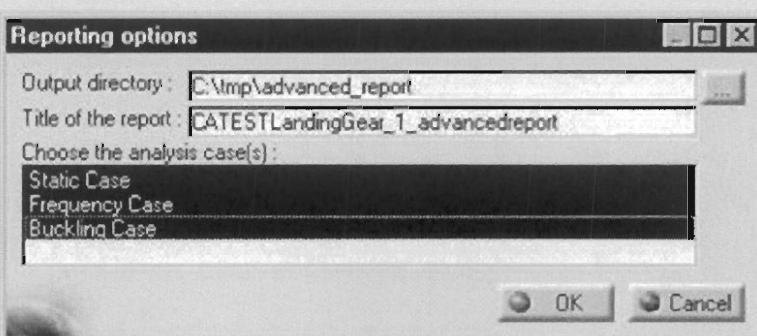
- Define the options for your Advanced Report.

1 Click on the "Advanced Reporting" icon



The Reporting Options box is displayed

2 Select an output directory

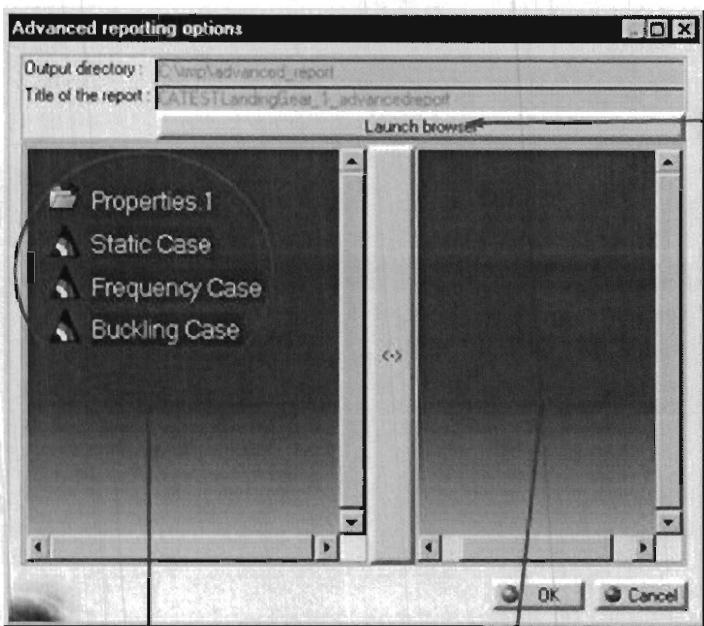


3 Enter a title for your report

4 Select the cases you want to include in your report

## Advanced Report (3/6)

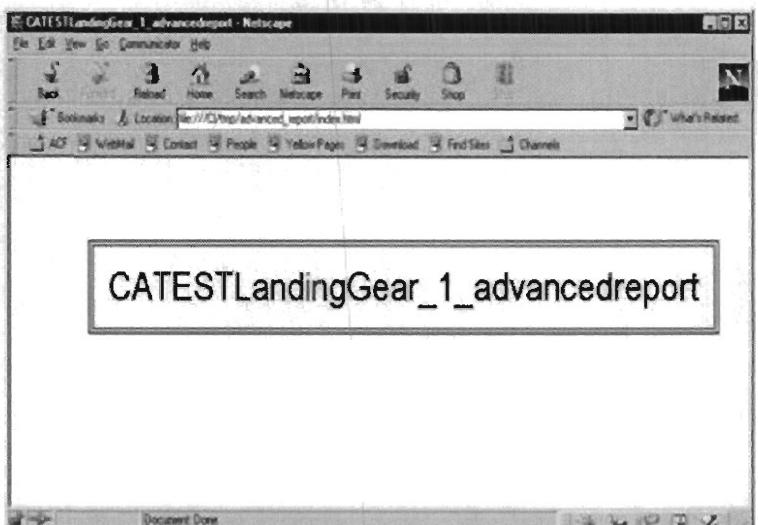
- Advanced Report Creation Panel.



The left part of the panel contains all the data and results available in the selected cases.

