MSC.Software Corporation

2 MacArthur Place
Santa Ana, CA 92707, USA
Tel: (714) 540-8900
Fax: (714) 784-4056
Web: http://www.mscsoftware.com

Tokyo, Japan
Tel: 81-3-3505-0266
Fax: 81-3-3505-0914

United States
MSC.Patran Support
Tel: 1-800-732-7284
Fax: (714) 979-2990

Munich, Germany
Tel: (+49)-89-43 19 87 0
Fax: (+49)-89-43 61 716

CATIA V5 Structural Analysis for the Designer

CAT509 Workshops

March 2002

Part Number: DAS*V2002*Z*Z*Z*SM-CAT509-WBK

DISCLAIMER

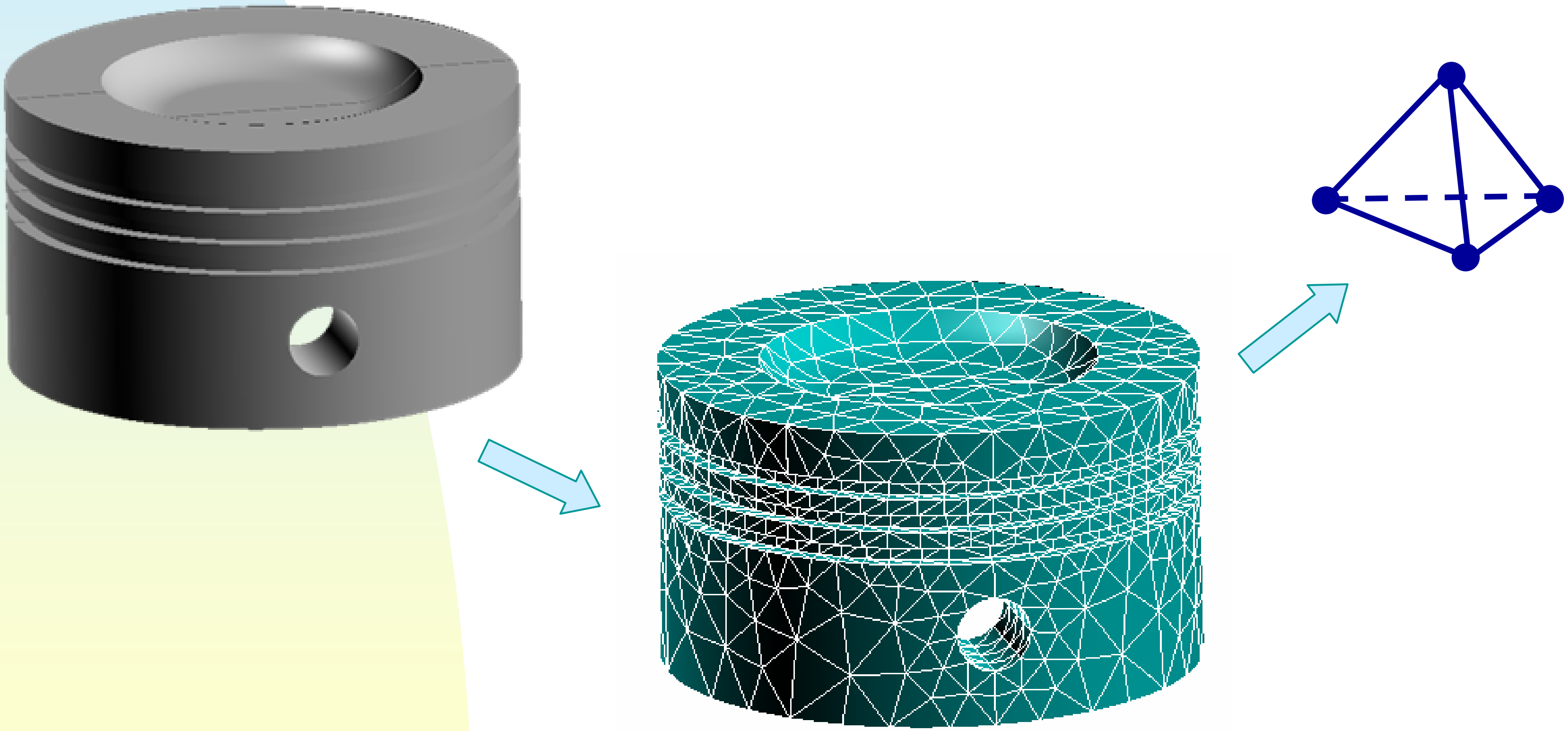
MSC.Software Corporation reserves the right to make changes in specifications and other information contained in this document without prior notice.

The concepts, methods, and examples presented in this text are for illustrative and educational purposes only, and are not intended to be exhaustive or to apply to any particular engineering problem or design. MSC.Software Corporation assumes no liability or responsibility to any person or company for direct or indirect damages resulting from the use of any information contained herein.

TABLE OF CONTENTS

WORKSHOP 1

FEM REVIEW



- Quiz yourself on the FEM:

1. How can preliminary structural analysis improve the design process?

2. Briefly describe the Finite Element Method (FEM).

3. Simple pieces that represent a more complex structure are called ____________ ____________ .

4. The simple pieces mentioned above are connected together at ____________ .

5. The assembly of #3 and #4 is called a ___________ ___________ ____________ .

- Quiz yourself on FEA:

1. What are the six main steps in pre-processing a finite element analysis (FEA)?

2. Name a load type that would be applied in FEA.

3. Name a constraint (restraint) type that would be applied in FEA.

4. What step in FEA comes between pre- and post-processing?

5. What are the two main steps in FEA post-processing?

6. How are FEA results displayed?

- Quiz yourself on CATIA structural analysis:

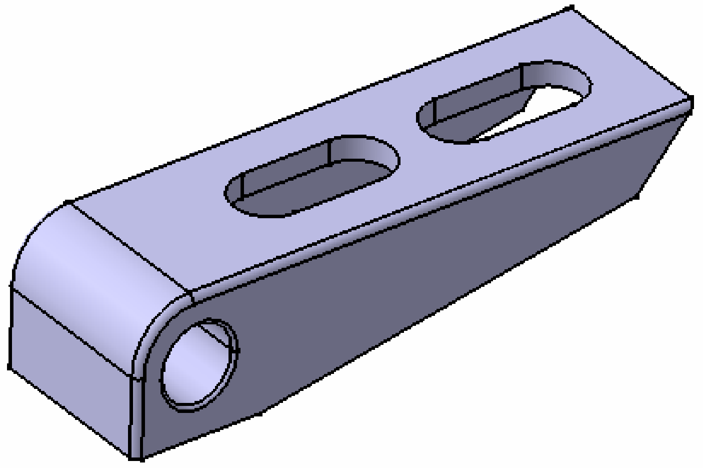
1. What are the 3 types of analysis supported by the CATIA structural analysis tools?

2. Write the name or sketch at least one linear and one parabolic element supported by the CATIA structural analysis tools.

3. What is the name of your instructor? (extra credit)

WORKSHOP 2

FOOT PEG



Problem Description

- A new All Terrain Vehicle (ATV) is being designed to carry two people – a driver and a passenger. An area of concern is the Foot Peg for the passenger on the ATV. The Foot Peg needs to be small due to limited space on the ATV yet able to handle the force of the passenger during the ride.

- Analyze the Foot Peg as an aluminum part in the preliminary design phase to check for part failure in a static condition.

Suggested Exercise Steps

1. Open the existing CATIA part in the Part Design workbench.
2. Apply aluminum material properties to the part.
3. Create a new CATIA analysis document (.CATAnalysis).
4. Apply the restraint condition.
5. Apply the load condition.
6. Compute the analysis.
7. Visualize the analysis results.
8. Save the analysis document.

Step 1. Open the part

Open the Foot Peg part in the Part Design workbench.

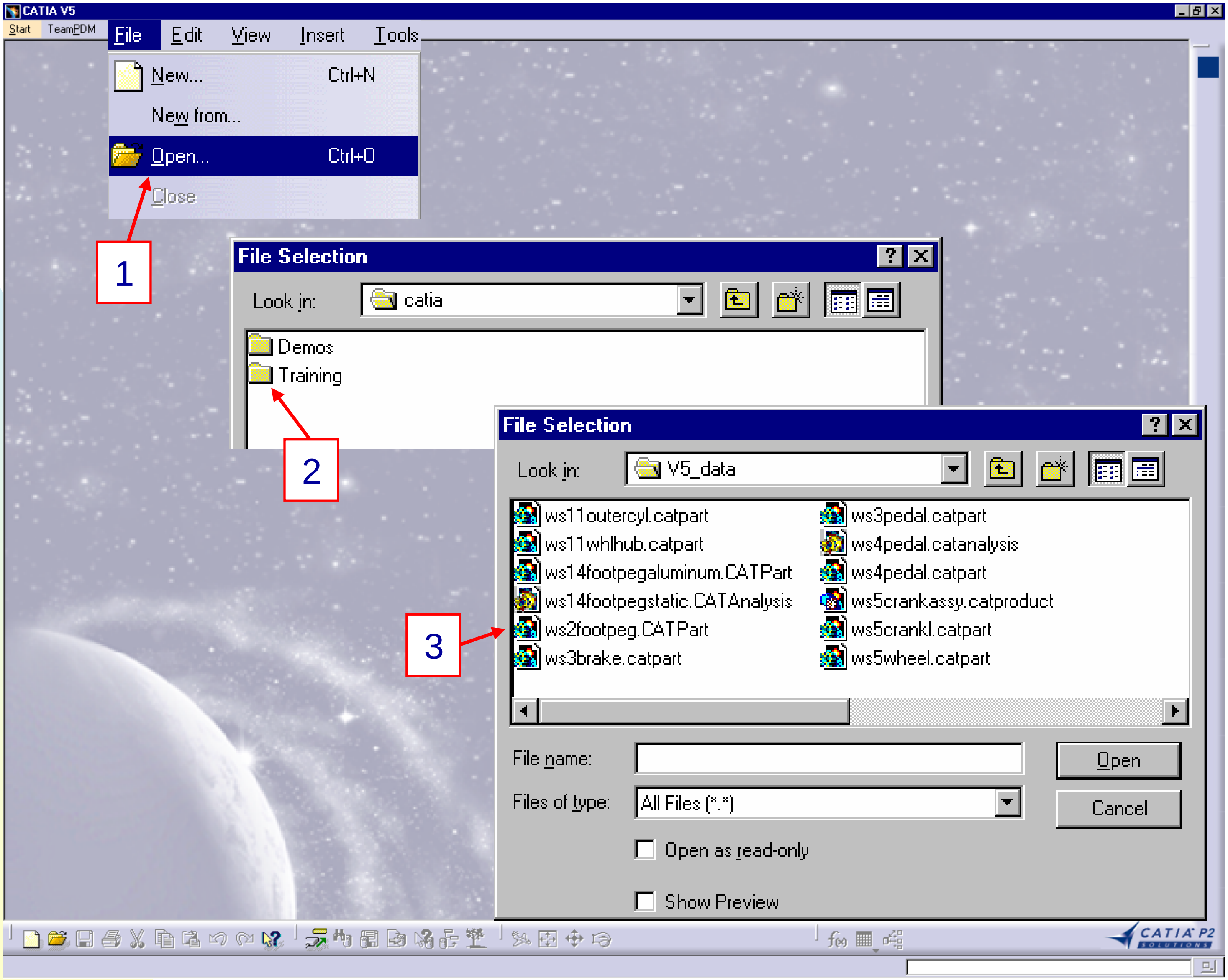
Steps:

1. Select File and Open… from the top pull-down menu.

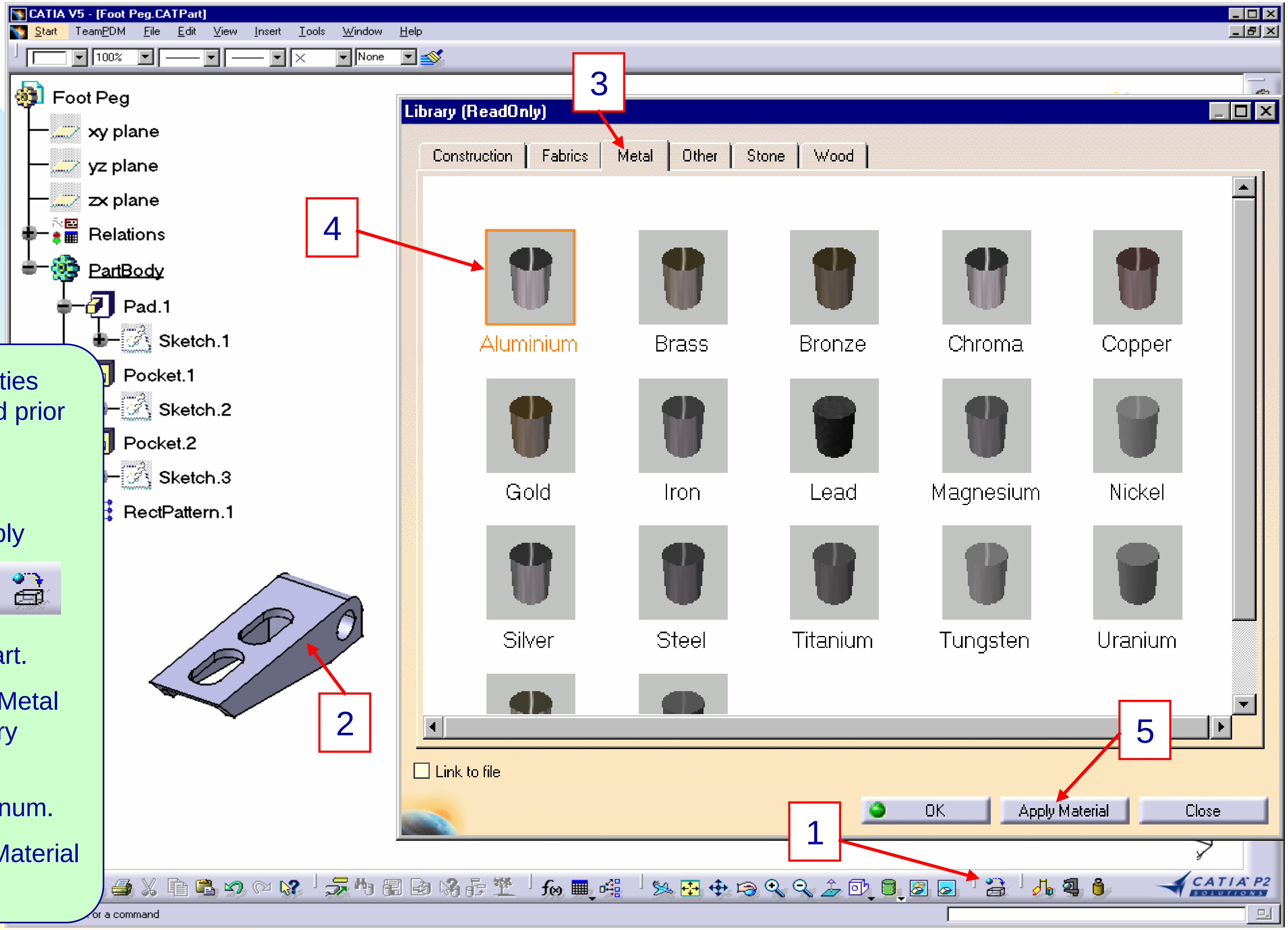
2. Access the class workshop directory using the typical Windows interface.

3. Open the ws2footpeg.CATPart by double-clicking.

By default, the Foot Peg and any other CATPart document is opened in Part Design workbench.



Step 2. Apply material properties



Material properties must be applied prior to analysis.

Steps:

1. Click the Apply Material icon.

2. Select the part.

3. Activate the Metal tab in the Library window.

4. Select Aluminum.

5. Click Apply Material button…OK.

Step 2. Apply material properties

Apply the customized render mode to view the Aluminum material display.

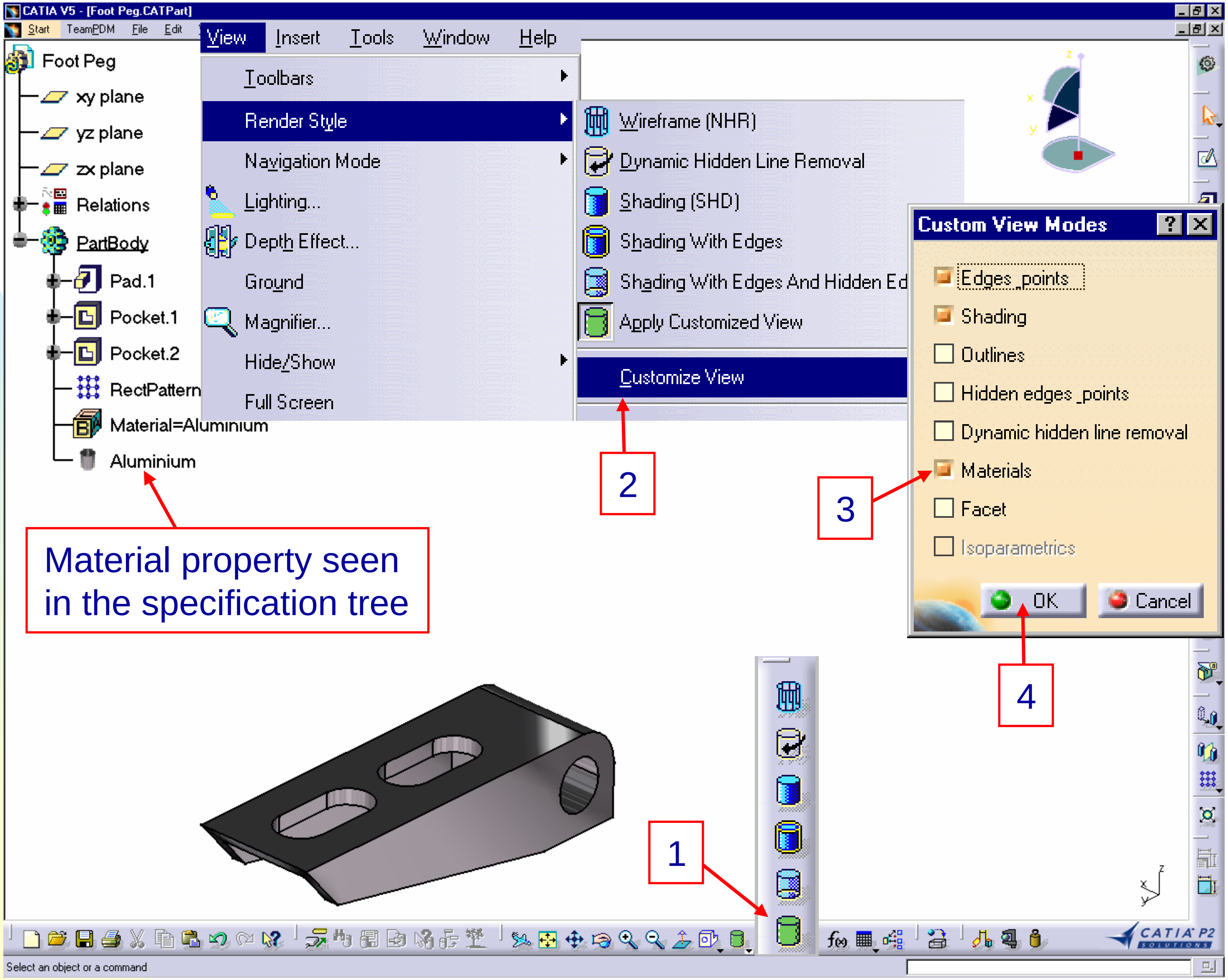
Steps:

1. Click the Customized View Parameters icon.

2. If material display is not seen, select Customize View under Render Style from the View pull-down menu.

3. Activate the Materials box in the Custom View Modes definition window.

4. Click OK.



Step 3. Create analysis document

Create a CATAnalysis document that will contain the information for our static analysis of the Foot Peg.

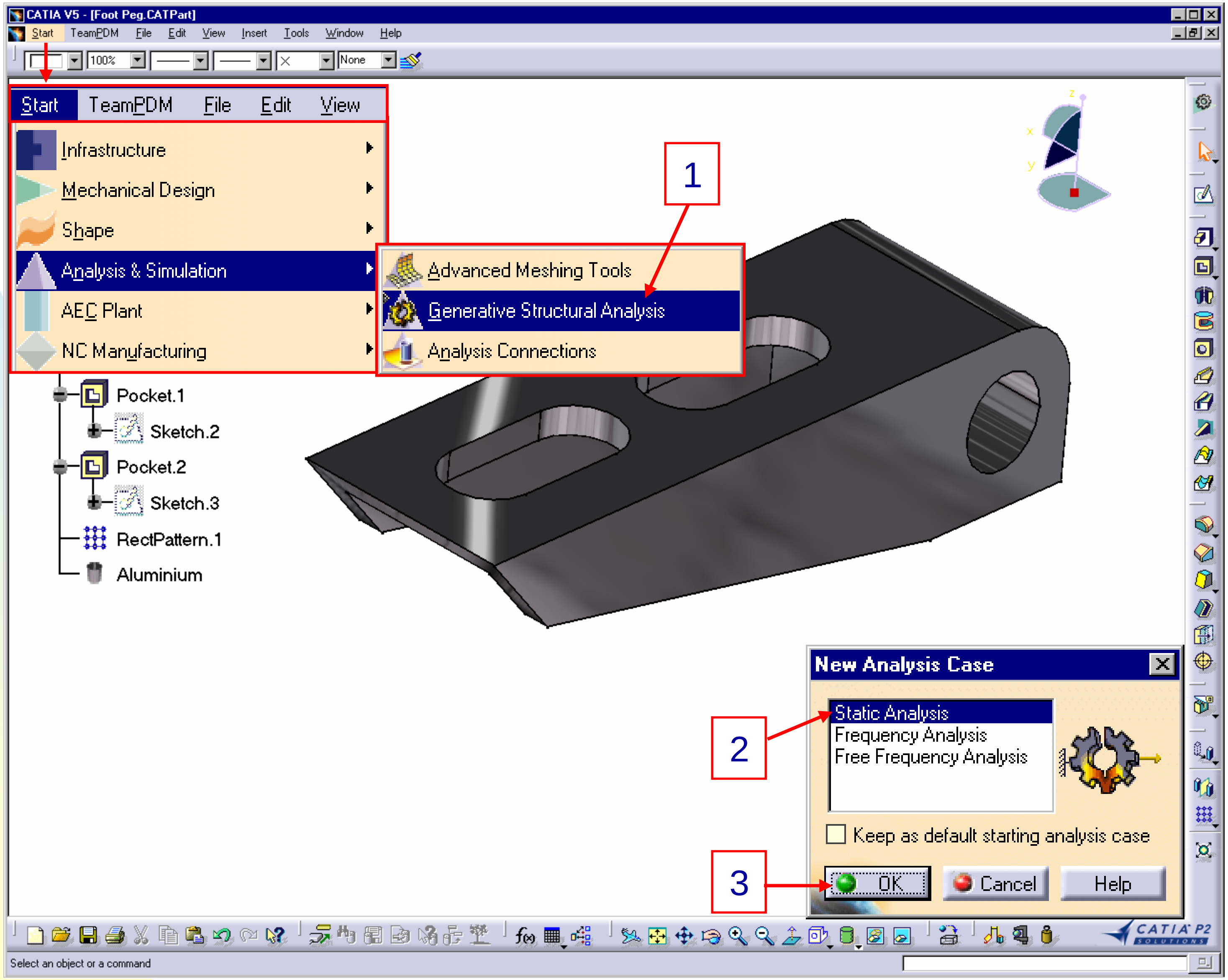
Steps:

1. Select the GSA workbench from the Start menu.

2. Highlight the Static Analysis case.

3. Click OK.

The new CATAnalysis document is now active in the GSA workbench.

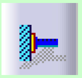


Step 4. Apply restraint condition

Apply a clamp restraint to the rear face of the Foot Peg to represent clamping (no motion) of the part at that face.

Steps:

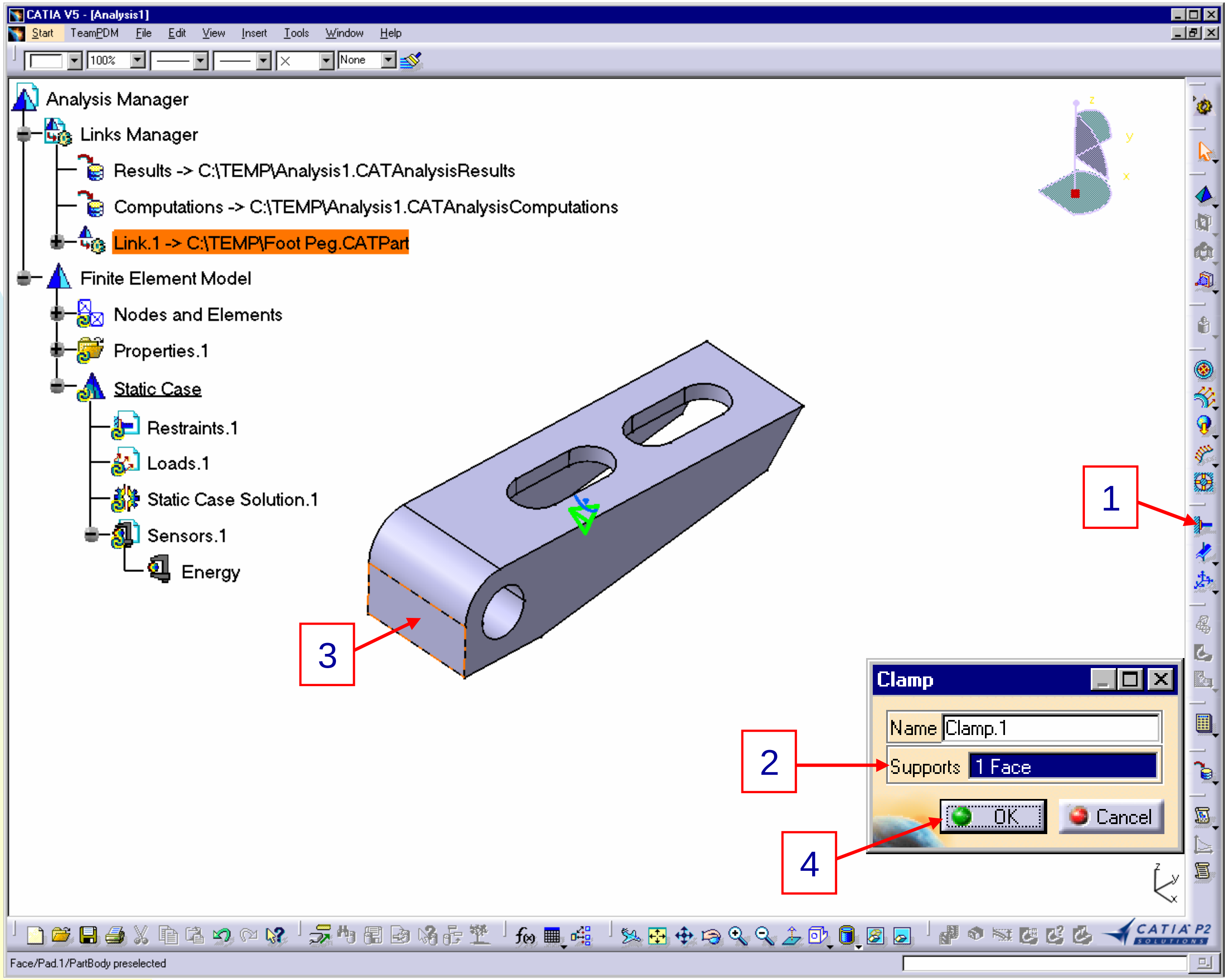
1. Select the clamp icon from the GSA workbench.



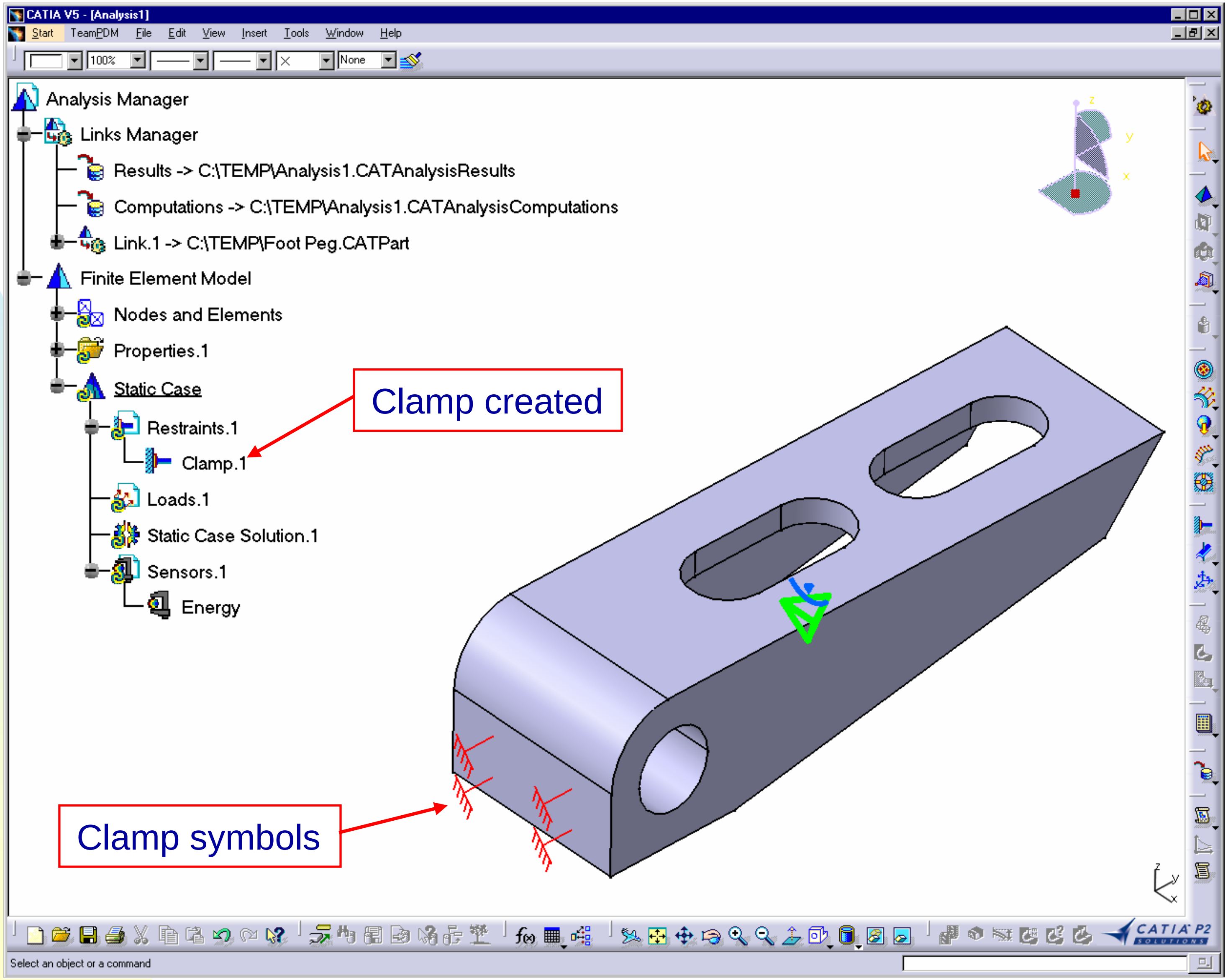
2. Be sure Supports field is highlighted.

3. Select geometry to clamp (rear face).

4. Click OK.



Step 4. Apply restraint condition



The clamp restraint is created and seen in the specification tree.

Symbols appear on the part showing the clamp restraint applied to the rear surface of the Foot Peg.

Step 5. Apply load condition

Apply a force load of 550lbf to the Foot Peg top face in a direction normal to the face pushing downward.

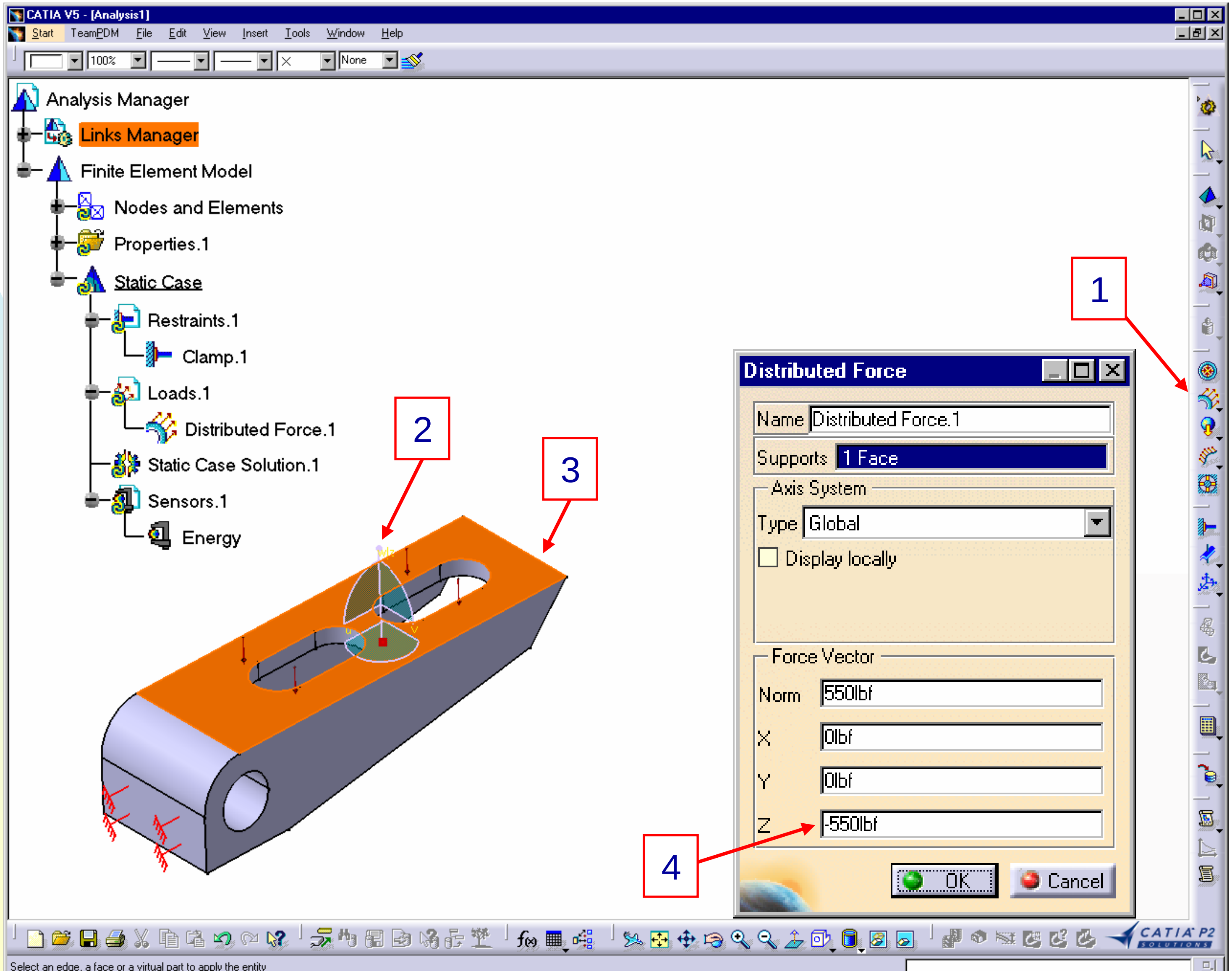
Steps:

1. Select the force icon from the GSA workbench. 

2. Drag and drop the compass on to the top face to establish an axis system normal to the face.

3. The top face is highlighted and force vectors shown.

4. Key in value -550lbf for the Z vector…OK.

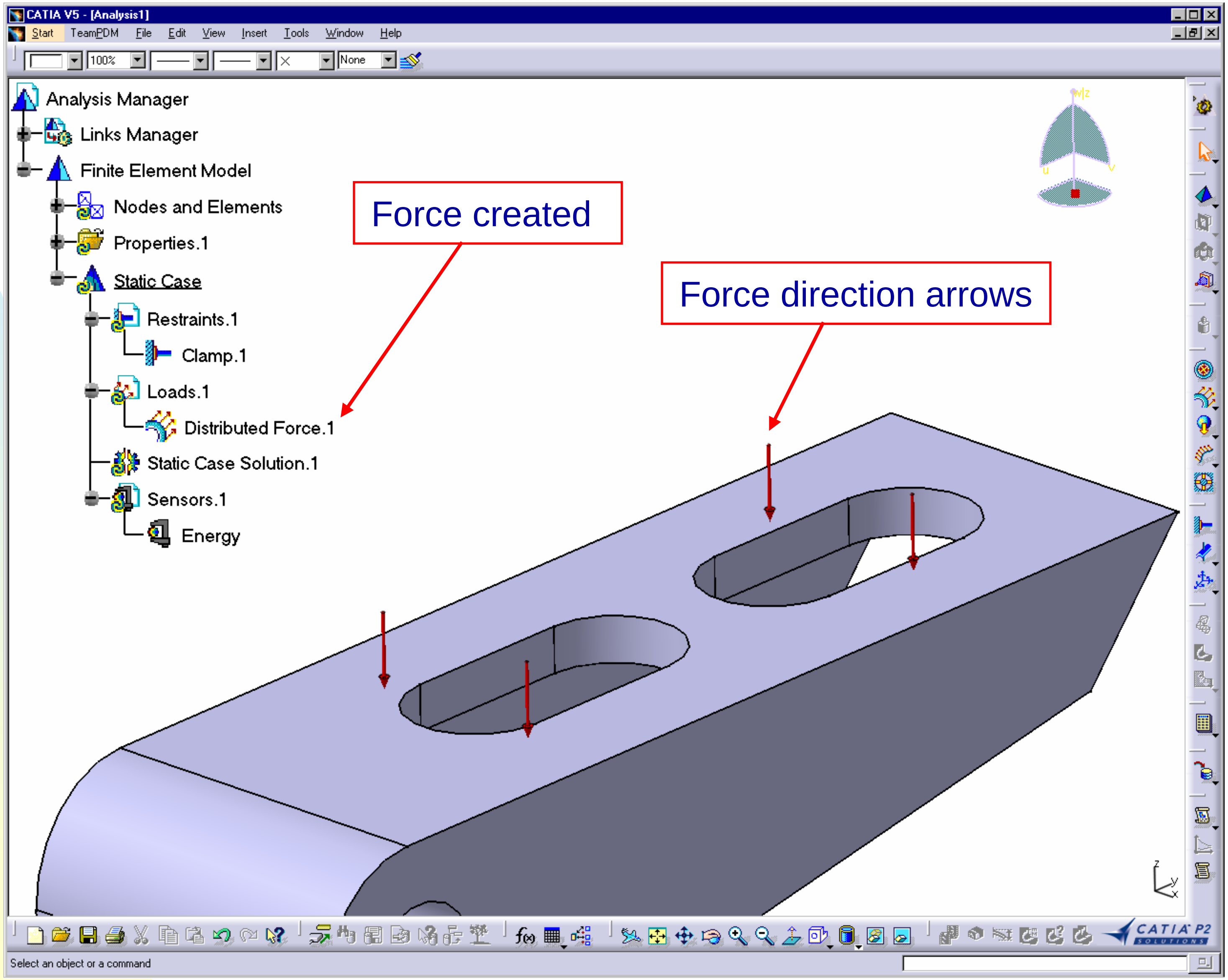


Step 5. Apply load condition

The force load is created and seen in the specification tree.

The force load is applied to the top face in a downward direction as shown by the vector arrows.

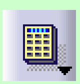
Hint: Drag and drop the compass back to its normal position away from the part after use.



Step 6. Compute the analysis

After restraint and load conditions are applied, the analysis can be computed.

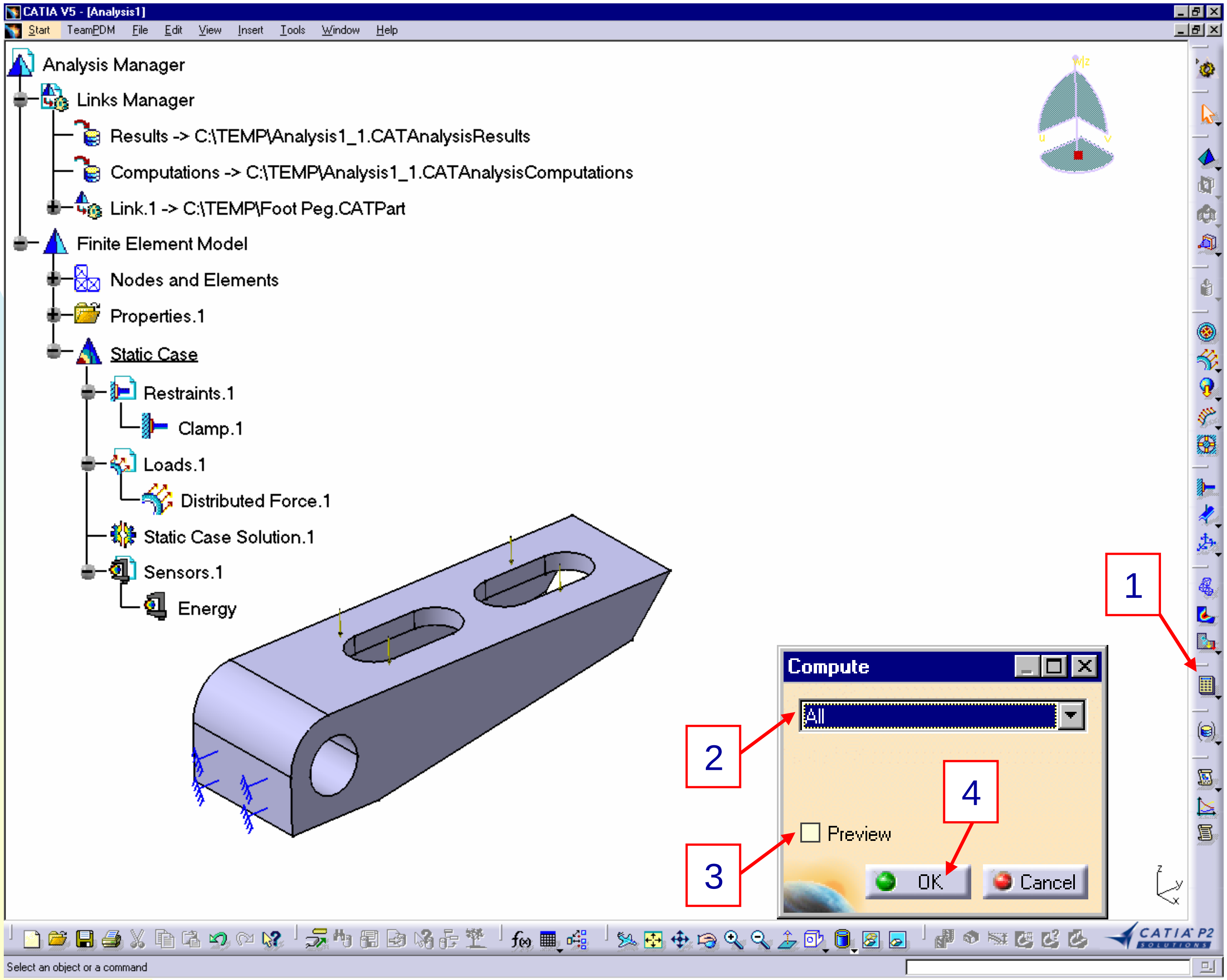
Steps:

1. Select the Compute icon. 

2. Specify that All parameters should be used in the calculation.

3. Verify that the Preview box is **not checked**.

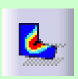
4. Click OK.



Step 7. Visualize analysis results

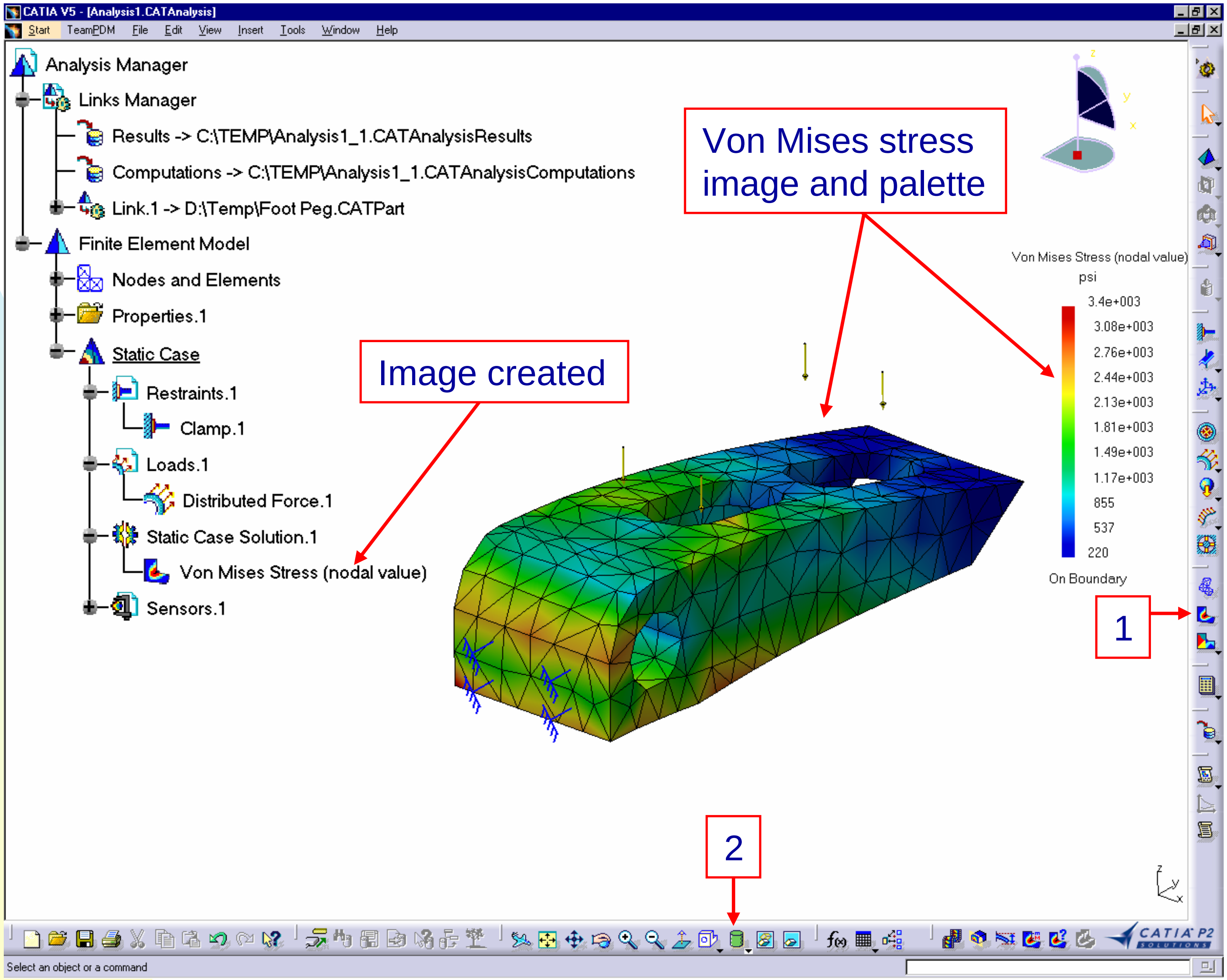
To visualize results, select the desired image. In this case, we want to see the Von Mises stress.

Steps:

1. Select the Von Mises stress image icon. 

2. Verify that the Customized View Parameters icon is active. 

The Von Mises stress image is displayed with color palette when the custom view mode is active.



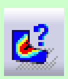
MSC SOFTWARE

Step 7. Visualize analysis results

For detailed results, query the maximum Von Mises stress values for the analysis.

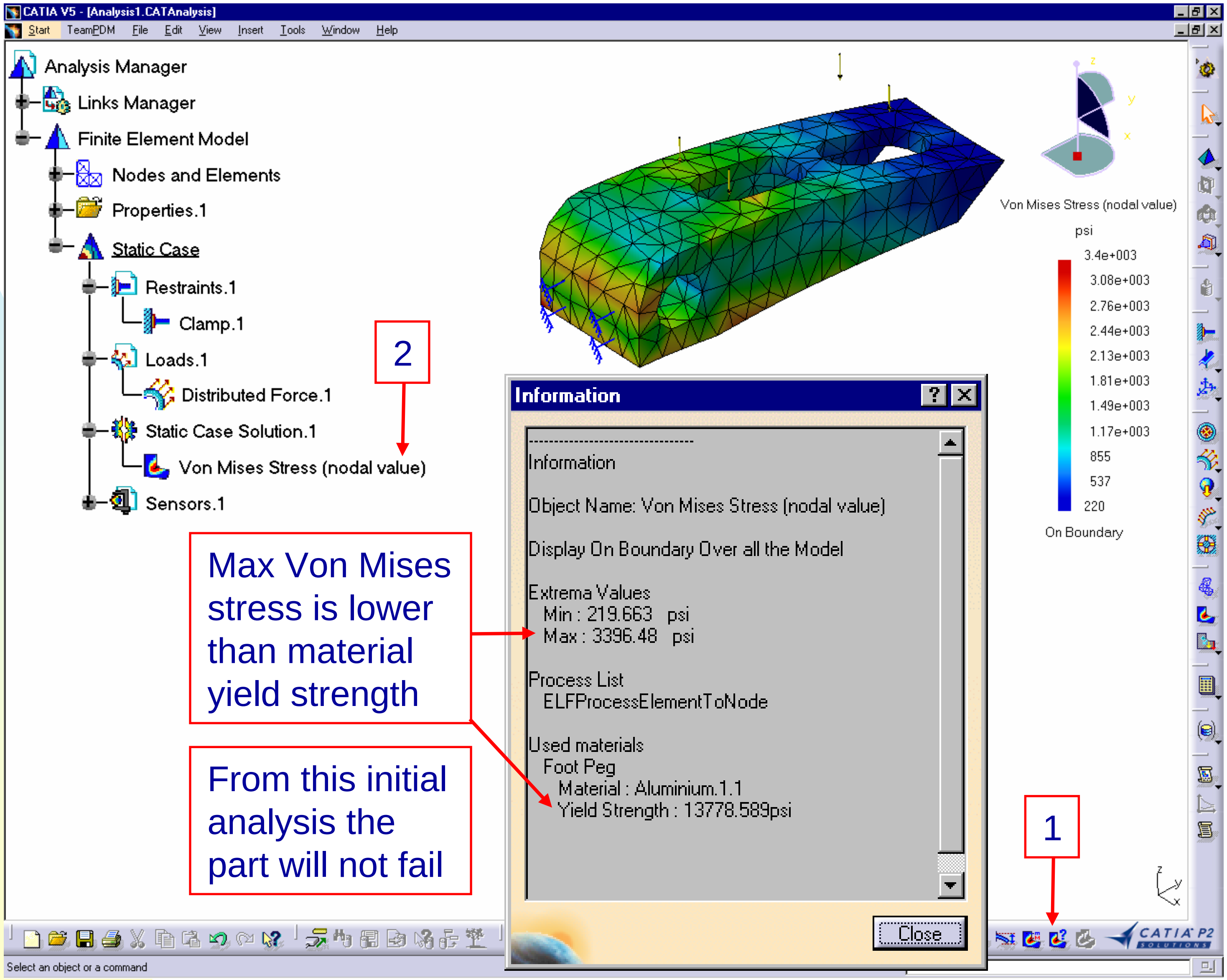
Steps:

1. Select the Informations icon.



2. Select the Von Mises stress image if necessary.

The information window shows the minimum and maximum Von Mises stress values as well as the material yield strength of aluminum.



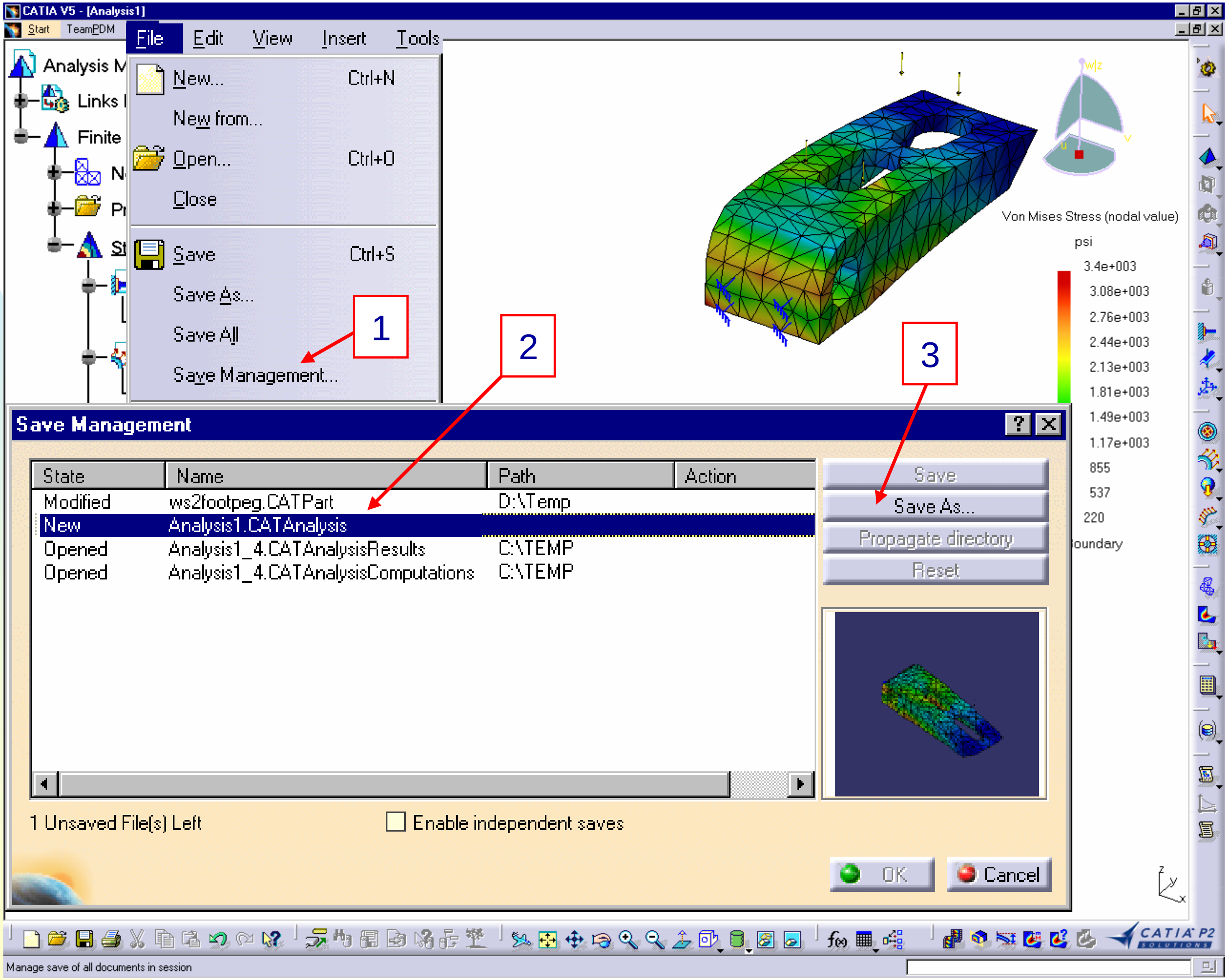
2

1

Max Von Mises stress is lower than material yield strength

From this initial analysis the part will not fail

Step 8. Save analysis document



Save the analysis document.

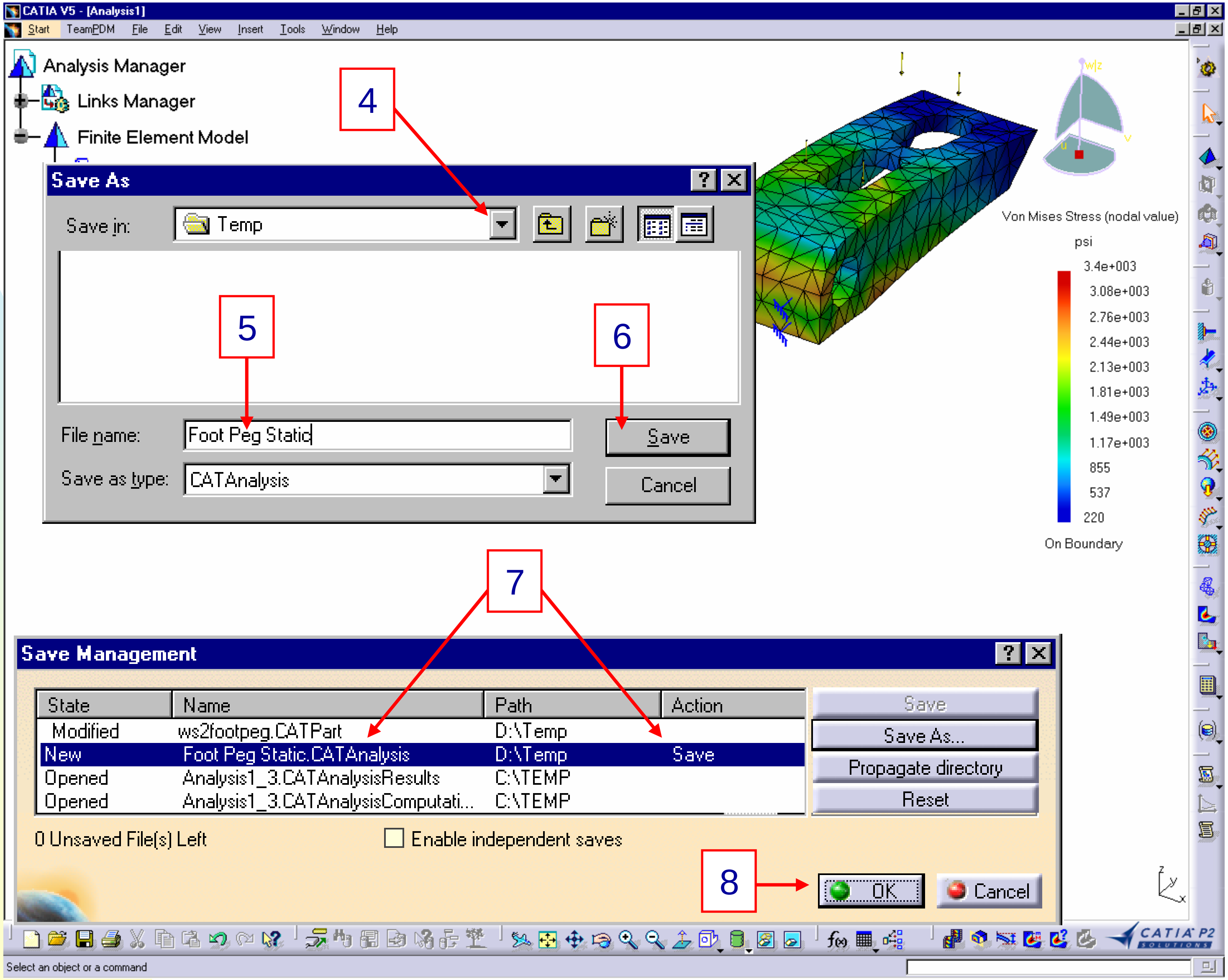
Steps:

1. Select Save Management from the File pull-down menu.

2. Highlight the CATAnalysis document in the list.

3. Click the Save As… button.

Step 8. Save analysis document



Steps:

4. Select the directory path.

5. Key in Foot Peg Static for the analysis document name.

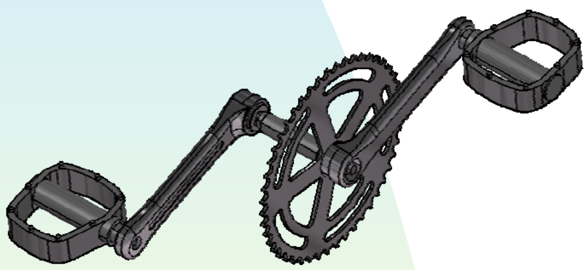
6. Click Save.

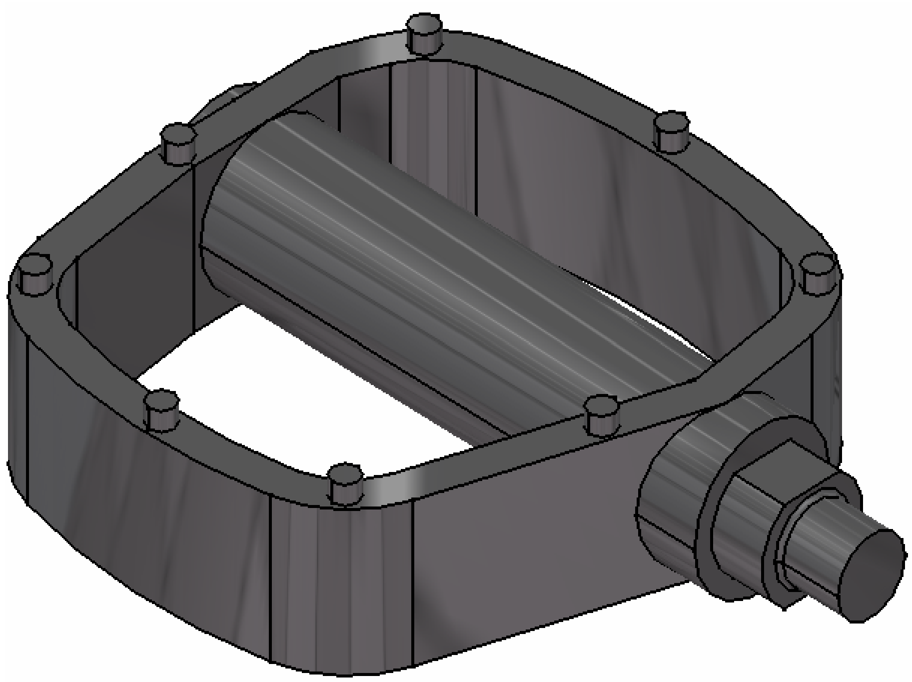
7. Notice the new name and Action "Save" for the analysis document.

8. Click OK to execute the noted Actions.

WORKSHOP 3

BICYCLE PEDAL STATIC ANALYSIS

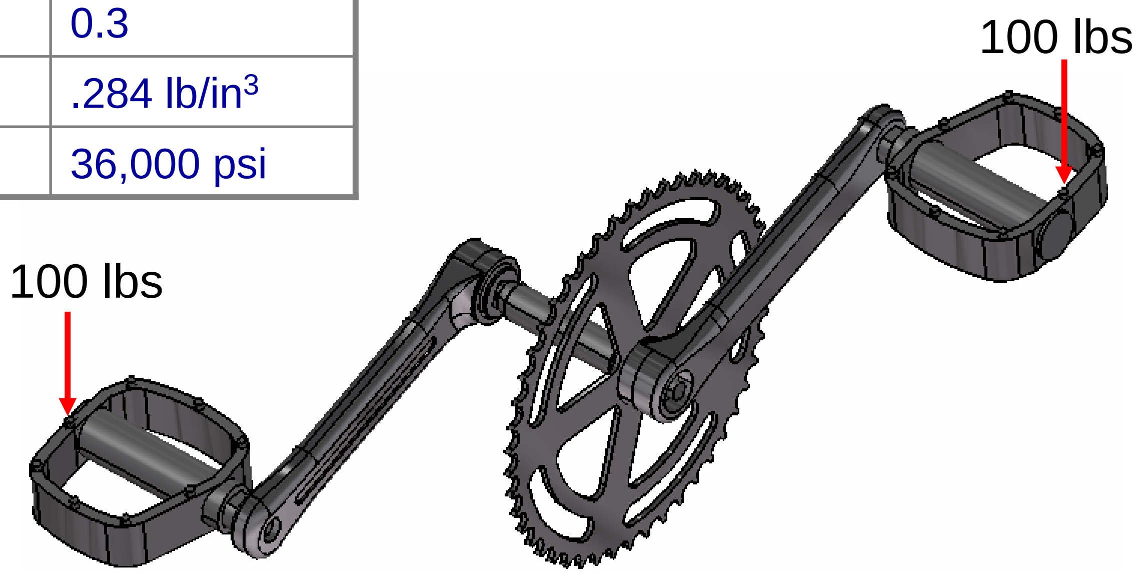




Problem Description

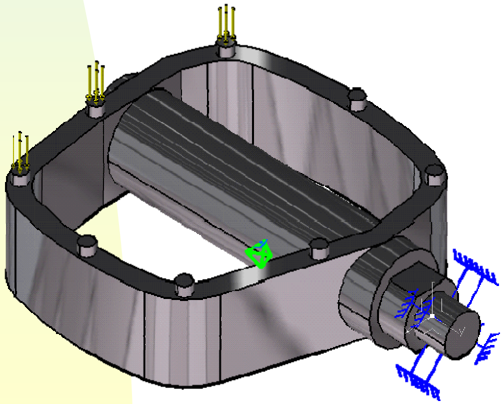
- Your job will be to analyze various components of a mountain bicycle. We will start with the pedal.
- Let's assume a 200 lb person riding this bike is standing, balanced evenly on each pedal. Material (Steel) properties as specified below. Use a rough analysis to determine where the high stress areas exist that will require additional mesh refinement.

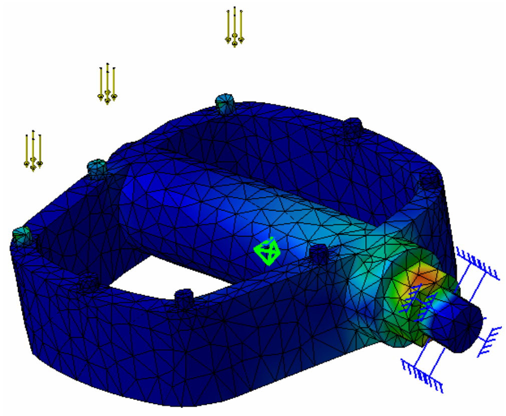
Elastic Modulus, E	29.0E6 psi
Poisson's Ratio, ν	0.3
Density	.284 lb/in^3
Yield Strength	36,000 psi



Suggested Exercise Steps

1. Open the existing CATIA part in the Part Design workbench.
2. Apply steel material properties to the part.
3. Create a new CATIA analysis document (.CATAnalysis).
4. Pre-process initial finite element mesh.
5. Apply a clamp restraint.
6. Apply a distributed force.
7. Compute the analysis.
8. Visualize the analysis results.
9. Save the analysis document.





Step 1. Open the existing CATIA part

Open the CATIA part ws3pedal.CATPart in the Part Design workbench.

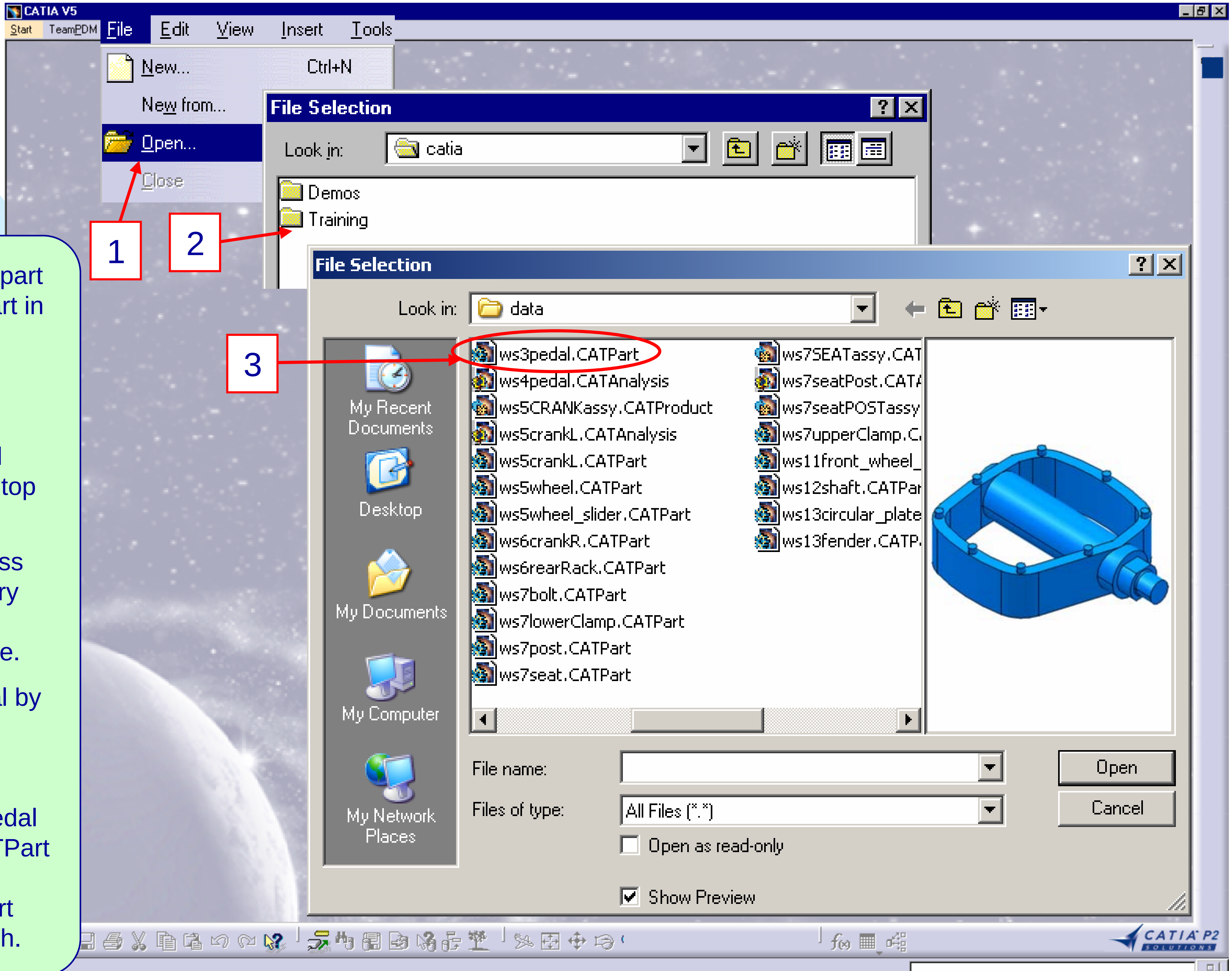
Steps:

1. Select File and Open… from the top pull-down menu.

2. Access the class workshop directory using the typical Windows interface.

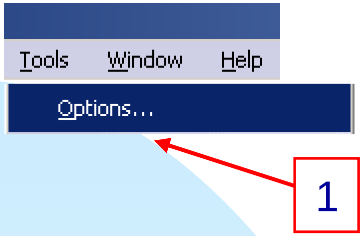
3. Open the pedal by double-clicking.

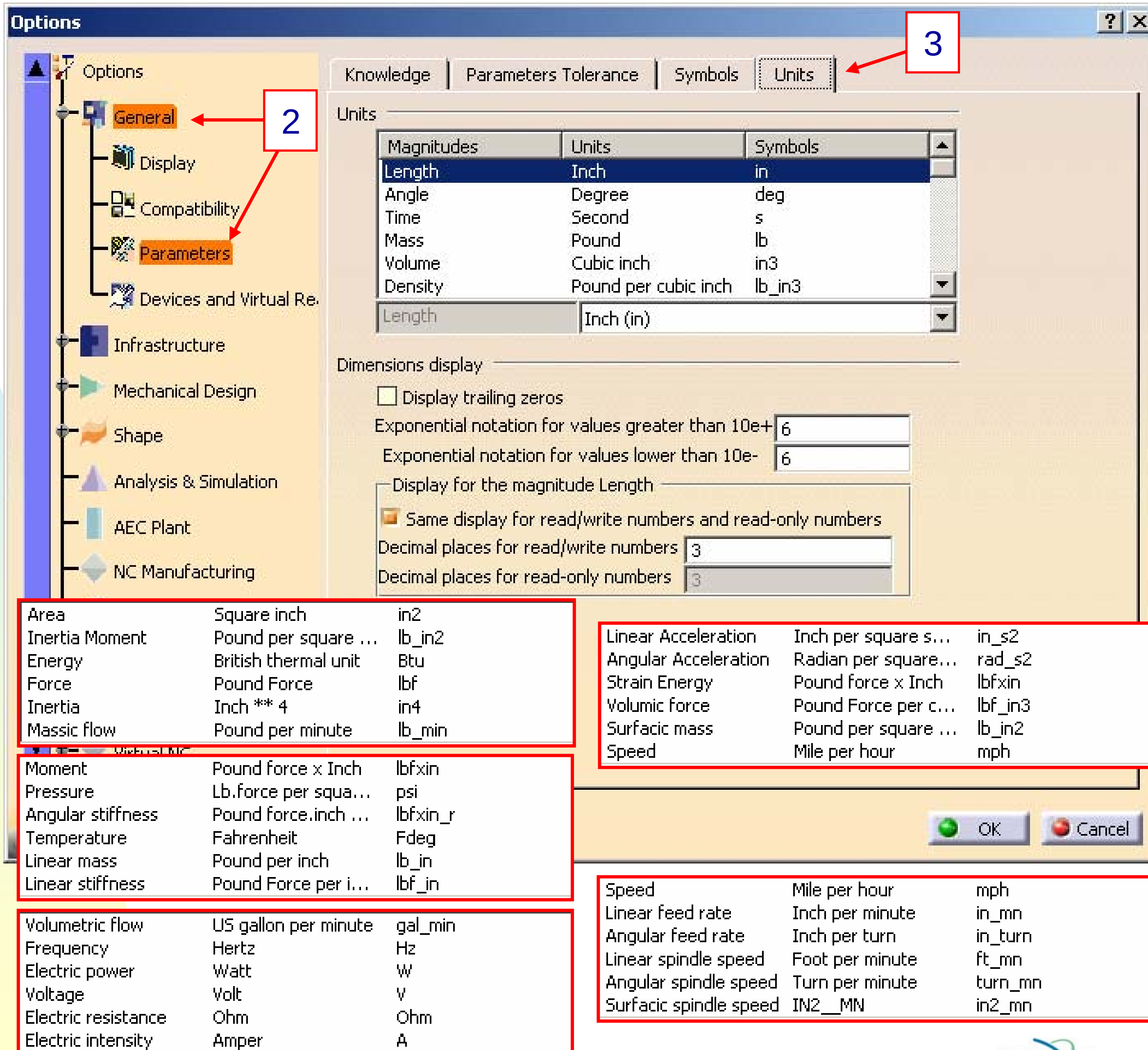
By default, the pedal and all other CATPart documents are opened in the Part Design workbench.



MSC SOFTWARE

Step 2. Apply steel material properties to the part





Before every session you should verify your session units.

Steps:

1. Select Tools from the menu then Options.

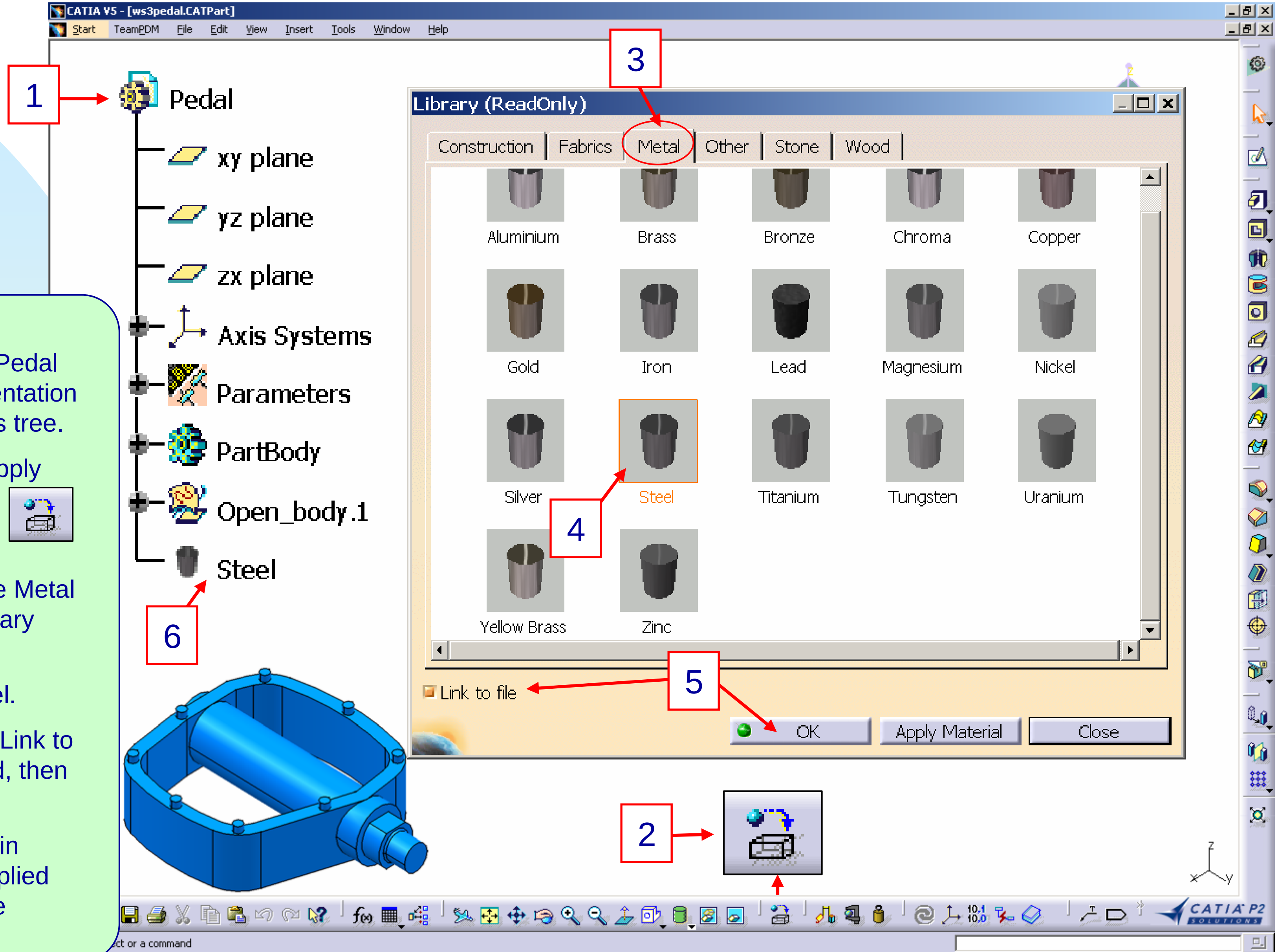
2. Select the General category then Parameters.

3. Select “Units” tab, change all units to the English system.

Notice there are many variables accessed by a scroll bar, verify and edit until all units are consistent. You must change each one separately, select OK.

MSC SOFTWARE

Step 2. Apply steel material properties to the part



Steps:

1. Select the Pedal "Part" representation in the features tree.

2. Click the Apply Material icon.

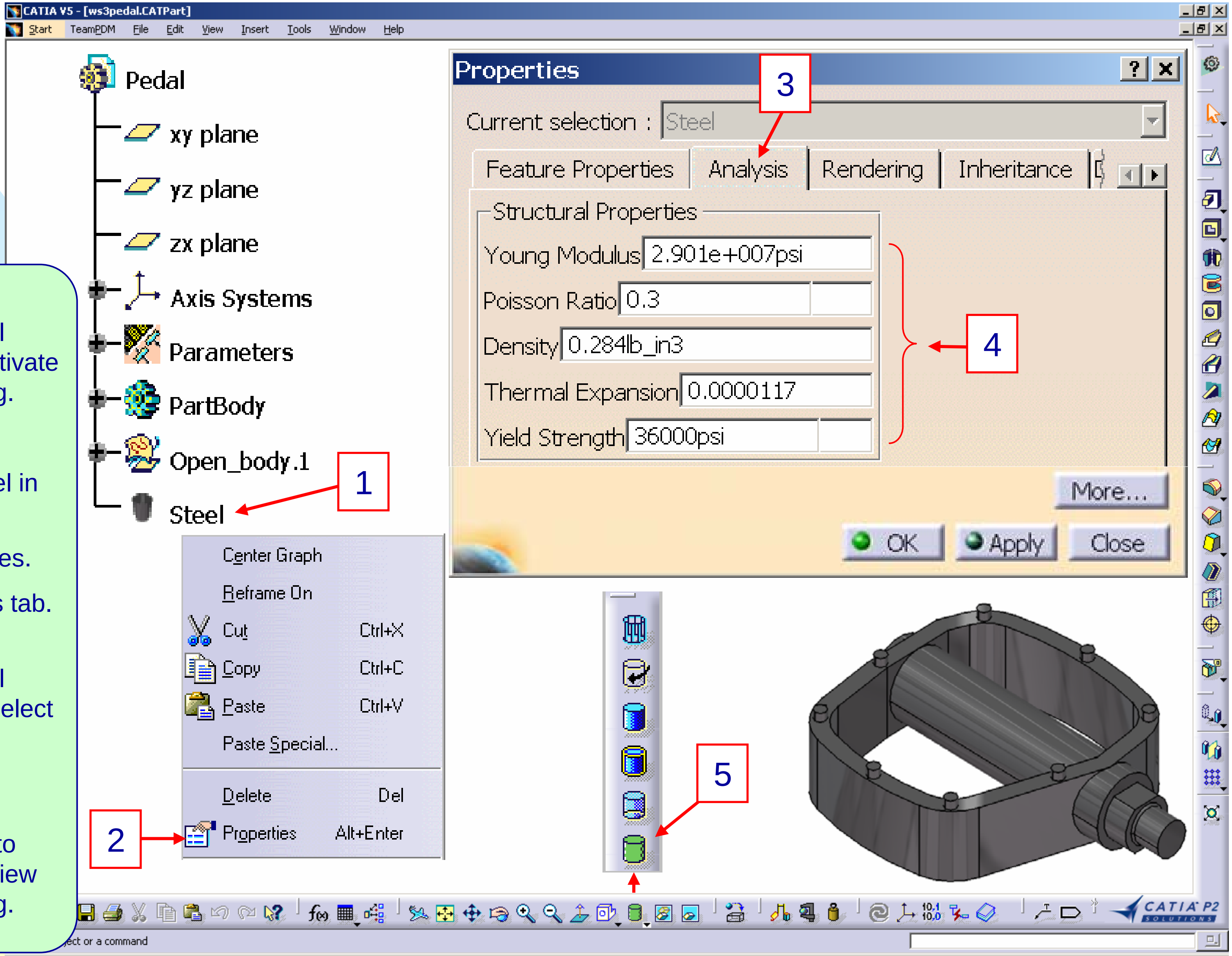
3. Activate the Metal tab in the Library window.

4. Select Steel.

5. Make sure Link to file is selected, then select OK.

6. Make certain material is applied properly in the features tree.

Step 2. Apply steel material properties to the part



Verify and edit structural material properties and activate material rendering.

Steps:

1. Right click Steel in the features tree.

2. Select Properties.

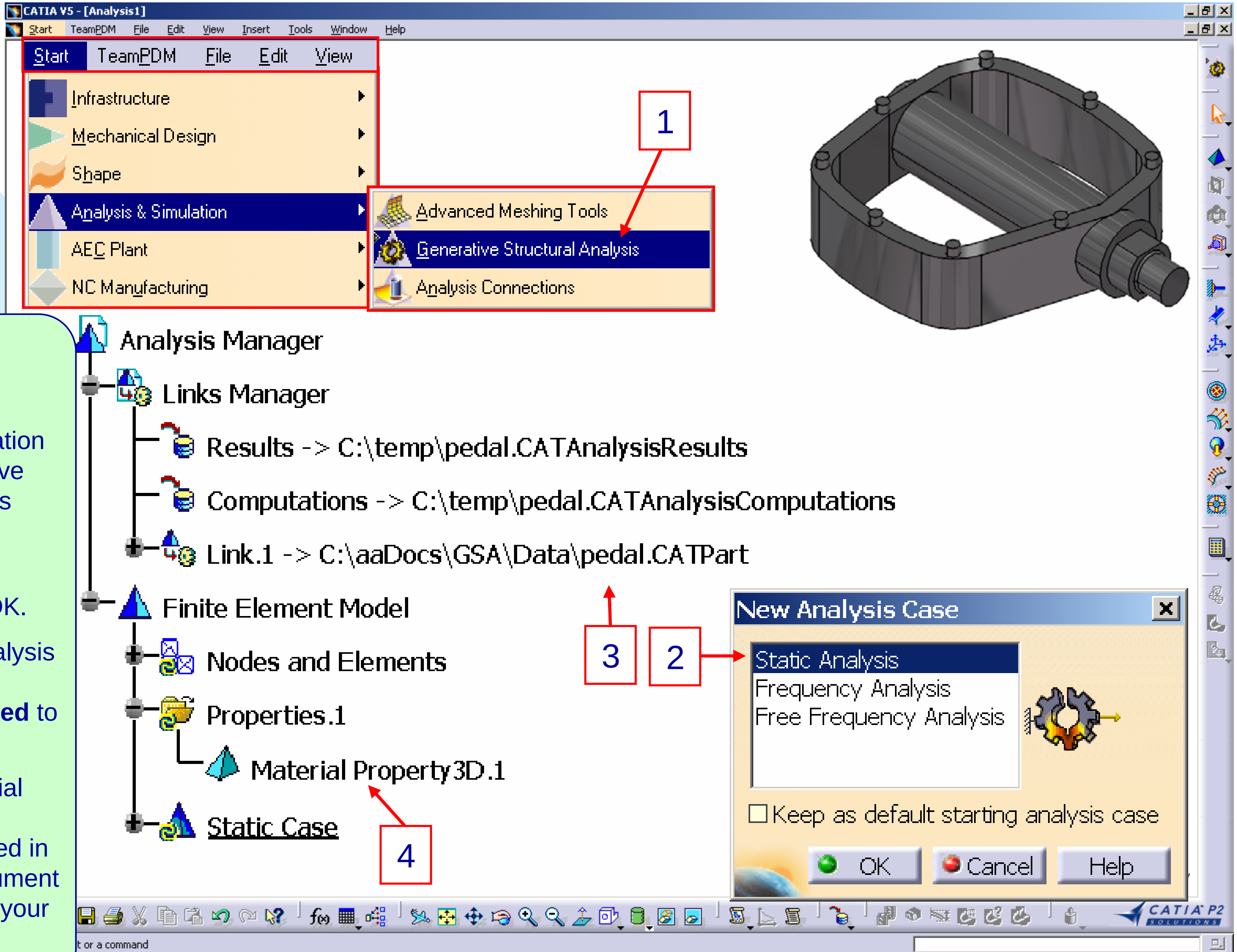
3. Select Analysis tab.

4. Verify and edit structural material properties here, select OK.

5. Click the Customized View Parameters icon to activate custom view material rendering.

MSC SOFTWARE

Step 3. Create a new CATIA analysis document



Steps:

1. From the Start menu select the Analysis & Simulation then the Generative Structural Analysis workbench.

2. Select Static Analysis, select OK.

3. Your Static Analysis document gets automatically **linked** to the CATPart.

4. Note the Material Property 3D.1 previously specified in the CATPart document shows up here in your CATAnalysis document.

Step 3. Create a new CATIA analysis document

Specify the External Storage directory locations.

Steps:

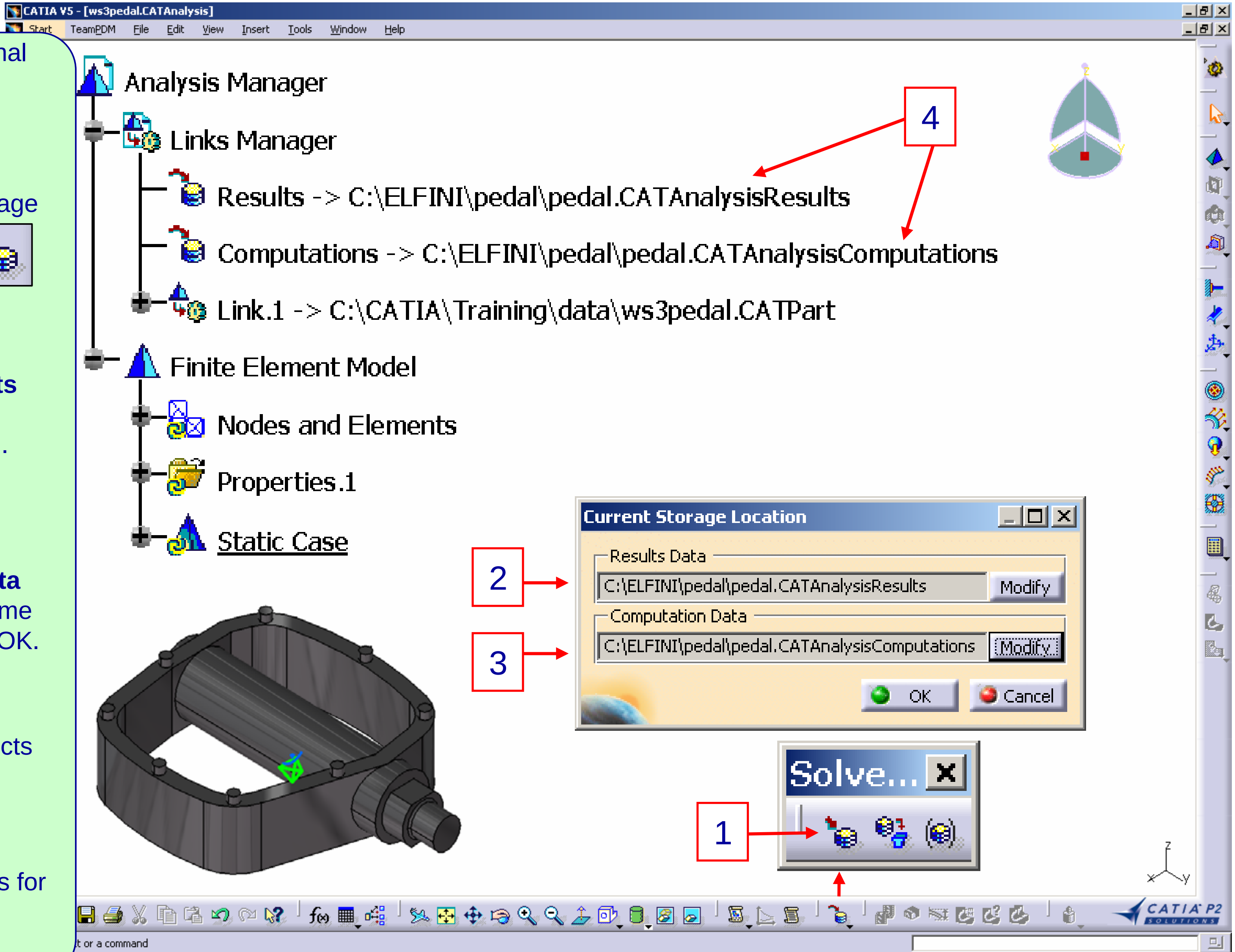
1. Select the Storage Location icon.

2. In the Current Storage Location modify the **Results Data** location and rename as shown.

3. In the Current Storage Location modify the **Computation Data l**ocation and rename as shown, select OK.

4. Note the Links Manager in the features tree reflects the paths.

You can create specific directories for additional organization.



MSC SOFTWARE

Step 4. Pre-process initial finite element mesh

Define the global finite element mesh properties.

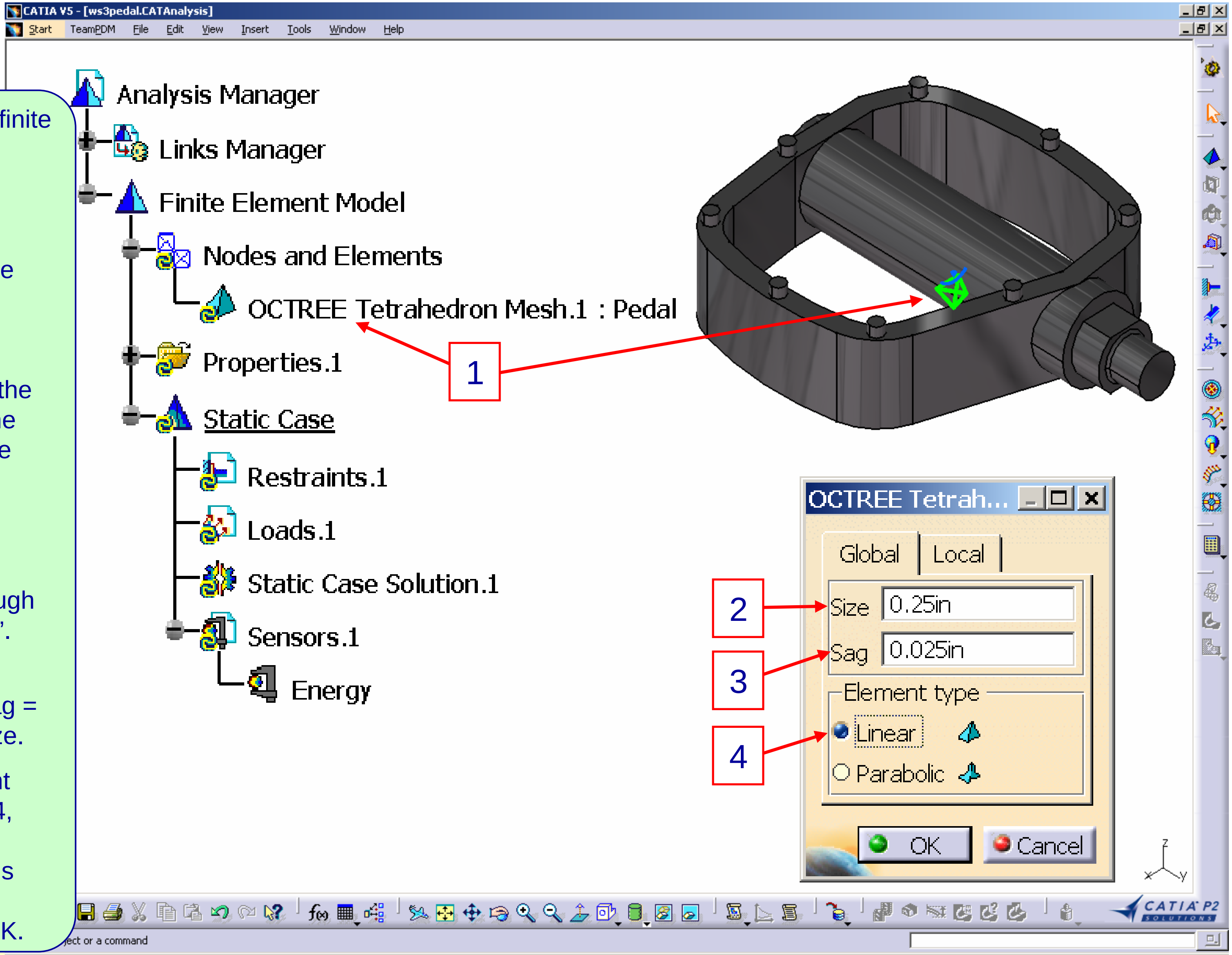
Steps:

1. Double Click the "OCTREE Tetrahedron Mesh.1:Pedal" representation in the features tree or the "Mesh" icon on the part.

2. Specify the recommended rough Global Size = .25".

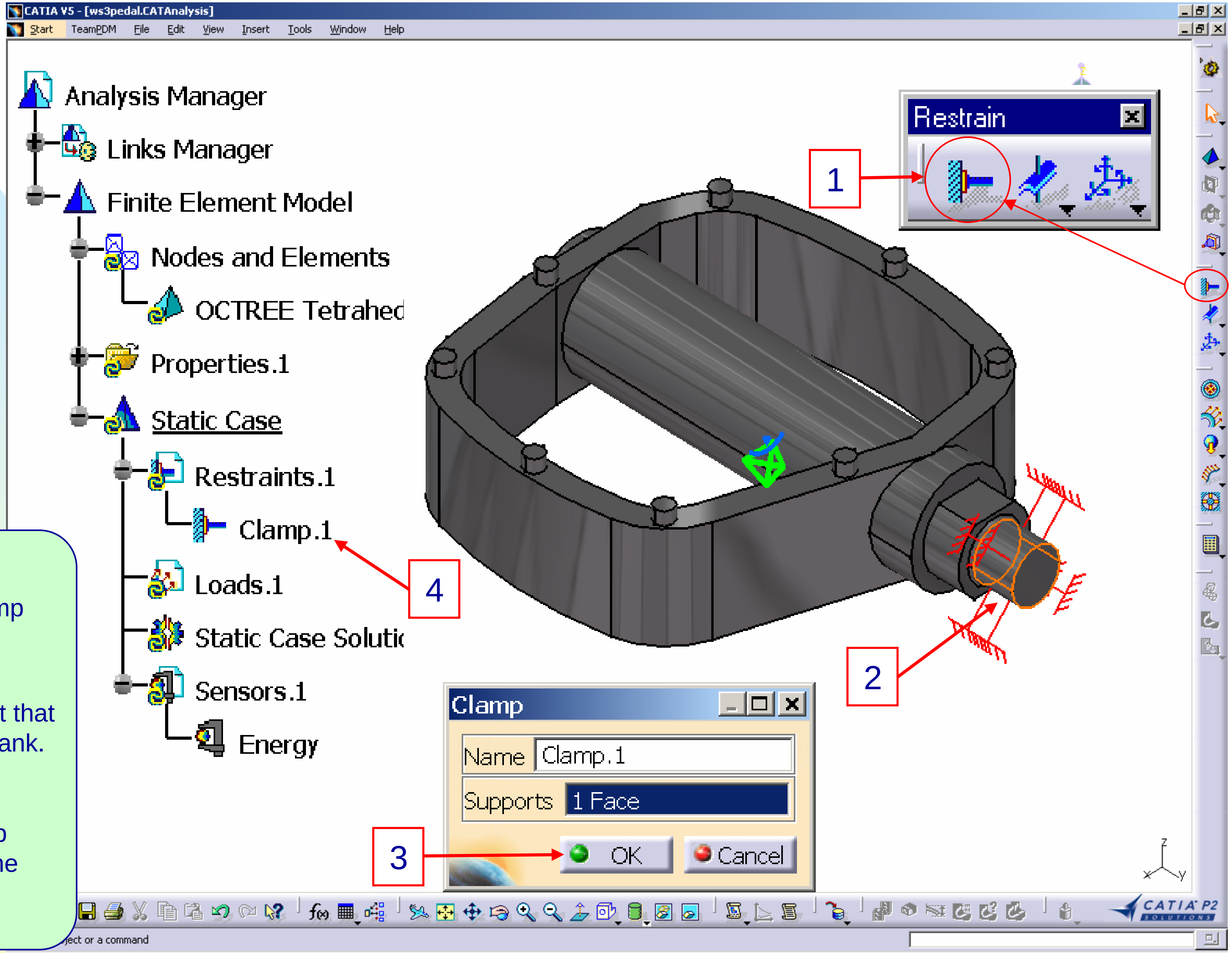
3. Specify the recommended Sag = 10% of Global Size.

4. Specify element type "Linear" (TE4, means 4 node tetrahedron) and is good for a rough analysis, select OK.



MSC SOFTWARE

Step 5. Apply a clamp restraint



Steps:

1. Select the Clamp Restraint icon.

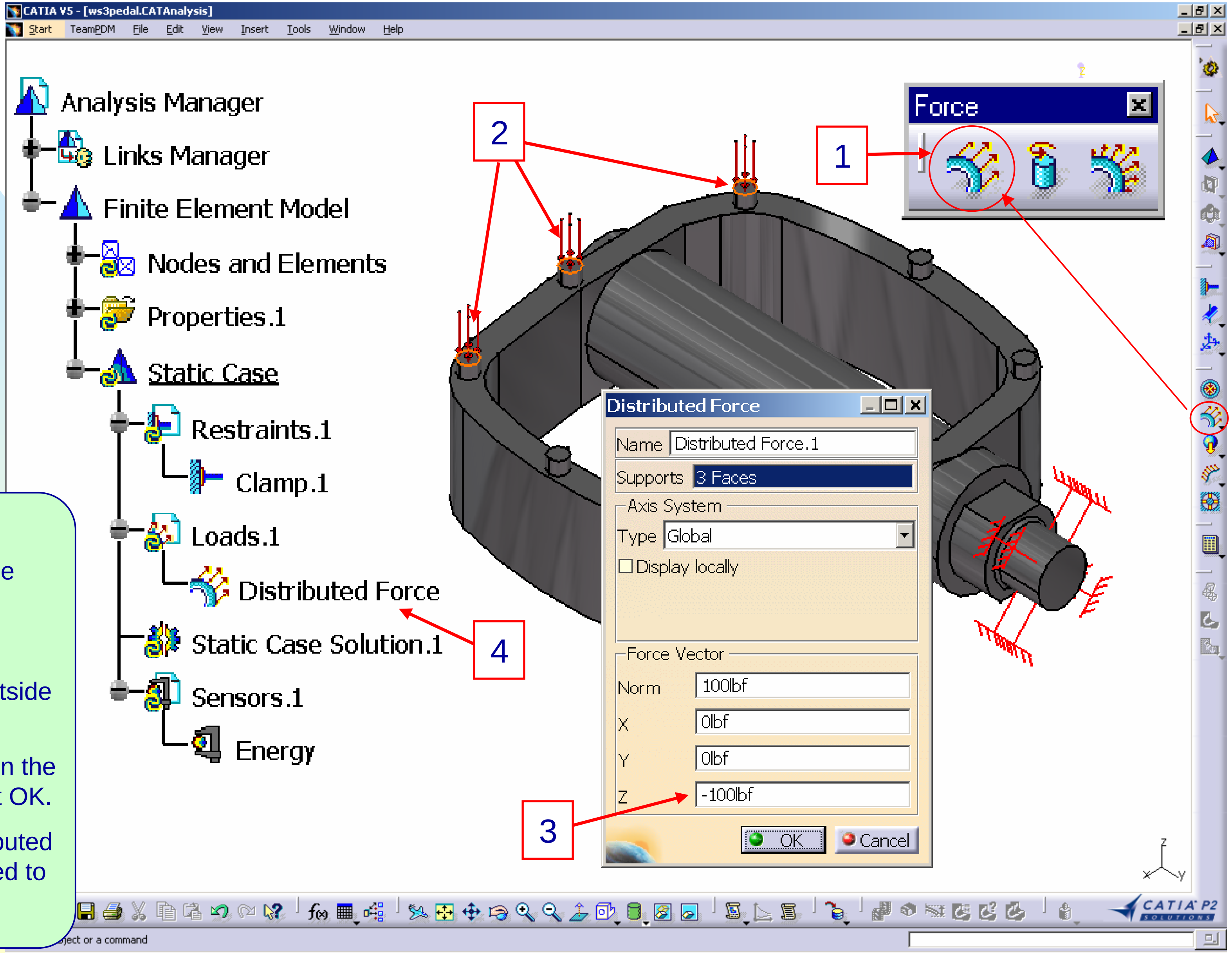
2. Select the shaft that attaches to the crank.

3. Select OK.

4. Note the Clamp object added to the features tree.

MSC SOFTWARE

Step 6. Apply a distributed force



Steps:

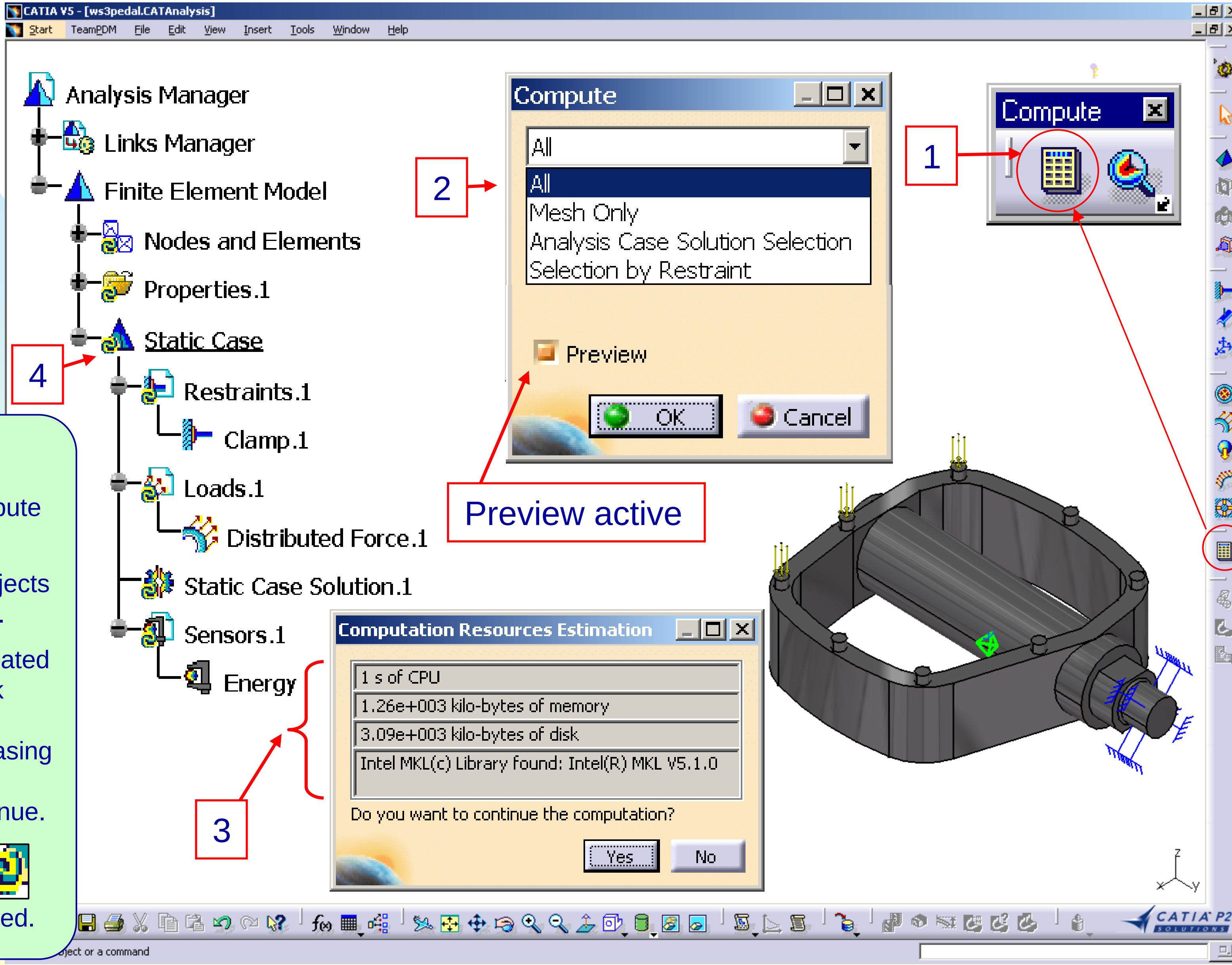
1. Select the Force icon.

2. Select the 3 outside foot grip pads.

3. Enter -100 lbs in the Z-direction, select OK.

4. Note the Distributed Force object added to the features tree.

Step 7. Compute the analysis



Steps:

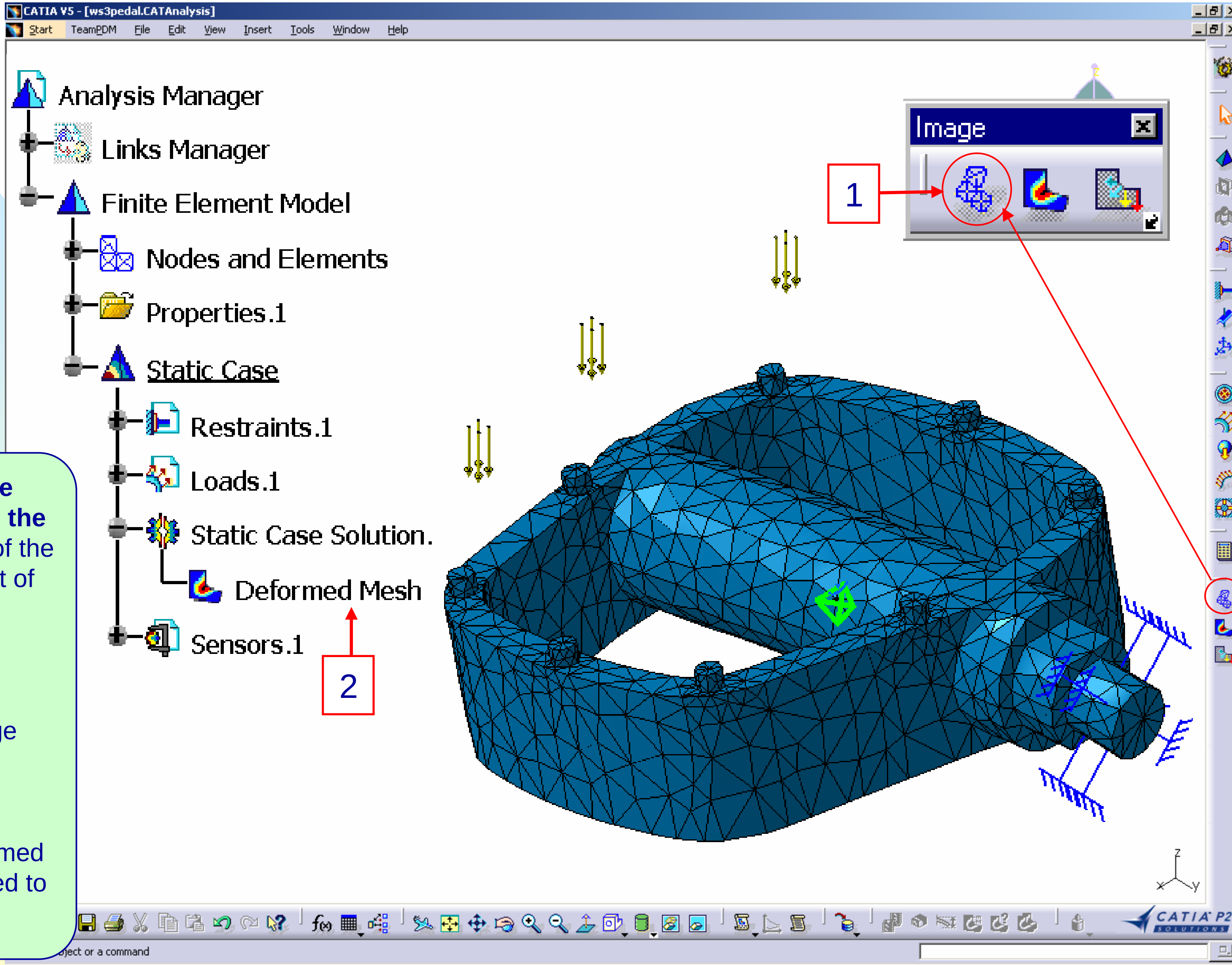
1. Select the Compute icon.

2. Compute All Objects defined, select OK.

3. Notice the estimated time, memory, disk requirement and Warning for decreasing computation time - select Yes to continue.

4. This symbol indicates computation required.

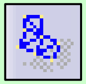
Step 8. Visualize the analysis results



Visualize the **finite element mesh in the deformed state** of the system as a result of loading.

Steps:

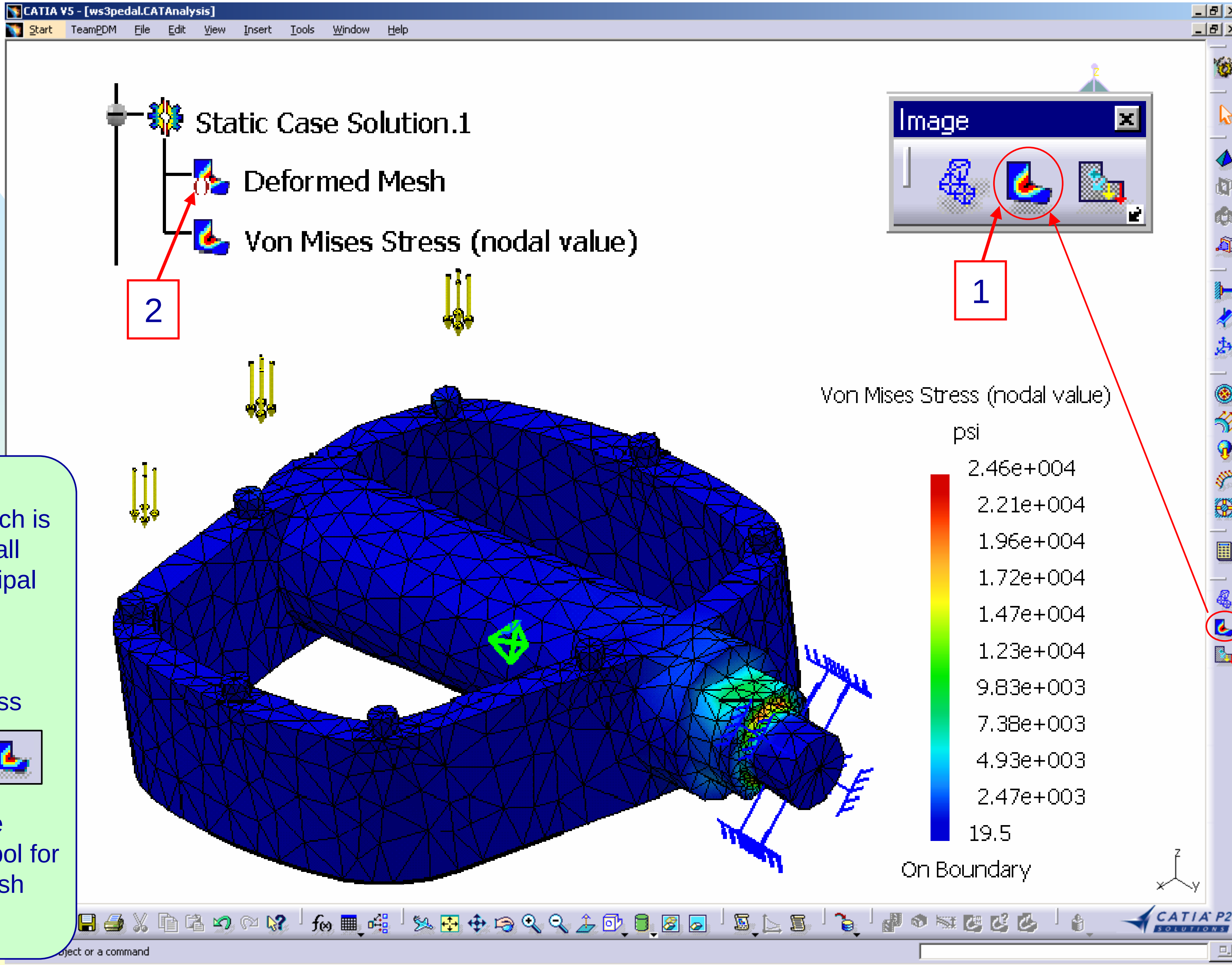
1. Select the Deformation Image Icon.



2. Note the Deformed Mesh Image added to the features tree.

MSC SOFTWARE

Step 8. Visualize the analysis results



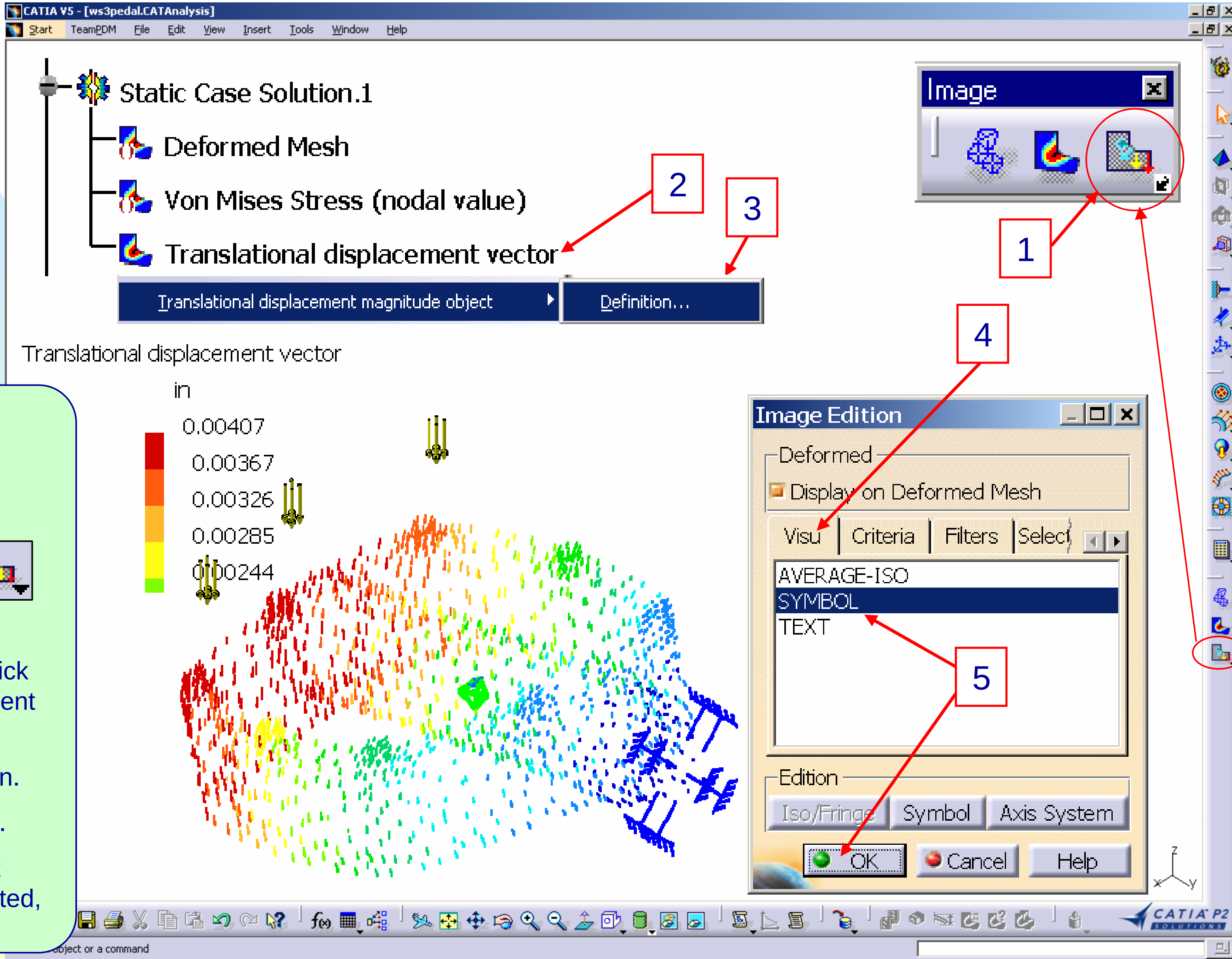
Visualize the **Von Mises stress** which is a combination of all primary and principal stresses.

Steps:

1. Select the Stress Von Mises icon.

2. Note the Image Deactivated symbol for the Deformed Mesh image.

Step 8. Visualize the analysis results



Visualize the **displacement** vectors.

Steps:

1. Select the Displacement icon.

2. Right mouse click Transl. displacement vector in the tree.

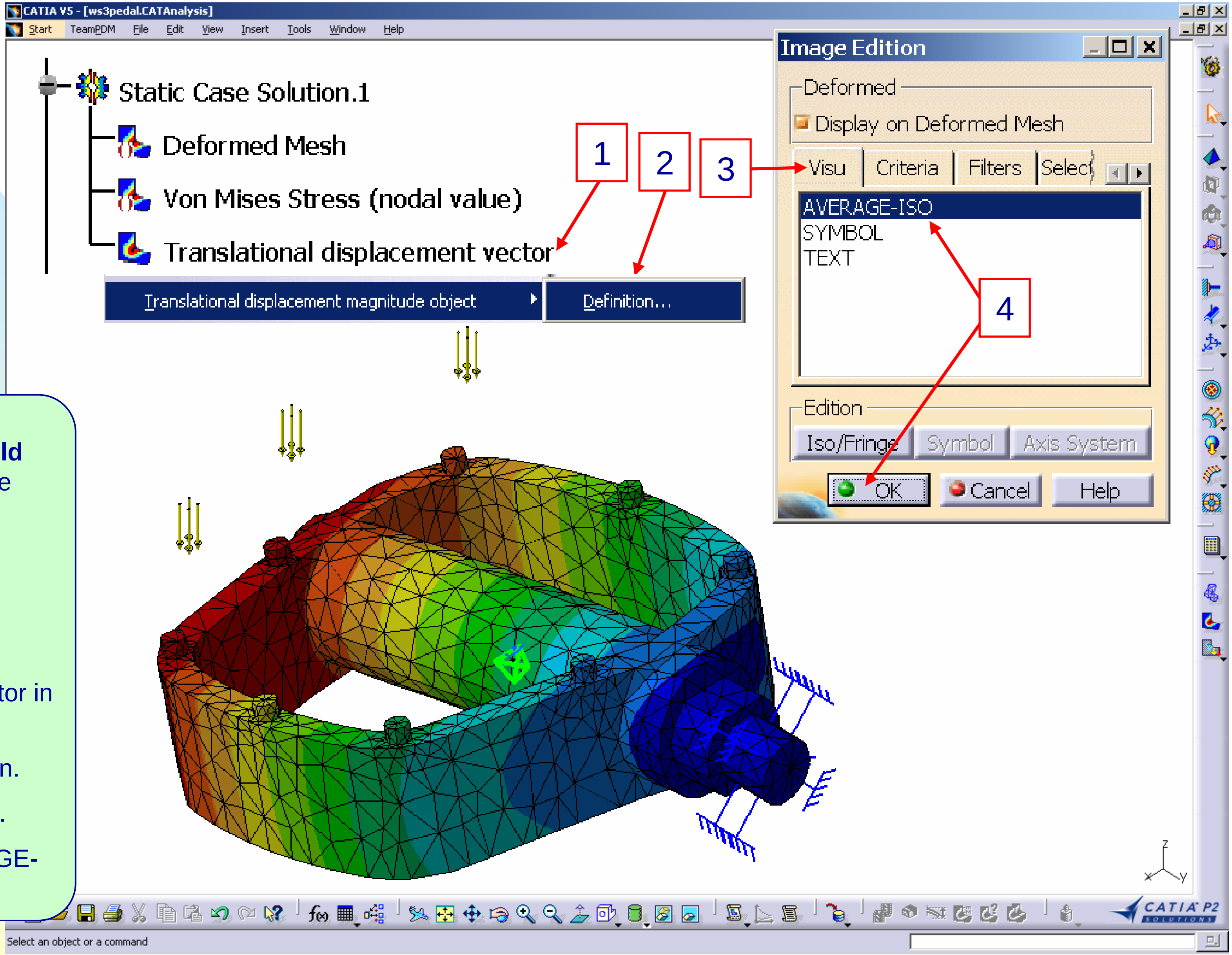
3. Select Definition.

4. Select Visu tab.

5. Note by default SYMBOL is selected, select OK.

MSC SOFTWARE

Step 8. Visualize the analysis results



Visualize the **displacement field patterns** using the AVERAGE-ISO definition.

Steps:

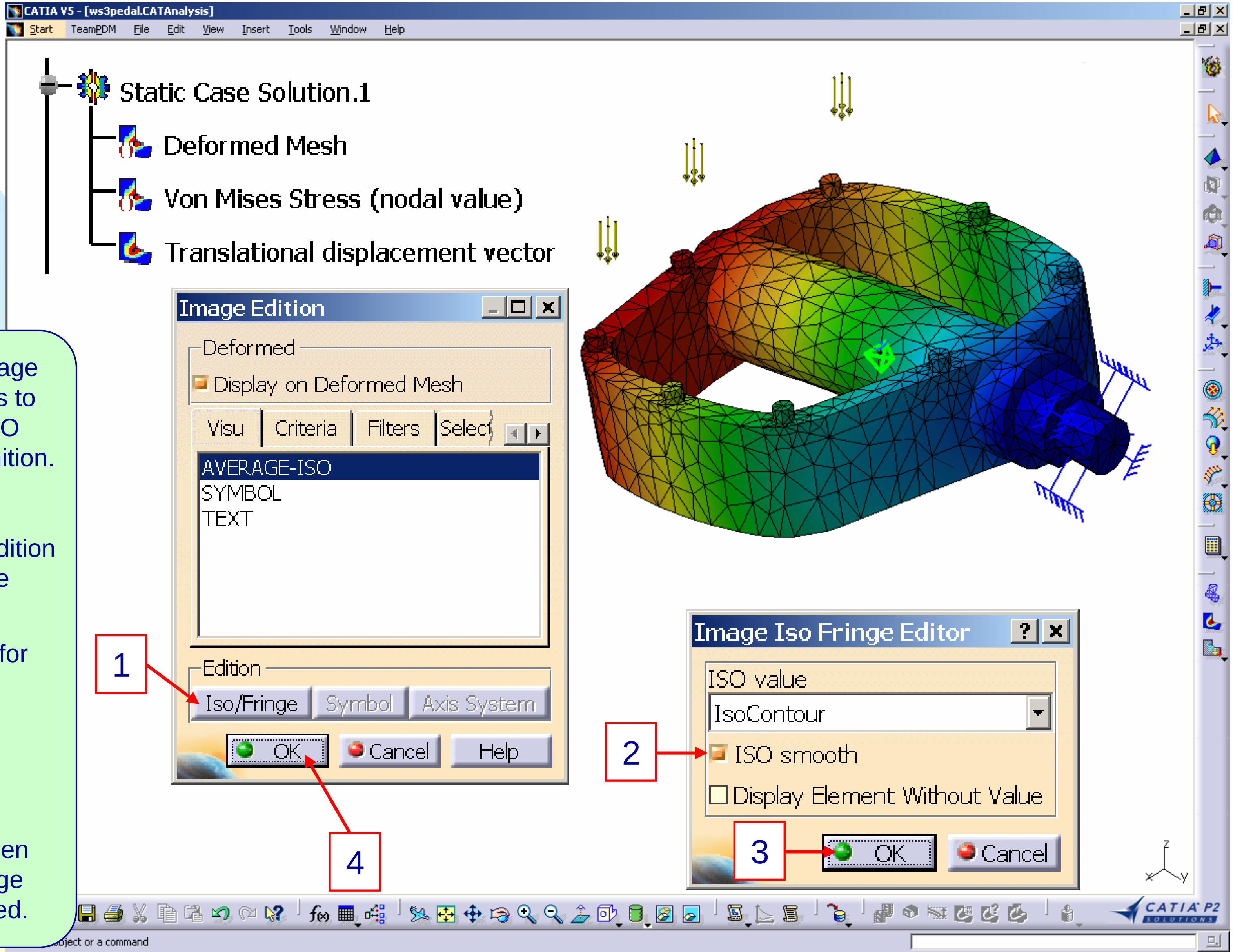
1. Right click Translational displacement vector in the features tree.

2. Select Definition.

3. Select Visu tab.

4. Select AVERAGE-ISO, select OK.

Step 8. Visualize the analysis results



Add additional image smoothing options to the AVERAGE-ISO visualization definition.

Steps:

1. In the Image Edition window, select the Iso/Fringe button.

2. Select the box for ISO smooth.

3. Select OK.

4. Select OK.

Transitions between colors on the image display are blended.

MSC SOFTWARE

Step 8. Visualize the analysis results

Visualize the **computation error** which represent scalar field quantities defined as the distribution of energy error norm estimates for a given computation.

Steps:

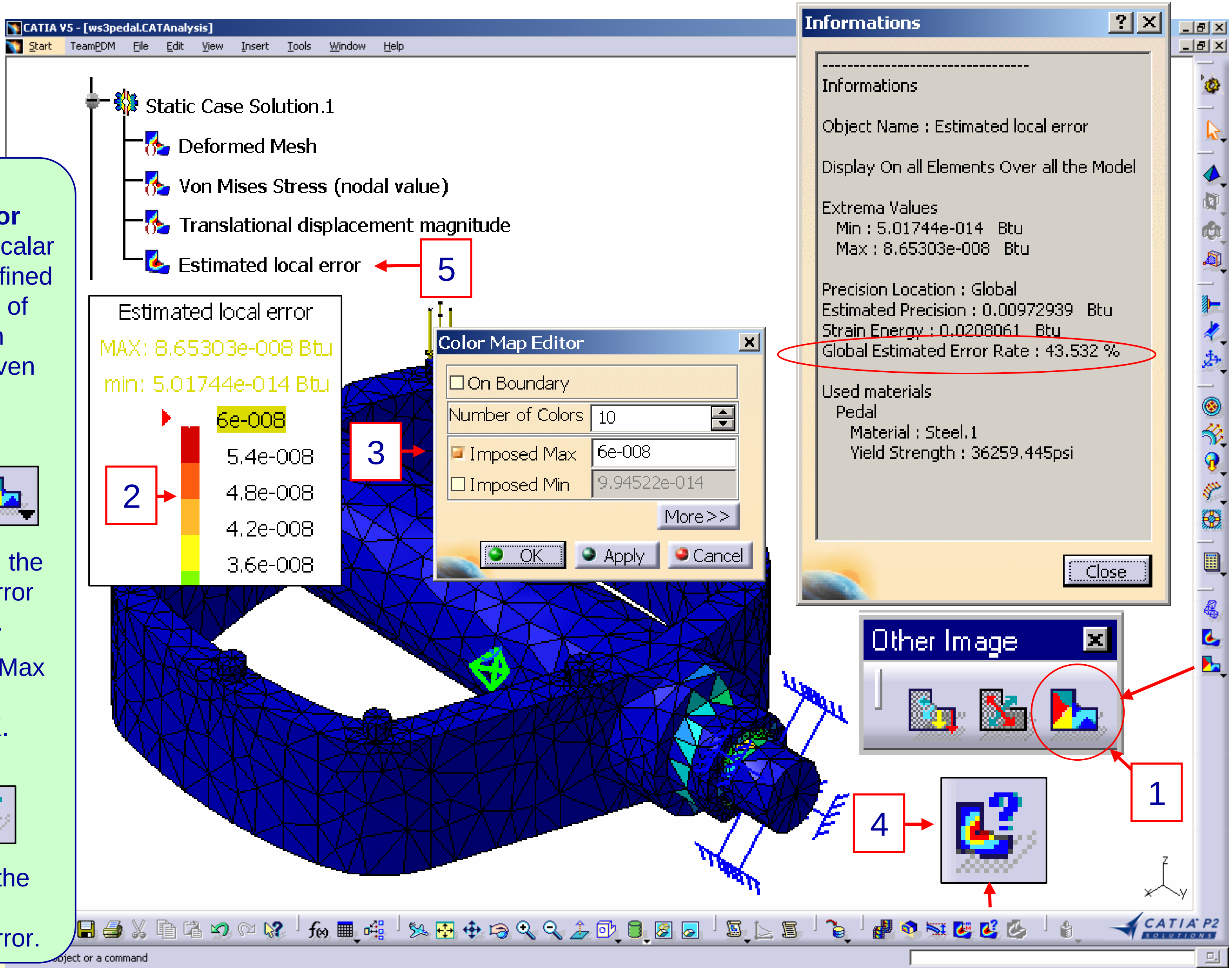
1. Select the Precision icon.

2. Double click on the Estimated local error color map palette.

3. Select Impose Max for the color map palette, select OK.

4. Select the Informations icon.

5. Then select in the features tree Estimated local error.

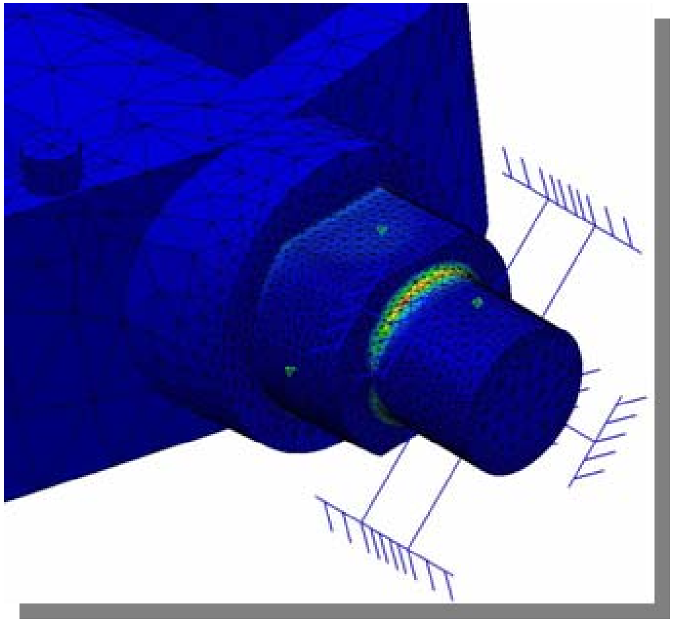


MSC SOFTWARE

Step 8. Visualize the analysis results

- **Conclusions**
  - You now know where the "hot spots" are but the stress and displacement results are questionable with a 43.5% Global Precision Error.
  - The next step is to refine the mesh in the critical areas. We will go over this in the next workshop #4.

	.25" Linear Mesh
Max Von Mises	24.6 ksi
Translational Displacement	.00407 inch
Error Estimate	8.65e-8
Global % Precision error Local % Precision error	43.5 % NA %



	Recommendation
Error Estimate	1.00e-8 (zero)
Global % Precision error Local % Precision error	20 % 10 %

Step 9. Save the analysis document

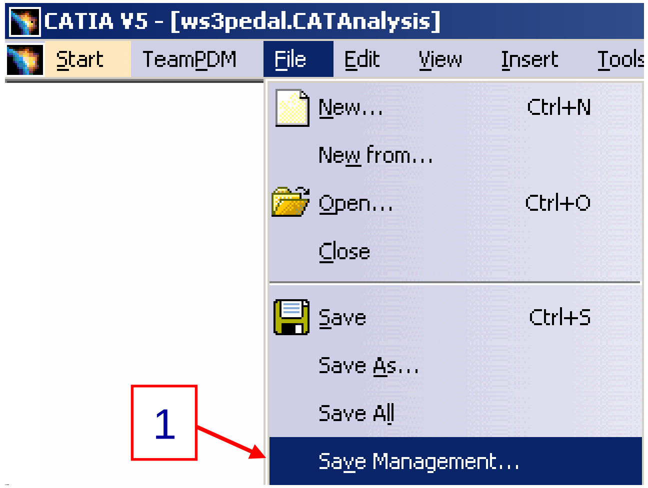
Steps:

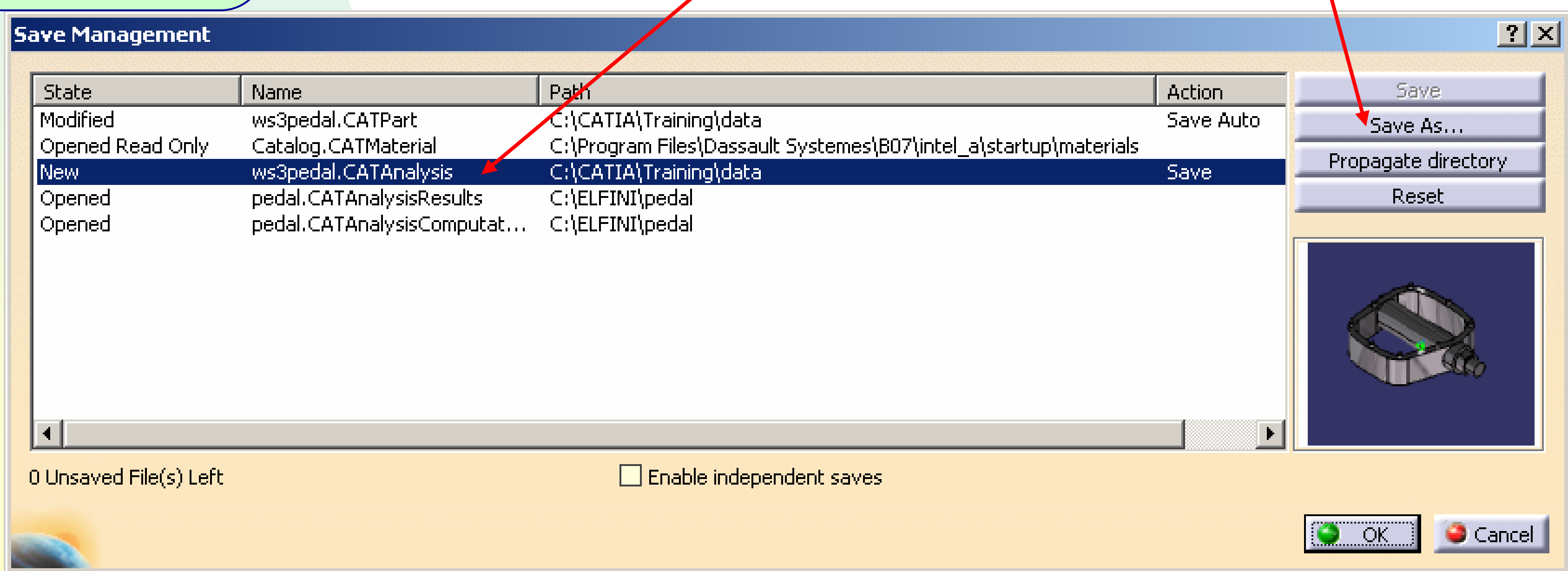
1. From the File menu select Save Management.

2. Select document you want to save.

3. Select Save As to specify name and path, select OK.

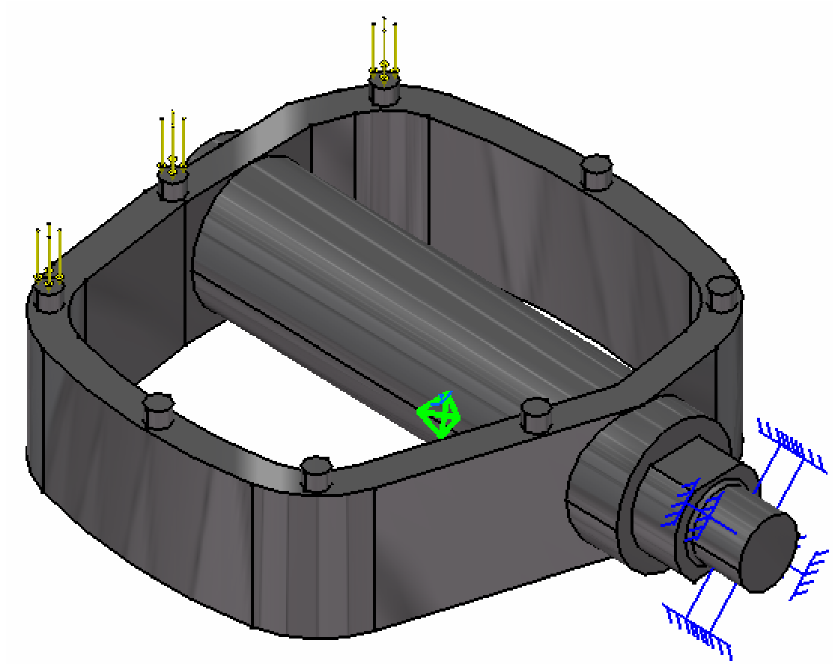
The pedal.CATPart and .CATAnalysis should each be saved under a new name in the work directory.





WORKSHOP 4

BICYCLE PEDAL MESH REFINEMENT AND ADAPTIVITY



Problem Description

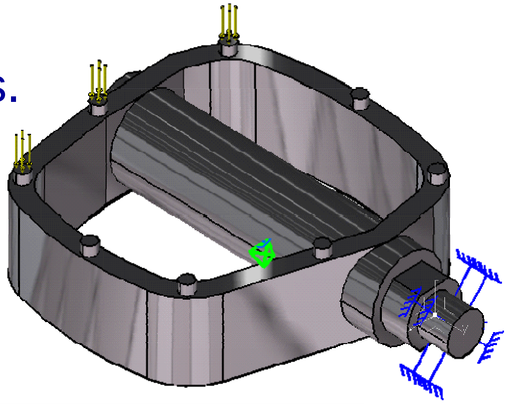
- Assume the same 200 lb person riding the bicycle is standing balanced evenly on each pedal. Material (Steel) properties are as specified below.
- Using the previous rough analysis, refine the mesh until you are comfortable with the results. Is this steel strong enough?

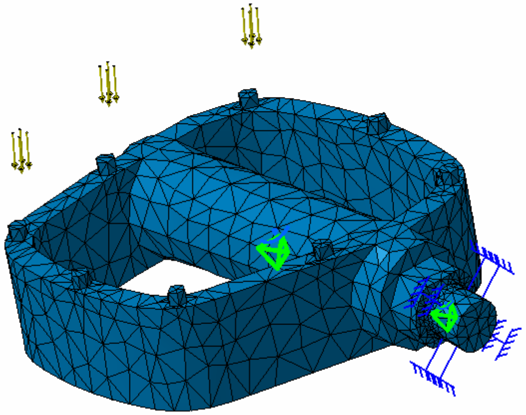
Steel ASTM A36

Elastic Modulus, E	29.0E6 psi
Poisson's Ratio, ν	0.3
Density	.284 lb/in^3
Yield Strength	36,000 psi

Suggested Exercise Steps

1. Open the existing CATIA analysis in the GSA workbench.
2. Change mesh to parabolic and add local meshing.
3. Compute the more precise analysis.
4. Search for point(s) of maximum Von Mises stress.
5. Search for point(s) of minimum precision.
6. Visualize the refined analysis results.
7. Create an adaptivity box with a 5% target.
8. Adapt and converge.
9. Visualize the adaptive analysis results.
10. Verify reactions.
11. Generate a basic analysis report.
12. Save the analysis document.





MSC SOFTWARE SIMULATING REALITY

Step 1. Open the existing CATIA analysis

Open the CATIA analysis document ws4pedal.CATAnalysis in the Generative Structural Analysis workbench.

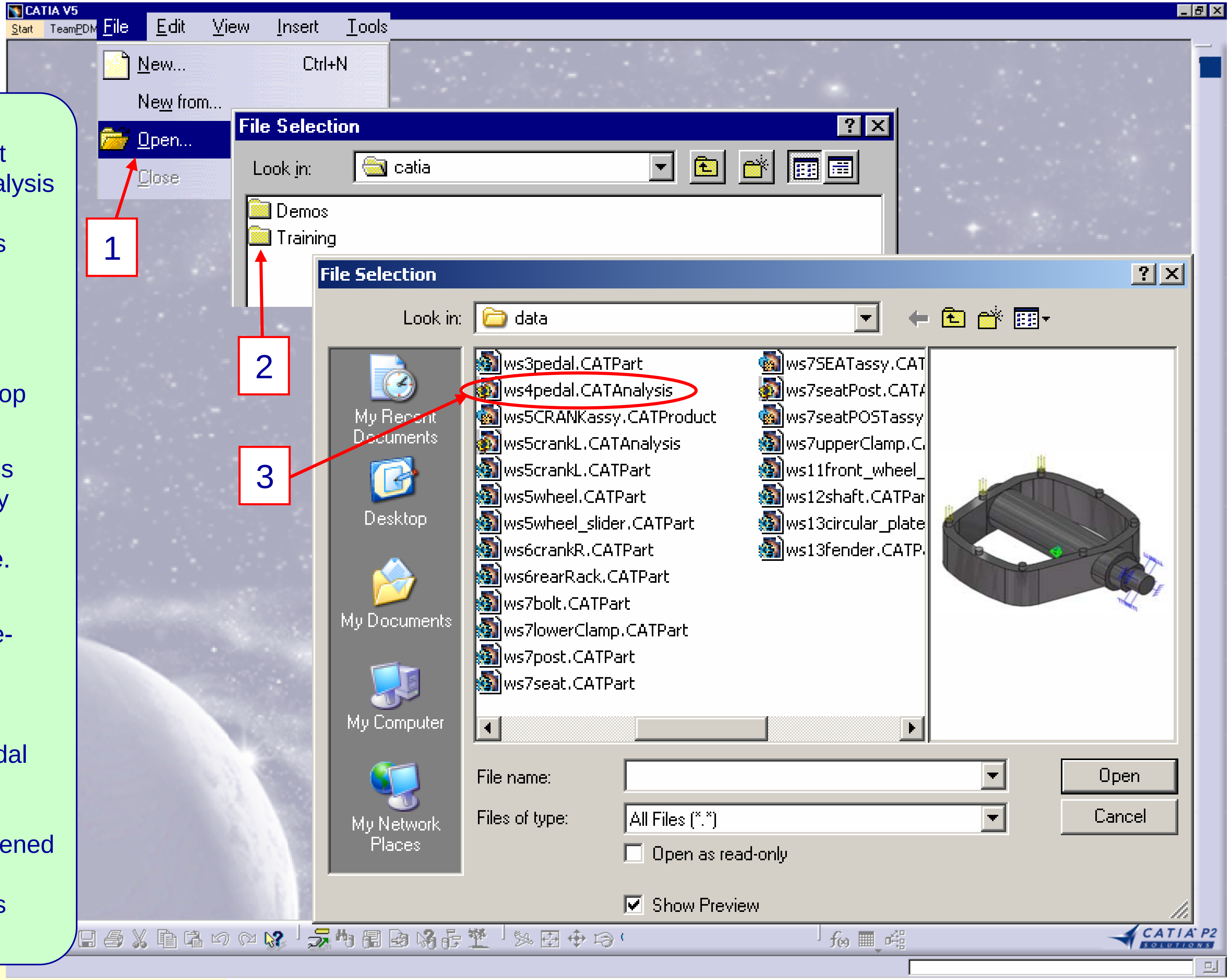
Steps:

1. Select File and Open… from the top pull-down menu.

2. Access the class workshop directory using the typical Windows interface.

3. Open the pedal analysis by double-clicking.

By default, the pedal and all other CATAnalysis documents are opened in the Generative Structural Analysis workbench.



Step 1. Open the existing CATIA analysis

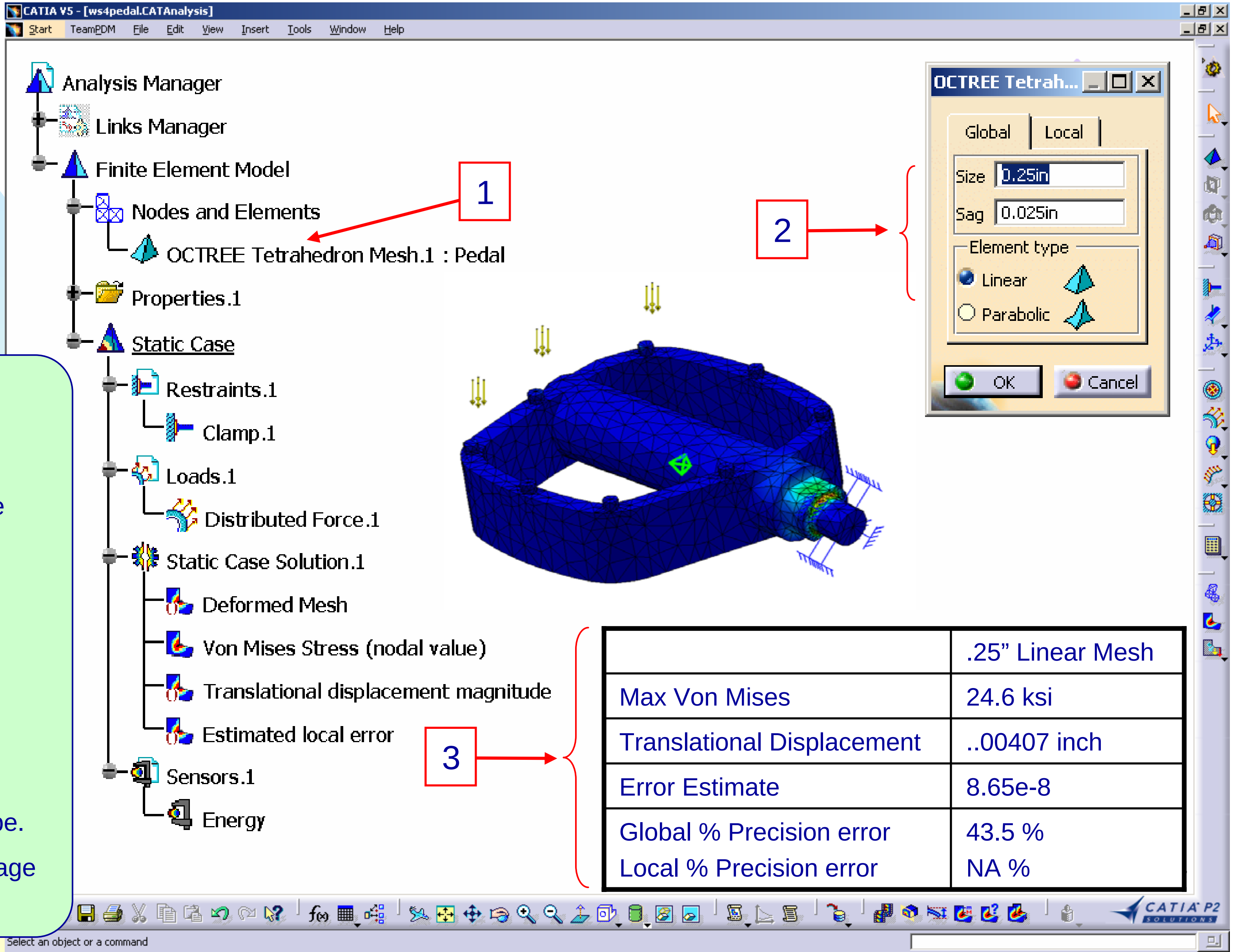
Summary of Workshop 3: the estimated percent error is not low enough (should be less than 10%).

Steps:

1. Double click OCTREE… in the features tree.

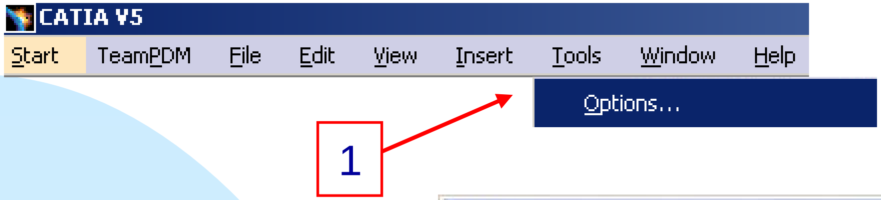
2. Note the Global mesh size of .25", Sag of .025" and Linear element type.

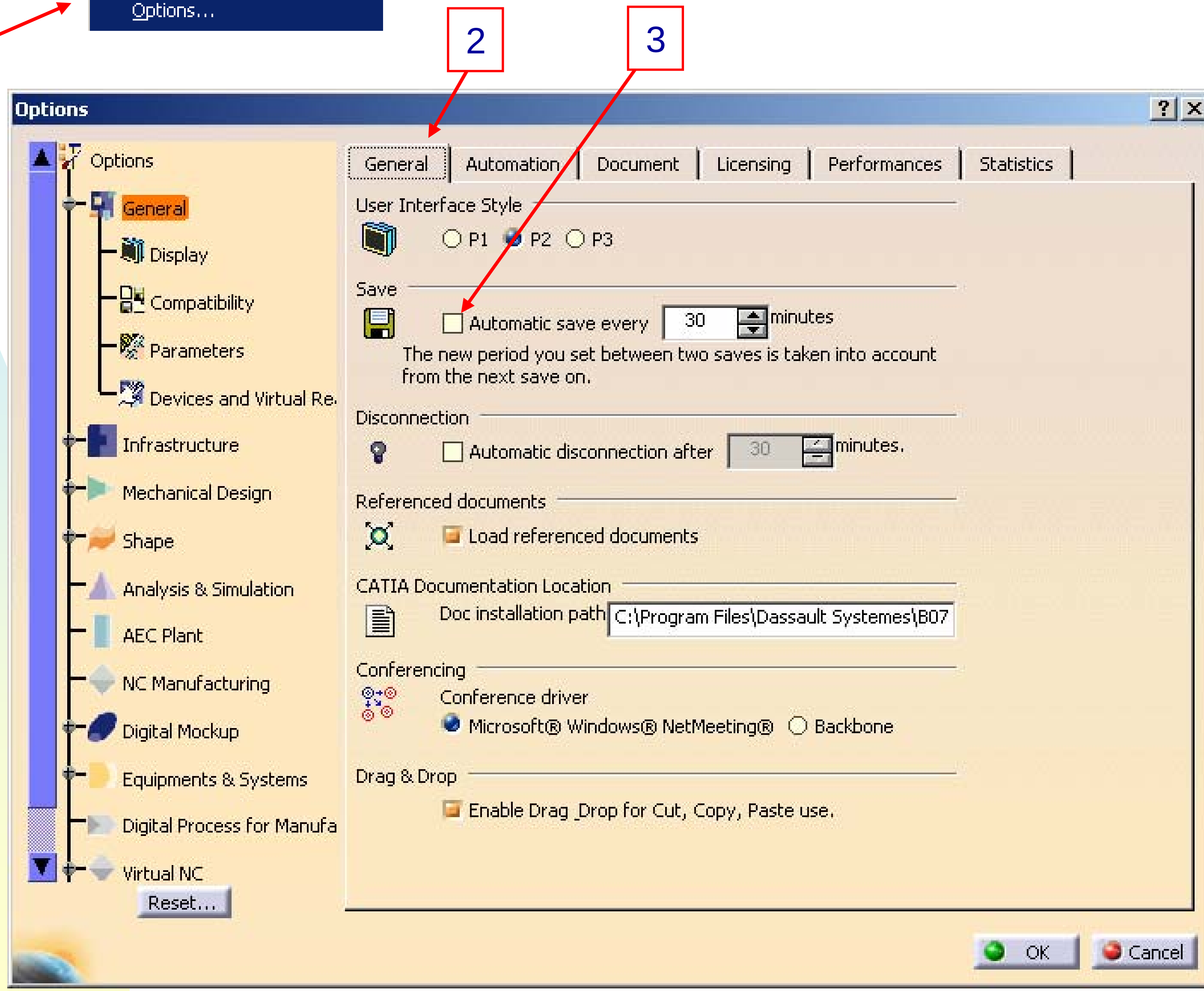
3. Summary of image results.



	.25" Linear Mesh
Max Von Mises	24.6 ksi
Translational Displacement	..00407 inch
Error Estimate	8.65e-8
Global % Precision error Local % Precision error	43.5 % NA %

Step 1. Open the existing CATIA analysis





You do not want "auto save" to start while computing a solution, best to turn it off.

Steps:

1. Select Tools from the menu then Options.

2. Select General and the tab General.

3. Deselect the Automatic save button, select OK.

Step 2. Change mesh to parabolic and add local meshing

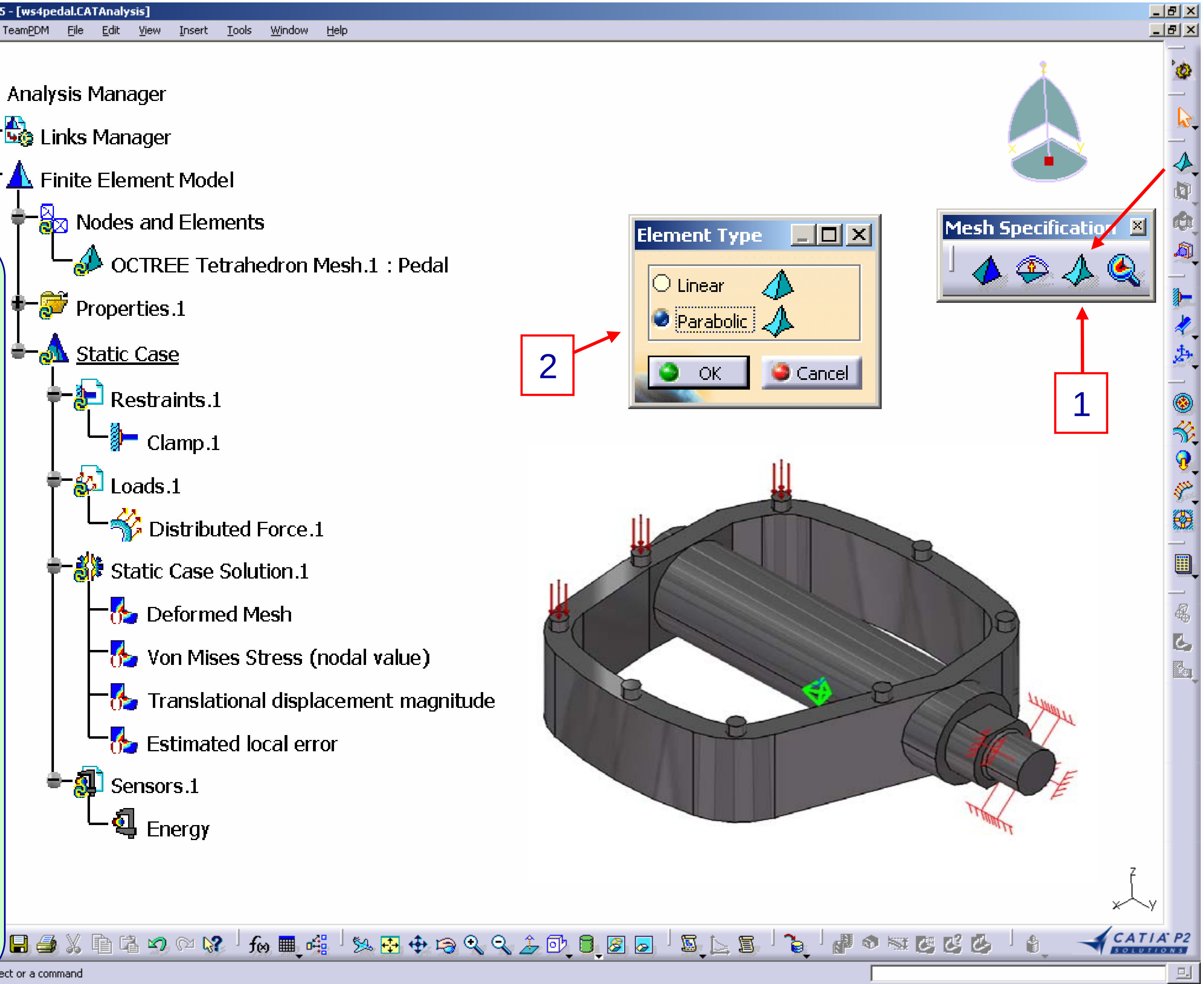
Change the element type from Linear (4-nodes) to Parabolic (10-nodes).

Steps:

1. Select the Change Element Type icon.

2. Select Parabolic then OK.

The best results are achieved using Parabolic elements even though your computation files will be large. Use Linear to locate "hot spots" and to verify a statically determinate model.



MSC SOFTWARE SIMULATING REALITY

Step 2. Change mesh to parabolic and add local meshing

Refine the local mesh size in the high stress area identified in Workshop 3.

Steps:

1. Select the Apply a Local Mesh Size icon.

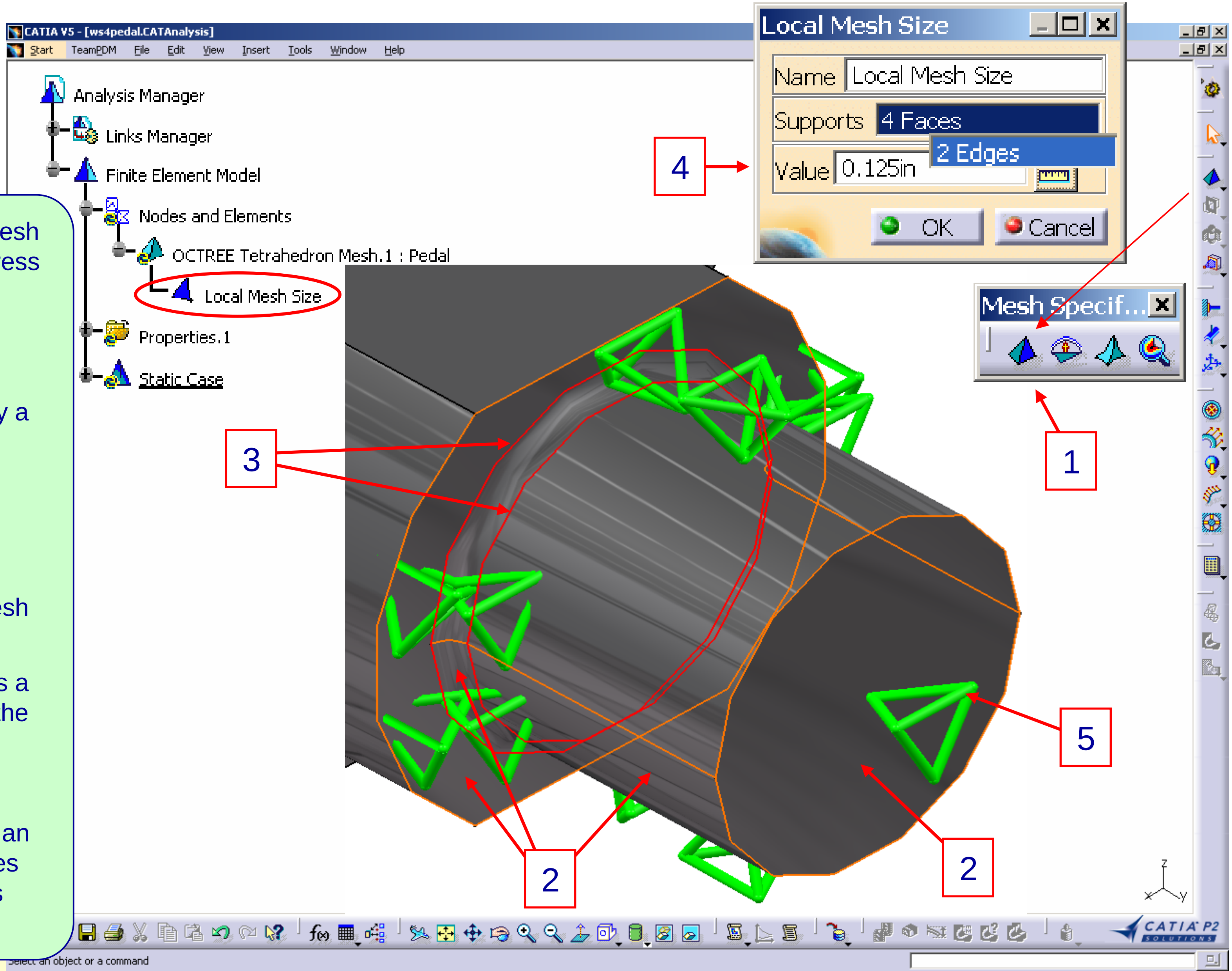
2. Select 4 faces.

3. Select 2 edges.

4. Key in .125” mesh value, select OK.

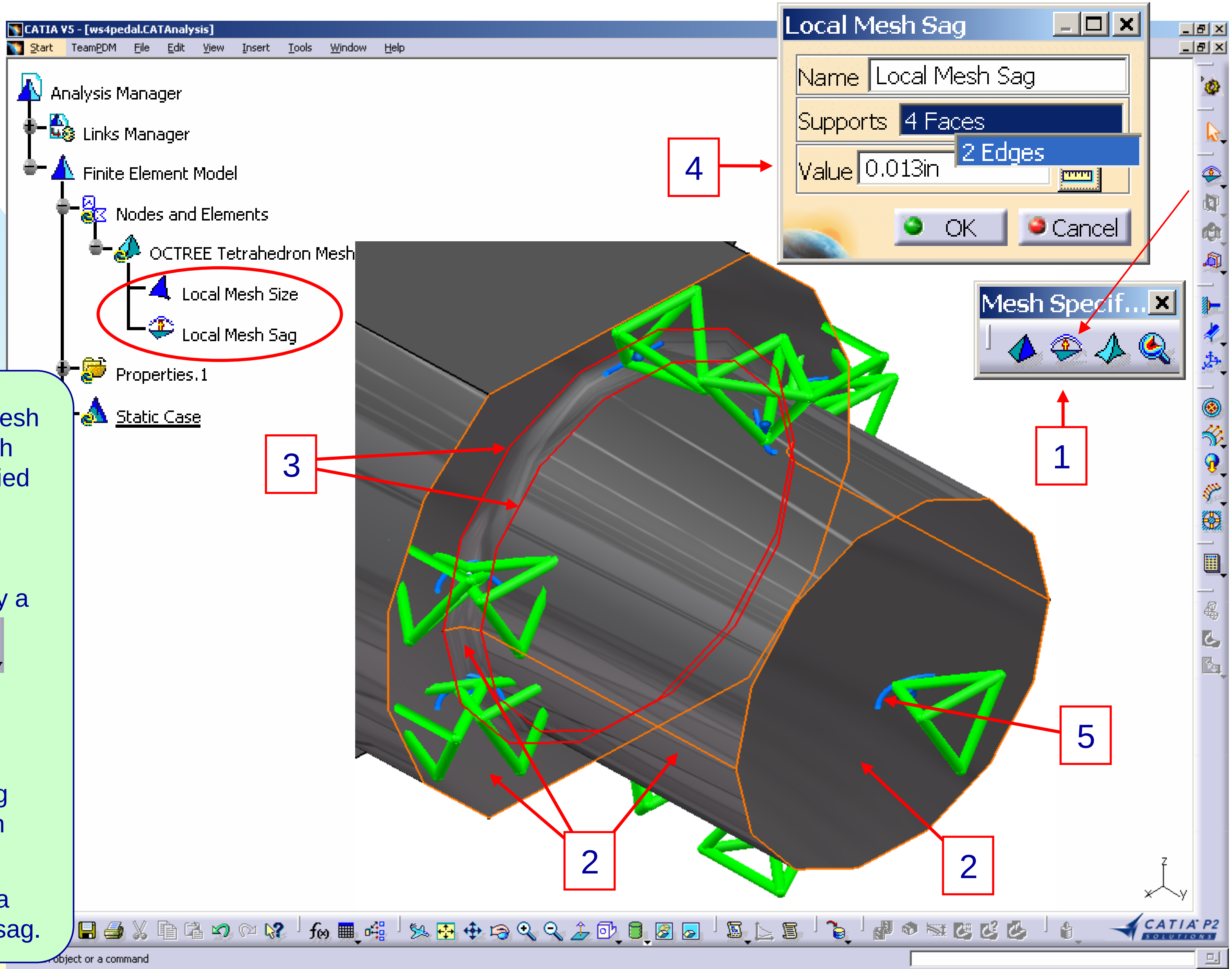
5. Green symbol is a representation of the mesh element.

Local meshing on an edge creates nodes along these edges (imposed edges).



MSC SOFTWARE

Step 2. Change mesh to parabolic and add local meshing



Refine the local mesh sag size in the high stress area identified in Workshop 3.

Steps:

1. Select the Apply a Local Mesh Sag icon.

2. Select 4 faces.

3. Select 2 edges.

4. Key in .013" sag size (10% of mesh size), select OK.

5. Blue symbol is a representation of sag.

MSC SOFTWARE

Step 3. Compute the more precise analysis

Important processes to consider before computing:

- RAM on your PC.
- Disk space for the computation.
- Paging space.
- Running with Intel MKL library installed.

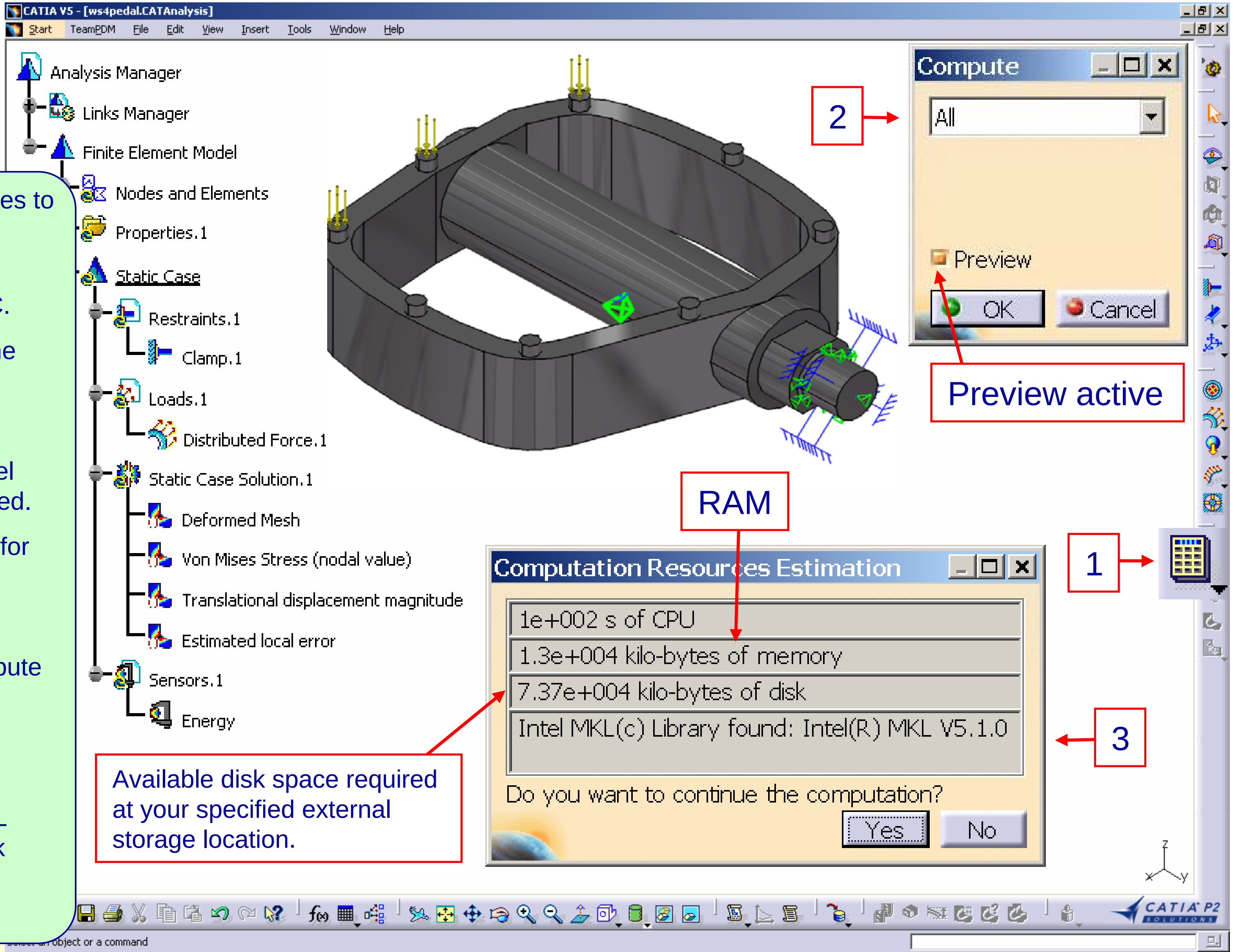
See Info Nuggets for details.

Steps:

1. Select the compute icon**.**

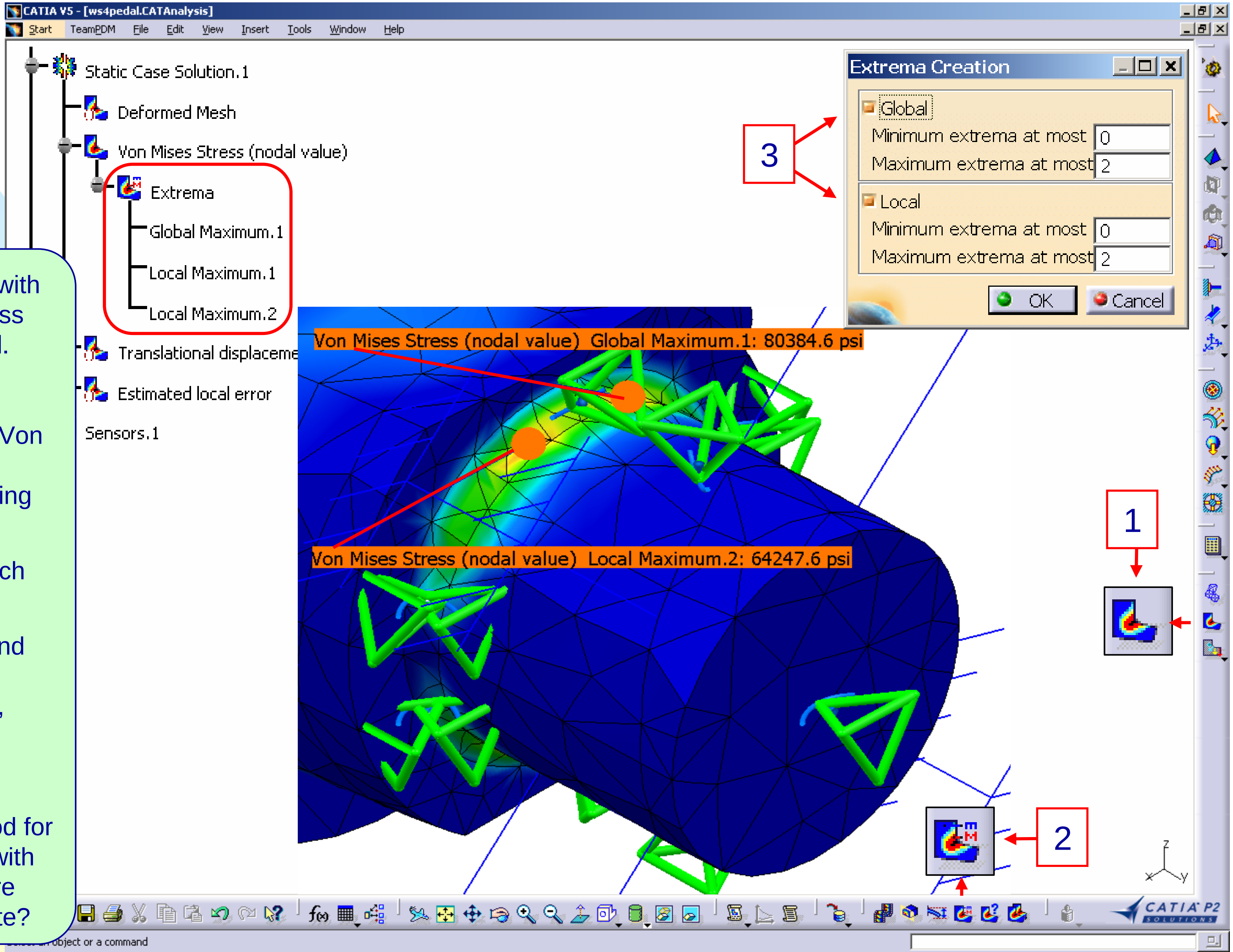
2. Compute All objects, click OK.

3. Note: Intel MKL library found. Click Yes to continue computation.



MSC SOFTWARE

Step 4. Search for point(s) of maximum Von Mises stress



Find the Element with the maximum stress value in the model.

Steps:

1. Clicking on the Von Mises stress icon activates the existing image**.**

2. Select the Search Extrema icon.

3. Select Global and Local, request 2 maximum at most, then select OK.

It doesn't look good for our A36 material with 36 ksi yield, but are the values accurate?

Step 5. Search for point(s) of minimum precision

Find the Element with the least accurate value in the model.

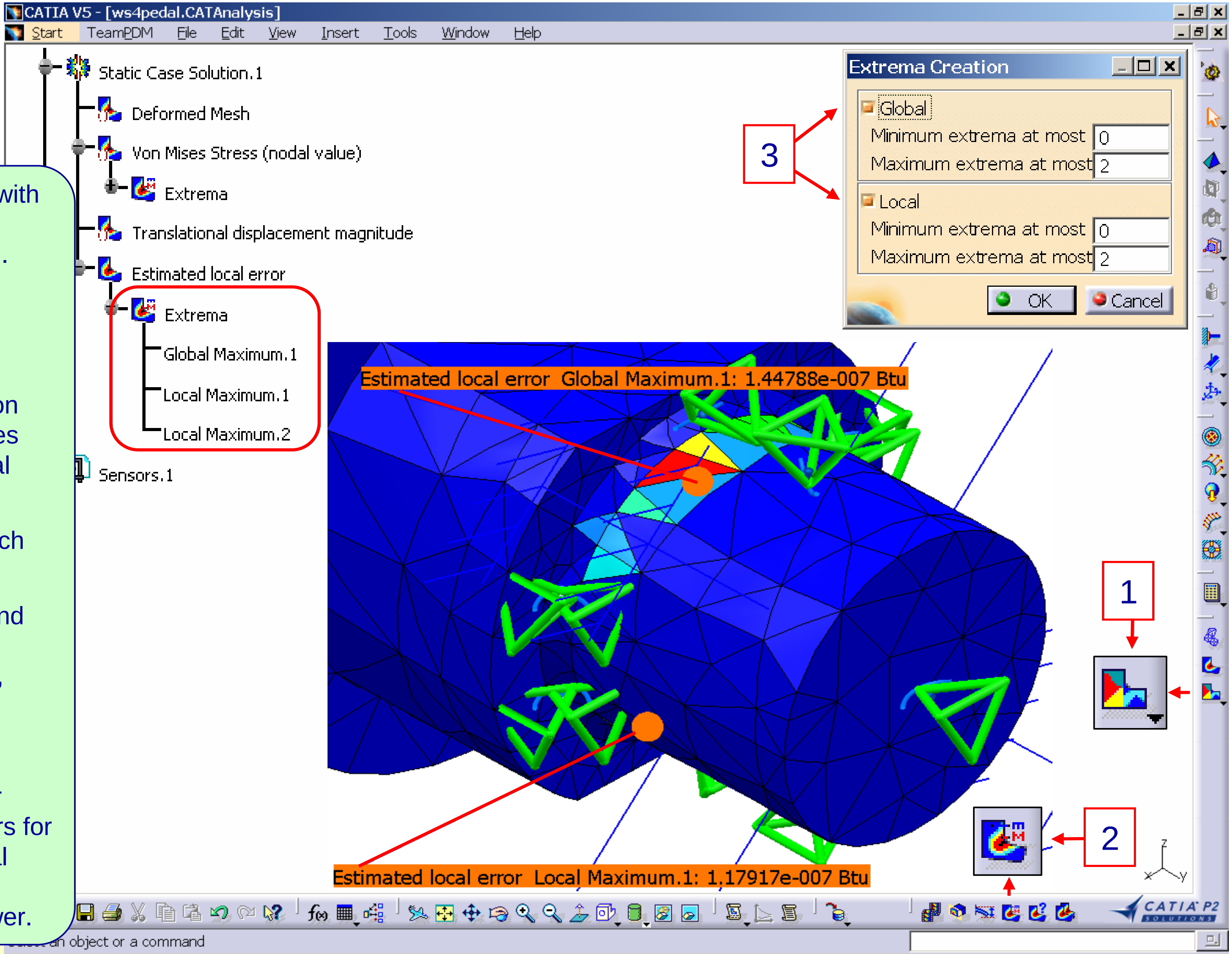
Steps:

1. Clicking on the Precision icon deactivates the Von Mises and activates the Estimated local error image.

2. Select the Search Extrema icon.

3. Select Global and Local, request 2 maximum at most, then select OK.

We are looking for very small numbers for the maximum local error, preferably a value of e-8 or lower.



Step 6. Visualize the refined analysis results

Find the Global estimated error rate percentage value.

Steps:

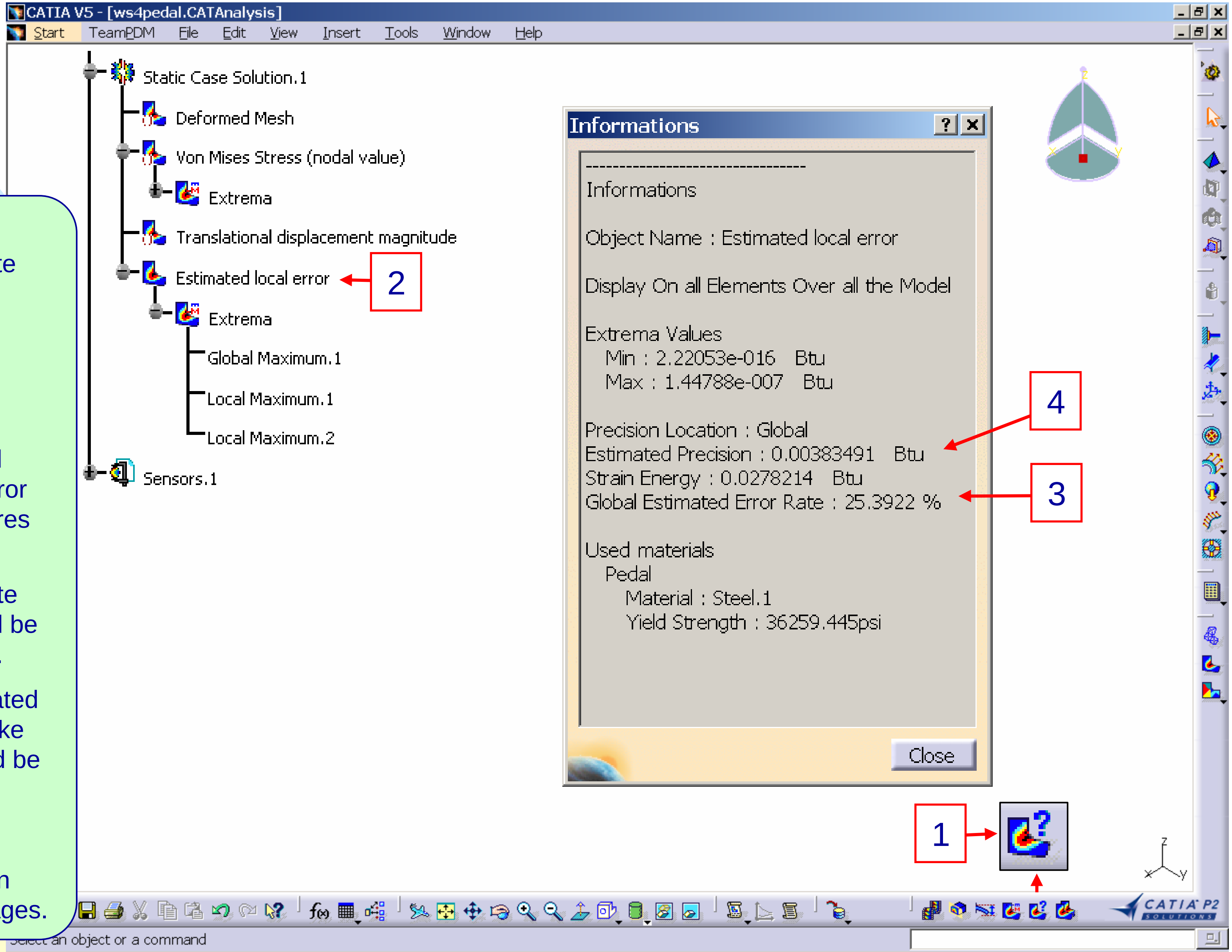
1. Click on the Information icon.

2. Select activated Estimated local error object in the features tree.

3. Note % error rate (global rate should be 20%, 10% locally).

4. Note the Estimated Precision (this is like epsilon and should be close to zero).

Review information from the other images.



Step 7. Create an adaptivity box with a 10% target

Steps:

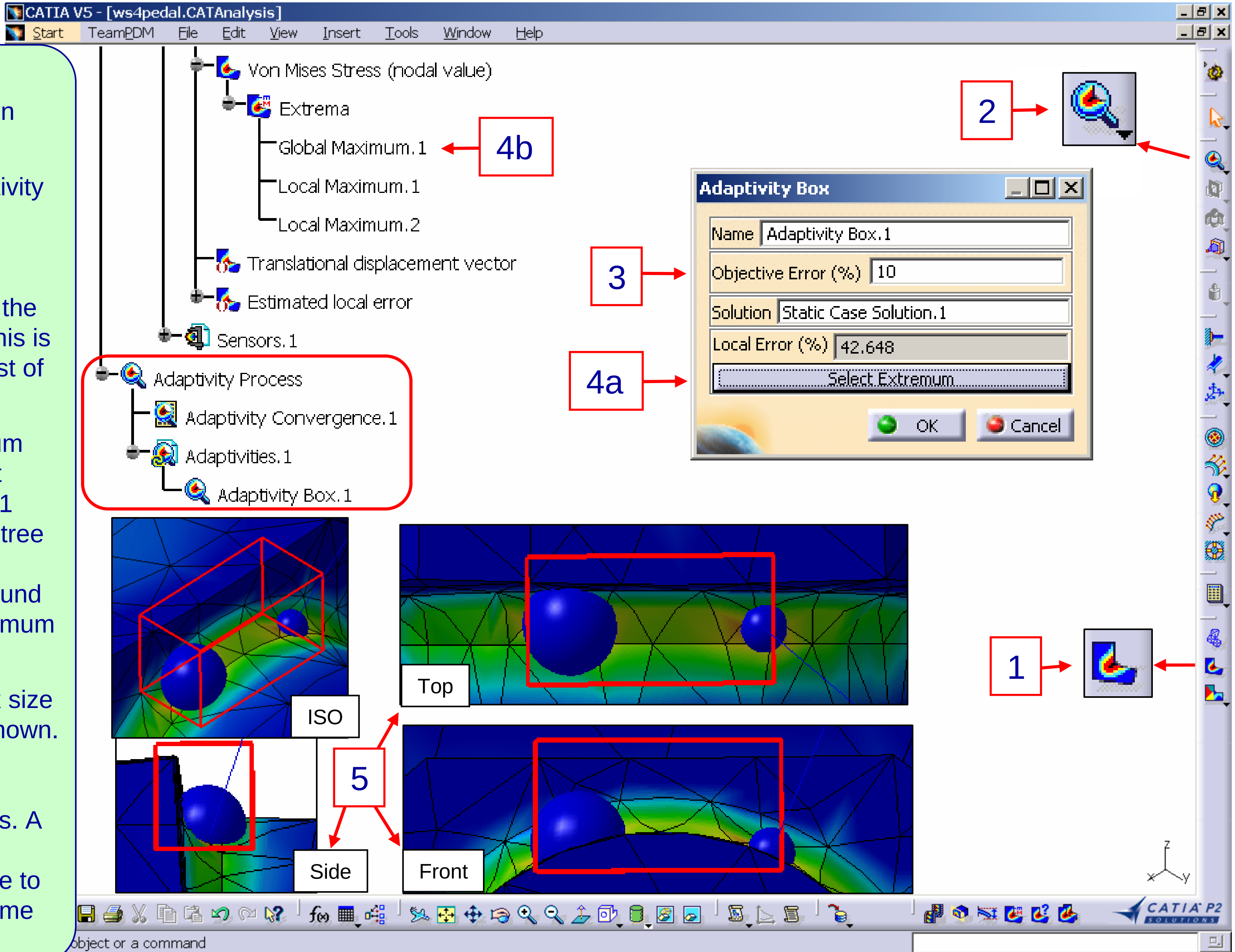
1. Activate the Von Mises Image.

2. Click the Adaptivity Box icon.

3. Key in 10% for the Objective Error (this is used in the interest of time 5% is best).

4. Select Extremum button then select Global Maximum.1 from the features tree (this centers the adaptivity box around the Extrema Maximum selected).

5. Manipulate box size and location as shown. The box should encompass the maximum symbols. A small box is recommended due to space and CPU time limits, select OK.



Step 8. Adapt and converge

Allow 2 iterations attempting to achieve the 10% target precision.

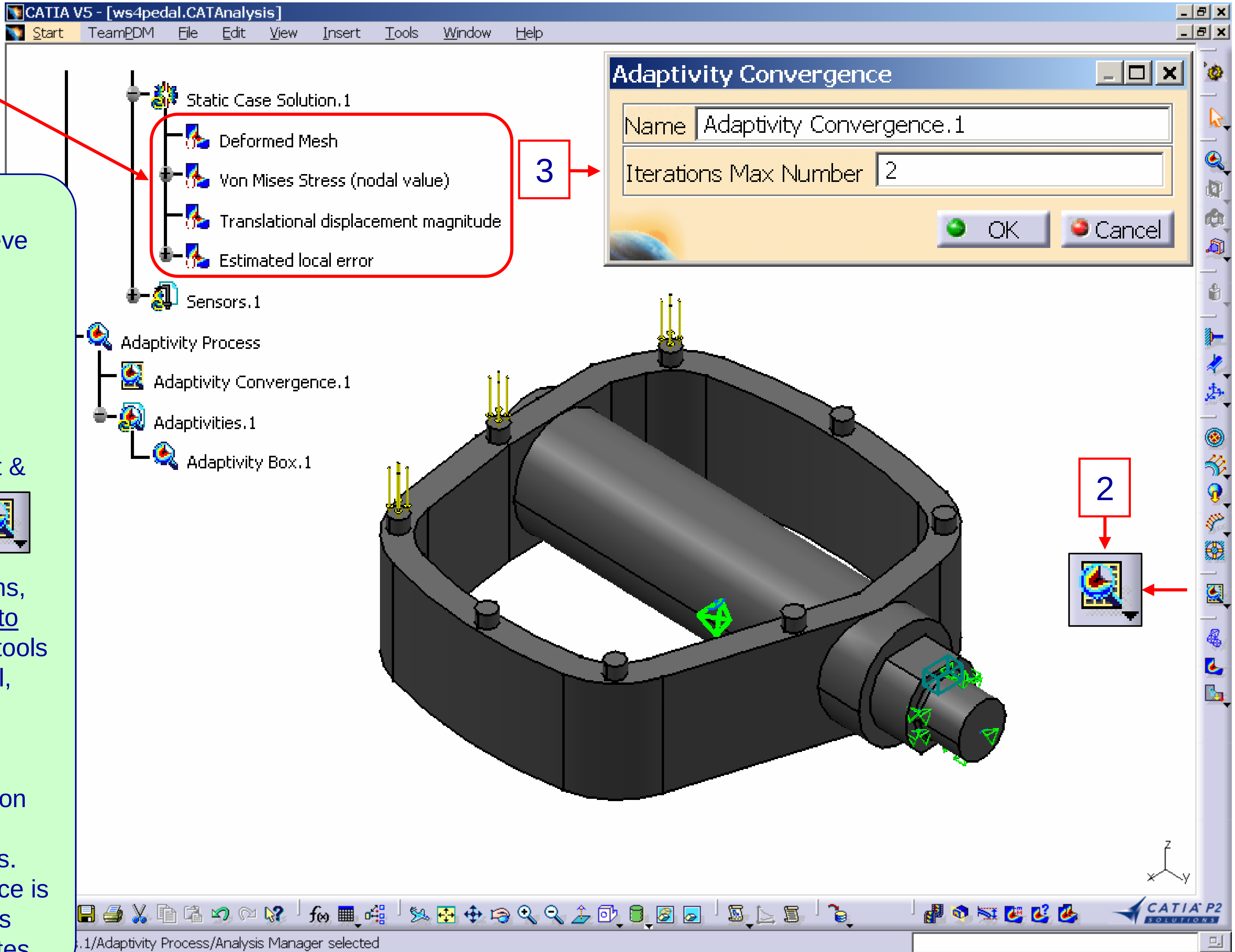
Steps:

1. Deactivate all images.

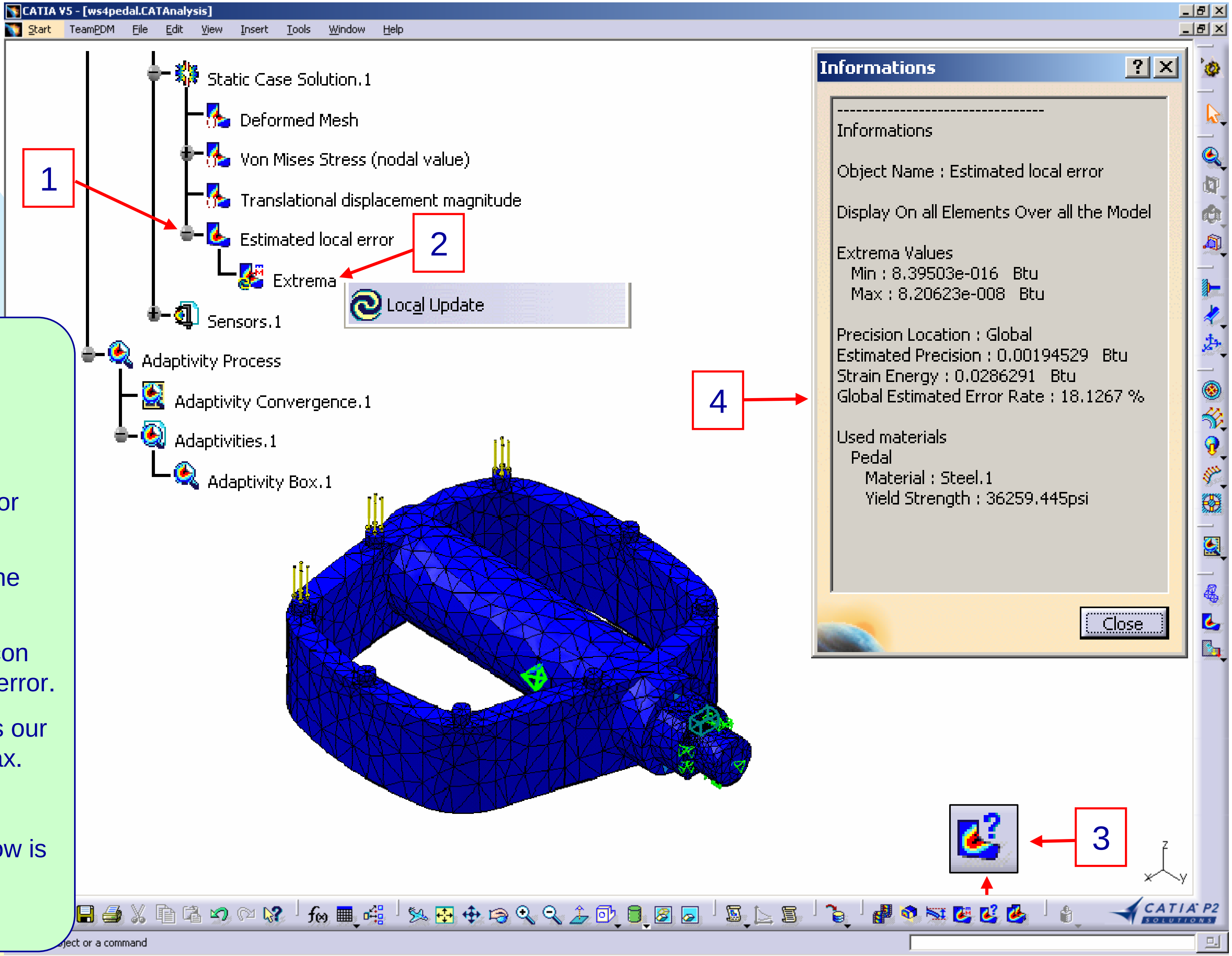
2. Select the Adapt & Converge icon.

3. Key in 2 Iterations, make sure your auto save is turned off: tools + options + general, select OK.

Note: no warnings on RAM, CPU time or space requirements. 1GB of paging space is recommended. This may take 5-7 minutes.



Step 9. Visualize the adaptive analysis results



Results for Global precision error.

Steps:

1. Activate the Estimated local error image.

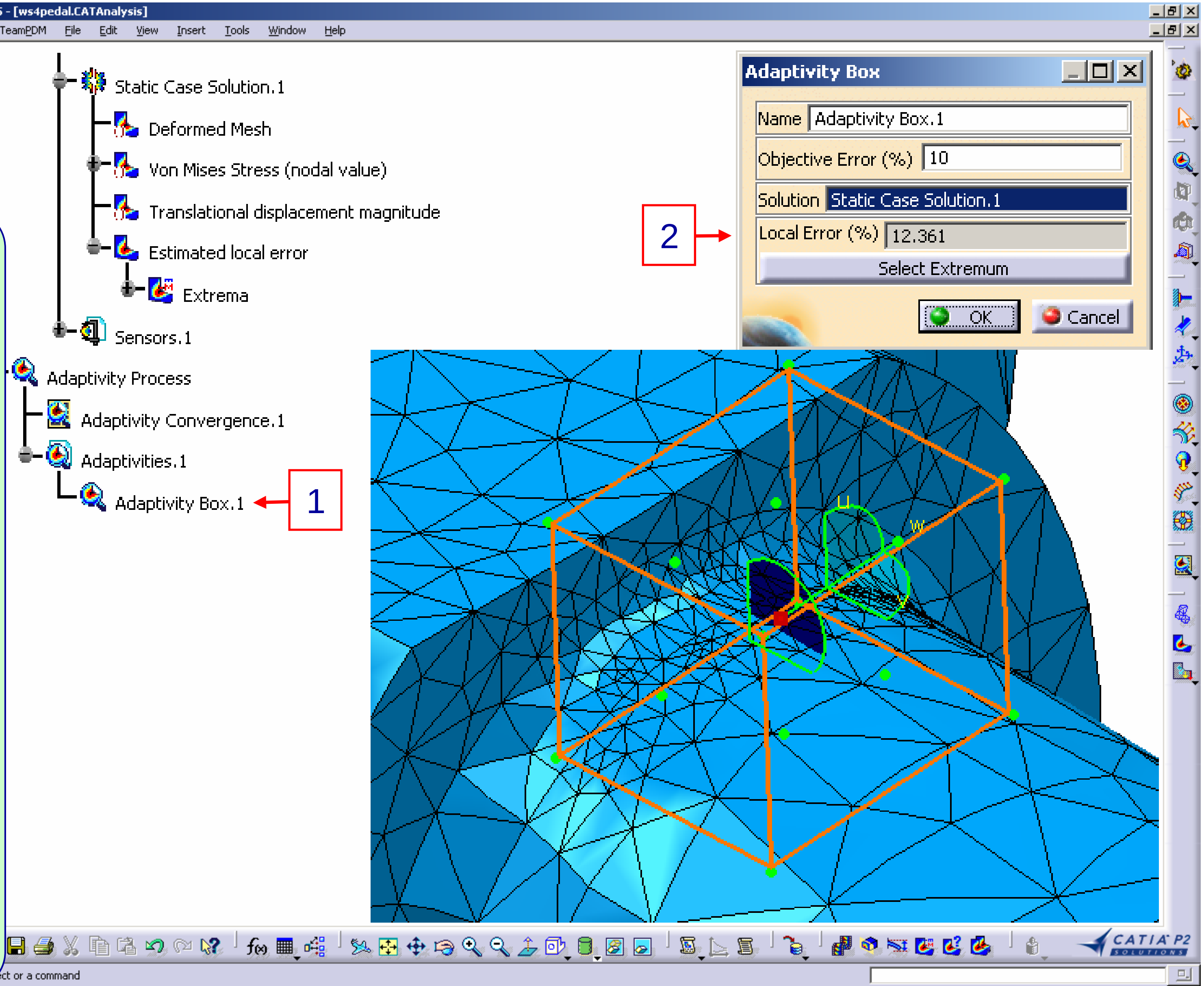
2. Locally update the extrema.

3. Select the info icon then the Est. local error.

4. Improved, meets our suggested 20% max.

Our real interest now is the adaptive local precision.

Step 9. Visualize the adaptive analysis results



Results for adaptive local precision.

Steps:

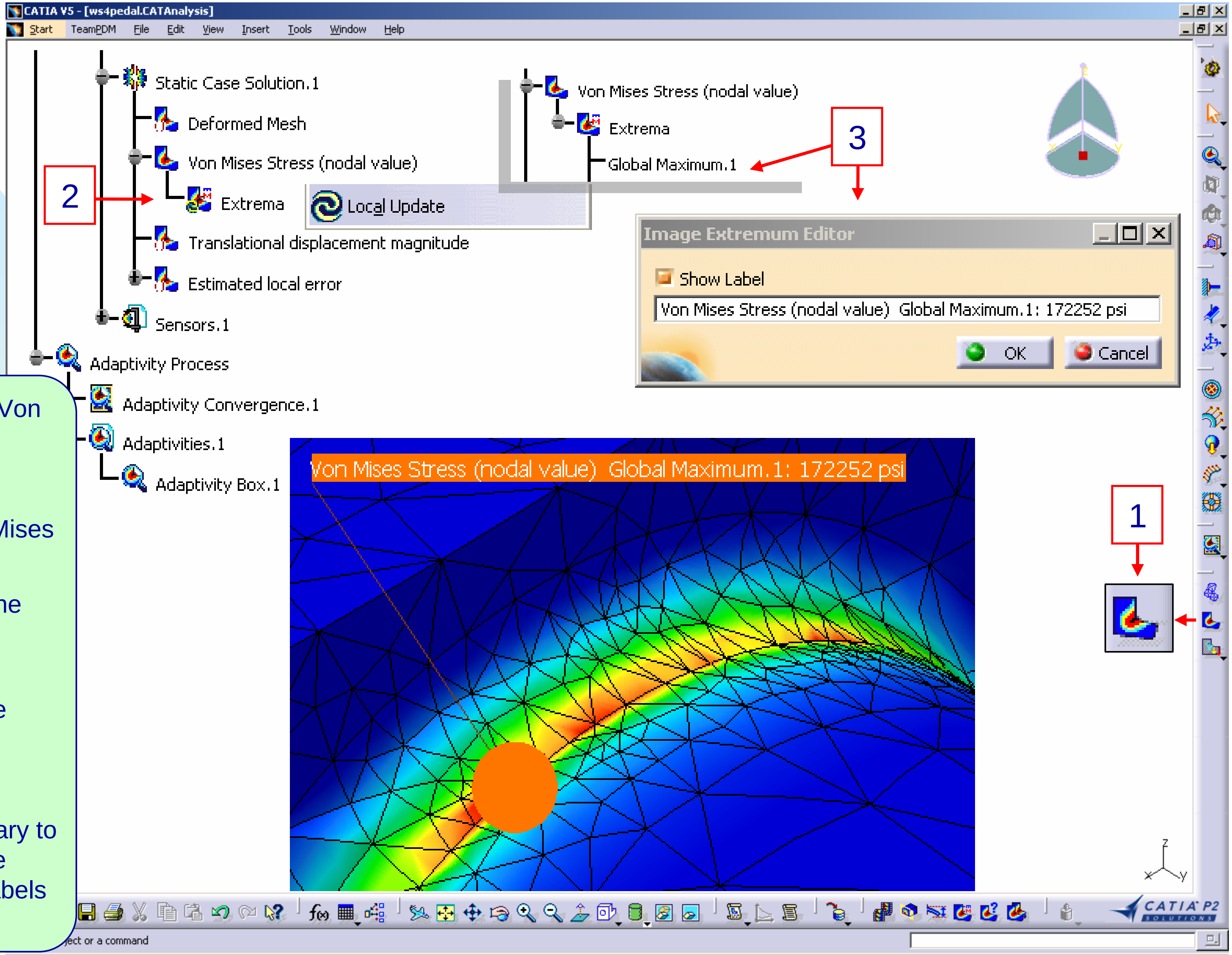
1. Double click Adaptivty Box.1.

2. Local Error not below 10%, but for this class we will continue with our results.

Time and CPU space permitting you should continue to adapt and converge until you get less than 10%.

Use the compass to move the adaptive box around. Notice the local error will update relative to the elements enclosed by the box.

Step 9. Visualize the adaptive analysis results



Precise maximum Von Mises stress.

Steps:

1. Select the Von Mises icon.

2. Locally update the Extrema object.

3. Double click on Global Max.1 in the features tree.

It might be necessary to delete and recreate Extrema to bring labels out of no show.

Step 9. Visualize the adaptive analysis results

Check cross sectional area stress values.

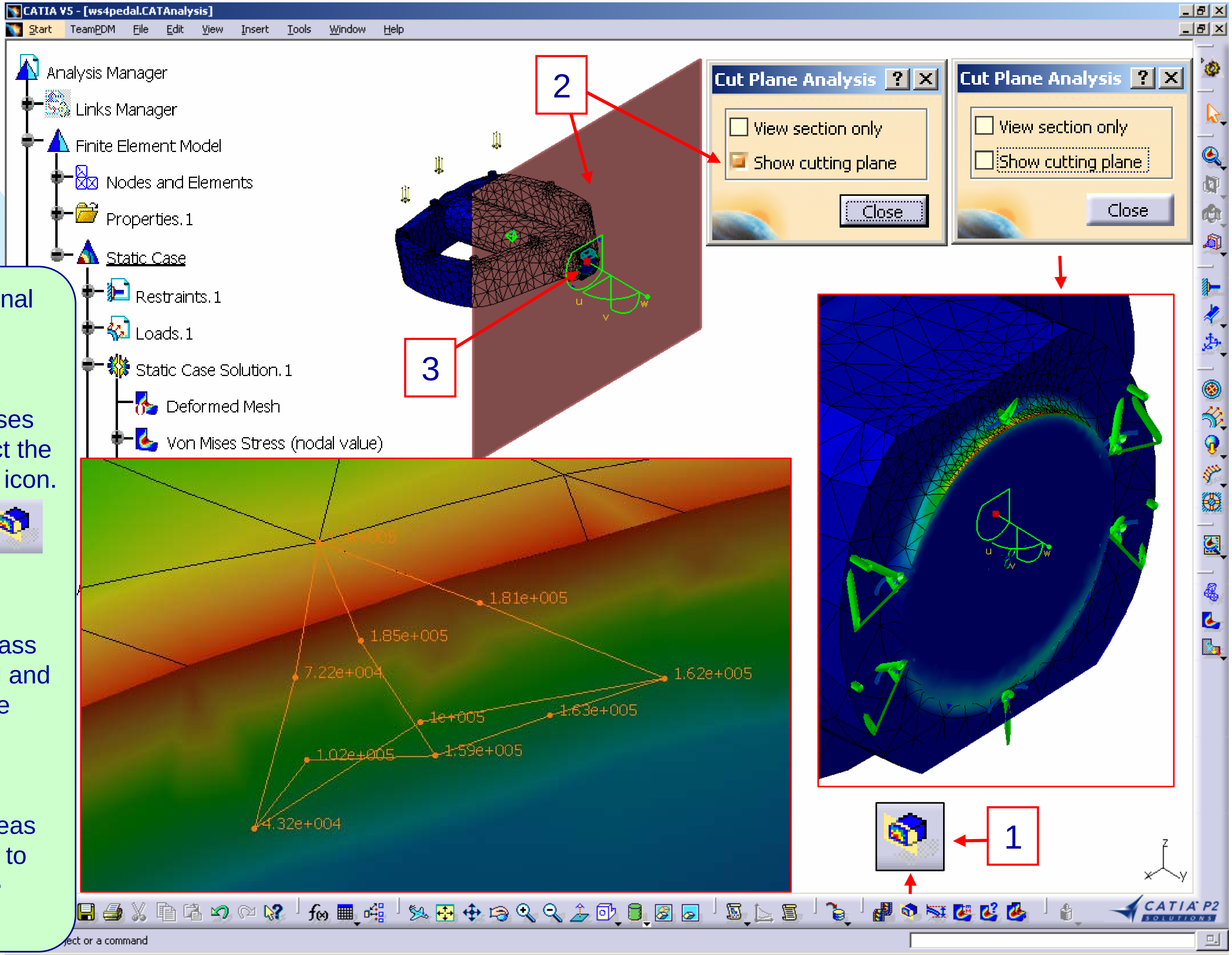
Steps:

1. With the Von Mises image active, select the Cut Plane analysis icon.

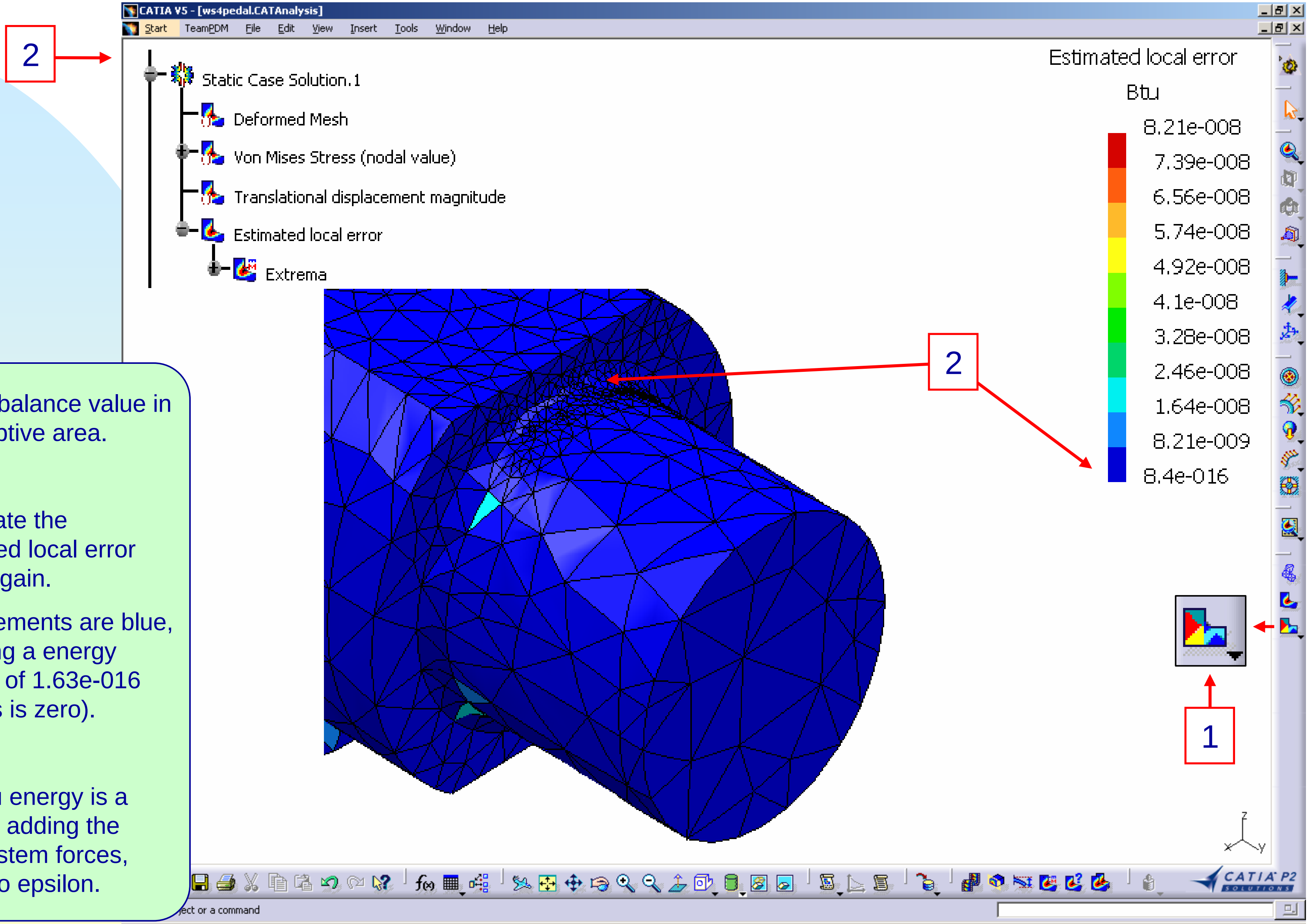
2. De-select Show cutting plane box.

3. Select the compass at the red dot, drag and locate normal to the shaft as shown.

Analyze various areas using the compass to drag and rotate the cutting plane.



Step 9. Visualize the adaptive analysis results



Energy balance value in the adaptive area.

Steps:

1. Activate the Estimated local error image again.

2. All elements are blue, indicating a energy balance of 1.63e-016 Btu (this is zero).

This Btu energy is a result of adding the FEM system forces, similar to epsilon.

Step 10. Verify reactions

Create an analysis sensor to verify reaction tensors on the clamp.