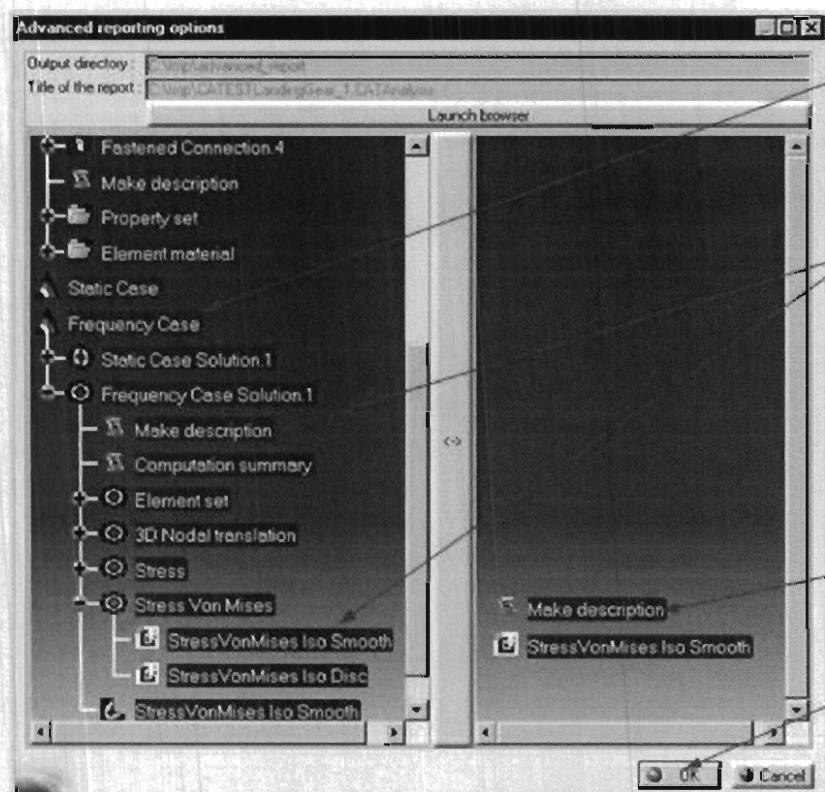
The right part of the panel will contain the structure of your report.

You can launch a browser to readily display the resulting report.



## Advanced Report (4/6)

- Build the structure of your Advanced Report using the data available in the left part of the Creation Panel.



1 Double click on the elements to extend the tree.

2 For each element you need in your report structure :  
Double click on it  
Or  
Select it and click the middle arrow

Each element selected will appear in the right part of the panel.

3 Proceed the same way to delete an element from your report structure.

4 Click OK when your report structure is completed.

Moving and zooming functions are available as in other CATIA windows (Middle mouse button to move + left button to zoom)

## Advanced Report (5/6)



- Visualization of the generated report. You can visualize it while creating the report structure by launching the browser from the creation panel.



The report is constituted by a set of files (html, jpg, etc.) located in the specified directory. A text document is also available, containing the same data without images.

## Advanced Report (6/6)



- A text document is also available, containing the same data than report without images. Its name is "ficel" without extension. You can open it with software like notepad. When you make a report, the data are added to it. You can also generate it by clicking the icon.

- It look like this.

```
STRUCTURE Computation
+
Number of nodes      : 1199
Number of elements   : 3492
Number of D.O.F.     : 3663
Number of Contact Elements : 0
Number of Kinematic relations : 6

cpu =      1.17 sec    elapsed =      1.30 sec    memory =      0.21 Mb

+
RESTRRAINT Computation
+
Name: RestraintSet.1
Number of S.P.C : 126

cpu =      0.03 sec    elapsed =      0.05 sec    memory =      0.04 Mb

+
LOAD Computation
+
Name: LoadSet.1
Applied load resultant :

Fx =  0.000e+000 N
Fy =  0.000e+000 N
Fz = -1.000e+003 N
Mx = -1.055e+002 Nm
My =  5.000e+000 Nm
```

## 4.3. Advanced Images

- Now, you have access to different advanced post-processing images on Properties, Loads, Static solutions and Frequency solutions.

- Contextual menu on *Properties, Loads, Static or Frequency solutions* in the tree. Click on *Generate Image*.