Steps:

1. Right click Sensor.1 in the features tree and select Create Sensor.

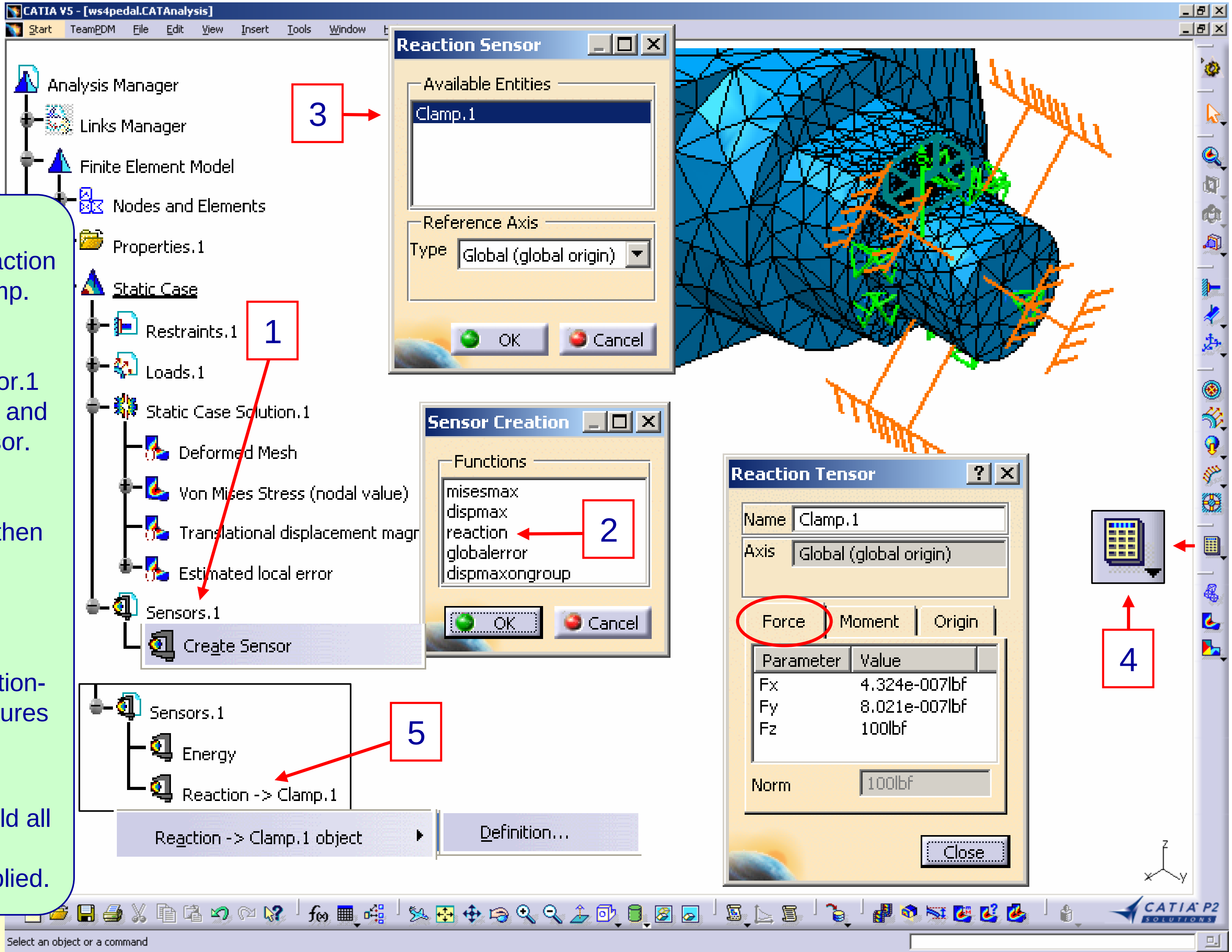
2. Select reaction.

3. Select Clamp.1 then OK (note: ref axis options).

4. Re-compute.

5. Right click Reaction-Clamp.1 in the features tree and select definition.

These values should all add up to zero and match our load applied.

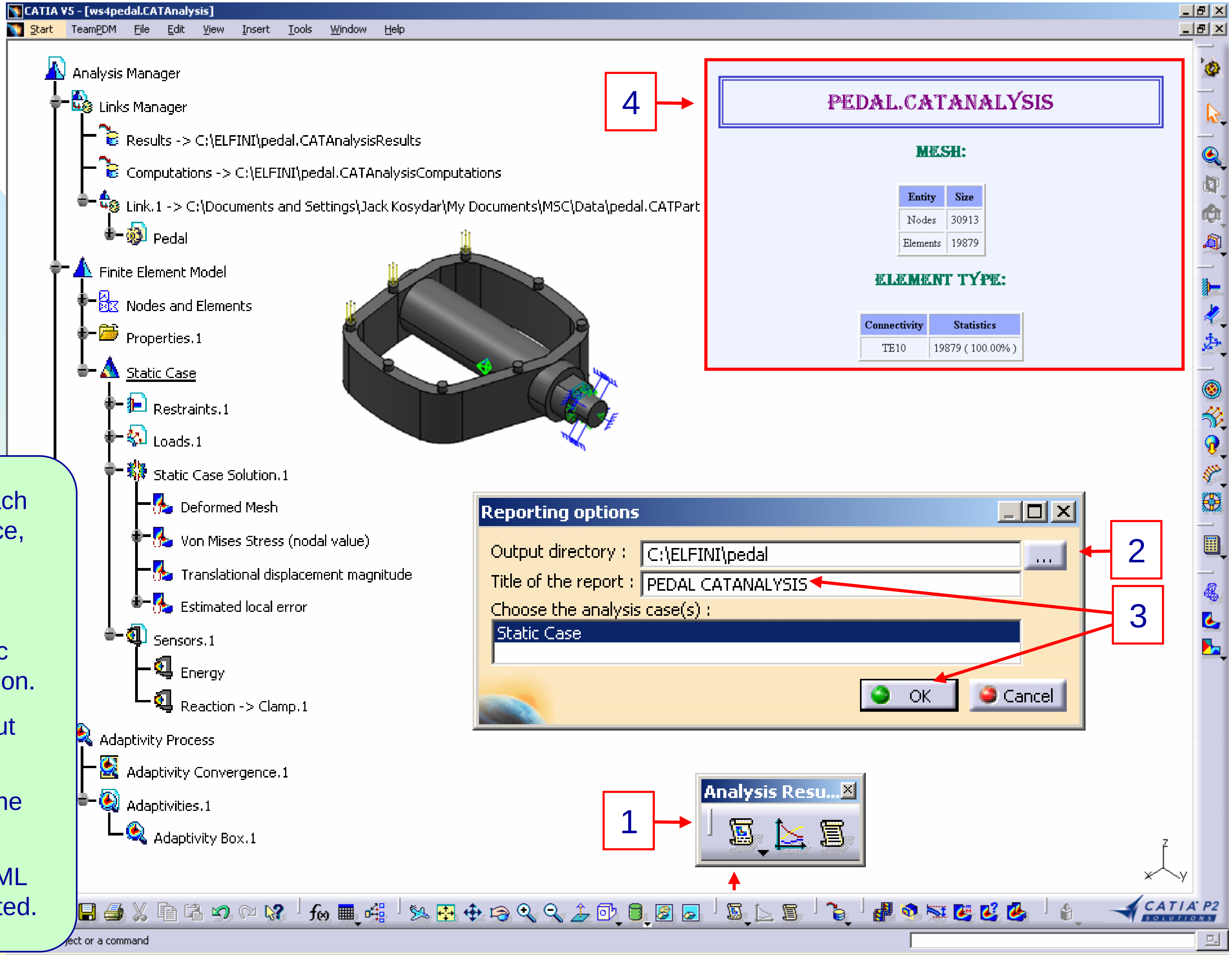


Step 11. Generate a basic analysis report

- **Conclusions**
  - The load set of a 200 lb man will overstress the pedal made of A36 steel. Use 4340 material and heat treat to 260-280 BHN for a yield strength equal to 217 ksi. You must change the material type and characteristics in the .CATPart document.
  - To add a different material to the CATIA material selector  or to create your own material catalog see Info Nugget – Materials Catalog.

	.25" Linear Mesh, .025 sag	.25"Linear Global Mesh, .025" sag .125" Parabolic Local Mesh, .013" sag Adapt and converge target of 5% locally
Max Von Mises	**24.6 ksi**	**172.3 ksi**
Translational Displacement	**.0047 inch**	**.00562 inch**
Error Estimate	**8.65e-8 Btu**	**8.4e-16 Btu local**
Global % Precision error **Local % Precision error**	**43.5 %** **NA %**	**18.1 %** **12.4 %**

Step 11. Generate a basic analysis report



After activating each image at least once, generate a report.

Steps:

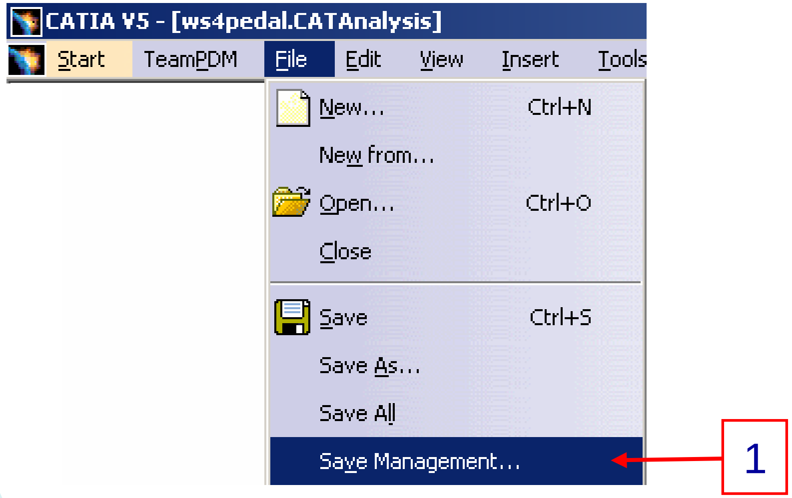
1. Select the Basic Analysis Report icon.

2. Select an Output directory.

3. Key in Title of the report, select OK.

4. Review the HTML report that is created.

Step 12. Save the analysis document

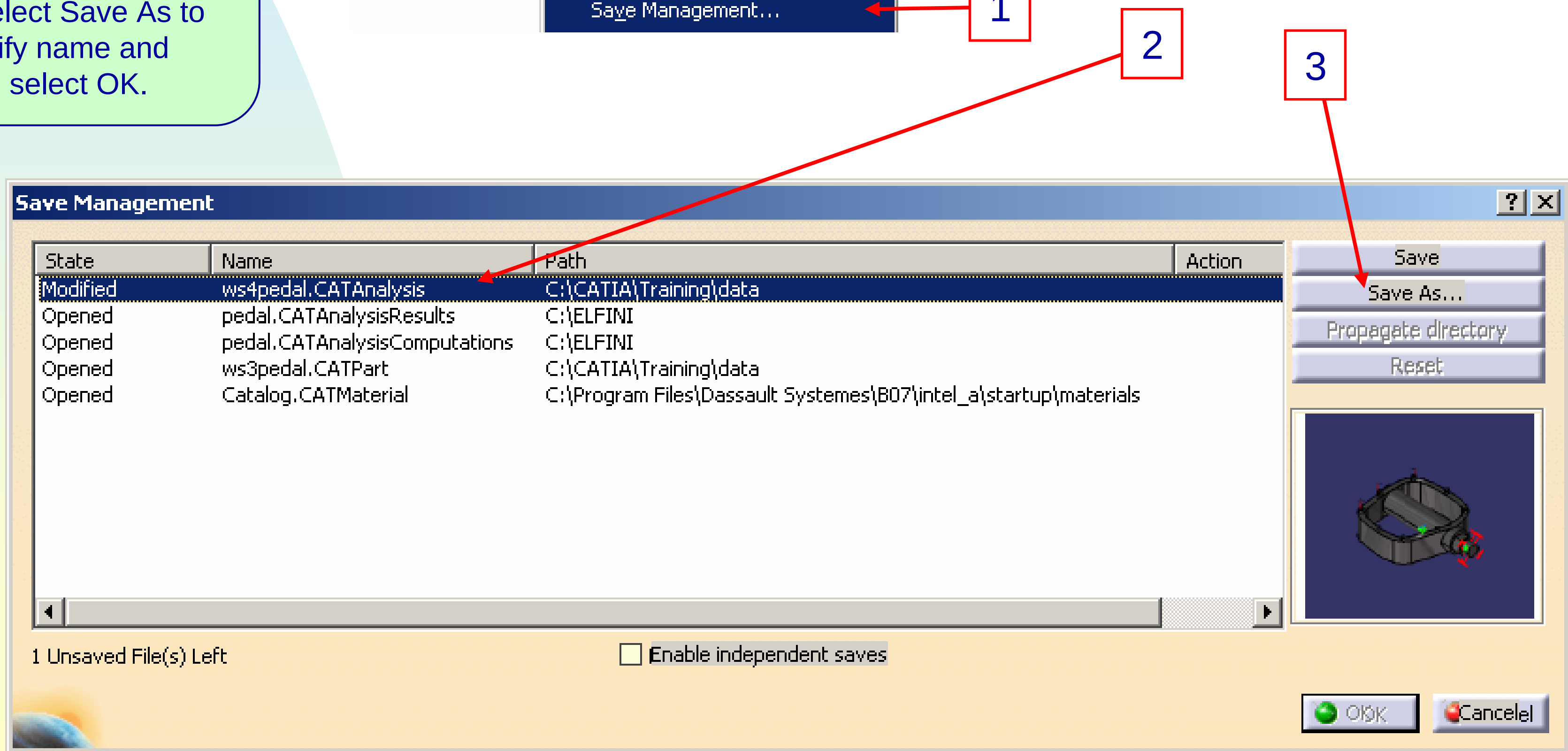


Steps:

1. From the File menu select Save Management.

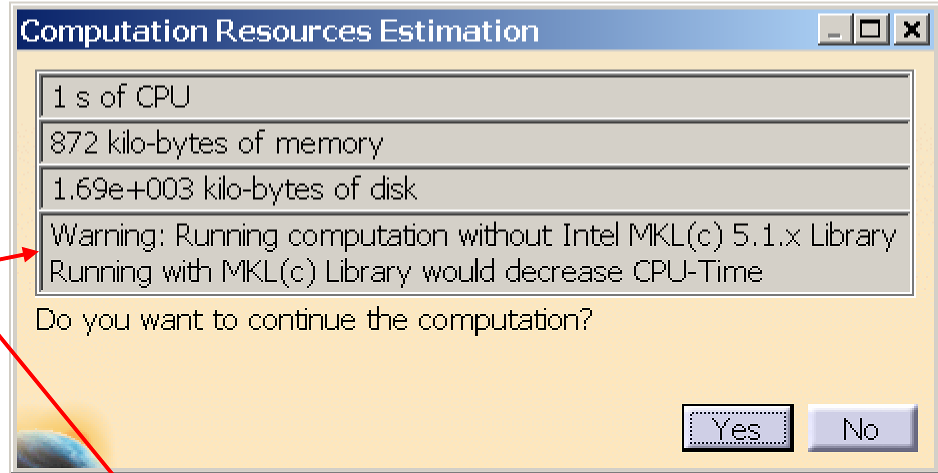
2. Highlight document you want to save.

3. Select Save As to specify name and path, select OK.



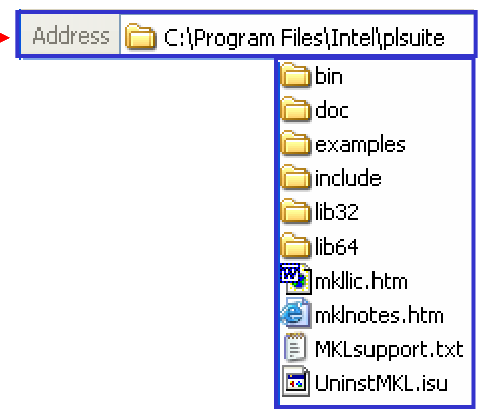
Info Nugget – Running with the Intel MKL Library

1



http://intel.com/home/tech/resource_library.htm

2



Installing the Intel Library to increase computing time.

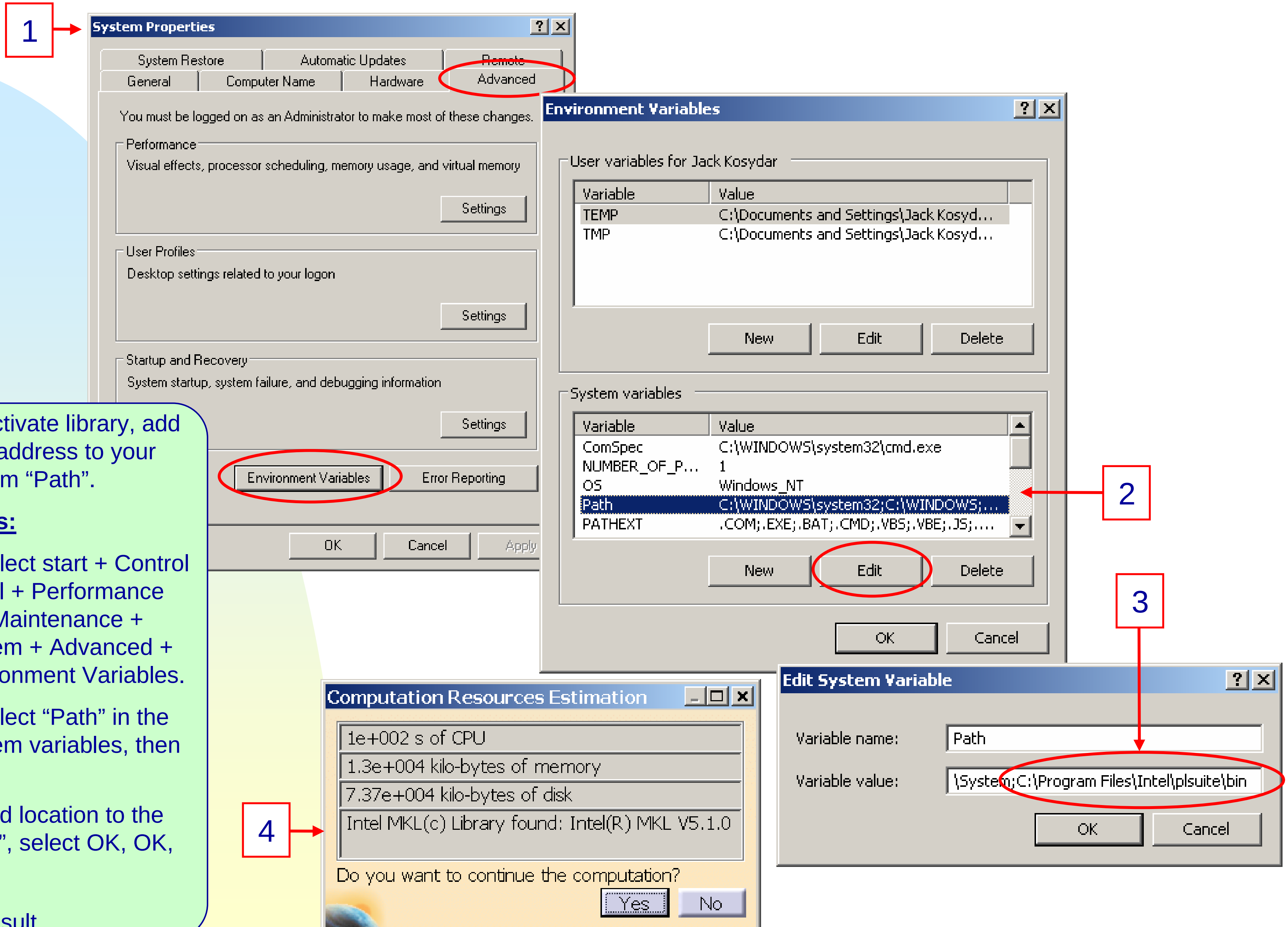
Steps:

1. This Intel Library should be downloaded and installed.

2. Example location of installed MKL51B.exe.

You must also add this location to your system “path”. See next page.

Info Nugget – Running with the Intel MKL Library



To activate library, add Intel address to your system "Path".

Steps:

1. Select start + Control Panel + Performance and Maintenance + System + Advanced + Environment Variables.

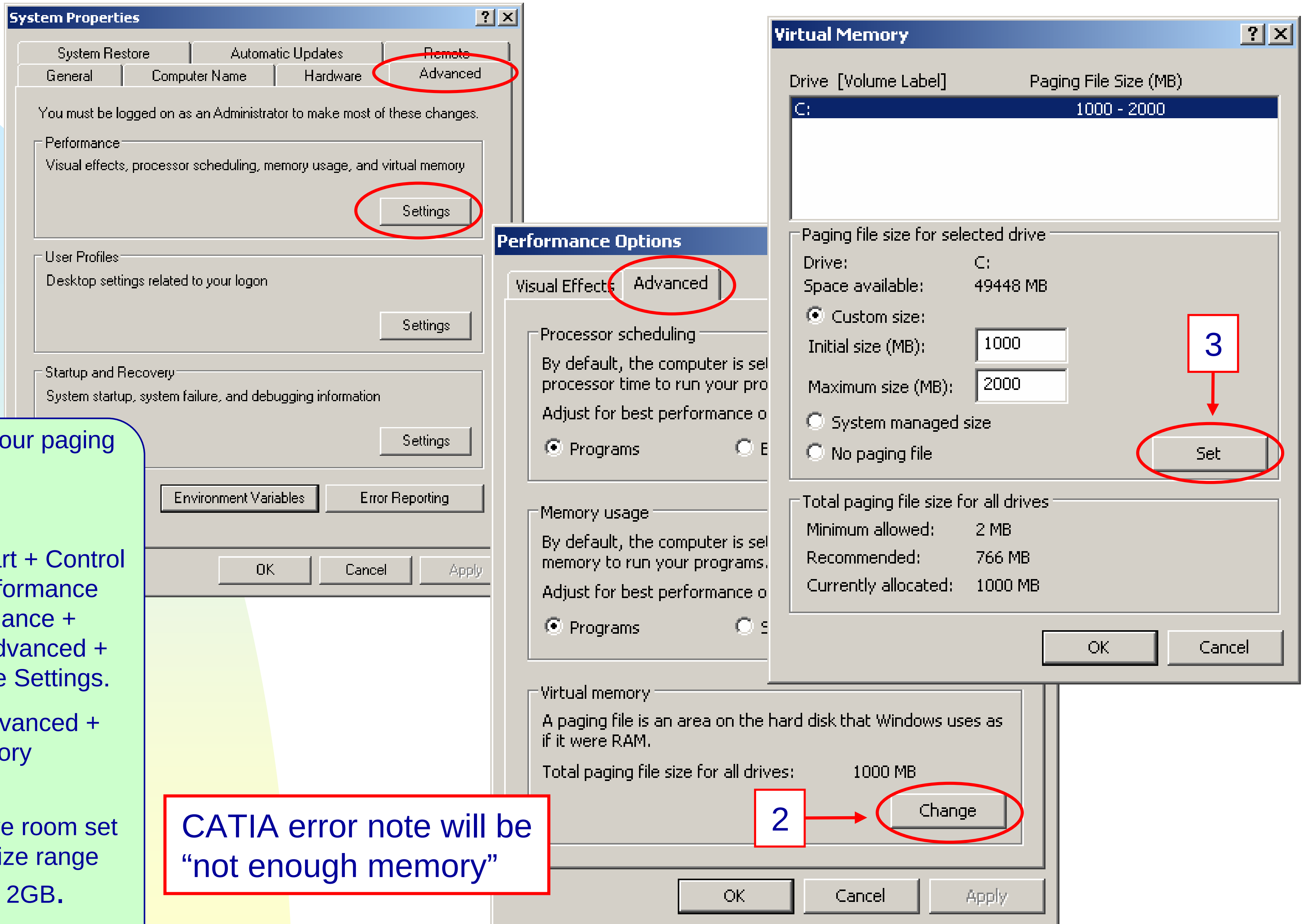
2. Select "Path" in the System variables, then Edit.

3. Add location to the "path", select OK, OK, OK.

4. Result

MSC SOFTWARE

Info Nugget – Paging Space



Increasing your paging space.

Steps:

1. Select start + Control Panel + Performance and Maintenance + System + Advanced + Performance Settings.

2. Select Advanced + Virtual memory Change.

3. If you have room set paging file size range from 1GB to 2GB.

4. Select OK, OK, OK.

CATIA error note will be “not enough memory”

Info Nugget – Batch Computing

Batch computing. This still seems to use your entire CPU resource unless you have a very powerful PC.

Steps:

1. Copy and Paste a duplicate of the CATIA launch icon on the desktop.

2. Rename to “CNextBatch”.

3. Add “–batch –e CATAnalysisBatch” to target location.

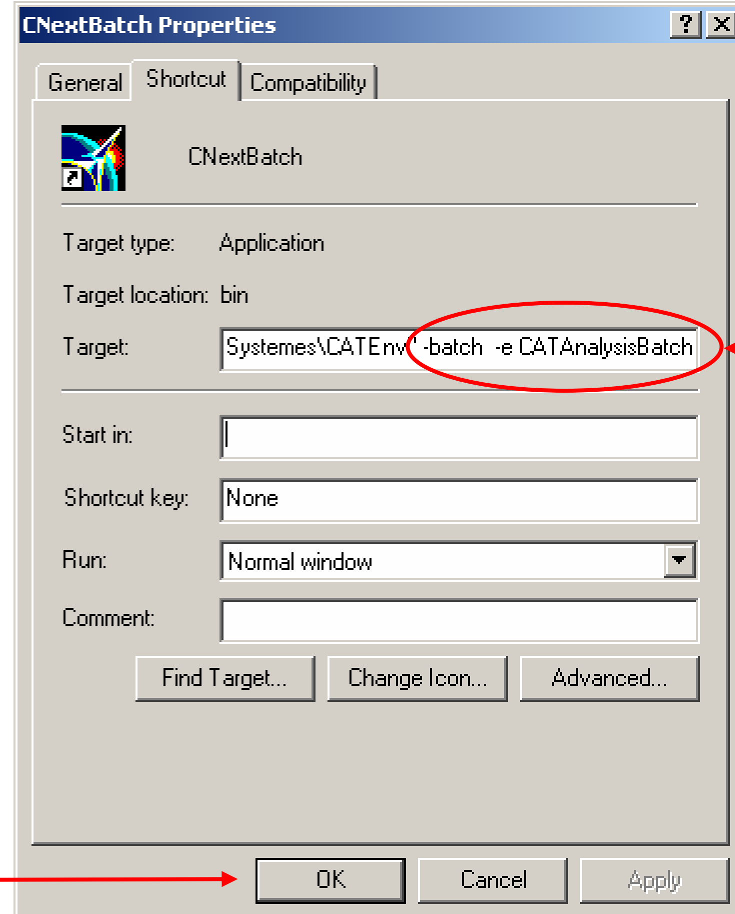
4. Select OK.

5. Result of double clicking

1

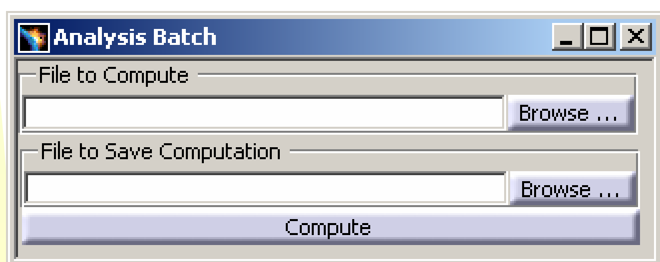
2

3

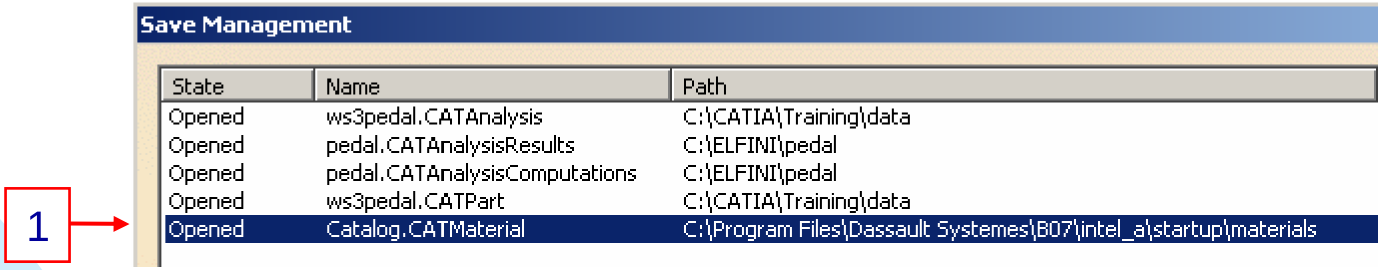


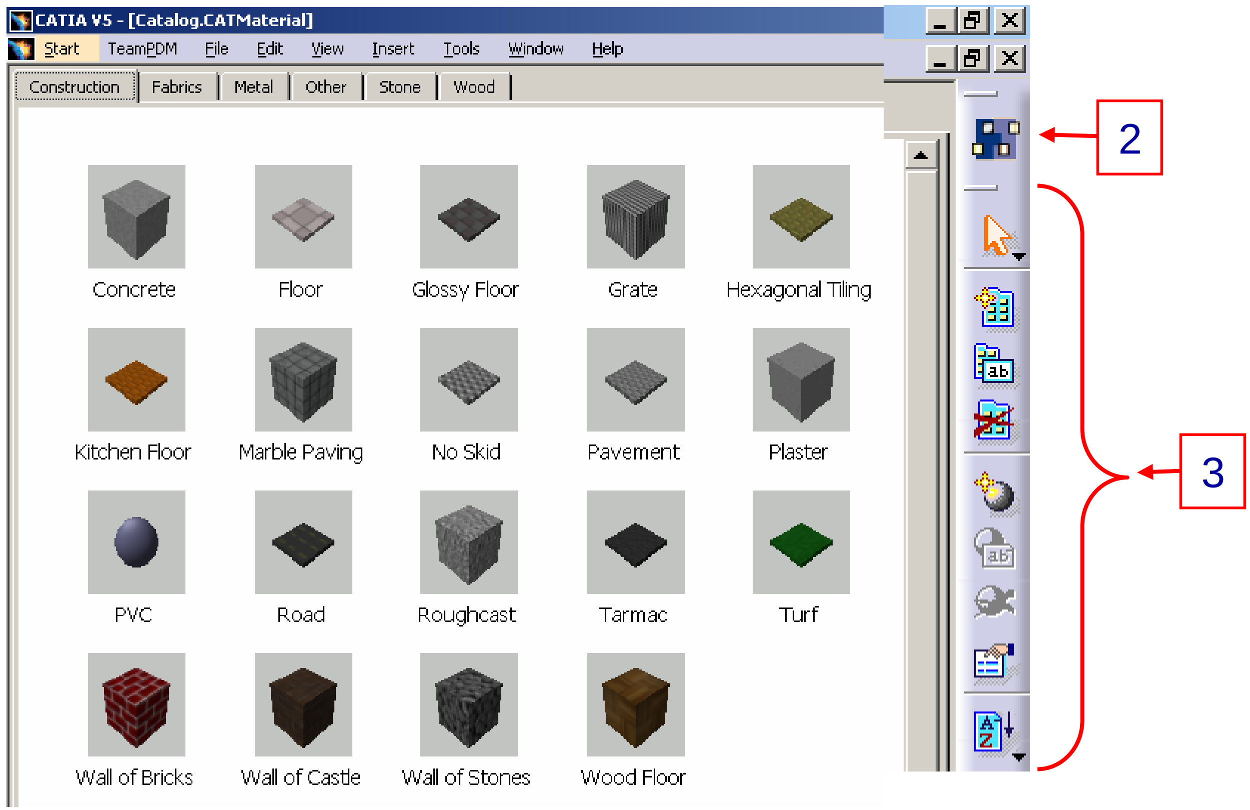
4

5



Info Nugget – Material Catalog





Edit the existing material catalog.

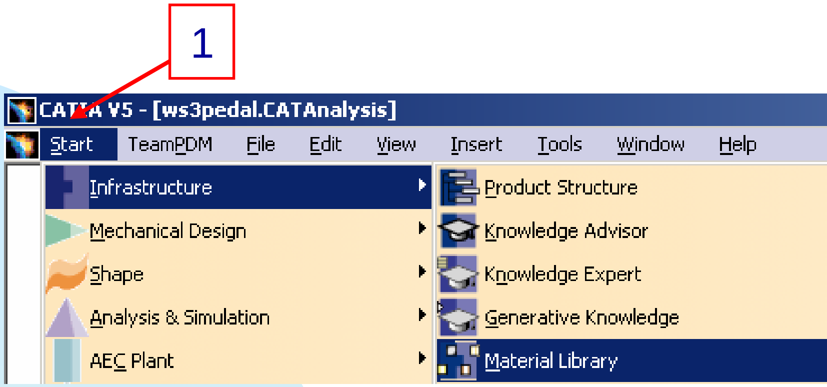
Steps:

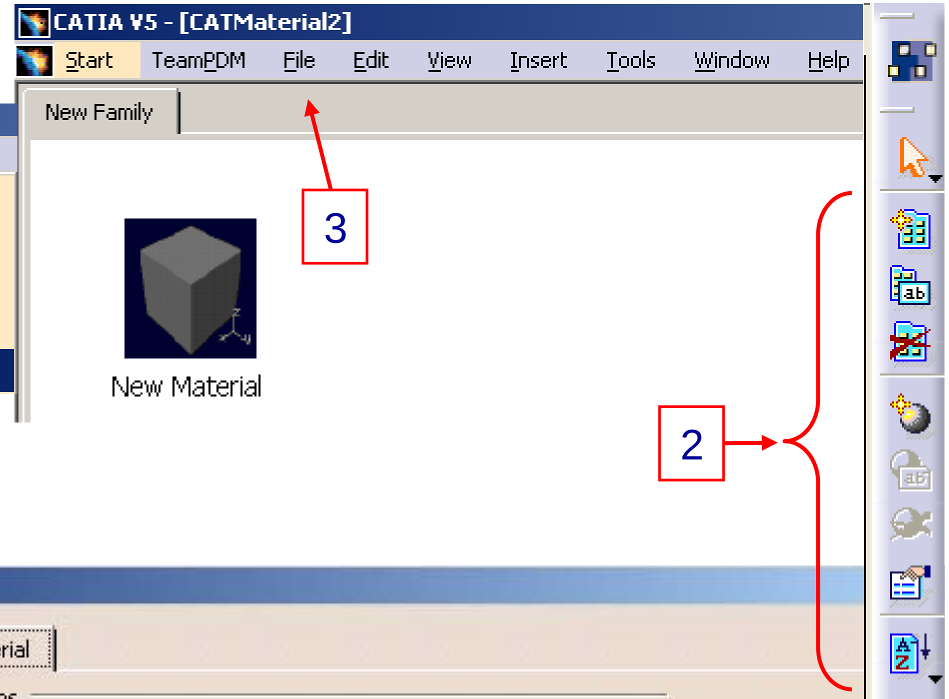
1. Locate the existing Catalog.CATMaterial file by Selecting File + Save management. The path will show if "Link to File" was selected when applying material.

2 Open the file in a CATIA session. The Material Library workbench will start.

3. Edit this file with the various tools provided and save.

Info Nugget – Personal Material Catalog





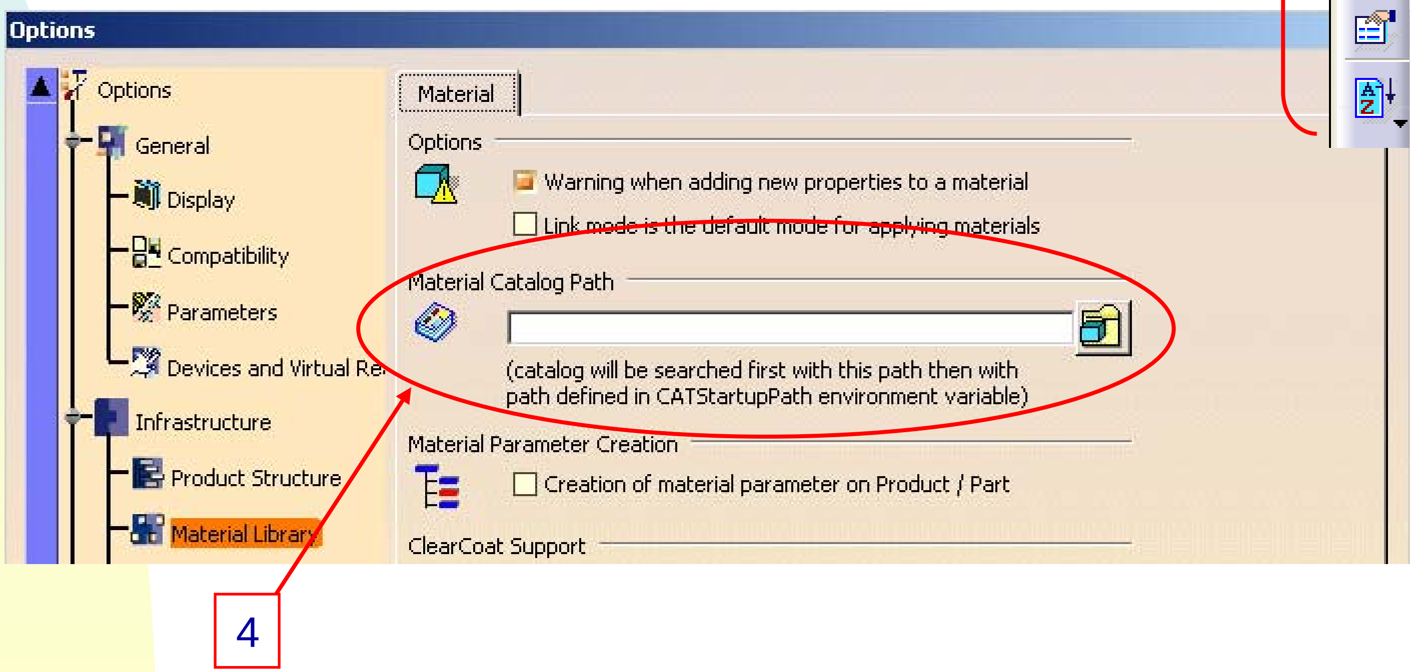
Steps:

1. Start Material Library workbench.

2. Create all your specific materials with the tools provided.

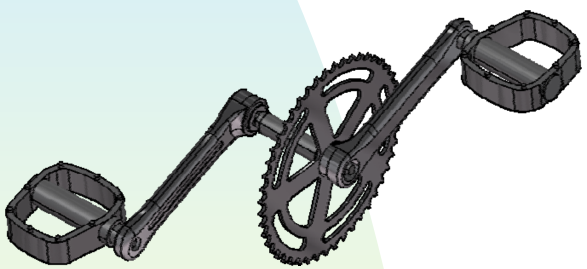
3. File + Save with the name **Catalog.CATMaterial**. It must have this exact name to work.

4. Modify Tools + options material catalog path to match where your personal material catalog is filed. This icon will then launch your materials. 



WORKSHOP 5

CRANK ANALYSIS USING VIRTUAL PARTS

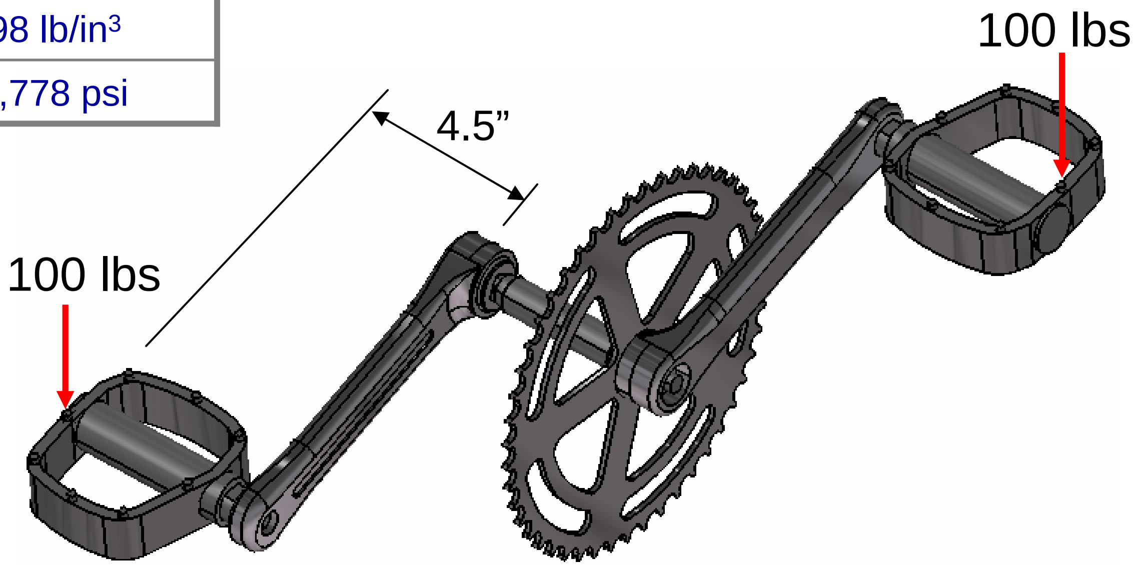


Problem Description

- The same 200 lb person riding this bike, standing balanced evenly on each peddle. Determine if the Crank material is capable of carrying this load.

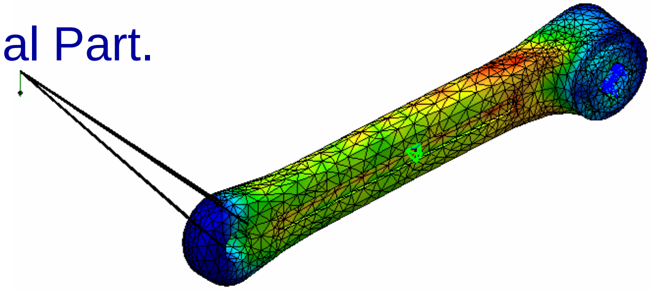
Aluminum

Elastic Modulus, E	10.15E6 psi
Poisson's Ratio, ν	0.346
Density	.098 lb/in^3
Yield Strength	13,778 psi



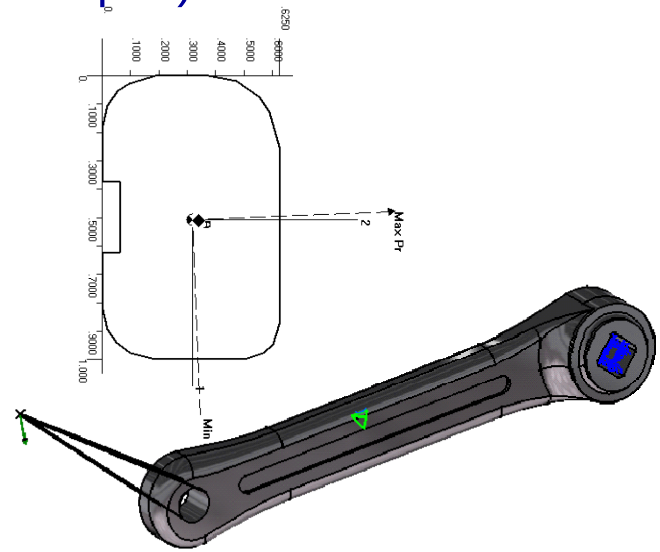
Suggested Exercise Steps

1. Open the existing CATIA part in the Part Design workbench.
2. Apply aluminum material properties to the part.
3. Create a new CATIA analysis document (.CATAnalysis).
4. Mesh globally with linear elements.
5. Apply a clamp restraint.
6. Simulate the pedal using a Smooth Virtual Part.
7. Apply a force to the Smooth Virtual Part.
8. Compute the initial analysis.
9. Visualize “hot spots” in the initial results.
10. Change mesh to parabolic and add local meshing.
11. Compute the more precise analysis.
12. Search for extrema points (max Von Mises, min precision).
13. Check local precision using adaptivity boxes.
14. Visualize final results (translations relative to user axis).
15. Save the analysis document.



2D DIAGRAM AND HAND CALCULATIONS

- Assume all 6 D.O.F. are restricted (clamped) where the crank attaches to the shaft.



SECTION PROPERTIES
AREA
A = 0.573283
MOMENTS OF INERTIA
I1 = 0.01650348
I2 = 0.04458936
I12 = 0.001097112
TORSIONAL CONST.
ABOUT CENTROID
J = 0.04158645
CENTROID RELATIVE
TO ORIGIN
HORI. = 0.50821763
VERT. = 0.3193818
SHEAR CENTER
REL. TO ORIGIN
HORI. = 0.51046425
VERT. = 0.34094304

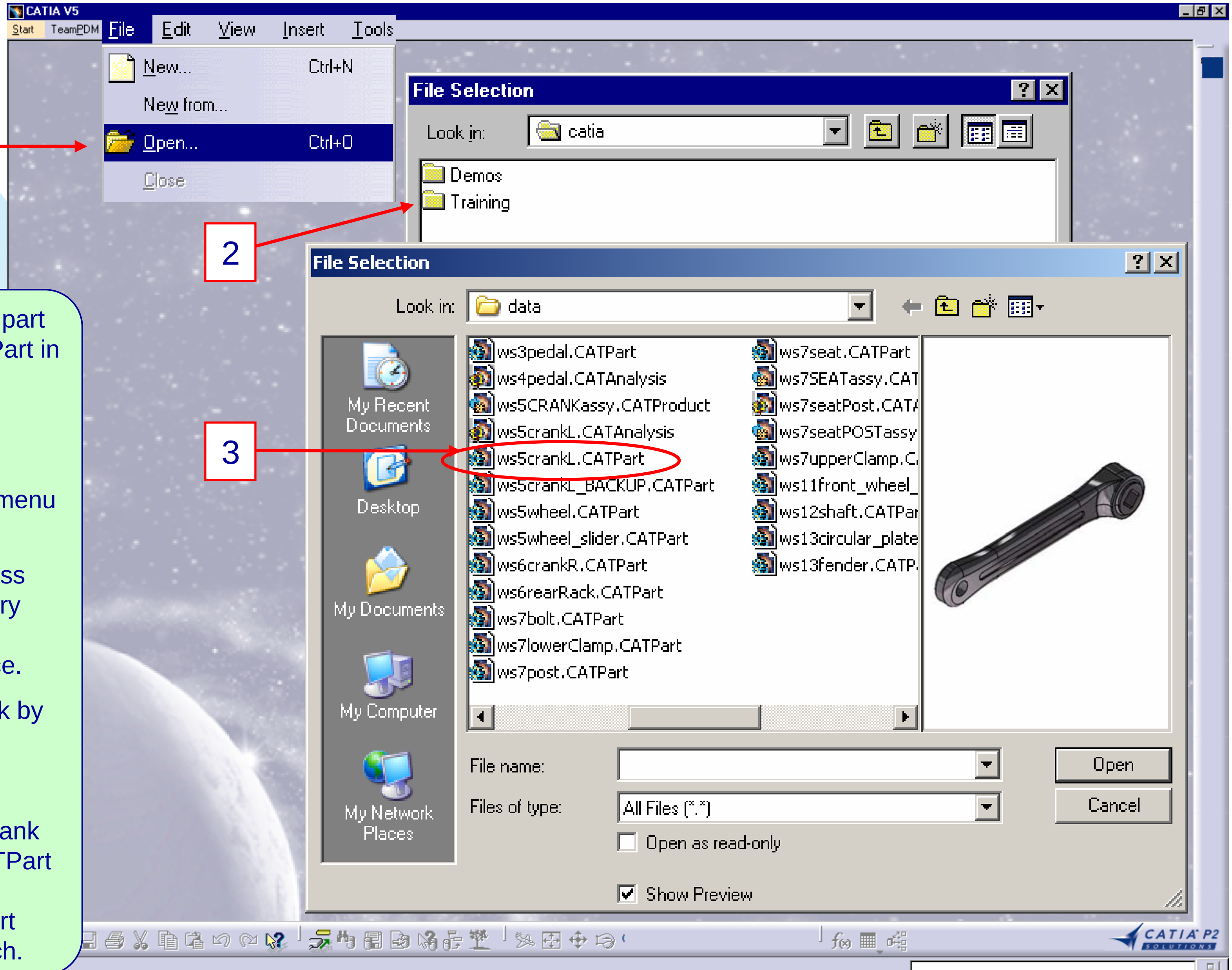
$$Moment = P \cdot L = 100 lbs \cdot 6.75 inches = 675\, inlbs$$

$$Bending = \frac{M \cdot c}{I} = \frac{675\, inlbs \cdot 0.508 inches}{0.0446\, in^4} = 7688 psi$$

$$Shear = \frac{T \cdot c}{J} = \frac{450\, inlbs \cdot 0.341\, inches}{0.0416\, in^4} = 3689 psi$$

$$Combined = \frac{\sigma_x + \sigma_y}{2} + \sqrt{\left(\frac{\sigma_x - \sigma_y}{2}\right)^2 + \tau^2_{xy}} = \frac{7688 psi}{2} + \sqrt{\left(\frac{7688 psi}{2}\right)^2 + 3689 psi^2} = 9172 psi$$

Step 1. Open the existing CATIA part



Open the CATIA part ws5crankL.CATPart in the Part Design workbench.

Steps:

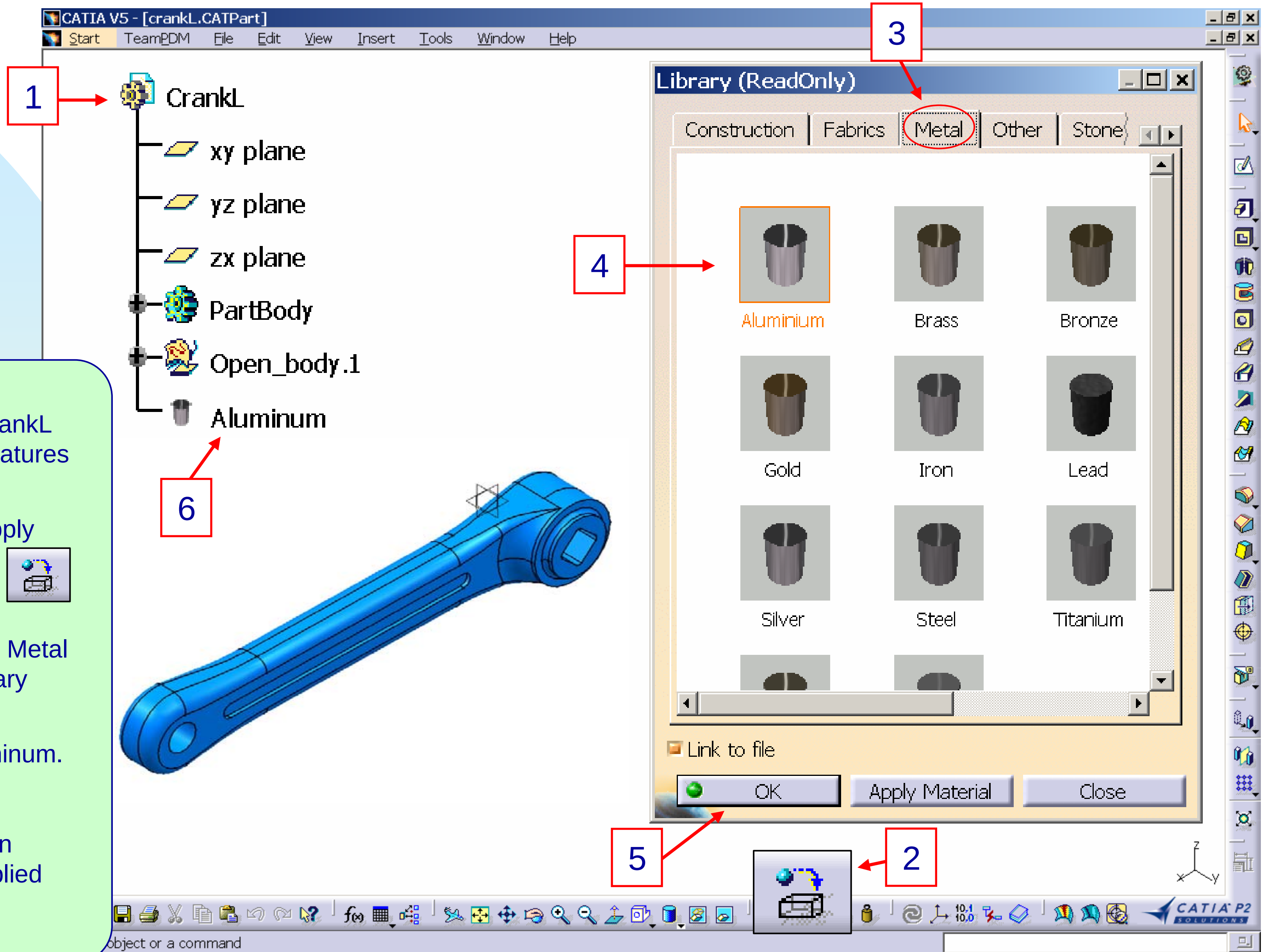
1. From the File menu select Open.

2. Access the class workshop directory using the typical Windows interface.

3. Open the crank by double-clicking.

By default, the crank and all other CATPart documents are opened in the Part Design workbench.

Step 2. Apply aluminum material properties to the part



Steps:

1. Click the CrankL "Part" in the features tree.

2. Click the Apply Material icon.

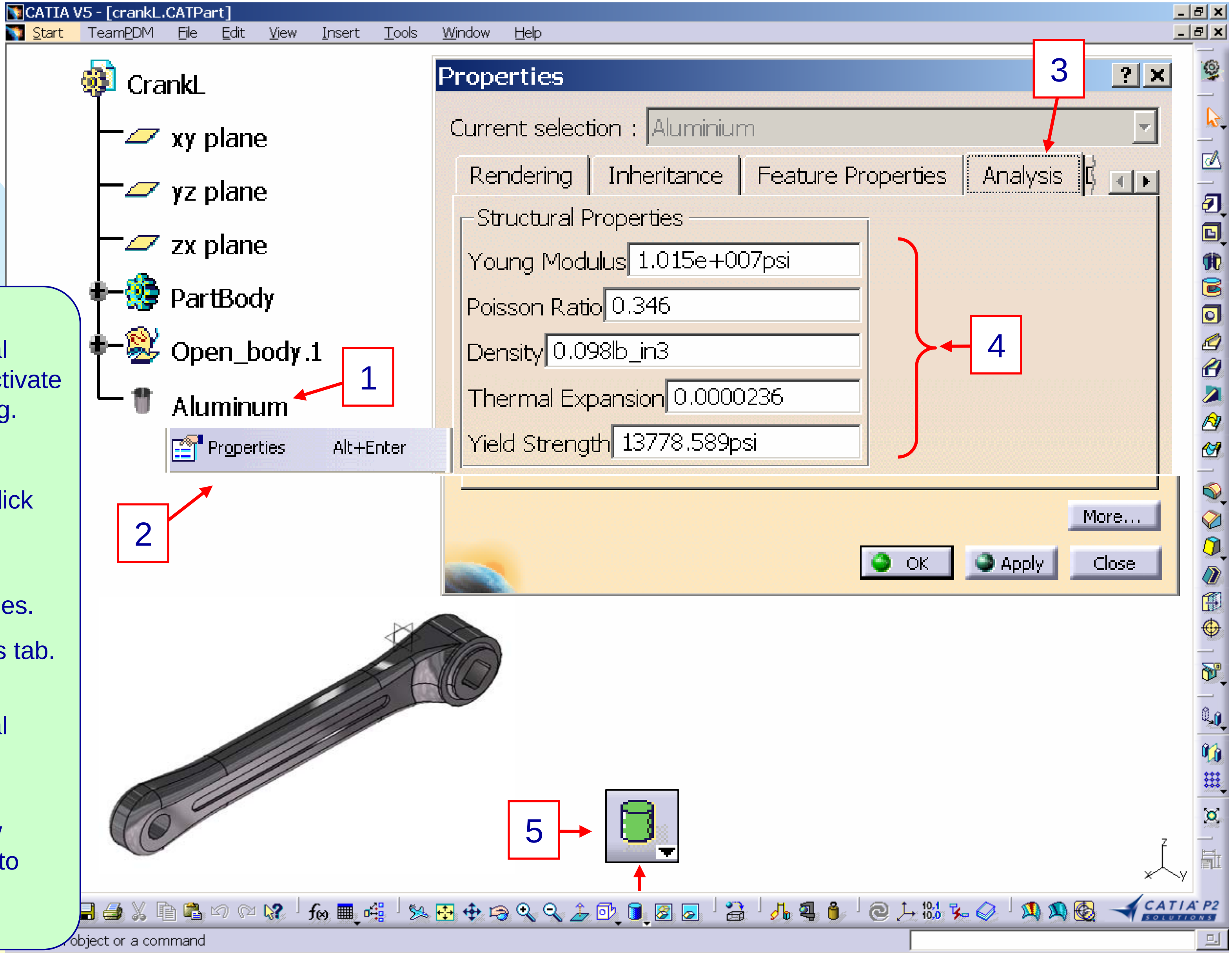
3. Activate the Metal tab in the Library window.

4. Select Aluminum.

5. Select OK.

6. Make certain material is applied properly in the features tree.

Step 2. Apply steel material properties to the part



Verify and edit structural material properties and activate material rendering.

Steps:

1. Right mouse click aluminum in the features tree.

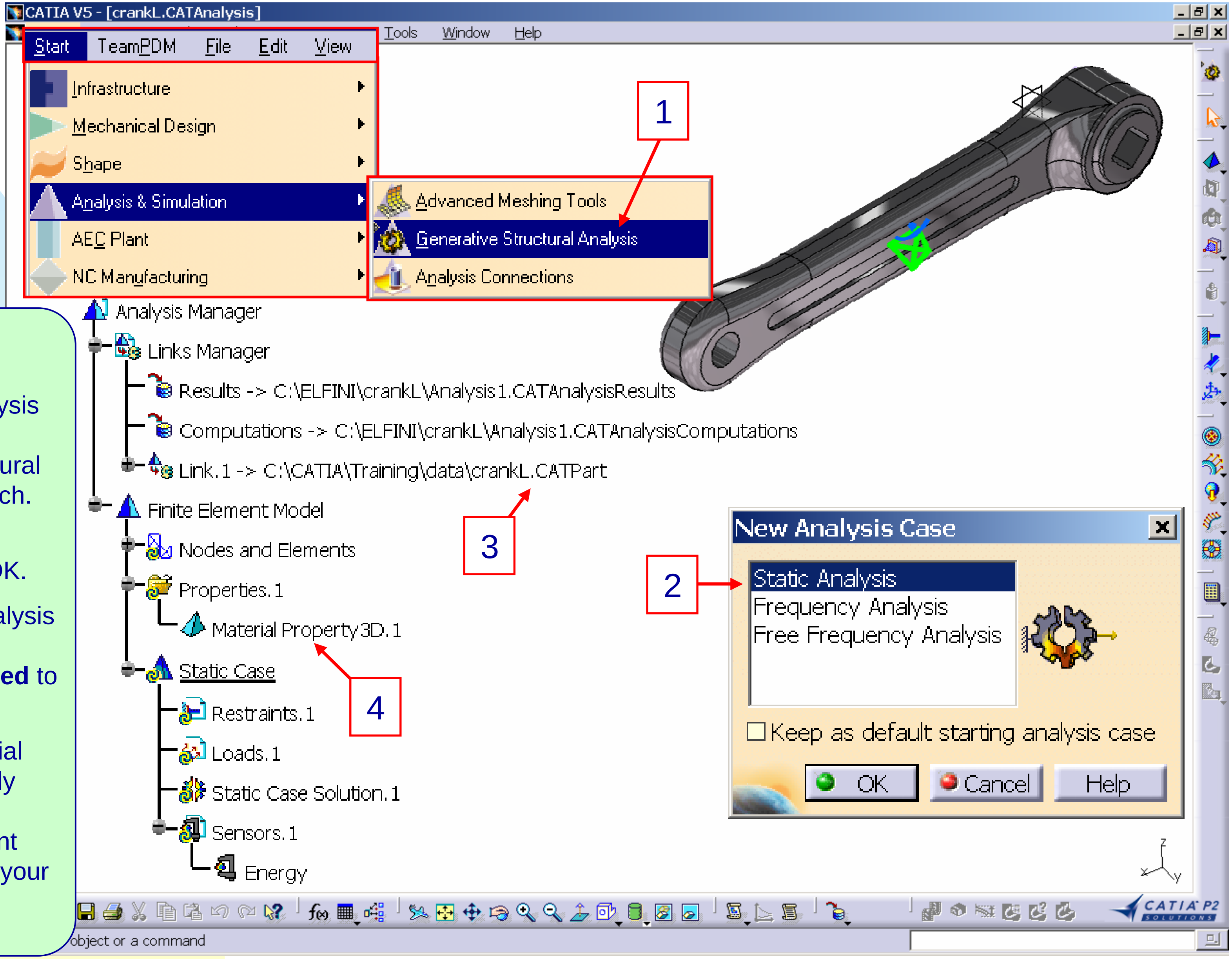
2. Select Properties.

3. Select Analysis tab.

4. Verify and edit structural material properties here.

5. Select the Customized View Parameters icon to activate material rendering.

Step 3. Create a new CATIA analysis document



Steps:

1. From the Start menu select Analysis & Simulation then Generative Structural Analysis workbench.

2. Select Static Analysis, select OK.

3. Your Static Analysis document gets automatically **linked** to the CATPart.

4. Note the material property previously specified in the CATPart document shows up here in your CATAnalysis document.

Step 3. Create a new CATIA analysis document

Specify the External Storage directory locations, results and computations names.

Steps:

1. Select the Storage Location icon.

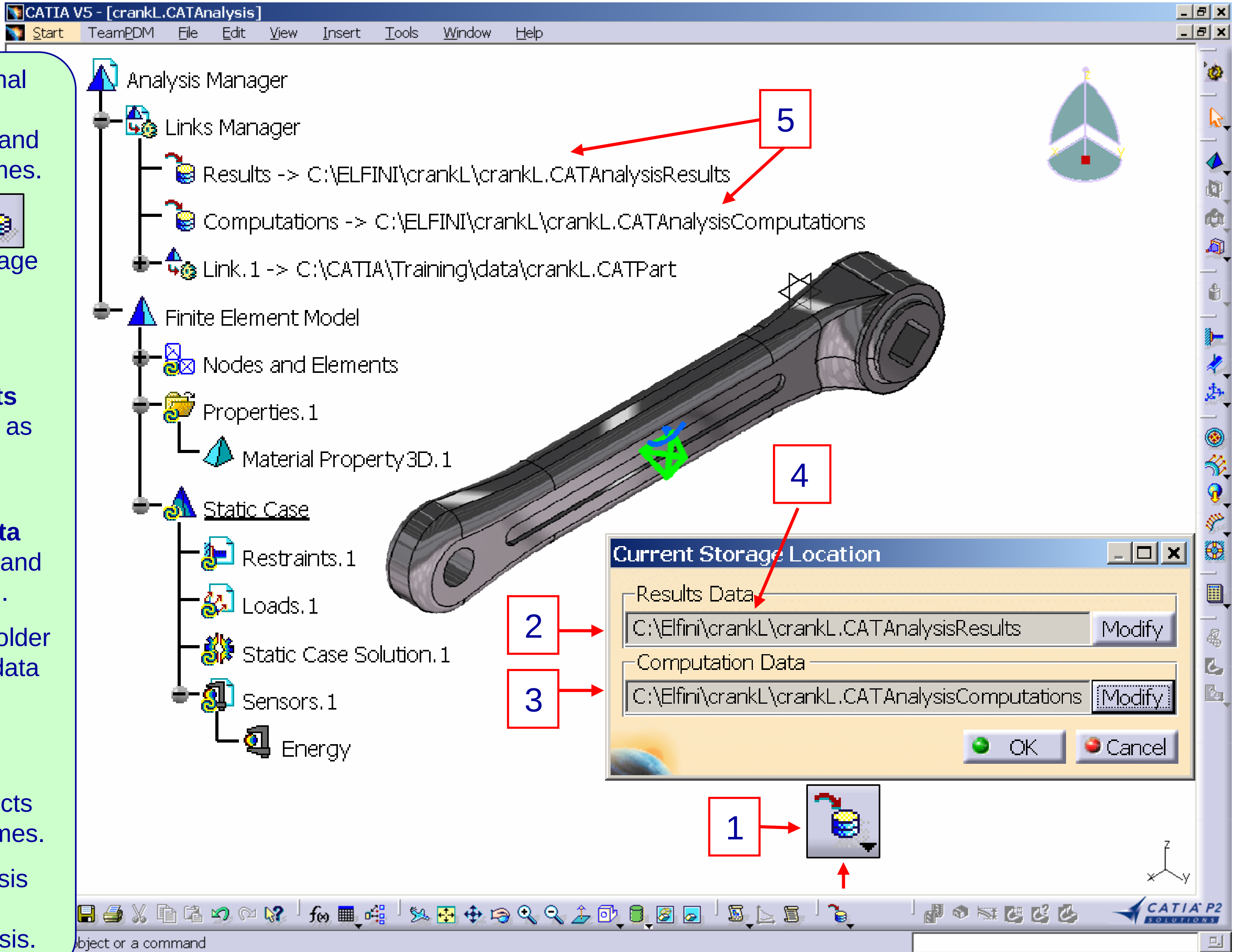
2. In the Current Storage Location modify the **Results Data** and rename as shown.

3. Modify the **Computation Data** Storage Location and rename as shown.

4. Create a new folder to keep analysis data segregated.

5. Note the Links Manager in the features tree reflects the paths and names.

6. Save the analysis document as crankL.CATAnalysis.



Step 4. Mesh globally with linear elements

Define the global finite element mesh properties.

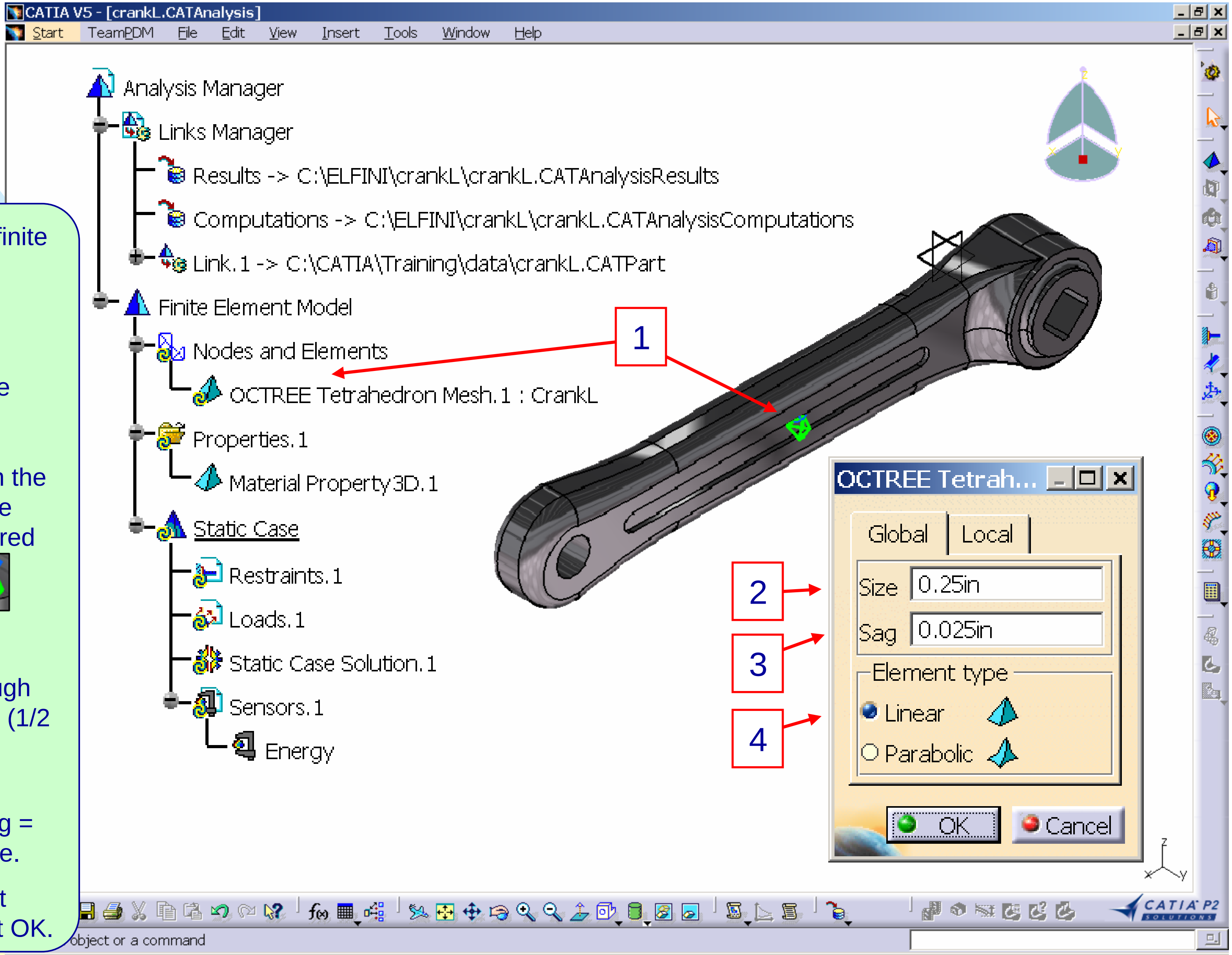
Steps:

1. Double Click the "OCTREE Tetrahedron Mesh.1:CrankL" in the features tree or the "Mesh" icon centered on the part. 

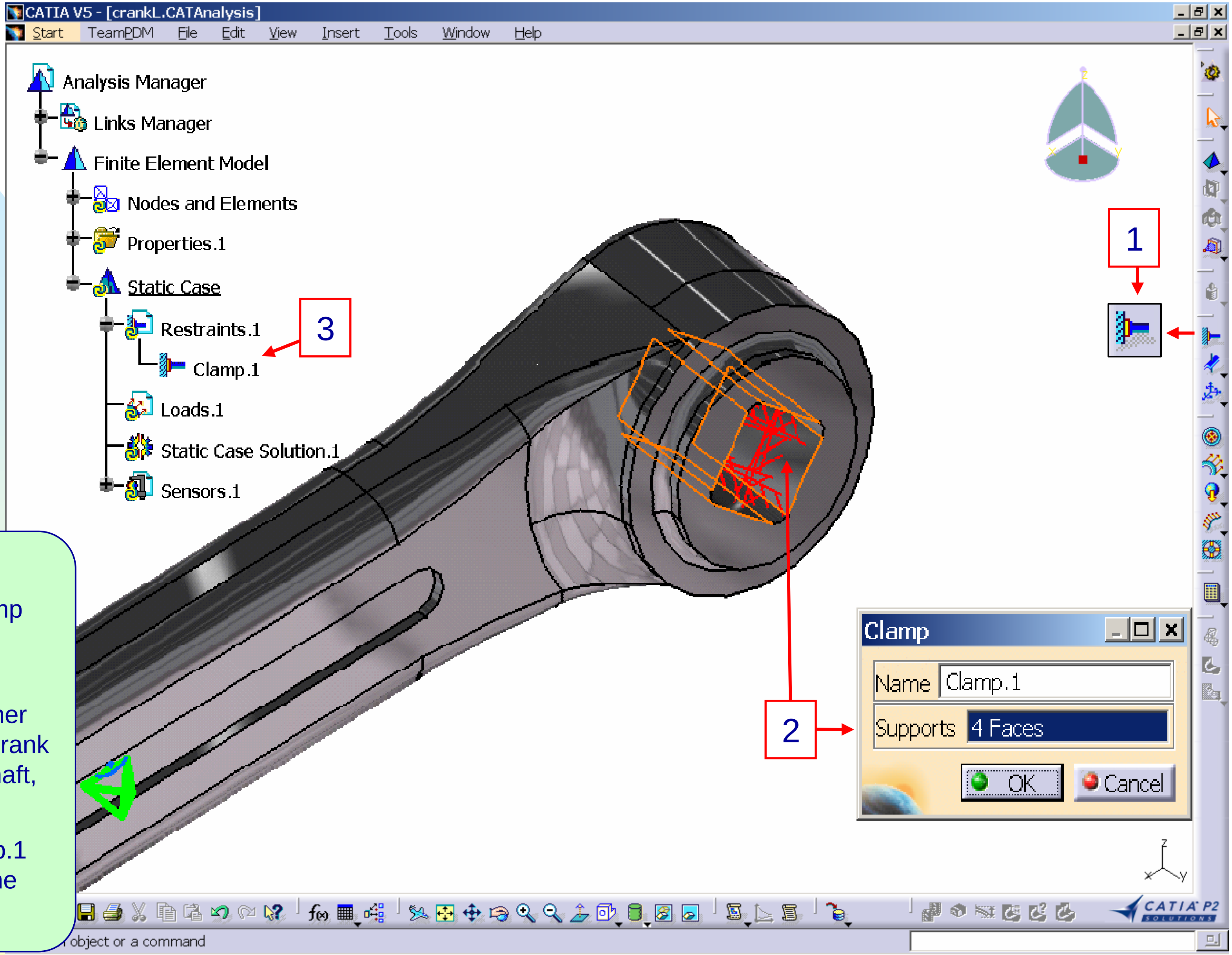
2. Specify the recommended rough Global Size = .25" (1/2 thinnest section).

3. Specify the recommended Sag = 10% of Global Size.

4. Specify element type Linear, select OK.



Step 5. Apply a clamp restraint



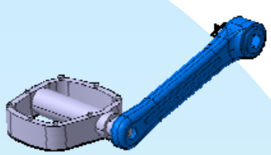
Steps:

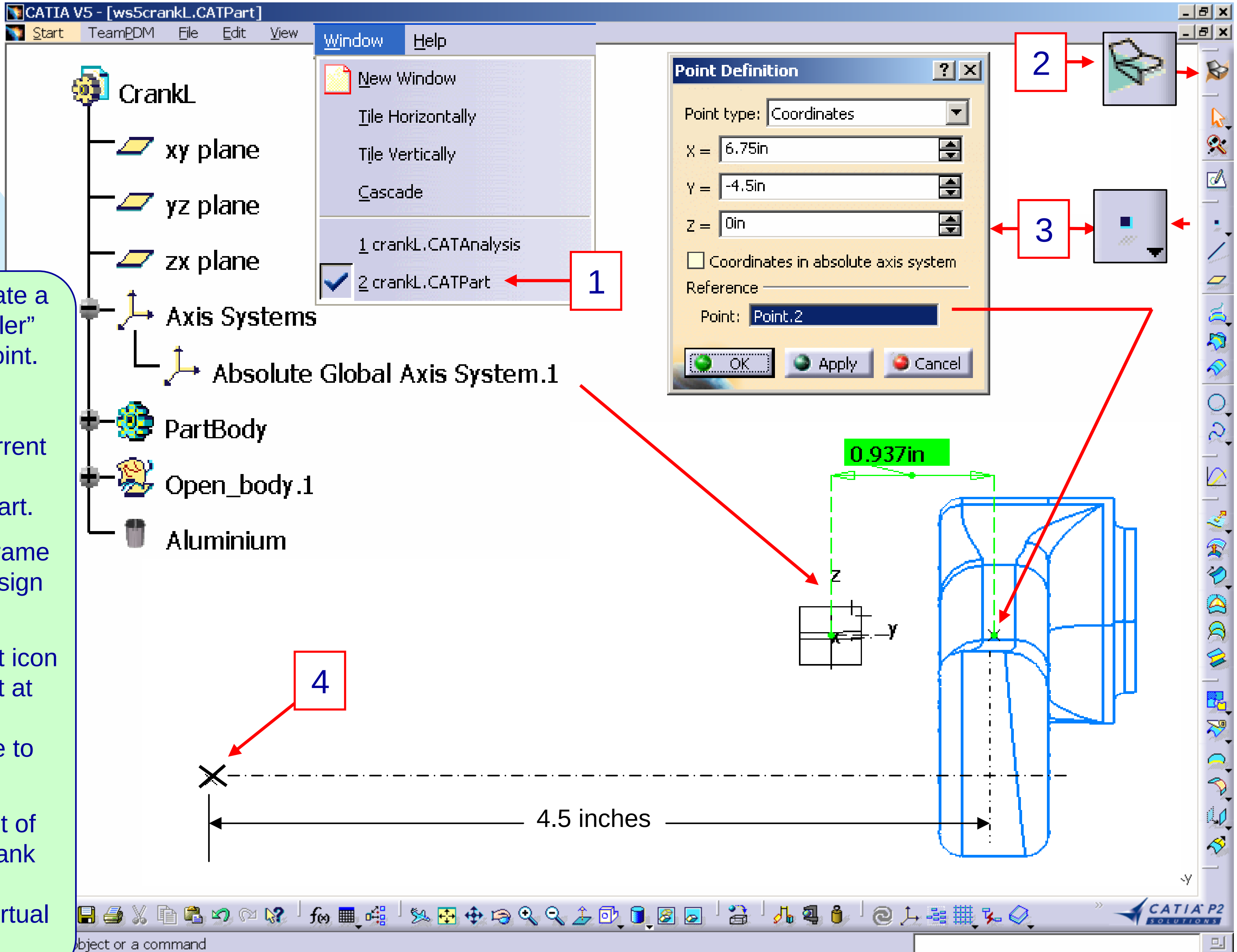
1. Select the Clamp Restraint icon.

2. Select the 4 inner faces where the crank attaches to the shaft, select OK.

3. Note the Clamp.1 object added to the features tree.

Step 6. Simulate the pedal using a Contact Virtual Part





We first must create a virtual "Part Handler" that is simply a point.

Steps:

1. Change the current document to ws5crankL.CATPart.

2. Start the Wireframe and Surfacing Design workbench.

3. Select the point icon and create a point at the coordinates shown. Reference to point.2. Click OK.

4. This is the point of load relative to crank centerline (Part Handler for our Virtual Part).

MSC SOFTWARE

Step 6. Simulate the pedal using a Smooth Virtual Part

Steps:

1. Change the current document back to crankL.CATAnalysis.

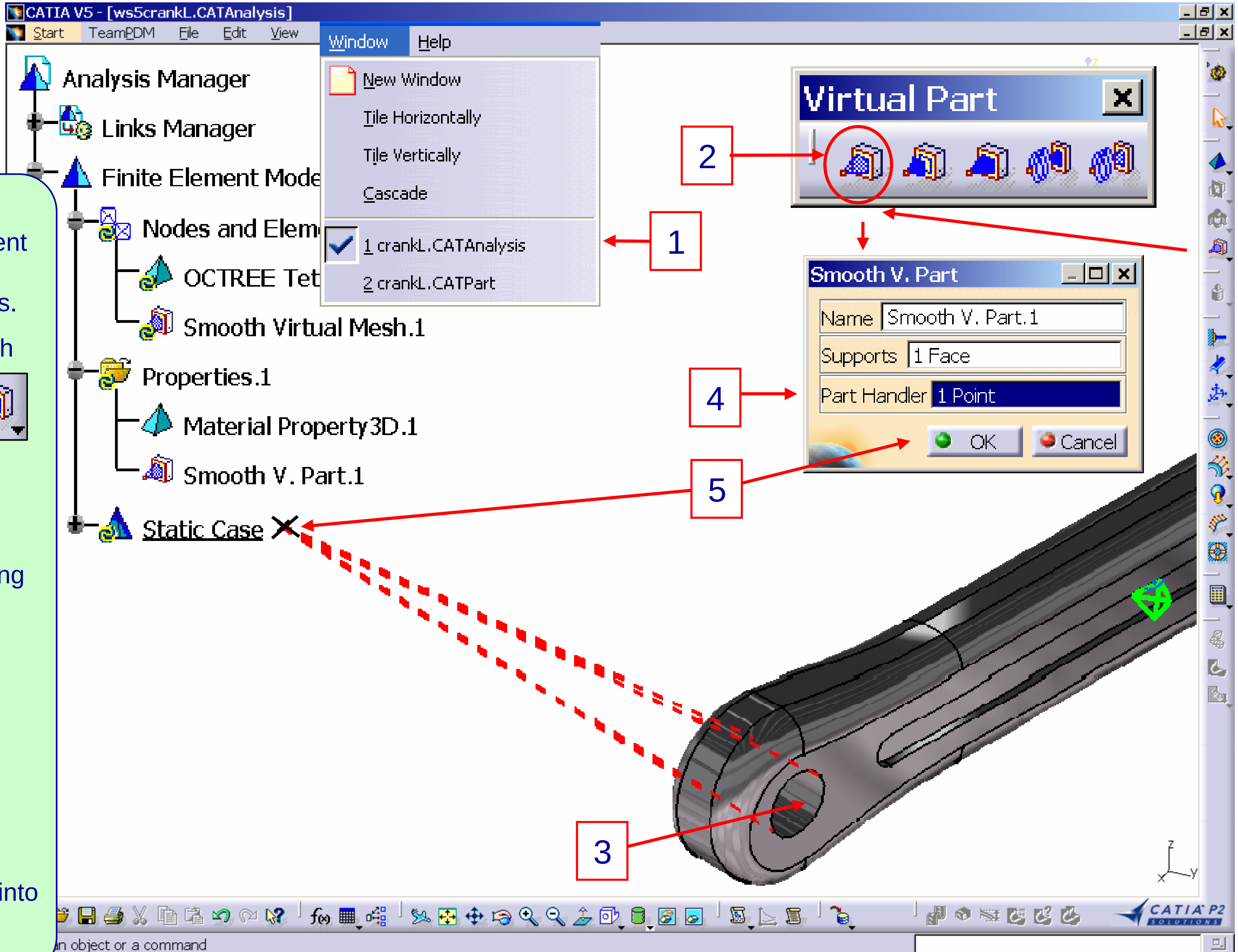
2. Select the smooth virtual part icon.

3. Select the face where the pedal attaches.

4. Activate by clicking in the Part Handler input box.

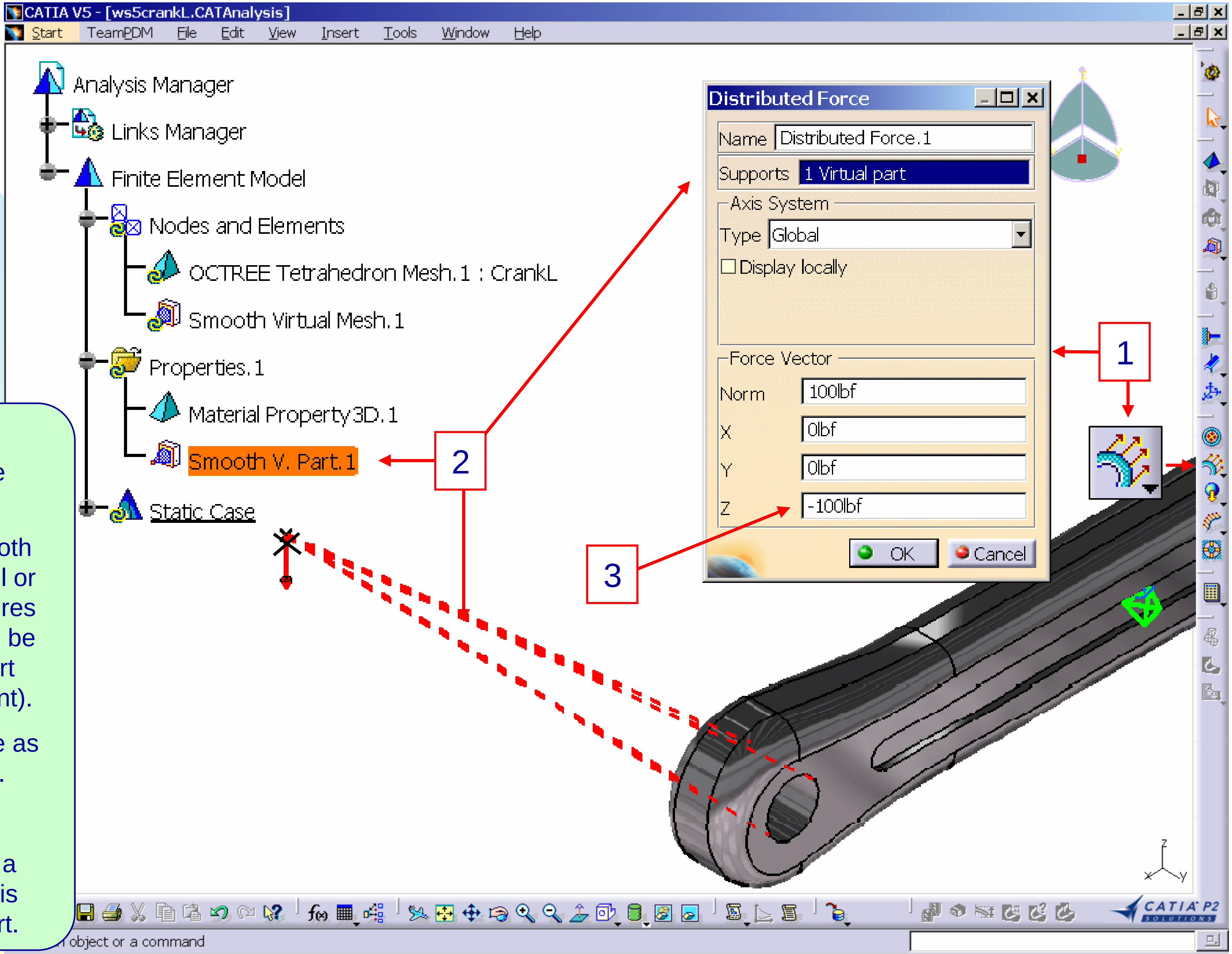
5. Select the Part Handler point previously created, select OK.

This smooth virtual part transmits load into the crank without adding stiffness.



MSC SOFTWARE

Step 7. Apply a force to the Smooth Virtual Part



Steps:

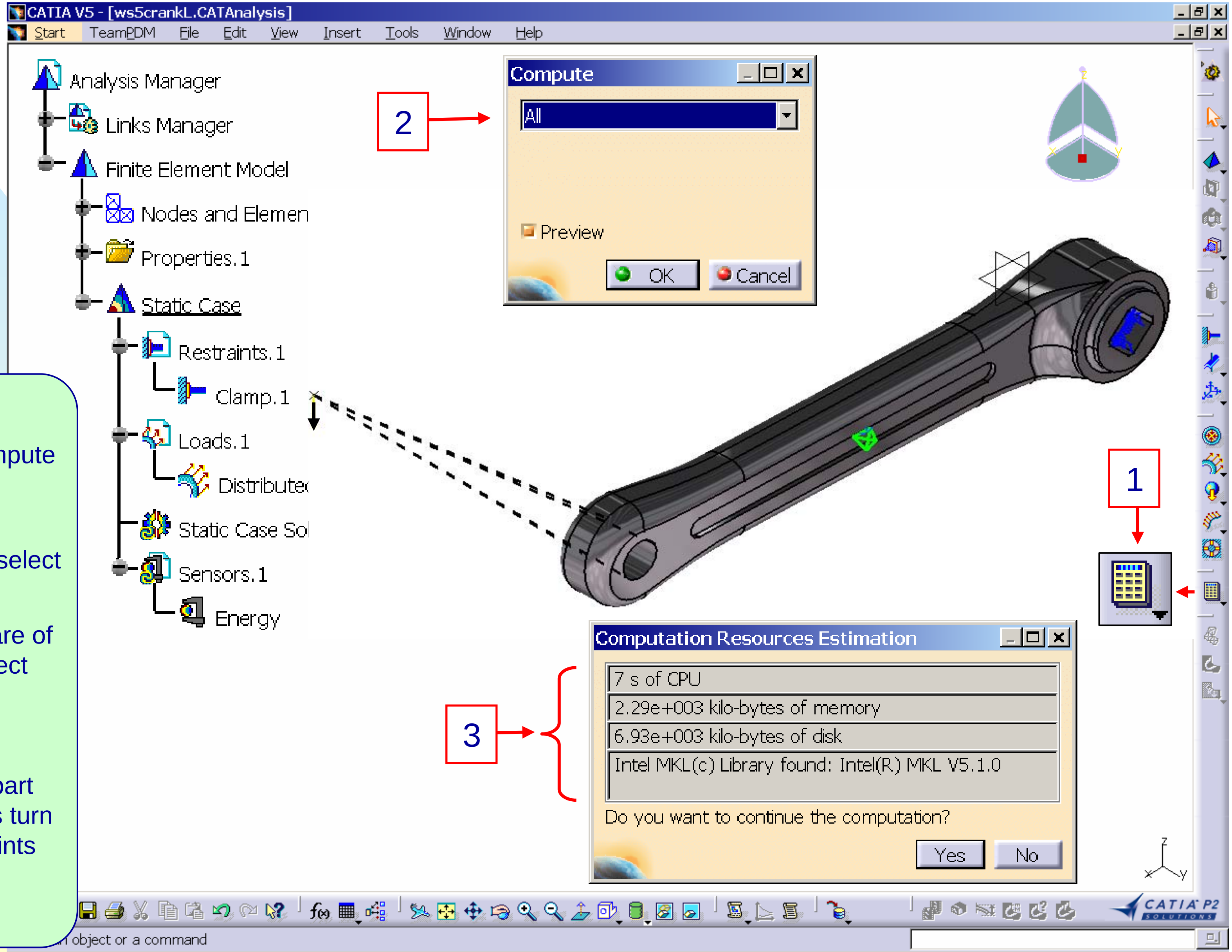
1. Select the force icon.

2. Select the smooth virtual part symbol or object in the features tree (the force will be applied at the “Part Handler” - the point).

3. Key in the force as shown, select OK.

The virtual part is a way to transmit this force into your part.

Step 8. Compute the initial analysis



Steps:

1. Select the Compute icon.

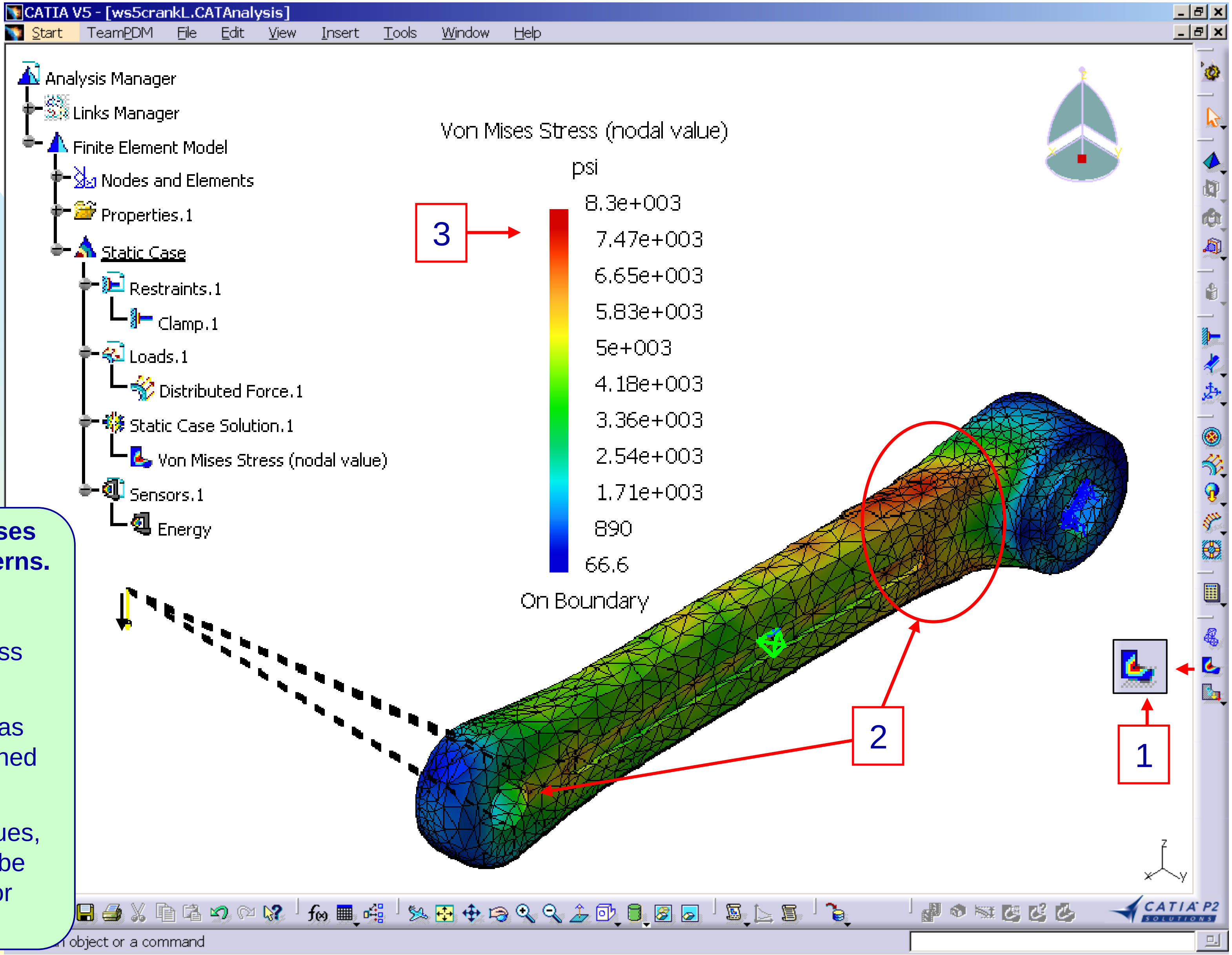
2. Compute All Objects defined, select OK.

3. Always be aware of these values, select Yes.

Note: the virtual part turns black, loads turn yellow and restraints turn blue.

Save often.

Step 9. Visualize "hot spots" in the initial analysis



Visualize **Von Mises stress field patterns.**

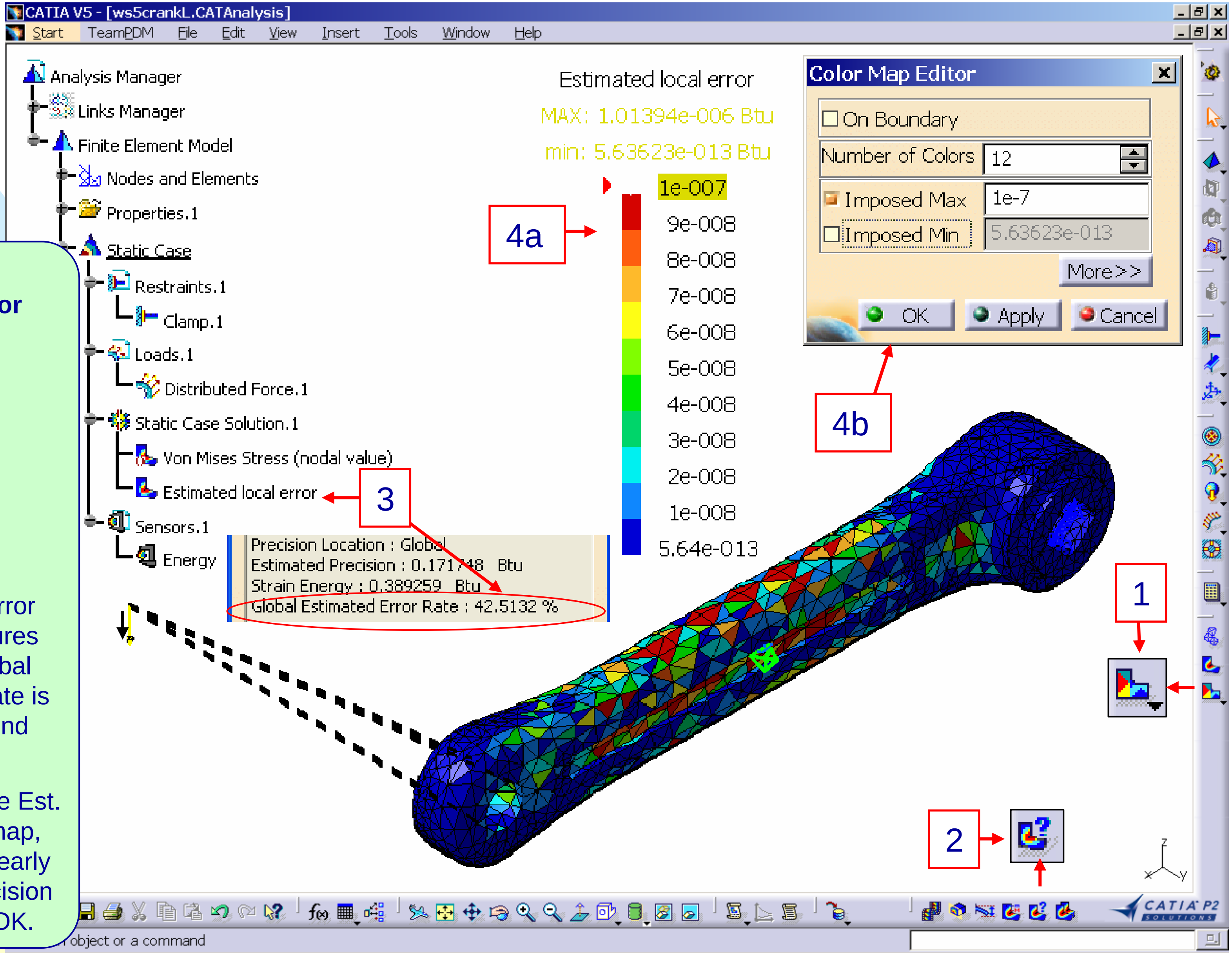
Steps:

1. Select the Stress Von Mises icon.

2. Note these areas requires local refined meshing.

3. Note these values, but they may not be precise enough for design.

Step 9. Visualize “hot spots” in the initial analysis



Visualize the **computation error map**

Steps:

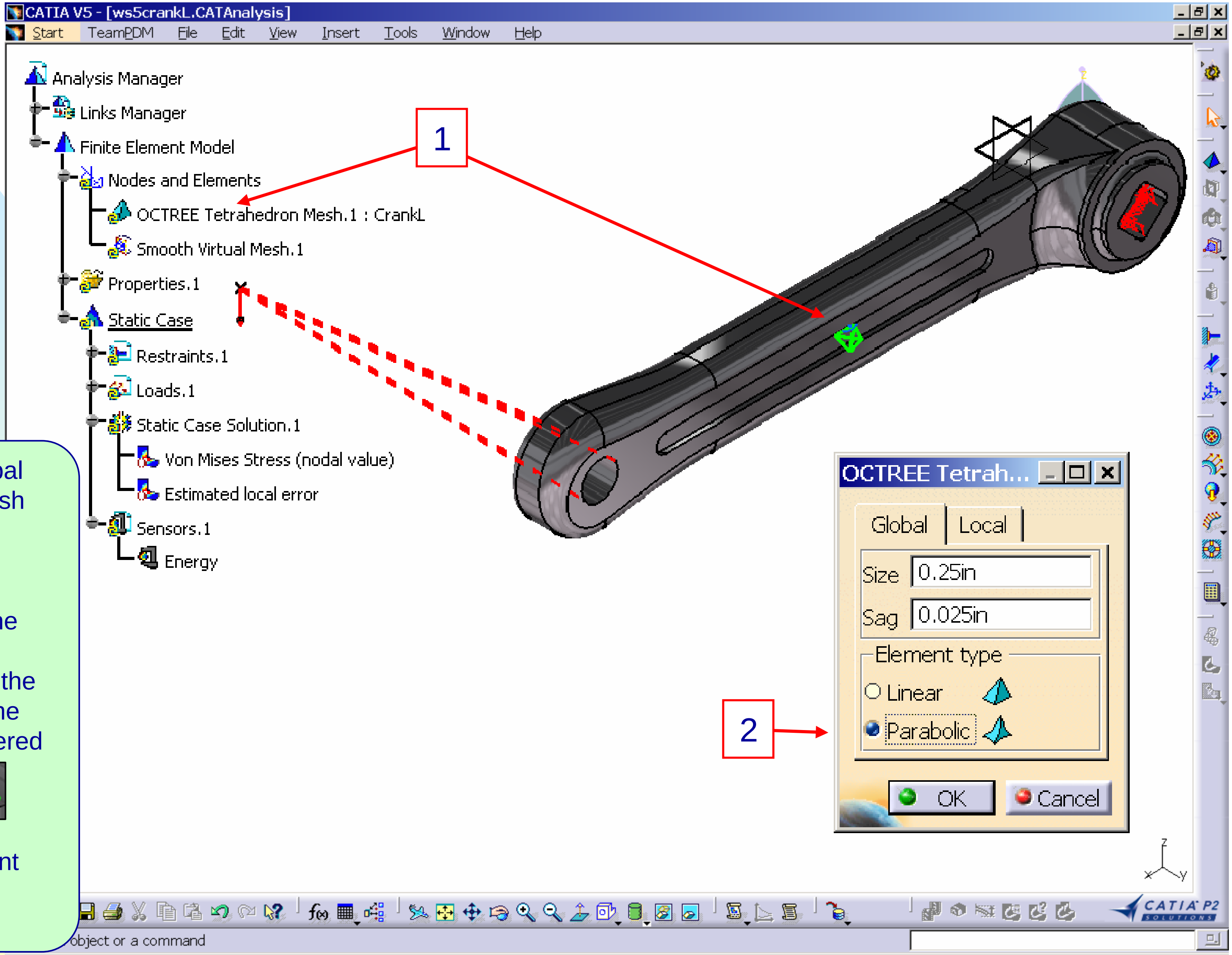
1. Select the Precision icon.

2. Select on the information icon.

3. Select the Estimated local error object in the features tree. Note the global estimated error rate is to high (recommend max 20%).

4. Double click the Est. local error color map, impose 1e-7 to clearly visualize low precision locations, select OK.

Step 10. Change mesh to parabolic and add local meshing



Redefine the global finite element mesh type.

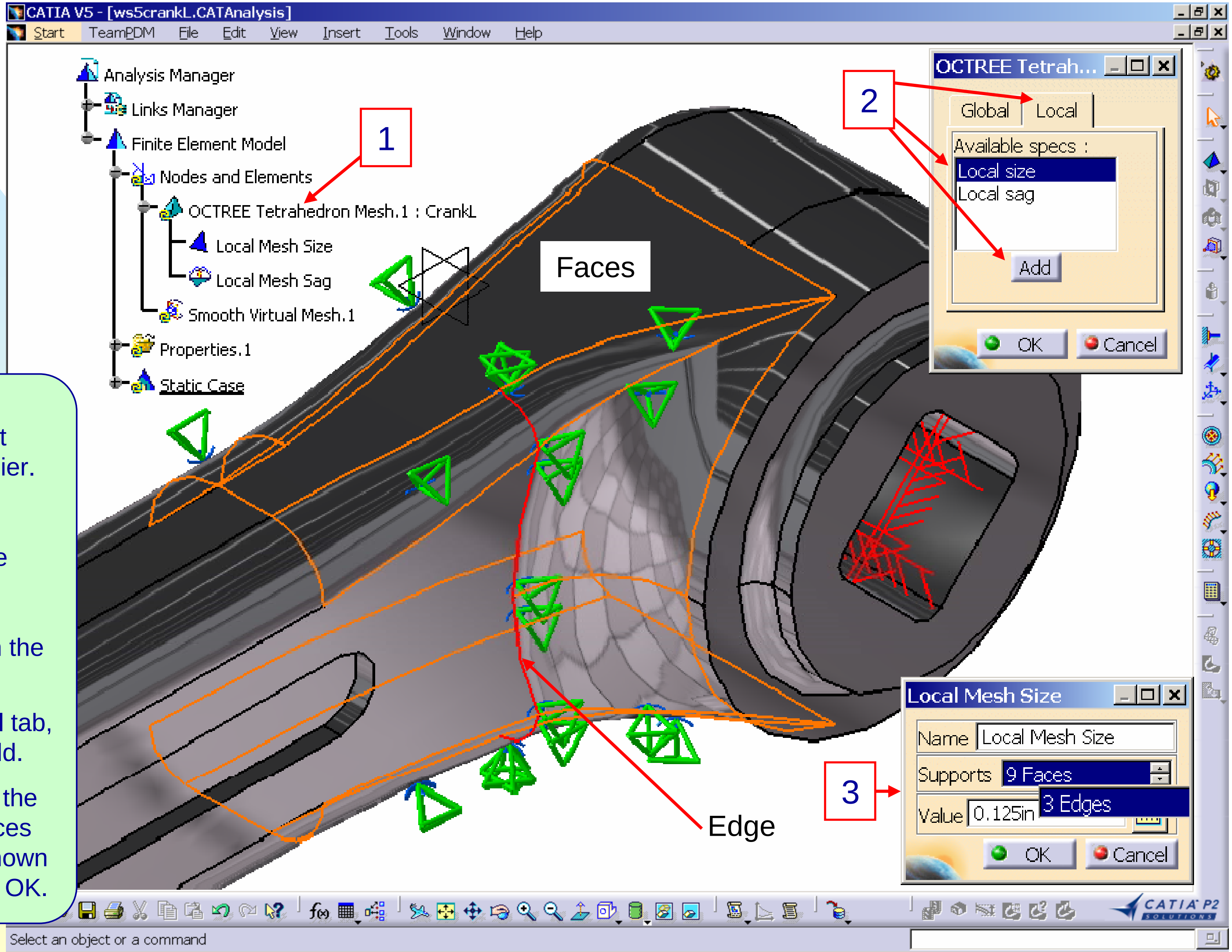
Steps:

1. Double Click the "OCTREE" representation in the features tree or the "Mesh" icon centered on the part.

2. Change element type to Parabolic, select OK.

MSC SOFTWARE SIMULATING REALITY

Step 10. Change mesh to parabolic and add local meshing



Locally refine the **mesh size** in a hot spot identified earlier.

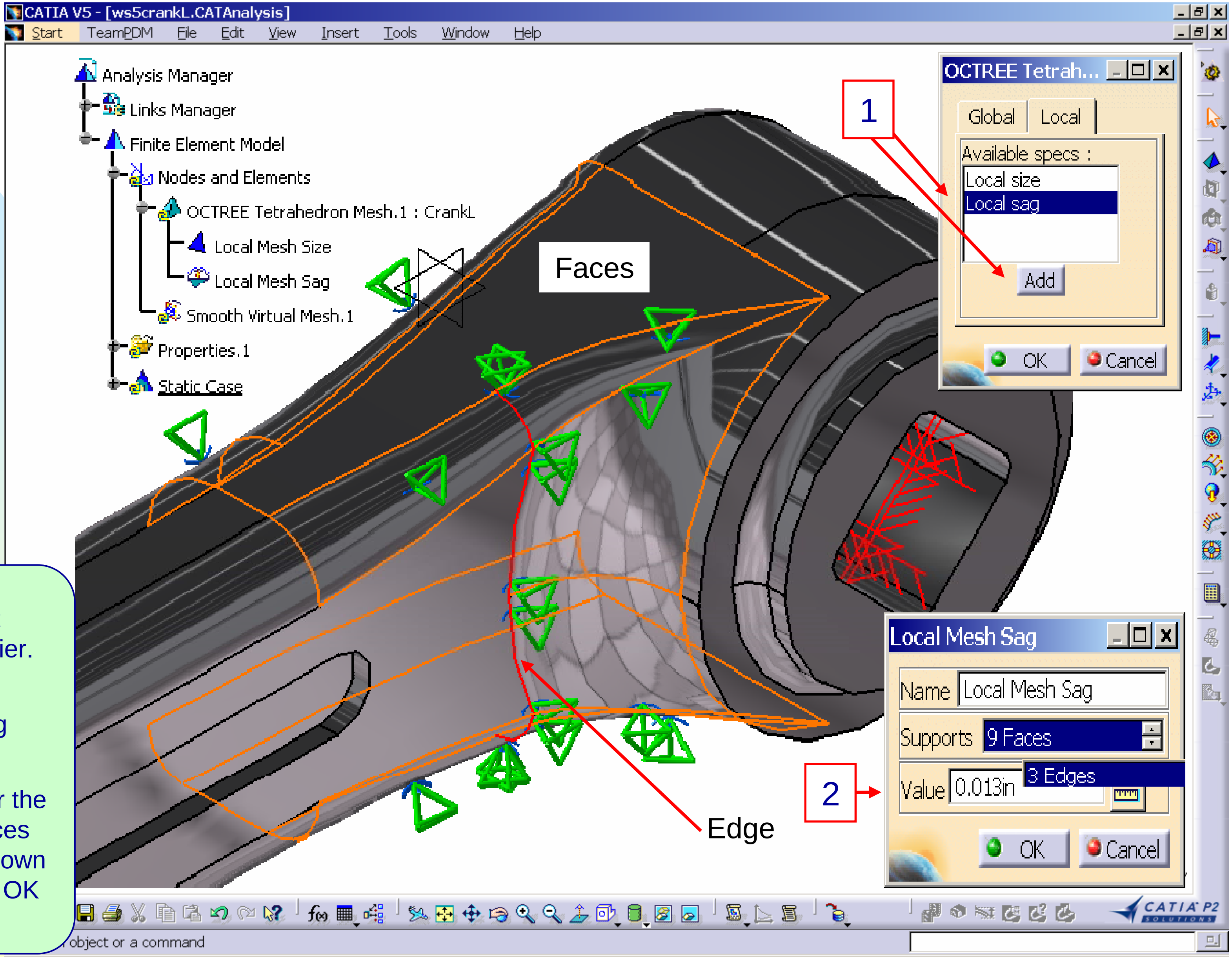
Steps:

1. Double Click the “OCTREE Tetrahedron Mesh.1:CrankL” in the features tree.

2. Select the Local tab, Local size then Add.

3. Key in .125” for the value, select 9 faces and 3 edges as shown highlighted, select OK.

Step 10. Change mesh to parabolic and add local meshing



Locally refine the **mesh sag** in a hot spot identified earlier.

Steps:

1. Select Local sag then Add.

2. Key in .013in for the value, select 9 faces and 3 edges as shown highlighted, select OK and OK.

MSC SOFTWARE

Step 10. Change mesh to parabolic and add local meshing

Locally refine the mesh size and sag in another hot spot identified earlier.

Steps:

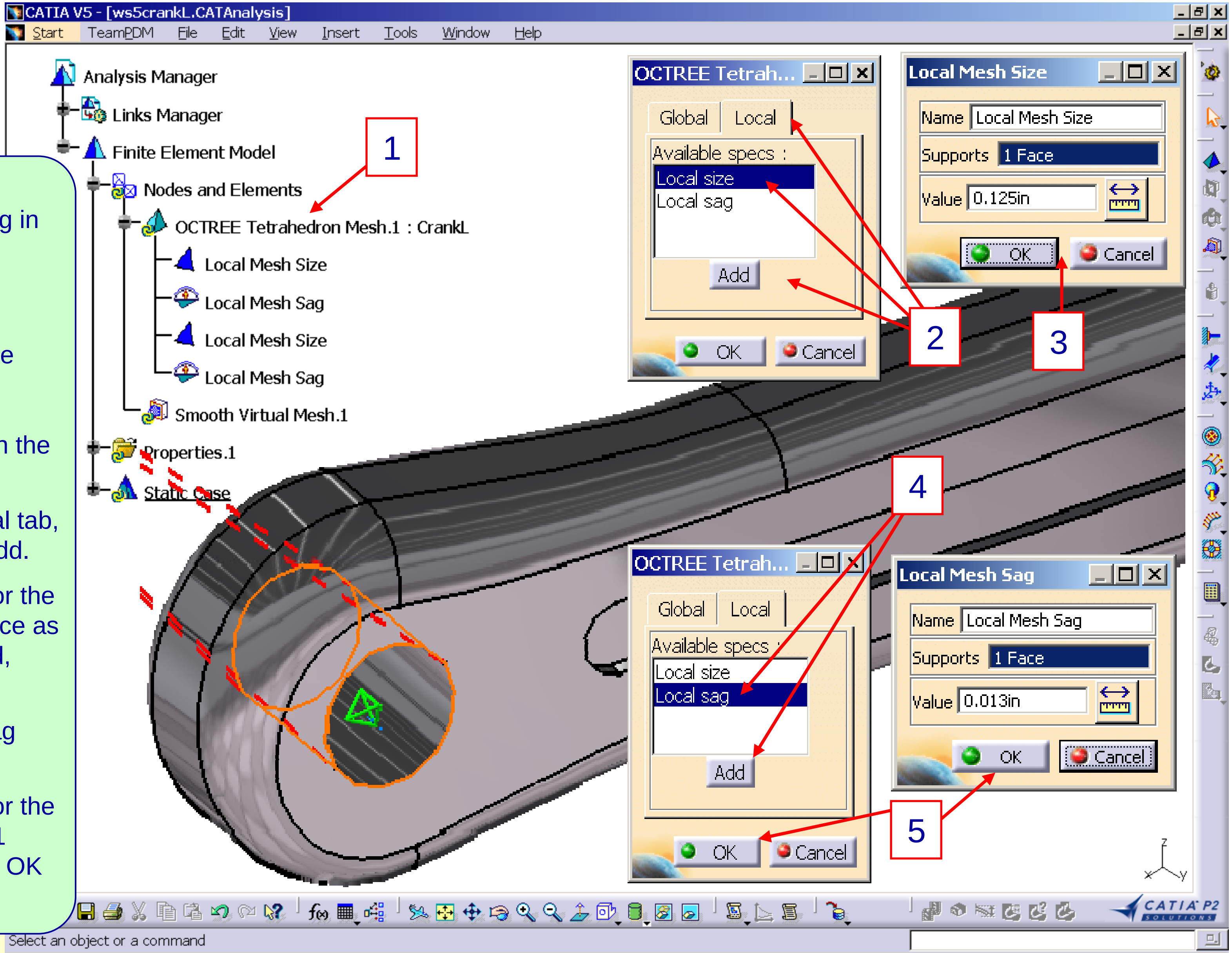
1. Double Click the "OCTREE Tetrahedron Mesh.1:CrankL" in the features tree.

2. Select the Local tab, Local size then Add.

3. Key in .125in for the value, select 1 face as shown highlighted, select OK.

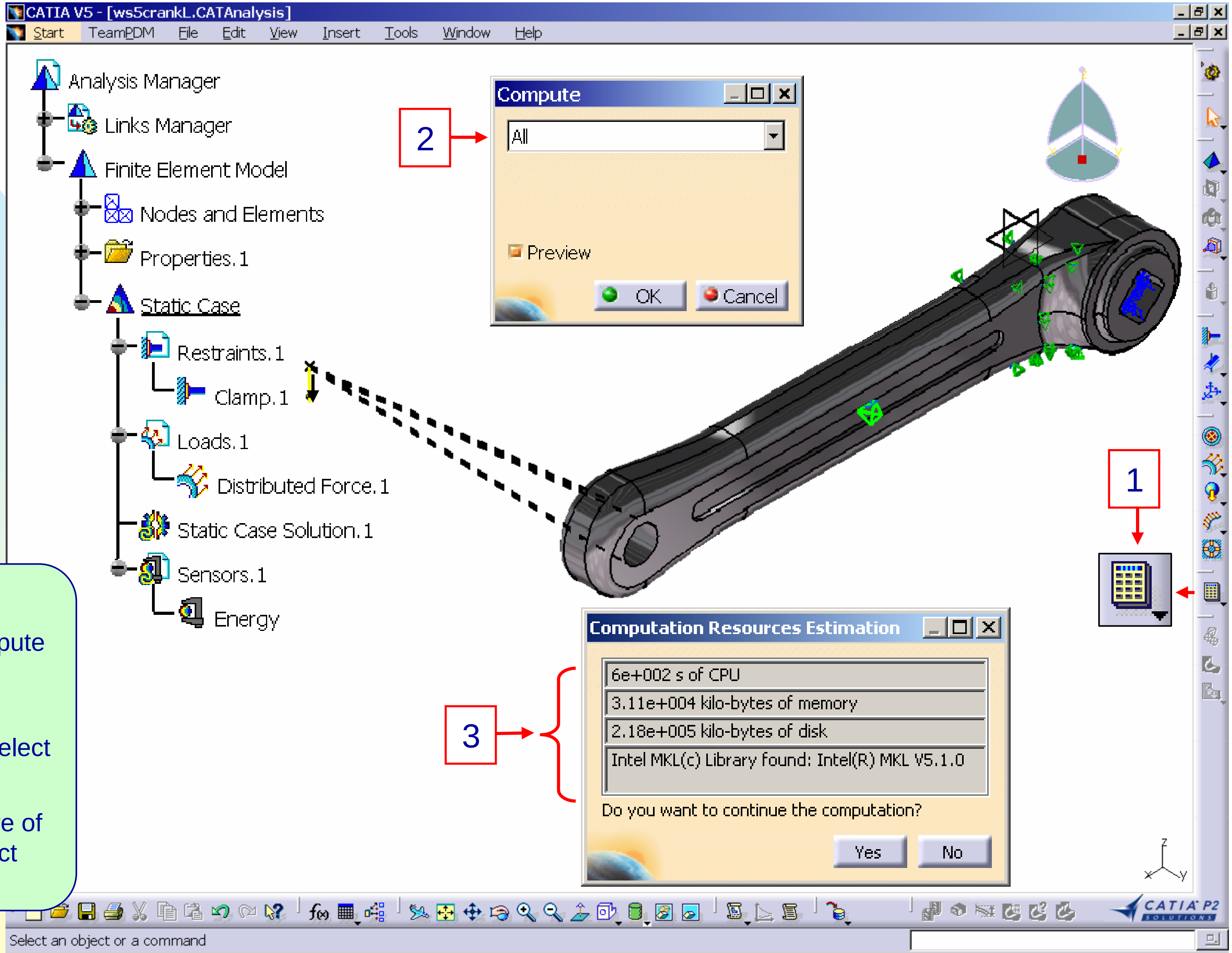
4. Select Local sag then Add.

5. Key in .013in for the value, select the 1 face again, select OK and OK.



MSC SOFTWARE

Step 11. Compute the more precise analysis



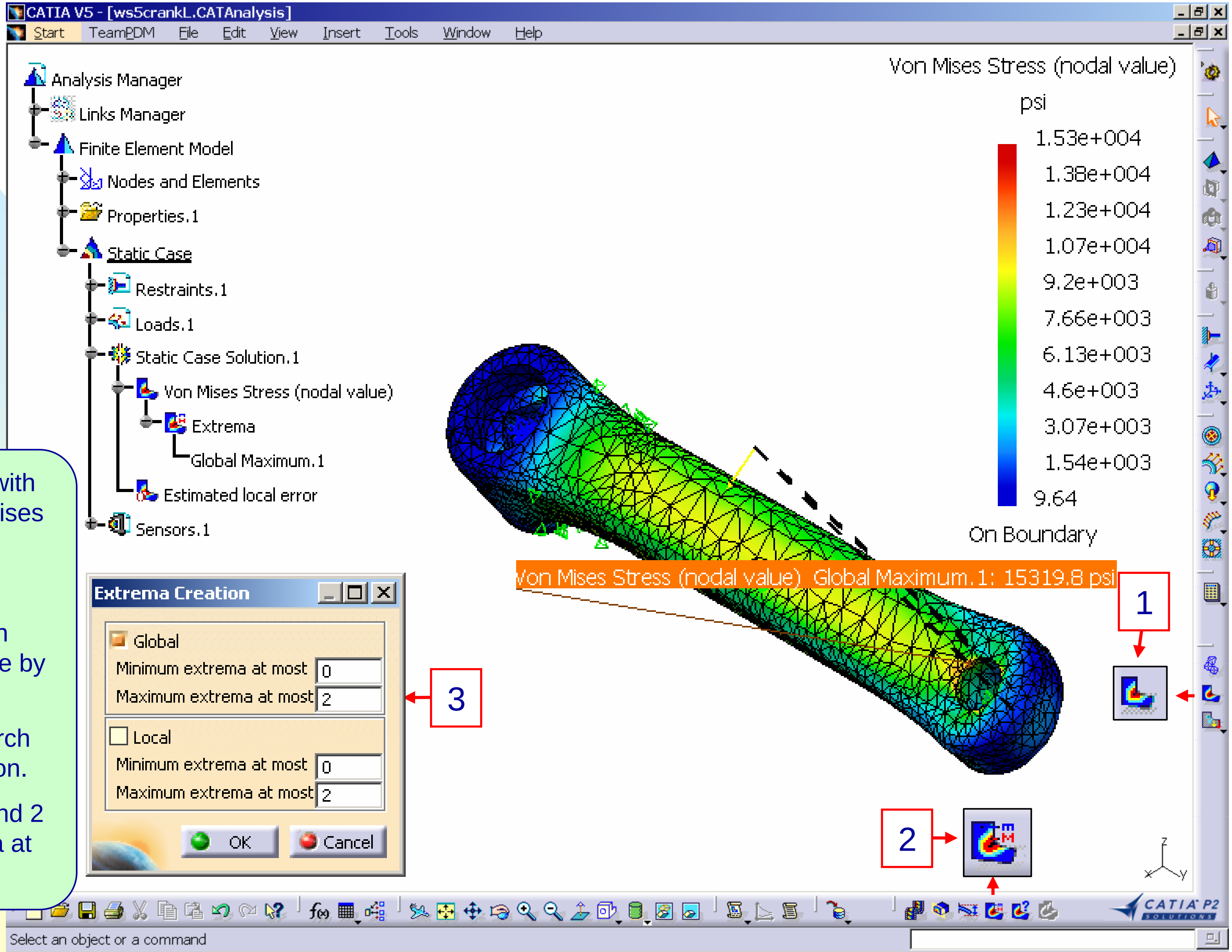
Steps:

1. Select the Compute icon.

2. Compute All Objects defined, select OK.

3. Always be aware of these values, select Yes.

Step 12. Visualize extremas



Find the element with the highest Von Mises stress.

Steps:

1. Activate the Von Mises stress image by selecting the icon.

2. Select the Search Image Extrema icon.

3. Select Global and 2 maximum extrema at most, select OK.

Step 12. Visualize extremas

Find the element with the highest Estimated error.

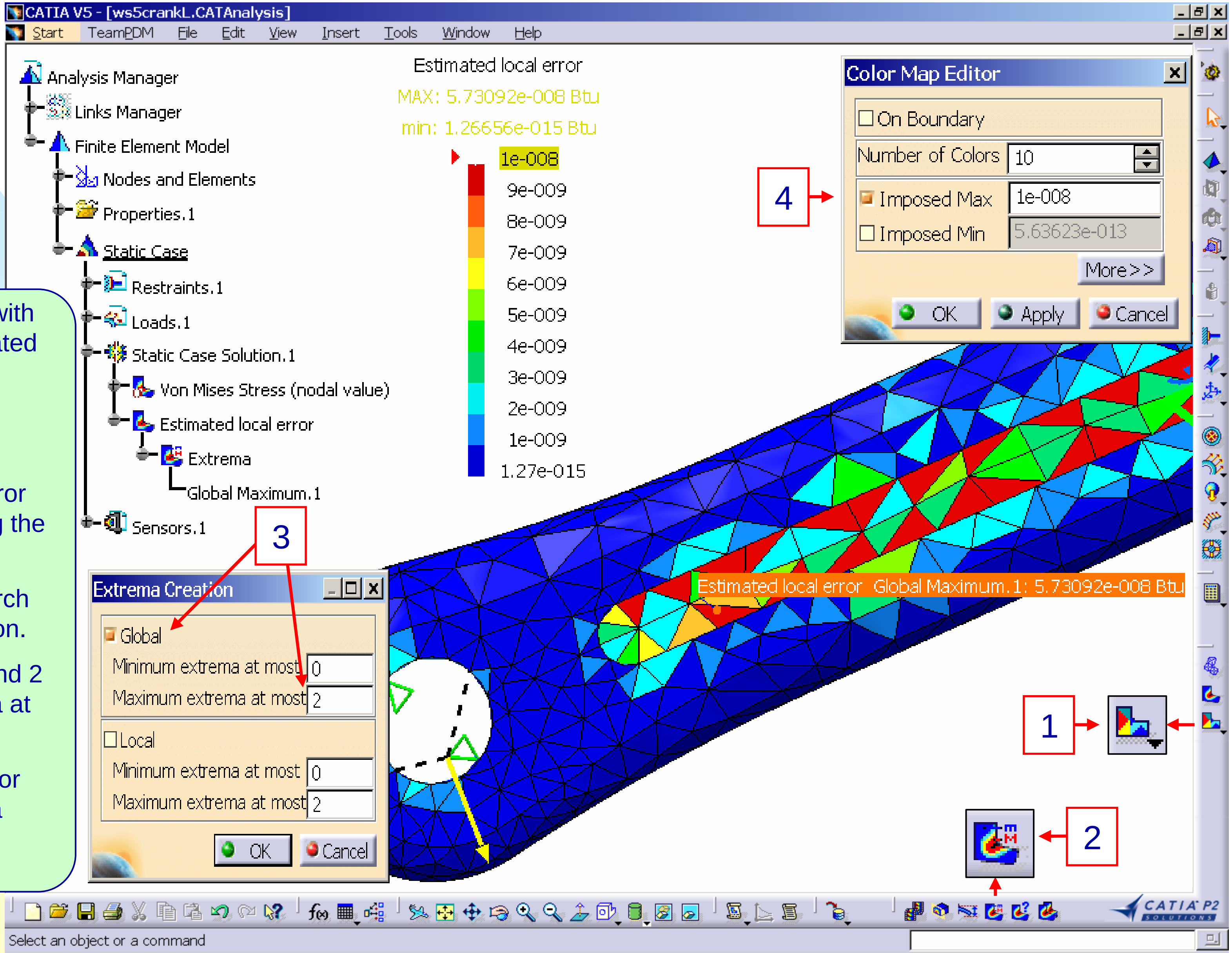
Steps:

1. Activate the Estimated local error image by selecting the Precision icon.

2. Select the Search Image Extrema icon.

3. Select Global and 2 maximum extrema at most, select OK.

4. Double click color map and impose a max 1e-008 (Btu value).



Step 13. Specify adaptivity boxes

Determine global and local error %.

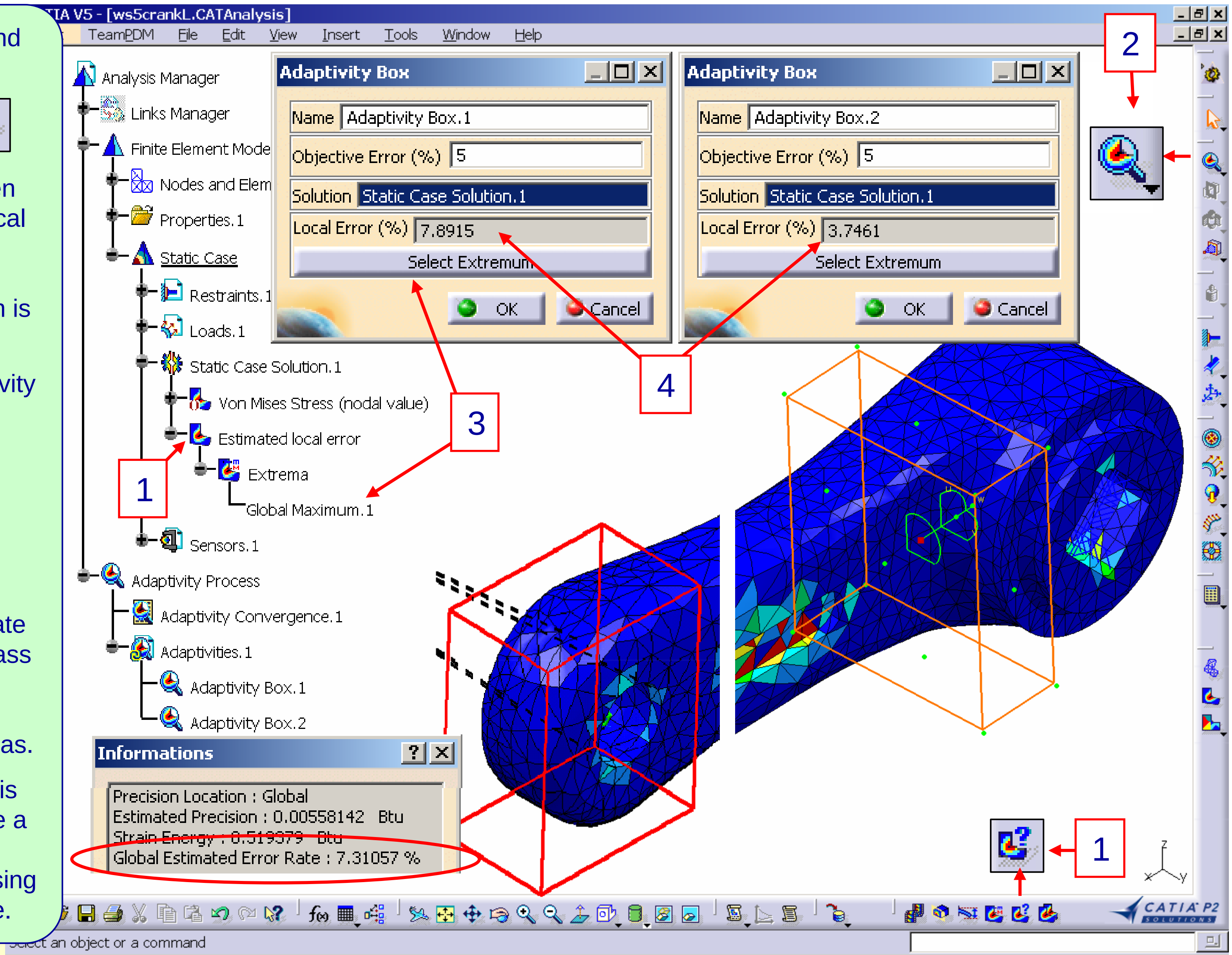
Steps:

1. Select the information icon then select Estimated local error object in the features tree to see that global precision is below 20%.

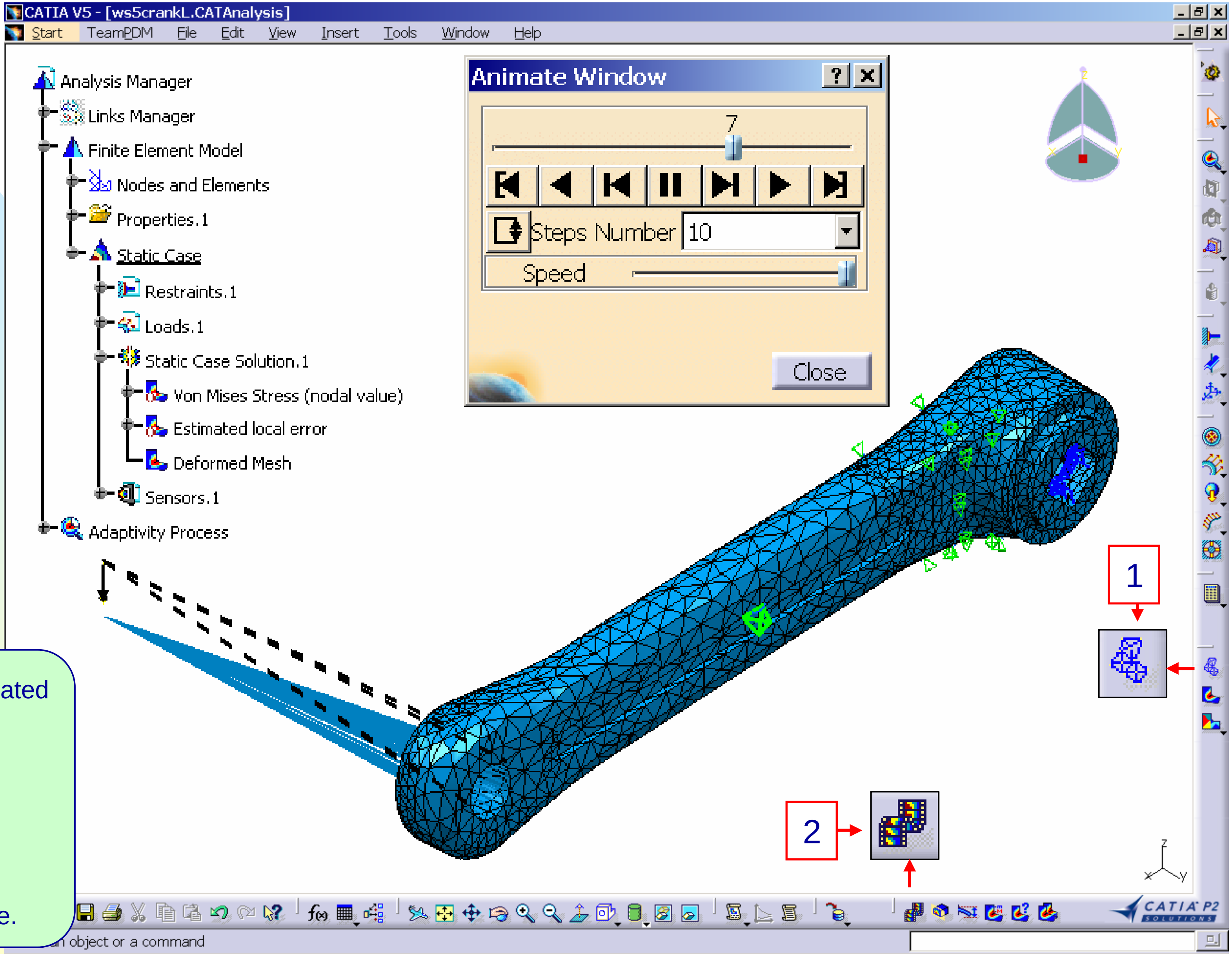
2. Select the adaptivity box icon.

3. First select the “Select Extremum” button then Global Maximum.1 in the features tree to locate box. Use the compass and green dots to locate and size box around meshed areas.

4. Since local error is below 10% we have a precise model. No need to compute using adapt and converge.



Step 14. Visualize final results



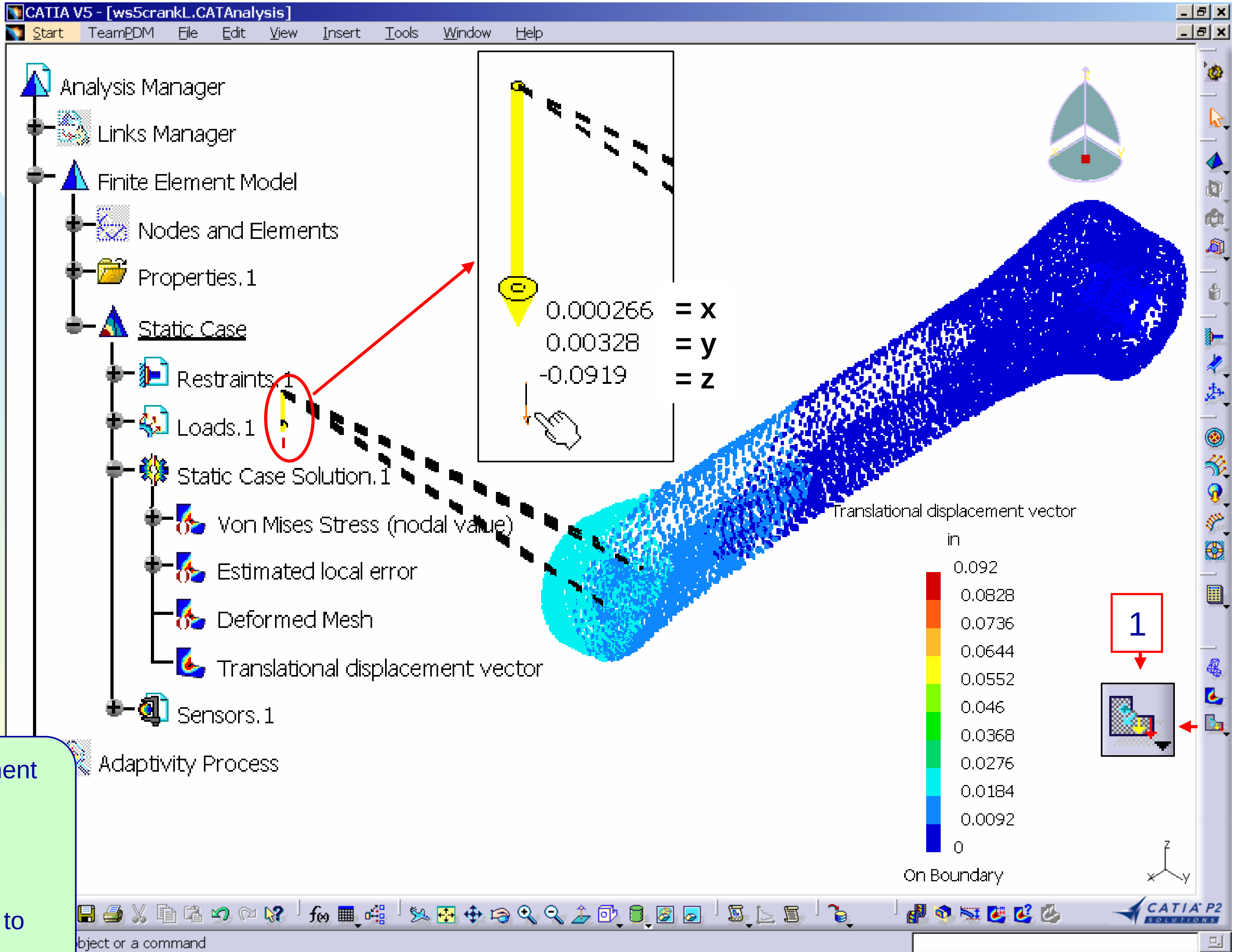
Visualize exaggerated **Deformation**.

Steps:

1. Select the Deformation icon.

2. Animate the deformation image.

Step 14. Visualize final results

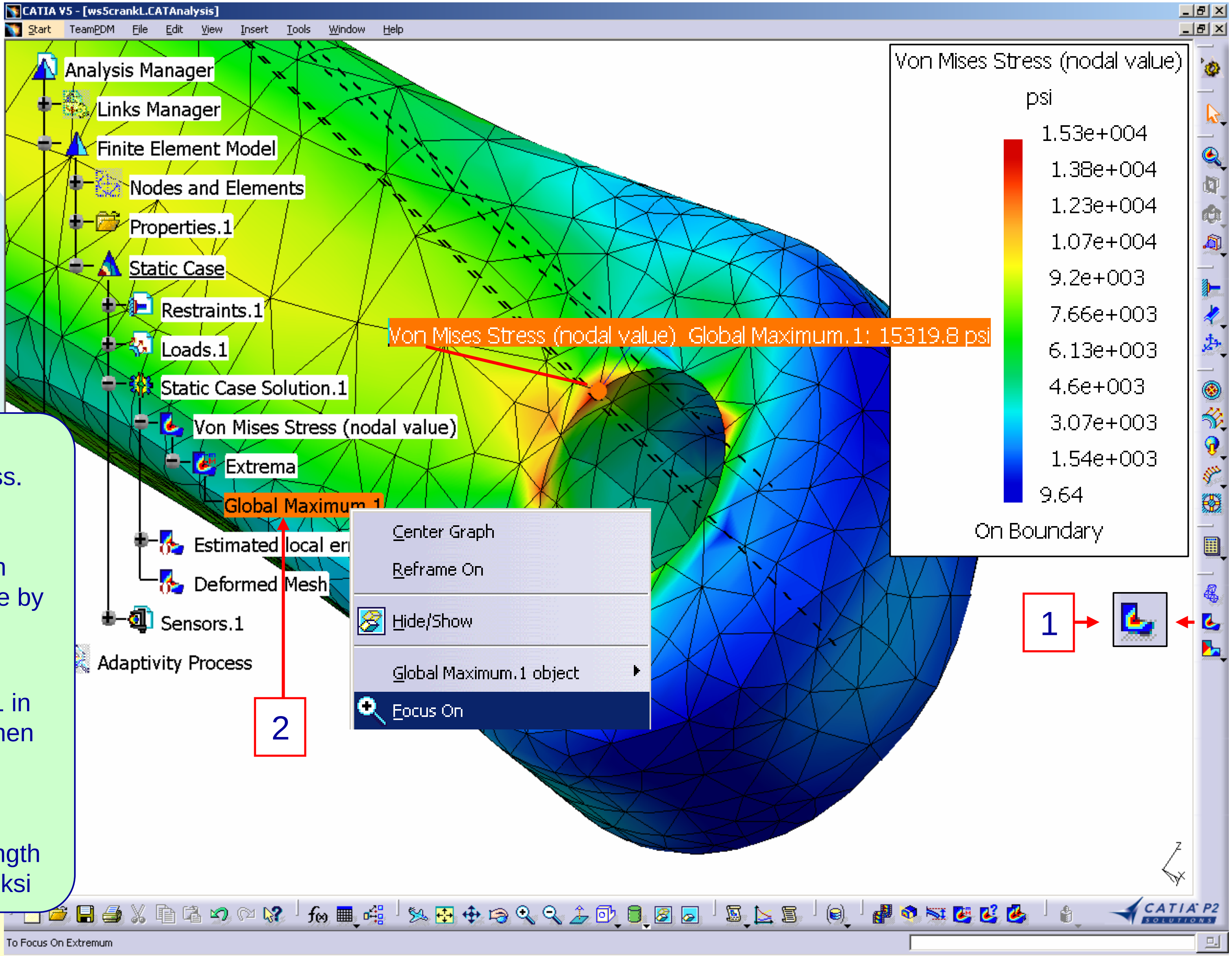


Add the displacement image

Steps:

1. Select the displacement icon to add this image.

Step 14. Visualize final results



Visualize the Von Mises design stress.

Steps:

1. Activate the Von Mises stress image by selecting the icon.

2. Right click on Global Maximum.1 in the features tree then select Focus on.

Material yield strength must exceed 15.3 ksi

Step 14. Visualize final results

- **Conclusions**
  - New material is required with a yield strength higher than 15.3 ksi.

Hand Calc's: 9.17 ksi Combined Stress	**.25" Linear Mesh, .025 sag**	**.25" Parabolic Global Mesh, .025" sag. .125" Parabolic Local Mesh, .013" sag. Adapt and converge not necessary.**
Max Von Mises	**8.30 ksi**	**15.3 ksi**
Translational Displacement	**? inch**	**-.0916" Z - direction at point of load**
Error Estimate	**1.01e-6 Btu**	**5.7e-8 Btu local**
Global % Precision error Local % Precision error	**42.5 % NA %**	**7.3 % 7.9 % and 3.7%**

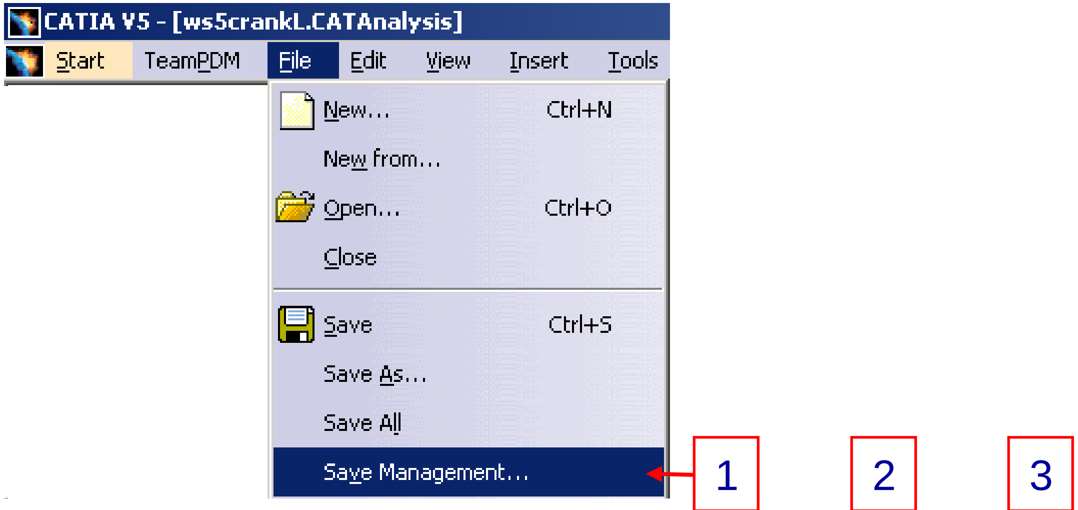
Step 15. Save the analysis document

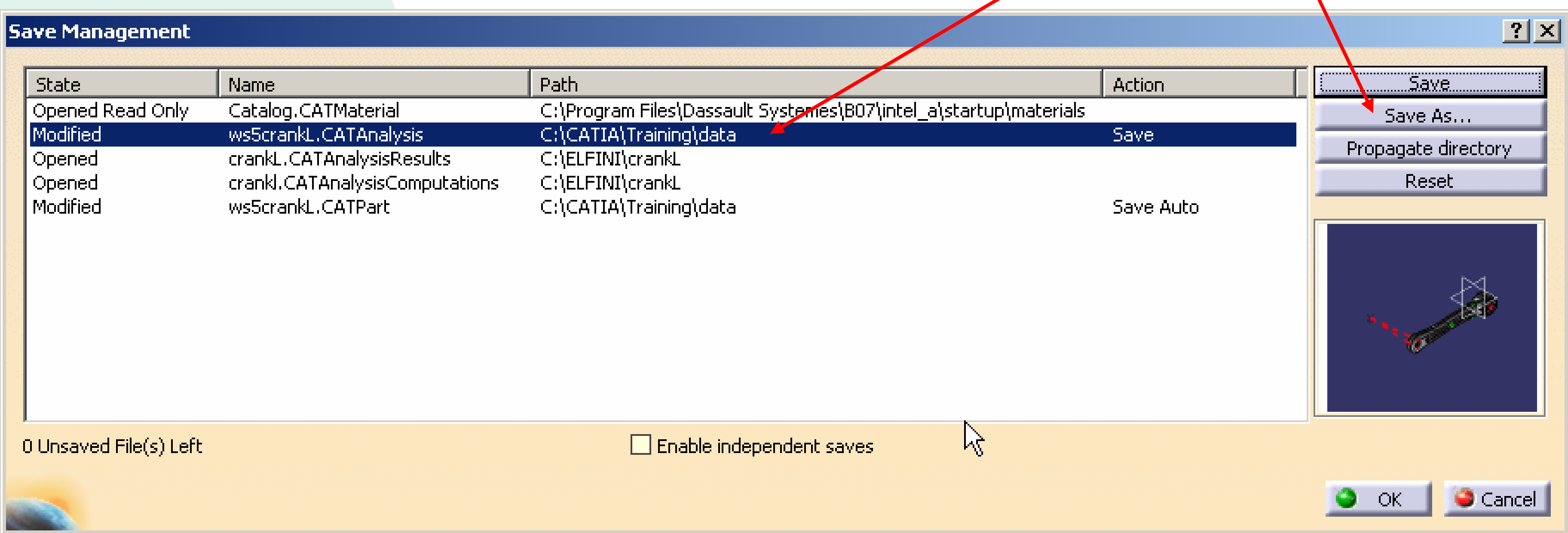
Steps:

1. Select Save Management from the File menu.

2. Highlight document you want to save.

3. Select Save As to specify name and path, select, OK





WORKSHOP 6

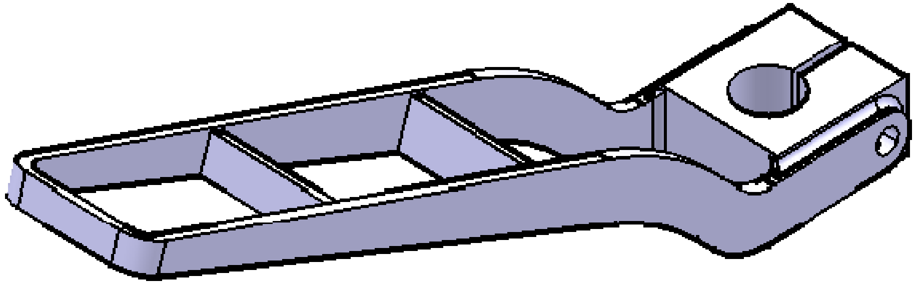
REAR RACK (MODAL) ANALYSIS

Problem Description

- Assume the dynamic characteristics of this bike with a 200 lb person traveling at 40 mph down a cobble stone road is: Mode 1=95 Hz, Mode 2 = 100 Hz, Mode 3 = 110 Hz, Mode 4 = 120 Hz, Mode 5 = 135 Hz.
- A rear rack accessory capable of supporting 150 lbs may be attached to the frame. You are asked to analyze this rack under dynamic loading.
- Perform a normal modes analysis to determine if the frequency of the bike is close to one of the natural frequencies of the rack. This is to avoid excessive vibrations and find “soft spots” (smooth, comfortable ride).

Aluminum

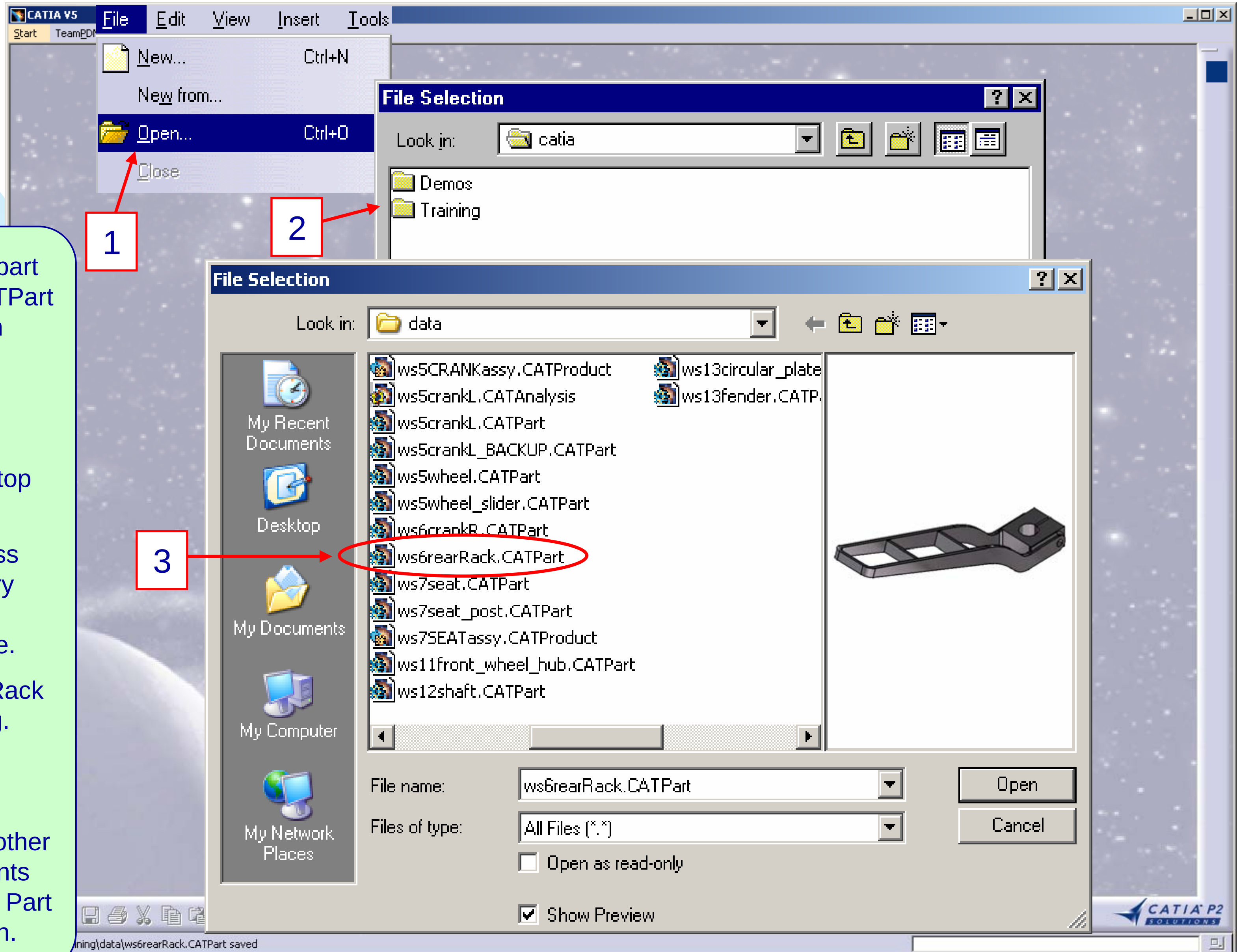
Elastic Modulus, E	10.15E6 psi
Poisson’s Ratio, ν	0.346
Density	.098 lb/in^3
Yield Strength	13,778 psi



Suggested Exercise Steps

1. Open the existing CATIA part in the Part Design workbench.
2. Apply aluminum material properties to the part.
3. Create a Frequency analysis document (.CATAnalysis).
4. Pre-process initial finite element mesh.
5. Apply a clamp restraint.
6. Apply a mass equipment load.
7. Compute the analysis.
8. Visualize the analysis results.
9. Generate a report of the results.
10. Save the analysis document.

Step 1. Open the existing CATIA part



Open the CATIA part ws6rearRack.CATPart in the Part Design workbench.

Steps:

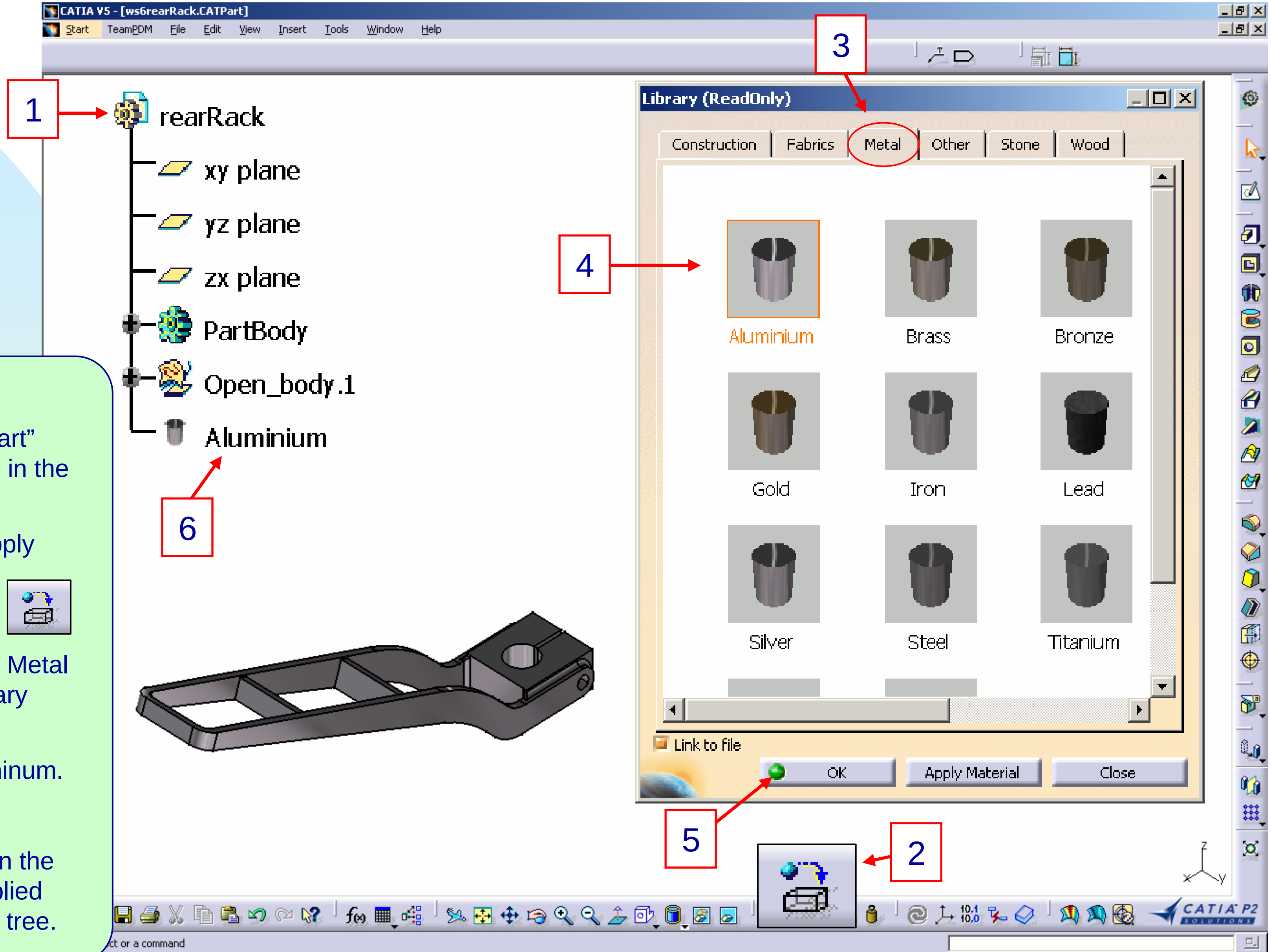
1. Select File and Open… from the top pull-down menu.

2. Access the class workshop directory using the typical Windows interface.

3. Open the rearRack by double-clicking.

By default, the rearRack and all other CATPart documents are opened in the Part Design workbench.

Step 2. Apply aluminum material properties to the part



Steps:

1. Click the "Part" representation in the features tree.

2. Click the Apply Material icon.

3. Activate the Metal tab in the Library window.

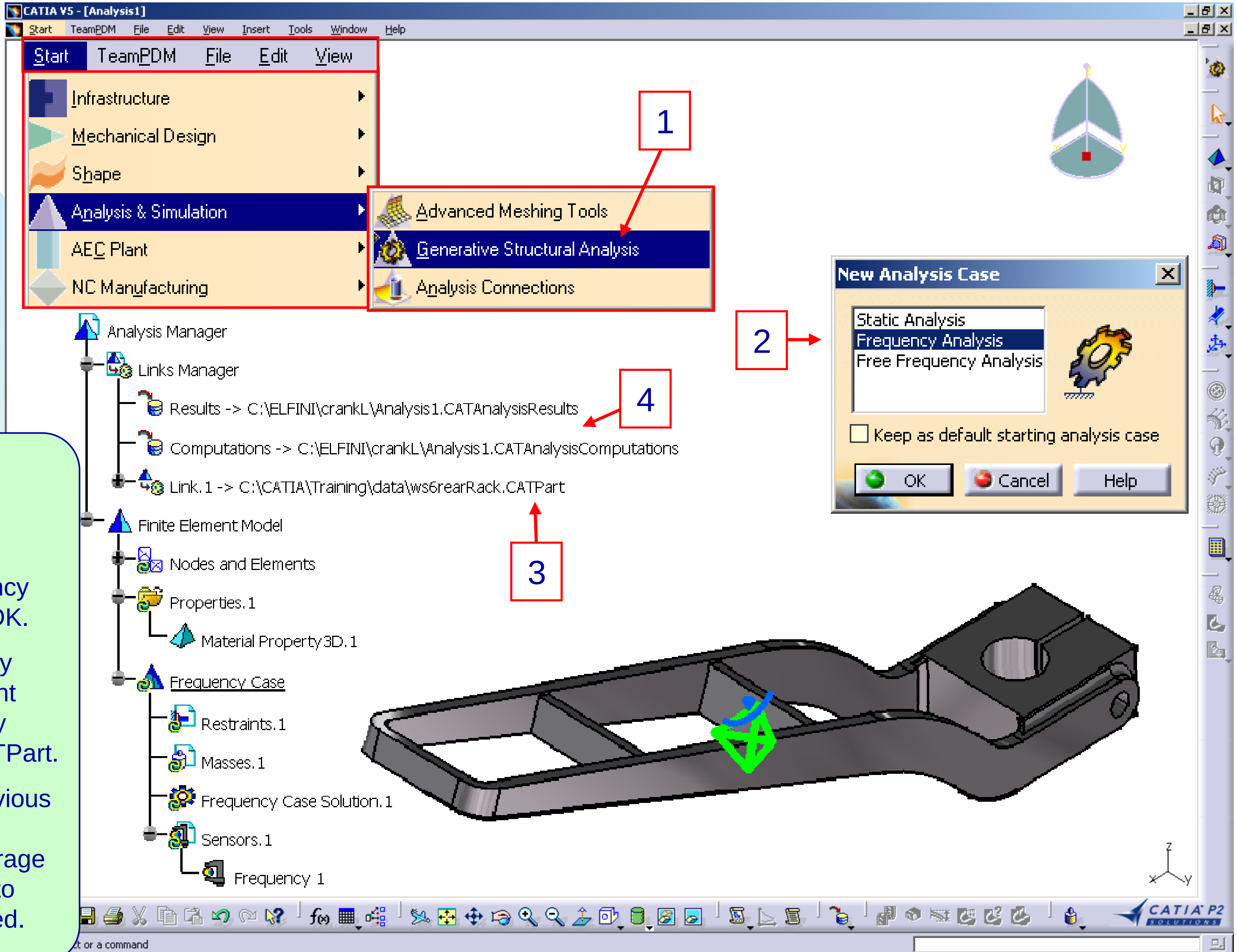
4. Select Aluminum.

5. Select OK.

6. Make certain the material is applied properly in the tree.

MSC SOFTWARE

Step 3. Create a Frequency analysis document



Steps:

1. Start a GSA workbench.

2. Select Frequency Analysis, select OK.

3. Your Frequency Analysis document gets automatically **linked** to the CATPart.

4. Note: your previous results and computations storage location defaults to your last path used.

Step 3. Create a Frequency analysis document

Specify unique External Storage directory locations.

Steps:

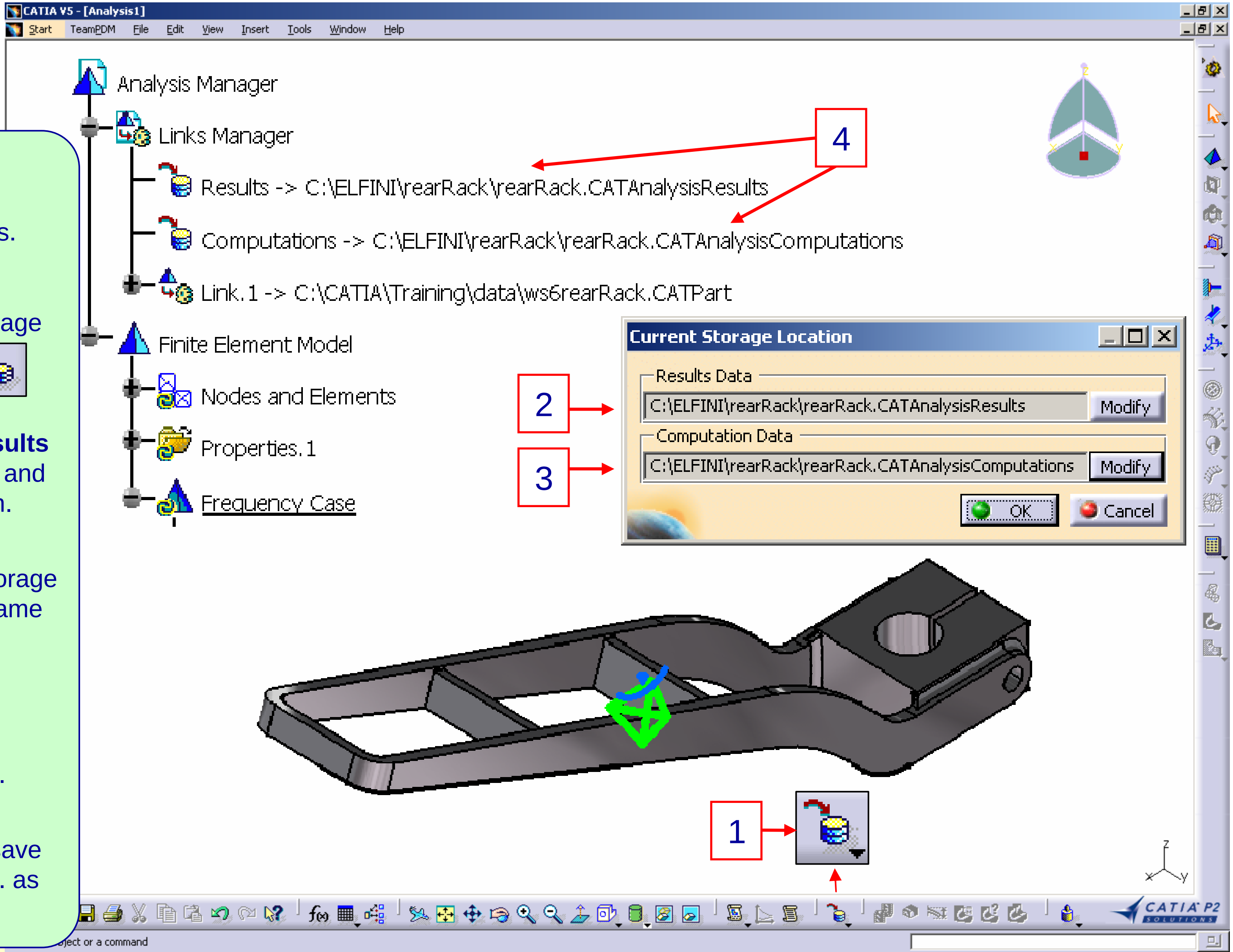
1. Select the Storage Location icon.

2. Modify the **Results** Storage Location and rename as shown.

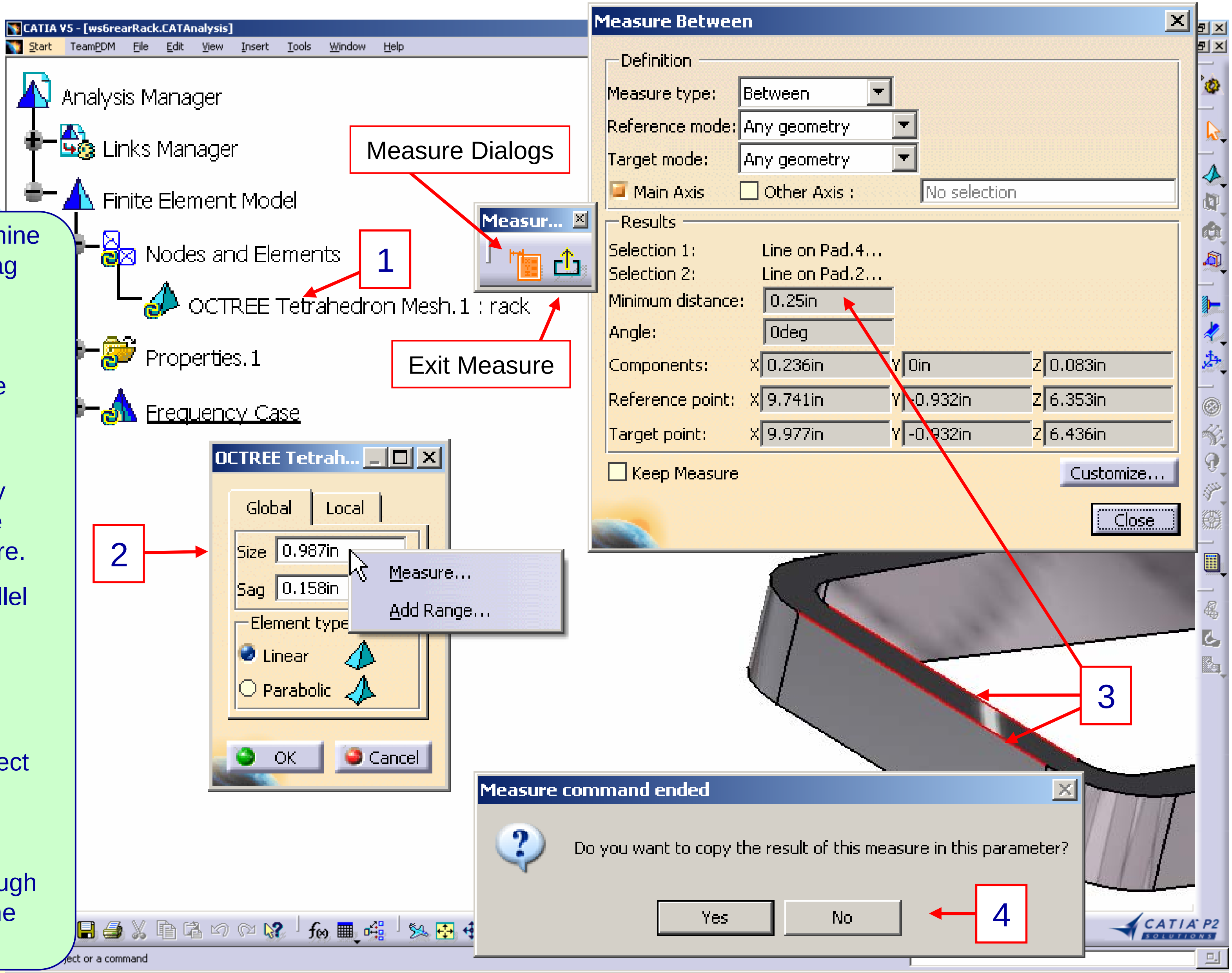
3. Modify the **Computation** Storage Location and rename as shown.

4. Note the Links Manager in the specification tree reflects the paths.

5. Use Save Management to save CATAnalysis doc. as “rearRack”.



Step 4. Pre-process initial finite element mesh



Measure to determine initial mesh and sag size.

Steps:

1. Double Click the “OCTREE” in the features tree.

2. Measure part by right clicking in the Size box + measure.

3. Select two parallel lines, note the distance = 0.25in, select Close.

4. Note the measurement, select NO.

Recommended rough Global Size = ½ the thinnest section.

Step 4. Pre-process initial finite element mesh

Define the global finite element mesh properties.

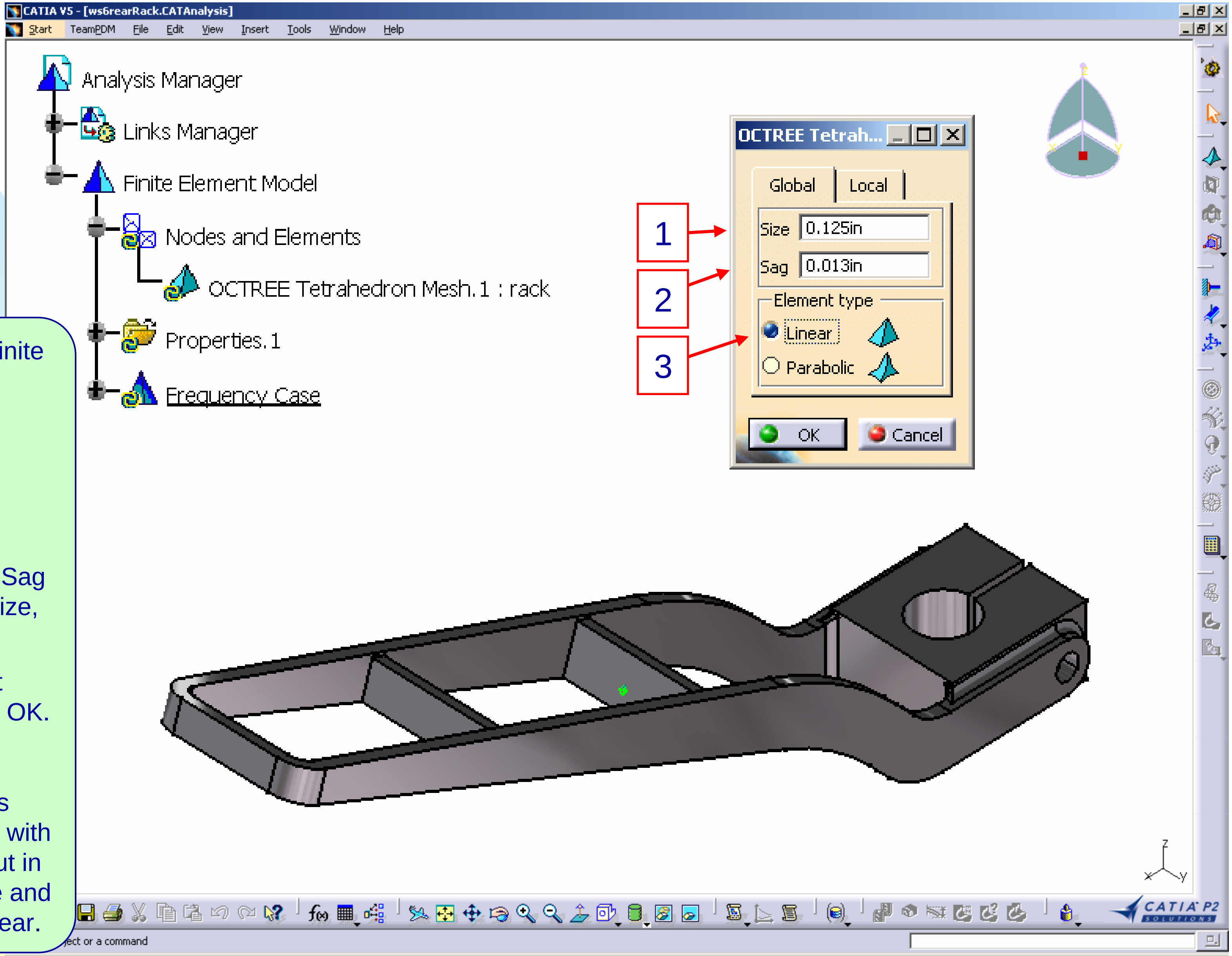
Steps:

1. Key in 0.125in global mesh size.

2. Recommended Sag = 10% of Global Size, key in 0.013in.

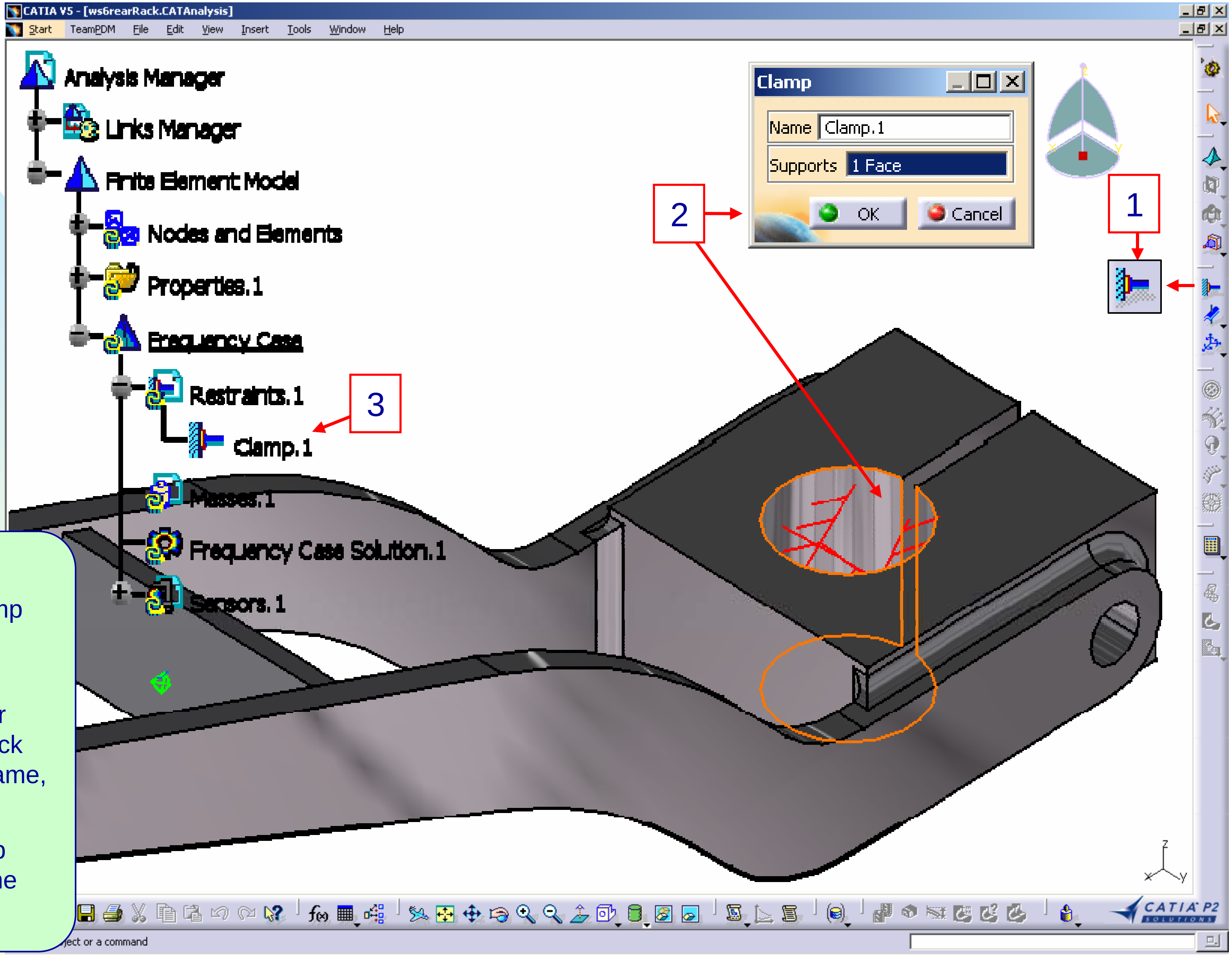
3. Specify element type Linear, select OK.

Parabolic elements yield better results with fewer elements, but in the interest of time and cpu space use Linear.



MSC SOFTWARE

Step 5. Apply a clamp restraint



Steps:

1. Select the Clamp Restraint icon.

2. Select the inner face where the rack attaches to the frame, select OK.

3. Note the Clamp object added to the specification tree.

MSC SOFTWARE

Step 6. Apply a mass equipment load

Steps:

1. Select the Mass icon.

2. Select the 2 faces as shown.

3. Enter 150 lbs as the mass, select OK.

4. Note the Distributed Mass object added to the specification tree.

English Mass Units:

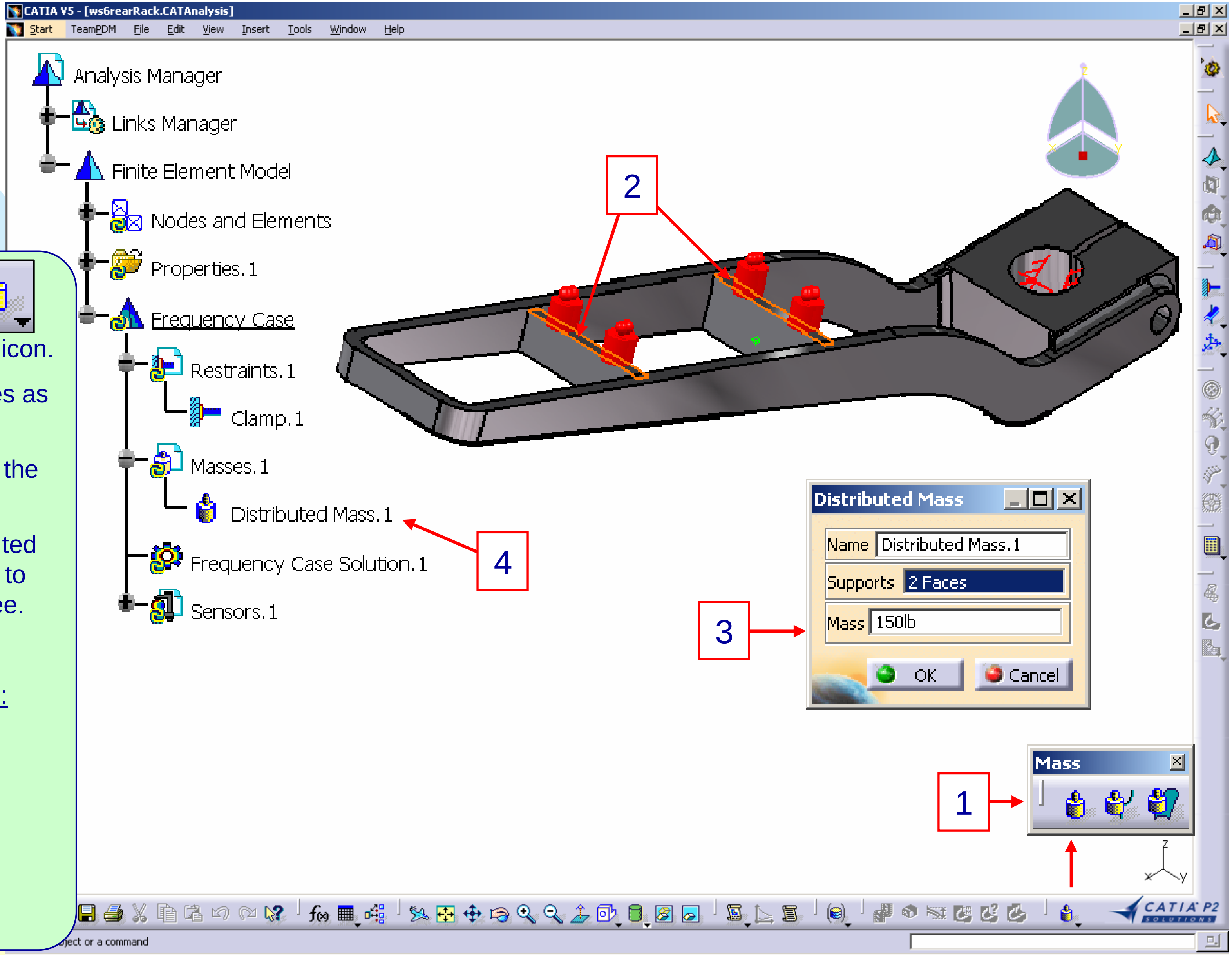
1g=386.1 in/sec2

Length=in

Time=sec

Density=lb/in3

Mass=lb



MSC SOFTWARE

Step 7. Compute the analysis

Specify the number of vibration modes to compute

Steps:

1. Double click on the Frequency Case Solution in the spec. tree.

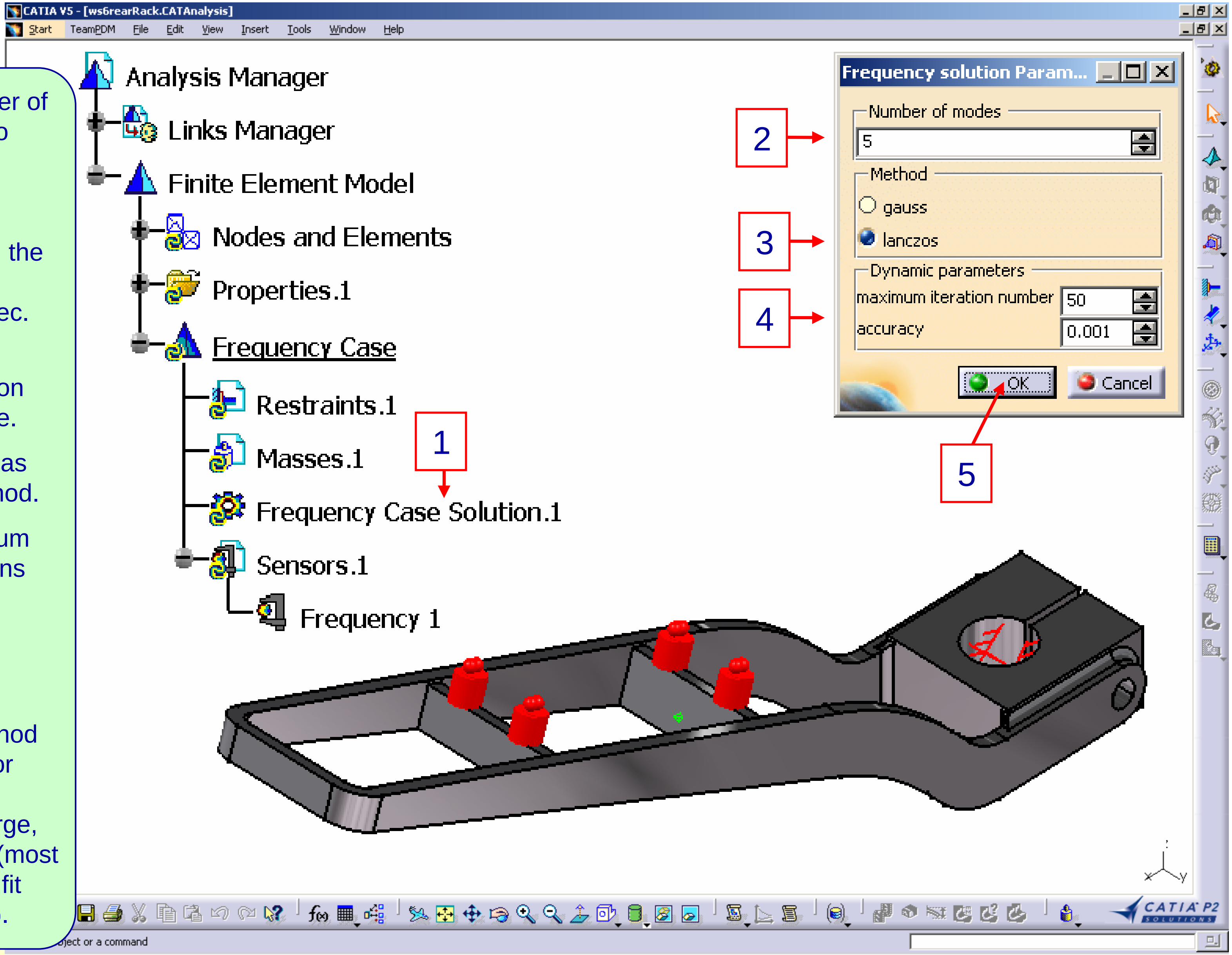
2. Key in 5 vibration modes to compute.

3. Select lanczos as the compute method.

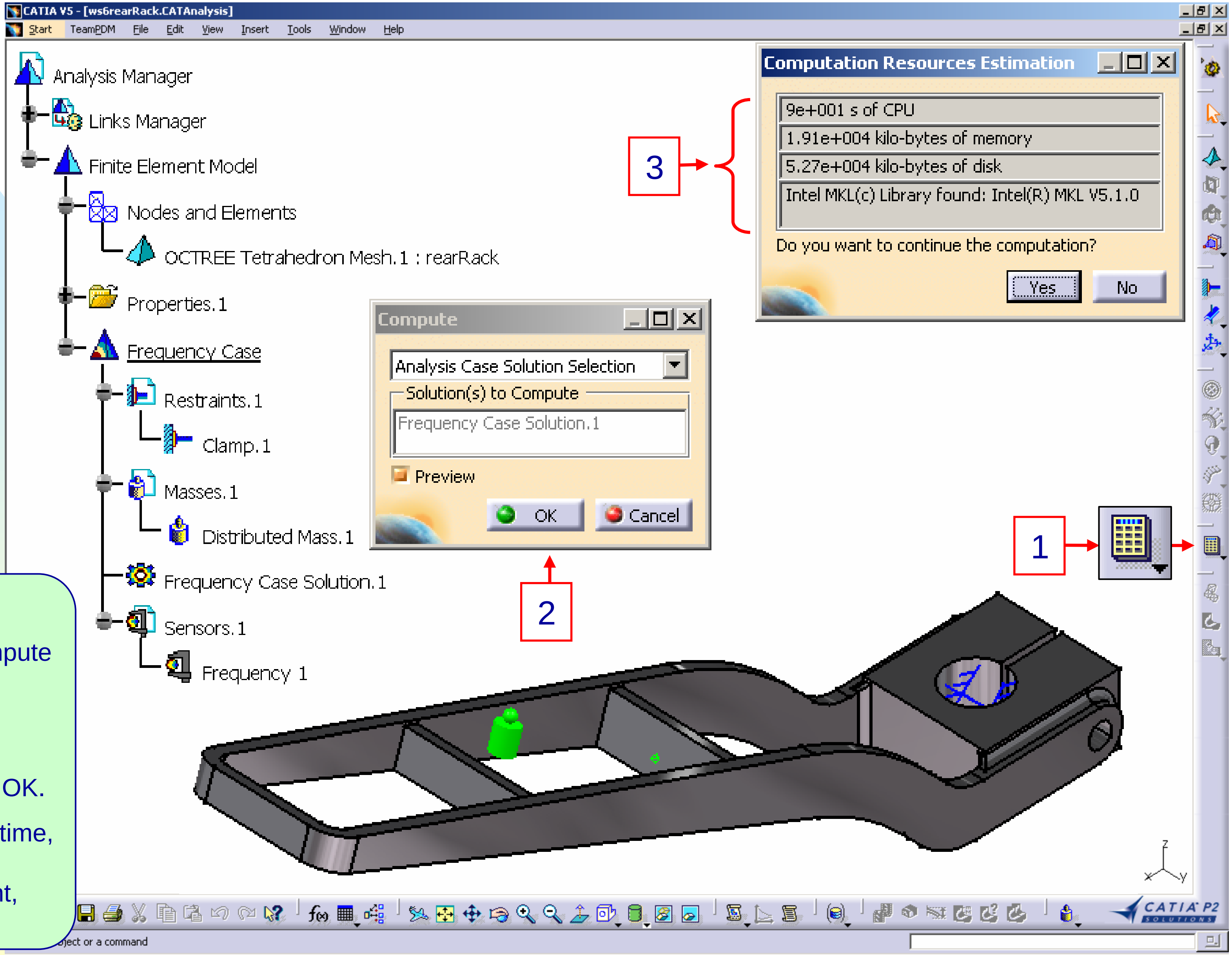
4. Specify maximum number of iterations and accuracy.

5. Select OK.

The Lanczos method is most efficient for computing a few Eigenvalues of large, sparce problems (most structural models fit into this category).



Step 7. Compute the analysis



Steps:

1. Select the Compute icon.

2. Compute the Frequency Case Solution.1, select OK.

3. Notice the est. time, memory and disk space requirement, select Yes.

Step 8. Visualize the analysis results

Visualize the maximum displacements to locate the areas of max strain energy.

Steps:

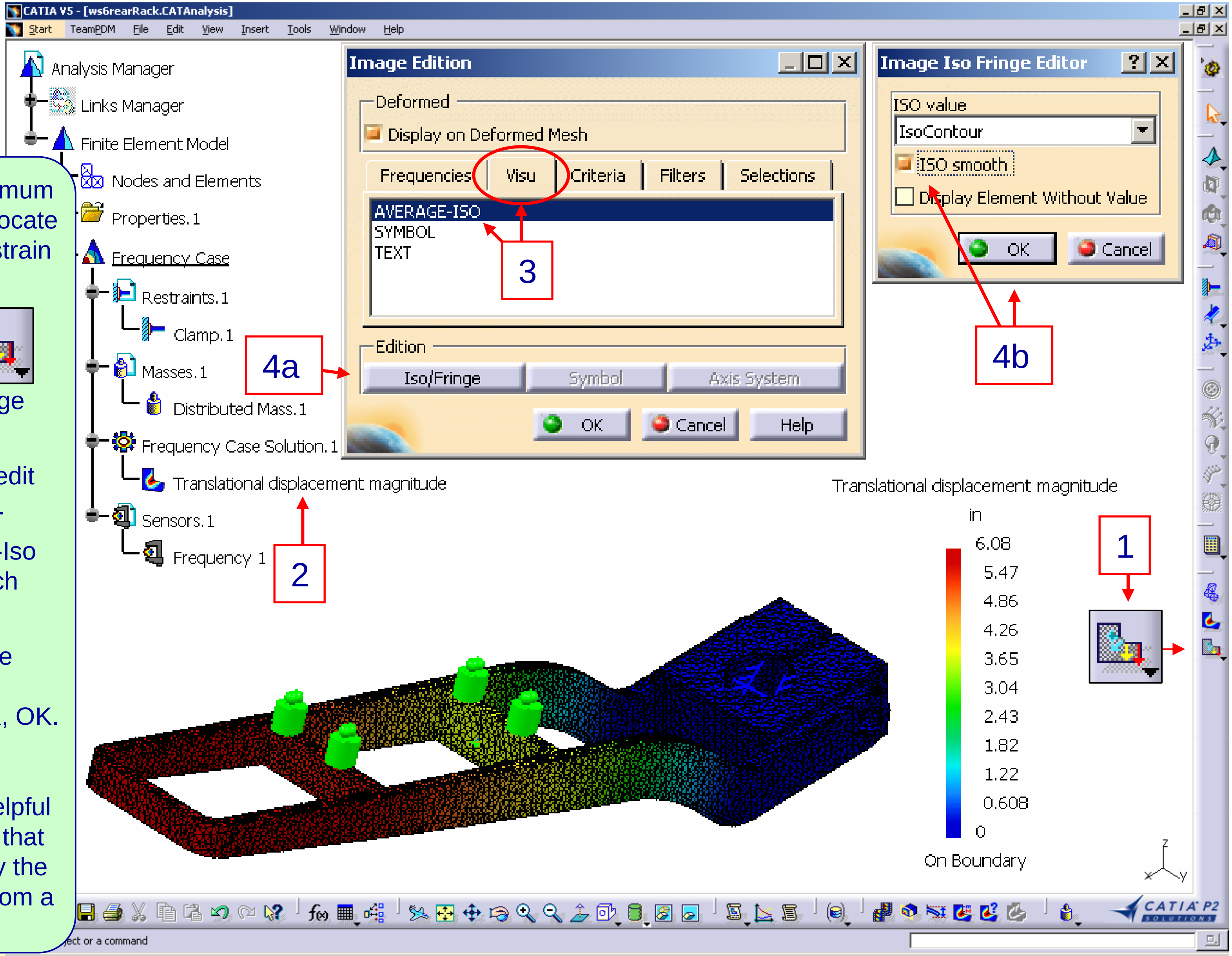
1. Select the Displacement Image Icon.

2. Double click to edit image parameters.

3. Select Average-Iso in Visu tab to switch display.

4. Select Iso/Fringe then select ISO smooth, select OK, OK.

Strain energy is helpful in finding the area that is most affected by the vibration from a natural frequency.



Step 8. Visualize the analysis results

Display all 5 dynamic modes.

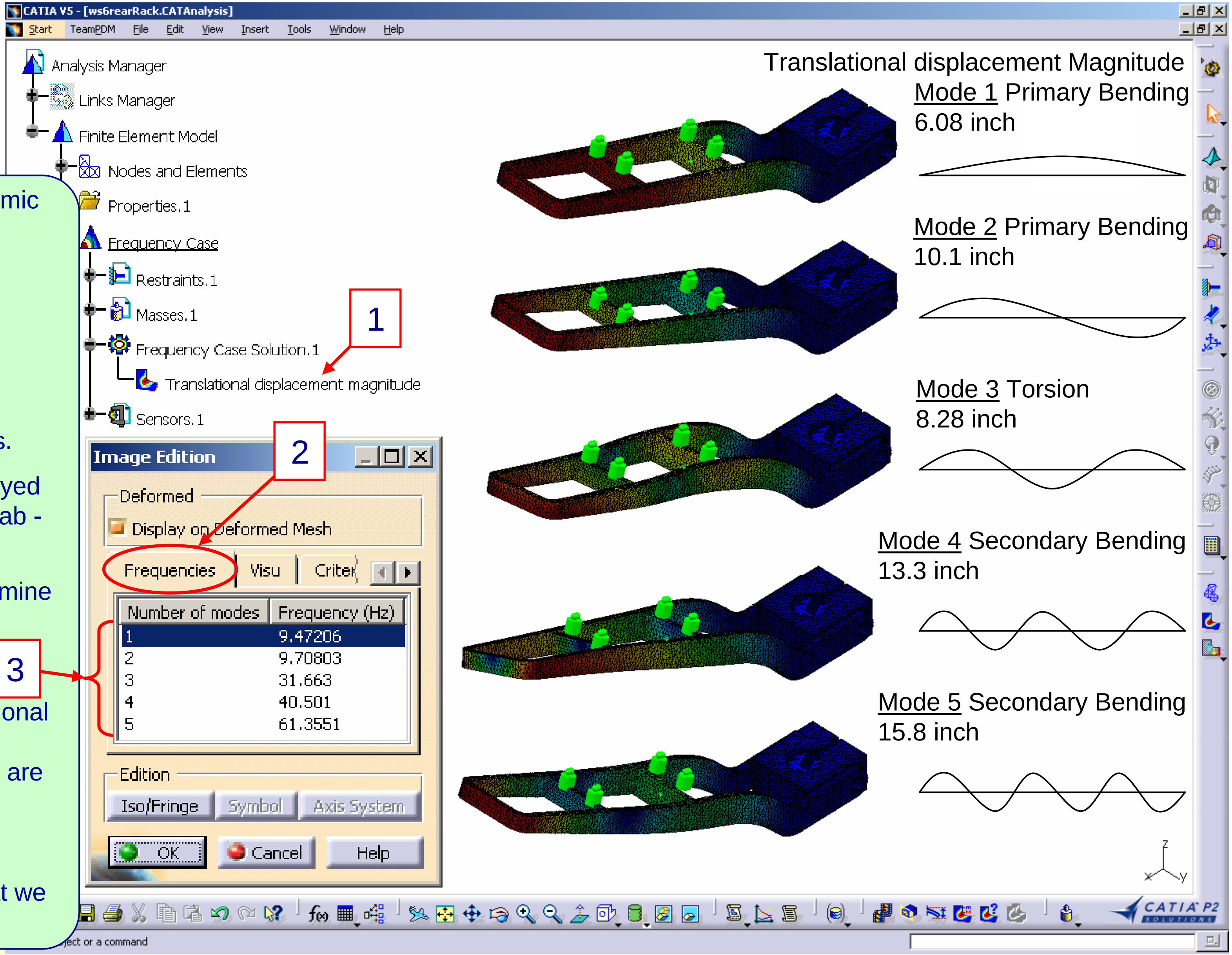
Steps:

1. Double click Translational displacement magnitude to edit image parameters.

2. View the displayed frequency under tab - Frequencies.

3. Select and examine each mode.

Note the Translational displacement magnitude values are arbitrary. The displacement distribution and Frequency is what we want.



MSC SOFTWARE

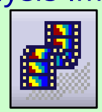
Step 8. Visualize the analysis results

Animate all 5 dynamic modes.

Steps:

1. The Translational displacement magnitude image must be active.

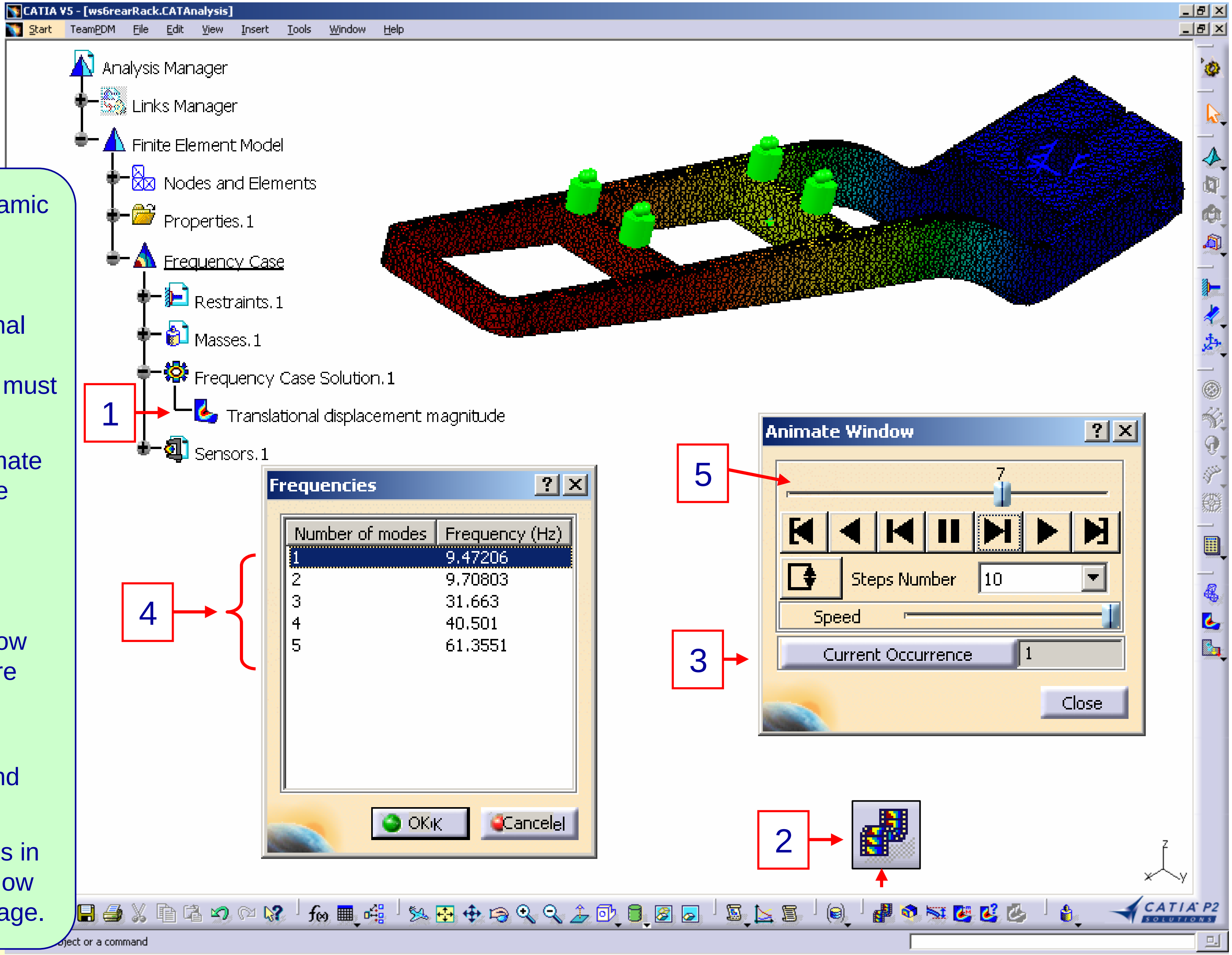
2. Select the Animate an Analysis Image icon.



3. Select Current Occurrence to know what mode you are animating.

4. Select different mode numbers and select OK.

5. Use the controls in the Animate Window to animate the image.



Step 8. Visualize the analysis results

Mode **5** has the greatest displacement, locate the element of maximum strain energy.

Steps:

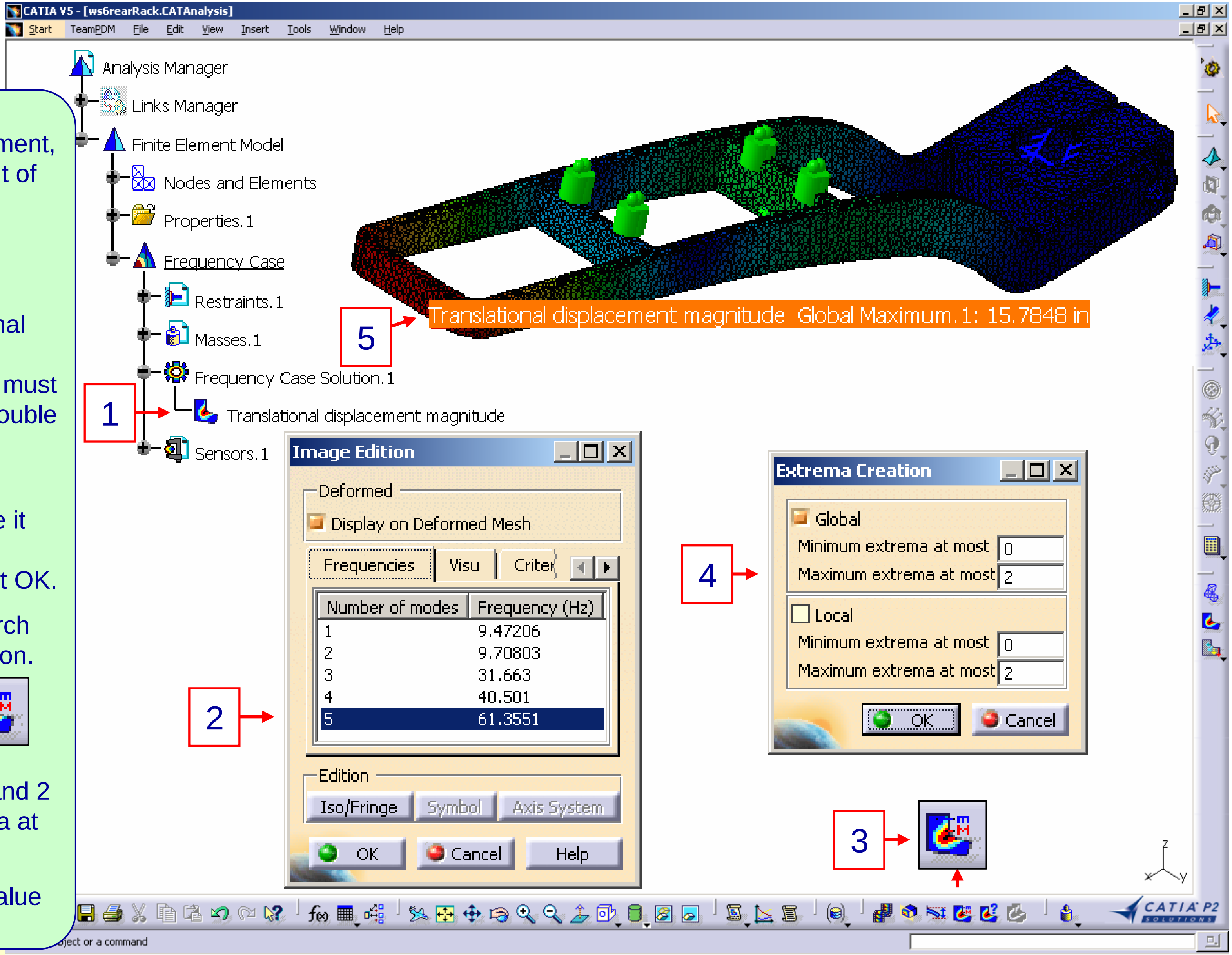
1. The Translational displacement magnitude image must be active. Then double clicked.

2. Select mode number 5 to make it the current occurrence, select OK.

3. Select the Search Image Extrema icon.

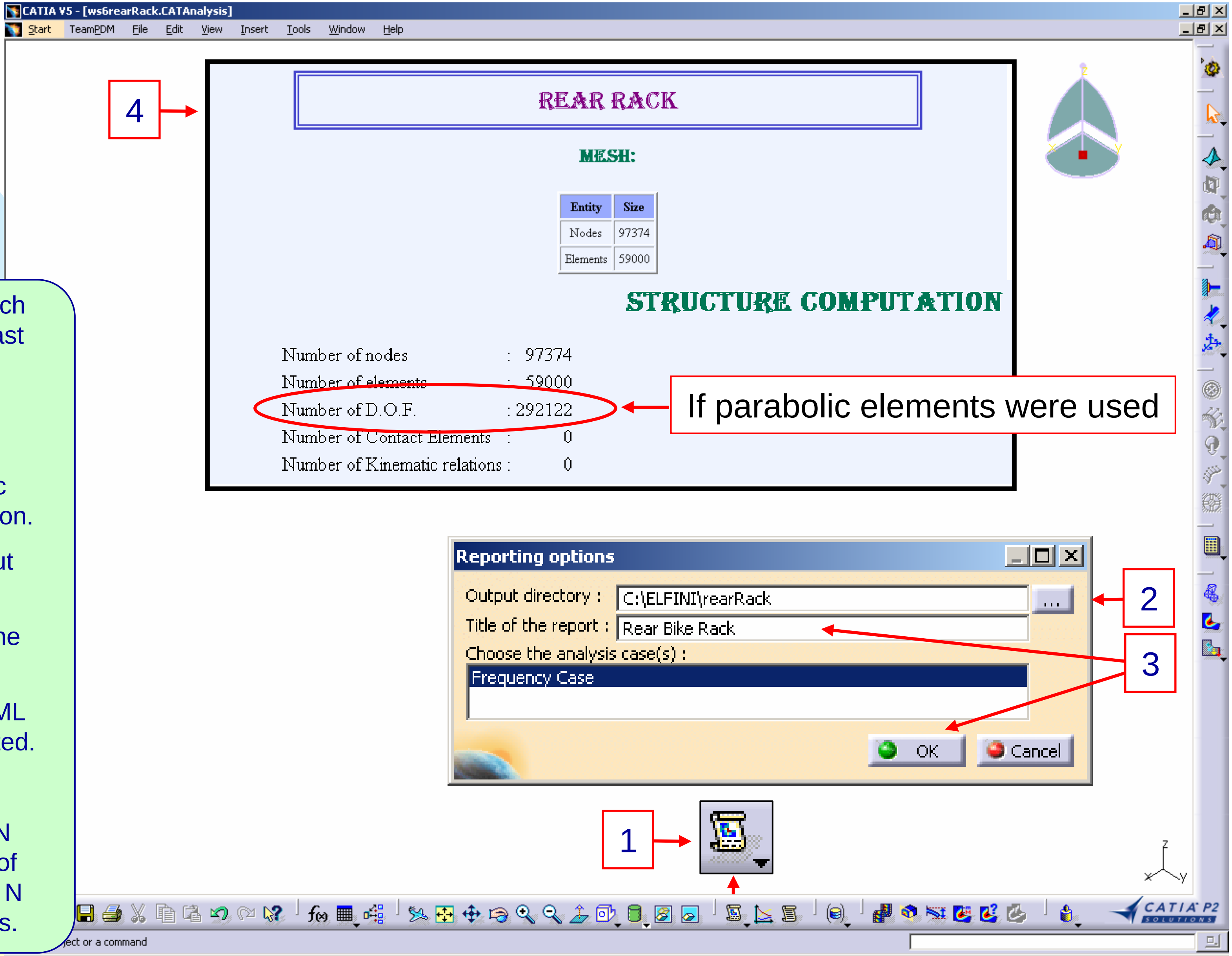
4. Select Global and 2 maximum extrema at most, select OK.

5. Location and value are displayed.



MSC SOFTWARE

Step 9. Generate a report



After activating each mode image at least once, generate a report.

Steps:

1. Select the Basic Analysis Report icon.

2. Select an Output directory.

3. Key in Title of the report, select OK.

4. Review the HTML report that is created.

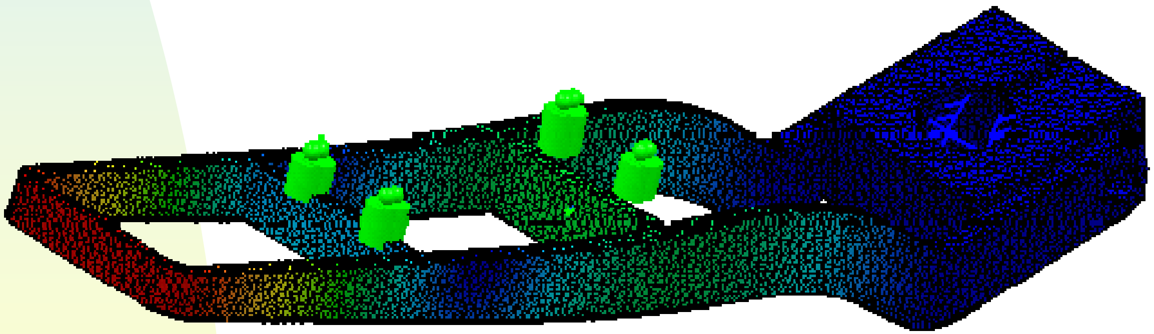
If a structure has N dynamic degrees of freedom there are N natural frequencies.

Step 9. Generate a report

Conclusions

- Comparing the natural frequency of the first 5 dynamic mode shapes shows a large difference. This verifies that we will have smooth ride “soft spots” during this load case.