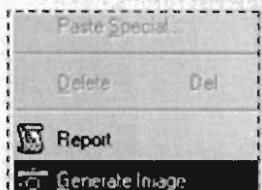
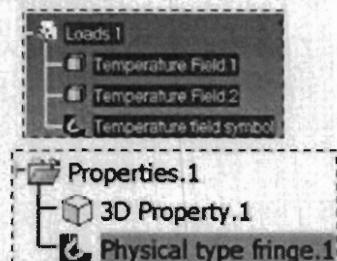
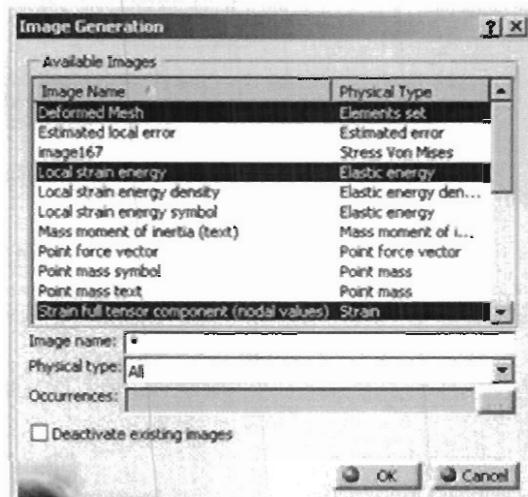


Image icons are created at the end of the tree.



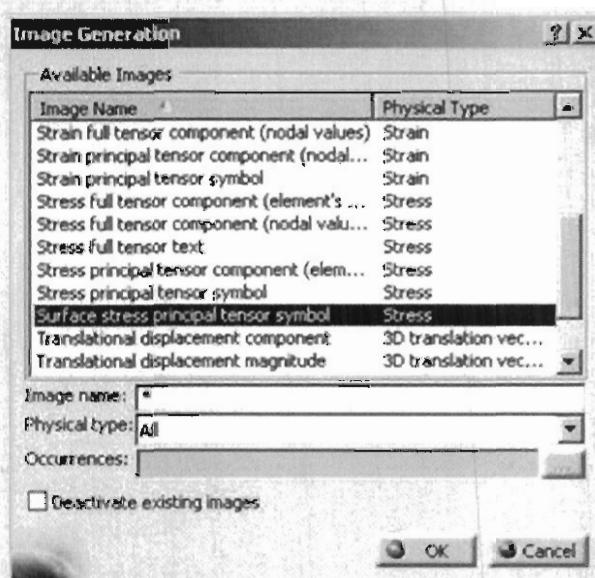
- Choose the Image to visualize.



- Properties : Beam properties, shell properties, contact, springs.
- Loads : Temperature field on mesh.
- Static Solutions : contact final clearance, frictional ratio, nodal masses and inertia.
- Frequency Solutions : Nodal masses and inertia.
- Combined Case Solution : images available on static sets.

## Post Processing : Surface Stress

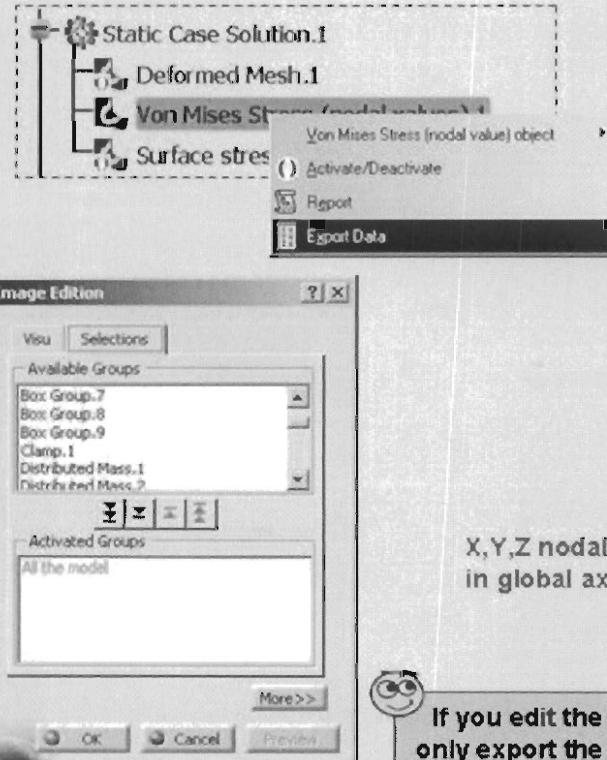
- A new “Surface Stress” image
  - Directly provide surface stress values computed at gauss points of element free faces
- Added Value
  - Provide surface stresses without surface coating mesh



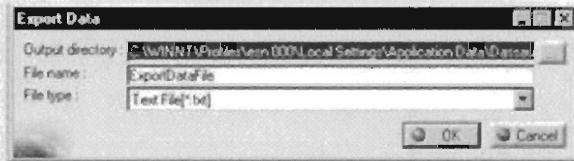
#### 4.4. Export of Nodal Values

- You can export nodal data from nodal values images (scalar, vector, tensor) with a text file or an .xls file.