Mode Number	Bike Frequency Hz (cycles/sec)	Rack Frequency Hz Parabolic Elements
1	95	9.47
2	100	9.71
3	110	31.66
4	120	40.50
5	135	61.36



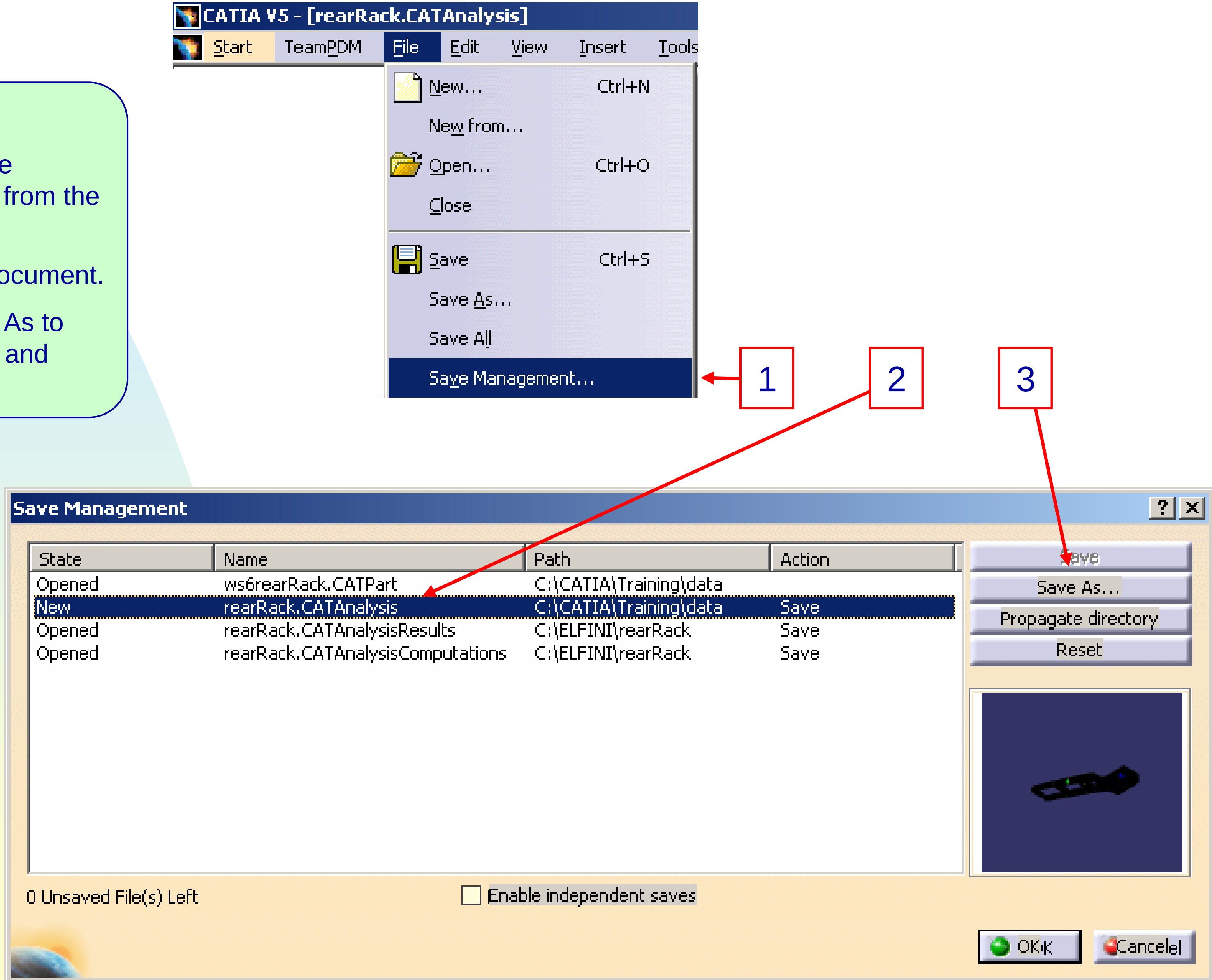
Step 10. Save the analysis document

Steps:

1. Select Save Management from the File menu.

2. Highlight document.

3. Click Save As to specify name and path…OK.



WORKSHOP 7

SEAT POST ASSEMBLY ANALYSIS

MSC SOFTWARE SIMULATING REALITY

Problem Description

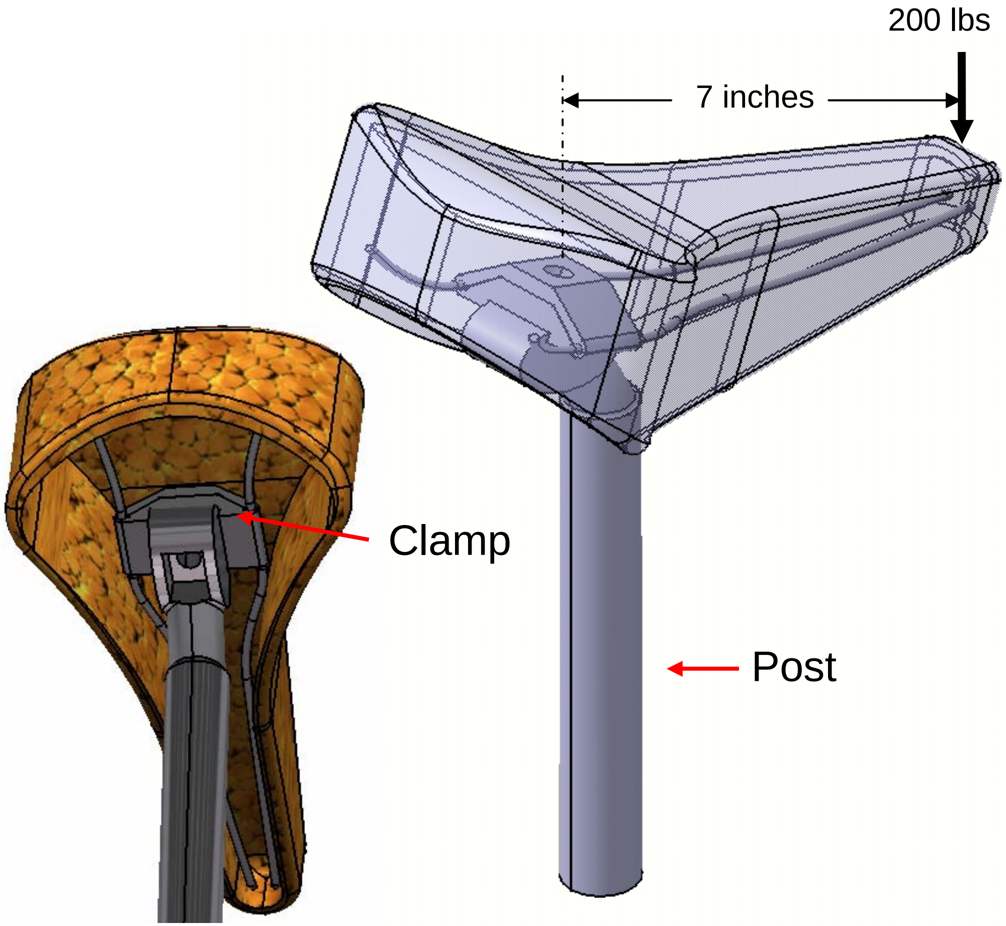
- The sales department has informed engineering that the seat post keeps breaking.
- Perform an assembled static analysis to determine why and recommend a solution. Be conservative by using a design case of 200 lbs forward on the seat.

Post is Aluminum

Elastic Modulus, E	10.15E6 psi
Poisson's Ratio, ν	0.346
Density	.098 lb_in3
Yield Strength	13,778 psi

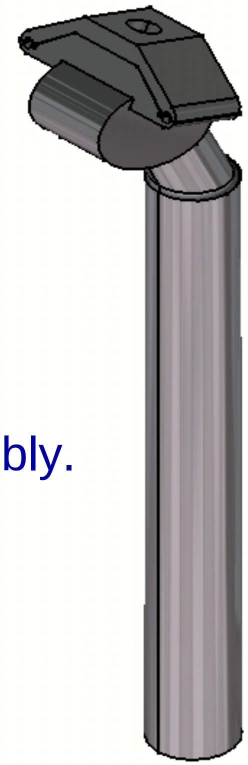
Lower and Upper clamp is Steel

Elastic Modulus, E	29.0E6 psi
Poisson's Ratio, ν	0.3
Density	.284 lb_in3
Yield Strength	36,000 psi



Suggested Exercise Steps

1. Open the existing CATIA product in the Assembly Design workbench.
2. Apply material properties to all parts.
3. Examine and verify assembly constraints.
4. Create an assembly static analysis document (.CATAnalysis).
5. Pre-process initial finite element mesh.
6. Apply Property Connections.
7. Apply a clamp restraint.
8. Apply a moment load.
9. Compute the analysis.
10. Visualize the analysis results.
11. Compute a Frequency (Modal) analysis for the assembly.
12. Generate a report.
13. Save the analysis document.
14. Appendix showing precise results.



Step 1. Open the existing CATIA product (assembly)

Open the CATIA product ws7seatPostassy.CAT Product in the Assembly Design workbench.

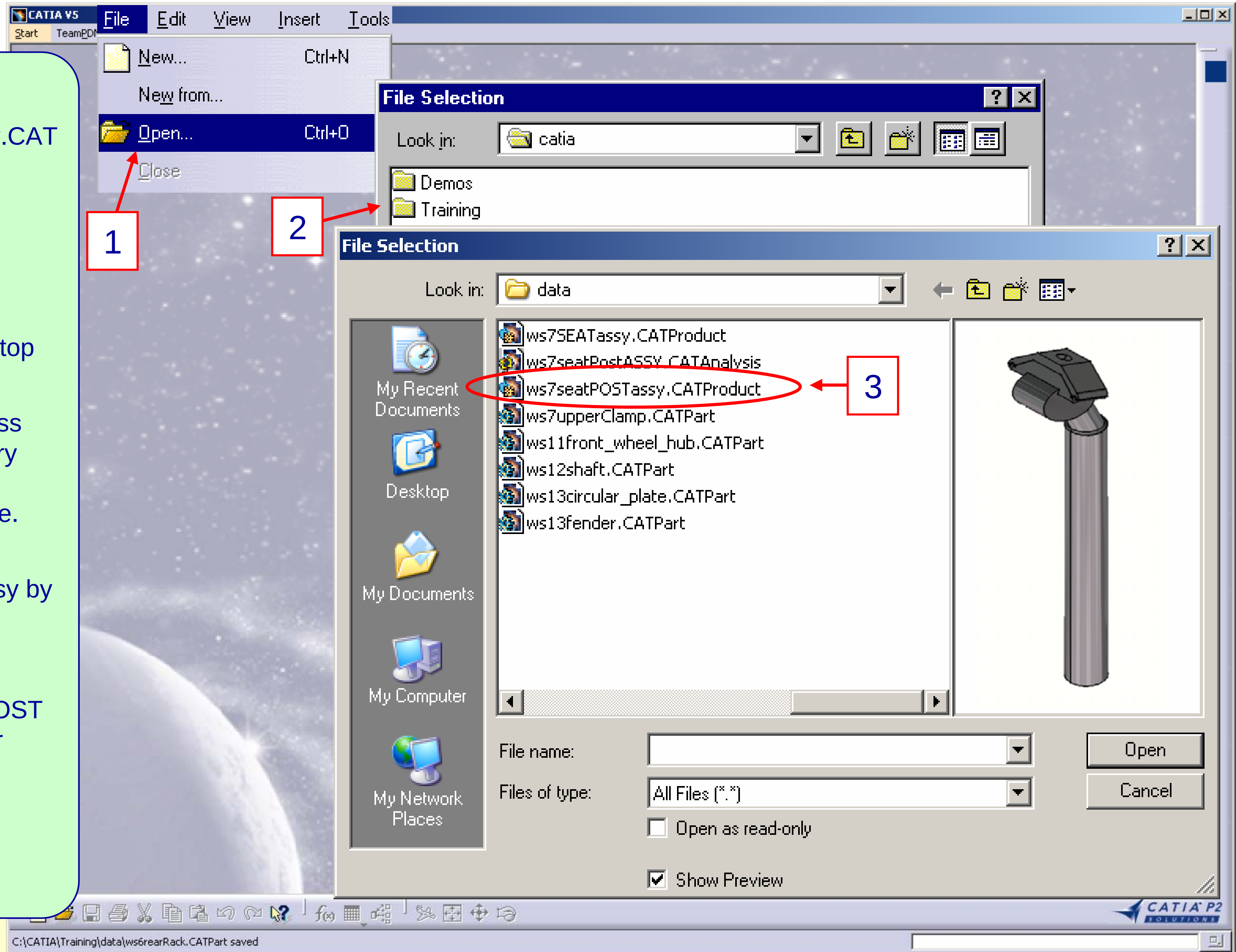
Steps:

1. Select File and Open… from the top pull-down menu.

2. Access the class workshop directory using the typical Windows interface.

3. Open the ws7seatPOSTassy by double-clicking.

By default, the POST assy and all other CATProduct documents are opened in the Assembly Design workbench.



Step 2. Apply material properties to all parts

Steps:

1. Double click the post "Part" representation in the features tree (it should turn blue). This makes the post "Defined in work" and launches you into the Part design workbench.

2. Click the Apply Material icon.

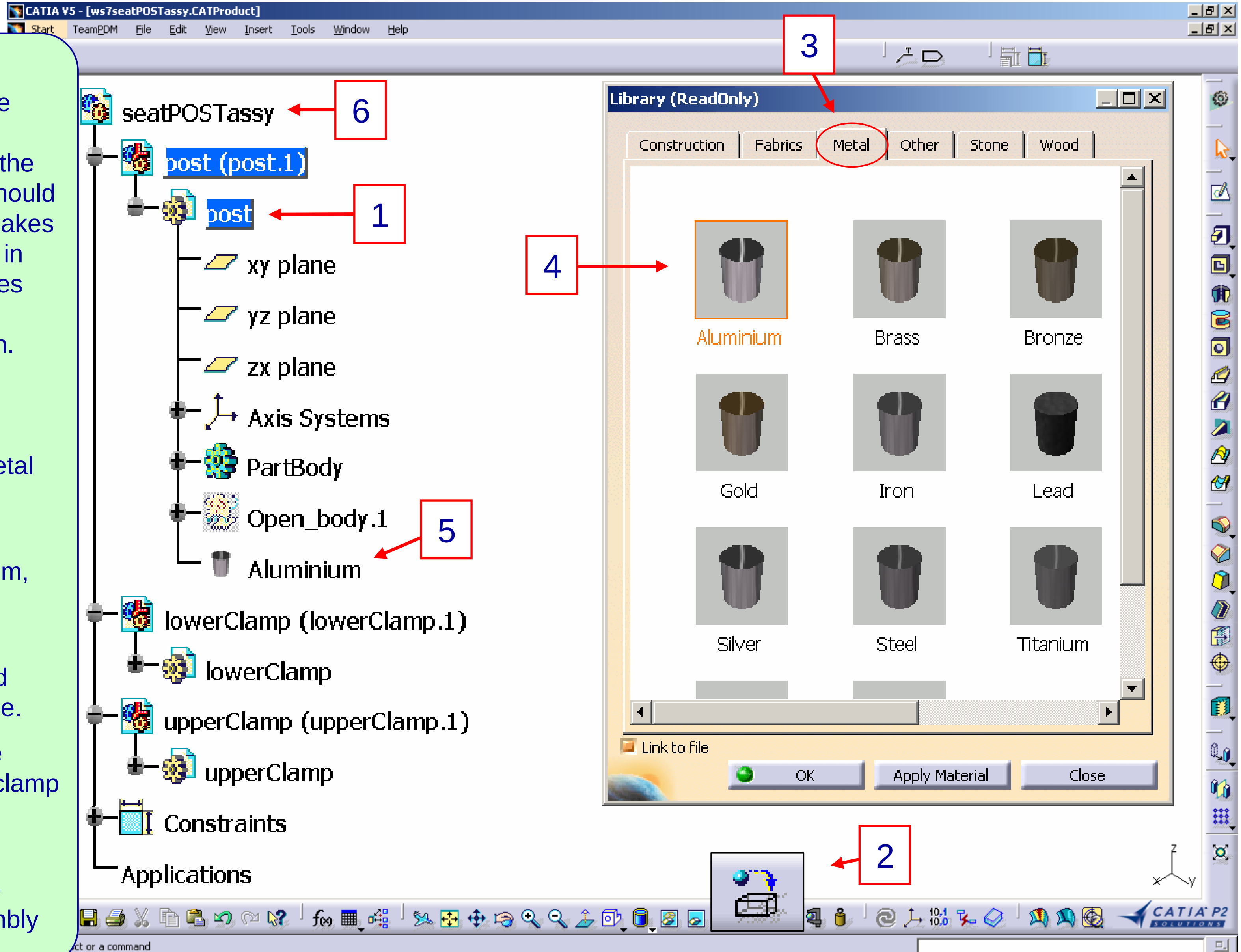
3. Activate the Metal tab in the Library window.

4. Select Aluminum, select OK.

5. Make certain material is applied properly in the tree.

Apply Steel to the lower and upper clamp parts.

6. Double click seatPOSTassy to access the assembly workbench.



MSC SOFTWARE

Step 3. Examine and verify assembly constraints

Steps:

1. You should be in the Assembly design workbench.

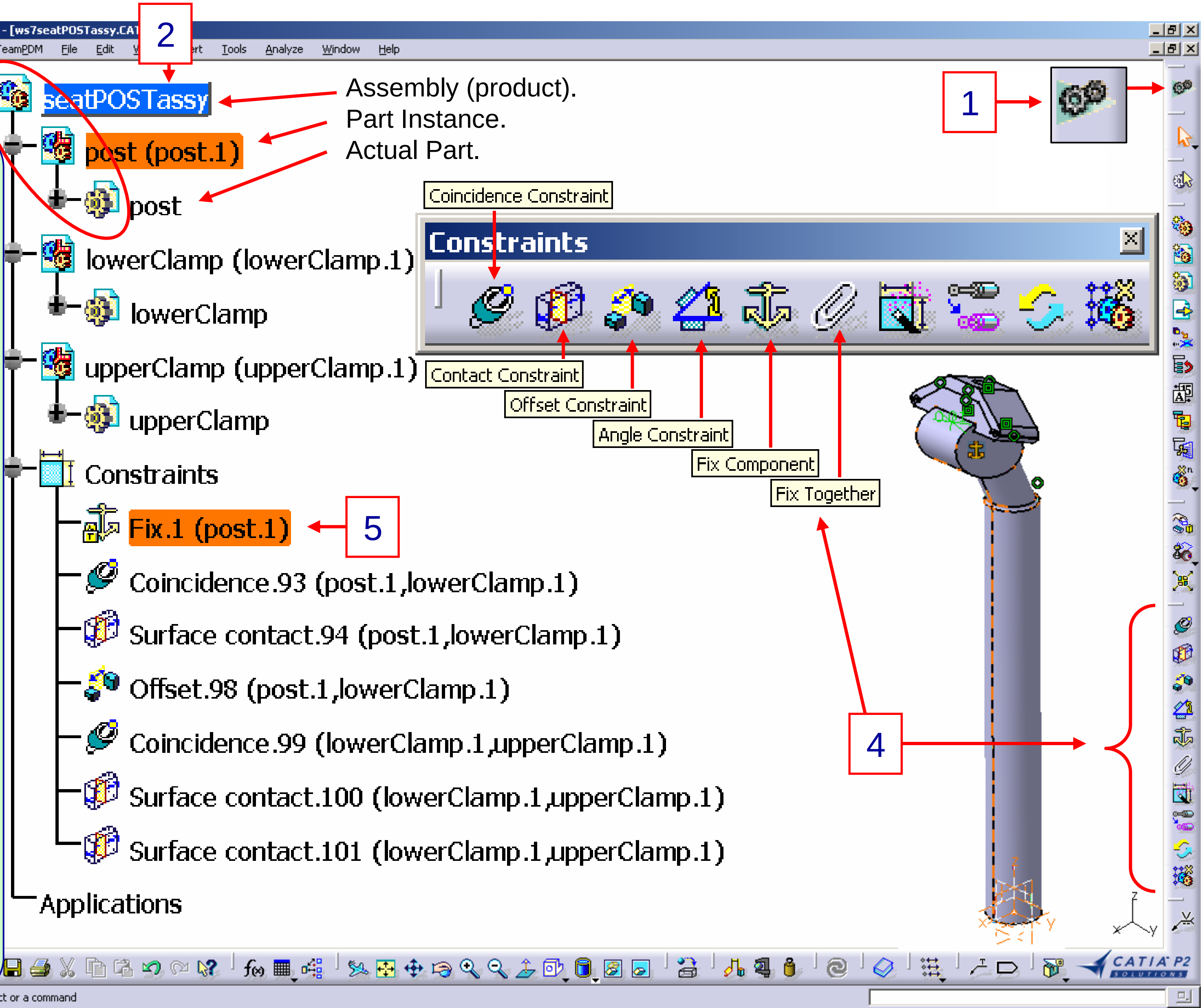
2. The seatPOSTassy object in the features tree is the Product and considered “Defined in work” when blue.

3. Notice the small differences in icons.

4. These are your main assembly tools.

5. Examine the Fix.1 constraint. Assembly constraints should start with an anchor.

If highlighting is not working check Tools + Options + General + Parameters + Symbols.



MSC SOFTWARE

Step 3. Examine and verify assembly constraints

Examine constraints between the post and lower clamp.

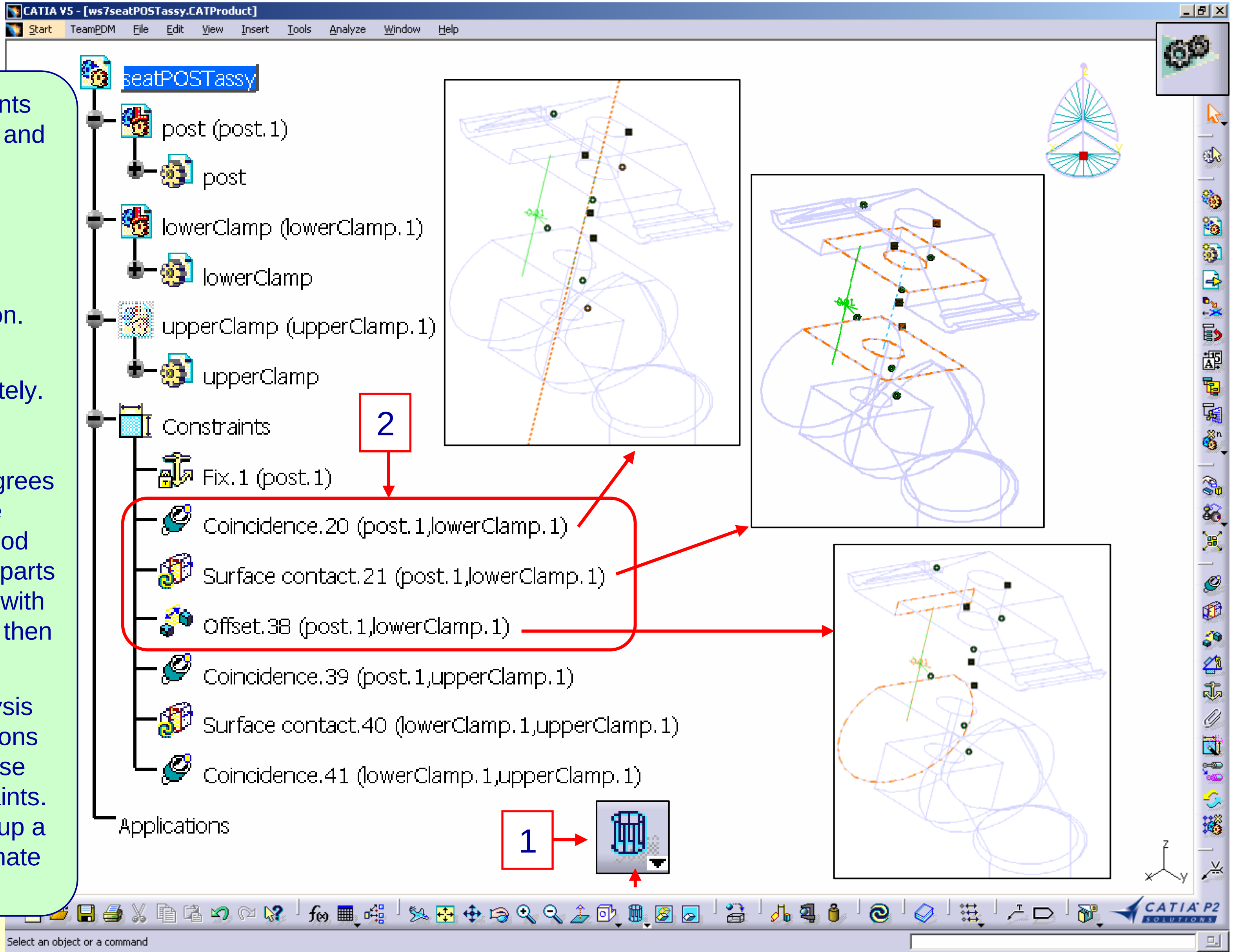
Steps:

1. Select the Wireframe (NHR) visualization option.

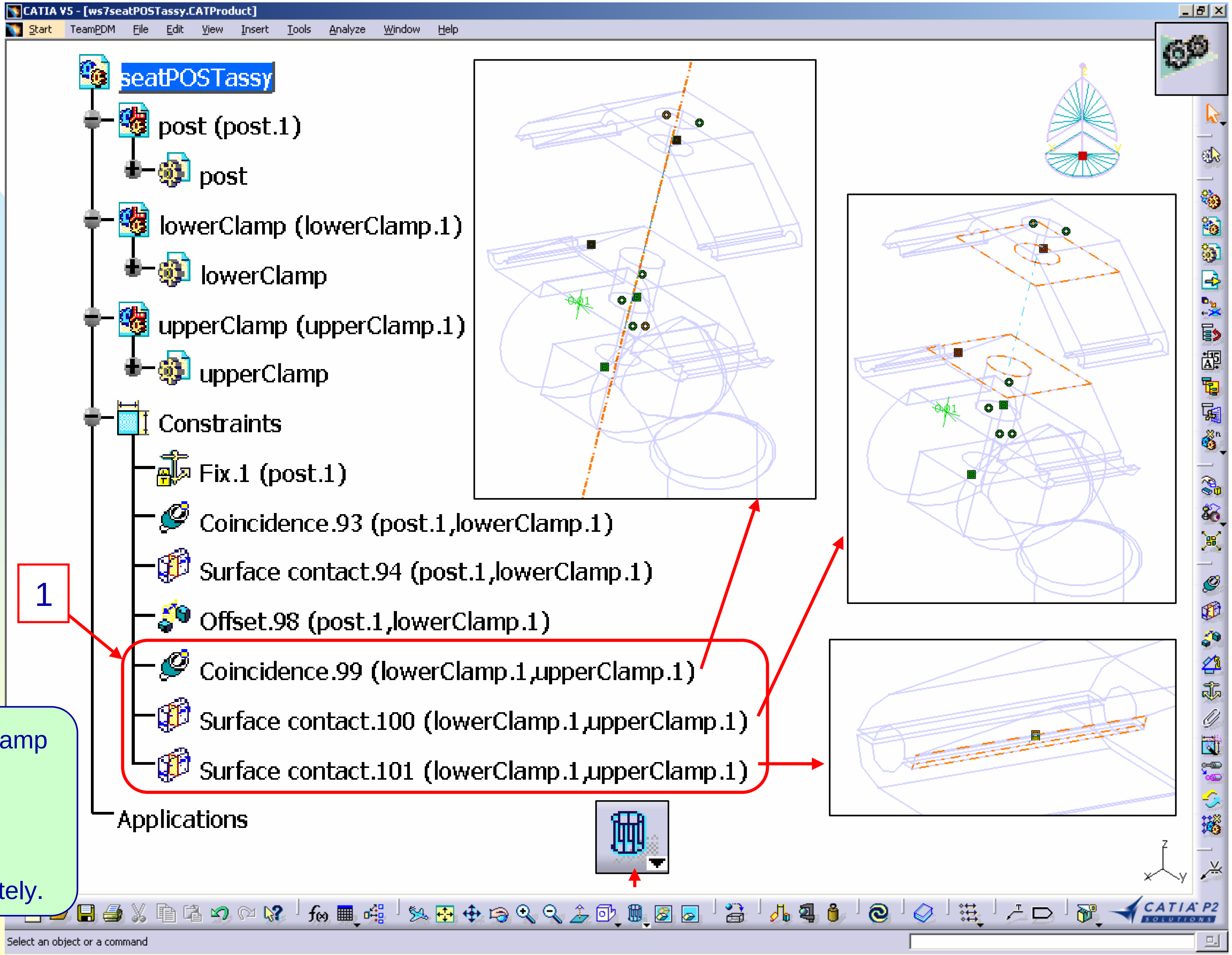
2. Highlight each constraint separately.

Basically all 6 degrees of freedom will be constrained. A good check is to move parts around arbitrarily with the compass and then update.

Also, stress analysis property connections are applied to these assembly constraints. The goal is to setup a statically determinate model.



Step 3. Examine and verify assembly constraints

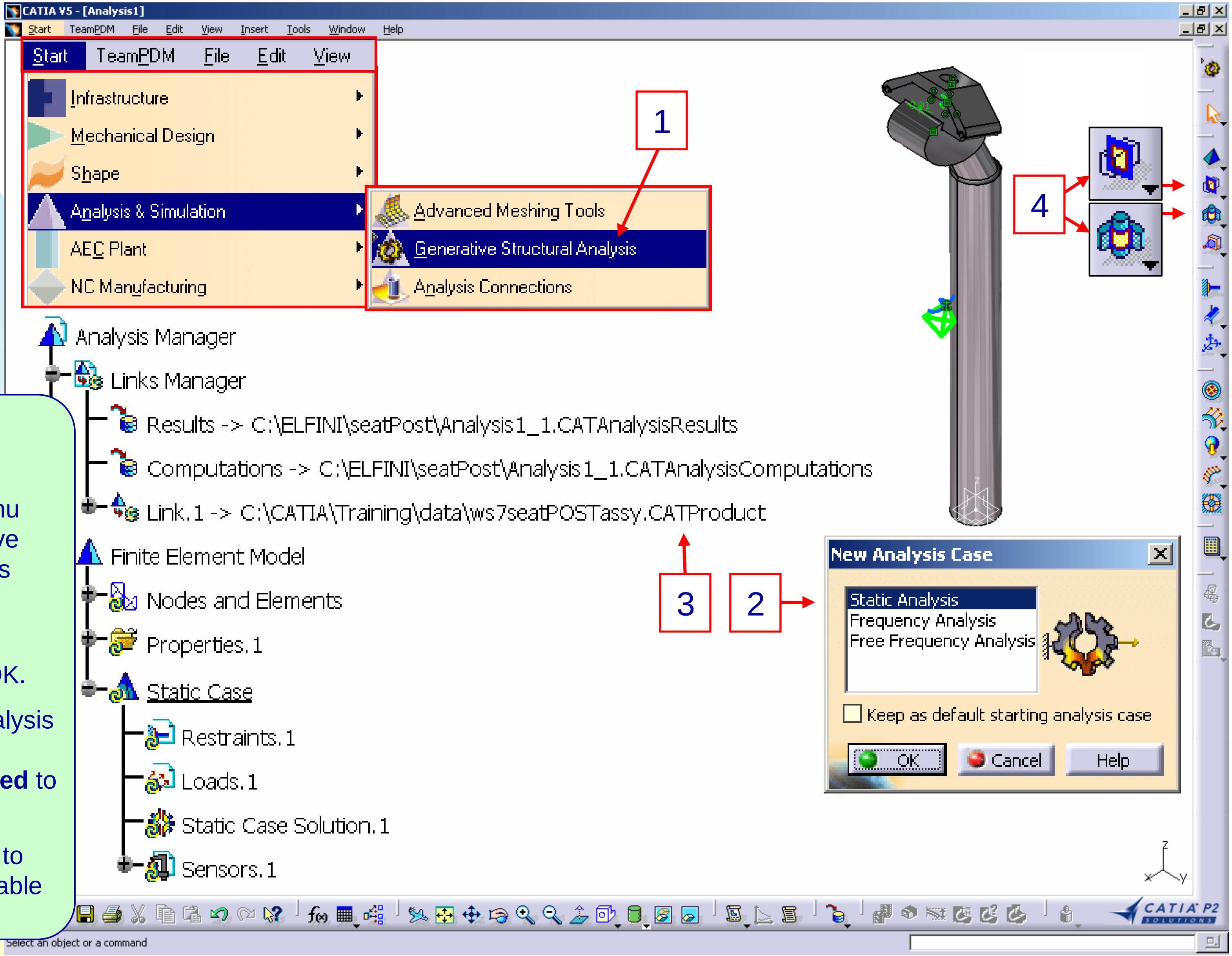


Examine upper clamp constraints.

Steps:

1. Highlight each constraint separately.

Step 4. Create an assembly static analysis document



Just like before.

Steps:

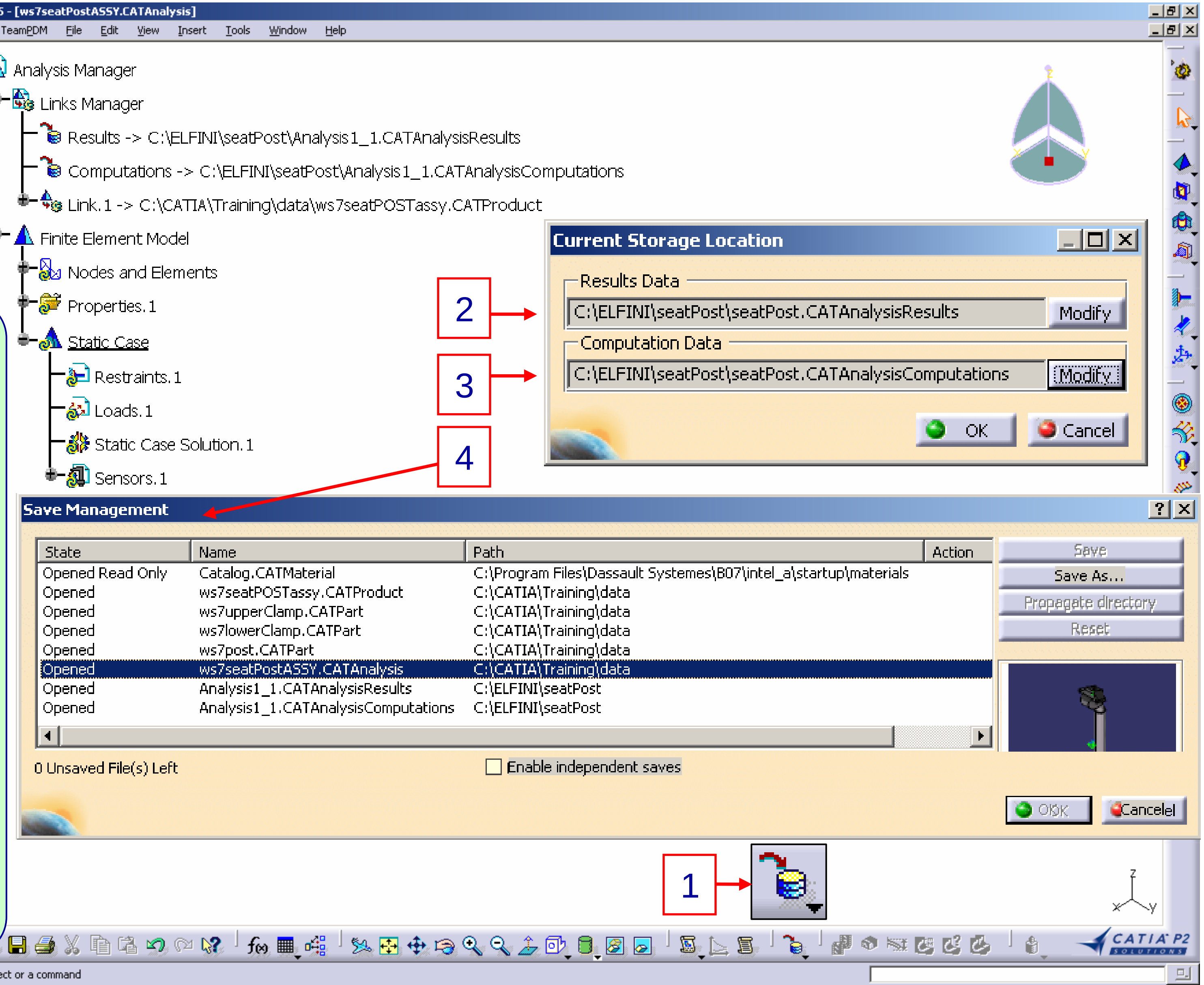
1. From Start menu select a Generative Structural Analysis workbench.

2. Select Static Analysis, select OK.

3. Your Static Analysis document gets automatically **linked** to the CATProduct.

4. One difference to notice is the available Connection icons.

Step 4. Create an assembly static analysis document



Specify unique External Storage directory locations.

Steps:

1. Select the Storage Location icon.

2. Modify the **Results** Storage Location and rename as shown.

3. Modify the **Computation** Storage Location and rename as shown.

4. Good idea to File + Save Management to specify where all documents will be saved.

Step 5. Pre-process initial finite element mesh

Define Linear global finite element mesh properties for all parts.

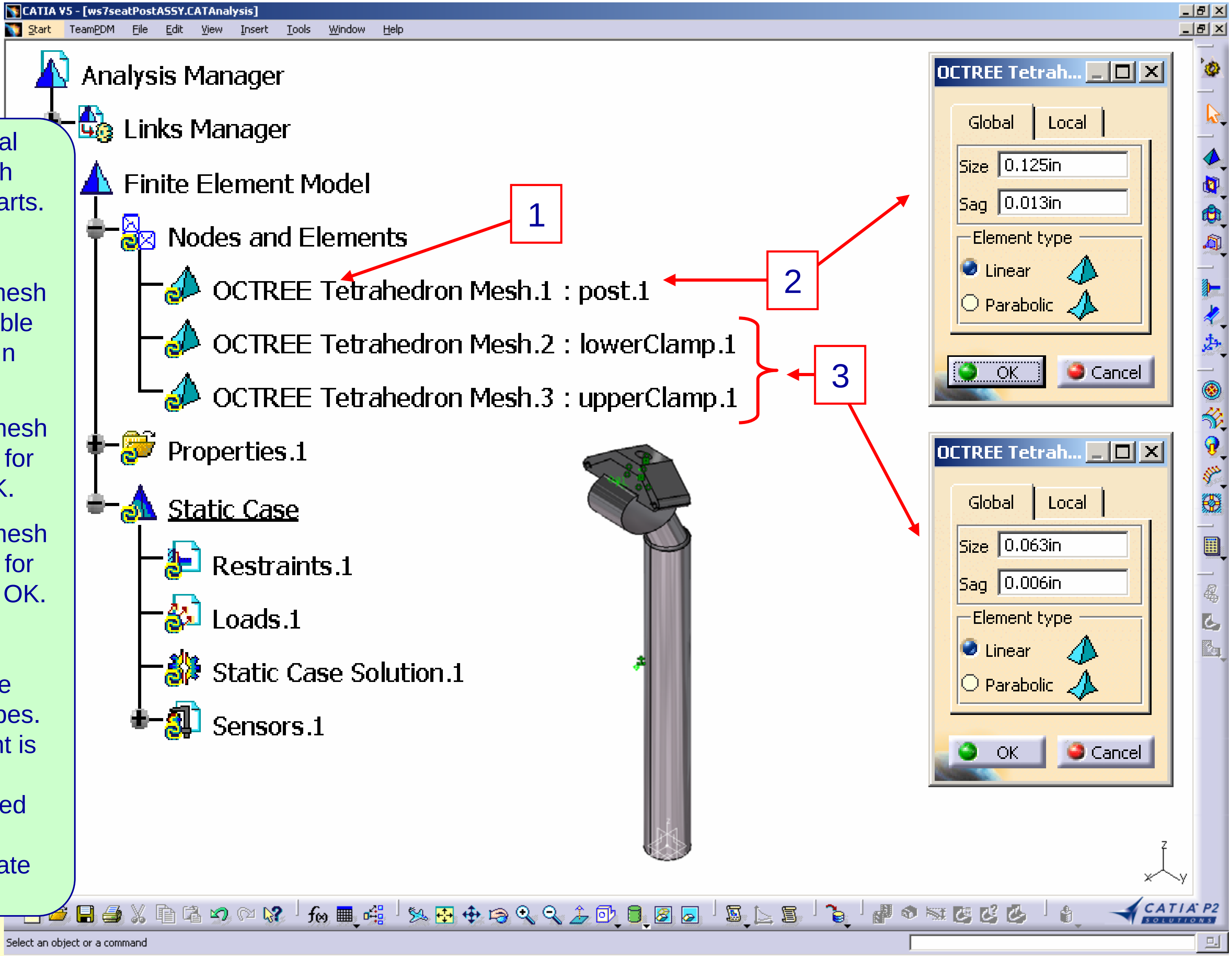
Steps:

1. Edit each part mesh individually by double clicking OCTREE in the features tree.

2. Specify global mesh and sag as shown for the post, select OK.

3. Specify global mesh and sag as shown for the clamps, select OK.

Each part can have unique element types. The Linear element is suggested for computational speed until we achieve a statically determinate model.



Step 6. Apply Property Connections

Define a Fastener Connection.

Steps:

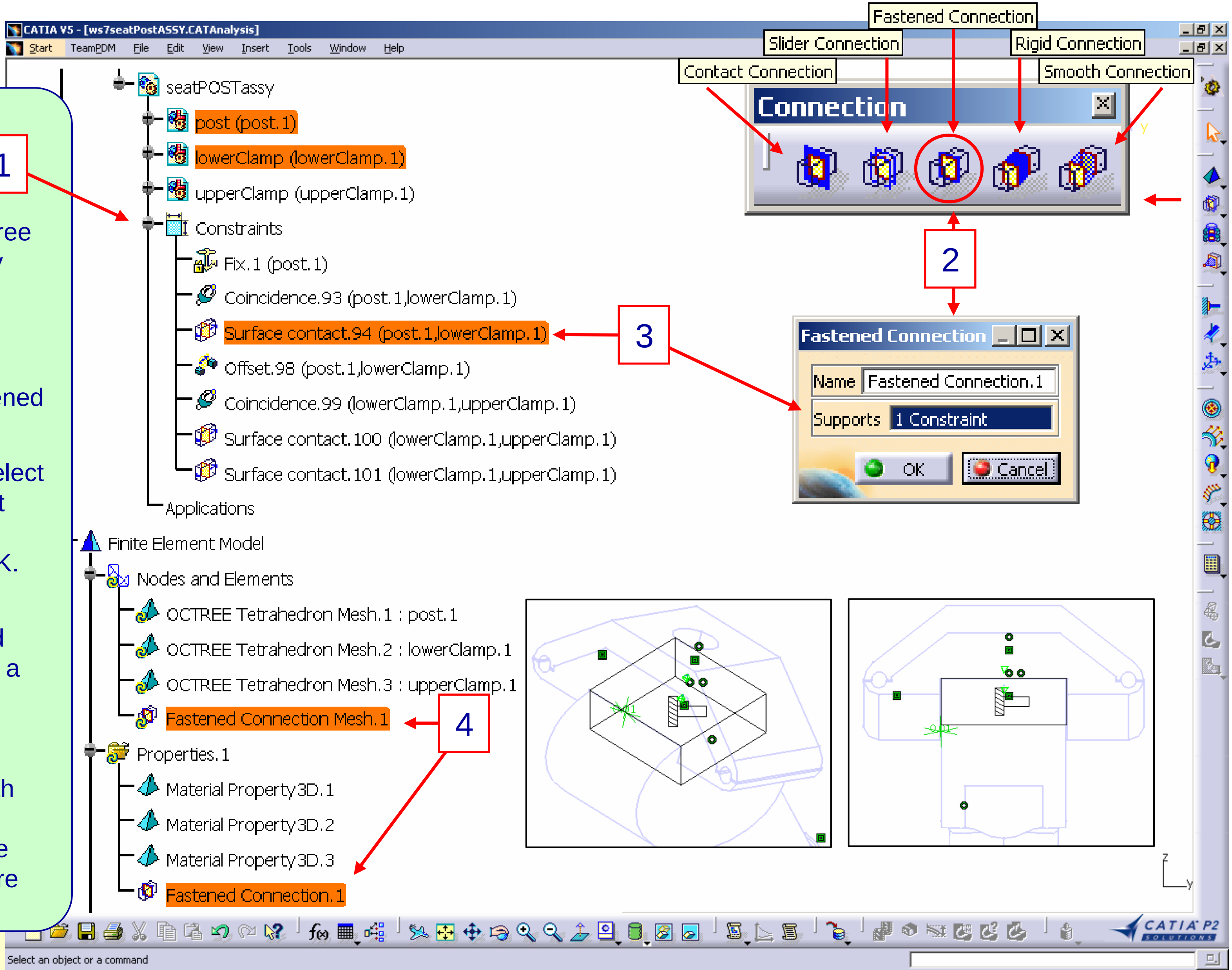
1. In the features tree open the assembly constraints by selecting the + symbol.

2. Select the Fastened Connection icon.

3. From the tree select the surface contact (post.1 to lower clamp.1), select OK.

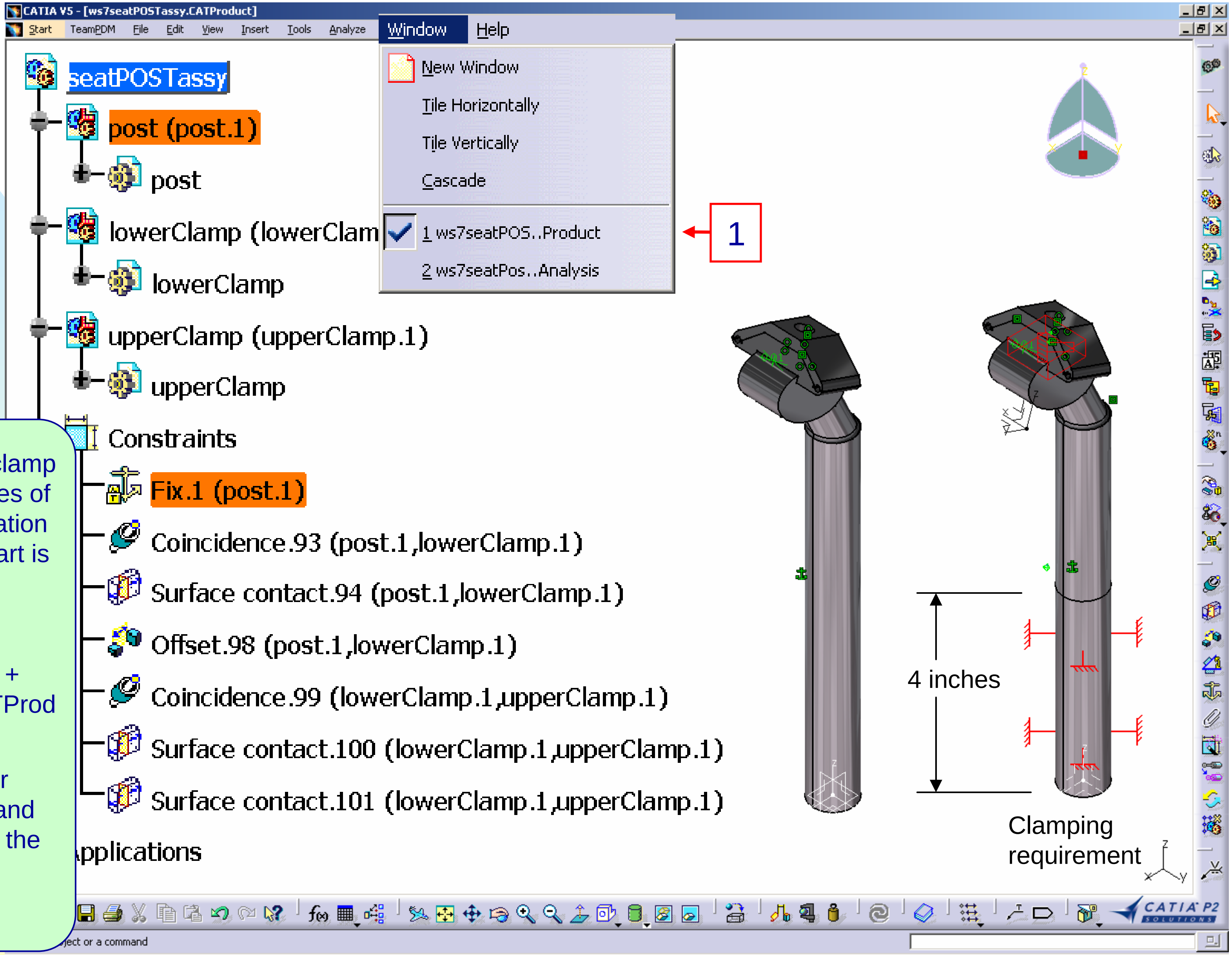
4. Note the mesh connection created between parts and a contact property.

For assemblies with many parts, connections can be renamed to be more meaningful.



MSC SOFTWARE

Step 7. Apply a clamp restraint



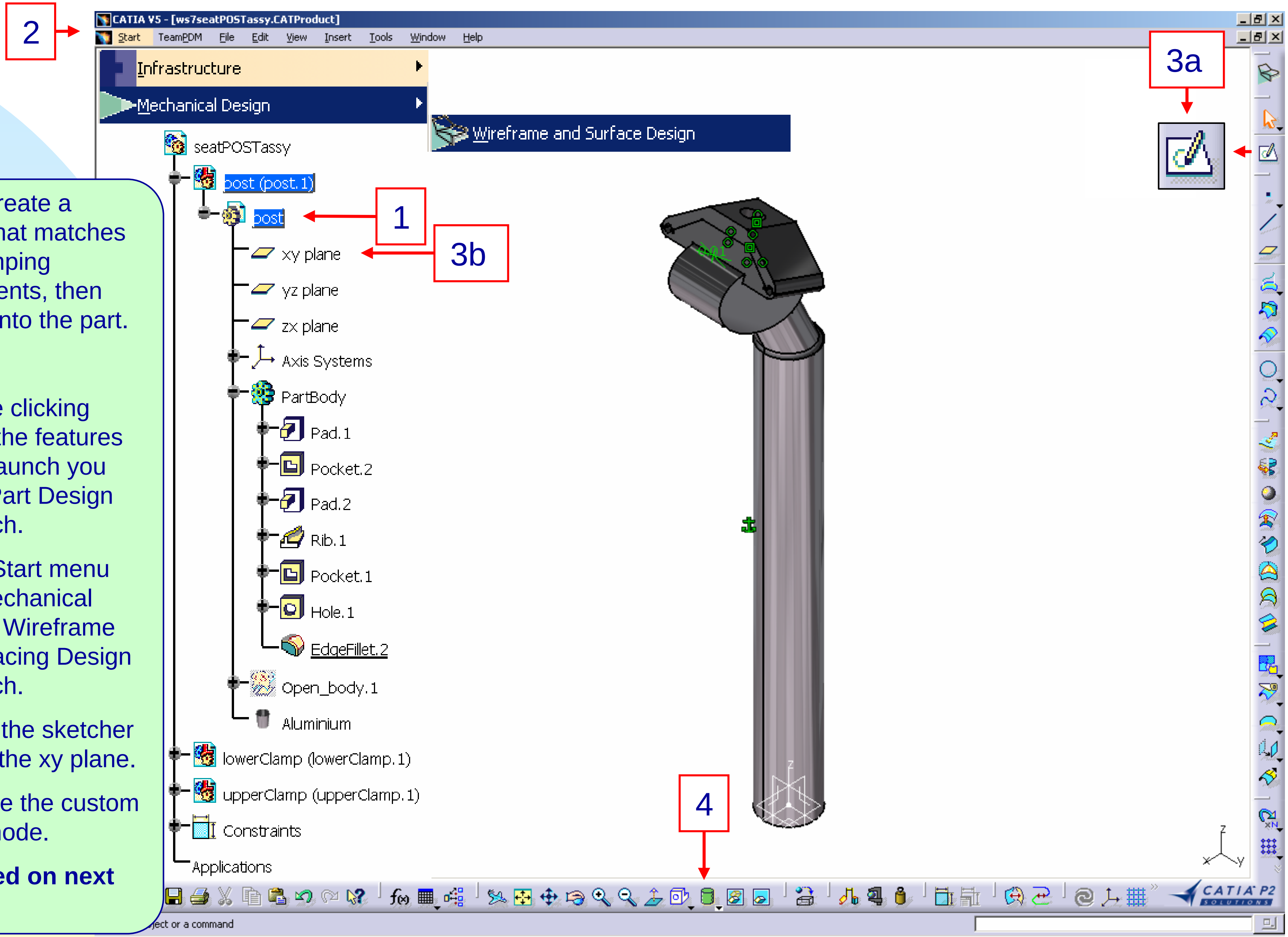
We only want to clamp the bottom 4 inches of the post. Modification of the post.CATPart is required.

Steps:

1. Select Window + ws7seatPost.CATProduct.

This changes your active document and launches you into the Assembly Design Workbench.

Step 7. Apply a clamp restraint



We will create a surface that matches your clamping requirements, then "sew" it onto the part.

Steps:

1. Double clicking "post" in the features tree will launch you into the Part Design workbench.

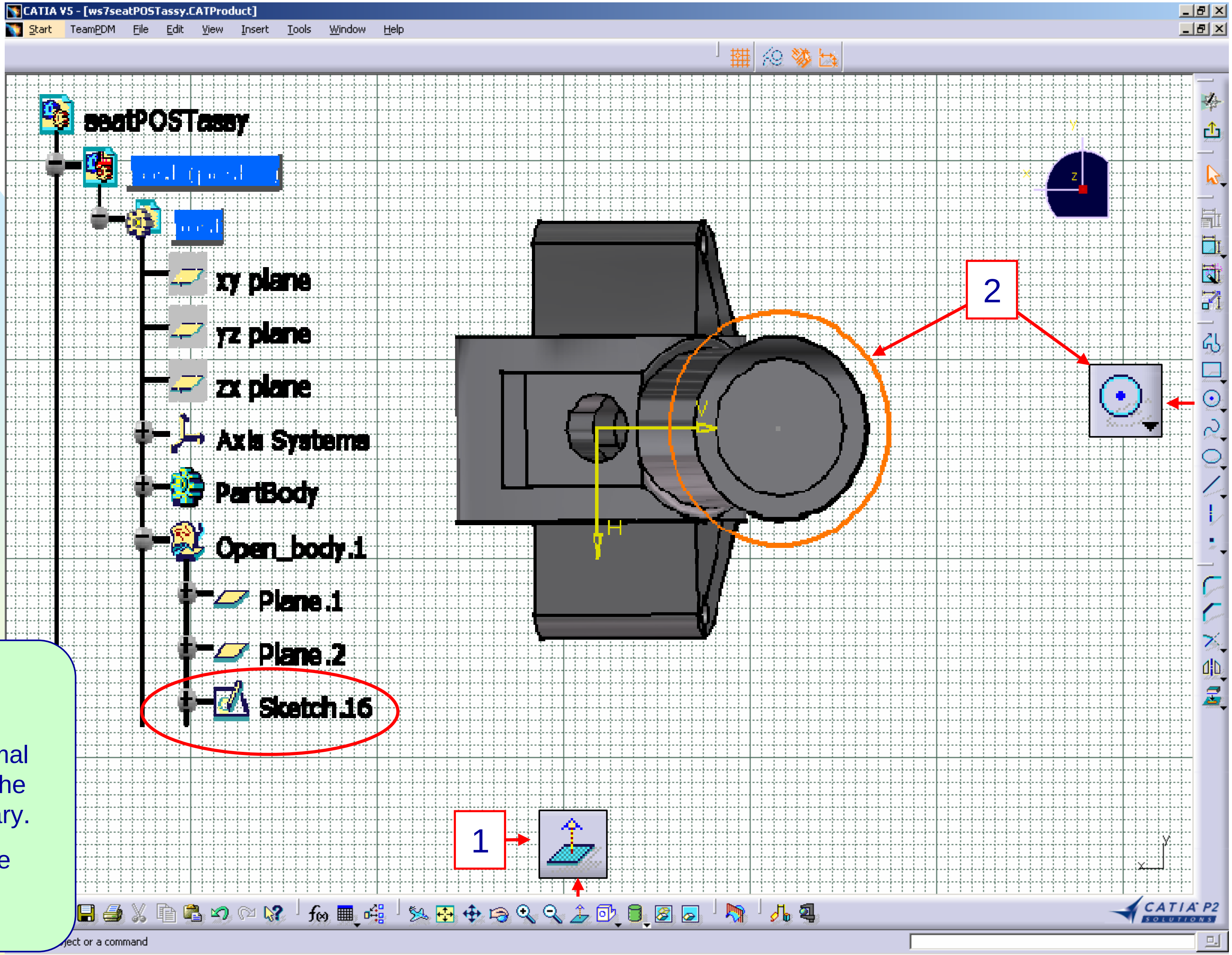
2. From Start menu select Mechanical Design + Wireframe and Surfacing Design workbench.

3. Select the sketcher icon and the xy plane.

4. Activate the custom display mode.

Continued on next page.

Step 7. Apply a clamp restraint



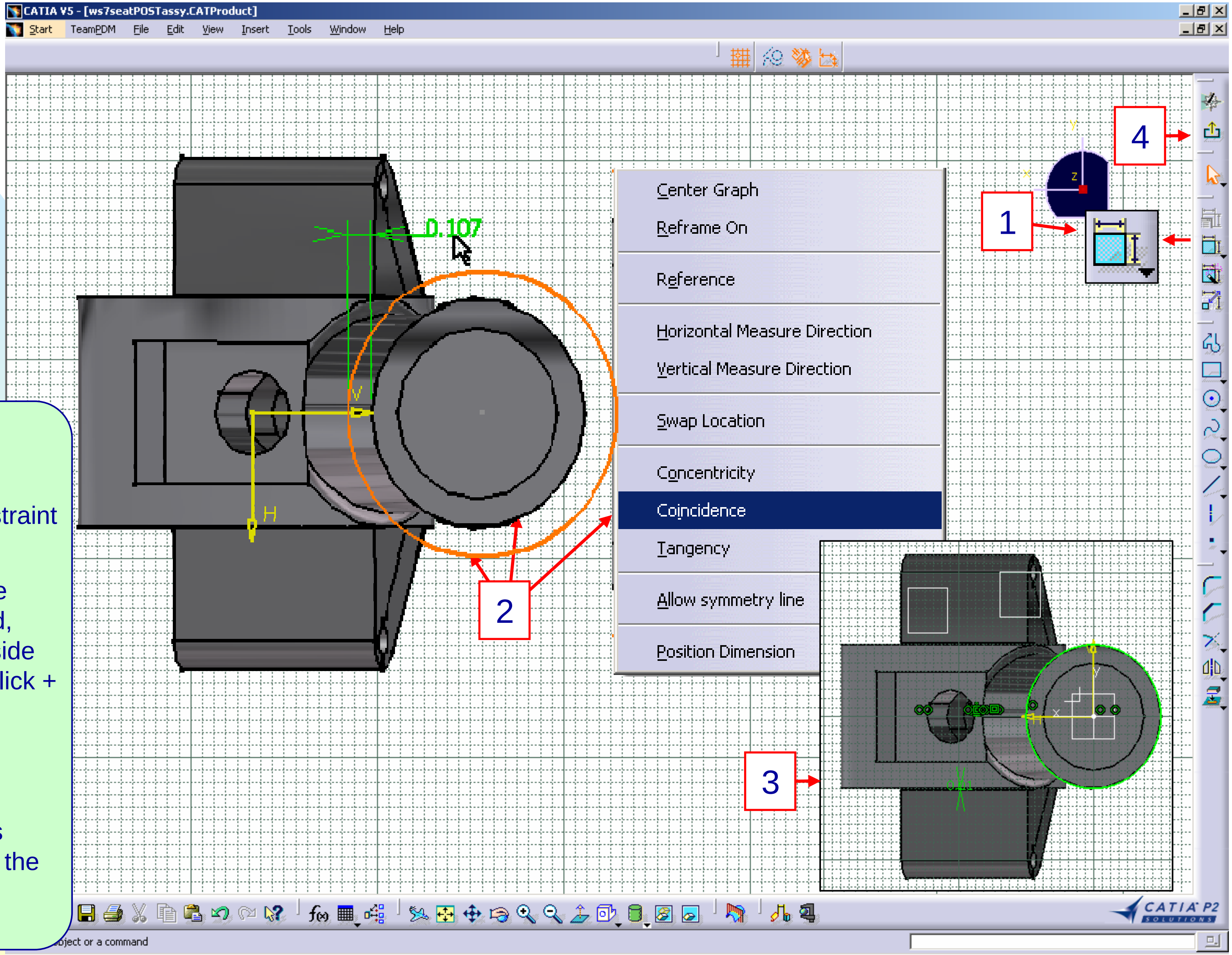
Sketch a shape.

Steps:

1. Select the normal view icon to see the bottom if necessary.

2. Select the circle icon and create a circle as shown.

Step 7. Apply a clamp restraint



Constrain circle.

Steps:

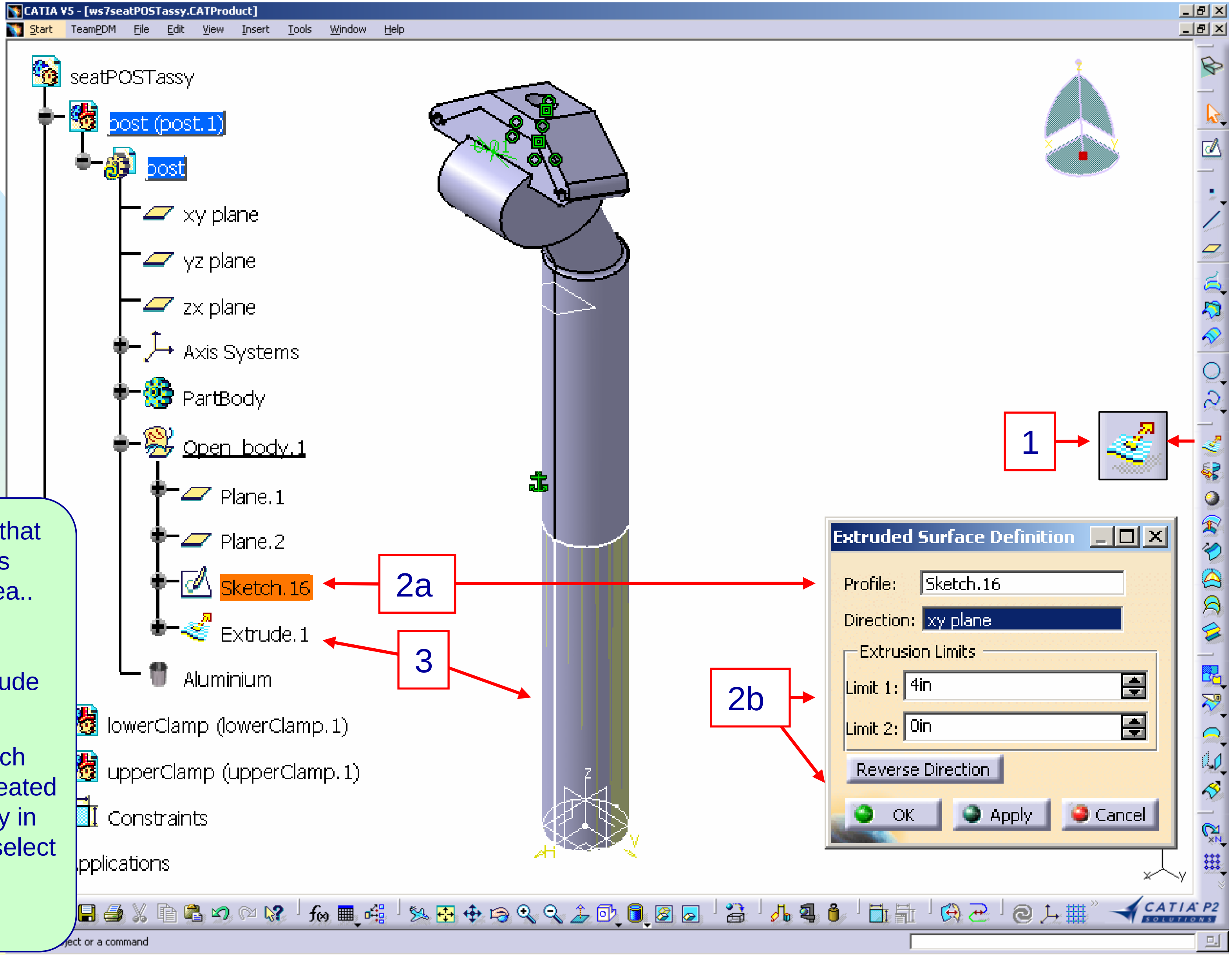
1. Select the constraint icon.

2. Select the circle previously created, then the post outside diameter. Right click + Coincidence.

3. Result

4. Select the exit sketcher icon, this takes you back to the Part Design workbench.

Step 7. Apply a clamp restraint



Create a surface that exactly represents your clamping area..

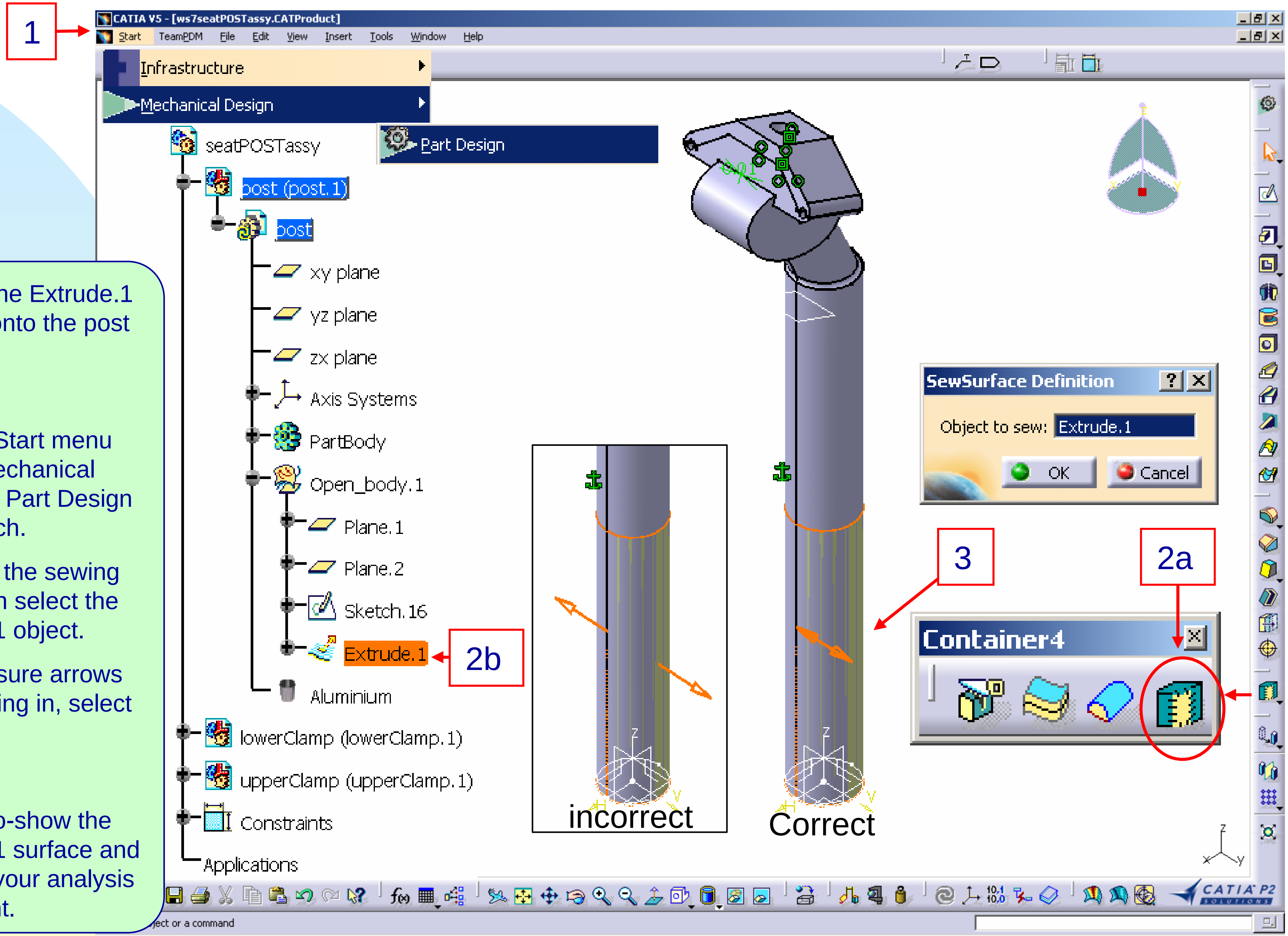
Steps:

1. Select the Extrude icon.

2. Select the sketch you previously created as the Profile. Key in limits as shown, select OK.

3. Result

Step 7. Apply a clamp restraint



Sewing the Extrude.1 surface onto the post part.

Steps:

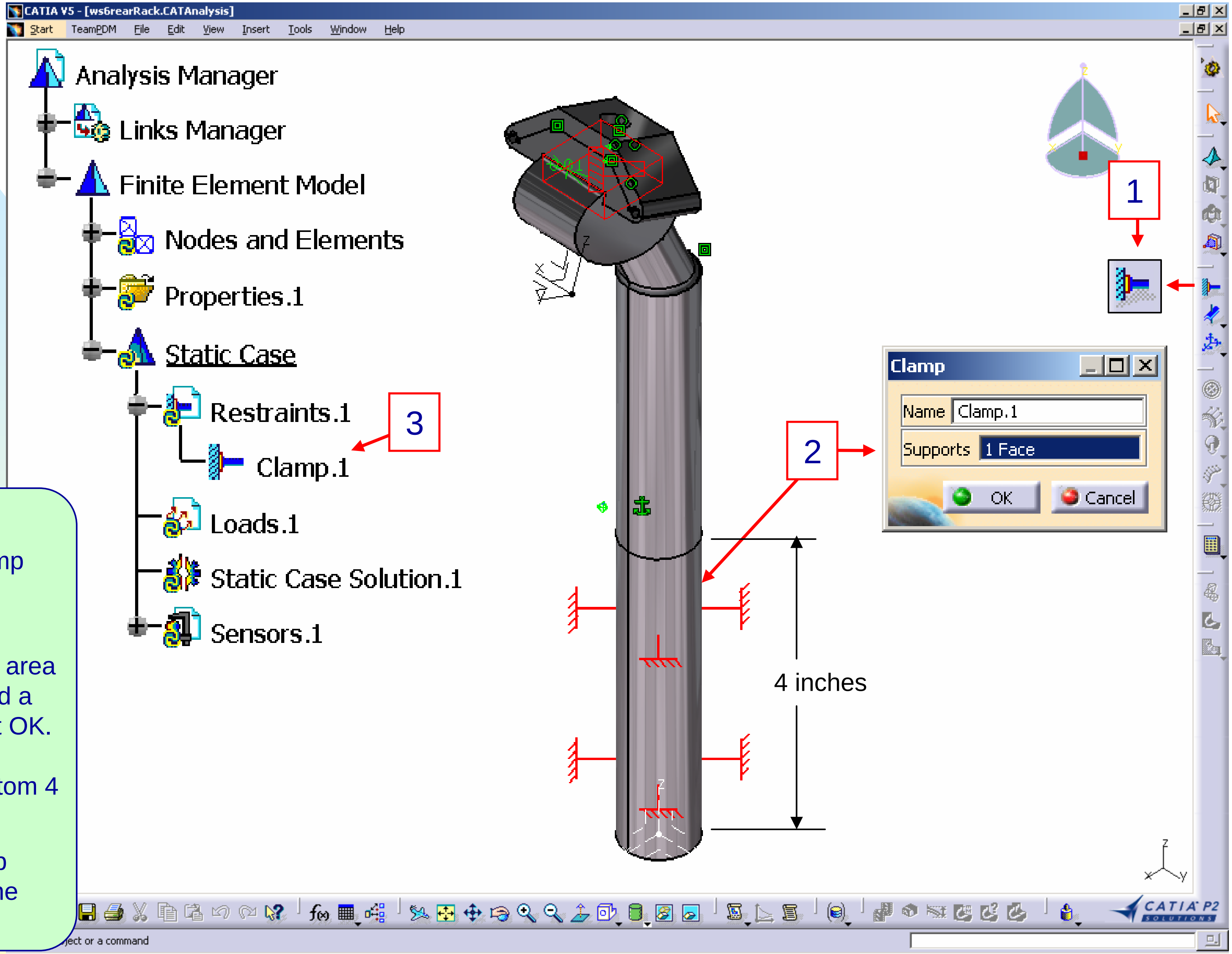
1. From Start menu select Mechanical Design + Part Design workbench.

2. Select the sewing icon, then select the Extrude.1 object.

3. Make sure arrows are pointing in, select OK.

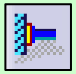
Finally no-show the Extrude.1 surface and activate your analysis document.

Step 7. Apply a clamp restraint



Steps:

1. Select the Clamp Restraint icon.



2. Select the post area that we just sewed a surface on, select OK. The clamp only applies to the bottom 4 inches.

3. Note the Clamp object added to the features tree.

Step 8. Apply a moment load

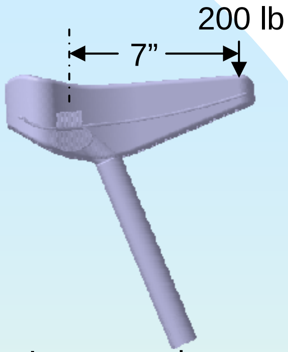
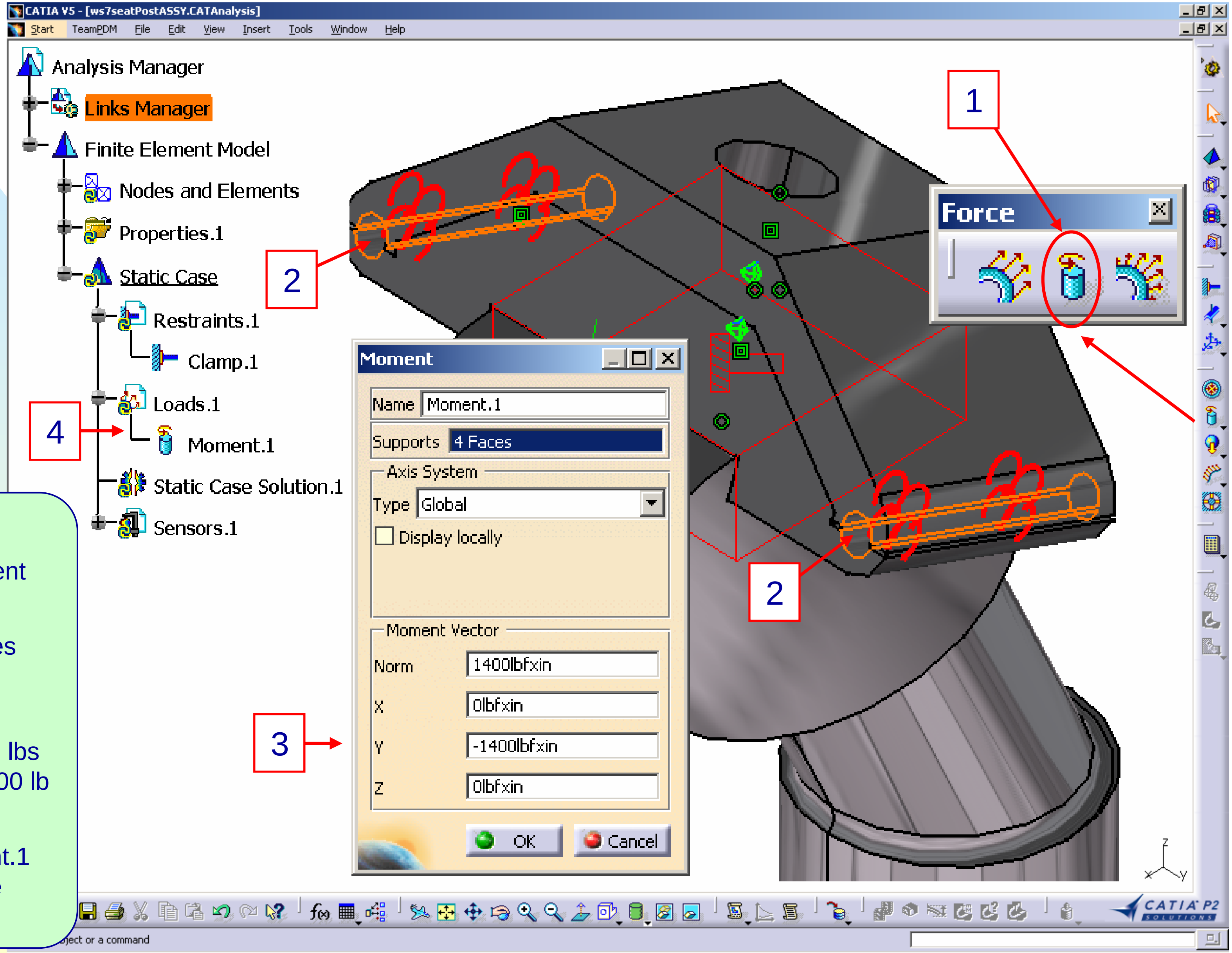


Image to show
The –y moment



Steps:

1. Select the Moment icon.

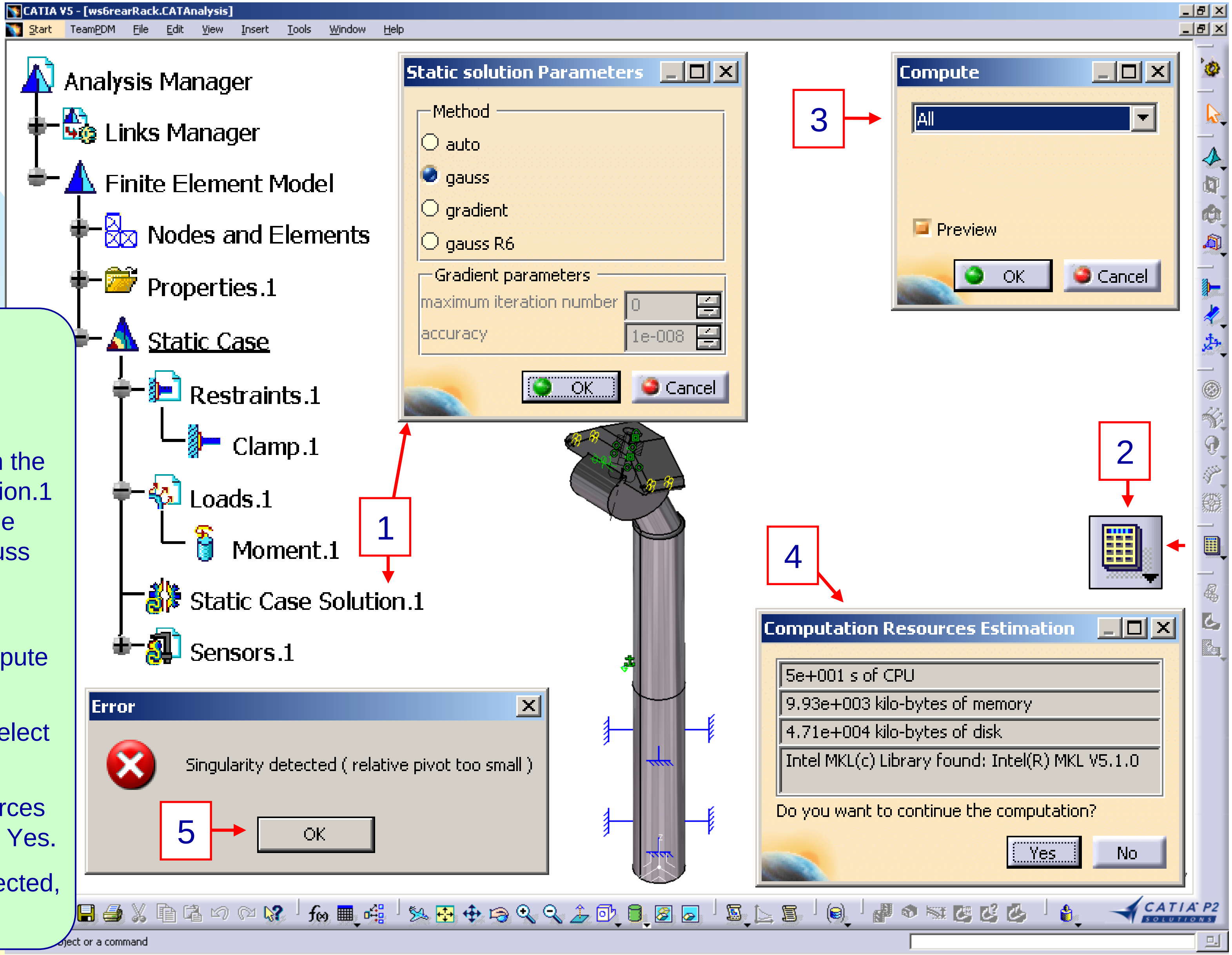
2. Select the 4 faces where the seat attaches.

3. Enter -1400 inch lbs about the y-axis (200 lb x 7 inches).

4. Note the Moment.1 object added to the features tree.

MSC SOFTWARE

Step 9. Compute the analysis



Debugging singularities.

Steps:

1. Double click on the Static Case Solution.1 in the features tree and verify the gauss solution type is selected.

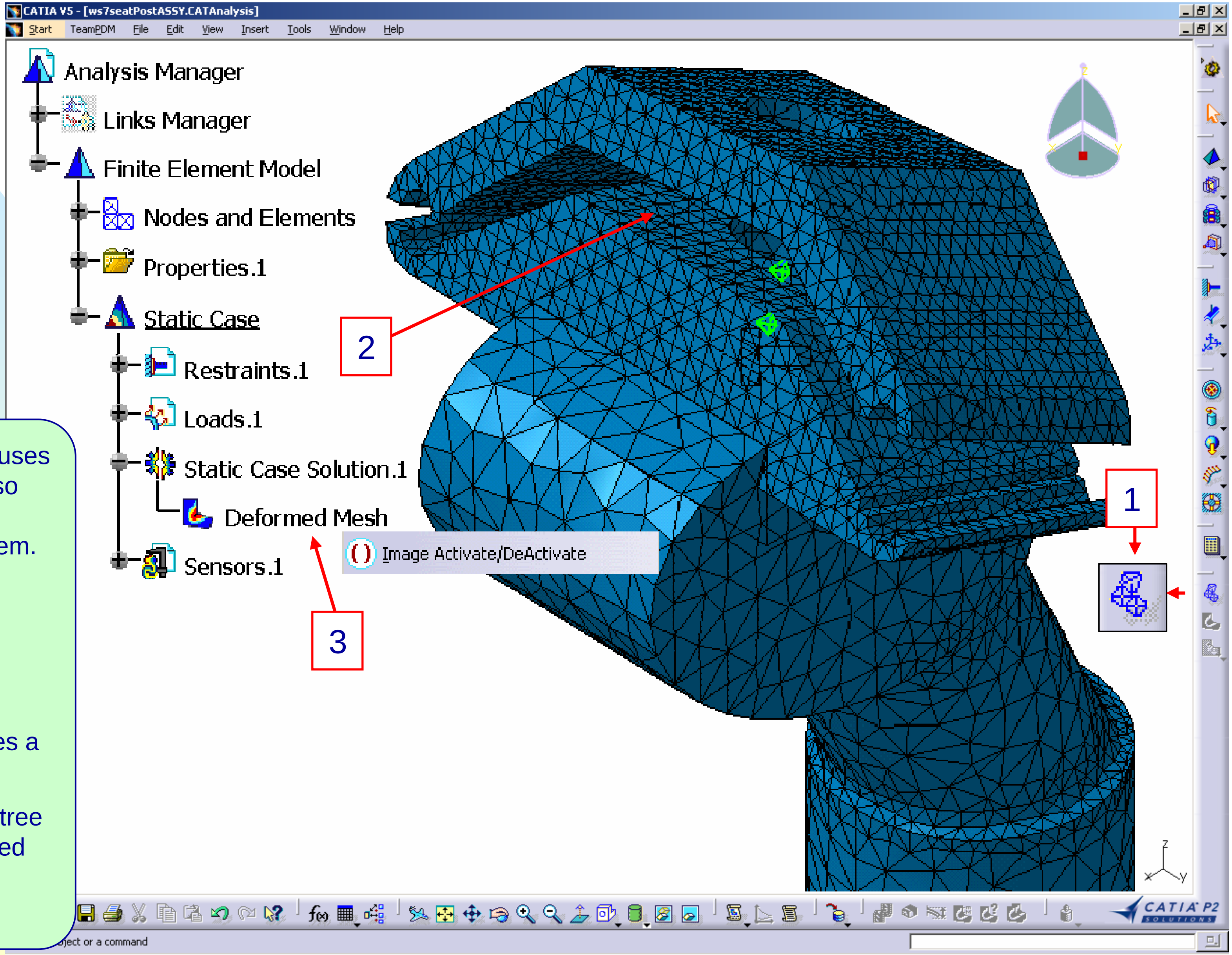
2. Select the compute icon.

3. Compute All, select OK.

4. Note the resources estimation, select Yes.

5. Singularity detected, select OK.

Step 9. Compute the analysis



Visualize what causes the singularities so you can properly restrain your system.

Steps:

1. Select the Deformation icon.

2. Note the upper clamp part requires a restraint.

3. In the features tree right click Deformed Mesh object then select Inactivate.

MSC SOFTWARE SIMULATING REALITY

Step 9. Compute the analysis

Define the second Fastener Connection making our model statically determinate.

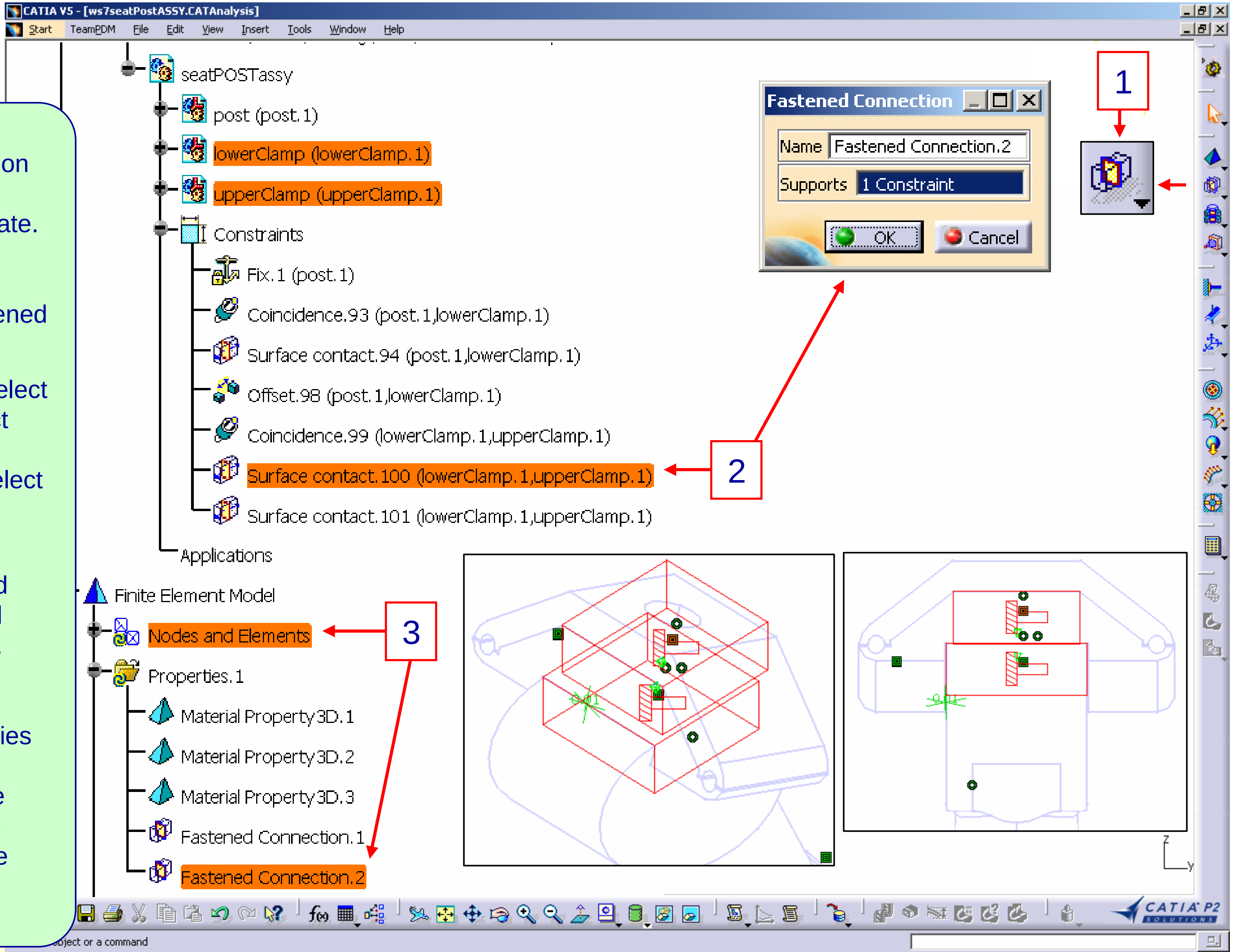
Steps:

1. Select the Fastened Connection icon.

2. From the tree select the surface contact (lower clamp.1 to upper clamp.1), select OK.

3. Note the mesh connection created between parts and contact properties.

Fastened part bodies are fastened to behave as a single body, however the deformability of the interface is considered.



MSC SOFTWARE

Step 9. Compute the analysis

Show the **Deformed Mesh.**

<u>Steps:</u>

1. Select the compute icon.

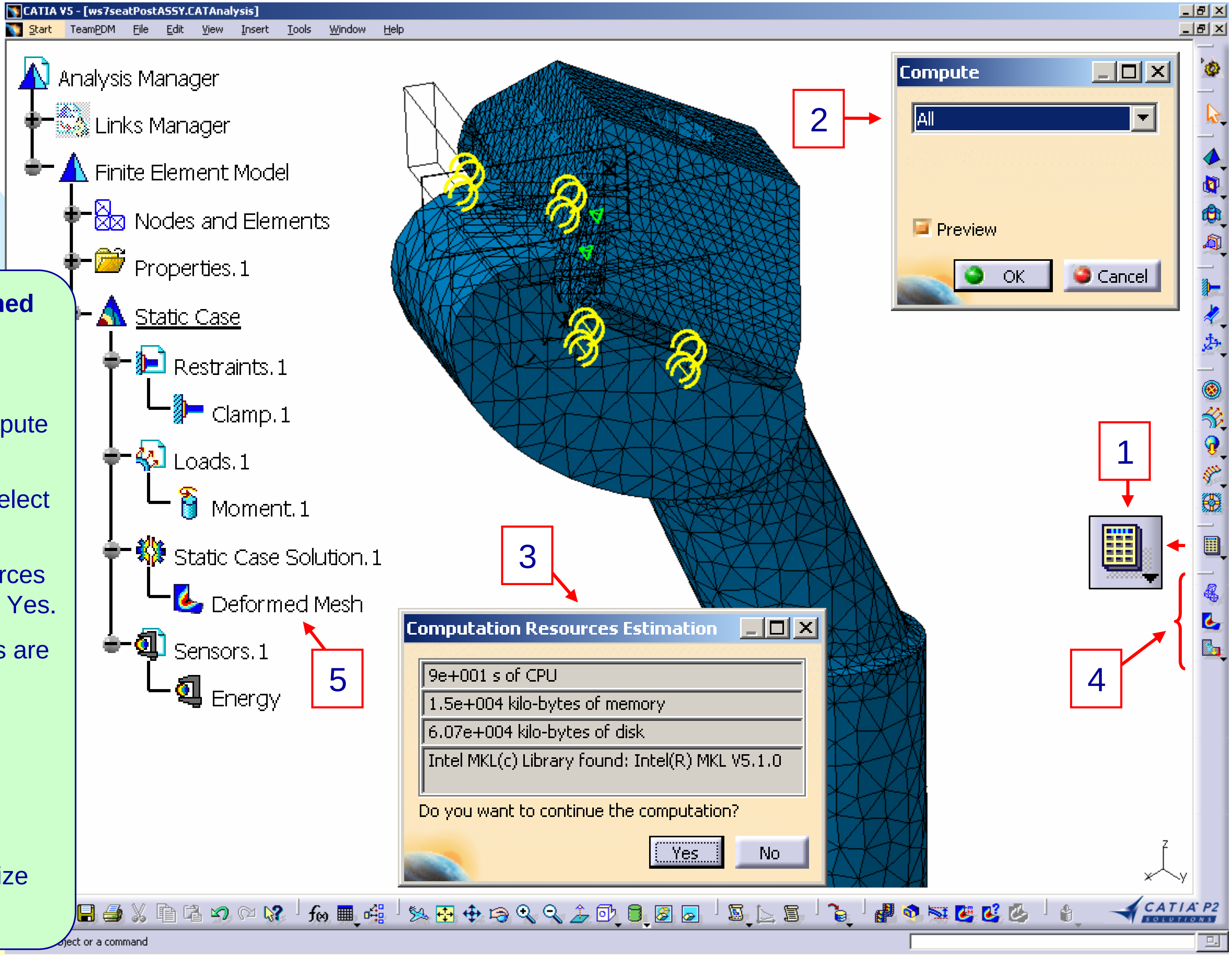
2. Compute All, select OK.

3. Note the resources estimation, select Yes.

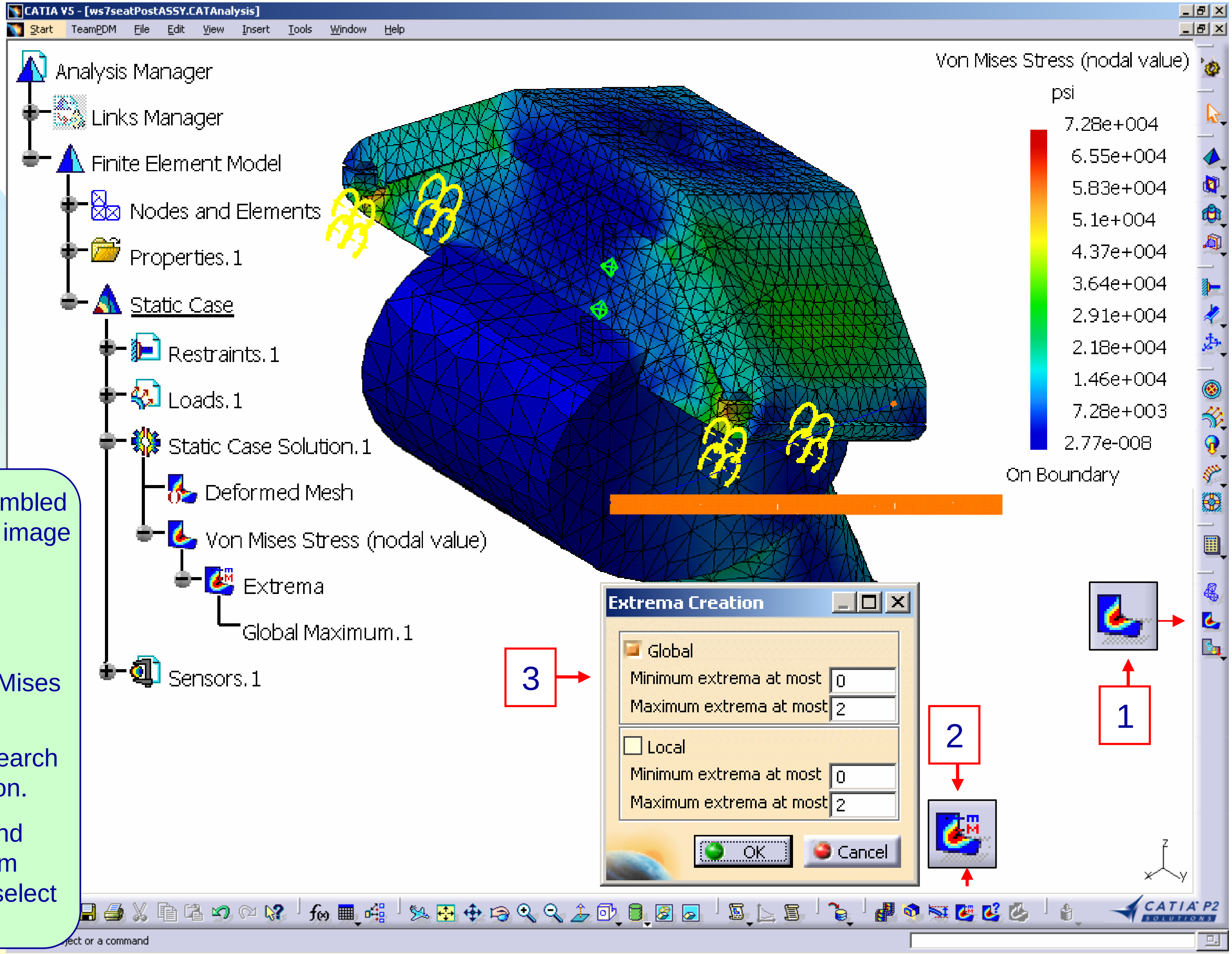
4. Note all images are available.

5. Activate the Deformed Mesh image.

Animate to visualize behavior.



Step 10. Visualize the analysis results



Visualize the assembled **Von Mises** stress image and **Maximum Extremas**.

Steps:

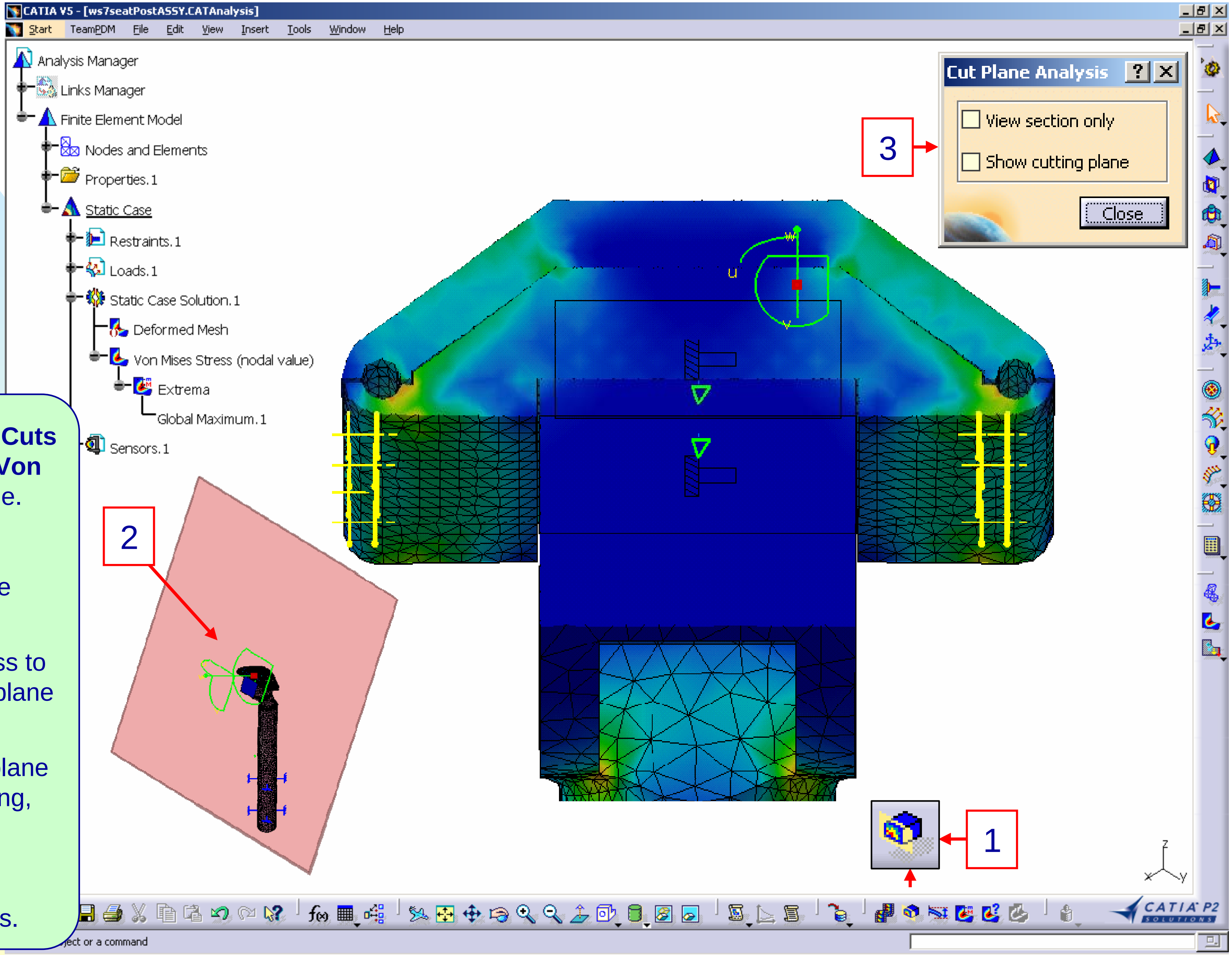
1. Select the Von Mises Icon.

2. Select on the Search Image Extrema icon.

3. Select Global and request 2 maximum extrema at most, select OK.

MSC Software

Step 10. Visualize the analysis results



Visualize **Section Cuts** of the assembled **Von Mises** stress image.

Steps:

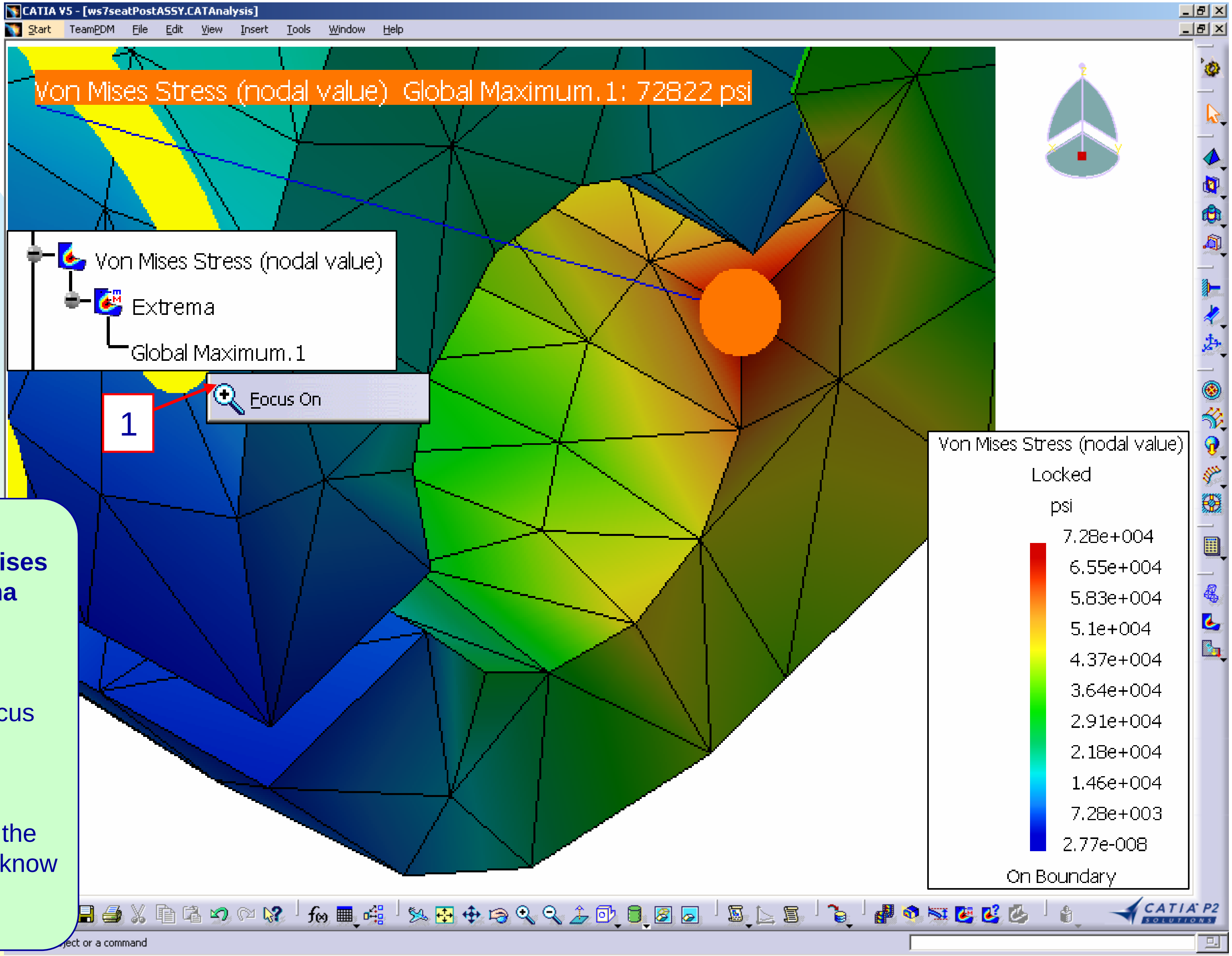
1. Select Cut Plane Analysis Icon.

2. Use the compass to locate the cutting plane as shown.

3. Modify the cut plane options to your liking, select close.

Review other areas.

Step 10. Visualize the analysis results



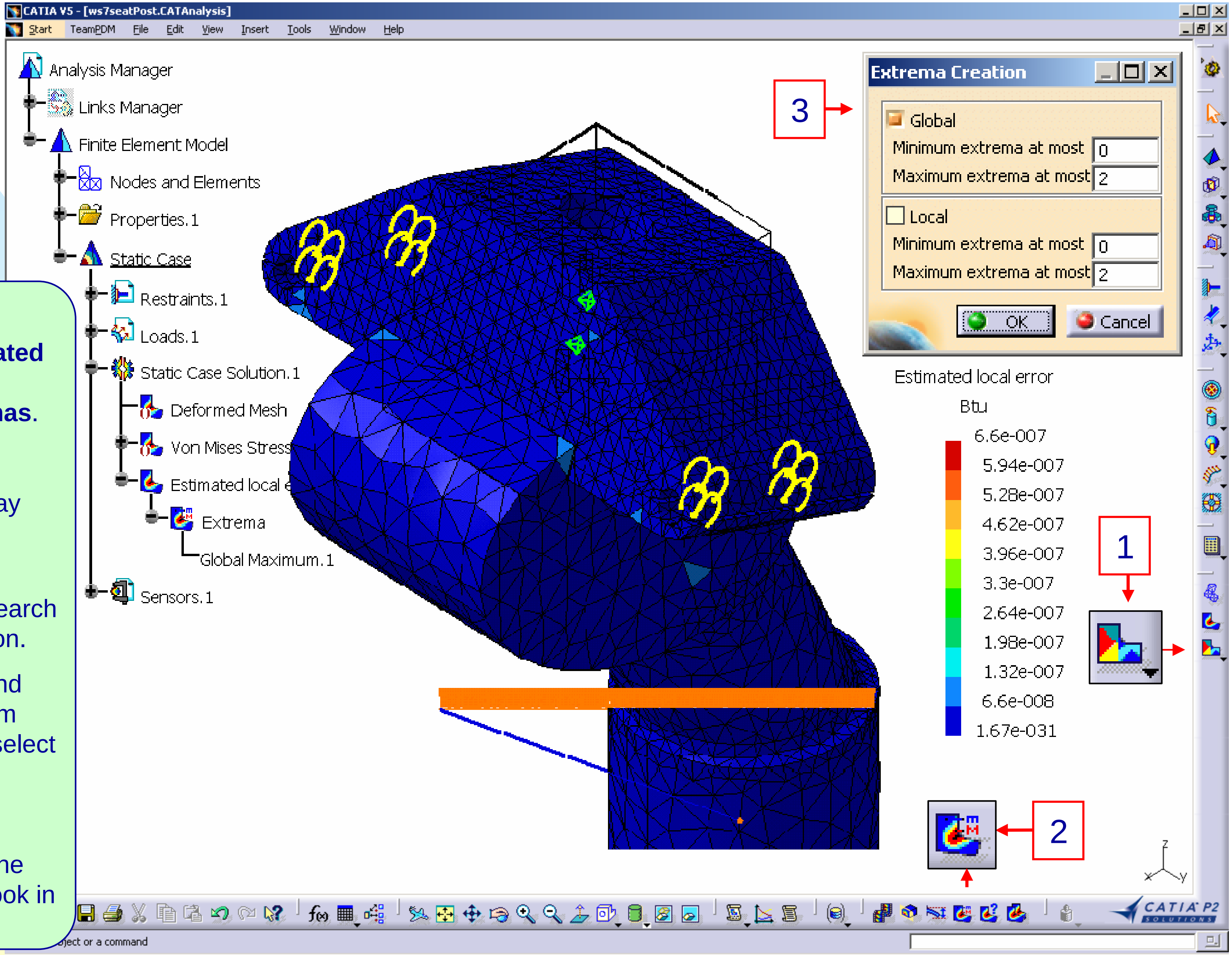
Focus on the assembled **Von Mises Maximum Extrema** stress**.**

Steps

1. Right click + Focus On.

Next we will verify the estimated error to know this is the precise design stress.

Step 10. Visualize the analysis results



Visualize the assembled **Estimated Error** image and **Maximum Extremas**.

Steps:

1. Select the display stress estimated precision Icon.

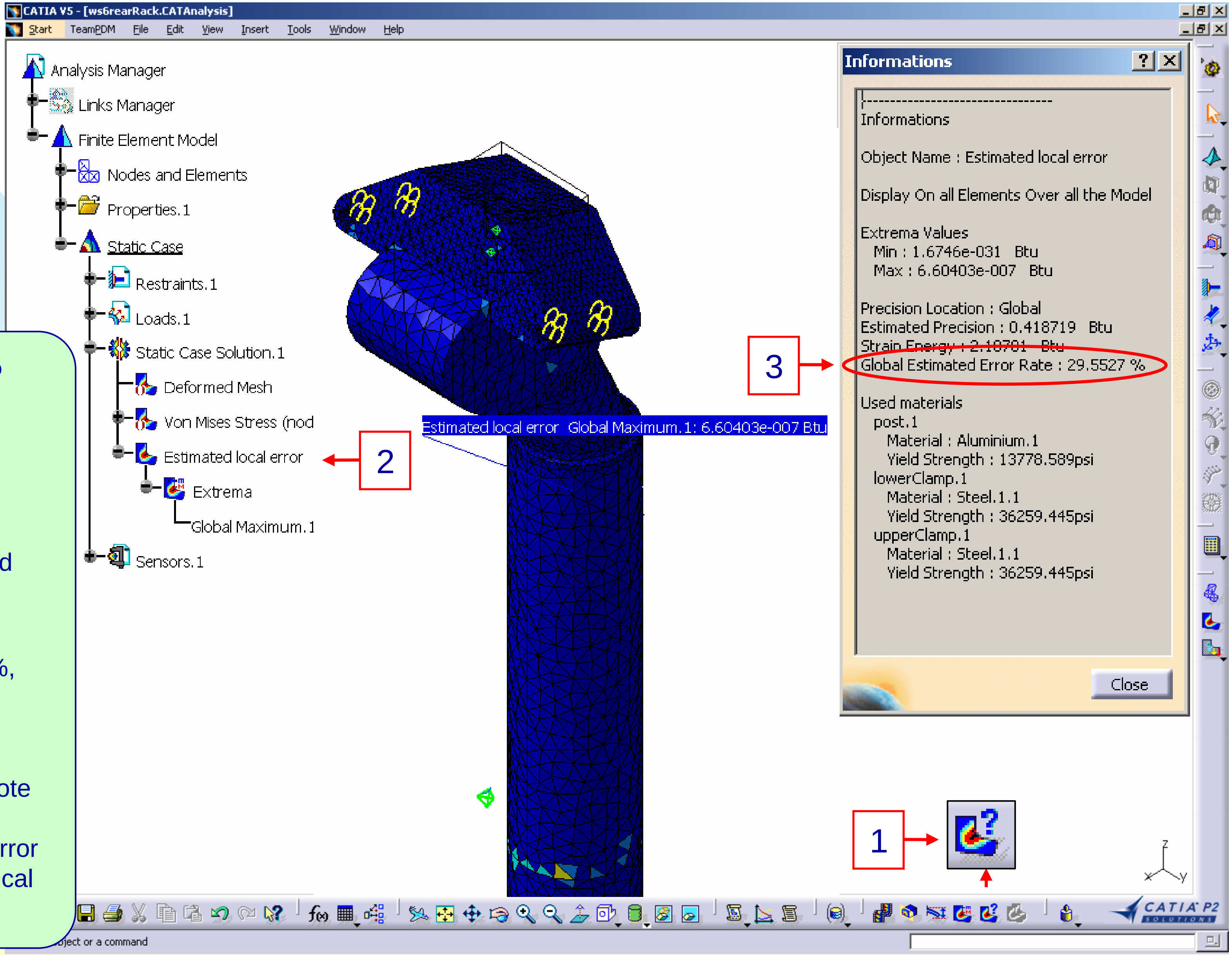
2. Select on the Search Image Extrema icon.

3. Select Global and request 2 maximum extrema at most, select OK.

If you do not see the Extrema symbol look in no show.

MSC SOFTWARE

Step 10. Visualize the analysis results



Find the **Global % Error**.

Steps:

1. Select the Informations icon.

2. Select Estimated local error in the features tree.

3. Note the error %, select close.

It is important to note that the estimated maximum global error is not near the critical clamping area.

Step 10. Visualize the analysis results

Create an **Adaptivity Box** to determine **Local % Estimated Error.**

Steps:

1. Verify that the Estimated Error image is active, and select the Adaptivity box Icon.

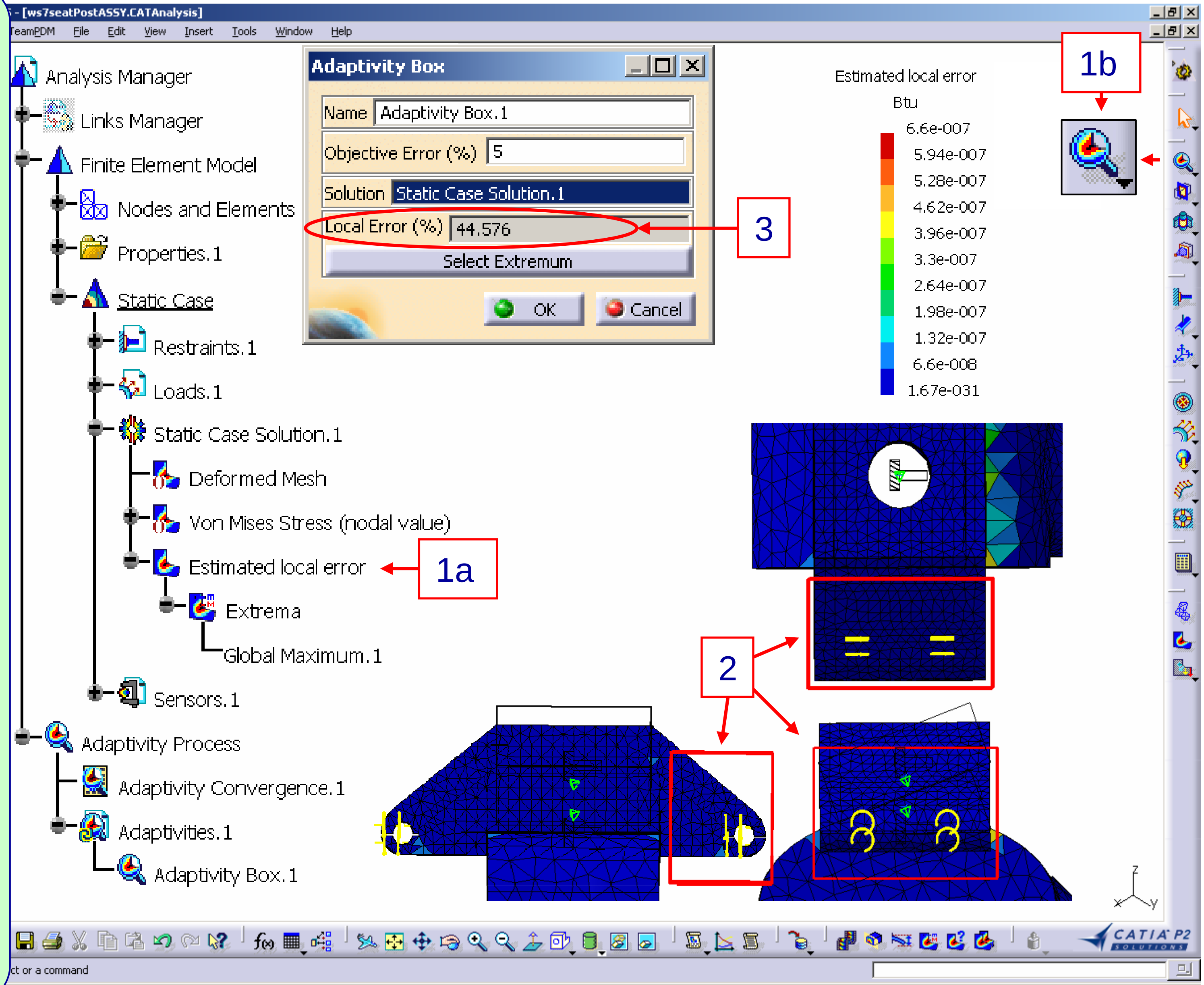
2. Locate and size the adaptivity box as shown.

3. Note the Local Error 45.7%.

Global error of 30% and local error of 45% is unacceptable. Recommend max of 20% and 10% respectively.

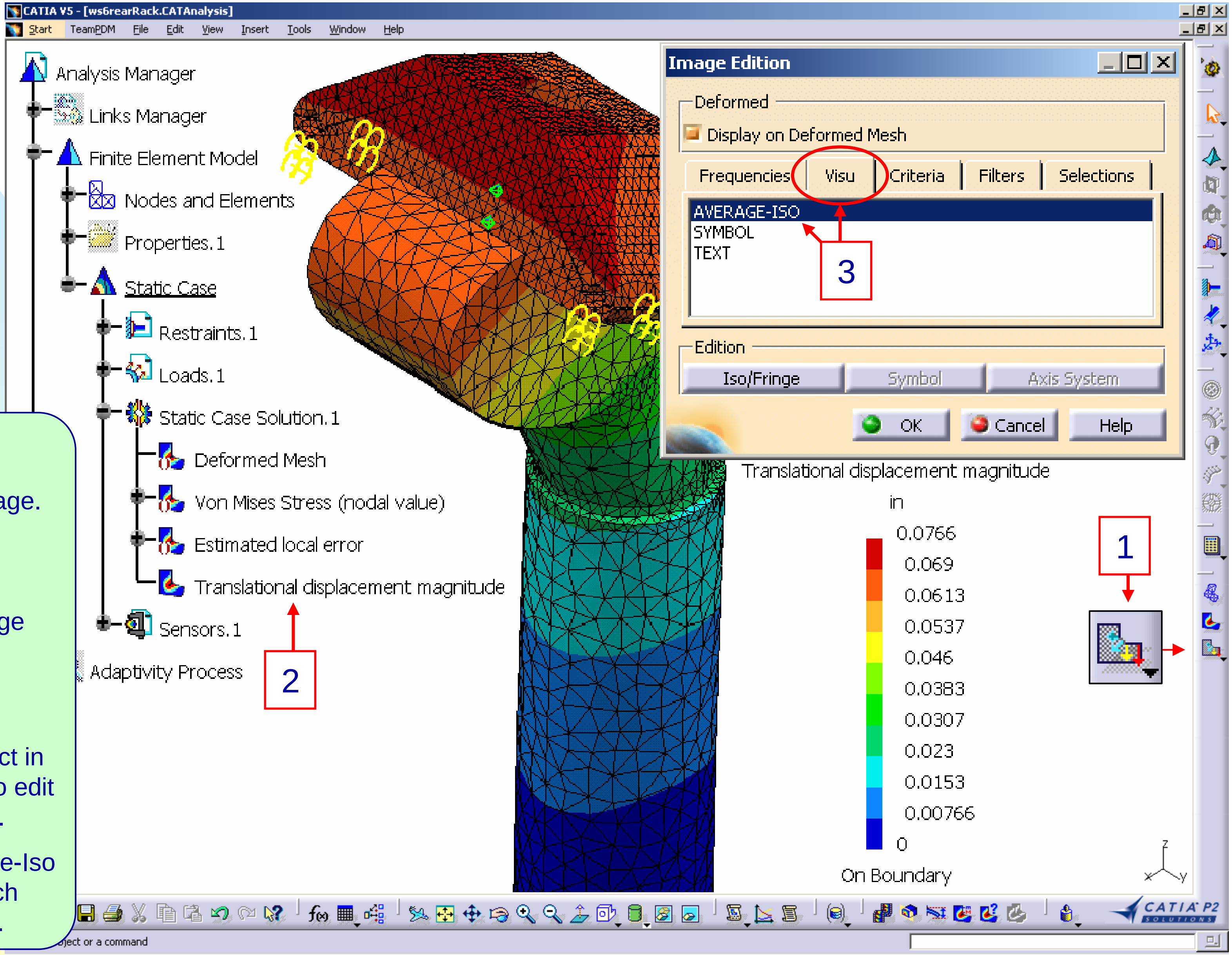
Changing to a Parabolic element type mesh will yield the precise values, but computational time is approx. 1 hour.

In the interest of time, stay with linear elements for this class.



MSC SOFTWARE

Step 10. Visualize the analysis results



Visualize the assembled **Displacement** image.

Steps:

1. Select the Displacement Image Icon.

2. Double click Translational displacement object in the features tree to edit image parameters.

3. Click on Average-Iso in Visu tab to switch display, select OK.

Step 10. Visualize the analysis results

Visualize the **Von Mises** stress for the post.

Steps:

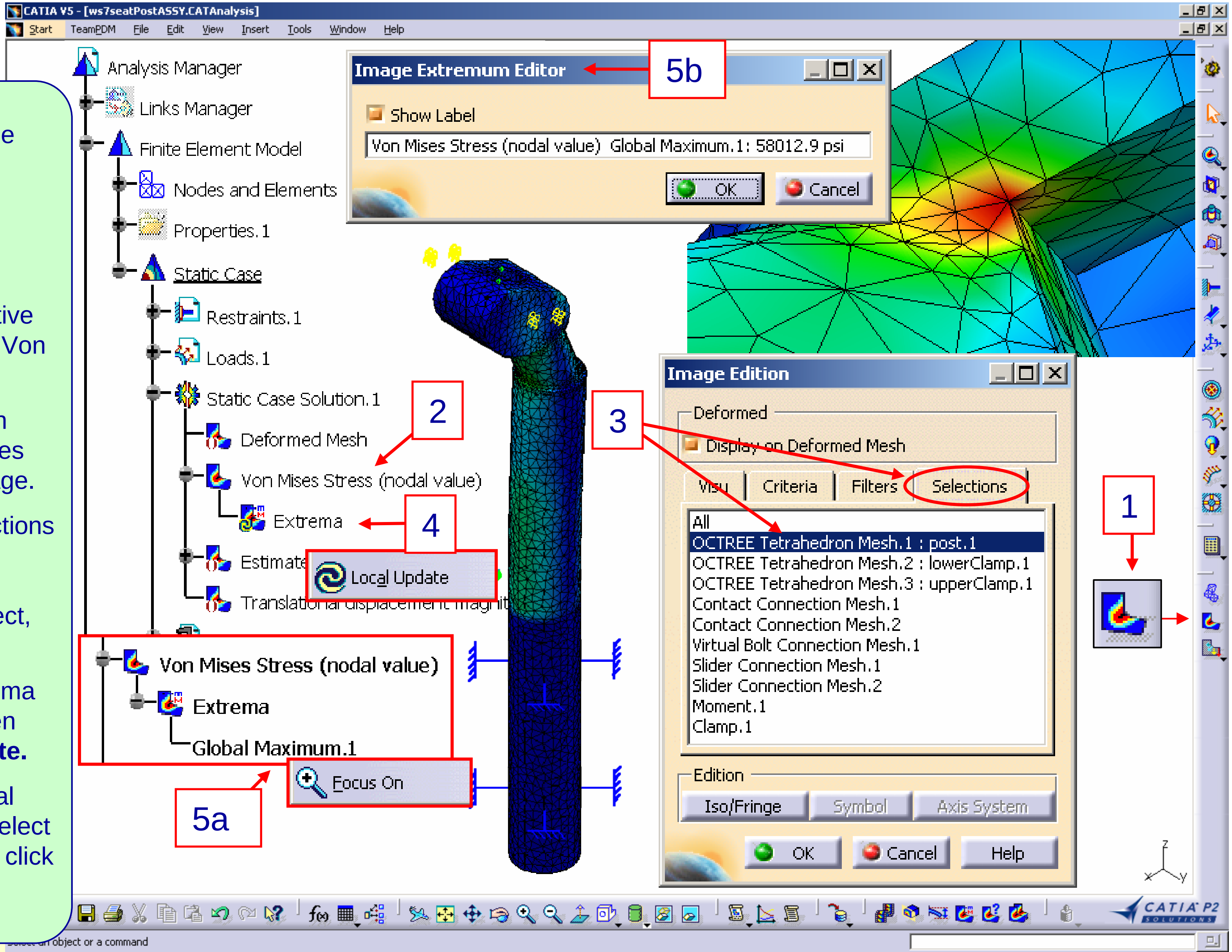
1. Select the Von Mises icon (this deactivates the active image and actives Von Mises).

2. Double click Von Mises in the features tree to edit the image.

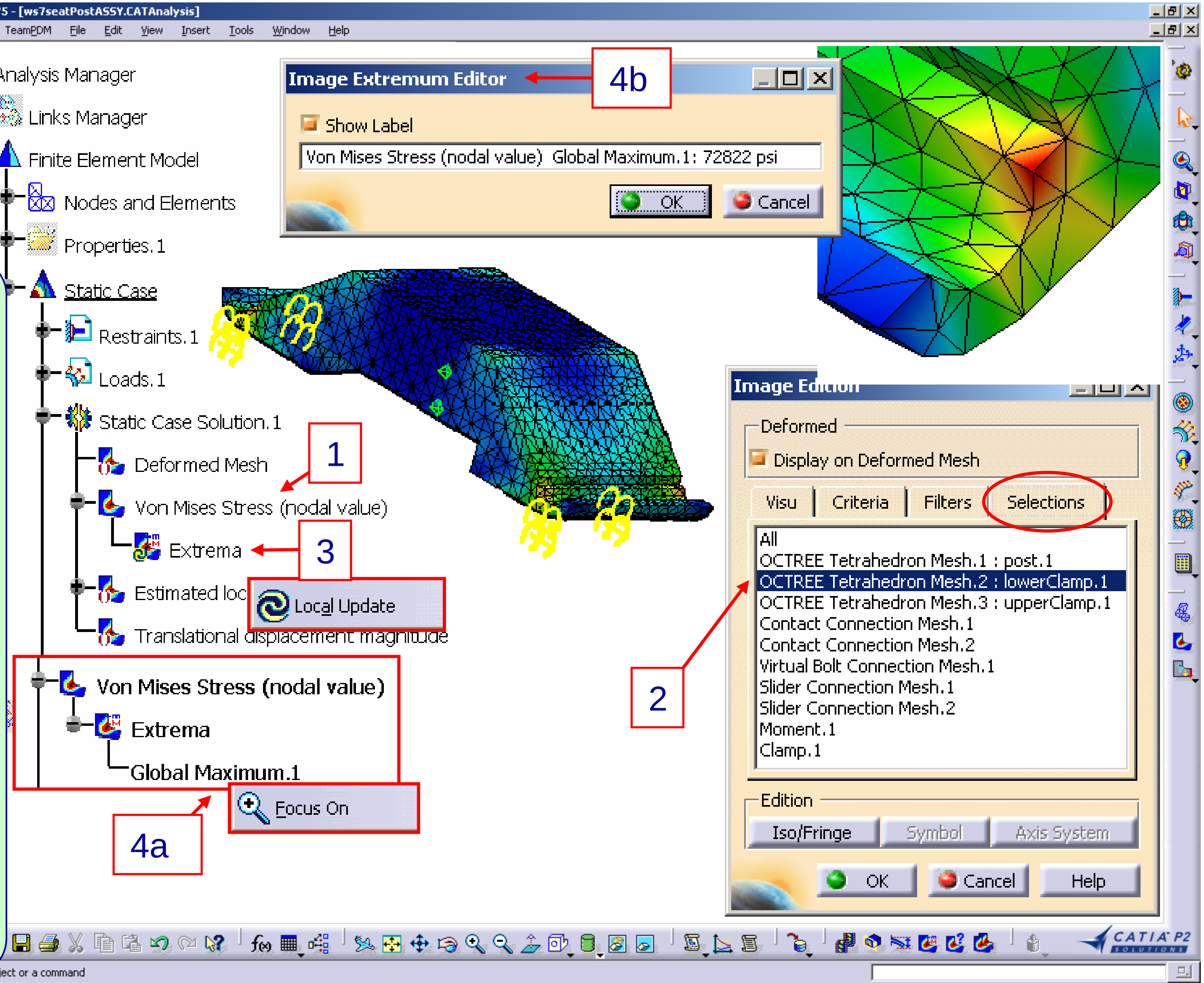
3. Select the Selections tab then OCTREE Tetrahedron Mesh.1:post.1 object, select OK.

4. Right click Extrema in features tree then select **Local update.**

5. Right click Global Maximum.1 then select **Focus on**. Double click to see the image extremum **editor**.



Step 10. Visualize the analysis results



Visualize the **Von Mises** stress for the lower Clamp.

Steps:

1. Double click Von Mises in the features tree to edit the image.

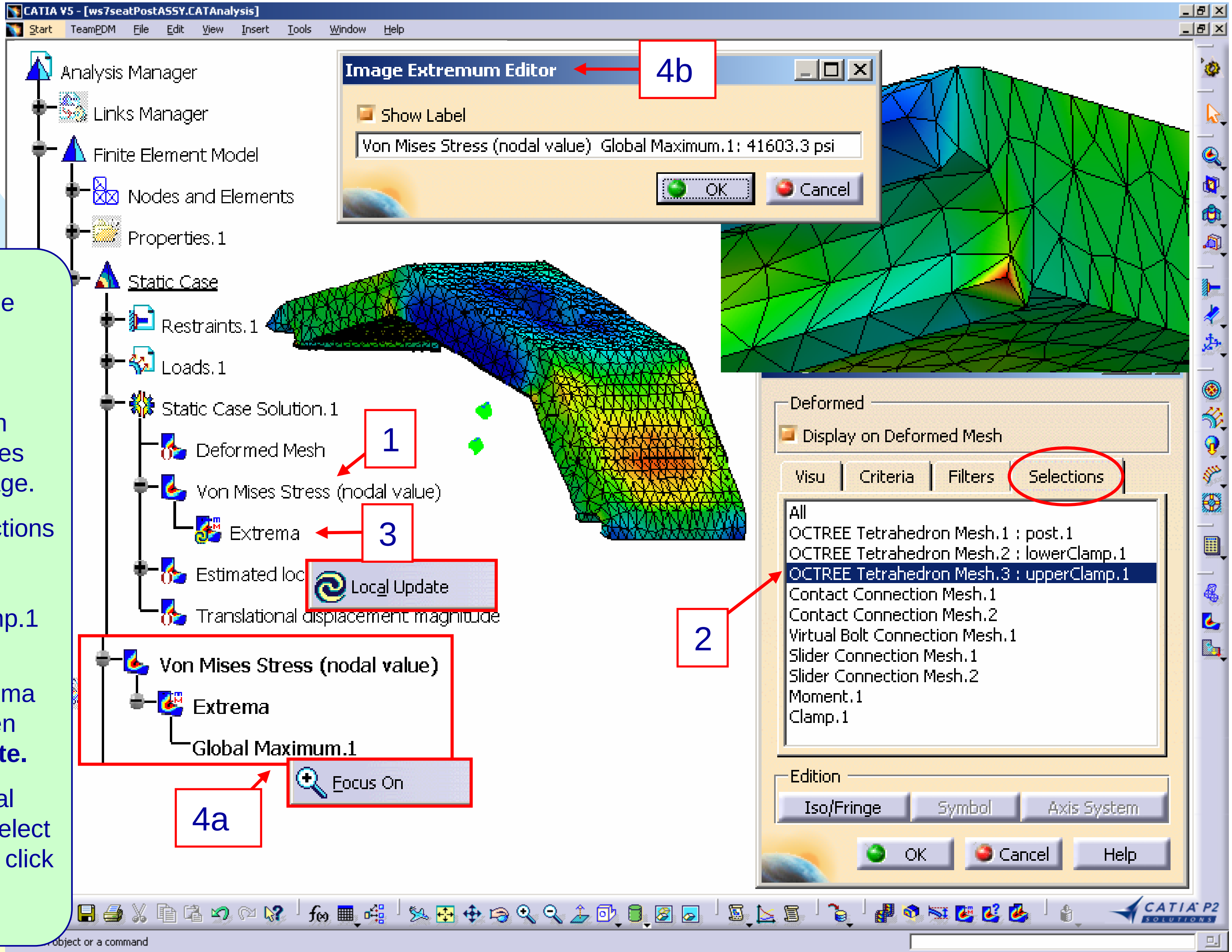
2. Select the Selections tab then OCTREE Tetrahedron Mesh.2:lowerClamp.1 object, select OK.

3. Right click Extrema in features tree then select **Local update.**

4. Right click Global Maximum.1 then select **Focus on**. Double click to see the image extremum **editor**.

MSC SOFTWARE

Step 10. Visualize the analysis results



Visualize the **Von Mises** stress for the upper Clamp.

Steps:

1. Double click Von Mises in the features tree to edit the image.

2. Select the Selections tab then OCTREE Tetrahedron Mesh.3:upperClamp.1 object, select OK.

3. Right click Extrema in features tree then select **Local update.**

4. Right click Global Maximum.1 then select **Focus on**. Double click to see the image extremum **editor**.

Step 10. Visualize the analysis results

Visualize the **Von Mises** stress localized at the clamp.

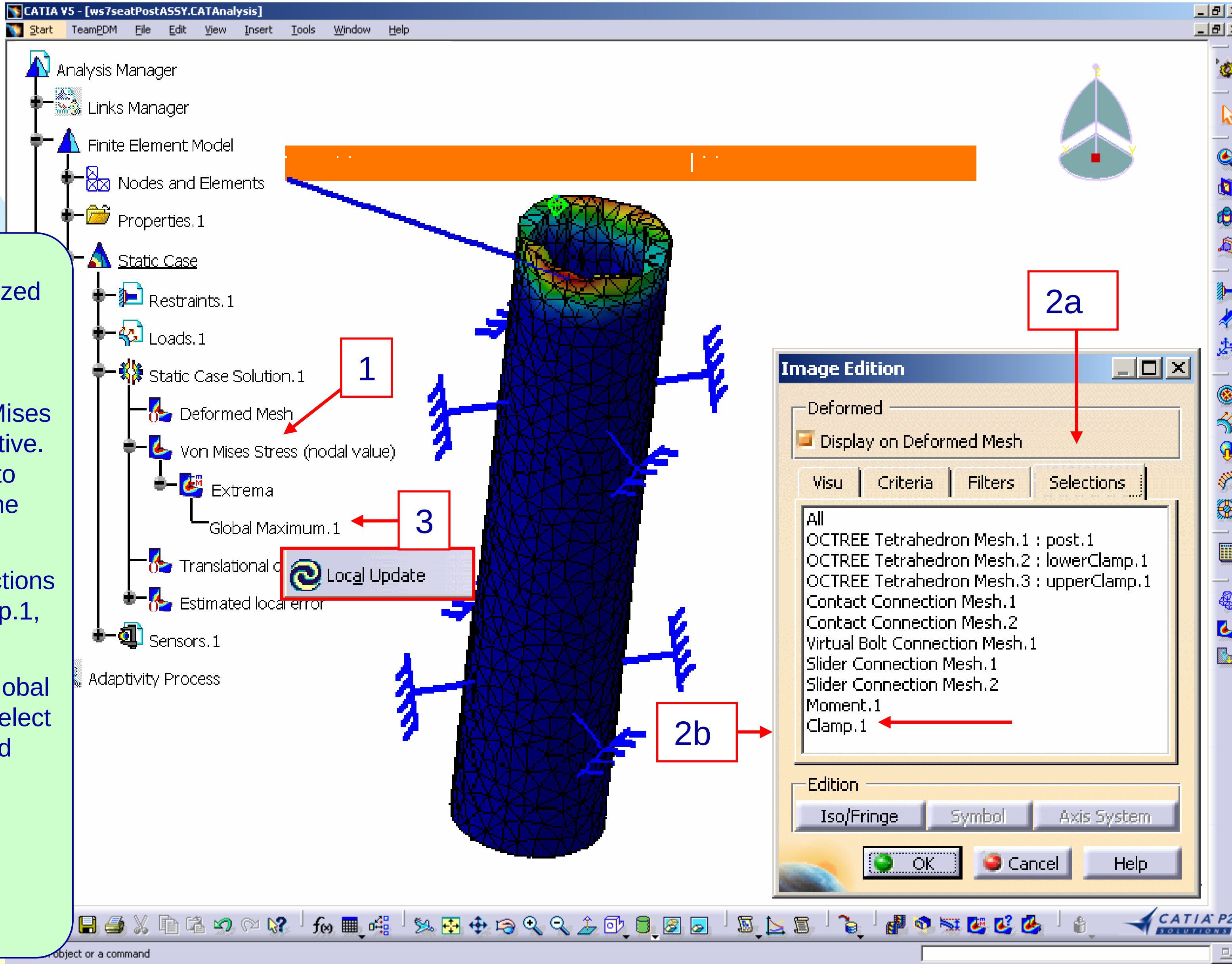
Steps:

1. Verify the Von Mises Stress image is active. Then double click to edit the image in the features tree.

2. Select the Selections tab, then the Clamp.1, select OK.

3. Right click on Global Maximum.1 then select Local update to find max stress on this selection.

Also examine the Moment.1.



MSC SOFTWARE

Step 10. Visualize the analysis results

Find the **Reactions** in the clamped area.

Steps:

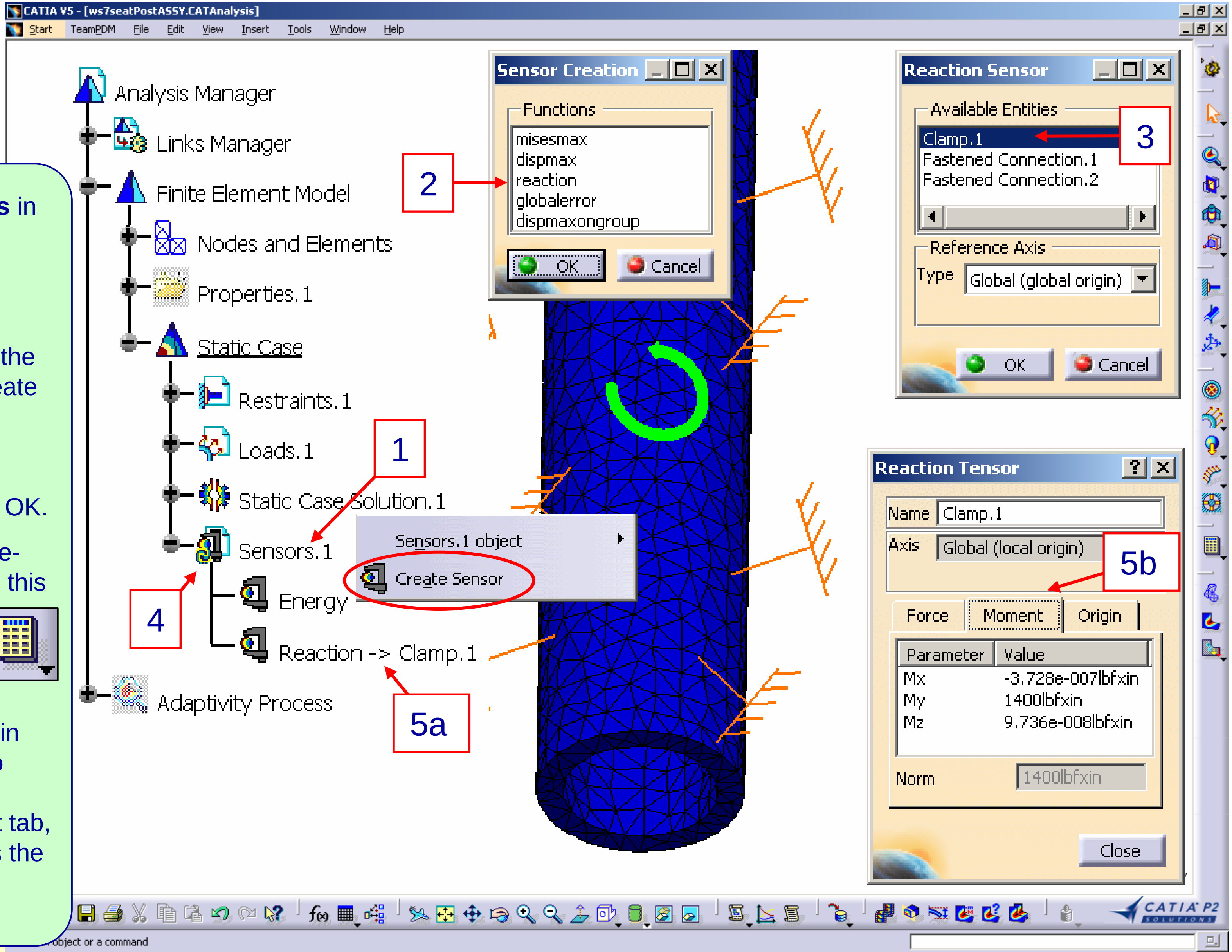
1. Right click the Sensor.1 object in the features tree + Create Sensor.

2. Select reaction.

3. Select Clamp.1, OK.

4. Note you must re-compute to update this sensor (it's fast).

5. Double click Reaction-Clamp.1 in the features tree to review the values. Select the Moment tab, verify that it equals the induced moment, select close.



Step 10. Visualize the analysis results

Find the **Bolt Loads.**

Steps:

1. Right click the Sensor.1 object in the features tree then select Create Sensor.

2. Select reaction.

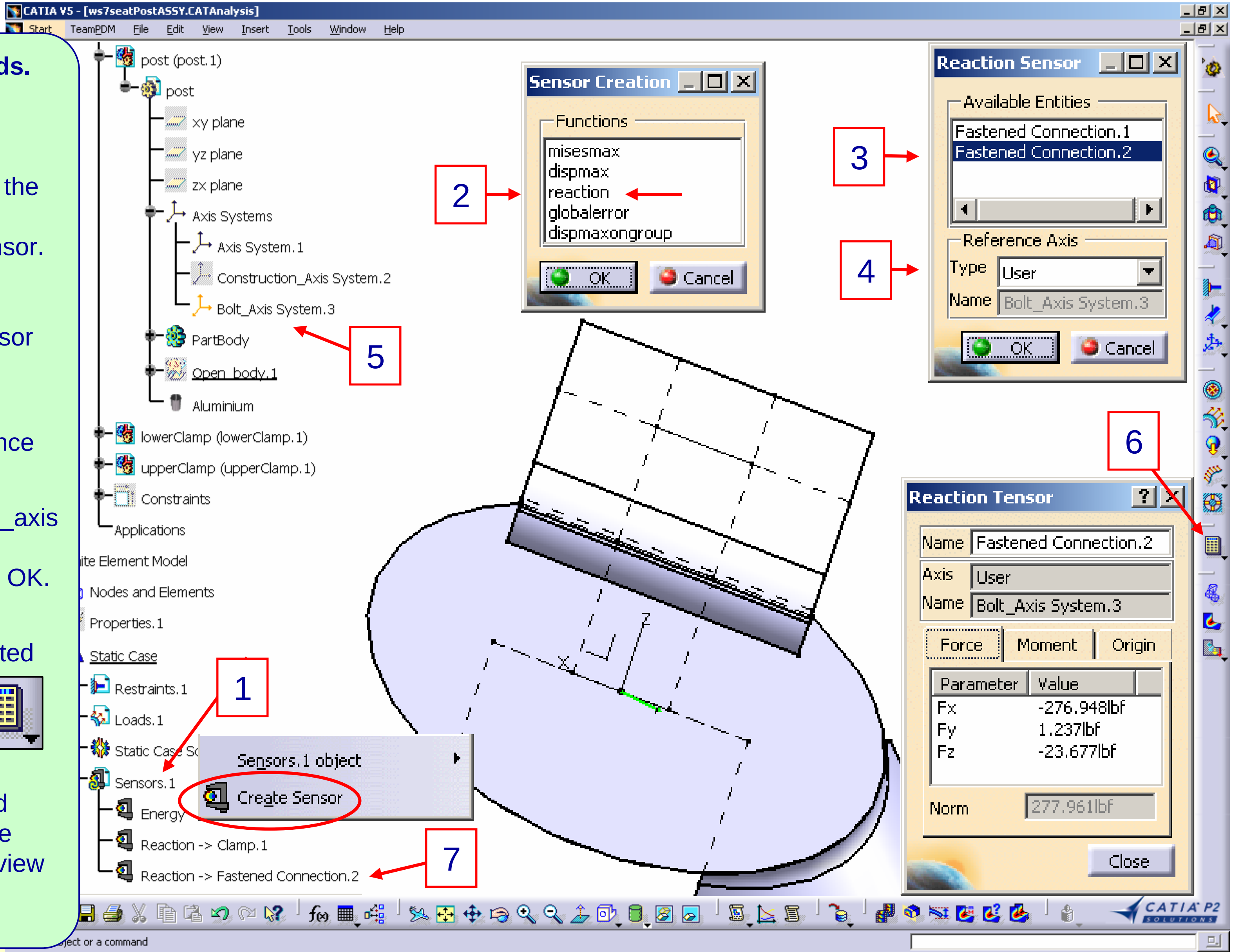
3. In Reaction sensor select Fastened Connection.2.

4. Change Reference Axis Type to User.

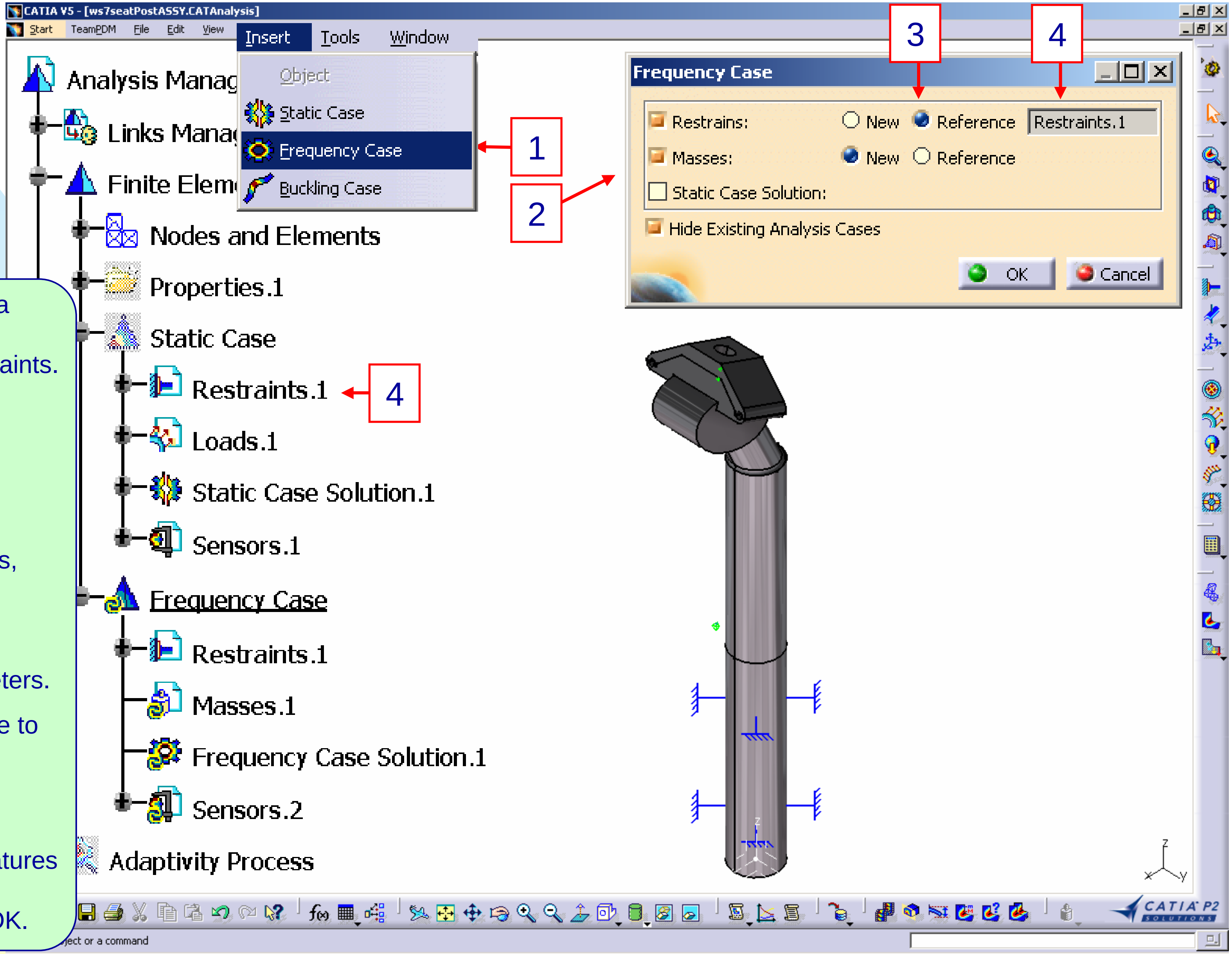
5. Select user Bolt_axis System.3 in the features tree, click OK.

6. Re-compute to update newly created sensors.

7. Double click Reaction-Fastened Connection.2 in the features tree to review the Forces and Moments.



Step 11. Frequency (Modal) analysis for the assembly



Start by inserting a new frequency analysis with restraints.

Steps:

1. From the Insert menu select Frequency Case.

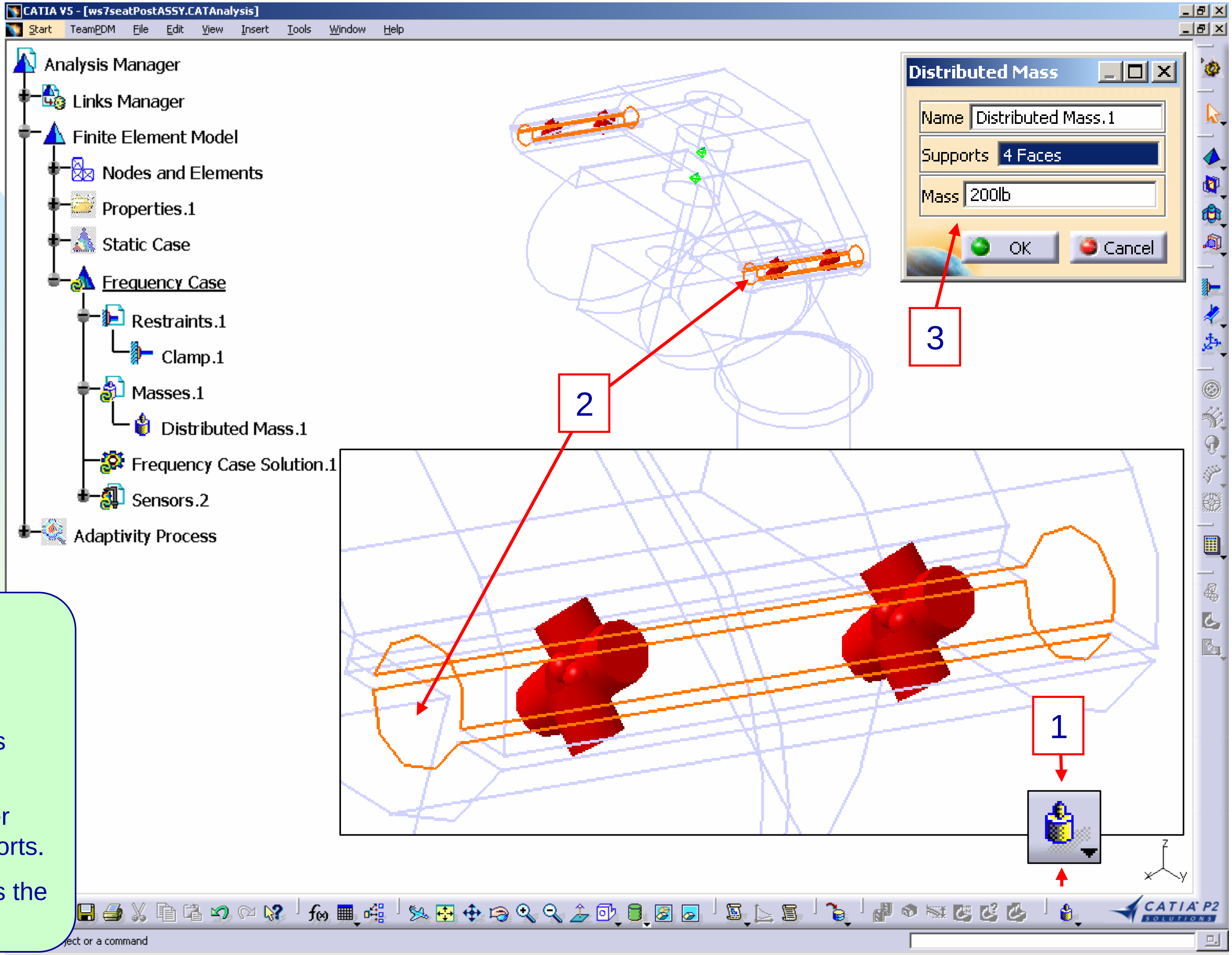
2. Select Restrains, Masses and Hide existing Analysis Cases to define frequency parameters.

3. Select reference to use static case restraints.

4. Select static restraints from features tree to specify reference select OK.

MSC SOFTWARE

Step 11. Frequency (Modal) analysis for the assembly



Create mass equipment.

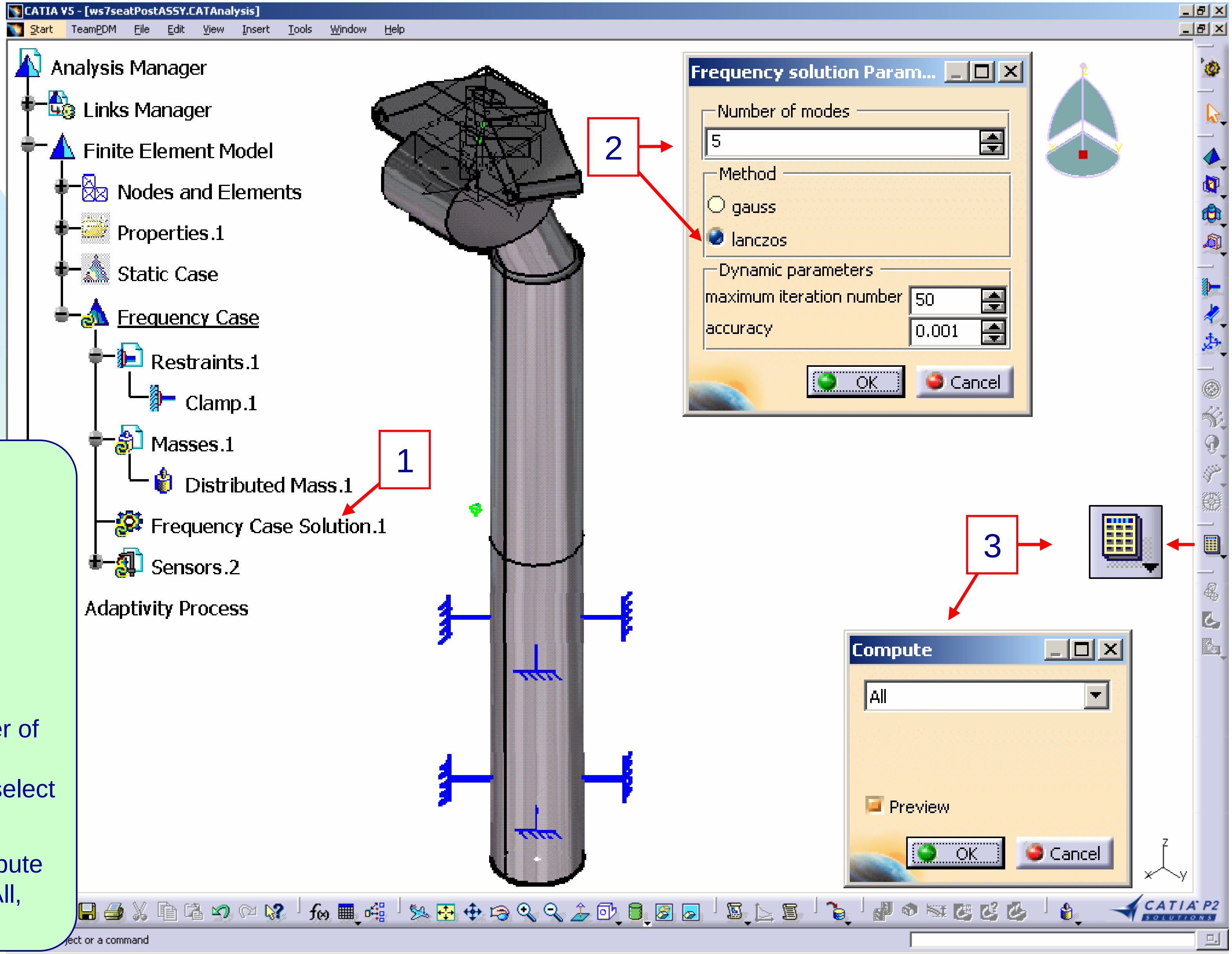
Steps:

1. Select the mass icon.

2. Select four inner faces as the supports.

3. Key in 200 lb as the mass, select OK.

Step 11. Frequency (Modal) analysis for the assembly



Specify solution parameters.

Steps:

1. Double click Frequency Case Solution.1 in the features tree.

2. Key in 5 number of modes. Select the lanczos method, select OK

3. Select the compute icon and specify All, select OK.

Step 11. Frequency (Modal) analysis for the assembly

Visualize the results.

Steps:

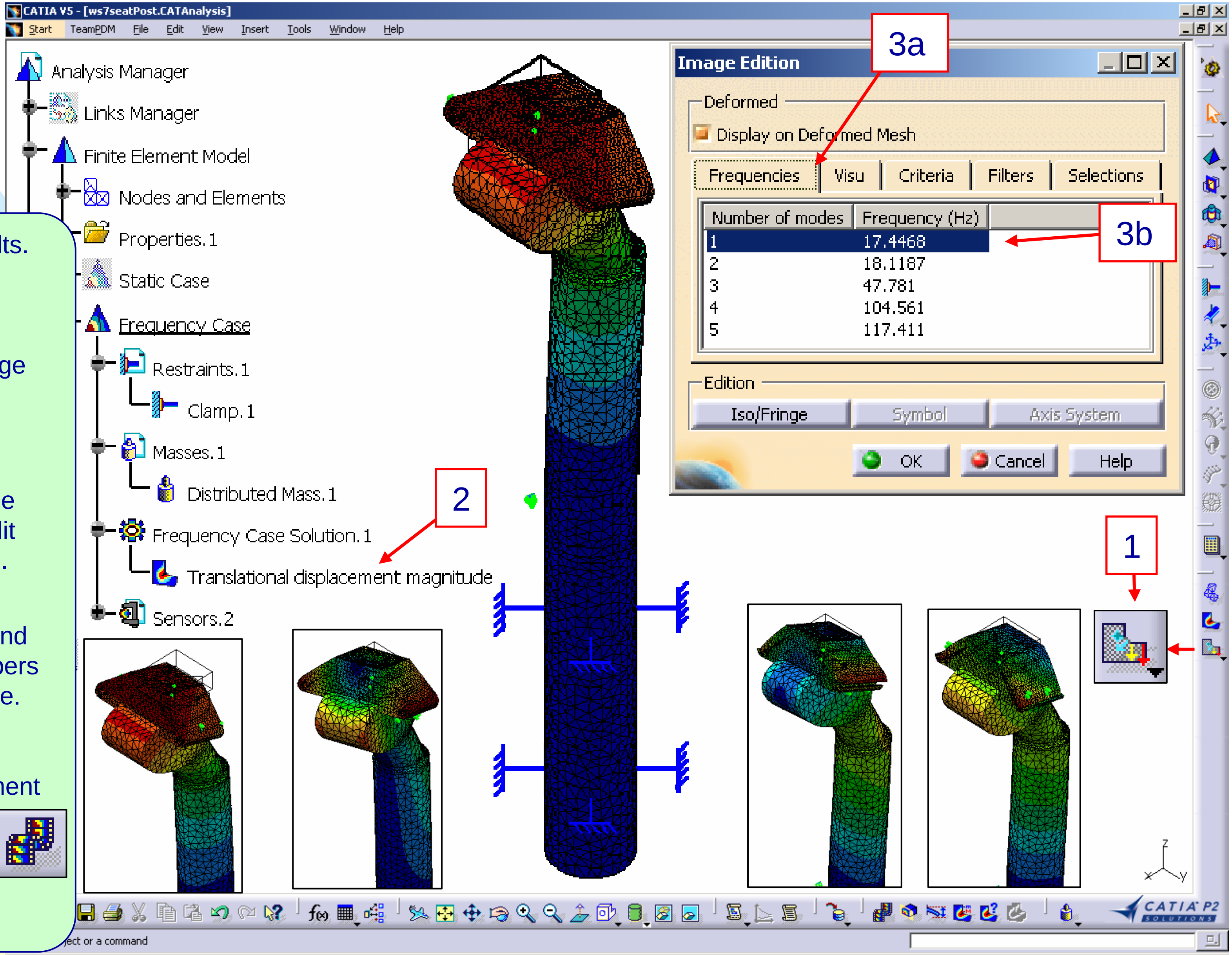
1. Select the Displacement image icon.

2. Double click Translational displacement in the features tree to edit image parameters.

3. Click the Frequencies tab and select mode numbers to view each image.

To view displacement at any calculated mode, animate.

Repeat with other modes.

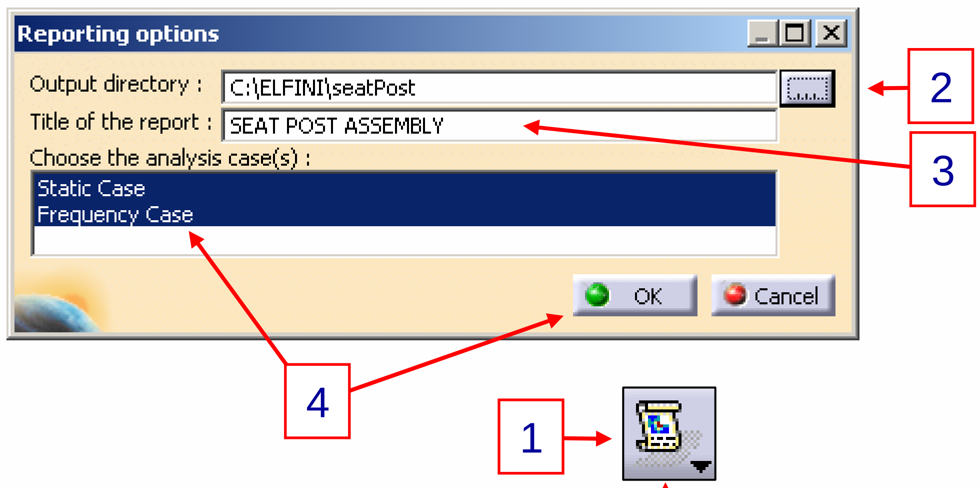


Step 12. Generate a report

Conclusions:

Maximum stress exceeds material yield. Select new material with yield values that exceed the analyzed Von Mises extrema using the parabolic element results.

	Von Mises extrema Linear elements, 44.5% precision	Von Mises extrema Parabolic elements, 9.8% precision
Post	58.0 ksi	98.2 ksi
Lower Clamp	72.8 ksi	144 ksi
Upper Clamp	41.6 ksi	101 ksi



After activating each image at least once, generate a report.

Steps:

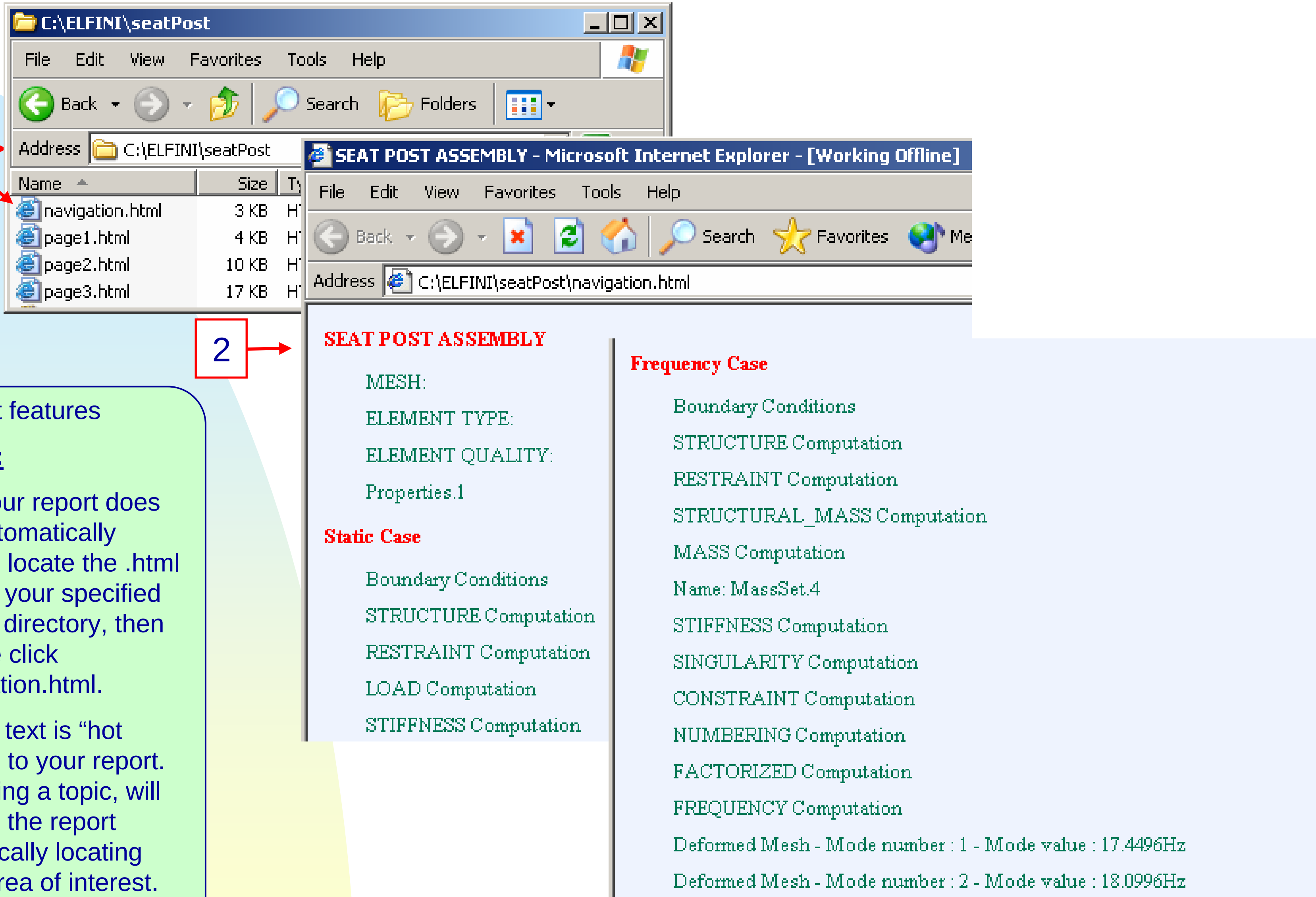
1. Select the Basic Analysis Report icon.

2. Select an Output directory.

3. Key in Title of the report.

4. Choose both analysis cases, select OK.

Step 12. Generate a report



Report features

Steps:

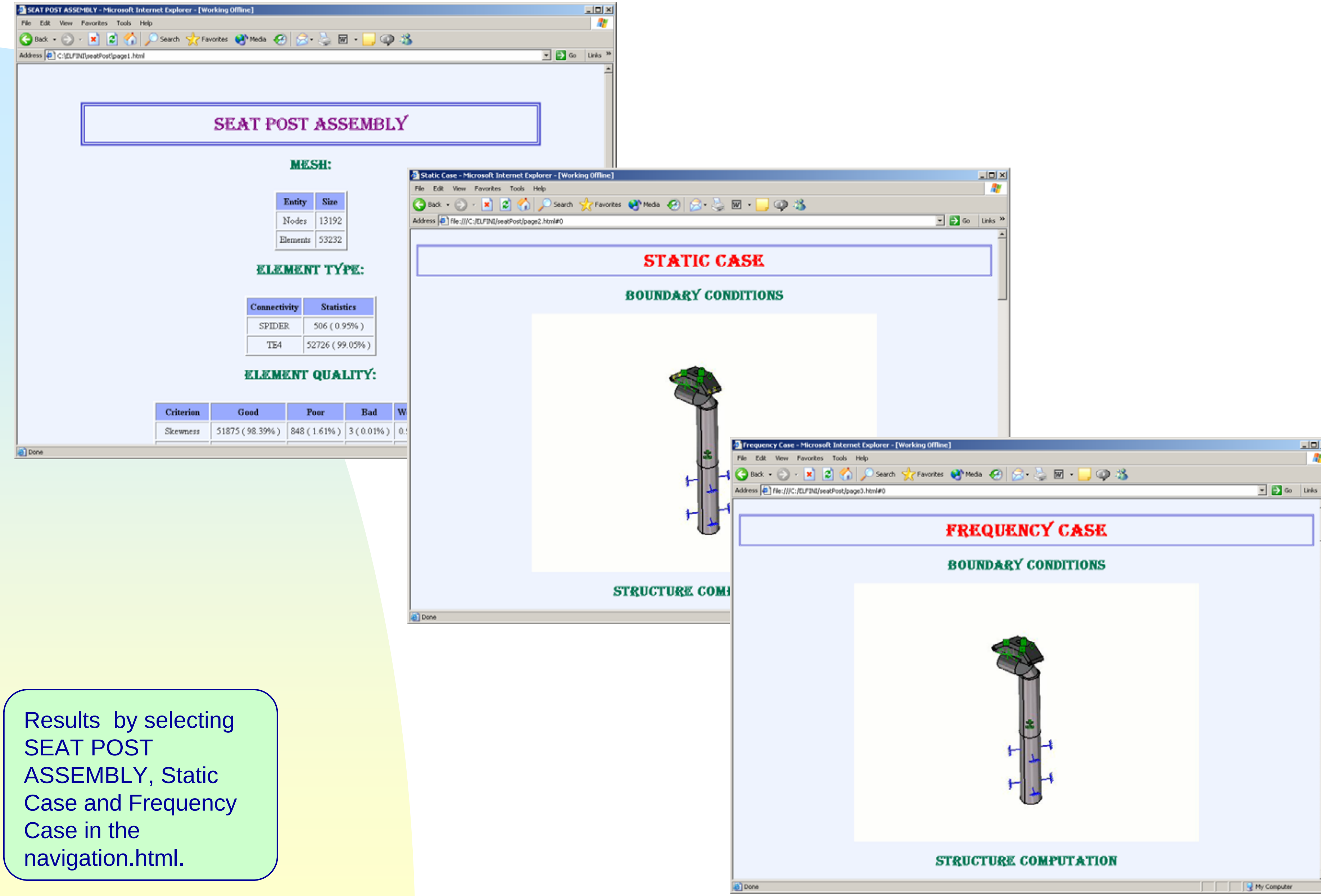
1. If your report does not automatically launch locate the .html files in your specified output directory, then double click navigation.html.

2. The text is “hot linked” to your report. Selecting a topic, will launch the report specifically locating your area of interest.

3. It may launch minimized.

MSC SOFTWARE

Step 12. Generate a report



Results by selecting SEAT POST ASSEMBLY, Static Case and Frequency Case in the navigation.html.

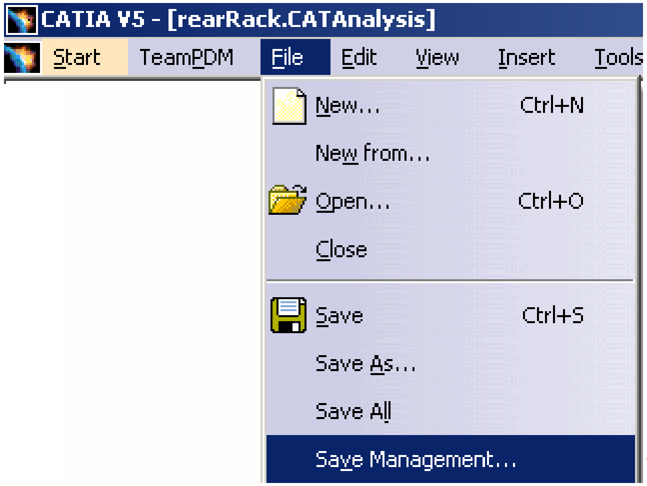
Step 13. Save the analysis document

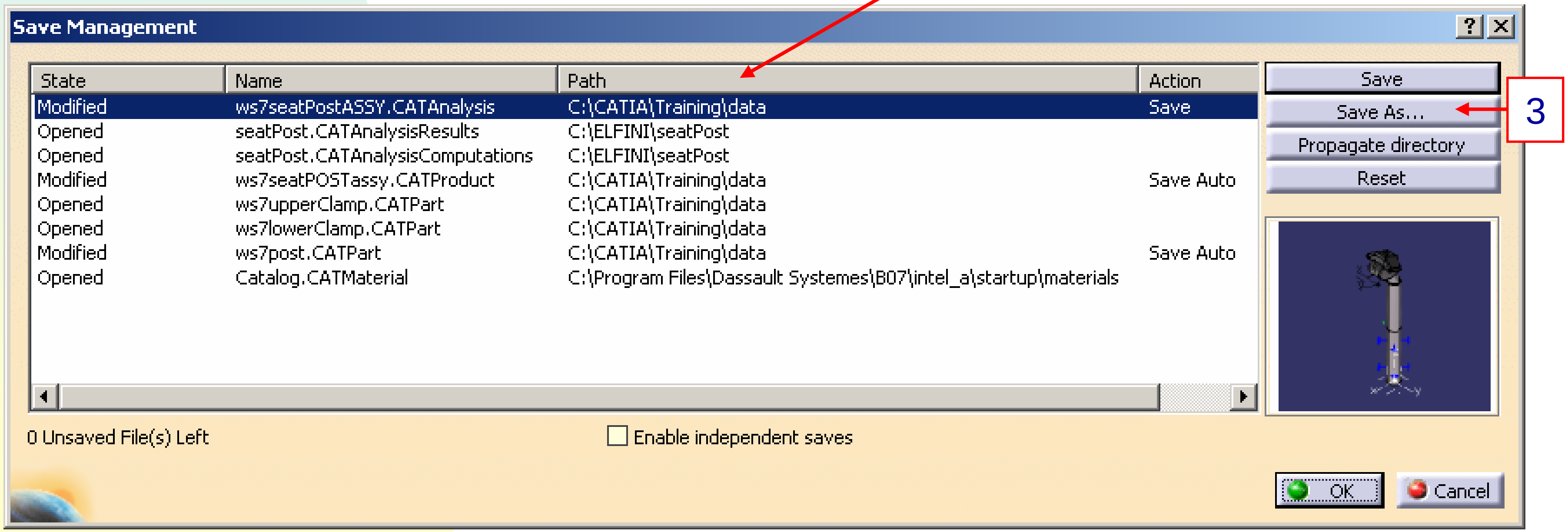
Steps:

1. From File menu select Save Management.

2. Highlight document.

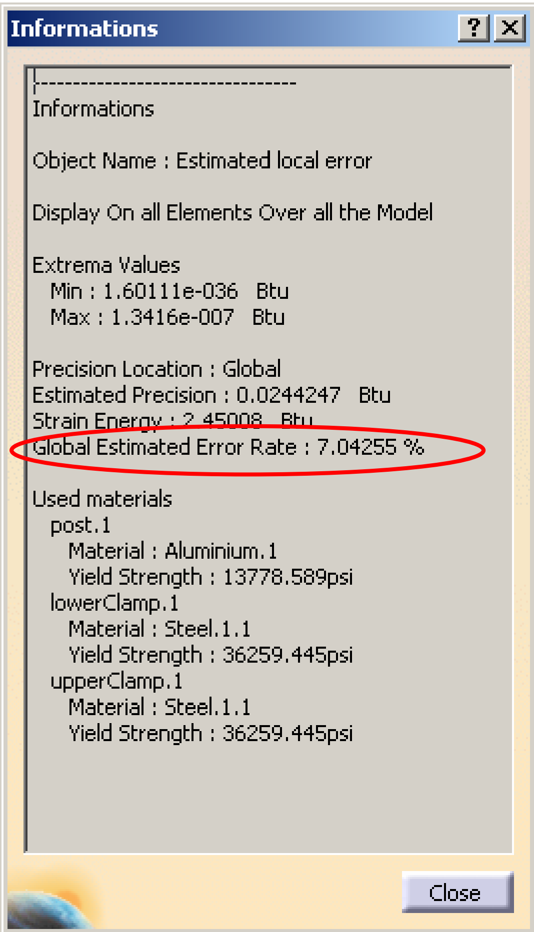
3. Click Save As to specify name and path, select OK.



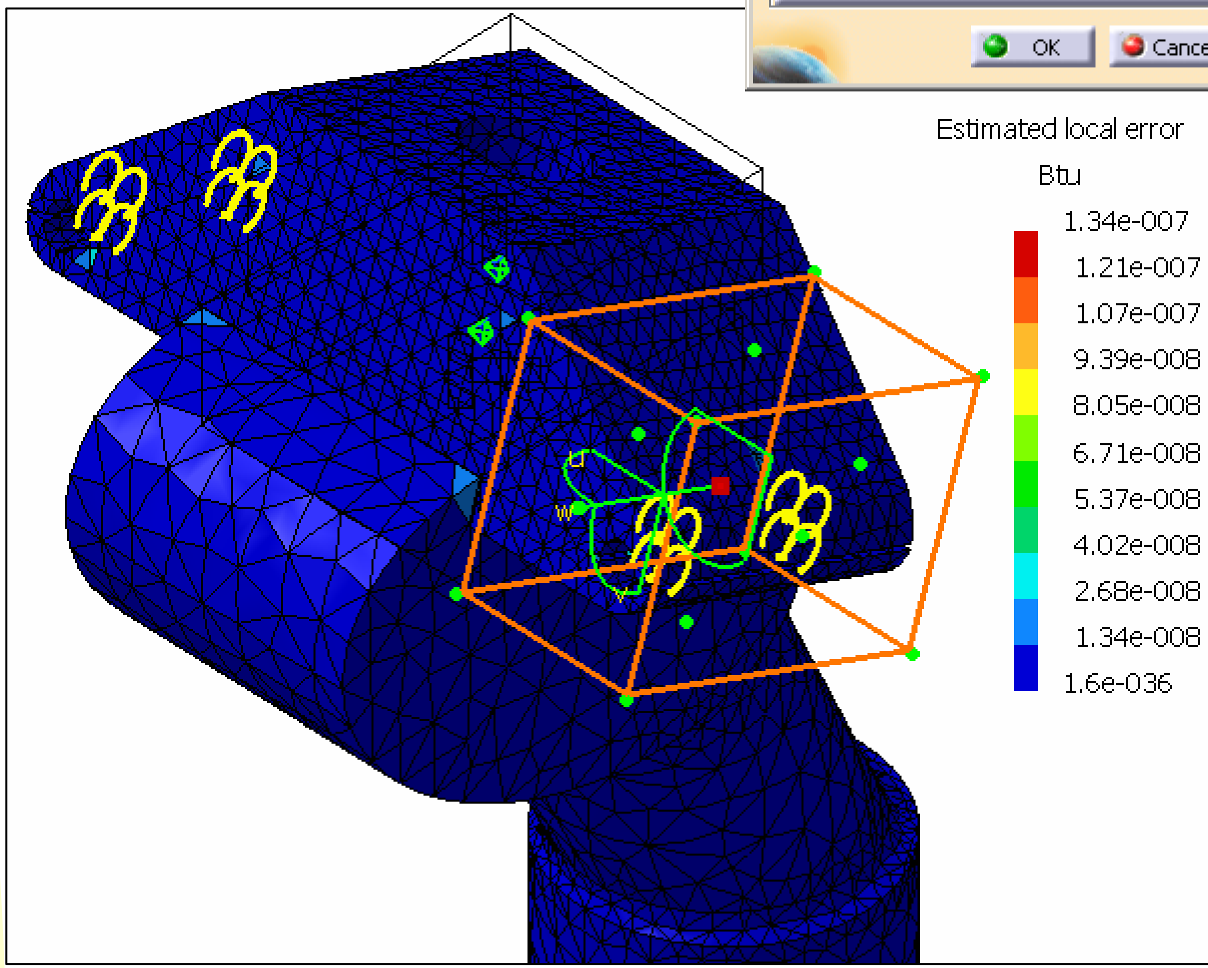


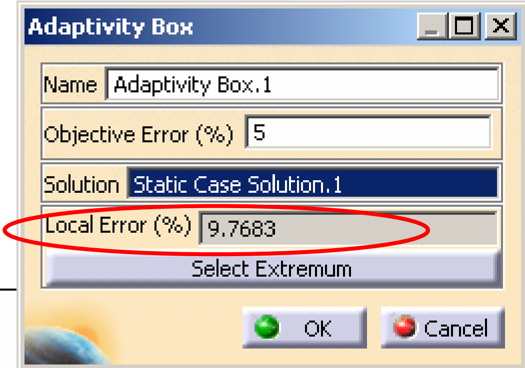
MSC SOFTWARE

Step 14. Precise Results



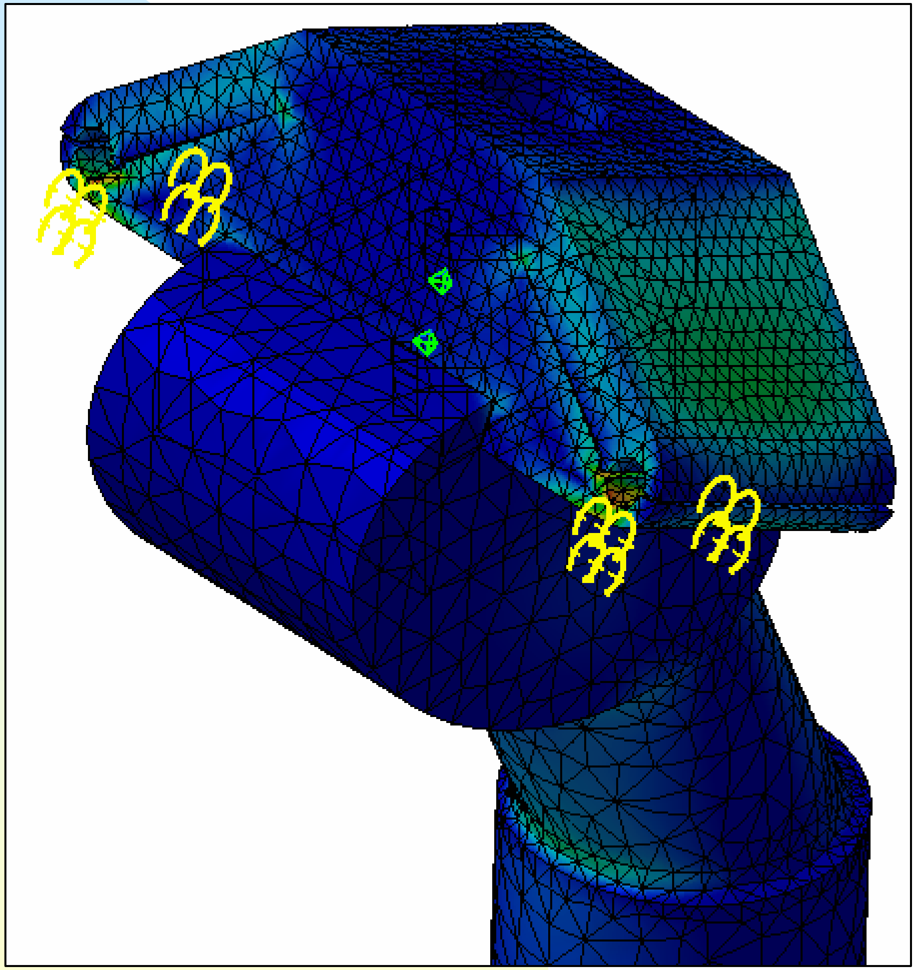
Parabolic elements

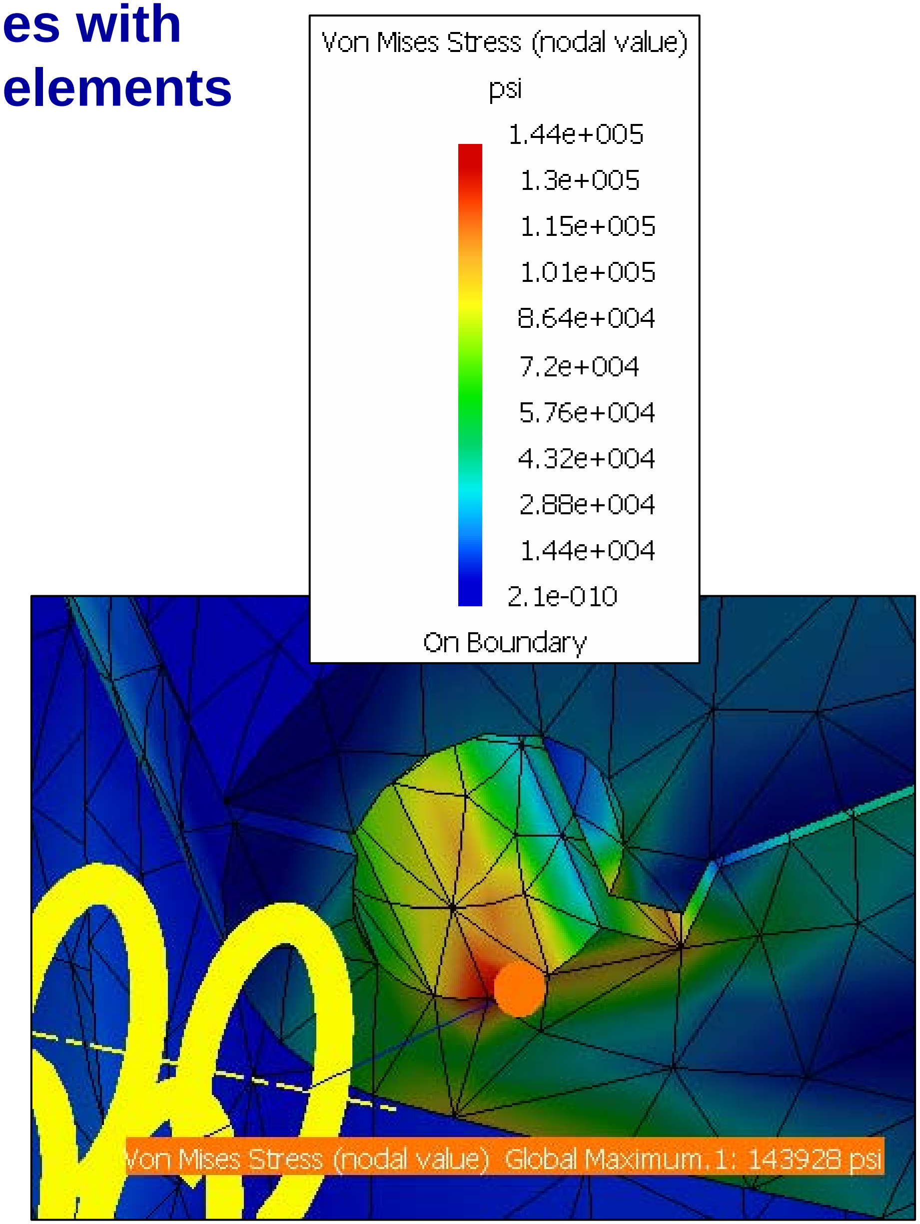




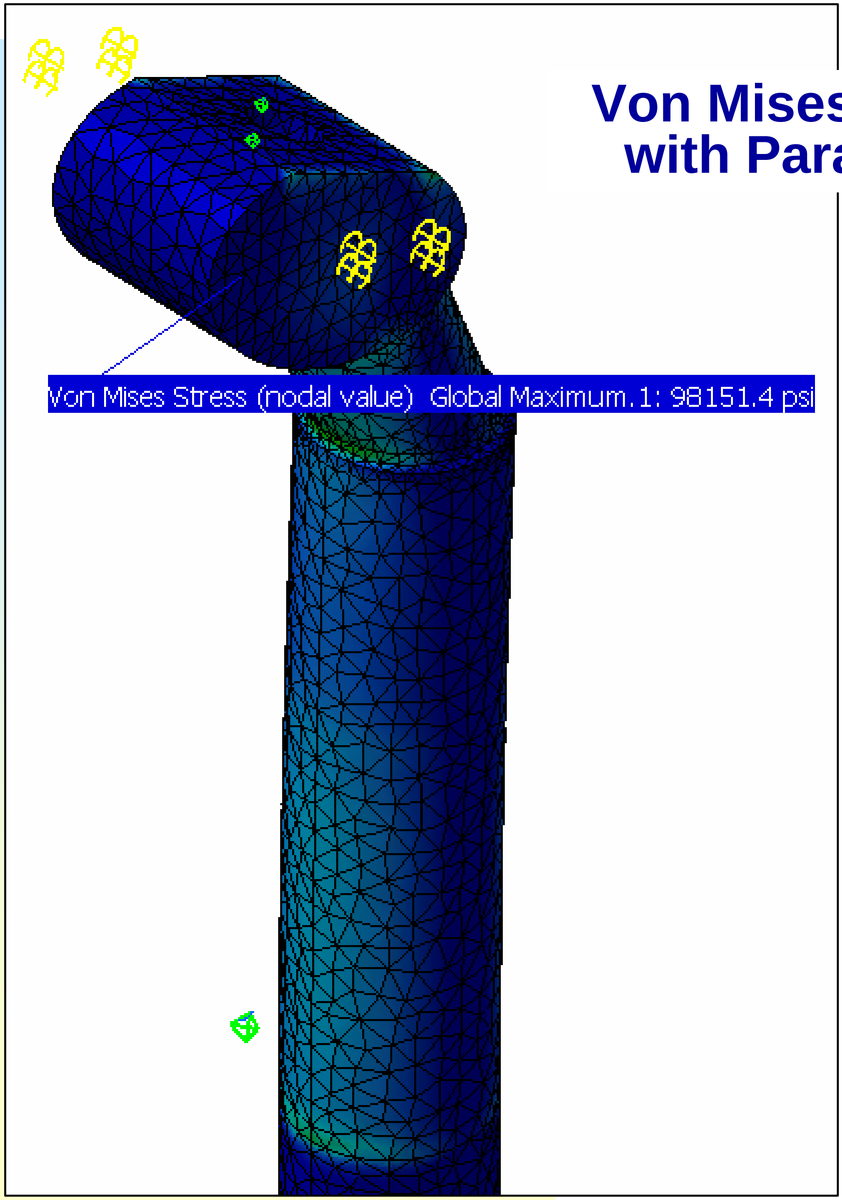
Step 14. Precise Results

Von Mises with Parabolic elements

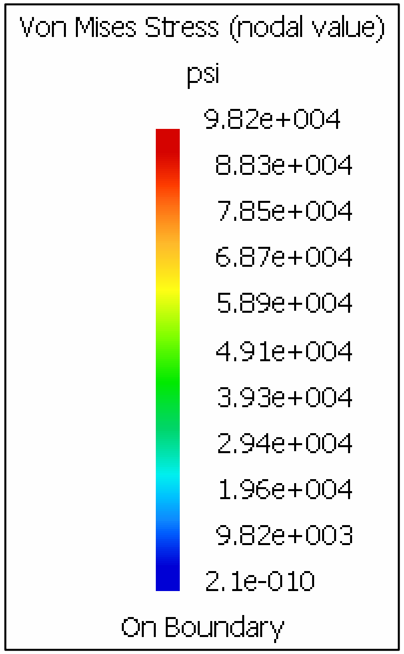


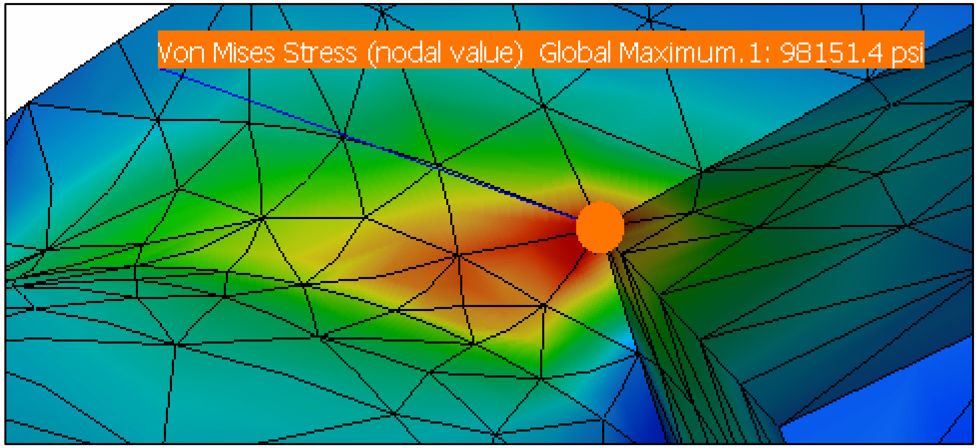


Step 14. Precise Results

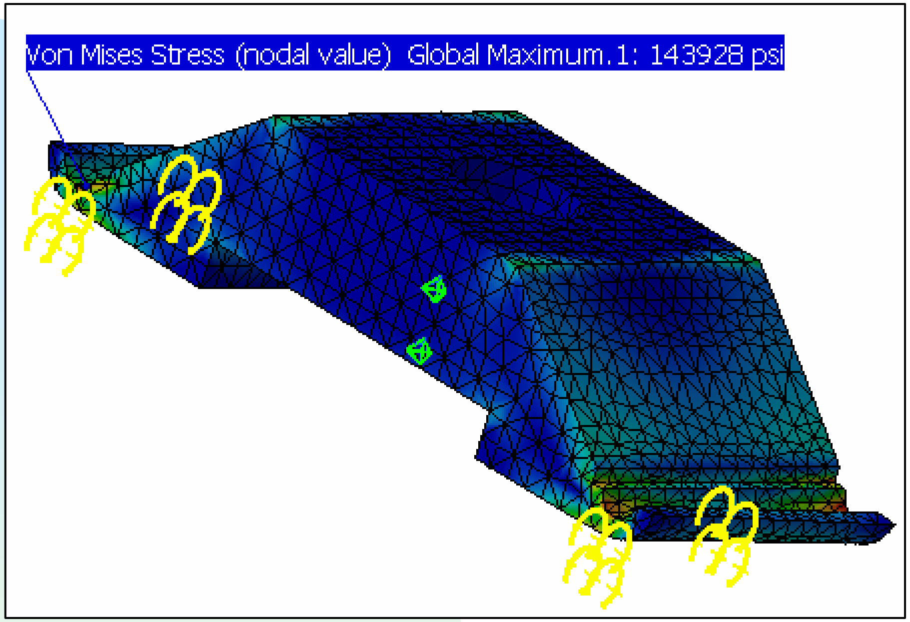


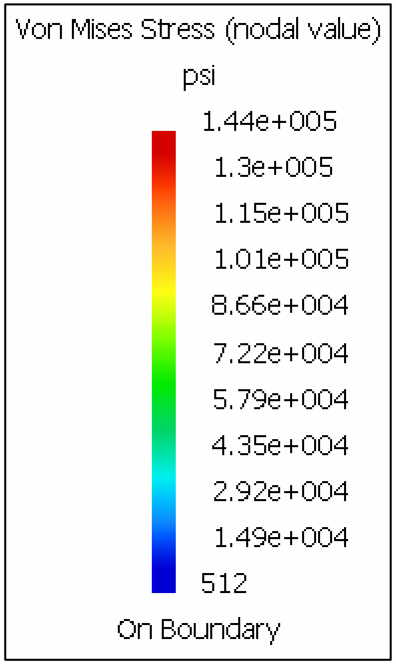
Von Mises "Post" Extrema with Parabolic elements



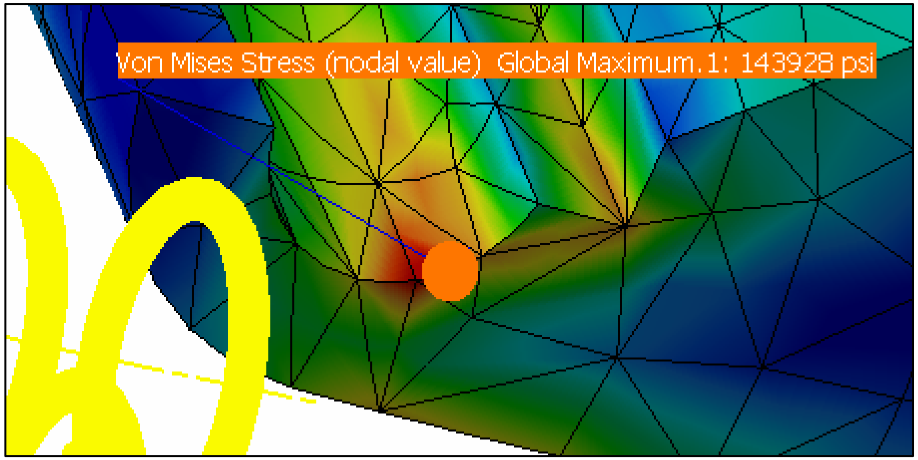


Step 14. Precise Results

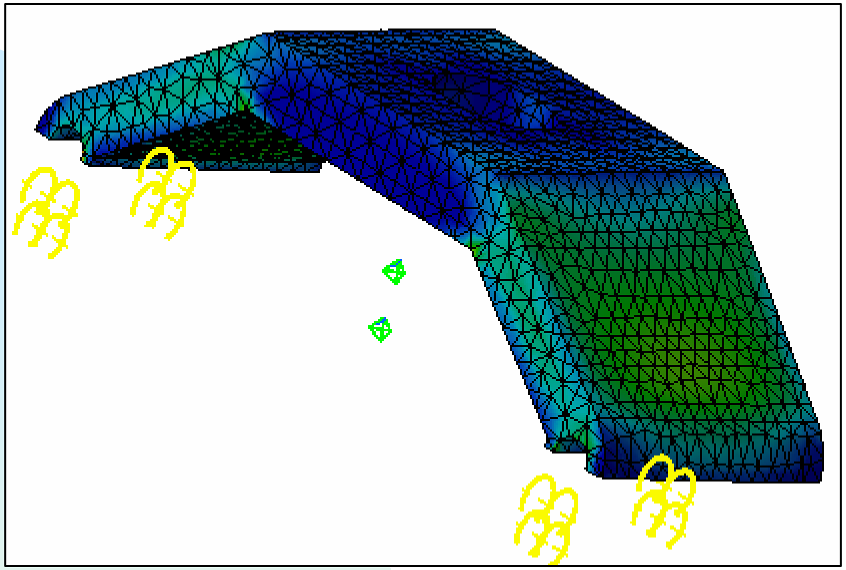


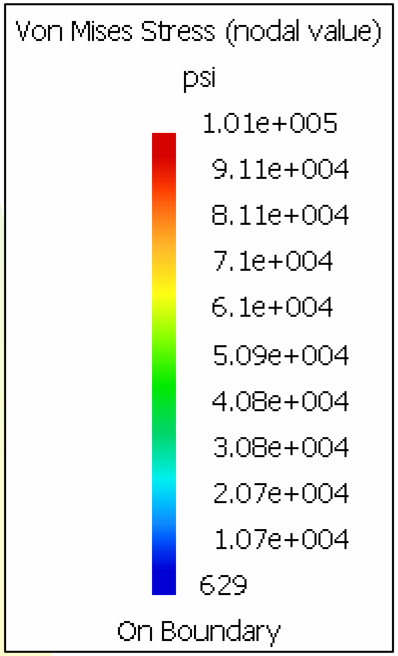


Von Mises "Lower Clamp" Extrema with Parabolic elements

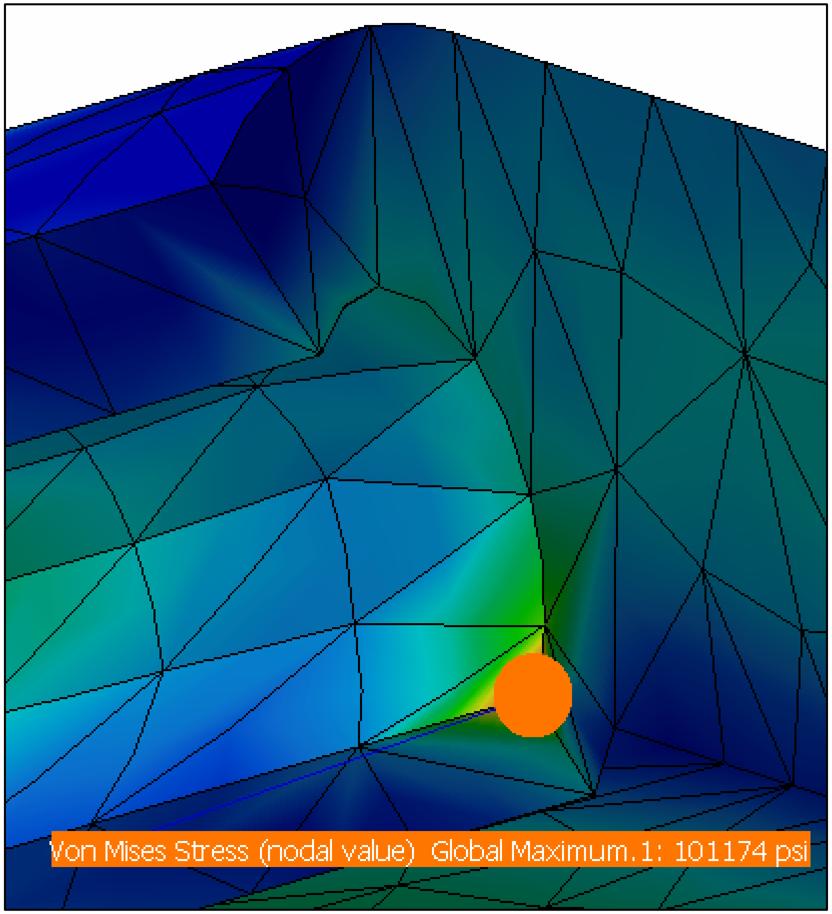


Step 14. Precise Results



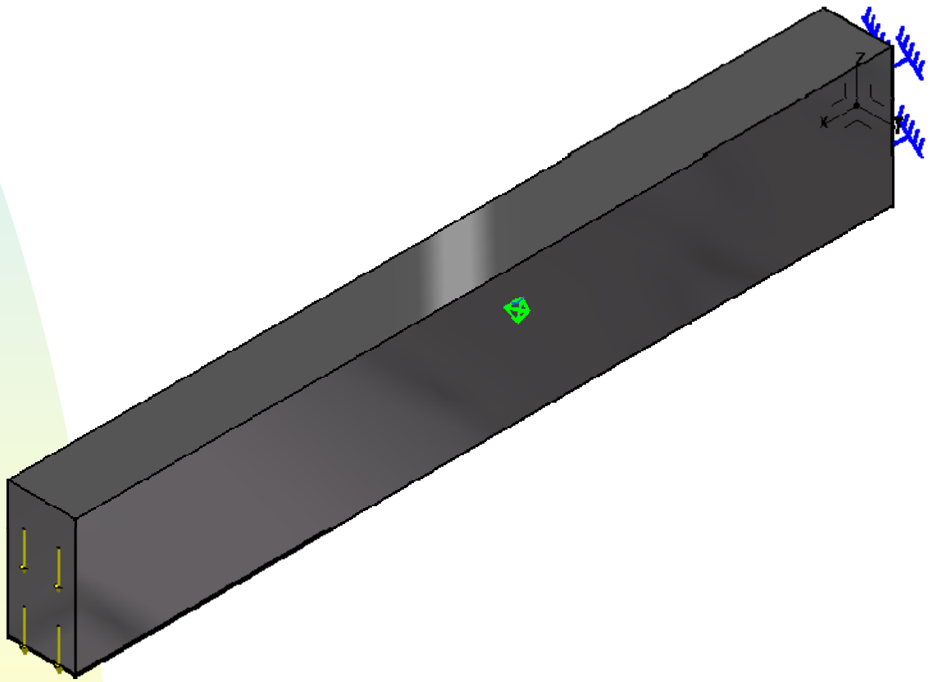


Von Mises “Upper Clamp” Extrema with Parabolic elements



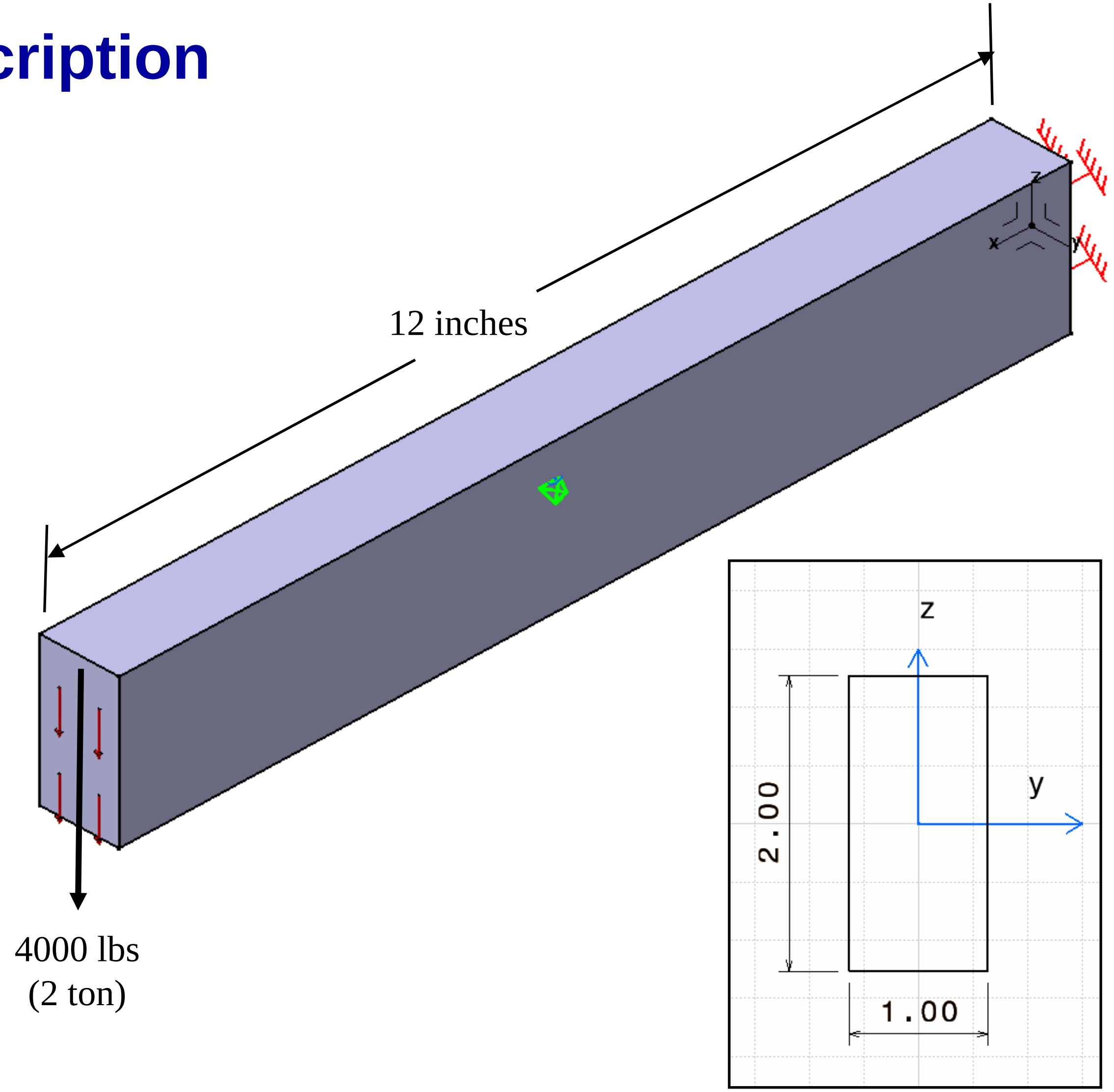
WORKSHOP 8

RECTANGULAR SECTION CANTILEVER BEAM



Problem Description

- Load case.



Material: Heat Treated 4340 Steel
Young Modulus = 29.0e6 psi
Poisson Ratio = .266
Density = .284 lb_in3
Yield Strength = 75000 psi

Problem Description

- Hand Calculations
- Displacement:

$$\text{Moment of Inertia} = I_y = \frac{b \cdot d^3}{12} = \frac{1\ \text{inch} \cdot 2^3\ \text{inch}}{12} = 0.667\ \text{inch}^4$$

$$\text{Displacement} = \delta = \frac{P \cdot L^3}{(3 \cdot E \cdot I)} = \frac{4000\ \text{lbs} \cdot 12^3\ \text{inch}}{(3 \cdot 29e^6\ \text{psi} \cdot 0.667\ \text{inch}^4)} = 0.119\ \text{inch}$$

- Bending Stress

$$\text{Bending Moment} = M_y = P \cdot L = 4000\ \text{lbs} \cdot 12\ \text{inch} = 48000\ \text{in lbs}$$

$$\text{Maximum Bending Stress} = \sigma_b = \frac{M_y \cdot y}{I_y} = \frac{48000\ \text{in lbs} \cdot 1.0\ \text{inch}}{0.667\ \text{inch}^4} = 72000\ \text{psi}$$

- Horizontal shear stress

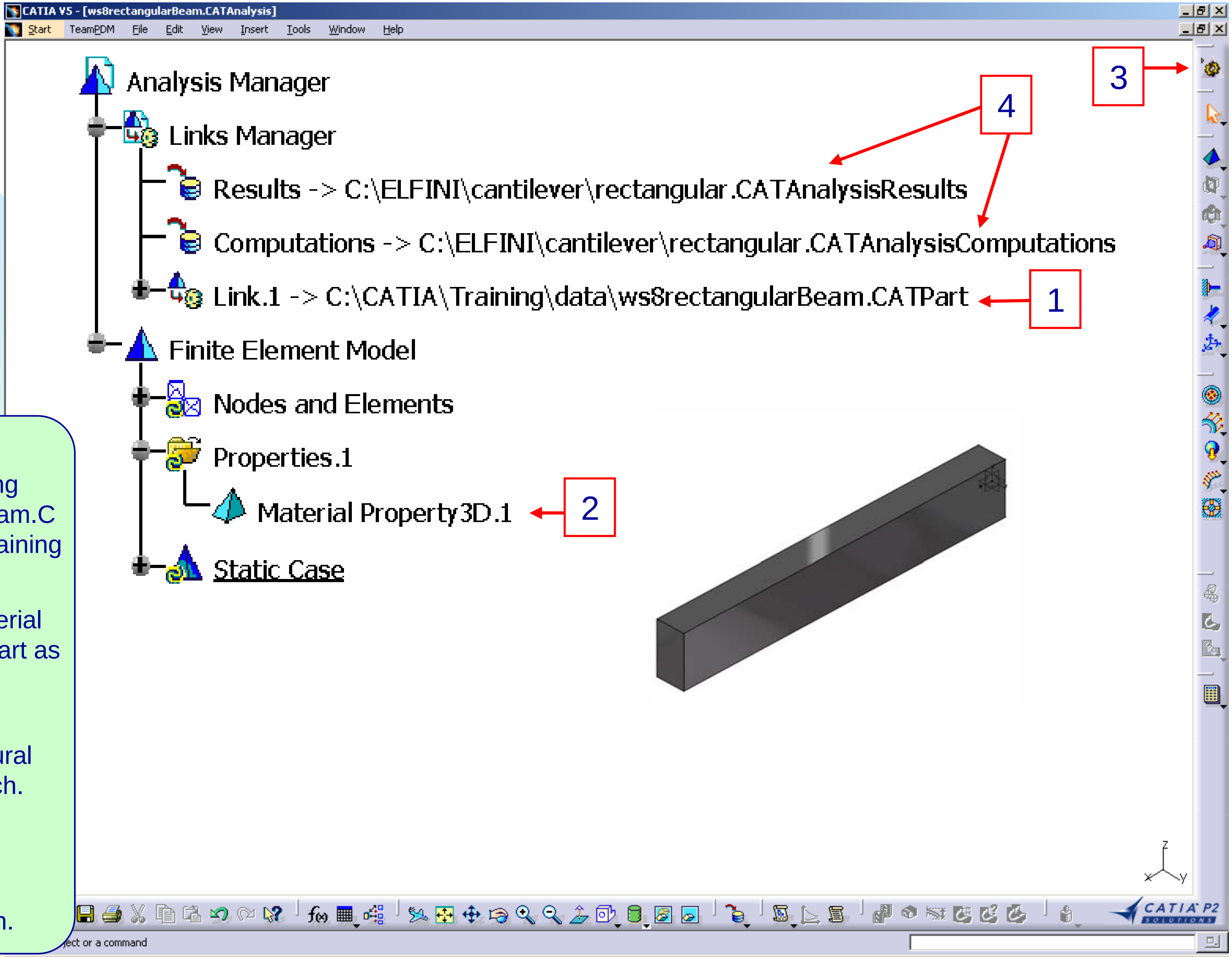
$$\text{Maximum horizontal shear stress at the neutral axis} = \tau = \frac{V \cdot a \cdot y}{I_y \cdot t}$$

$$\tau = \frac{4000\ \text{lbs} \cdot 1.0\ \text{in}^2 \cdot 0.50\ \text{in}}{0.667\ \text{in}^4 \cdot 1\ \text{in}} = 3000\ \text{psi}$$

Suggested Exercise Steps

1. Create a new CATIA analysis document (.CATAnalysis).
2. Mesh globally with linear elements.
3. Apply a clamp restraint.
4. Apply a distributed force.
5. Compute the initial analysis.
6. Check global and local precision (animate deformation, adaptive boxes and extremas).
7. Change mesh to parabolic.
8. Compute the precise analysis.
9. Visualize final results.
10. Save the analysis document.