1 Select **Export Data** in Contextual menu on a Post-processing visualization (*Von Mises Stress nodal value*, for example).



Enter the output directory, a file name and choose between a text file or an excel type file.



X,Y,Z nodal coordinates  
in global axis system

Microsoft Excel - Experiment.xls

Data in relation with the nodal value image

|    | A | B | C |
|----|---|---|---|
| 1  |   |   |   |
| 2  |   |   |   |
| 3  |   |   |   |
| 4  |   |   |   |
| 5  |   |   |   |
| 6  |   |   |   |
| 7  |   |   |   |
| 8  |   |   |   |
| 9  |   |   |   |
| 10 |   |   |   |
| 11 |   |   |   |
| 12 |   |   |   |
| 13 |   |   |   |
| 14 |   |   |   |
| 15 |   |   |   |
| 16 |   |   |   |
| 17 |   |   |   |
| 18 |   |   |   |
| 19 |   |   |   |
| 20 |   |   |   |
| 21 |   |   |   |
| 22 |   |   |   |
| 23 |   |   |   |
| 24 |   |   |   |
| 25 |   |   |   |

#### Data in relation with the nodal value image

If you edit the image and select a given selection, you will only export the nodal values of this selection (use Groups).

#### 4.5. Grouping for Post-processing

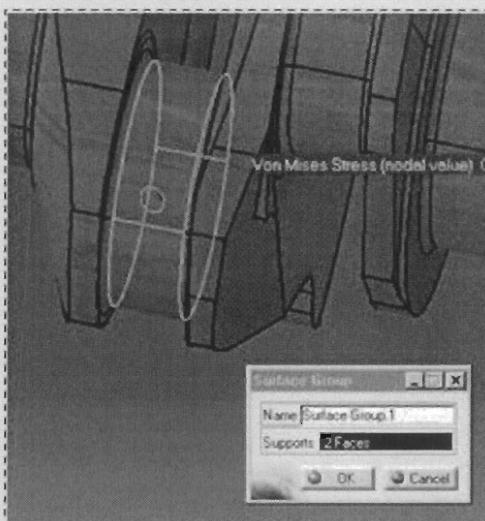
- Grouping elements allows you to generate images from a group of elements (points, lines, surfaces, bodies), or from a space allocation (box, sphere). So you have the possibility to focus post-processing analysis on critical areas.

1 Click a Group icon.



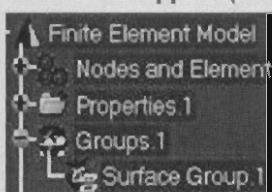
- Point group
- Line group
- Surface group
- Body group

**Box group**  
**Sphere group**



Double click on a result image object to open the *Image Edition* window (see next slide) : you can select the group.

2) Select the support (critical area for example).



A new feature set is created in the FEM tree.

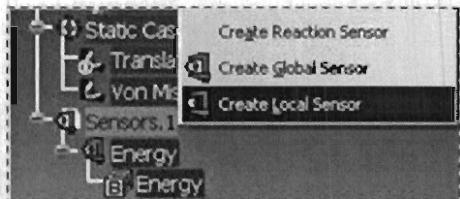


**Remember to deactivate results images before selecting geometrical entities for groups.**

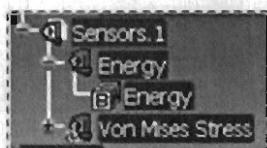
## 4.6. Local Sensors

- You can generate a local sensor: displacement, Von Mises extremum on a chosen geometry.
- It allows you to focus on specific area.