Step 1. Create a new CATIA analysis document



Steps:

1. Open the existing ws8rectangularBeam.CATPart from the training directory.

2. Apply steel material properties to the part as required.

3. Launch the Generative Structural Analysis workbench.

4. Specify the Computations and Results storage locations as shown.

Step 2. Mesh globally with linear elements

Define the global finite element mesh properties.

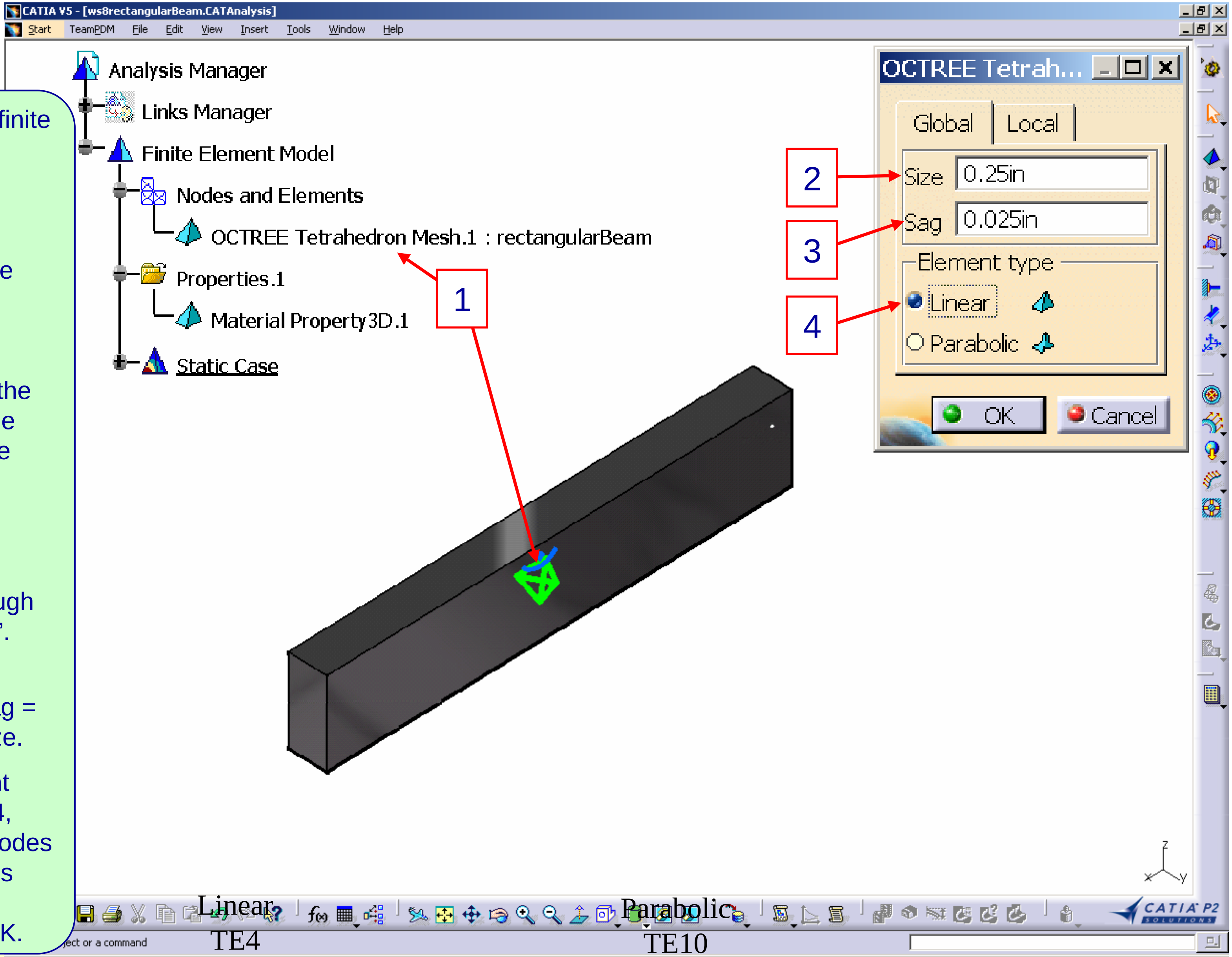
Steps:

1. Double Click the "OCTREE Tetrahedron Mesh.1:Pedal" representation in the features tree or the "Mesh" icon on the part.

2. Specify the recommended rough Global Size = .25".

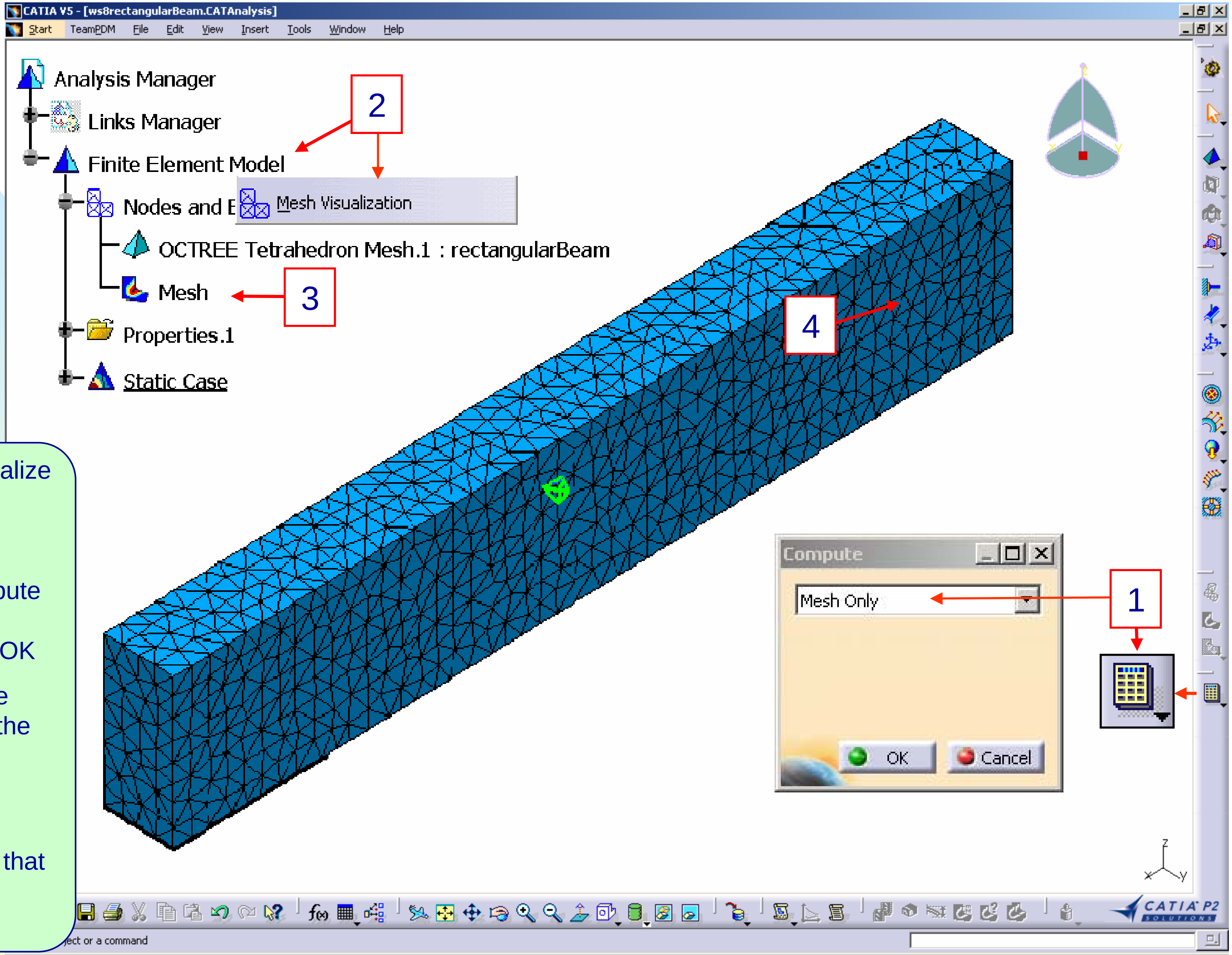
3. Specify the recommended Sag = 10% of Global Size.

4. Specify element type "Linear" (TE4, means 4 corner nodes tetrahedron) and is good for a rough analysis, select OK.



MSC SOFTWARE

Step 2. Mesh globally with linear elements



Compute and visualize the mesh only

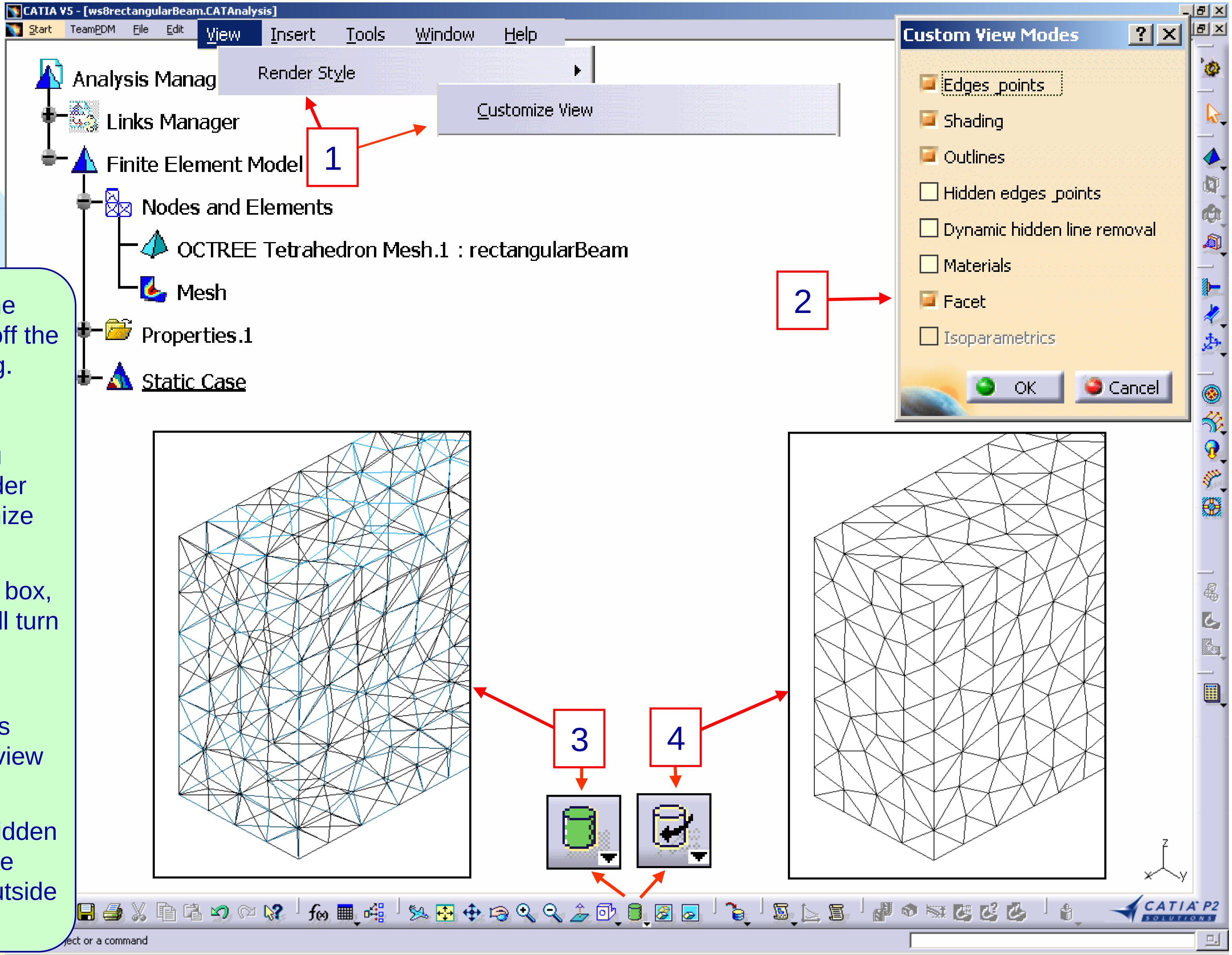
Steps:

1. Select the compute icon and compute mesh only, select OK

2. Right click Finite element Model in the features tree then select Mesh Visualization.

3. Note the image that get added to the features tree.

Step 2. Mesh globally with linear elements



Better visualize the mesh by turning off the material rendering.

Steps:

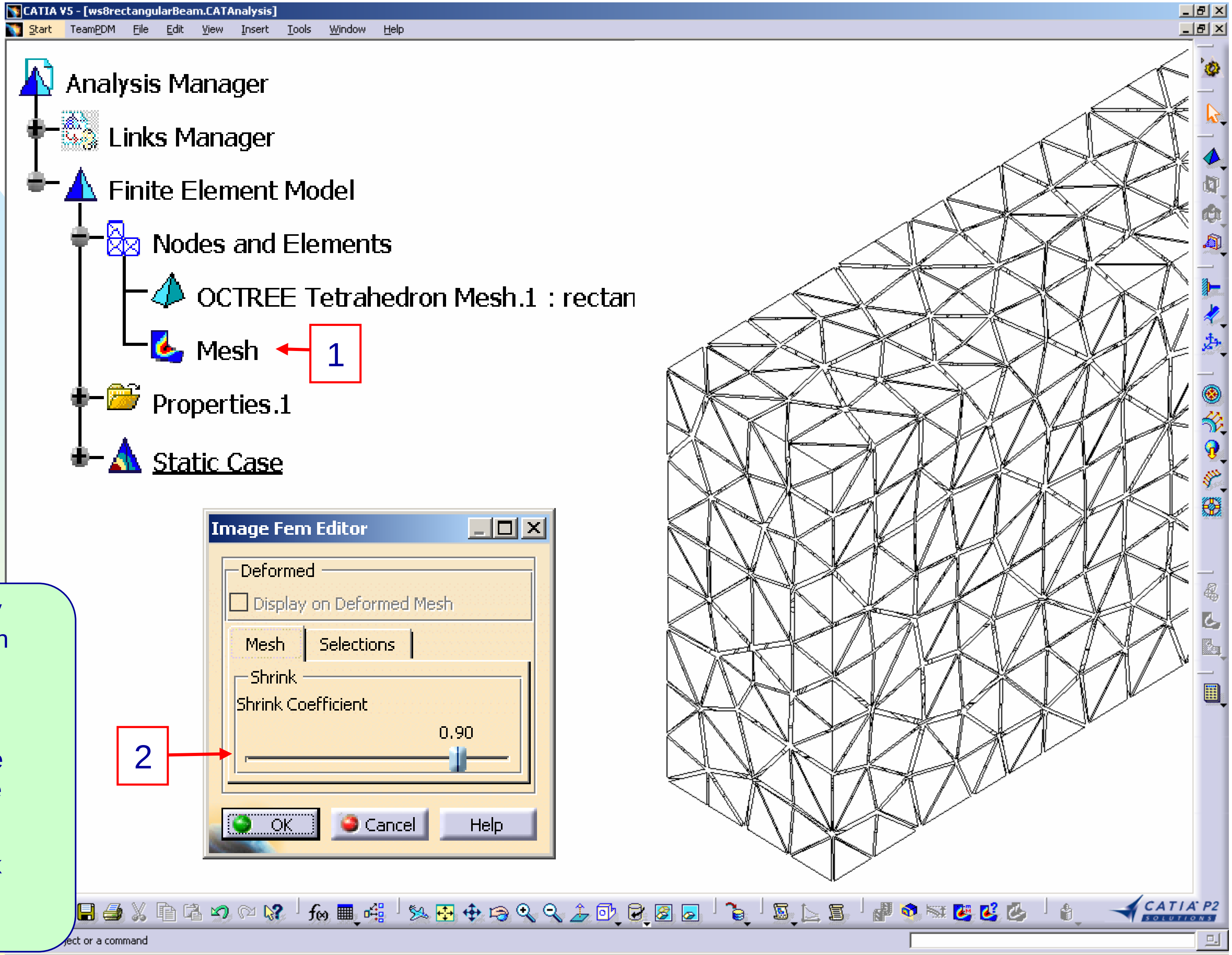
1. From the menu select View, Render Style and Customize View.

2. Click the Facet box, select OK (this will turn off the Materials rendering).

3. This icon shows your customized view parameters.

4. The dynamic hidden line removal image shows only the outside elements.

Step 2. Mesh globally with linear elements



Better visualize by shrinking the mesh elements.

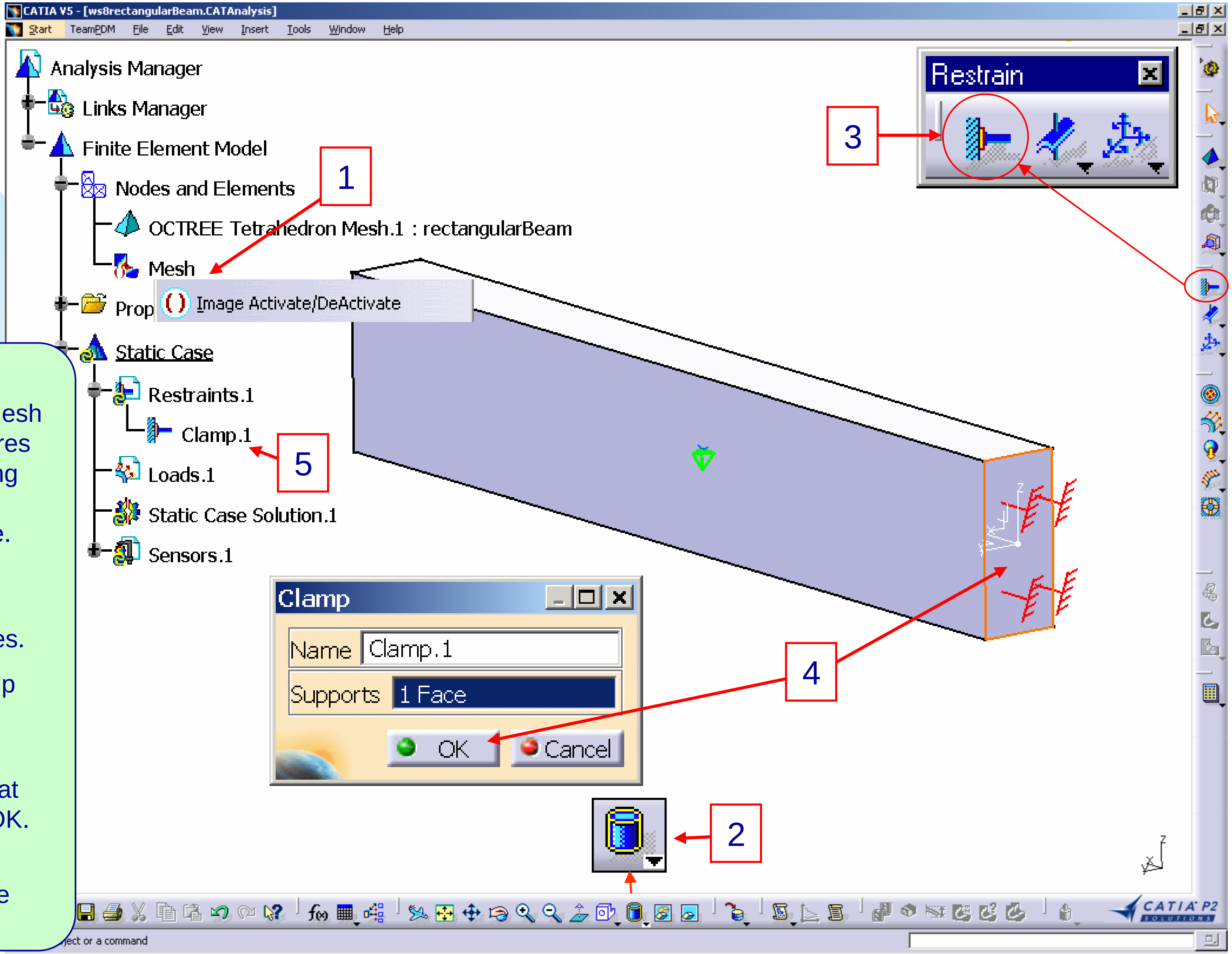
Steps:

1. Double click the Mesh object in the features tree.

2. Slide the Shrink Coefficient bar to 0.90%, select OK.

MSC SOFTWARE

Step 3. Apply a clamp restraint



Steps:

1. Inactivate the Mesh image in the features tree by right clicking then select Image activate/deactivate.

2. Change your display mode to Shading with Edges.

3. Select the Clamp Restraint icon.

4. Select the face at the origin, select OK.

5. Note the Clamp object added to the features tree.

Step 3. Apply a clamp restraint

Examine the details of what this clamp feature is doing at the nodes.

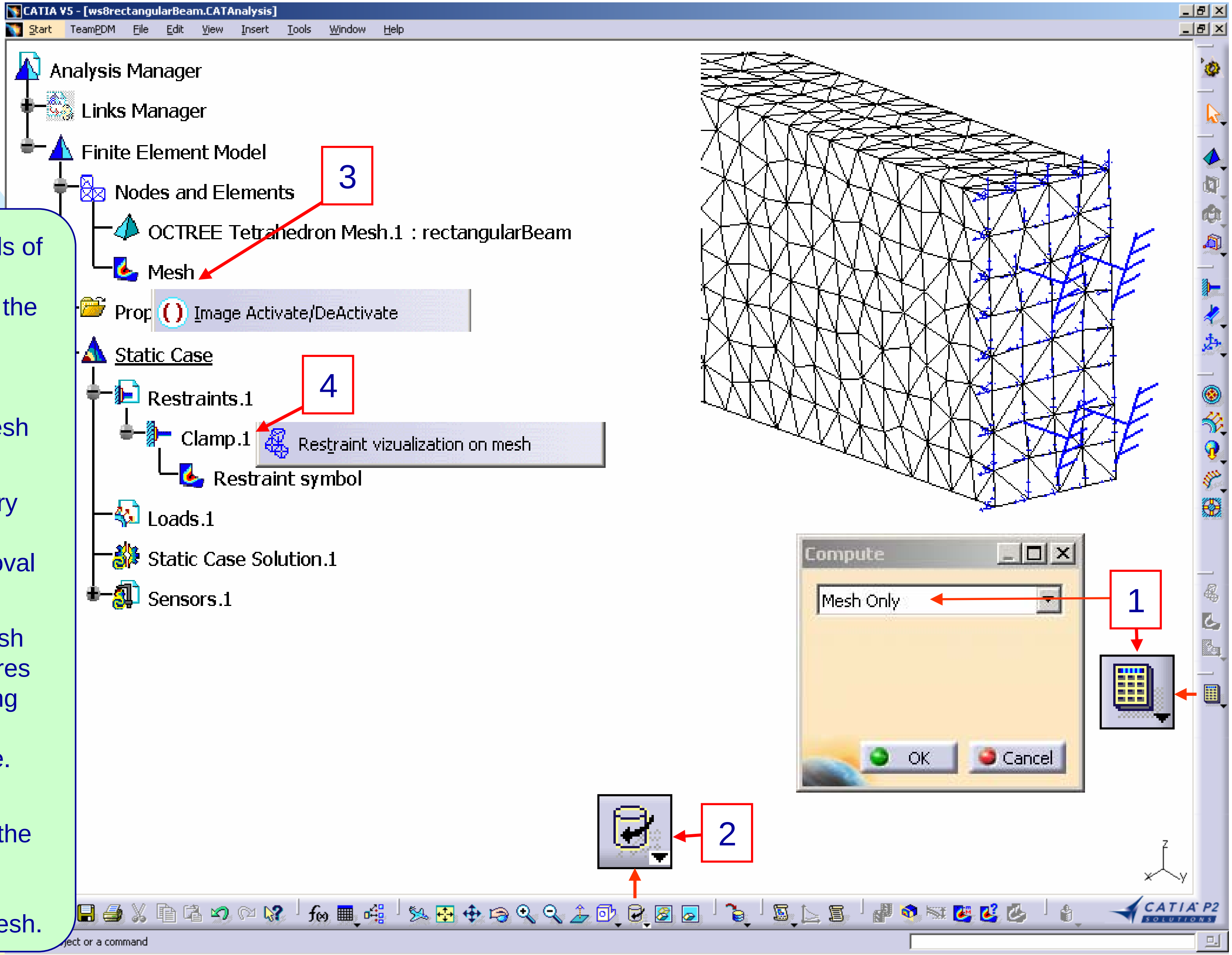
Steps:

1. Re-compute Mesh only.

2. Display geometry with the Dynamic Hidden Line Removal icon.

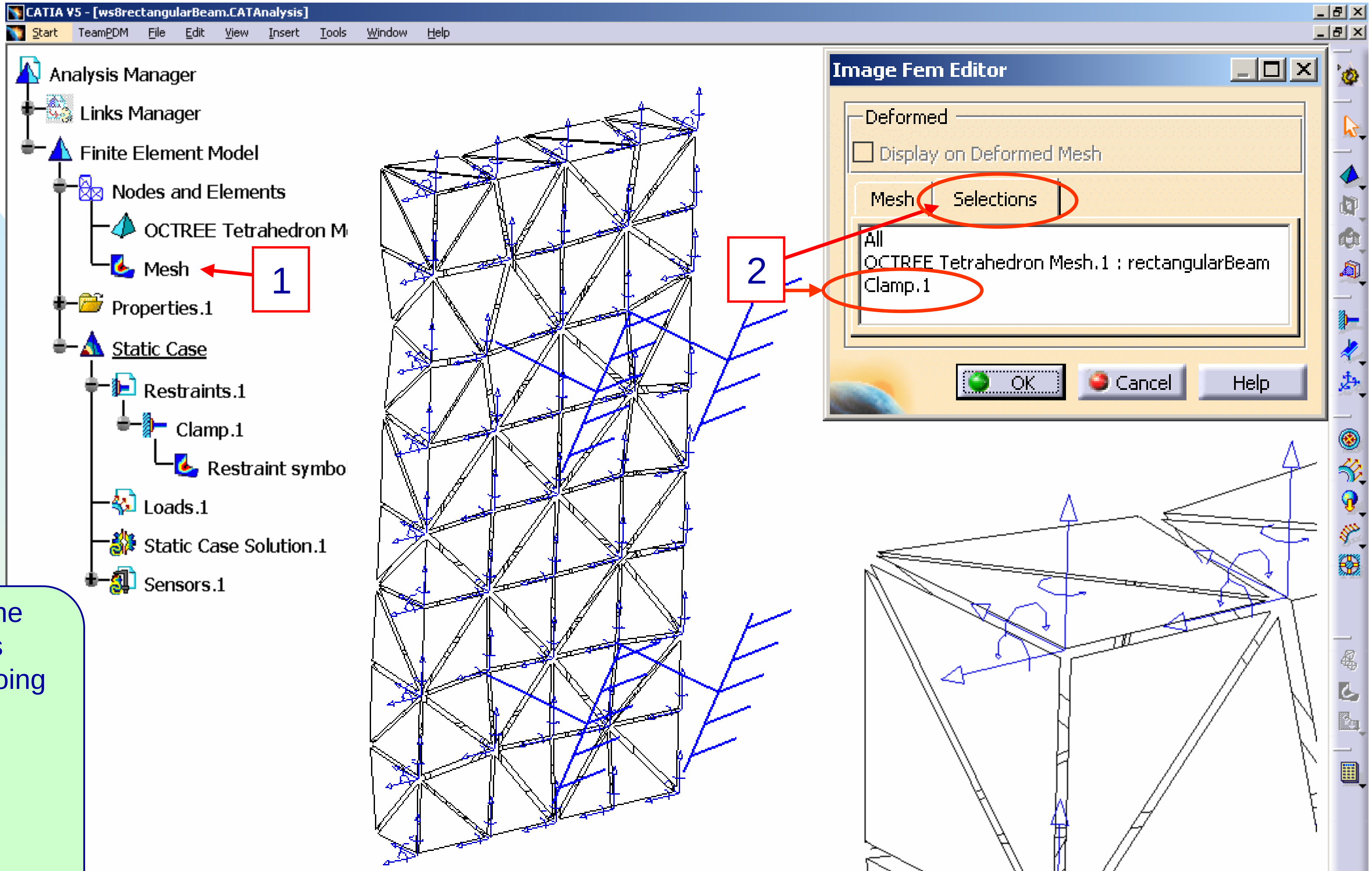
3. Activate the Mesh image in the features tree by right clicking then select Image activate/deactivate.

4. Right click the Clamp.1 object in the features tree then select Restraint visualization on mesh.



MSC SOFTWARE

Step 3. Apply a clamp restraint



Symbol indicates clamped,
or all 6 degrees of freedom restricted.

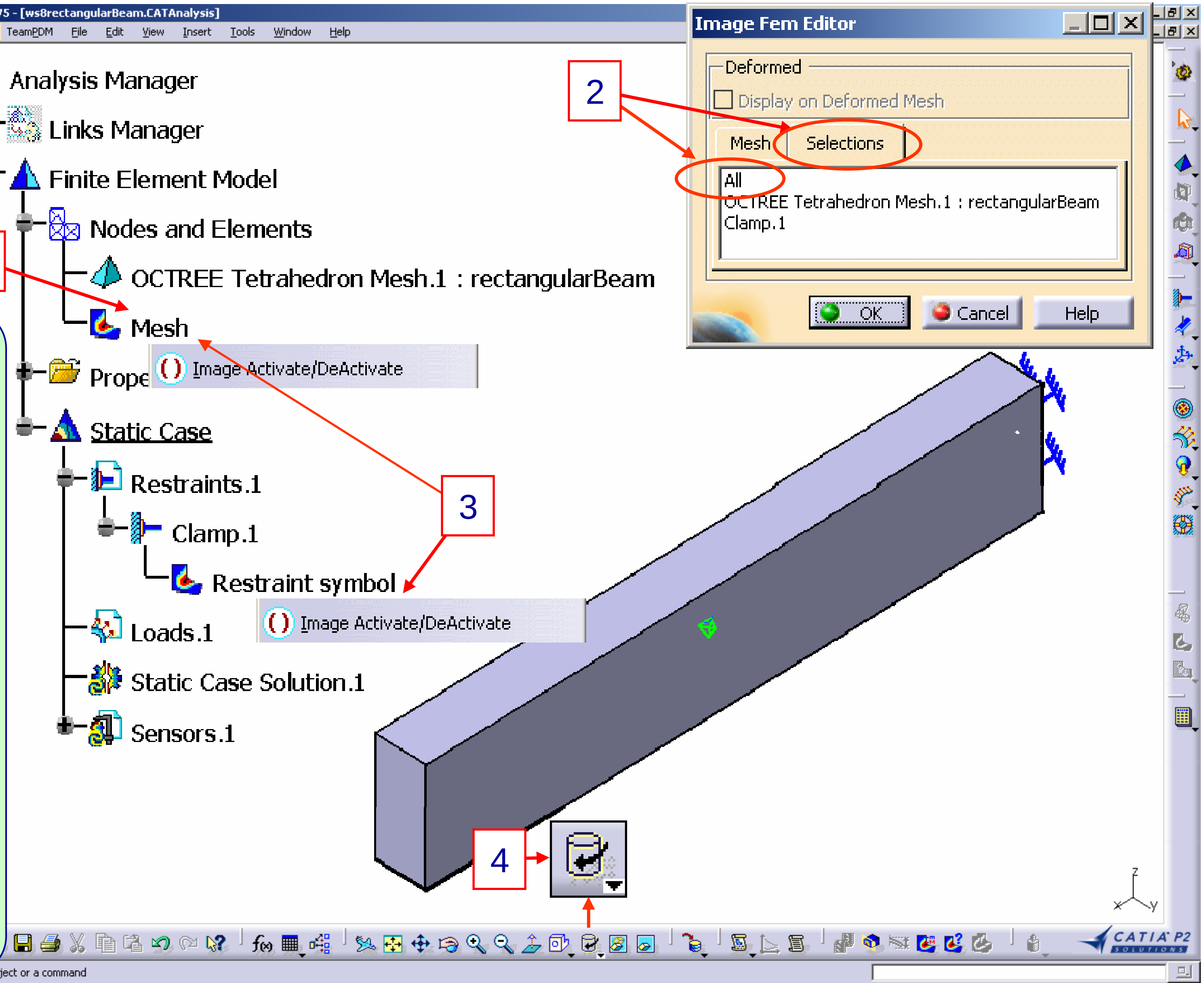
Further examine the details of what this clamp feature is doing at the nodes.

Steps:

1. Double click the Mesh object in the features tree.

2. Select the Selections tab and Clamp.1 in the Fem Editor, select OK.

Step 4. Apply a distributed force



Steps:

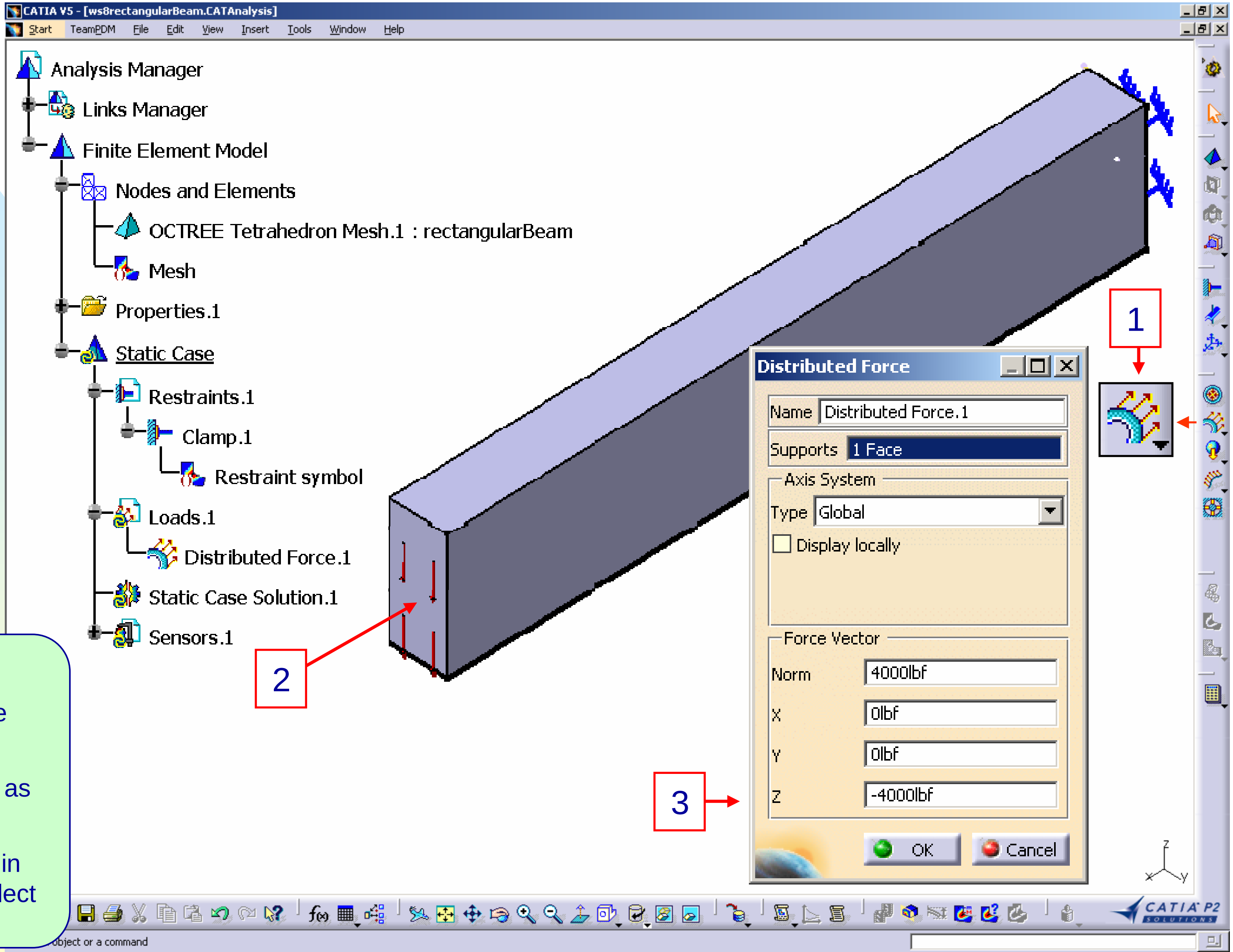
1. Double click the Mesh object in the features tree.

2. Select the Selections tab and “All” in the Fem Editor, select OK..

3. DeActivate the “Restraint symbol” and the “Mesh” image in the features tree by right clicking then select Image activate/deactivate.

4. Display geometry with the Dynamic Hidden Line Removal icon.

Step 4. Apply a distributed force



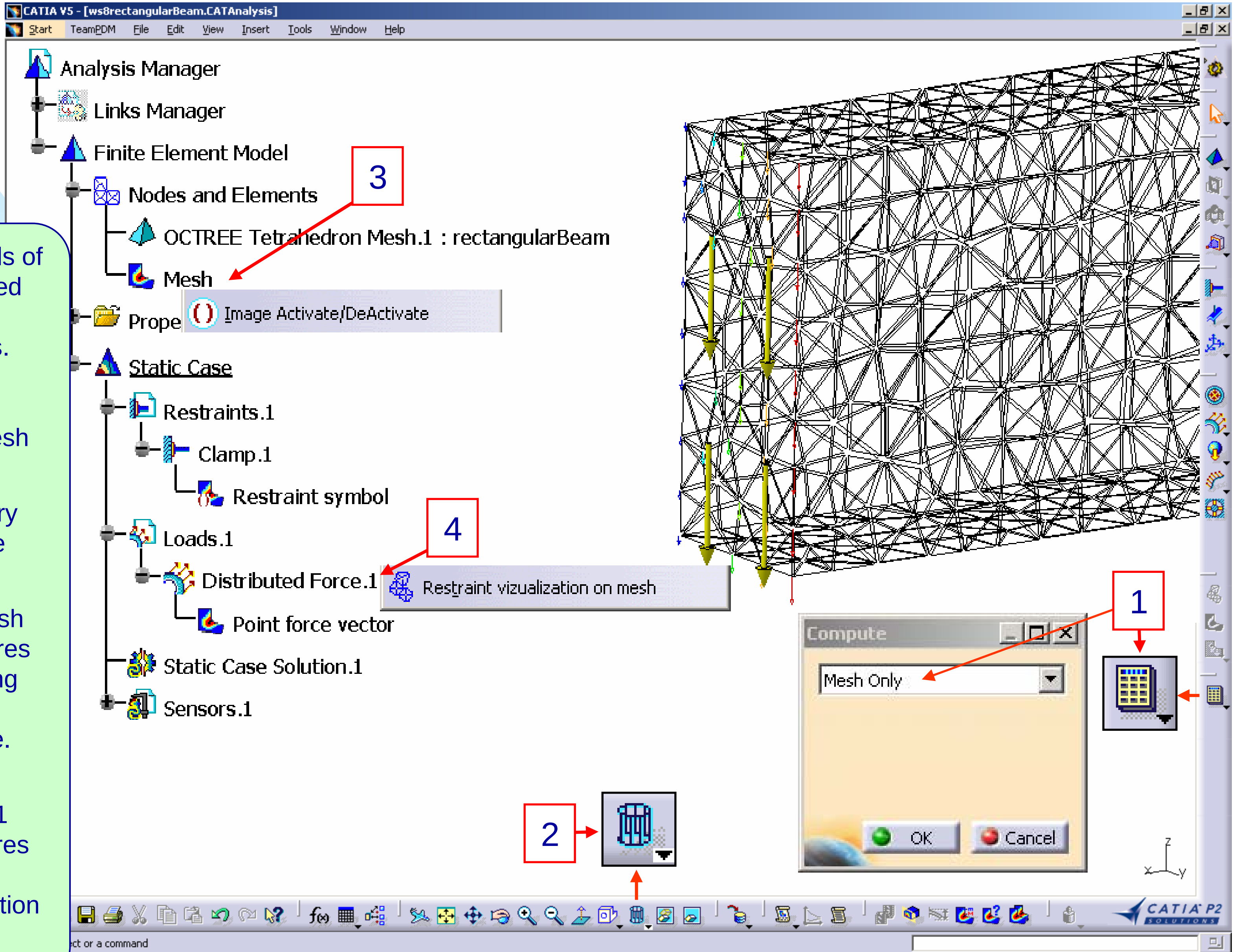
Steps:

1. Select the Force icon.

2. Select end face as shown.

3. Enter -4000 lbs in the Z-direction, select OK.

Step 4. Apply a distributed force



Examine the details of what this Distributed Force.1 feature is doing at the nodes.

Steps:

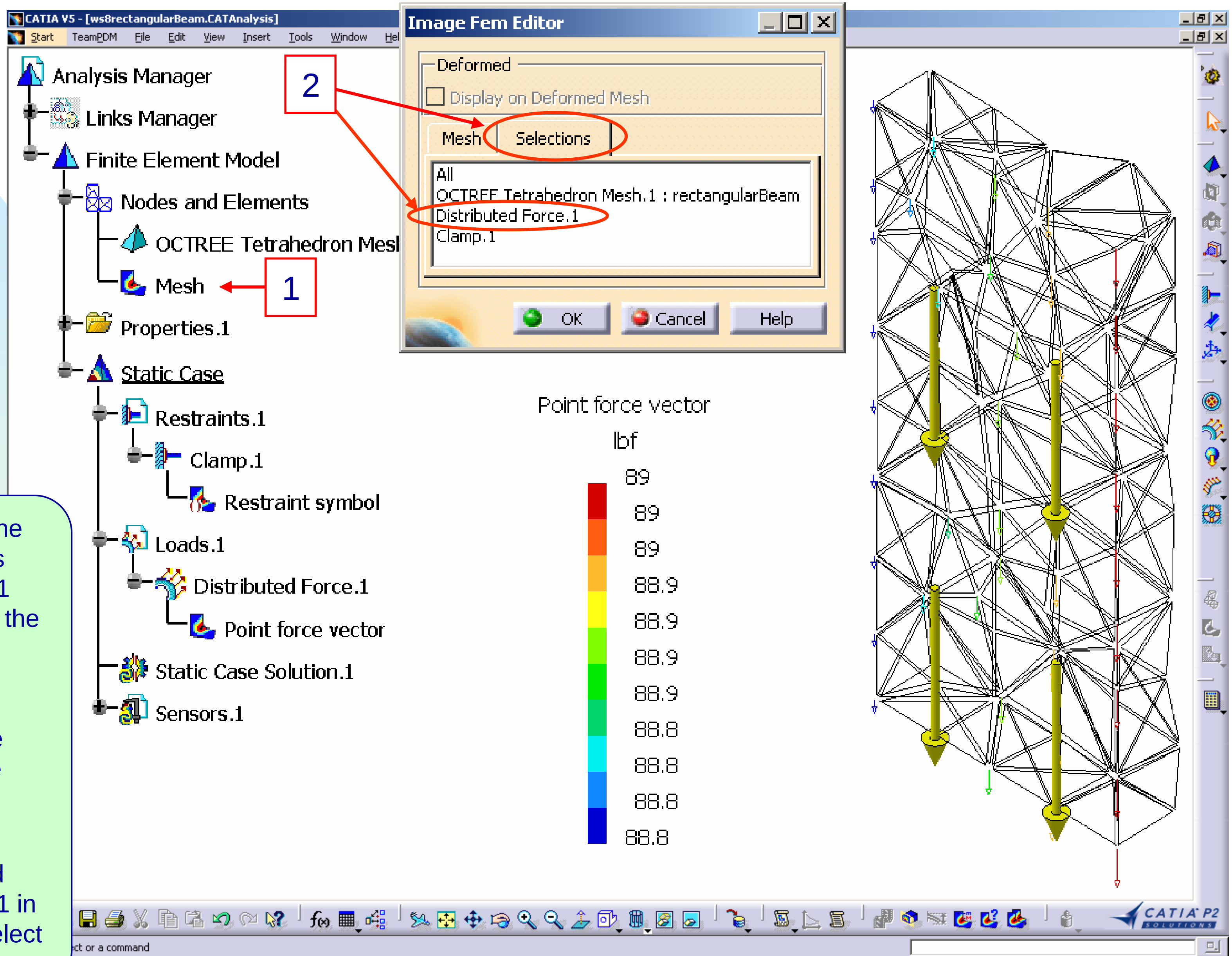
1. Re-compute Mesh only.

2. Display geometry with the Wireframe (NHR) icon.

3. Activate the Mesh image in the features tree by right clicking then select Image activate/deactivate.

4. Right click Distributed Force.1 object in the features tree then select Restraint visualization on mesh.

Step 4. Apply a distributed force



Further examine the details of what this Distributed Force.1 feature is doing at the nodes.

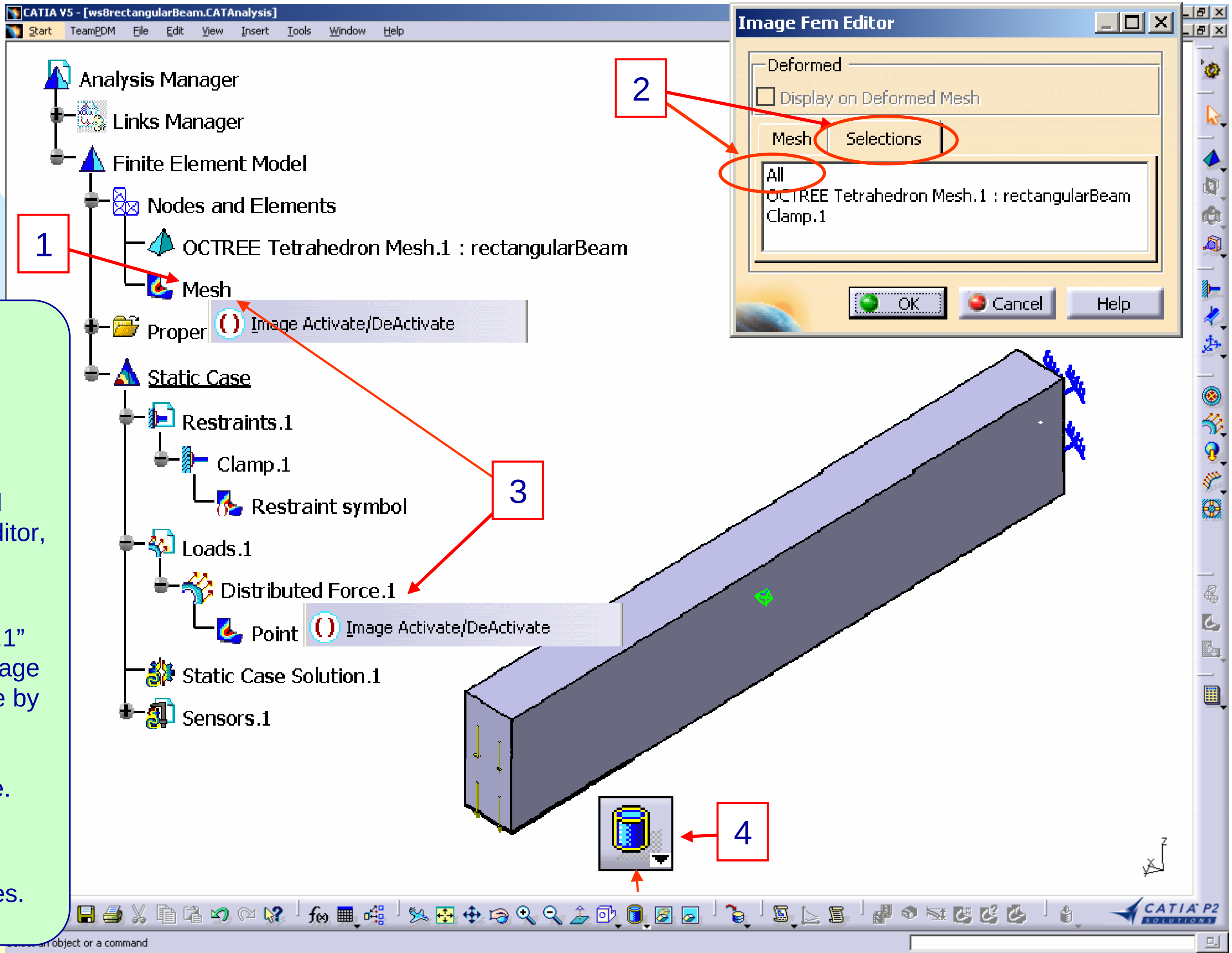
Steps:

1. Double click the Mesh object in the features tree.

2. Select the Selections tab and Distributed Force.1 in the Fem Editor, select OK.

MSC SOFTWARE

Step 5. Compute the initial analysis



Steps:

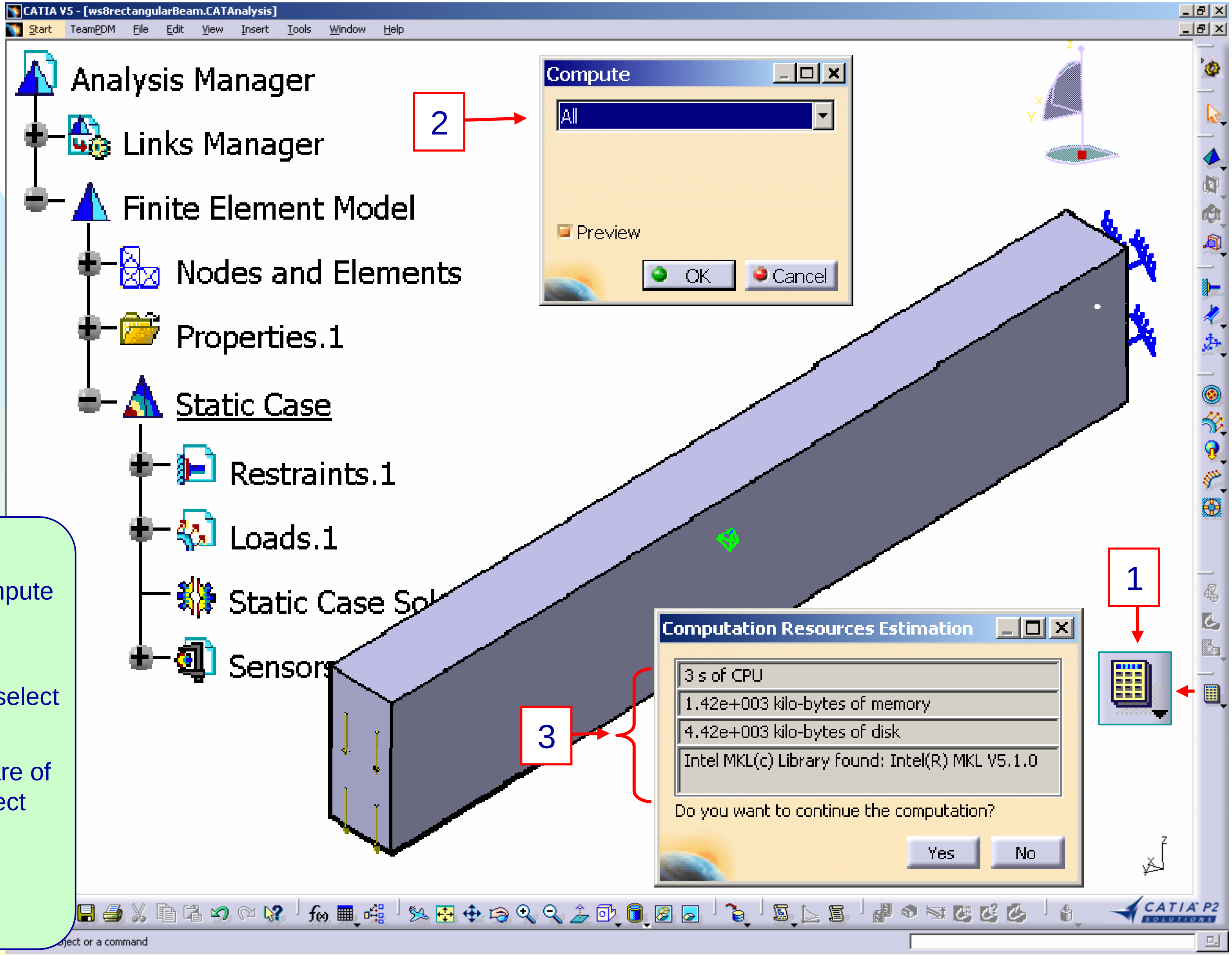
1. Double click the Mesh object in the features tree.

2. Select the Selections tab and "All" in the Fem Editor, select OK..

3. DeActivate the "Distributed Force.1" and the "Mesh" image in the features tree by right clicking then select Image activate/deactivate.

4. Change your display mode to Shading with Edges.

Step 5. Compute the initial analysis



Steps:

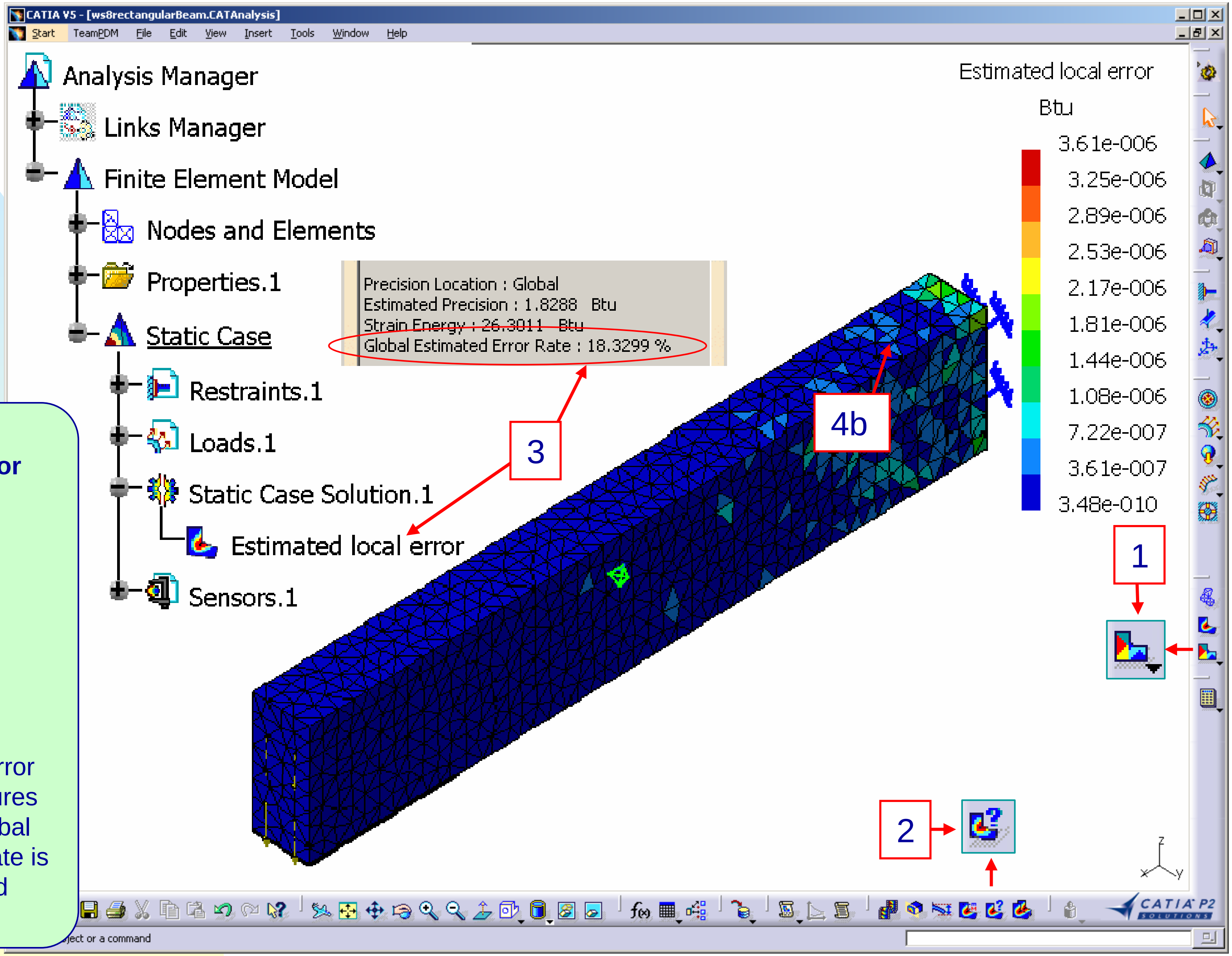
1. Select the Compute icon.

2. Compute All Objects defined, select OK.

3. Always be aware of these values, select Yes.

Save often.

Step 6. Check global and local precision



Visualize the **computation error map.**

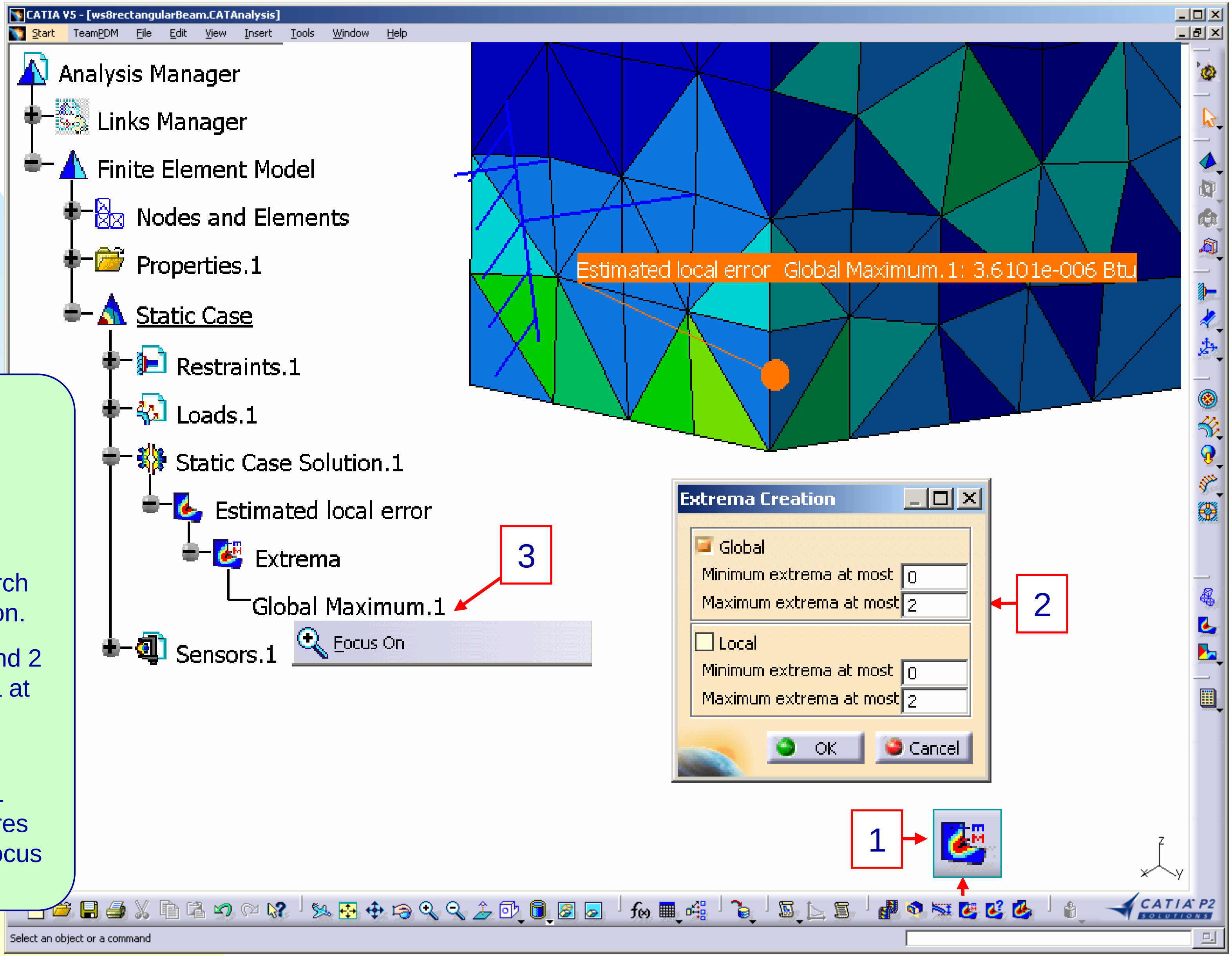
Steps:

1. Select the Precision icon.

2. Select on the information icon.

3. Select the Estimated local error object in the features tree. Note the global estimated error rate is good (recommend max 20%).

Step 6. Check global and local precision



Find the global element with the highest estimated error.

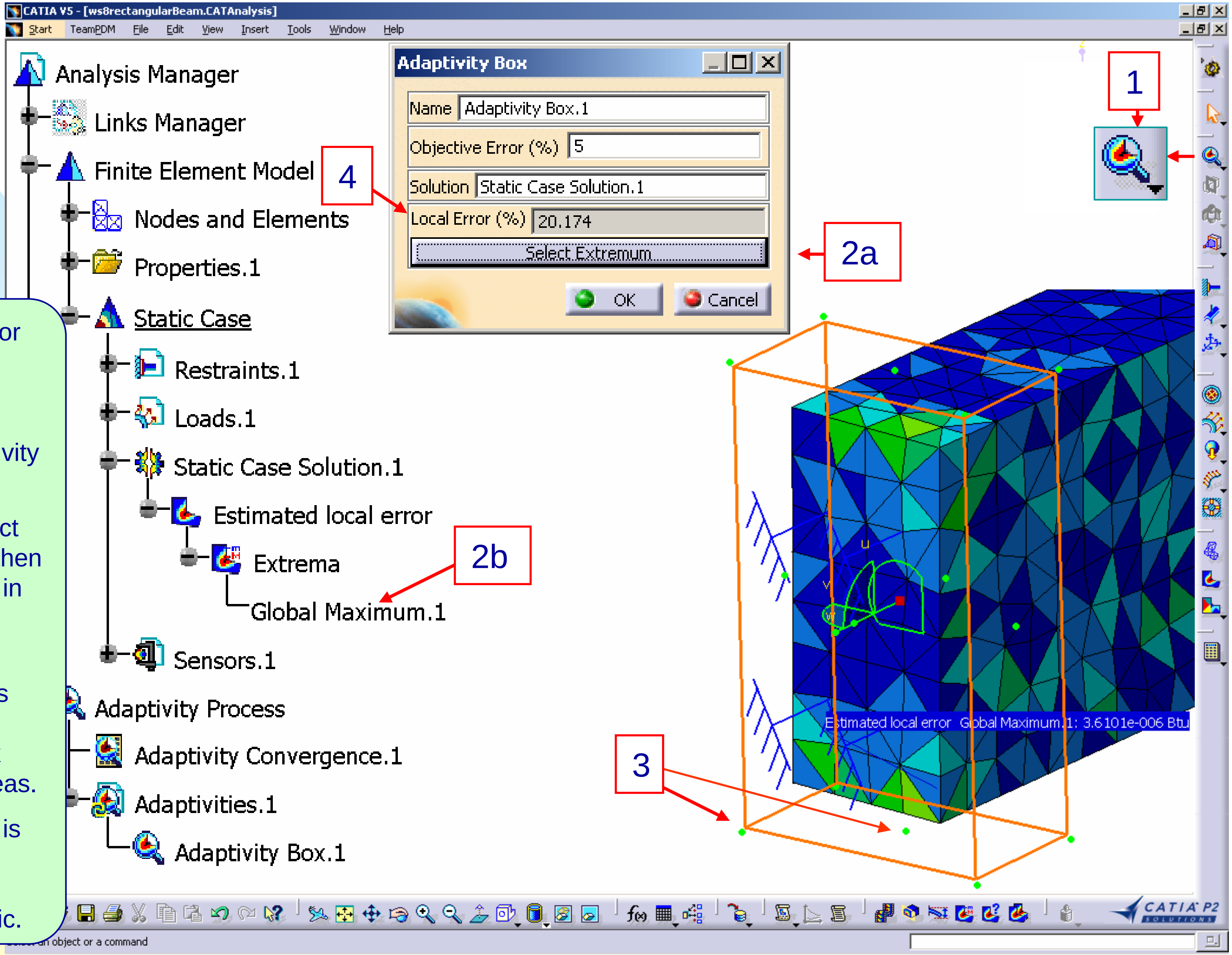
Steps:

1. Select the Search Image Extrema icon.

2. Select Global and 2 maximum extrema at most, select OK.

3. Right click the Global Maximum.1 object in the features tree then select Focus On.

Step 6. Check global and local precision



Determine local error percentage (%).

Steps:

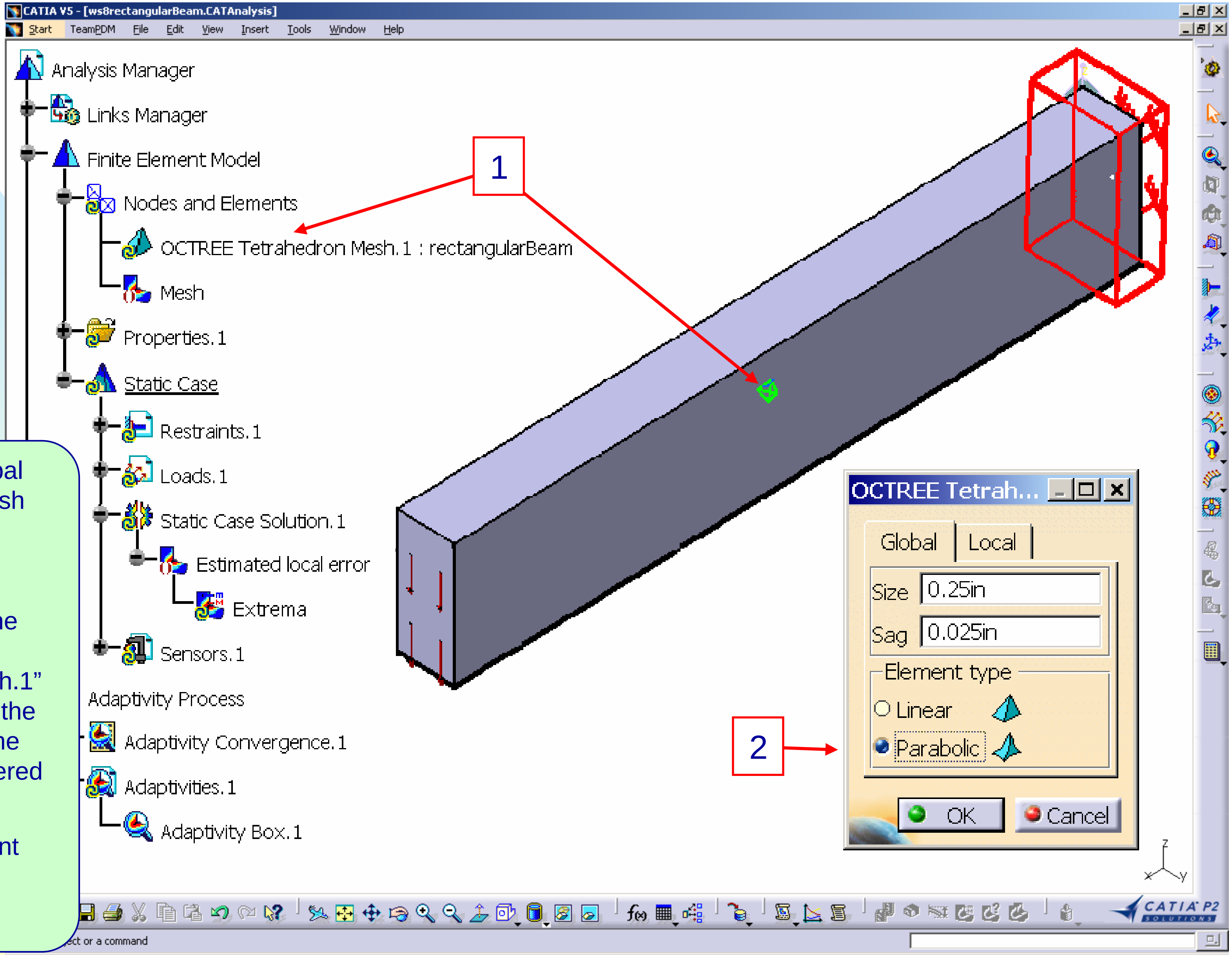
1. Select the adaptivity box icon.

2. Select the "Select Extremum" button then Global Maximum.1 in the features tree to locate box.

3. Use the compass and green dots to locate and size box around meshed areas.

4. Since local error is above 10% try changing the mesh element to Parabolic.

Step 7. Change mesh to parabolic



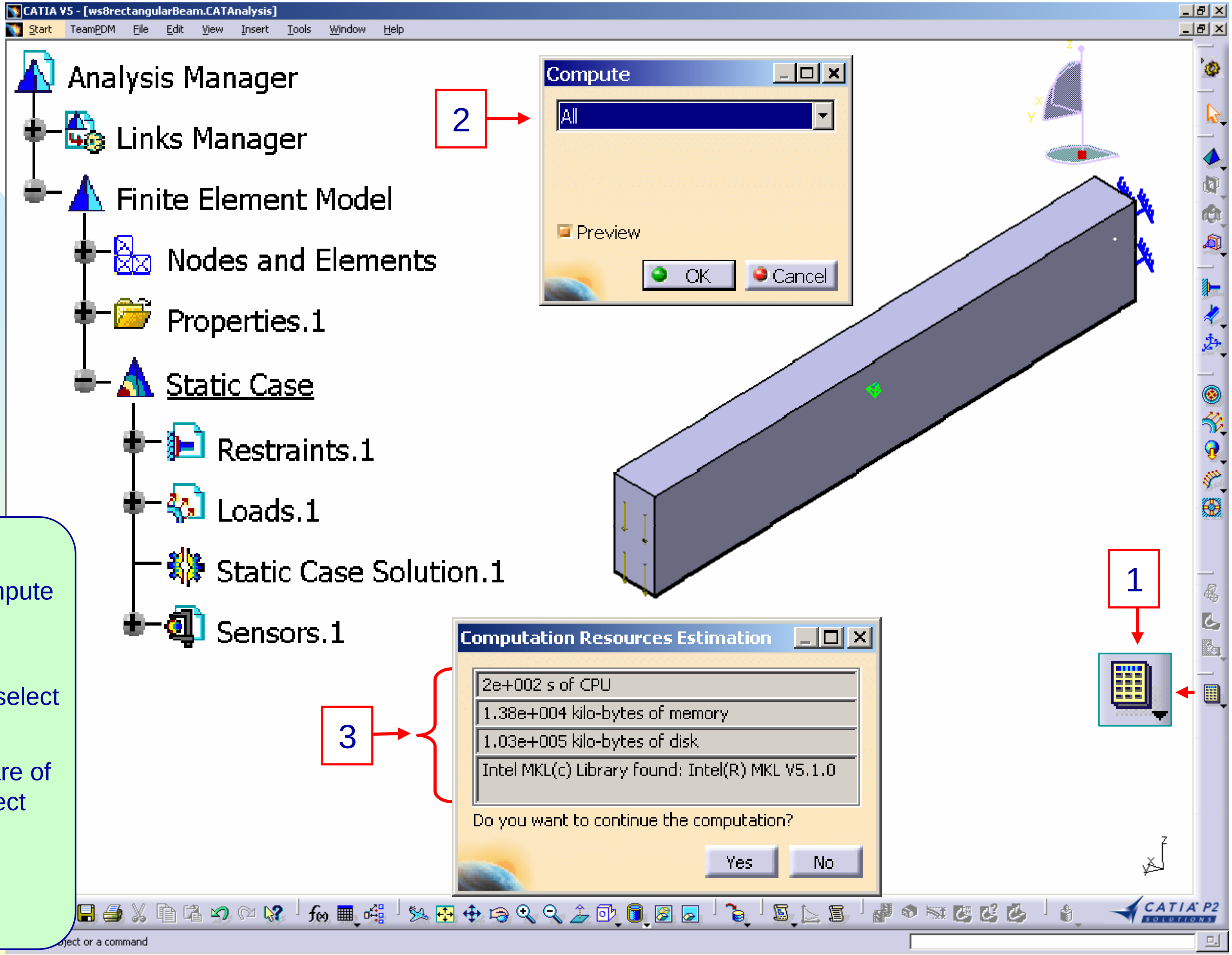
Redefine the global finite element mesh type.

Steps:

1. Double Click the "OCTREE Tetrahedron Mesh.1" representation in the features tree or the "Mesh" icon centered on the part.

2. Change element type to Parabolic, select OK.

Step 8. Compute the precise analysis



Steps:

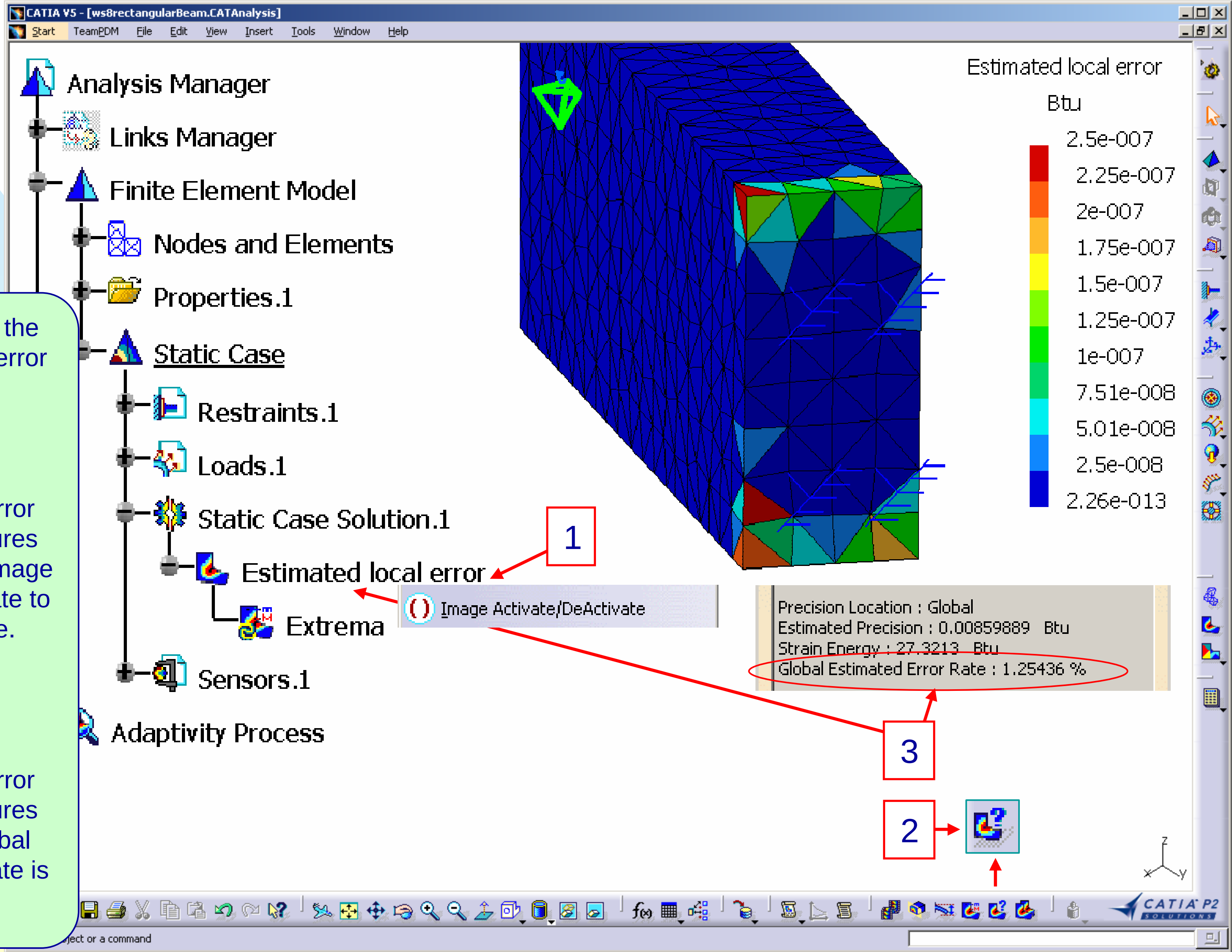
1. Select the Compute icon.

2. Compute All Objects defined, select OK.

3. Always be aware of these values, select Yes.

Save often.

Step 8. Compute the precise analysis



Check how much the global estimated error has improved

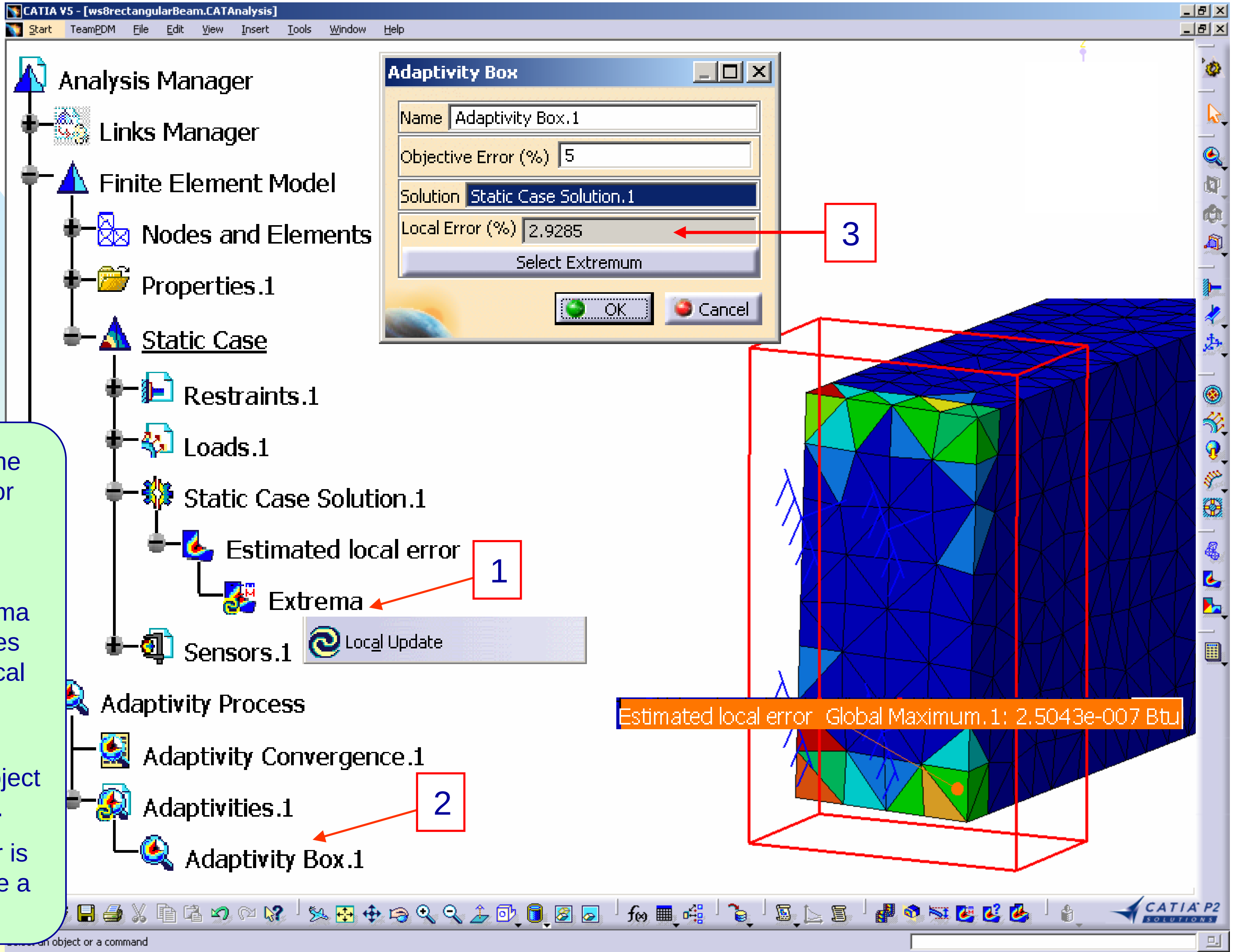
Steps:

1. Right click the Estimated local error object in the features tree then select Image Activate/DeActivate to activate the image.

2. Select on the information icon.

3. Select the Estimated local error object in the features tree. Note the global estimated error rate is very good.

Step 8. Compute the precise analysis



Check how much the local estimated error has improved.

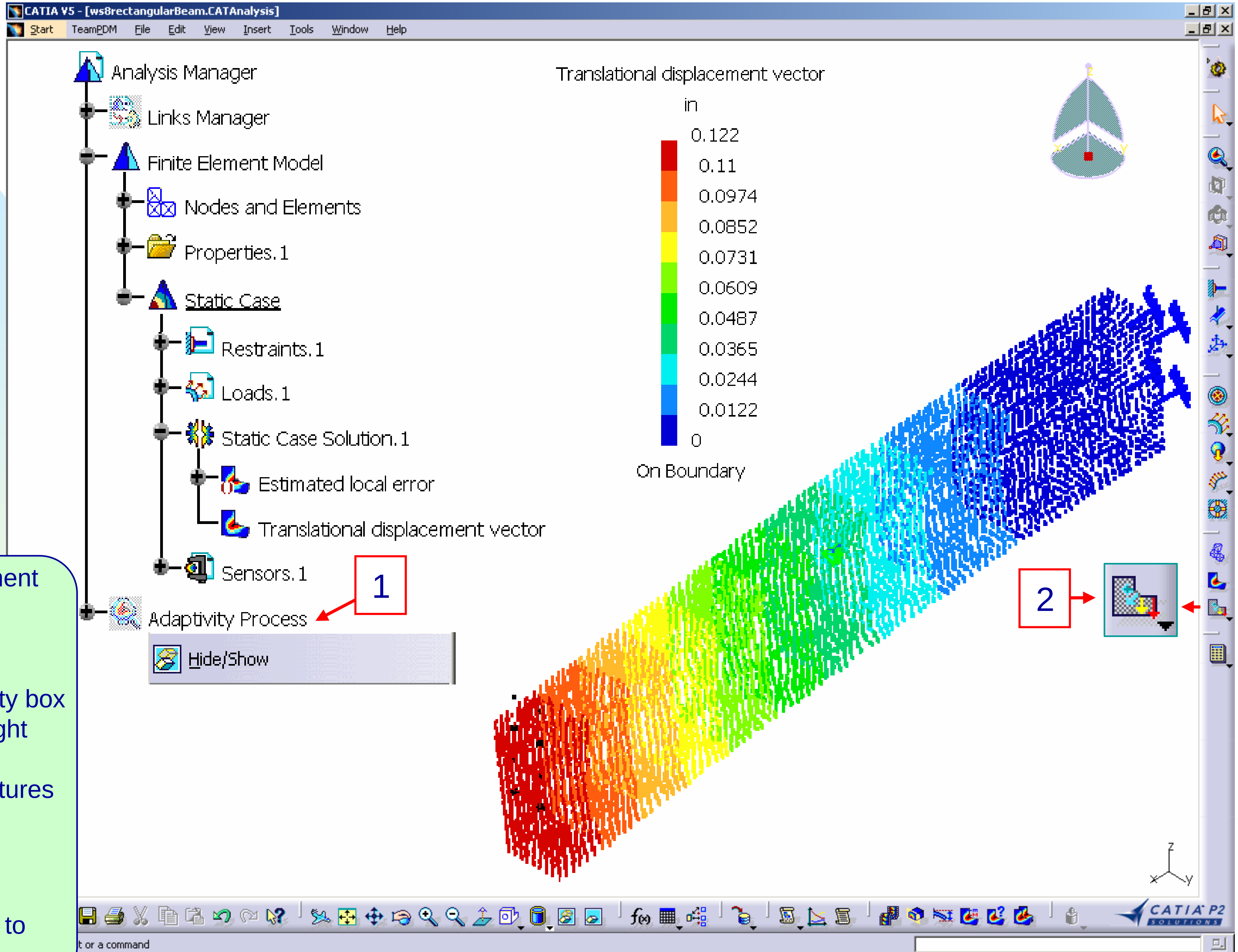
Steps:

1. Right click Extrema object in the features tree then select Local Update.

2. Double click the Adaptivity Box.1 object in the features tree.

3. Since local error is below 10% we have a precise model.

Step 9. Visualize final results



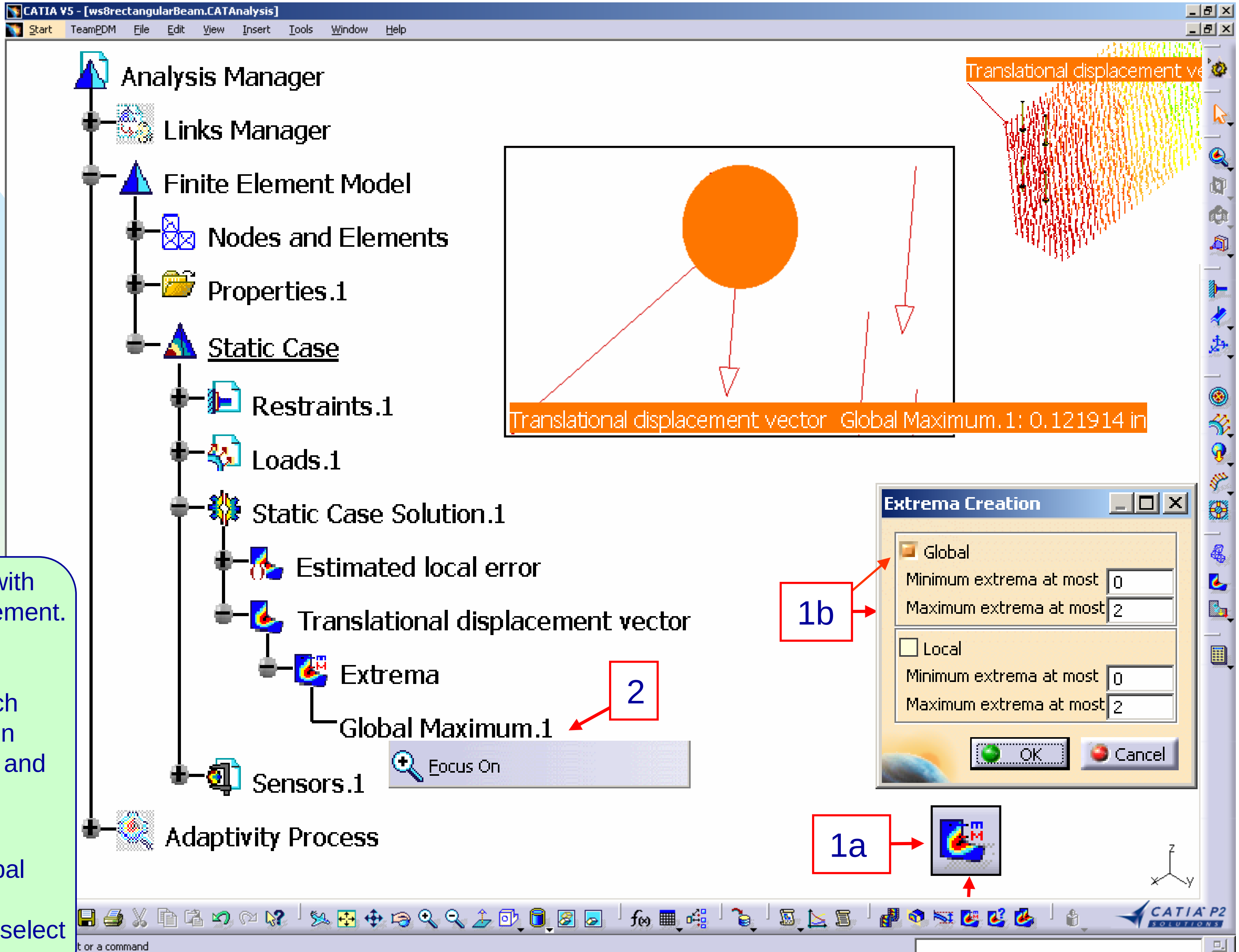
Add the displacement image.

Steps:

1. Put the adaptivity box into no show by right clicking Adaptivity Process in the features tree then select Hide/Show.

2. Select the displacement icon to add this image.

Step 9. Visualize final results



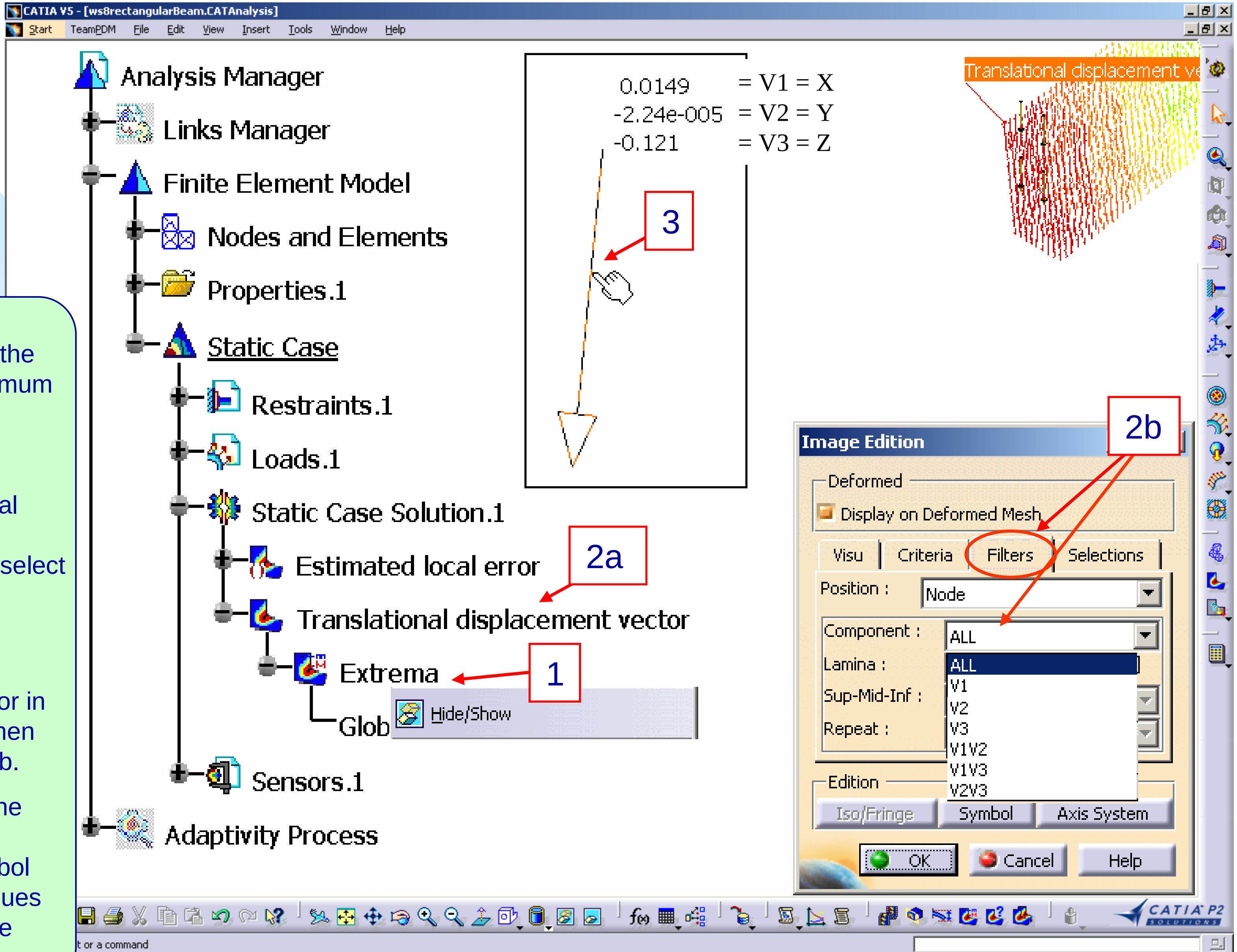
Find the element with maximum displacement.

Steps:

1. Select the search image extrema icon then select Global and key in 2 Maximum extrema at most.

2. Right click Global Maximum.1 in the features tree then select Focus On.

Step 9. Visualize final results



Find x, y, z displacements for the element with maximum displacement.

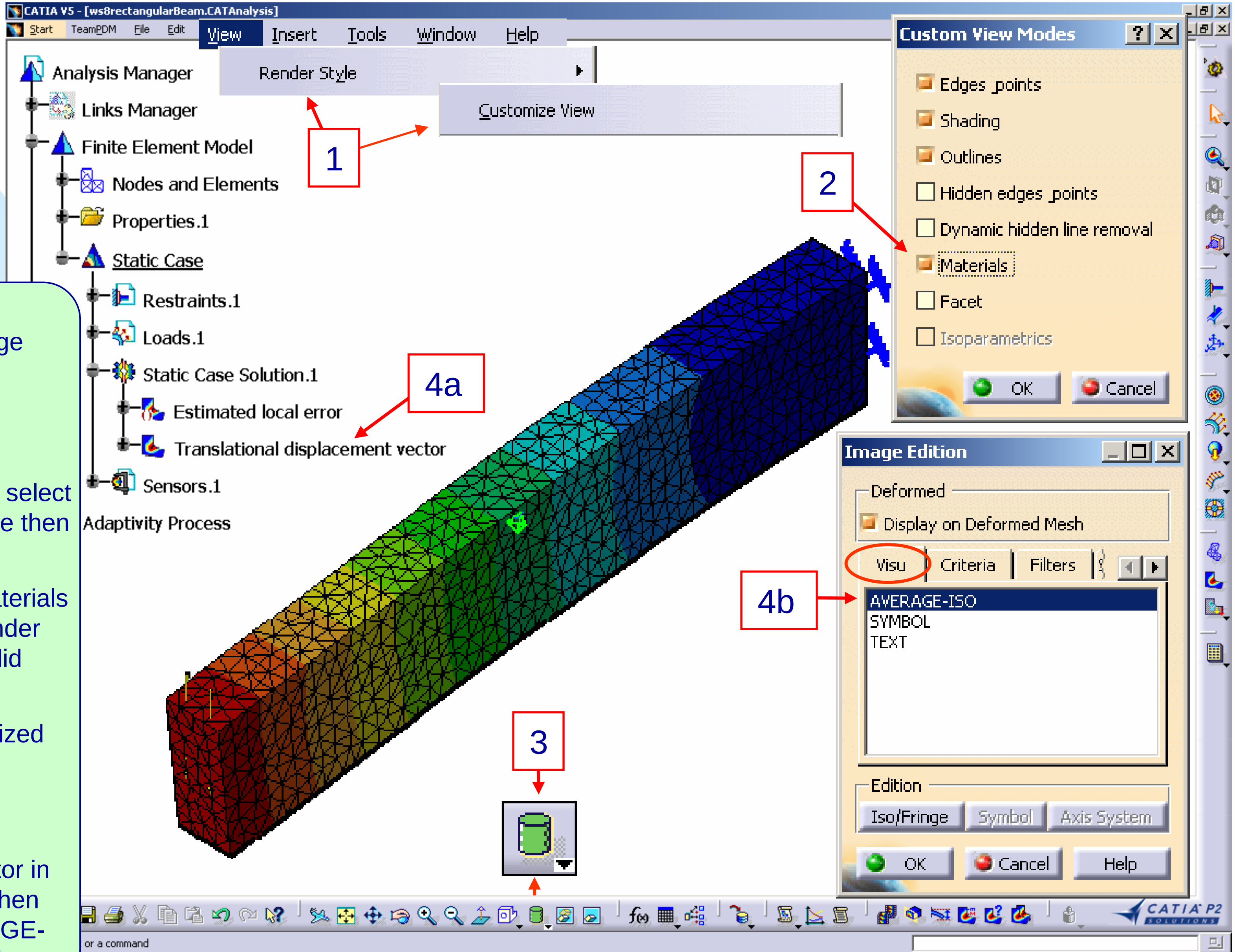
Steps:

1. Right click Global Maximum.1 in the features tree then select Hide/Show.

2. Double click Translational displacement vector in the features tree then select the filters tab.

3. By positioning the cursor on a displacement symbol the component values show relative to the current Filter.

Step 9. Visualize final results



Change the displacement image from symbols to Average-ISO

Steps:

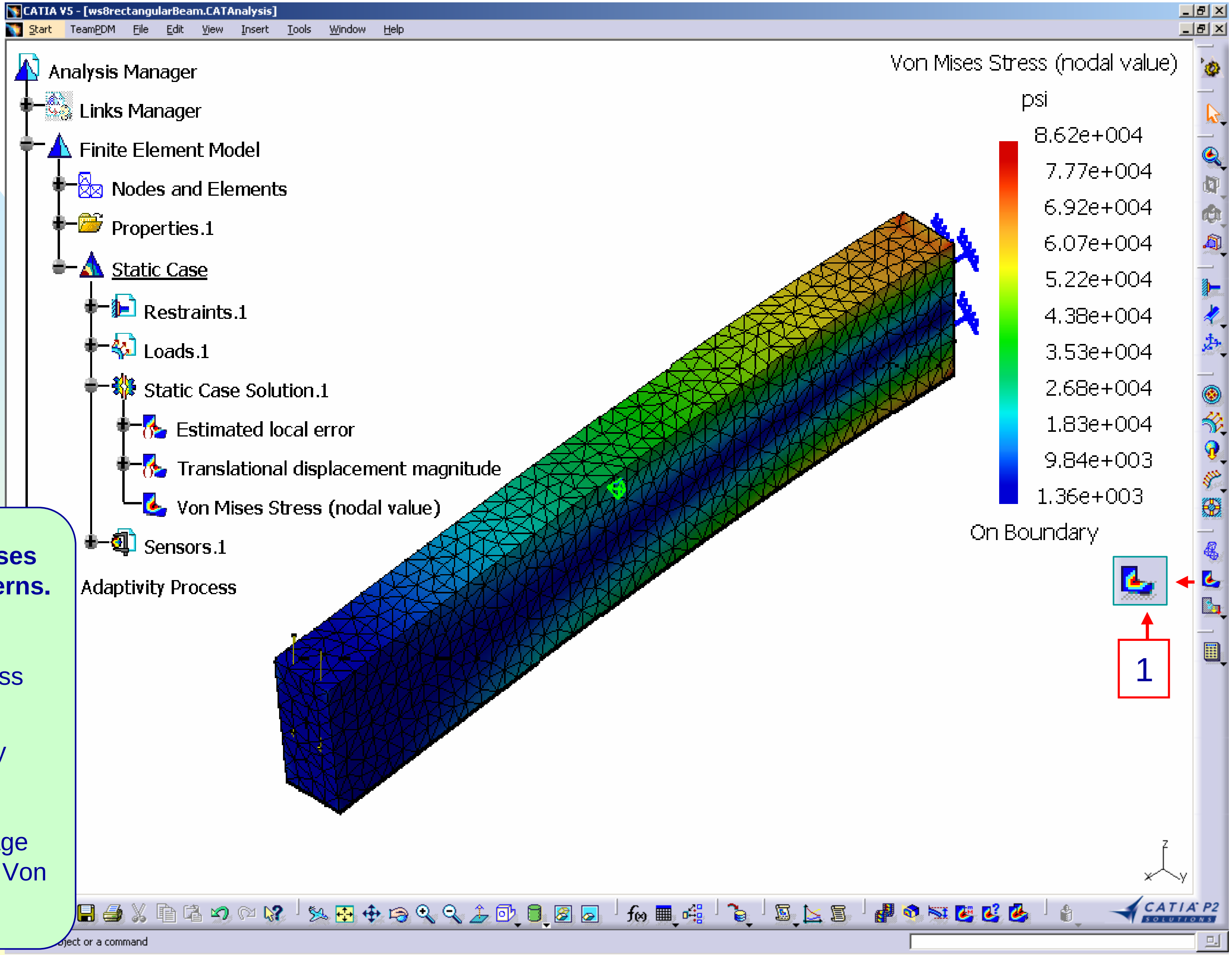
1. From the menu select View, Render Style then Customize View.

2. Click on the Materials box so we can render our image with solid colors.

3. Display customized view parameters.

4. Double click Translational displacement vector in the features tree then select the AVERAGE-ISO in the Visu tab.

Step 9. Visualize final results



Visualize **Von Mises stress field patterns.**

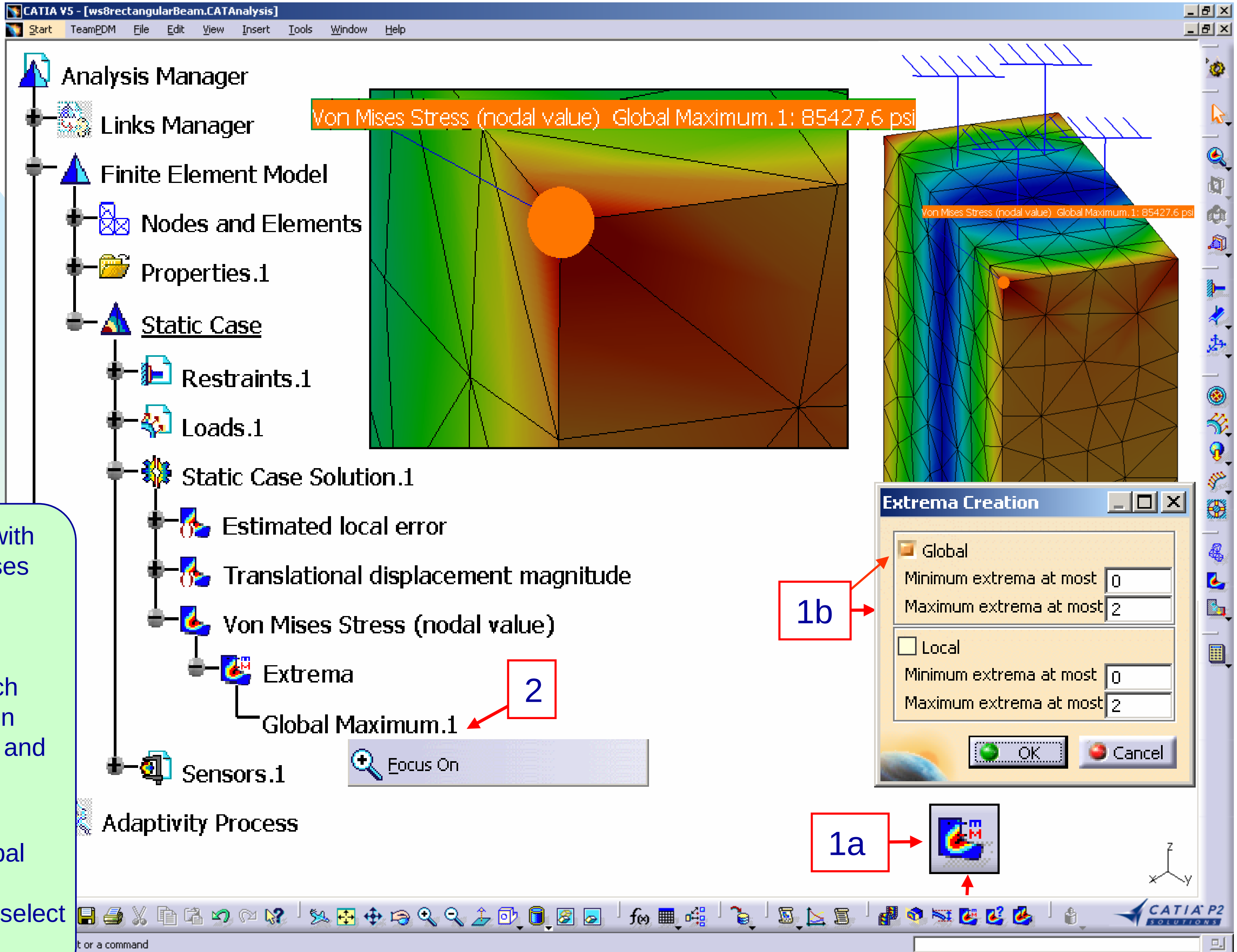
Steps:

1. Select the Stress Von Mises icon.

This automatically deactivates the Translational displacement image and activates the Von Mises image.

MSC SOFTWARE

Step 9. Visualize final results



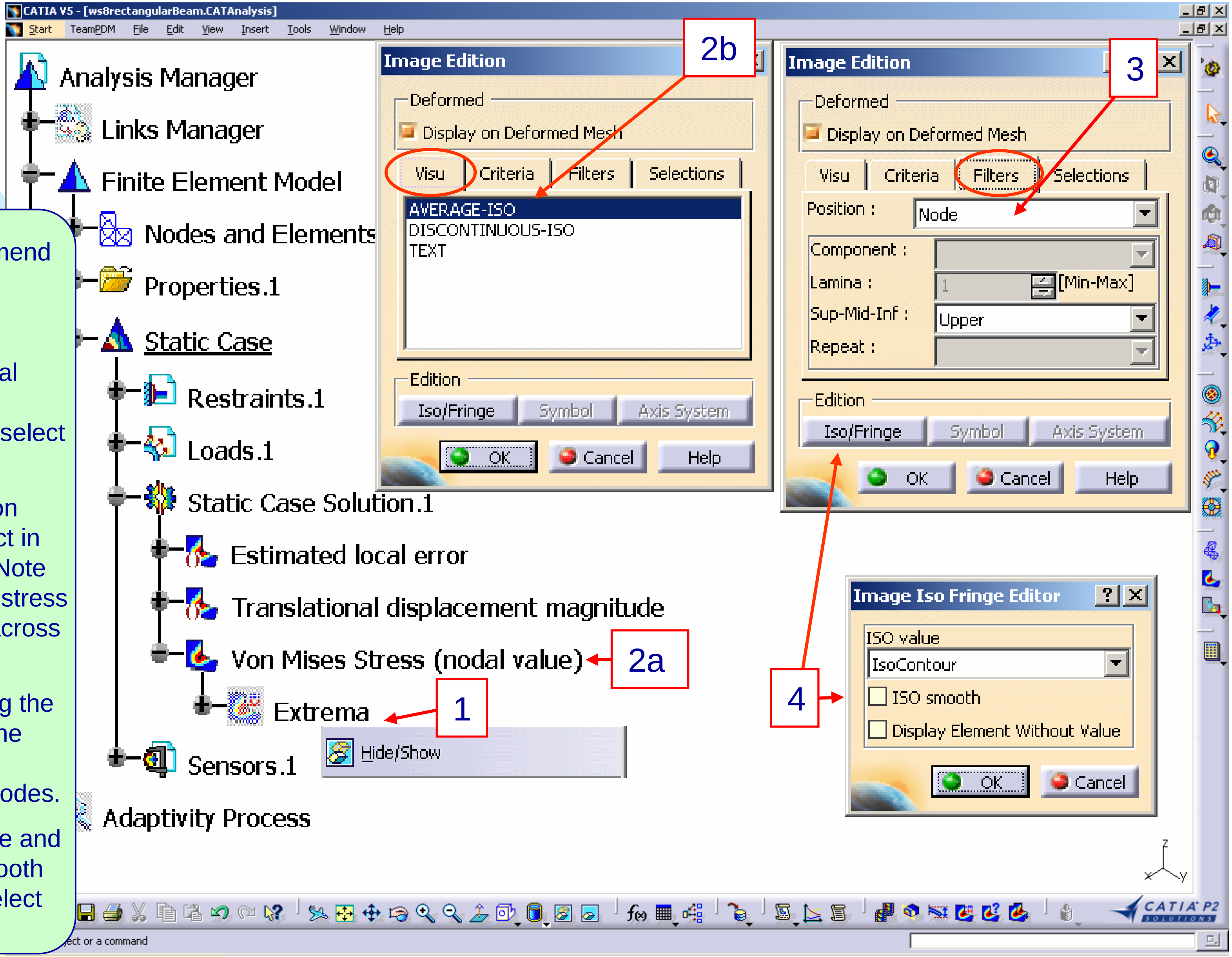
Find the element with maximum Von Mises Stress.