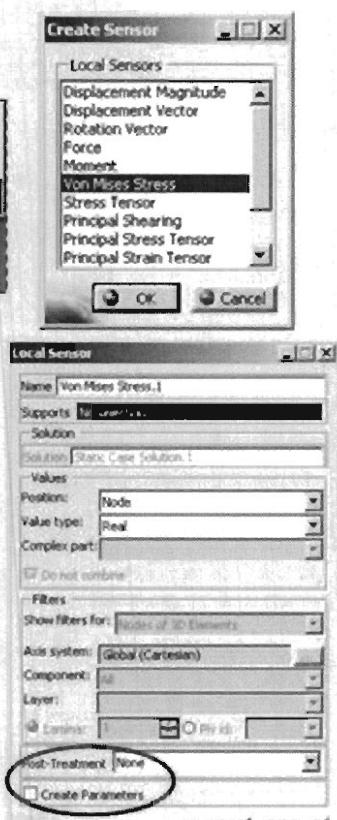
- Create a local sensor using sensors contextual menu.



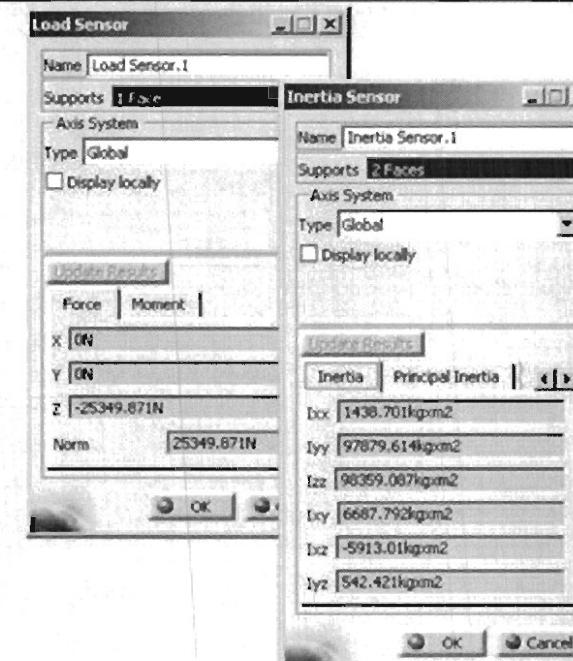
- Double click on the created local sensor.



- Enter Post Treatment : maximum, minimum or average and check to create parameters



V5R16  
Load sensor provides resulting force vector and moment vector: Fx, Fy, Fz, Mx, My, Mz. Inertia sensor provides mass and six inertia coefficients Ixx, Ixy, Ixz, Iyy, Iyz, Izz. These are available under Reaction Sensor.

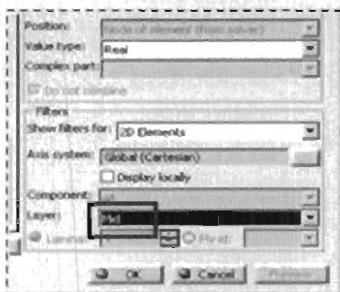


## 4.7. Image Template

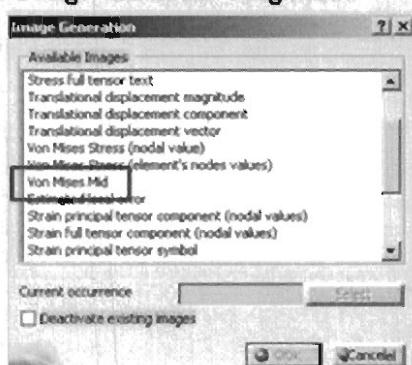
You can save your customised image as Image Template in order to store it

- It will be available in the Generate Images list
- Allows to benefit from previous Image customization

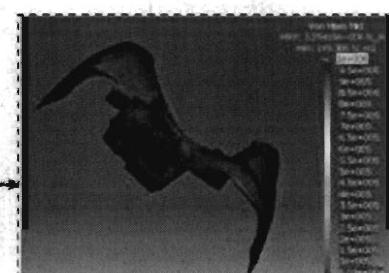
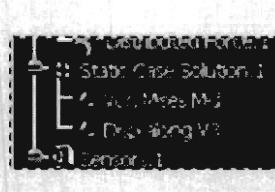
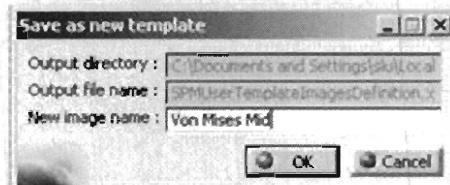
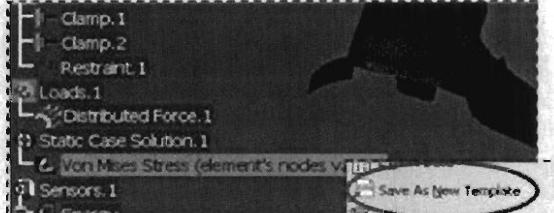
- Customise a standard image (ex. Von Mises with Mid Layer)



- This image is now available Through Generate Image List



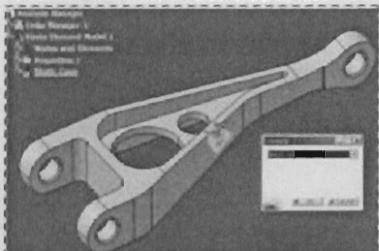
- Save it as New Template using its contextual menu and name it as 'Von Mises Mid'



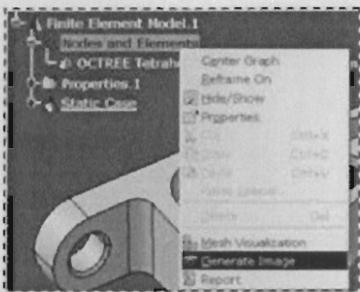
## 4.8. FEM Model Images

Images are available for Nodes and Elements

- If needed, generate the mesh: launch a computation for mesh only



- In Nodes and Elements Contextual Menu, select generate Image



- Display any available FEM image

| Image Names                   | Meaning                                   |
|-------------------------------|---|
| <b>Mesh</b>                   | Mesh                                      |
| <b>Elements text</b>          | Elements numbers                          |
| <b>Nodes text</b>             | Nodes numbers                             |
| <b>Degrees of freedom</b>     | Nodal symbol of fixed degrees of freedom  |
| <b>Local axis</b>             | Symbol of local axis                      |
| <b>Physical type fringe</b>   | Fringe image of the element physical type |
| <b>Coordinate symbol node</b> | Nodal coordinate symbol                   |

## 5. EST Functionalities (1/3)

All the CATIA functionalities are available with EST, except Property connections.

There are functionalities which run ONLY with EST :

- Buckling case.
- Inserting a new static case and a new frequency case (except for inserting a first static case and a first frequency case).
- Distributed mass visualization on mesh.
- Line mass density visualization on mesh.
- Surface mass density visualization on mesh
- Generate Image of masses objects.
- Basic analysis report of masses objects.
- Restraint visualization on mesh.
- Generate Image of restraint objects.
- Basic analysis report of restraint objects.
- Load visualization on mesh.
- Generate Image of load objects.
- Basic analysis report on load objects.
- Analysis Cases Solution Selection.
- Definition parameters of an Analysis Case.
- Generate Image of Static Analysis objects.
- Global analysis report of an Analysis Case.

## EST Functionalities (2/3)



There are functionalities which run ONLY with EST :

- Report on a deformed mesh.
- FEM Image editor.
- Selections tab in Image Edition for von Mises and Principal stresses.
- Image Iso Fringe Editor.
- Report on a Stress Von Mises / Principal stresses feature.
- Selections tab in Symbol Editor.
- Image Axis System.
- Report on a translational displacement vector.
- Report on Estimated local error feature.
- Advanced reporting.
- Static constrained case
- Combined case
- Imported composite property
- Groups functionalities
- Periodicity condition
- Inertia on virtual parts
- Bearing Load, Imported forces, Imported moments
- Force density
- Creating temperature field

## EST Functionalities (3/3)



There are functionalities which run ONLY with EST :

- Advanced reporting
- Image layout
- Simplified representation
- Local 2D property
- Local beam property
- Local adaptivity

## To Sum Up ...

In this course you have seen :

- Extra tools offered with advanced option EST :
- Advanced Pre-Processing
- Multi Analysis case management (including Buckling)
- Advanced Post-Processing

## Frequently Asked Questions

- ? Is the Contact Analysis no linear one
  - For the moment, contact analysis is linear (without friction effects).
- ? How to share analysis results
  - You can generate an html report, and users can specify its content thanks to Advanced Reporting function.
- ? How to launch a batch analysis
  - You can launch batch analysis with a macro written in VB Script.