Steps:

1. Select the search image extrema icon then select Global and key in 2 Maximum extrema at most.

2. Right click Global Maximum.1 in the features tree then select Focus On.

Step 9. Visualize final results



Find exact recommend design stress.

Steps:

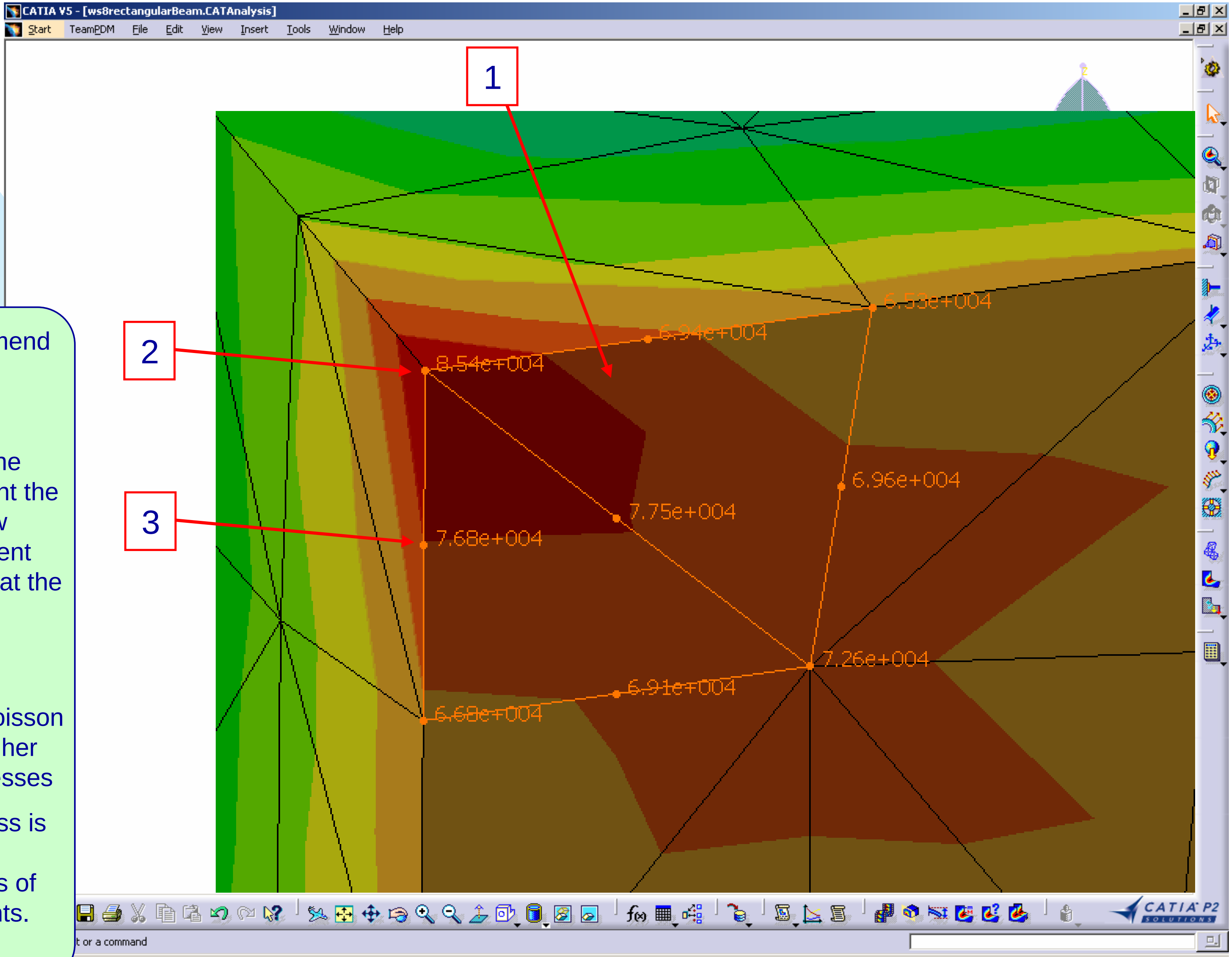
1. Right click Global Maximum.1 in the features tree then select Hide/Show.

2. Double click Von Mises Stress object in the features tree. Note you are looking at stress values averaged across elements.

3. Also by selecting the Filters tab notice the stress output is calculated at the nodes.

4. Select Iso/Fringe and select the ISO smooth box to turn it off select OK twice.

Step 9. Visualize final results



Find exact recommend design stress.

Steps:

1. By positioning the cursor on a element the stress values show relative to the current Filter (in this case at the nodes).

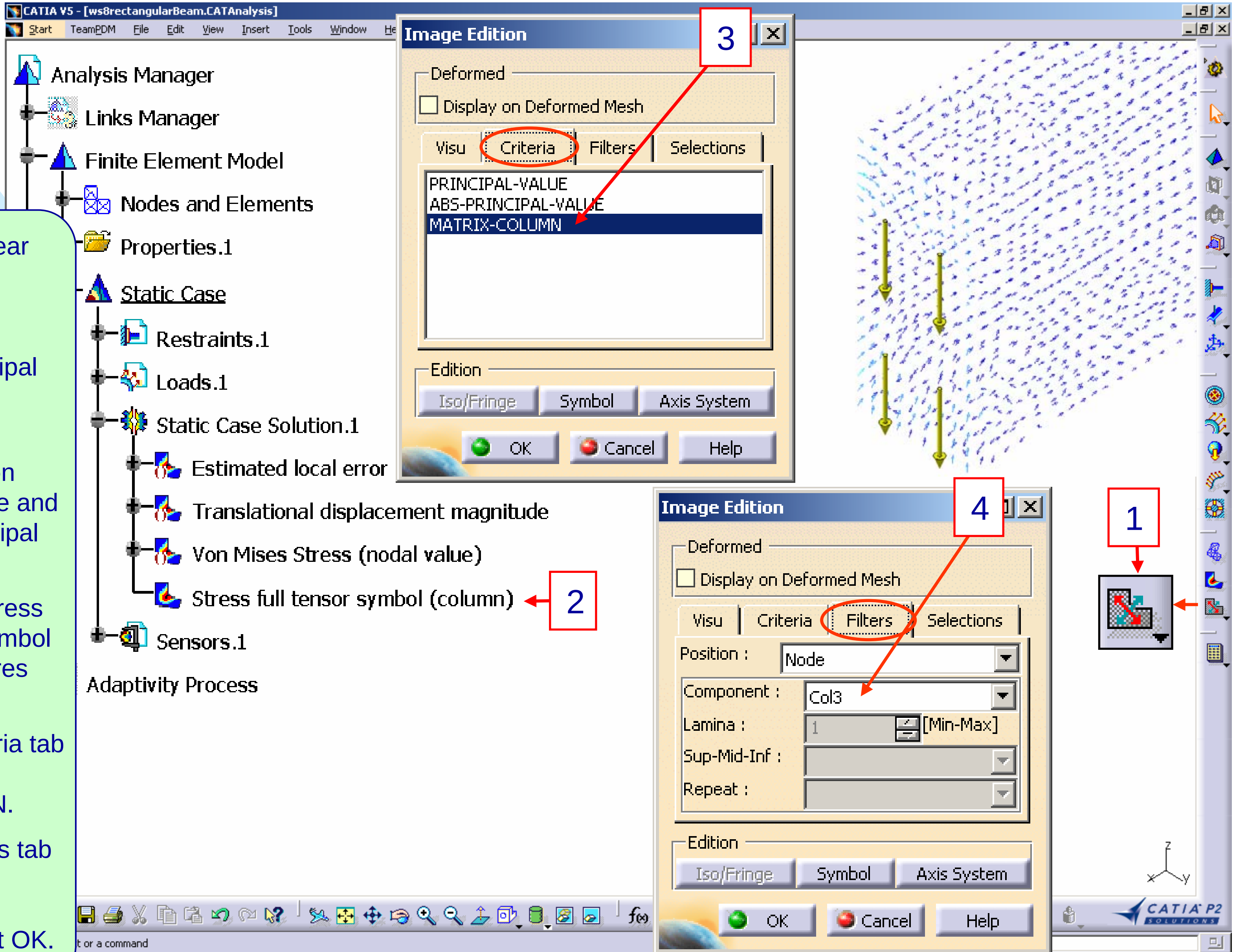
2. The maximum extrema stress is uninfluenced by poisson effects yielding higher than expected stresses

3. The design stress is found at the intermediate nodes of the bottom elements.

69400 – 76800 psi

MSC SOFTWARE

Step 9. Visualize final results



Find horizontal shear stress.

Steps:

1. Select the Principal Stress icon.

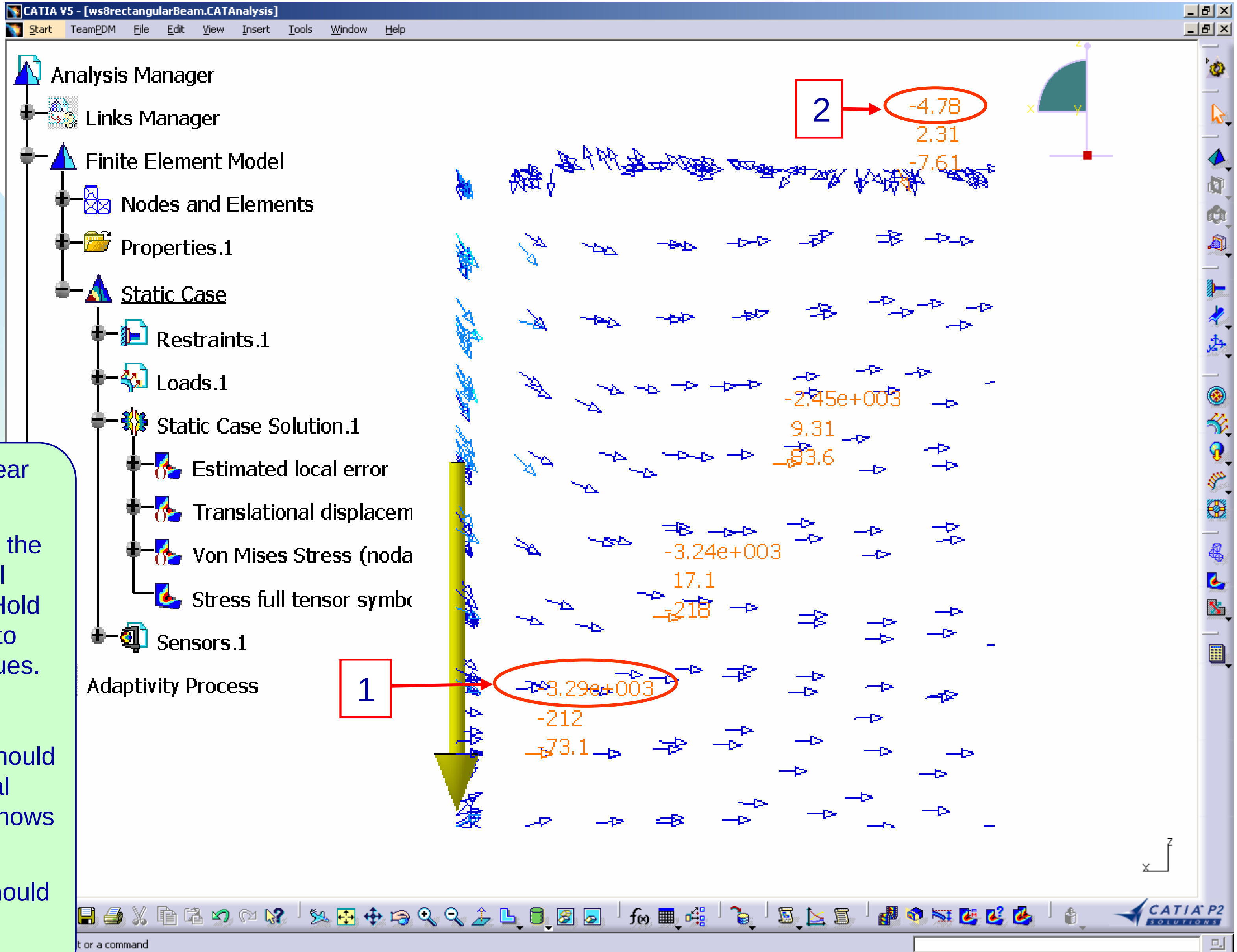
This automatically deactivates the Von Mises stress image and activates the Principal Stress image.

2. Double click Stress principal tensor symbol object in the features tree.

3. Select the Criteria tab and then select MATRIX-COLUMN.

4. Select the Filters tab and with the arrow select the Col3 Component, select OK.

Step 9. Visualize final results



Find horizontal shear stress.

Hold the cursor on the tensor symbols will show the values. Hold the Ctrl key down to select multiple values.

Steps:

1. Highest value should occur at the neutral axis. This model shows 3290 psi

2. Lowest value should occur on the outer edges.

Conclusions

- CATIA V5 GSA workbench is validated for a rectangular cantilever beam scenario. To be conservative, increase material strength to a minimum yield of 77000 psi for the described load case.

	Hand Calculations	.25” Parabolic Global Mesh, .025” sag
Global % Precision error Local % Precision error	NA NA	1.25 % 2.93 %
Error Estimate	NA	2.5e-7 Btu global
Translational Displacement	-0.119 inch	-0.121 inch (Z - direction)
Max Von Mises Stress	72000 psi	69400 - 76800 psi
Horizontal Shear Stress	3000 psi	3290 psi

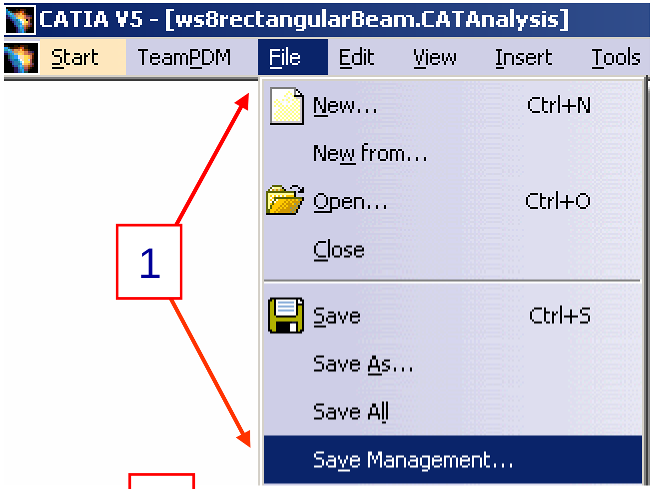
Step 10. Save the analysis document

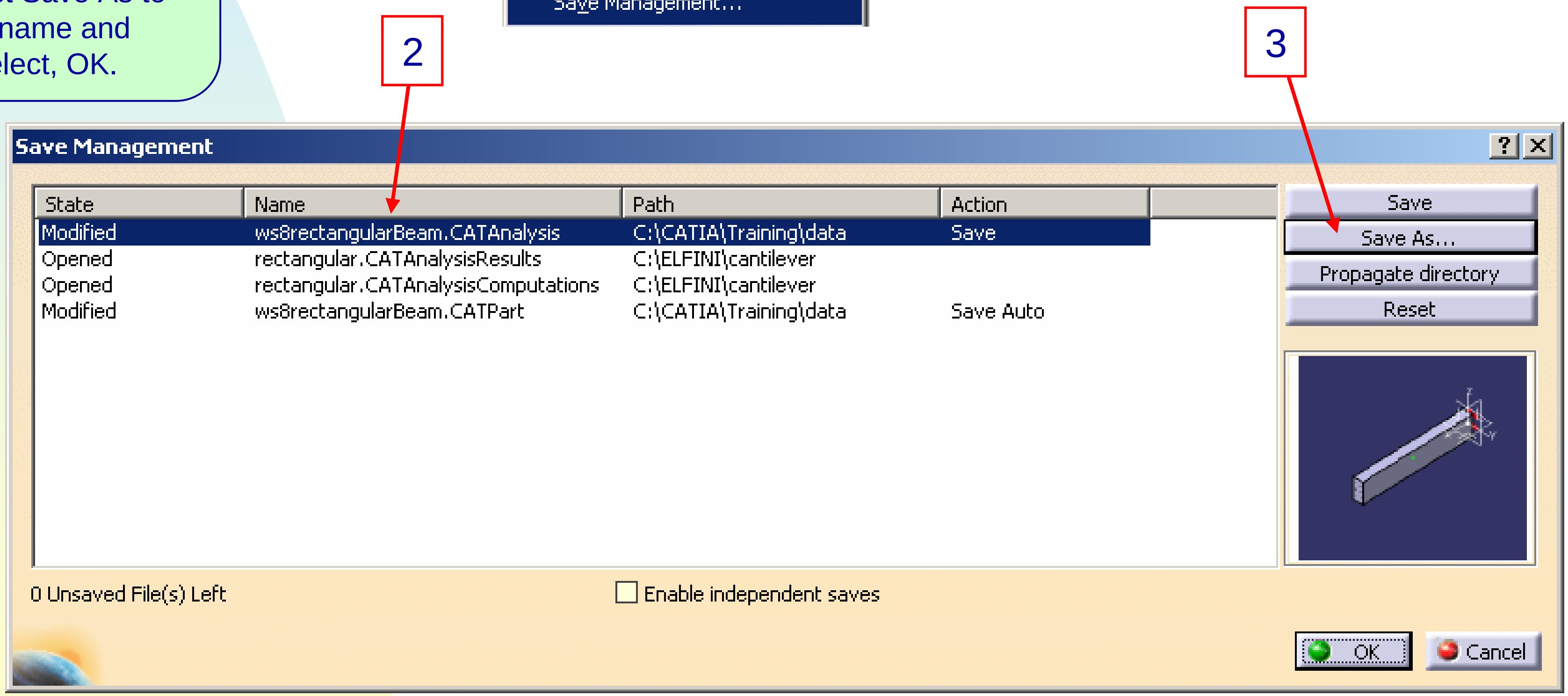
Steps:

1. Select Save Management from the File menu.

2. Highlight document you want to save.

3. Select Save As to specify name and path, select, OK.





List of Symbols and Definitions

- Greek letters.

α =	Angular acceleration (radians/sec/sec); included angle of beam curvature (degrees); form factor.
δ =	Perpendicular deflection (in.), bending (b) or shear (s).
ε =	Unit strain, elongation or contraction (in./in.)
ε_s =	Unit shear strain (in./in.).
ν =	Poisson's ratio (aluminum = .346 usually, steel = .266 usually); unit shear force.
ϕ =	Unit angular twist (radians/linear inch); included angle; angle of rotation.
σ =	Normal stress, tensile or compressive (psi); strength (psi).
σ_b =	Bending stress (psi).
σ_y =	Yield strength (psi).
τ =	Shear stress (psi); shear strength (psi).
θ =	Angle of twist (radians; 1 radian = 57.3 degrees); angle of rotation (radians); slope of tapered beam; any specified angle.

List of Symbols and Definitions

- Letters.

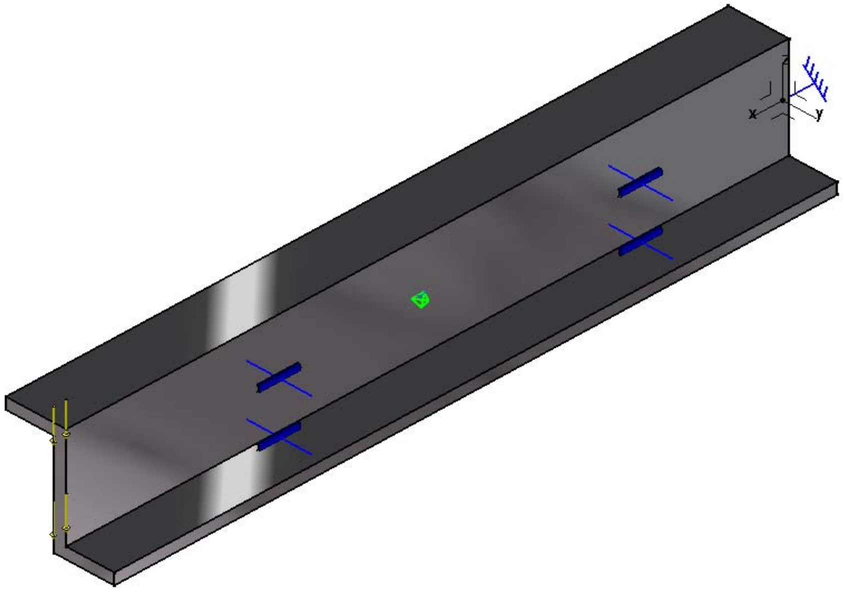
a = area of section where stress is desired or applied (in2)
b = width of section (in)
c = distance from neutral axis to extreme fiber (in)
d = depth of section (in)
e = eccentricity of applied load (in)
f = force per linear inch (in)
g = acceleration of gravity (386.4 inch/sec2)
h = height (in)
k = any specified constant or amplification factor
m = mass
n = distance of section’s neutral axis from ref axis (in)
p = internal pressure (psi)
r = radius (in); radius of gyration
t = thickness of section (in)
w = uniformly distributed load (lbs/linear inch)
y = distance of area’s center of gravity to neutral axis of entire section (in)

A = area (in2); total area of cross-section
E = modulus of elasticity, tension (psi)
F = total force (lbs); radial force (lbs)
I = moment of inertia (in4)
J = polar moment of inertia (in4)
L = length of member (in)
M = bending moment (in-lbs)
P = concentrated load (lbs)
Q = shear center
R = reaction (lbs)
S = section modulus (in3) = I/c
T = torque or twisting moment (in-lbs
V = vertical shear load (lbs)
W = total load (lbs); weight (lbs)

C.G. = center of gravity
D.O.F = degrees of freedom
N.A. = neutral axis

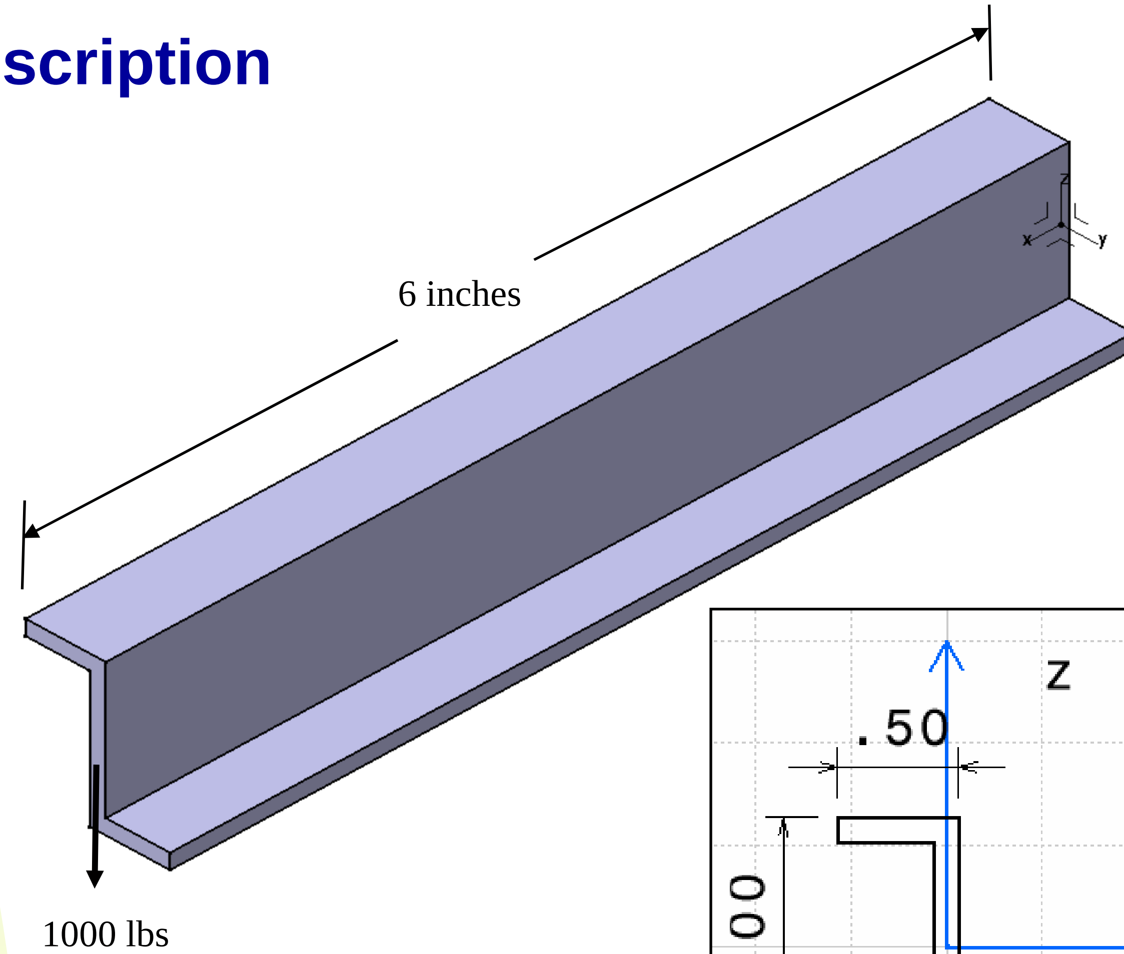
WORKSHOP 8b

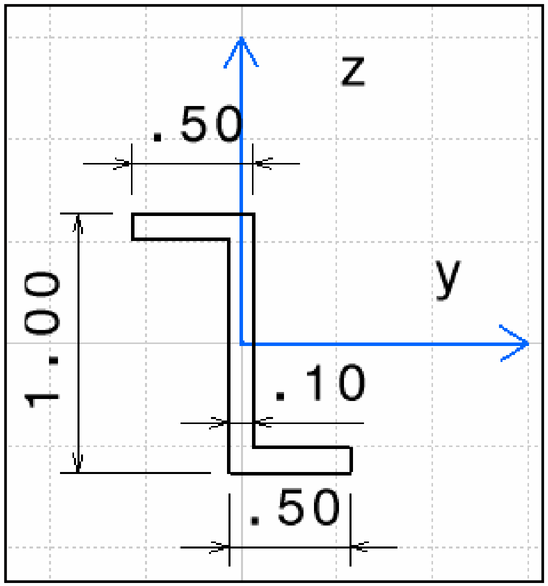
Z-SECTION CANTILEVER BEAM



Problem Description

- Load case.





Material: Aluminum
Young Modulus = 10.15e6 psi
Shear Modulus = 3.77e6 psi
Poisson Ratio = .346
Density = .098 lb_in3
Yield Strength = 13778 psi

Problem Description

- Bending and shear displacement

$$\text{Moment of Inertia} = I_y = \frac{B \cdot D^3 - ((D - (2 \cdot T))^3 \cdot A)}{12} = \frac{0.5in \cdot 1.0in^3 - ((1.0in - (2.0 \cdot 0.1in))^3 \cdot 0.4in)}{12} = 0.0246\ \text{inch}^4$$

$$\text{Bending Displacement} = \partial = \frac{P \cdot L^3}{(3 \cdot E \cdot I)} = \frac{1000\ \text{lbs} \cdot 6^3\ \text{inch}}{(3 \cdot 10.15e^6\ \text{psi} \cdot 0.0246\ \text{inch}^4)} = 0.288\ \text{inch}$$

$$\text{Shear Displacement} = \partial = \frac{P \cdot L}{(E_s \cdot A)} = \frac{1000\ \text{lbs} \cdot 6\ \text{inch}}{(3.77e^6\ \text{psi} \cdot 0.1\ \text{inch}^2)} = 0.0159\ \text{inch}$$

$$\partial_{Combined} = 0.288in + 0.0159in = 0.304in$$

- Bending stress

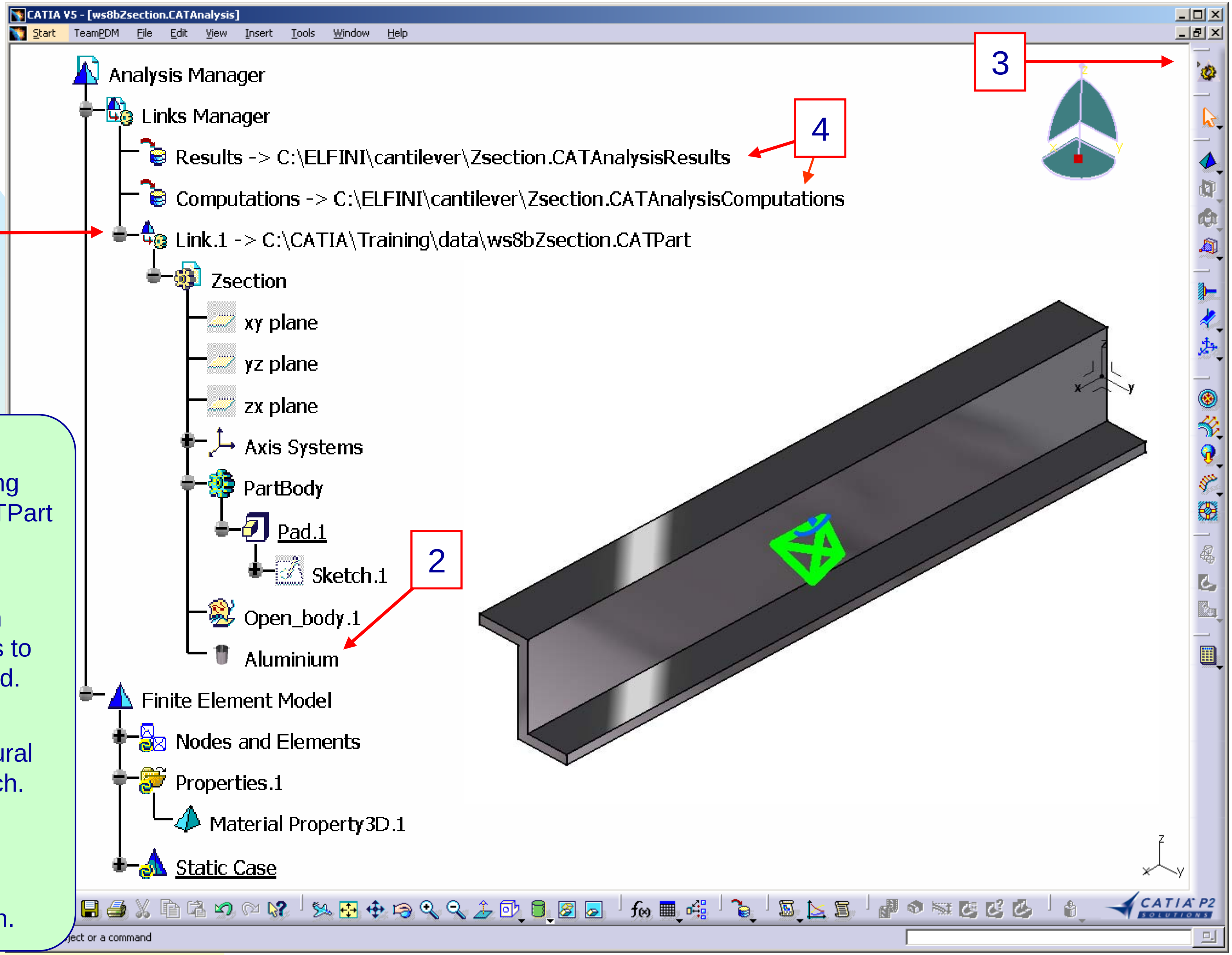
$$\text{Bending Moment} = M_y = P \cdot L = 1000\ \text{lbs} \cdot 6\ \text{inch} = 6000\ \text{in lbs}$$

$$\text{Maximum Bending Stress} = \sigma_b = \frac{M_y \cdot y}{I_y} = \frac{6000\ \text{in lbs} \cdot 0.5\ \text{inch}}{0.0246\ \text{inch}^4} = 122000\ \text{psi}$$

Suggested Exercise Steps

1. Create a new CATIA analysis document (.CATAnalysis).
2. Mesh globally with linear elements.
3. Apply a clamp restraint.
4. Apply a distributed force.
5. Compute the initial analysis.
6. Check global and local precision (animate deformation, adaptive boxes and extremas).
7. Change mesh to parabolic, possibly add local meshing.
8. Compute the precise analysis.
9. Visualize final results.
10. Save the analysis document.

Step 1. Create a new CATIA analysis document



Steps:

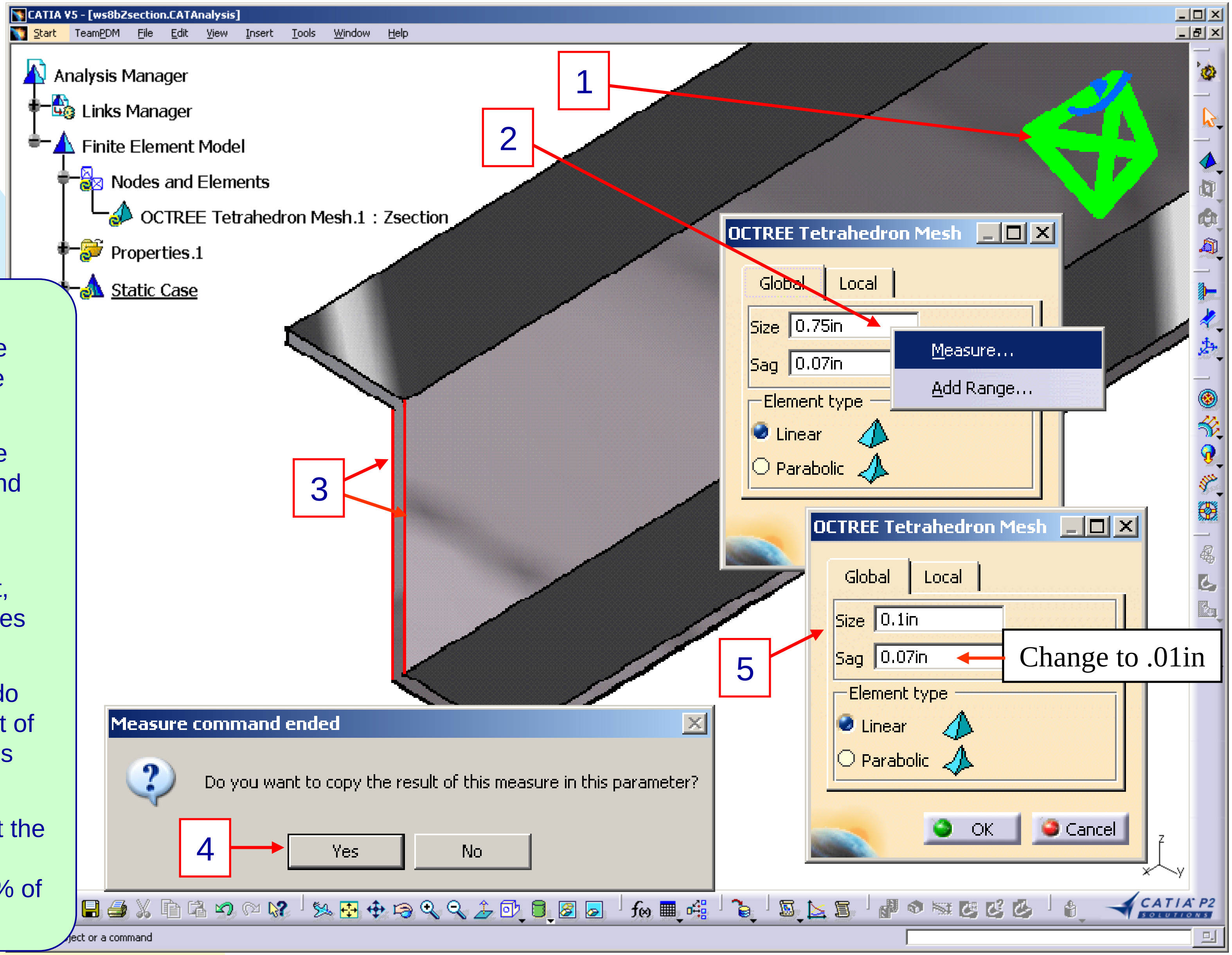
1. Open the existing ws8bZsection.CATPart from the training directory.

2. Apply aluminum material properties to the part as required.

3. Launch the Generative Structural Analysis workbench.

4. Specify the Computations and Results storage locations as shown.

Step 2. Mesh globally with linear elements



Steps:

1. Double Click the "Mesh" icon on the part.

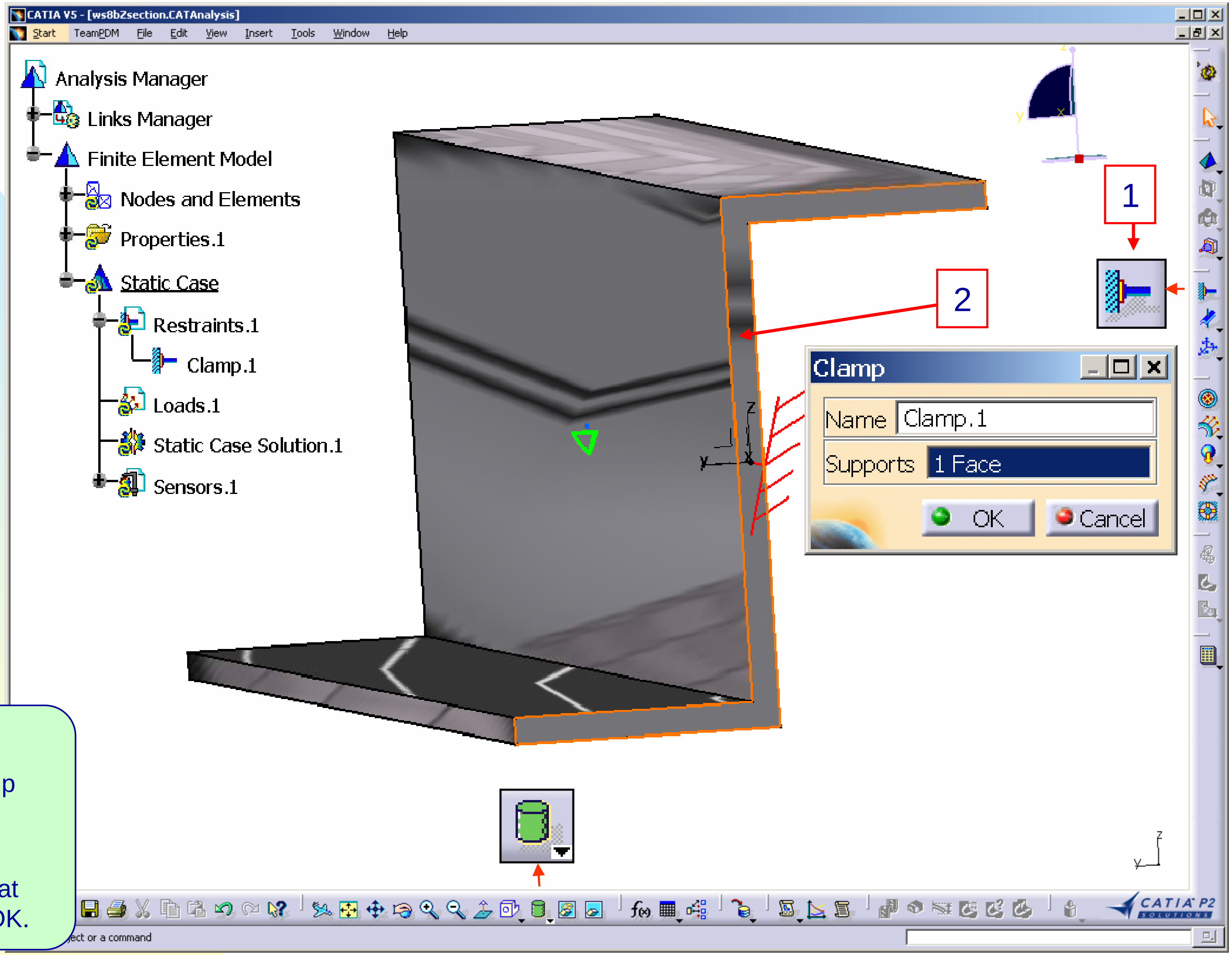
2. Right click in the Global Size box and select Measure.

3. Note measure between is current, select the two edges indicated.

4. Click Yes, you do want to copy result of this measure in this parameter.

5. Result, also edit the Sag, key in the recommended 10% of mesh size (.01in).

Step 3. Apply a clamp restraint

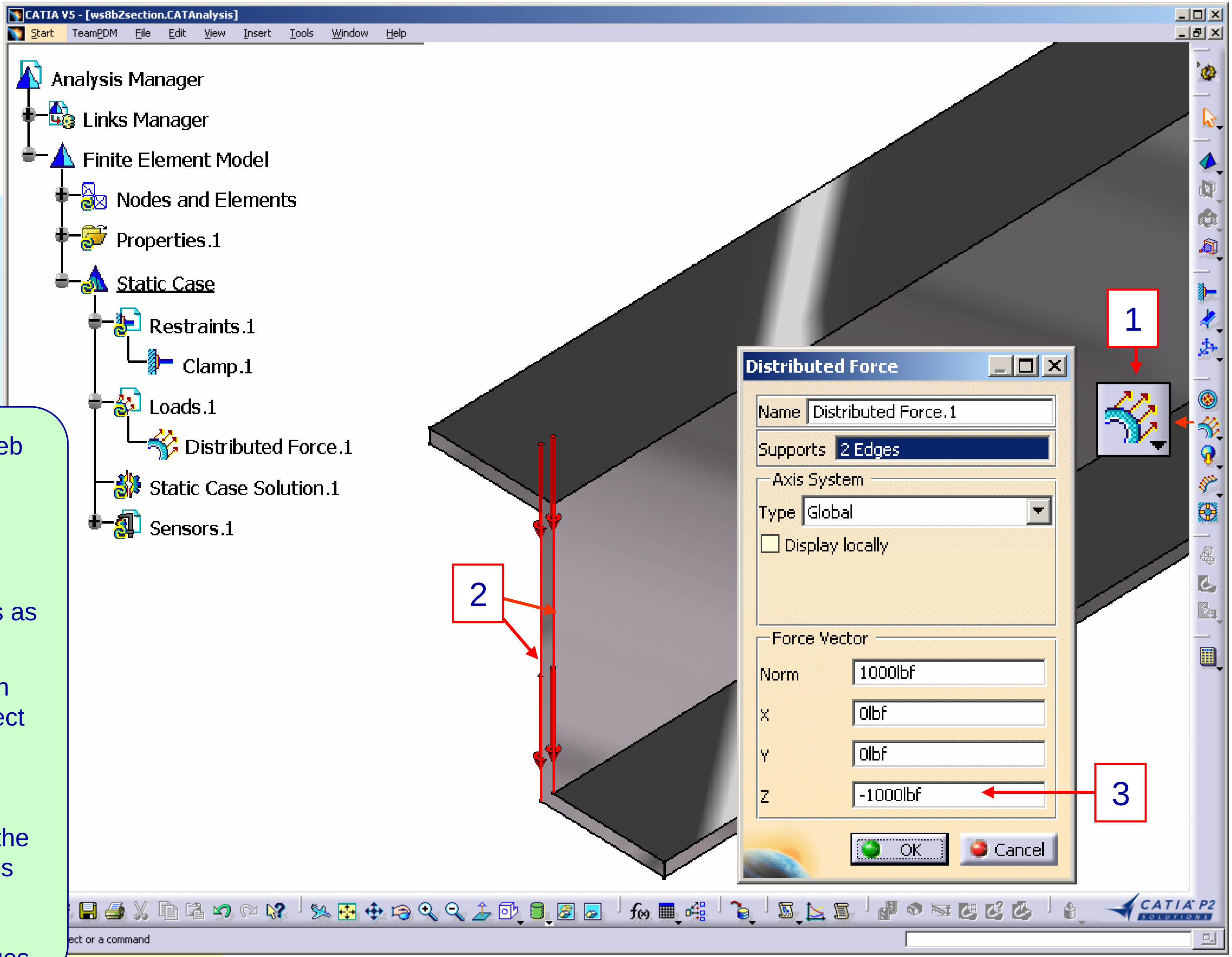


Steps:

1. Select the Clamp Restraint icon.

2. Select the face at the origin, select OK.

Step 4. Apply a distributed force



Load only on the web

Steps:

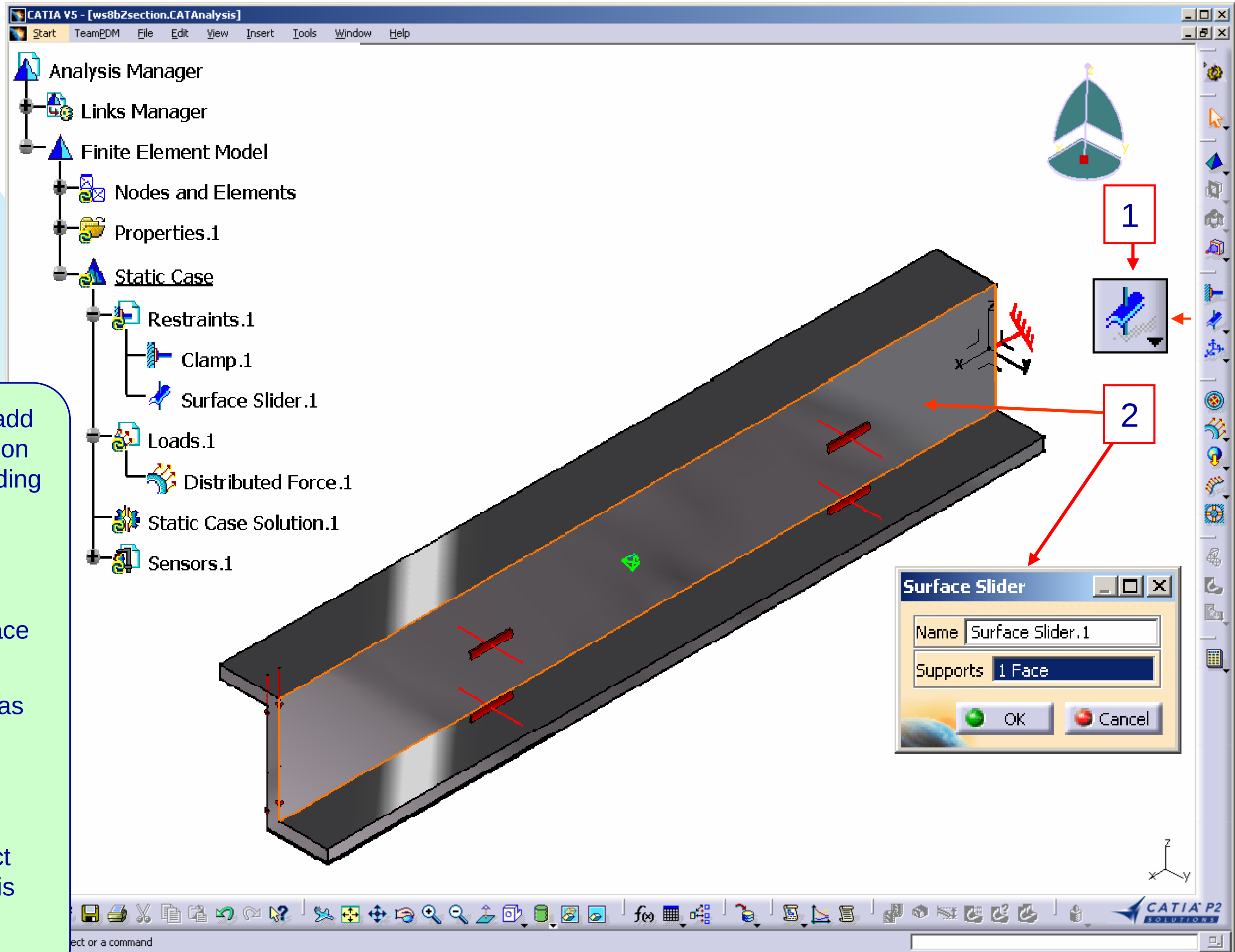
1. Select the Force icon.

2. Select two edges as shown.

3. Enter -1000 lbs in the Z-direction, select OK.

If you try selecting the face the whole cross section will select, causing inaccurate loading on the flanges.

Step 4. Apply a distributed force



It is necessary to add a boundary condition that forces all bending to occur in the x-z plane.

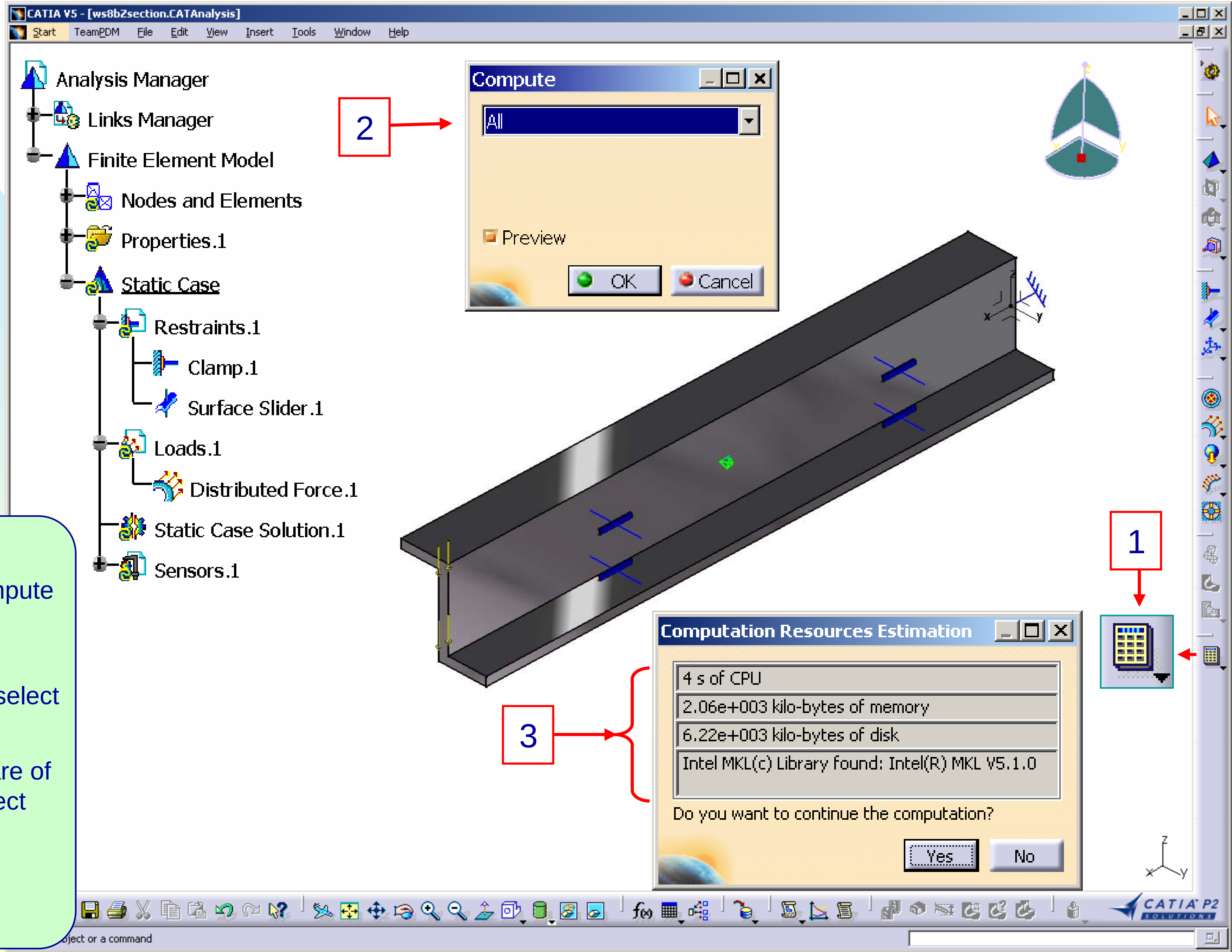
Steps:

1. Select the Surface Slider icon.

2. Select the face as shown.

Unsymmetrical sections will deflect laterally without this Surface Slider restraint.

Step 5. Compute the initial analysis



Steps:

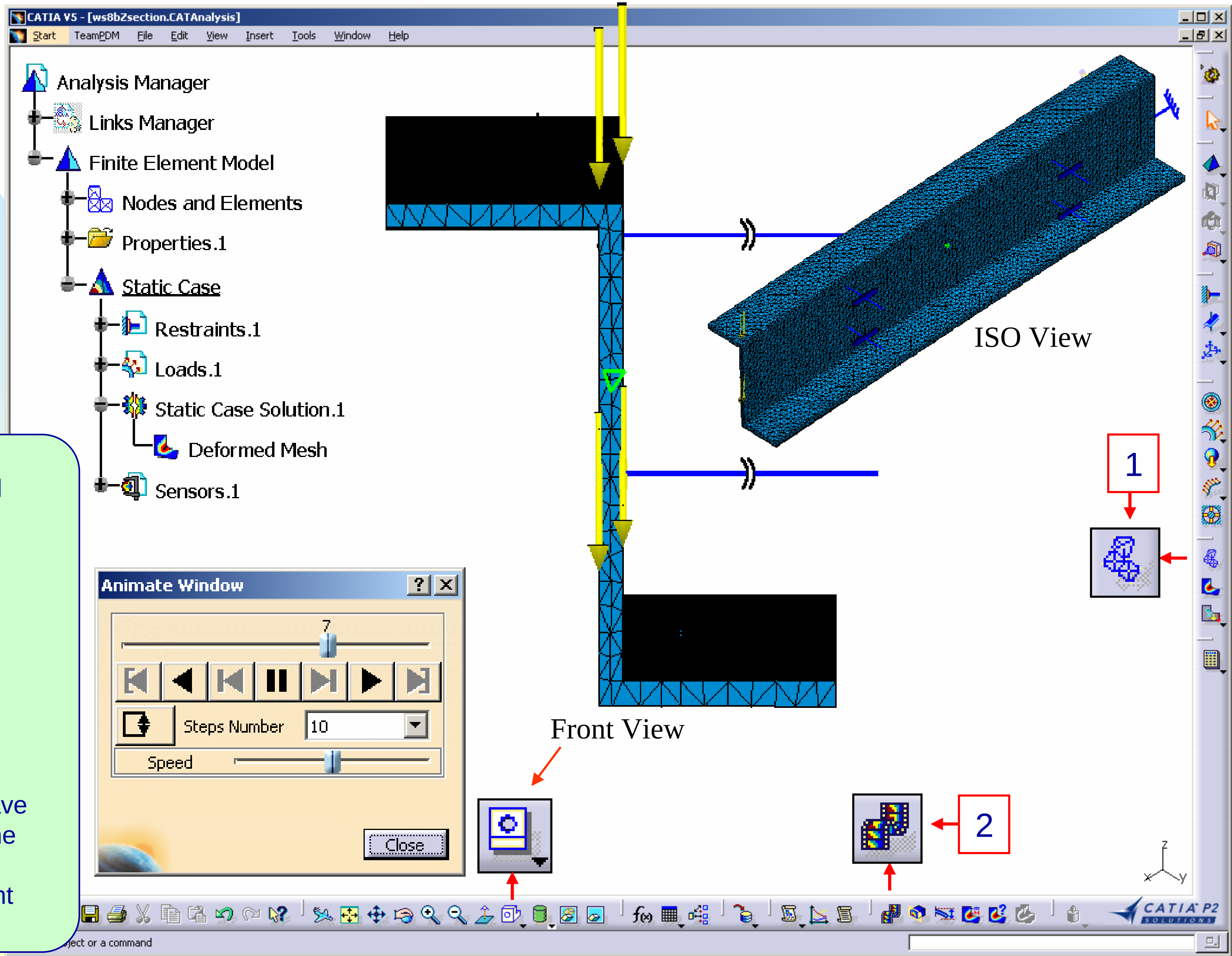
1. Select the Compute icon.

2. Compute All Objects defined, select OK.

3. Always be aware of these values, select Yes.

Save often.

Step 6. Check global and local precision



Visualize the **Deformation** and animate.

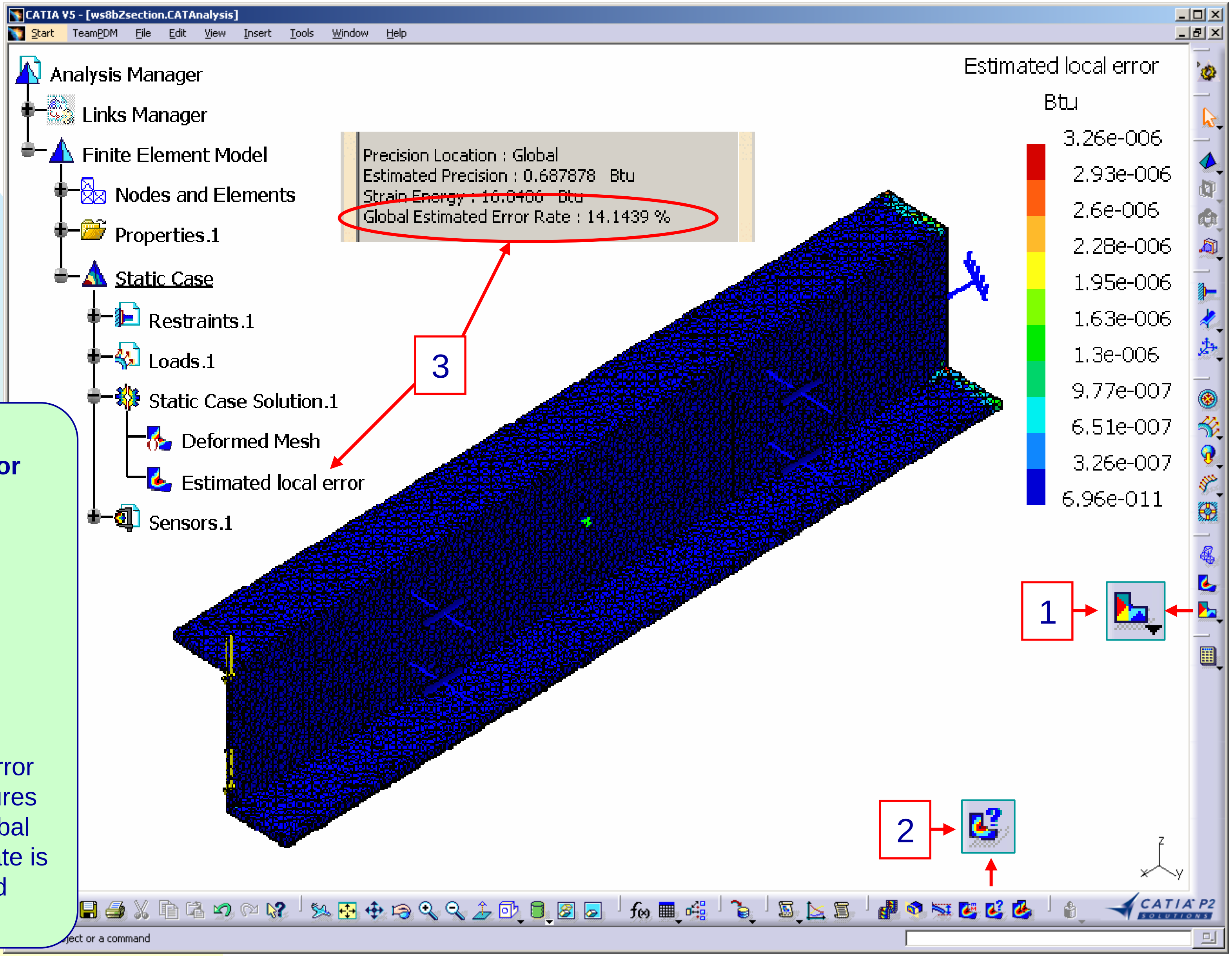
Steps:

1. Select the Deformation icon.

2. Select on the Animate icon.

Verify that you have no deflection in the y-direction by animating the front view.

Step 6. Check global and local precision



Visualize the **computation error map.**

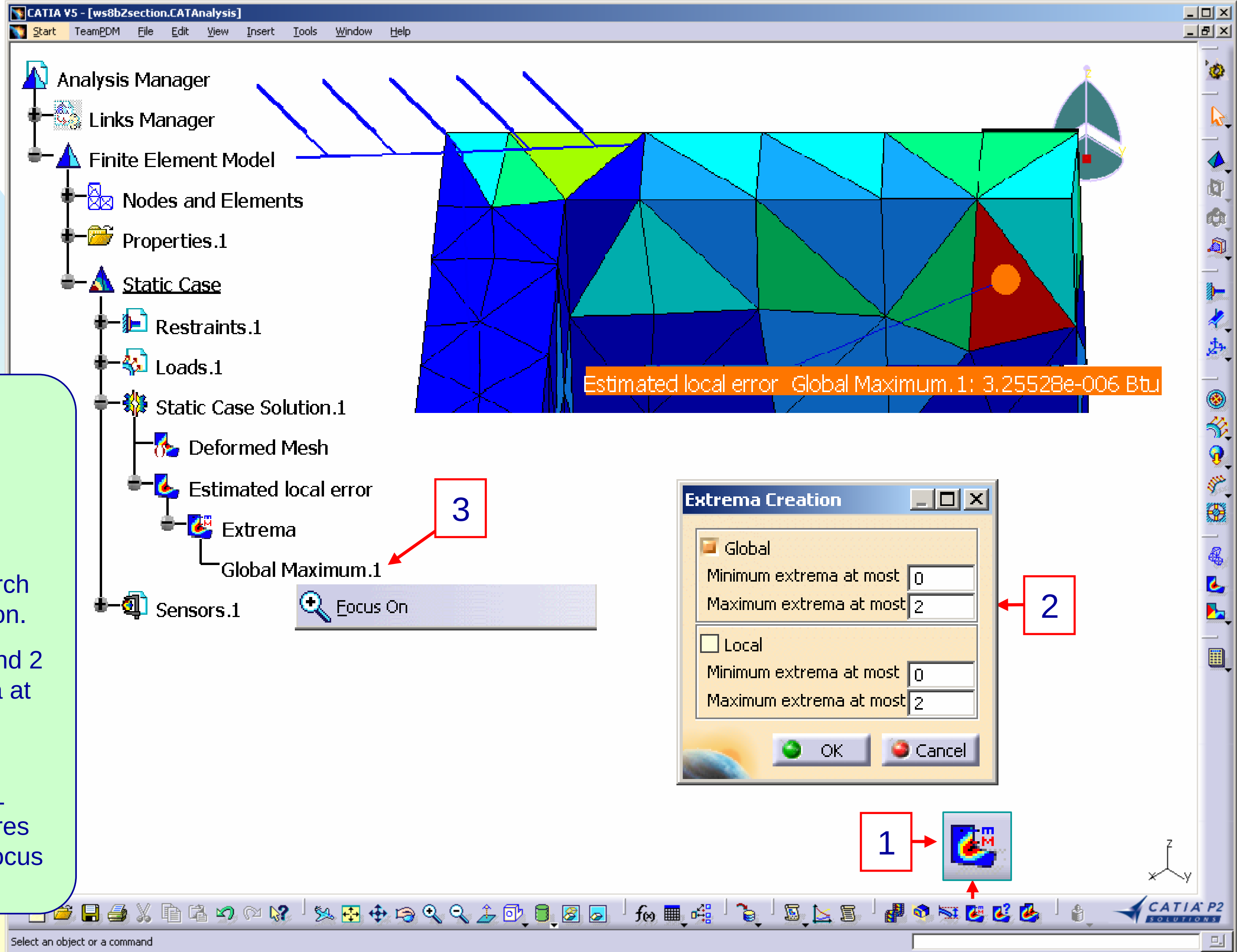
Steps:

1. Select the Precision icon.

2. Select on the information icon.

3. Select the Estimated local error object in the features tree. Note the global estimated error rate is good (recommend max 20%).

Step 6. Check global and local precision



Find the global element with the highest estimated error.

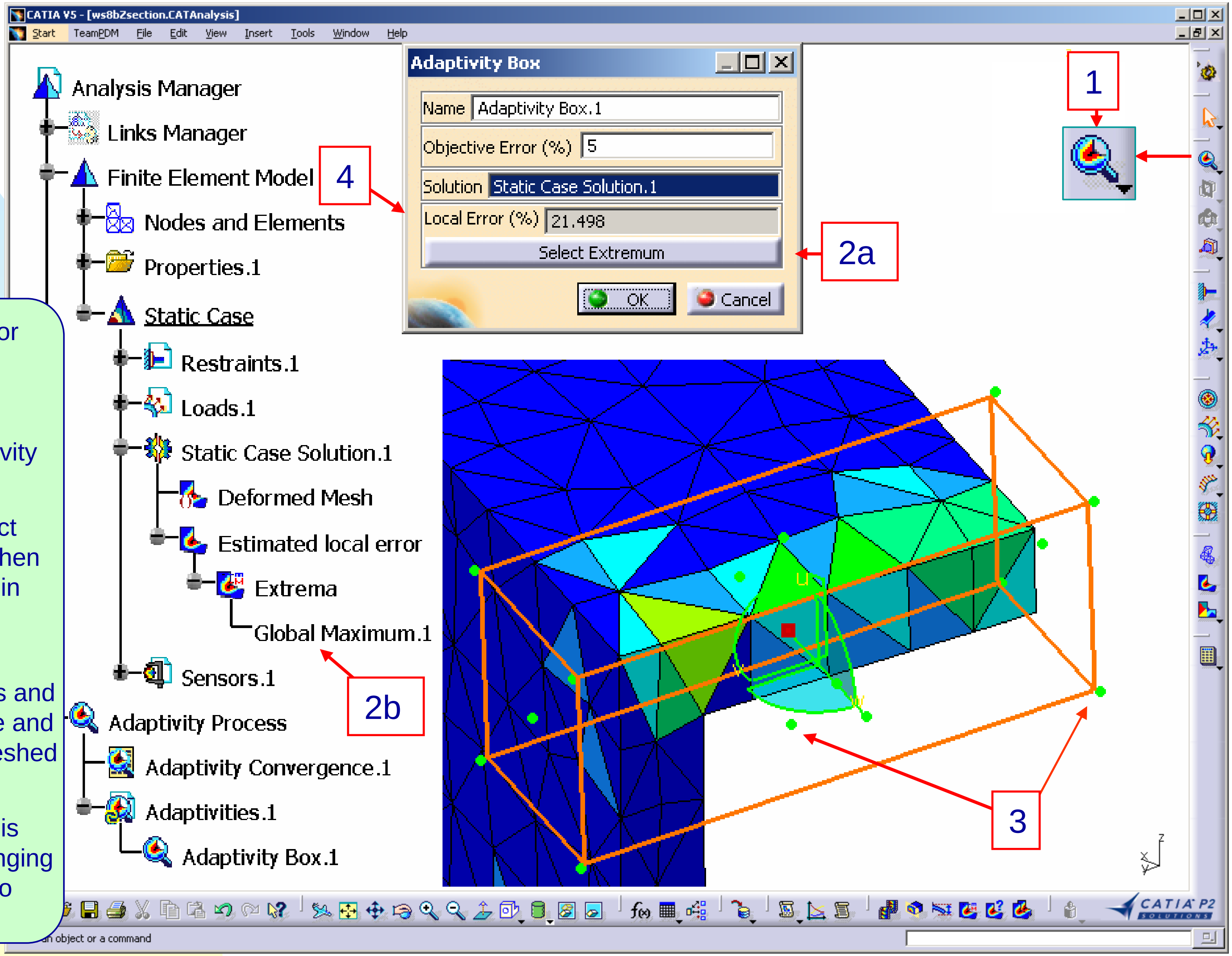
Steps:

1. Select the Search Image Extrema icon.

2. Select Global and 2 maximum extrema at most, select OK.

3. Right click the Global Maximum.1 object in the features tree then select Focus On.

Step 6. Check global and local precision



Determine local error percentage (%).

Steps:

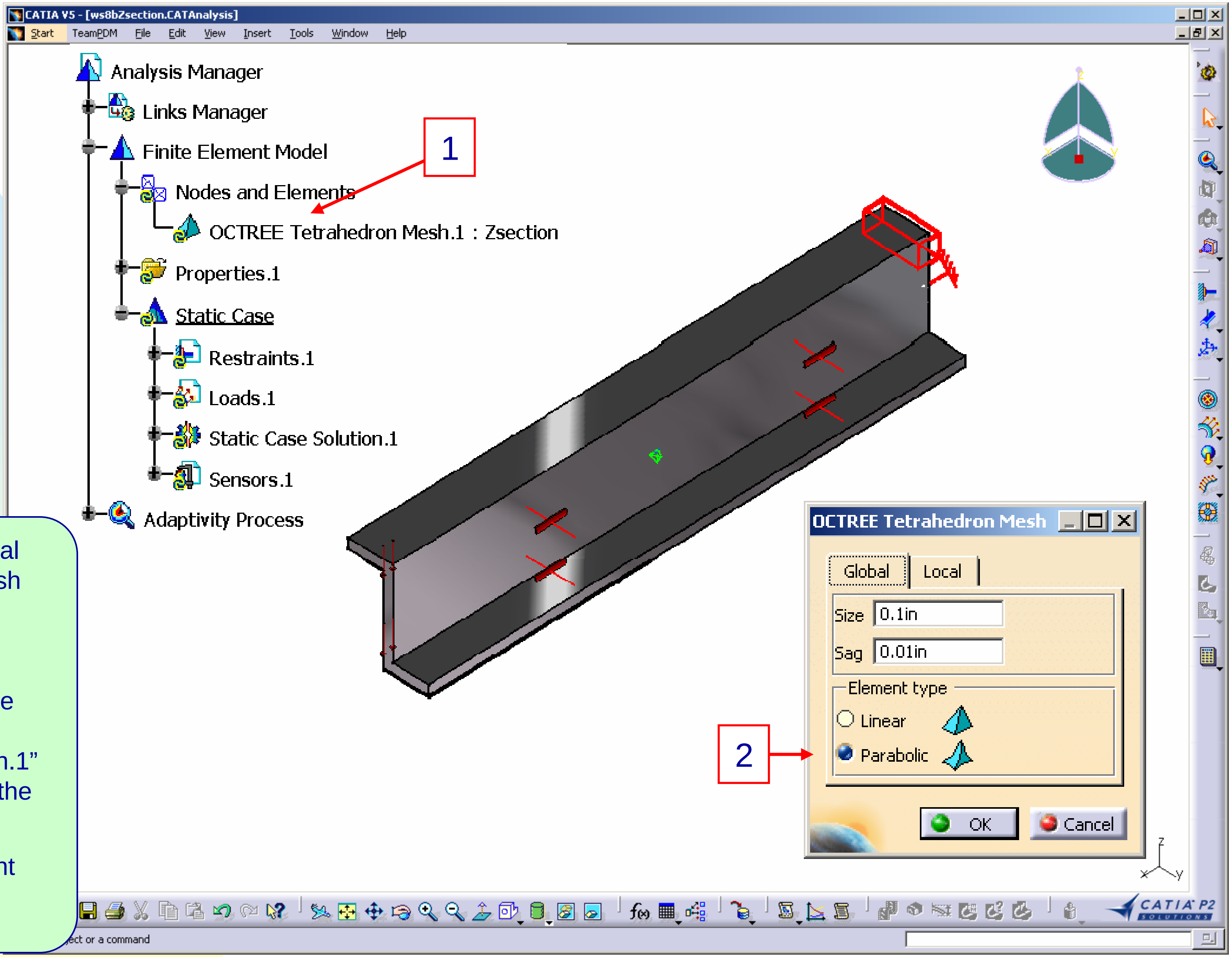
1. Select the adaptivity box icon.

2. Select the "Select Extremum" button then Global Maximum.1 in the features tree to locate box.

3. Use the compass and green dots to locate and size box around meshed areas.

4. Since local error is above 10% try changing the mesh element to Parabolic.

Step 7. Change mesh to parabolic



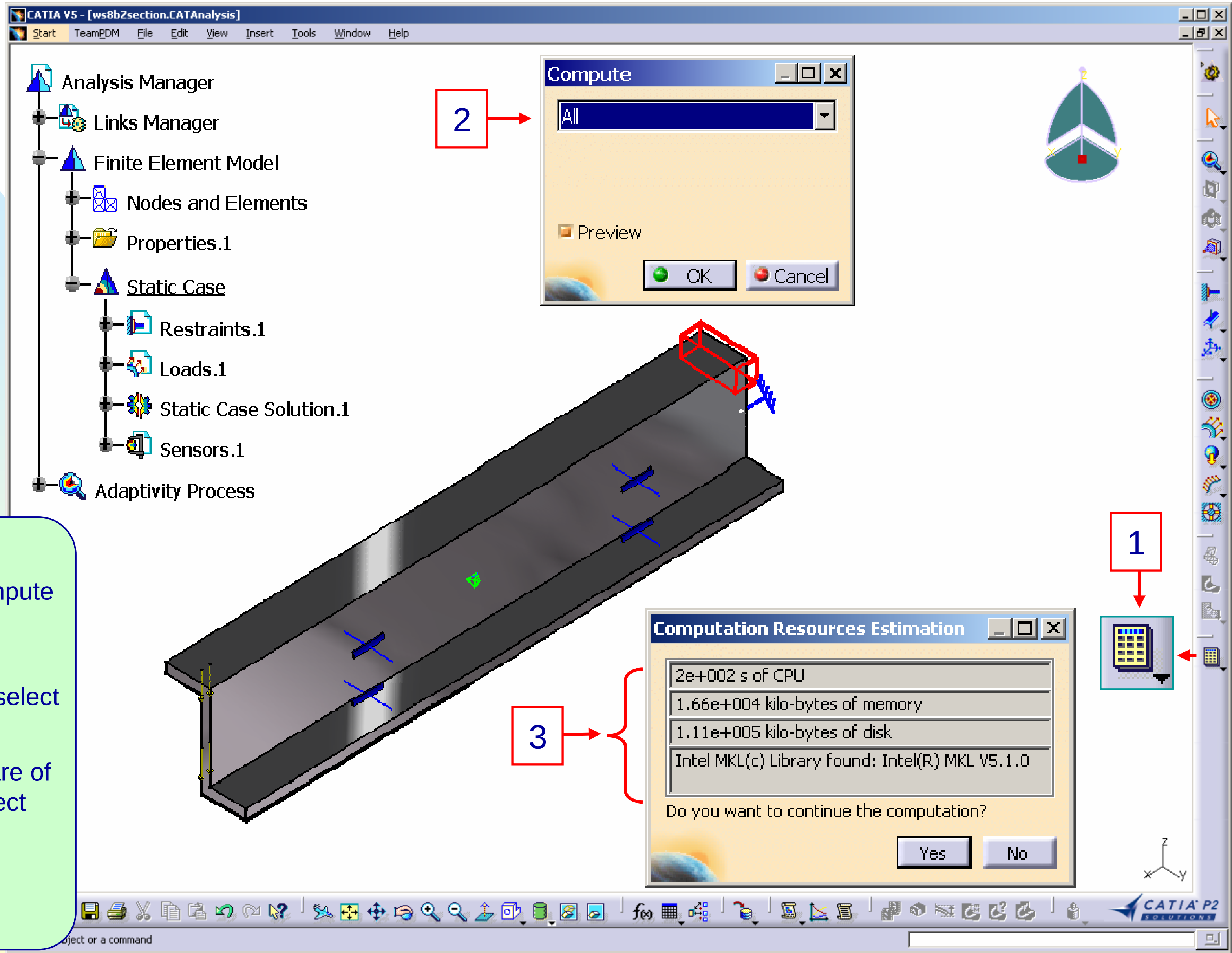
Redefine the global finite element mesh type.

Steps:

1. Double Click the "OCTREE Tetrahedron Mesh.1" representation in the features tree.

2. Change element type to Parabolic, select OK.

Step 8. Compute the precise analysis



Steps:

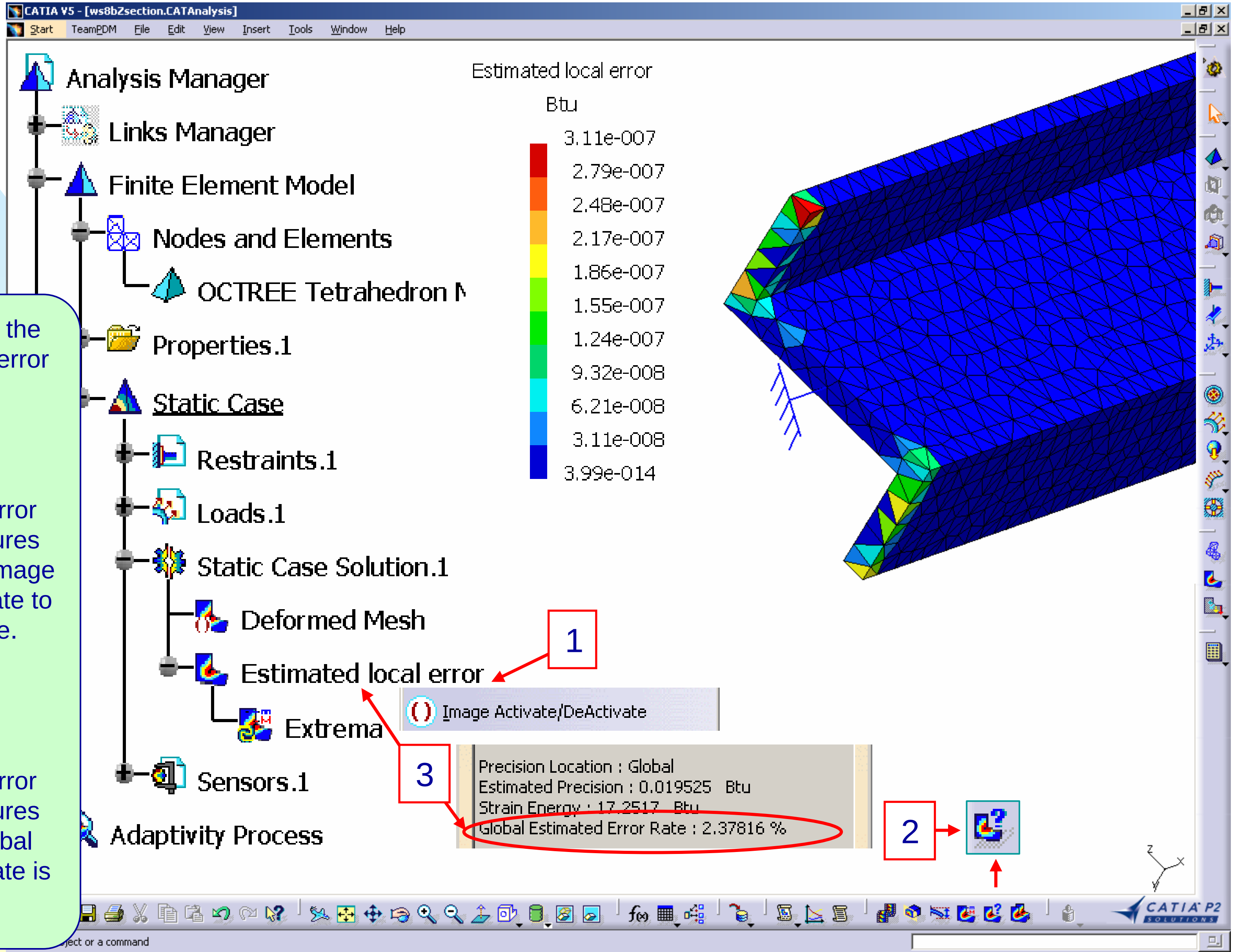
1. Select the Compute icon.

2. Compute All Objects defined, select OK.

3. Always be aware of these values, select Yes.

Save often.

Step 8. Compute the precise analysis



Check how much the global estimated error has improved

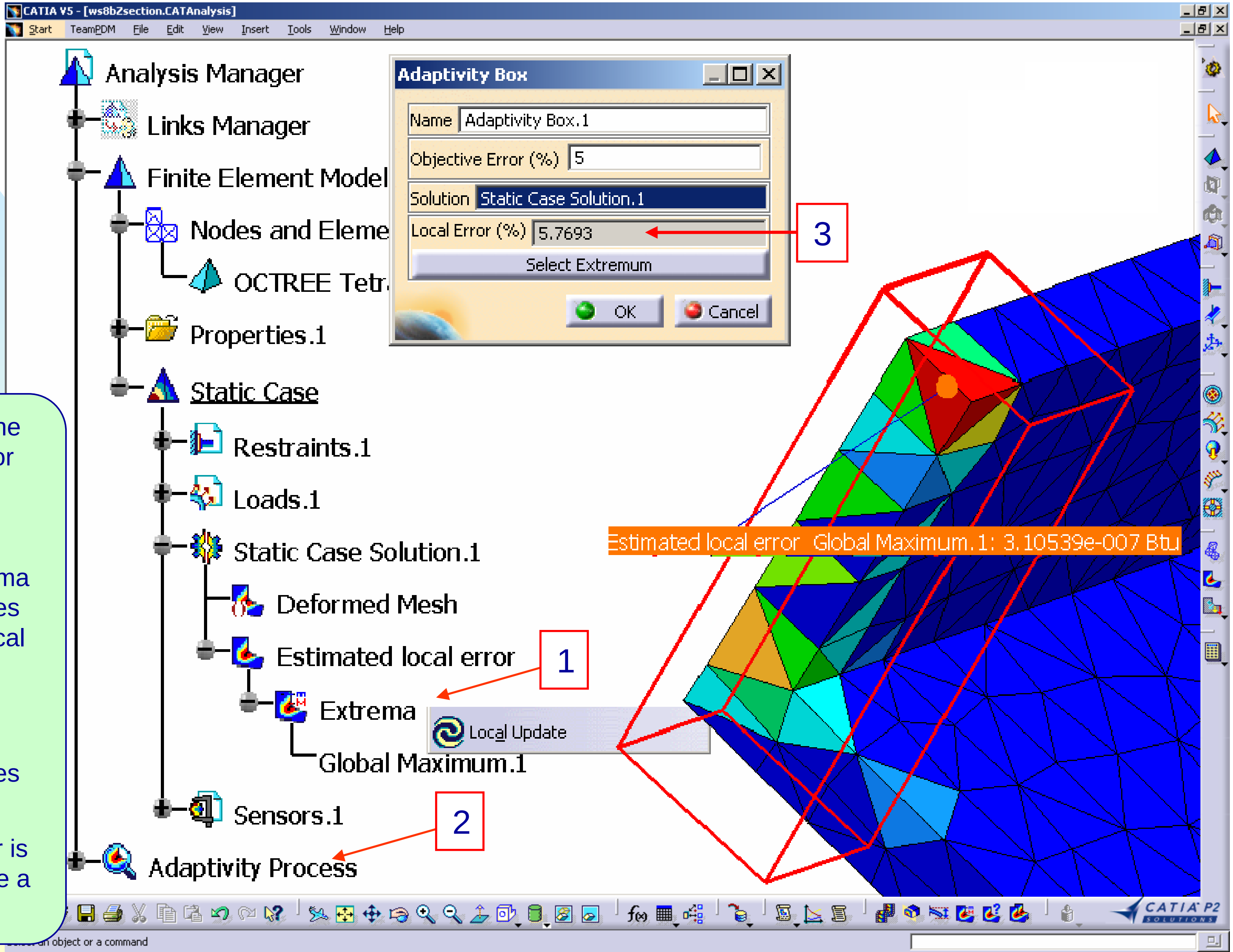
Steps:

1. Right click the Estimated local error object in the features tree then select Image Activate/DeActivate to activate the image.

2. Select on the information icon.

3. Select the Estimated local error object in the features tree. Note the global estimated error rate is very good.

Step 8. Compute the precise analysis



Check how much the local estimated error has improved.

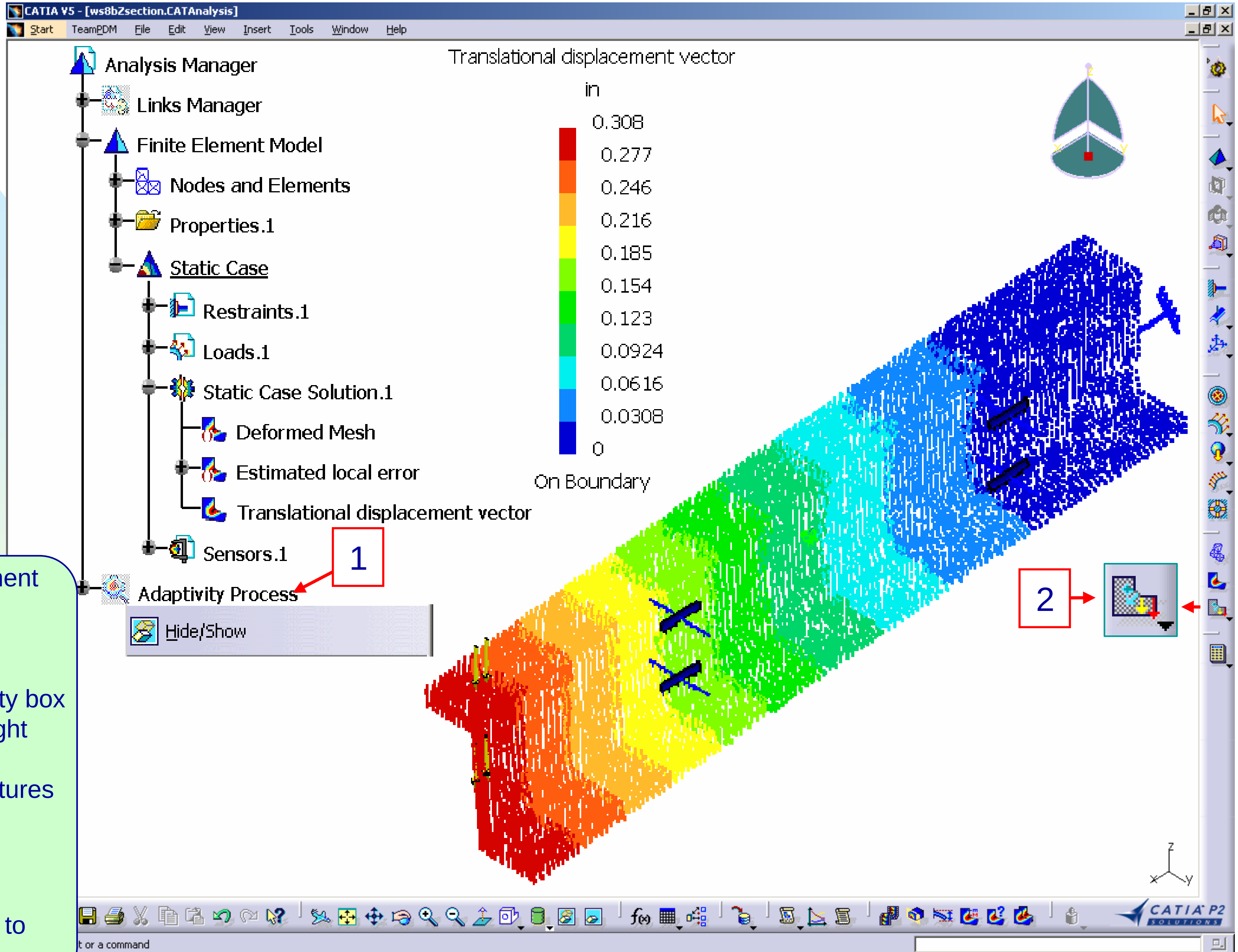
Steps:

1. Right click Extrema object in the features tree then select Local Update.

2. Double click the Adaptivity Process object in the features tree.

3. Since local error is below 10% we have a precise model.

Step 9. Visualize final results



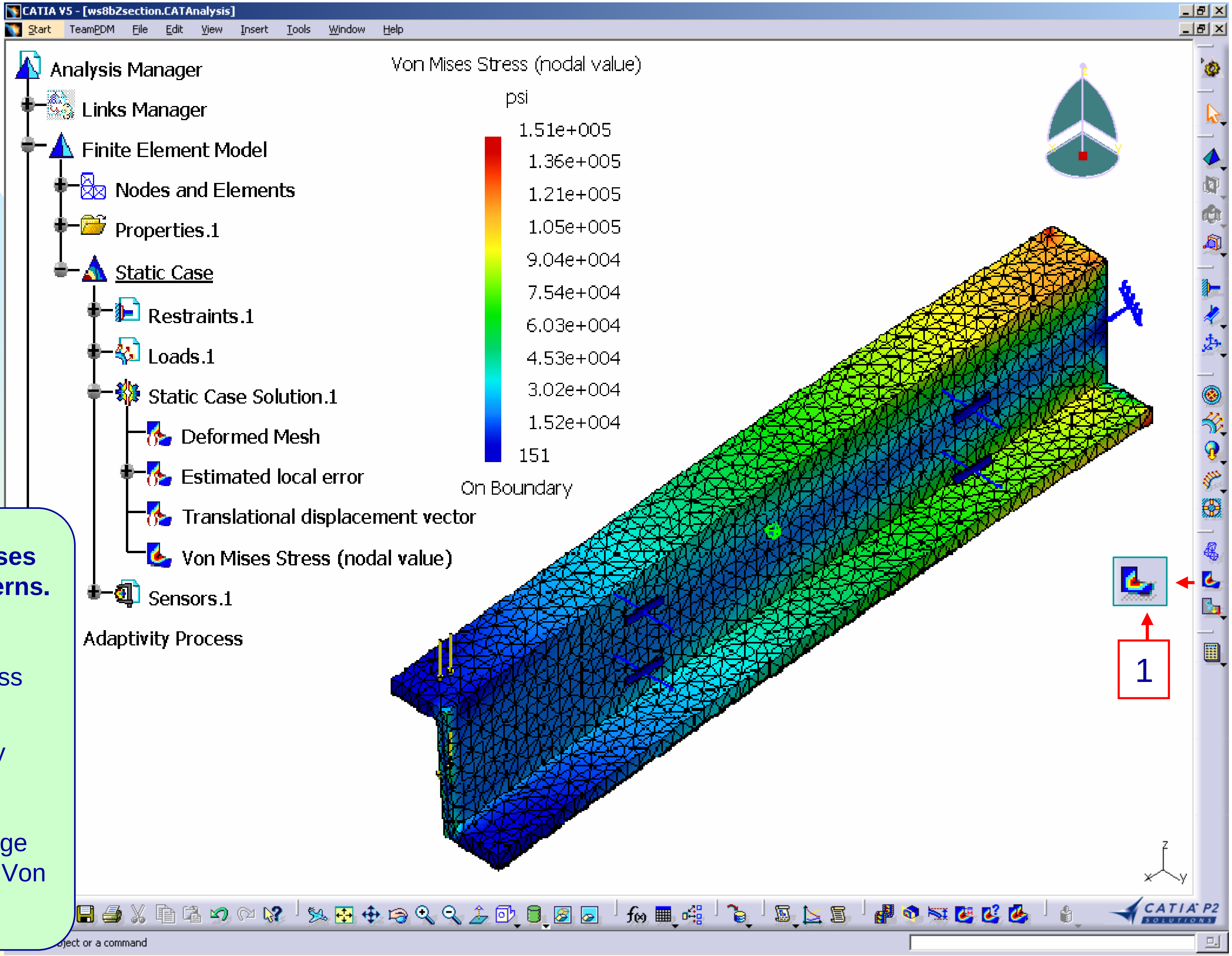
Add the displacement image.

Steps:

1. Put the adaptivity box into no show by right clicking Adaptivity Process in the features tree then select Hide/Show.

2. Select the displacement icon to add this image.

Step 9. Visualize final results



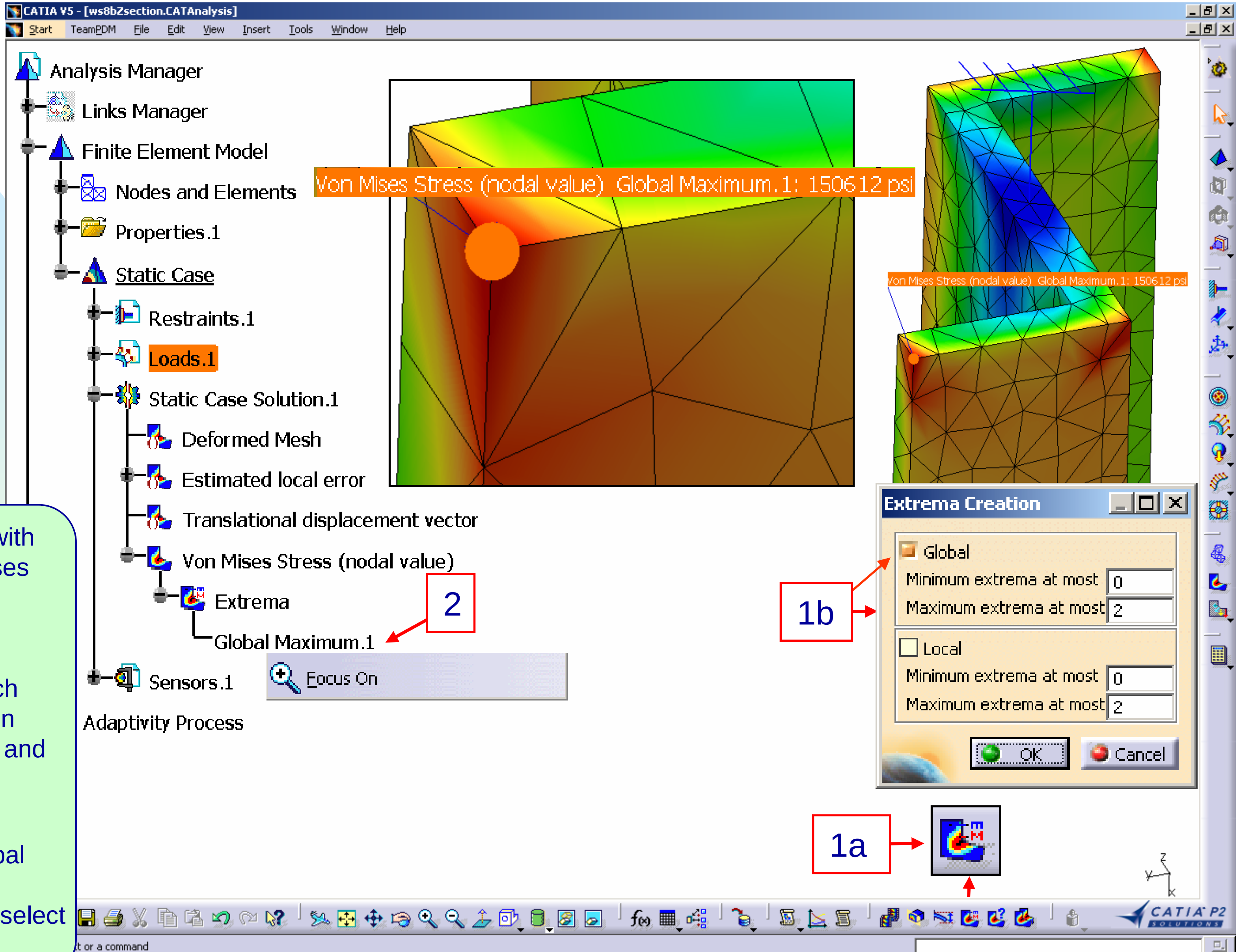
Visualize **Von Mises stress field patterns.**

Steps:

1. Select the Stress Von Mises icon.

This automatically deactivates the Translational displacement image and activates the Von Mises image.

Step 9. Visualize final results



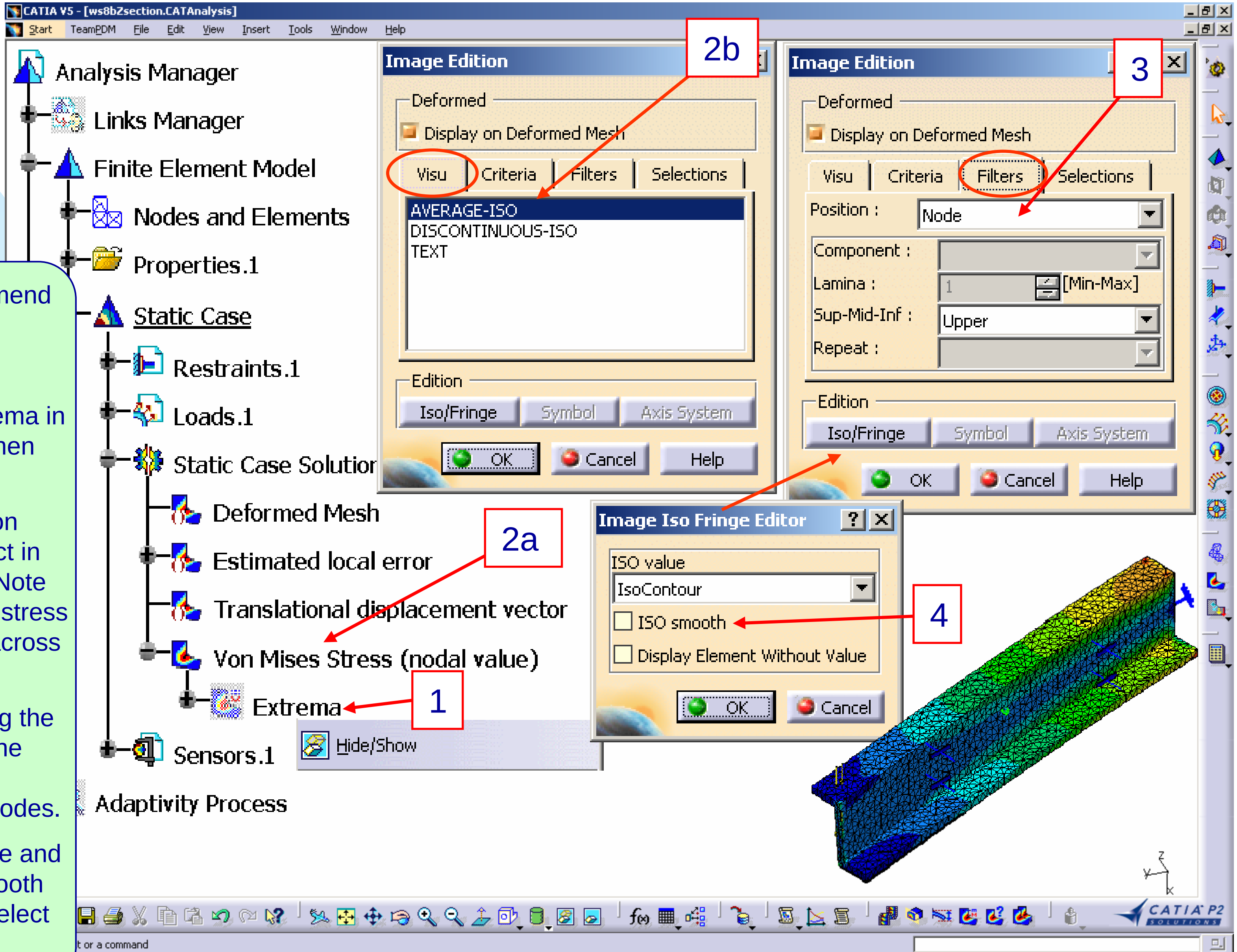
Find the element with maximum Von Mises Stress.

Steps:

1. Select the search image extrema icon then select Global and key in 2 Maximum extrema at most.

2. Right click Global Maximum.1 in the features tree then select Focus On.

Step 9. Visualize final results



Find exact recommend design stress.

Steps:

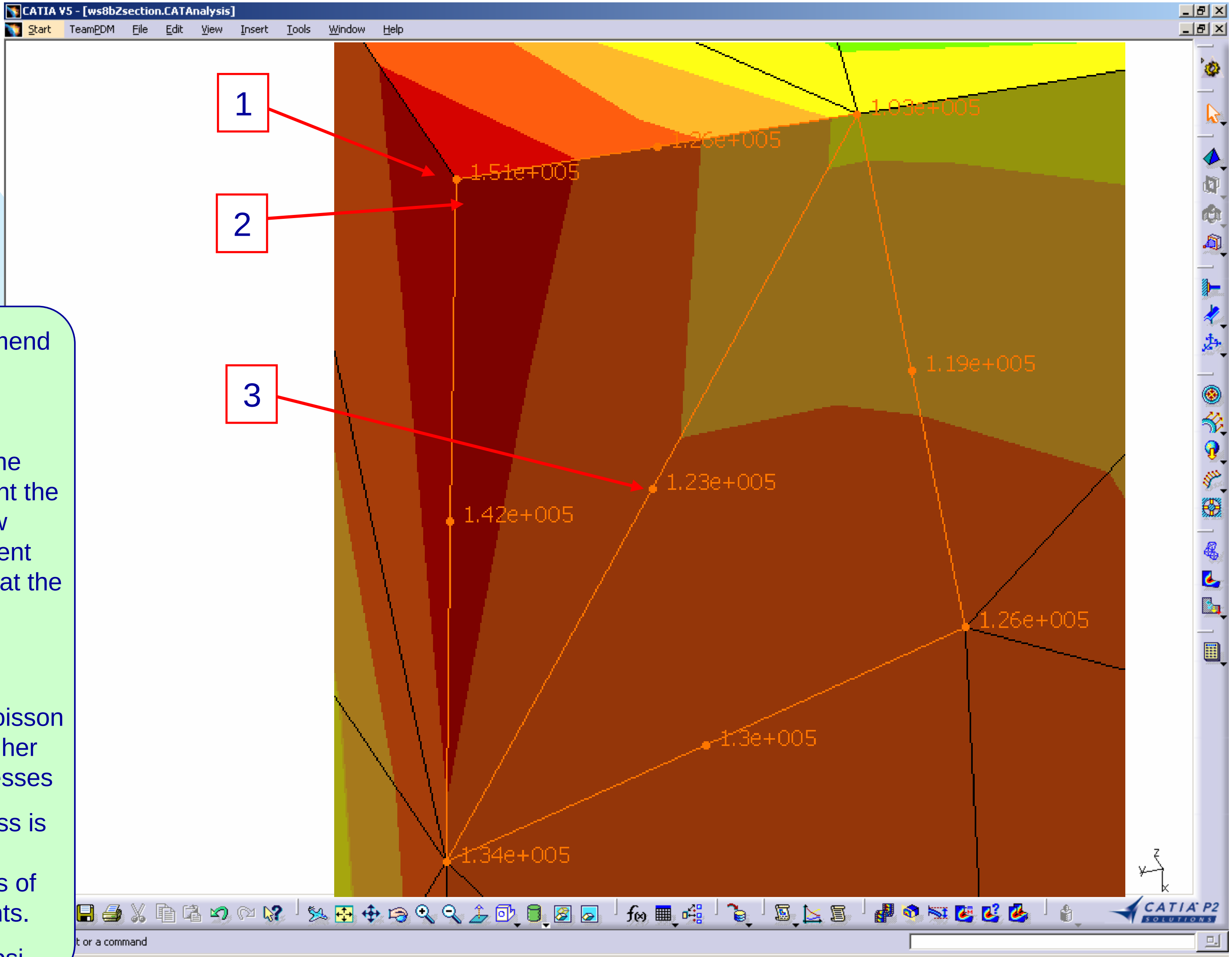
1. Right click Extrema in the features tree then select Hide/Show.

2. Double click Von Mises Stress object in the features tree. Note you are looking at stress values averaged across elements.

3. Also by selecting the Filters tab notice the stress output is calculated at the nodes.

4. Select Iso/Fringe and select the ISO smooth box to turn it off, select OK twice.

Step 9. Visualize final results



Find exact recommend design stress.

Steps:

1. By positioning the cursor on a element the stress values show relative to the current Filter (in this case at the nodes).

2. The maximum extrema stress is uninfluenced by poisson effects yielding higher than expected stresses

3. The design stress is found at the intermediate nodes of the bottom elements.

123000 - 134000 psi

Step 9. Visualize final results

Conclusions

- CATIA V5 GSA workbench is validated for a Z-section cantilever beam scenario. To be conservative, increase material strength to a minimum yield of 134000 psi for the described load case.

	Hand Calculations	.1 inch Parabolic Global Mesh, .01 inch sag
Global % Precision error Local % Precision error	NA NA	2.38 % 5.77 %
Error Estimate	NA	3.11e-7 Btu global
Translational Displacement	-0.304 inch	-0.308 inch (Z - direction)
Max Von Mises Stress	122000 psi	123000 - 134000 psi

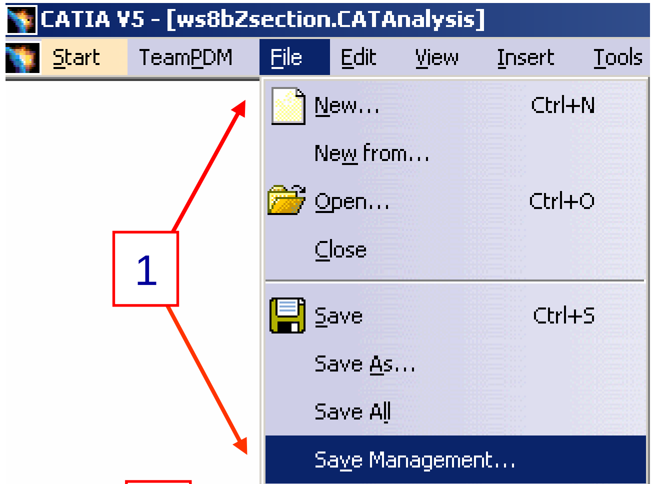
Step 10. Save the analysis document

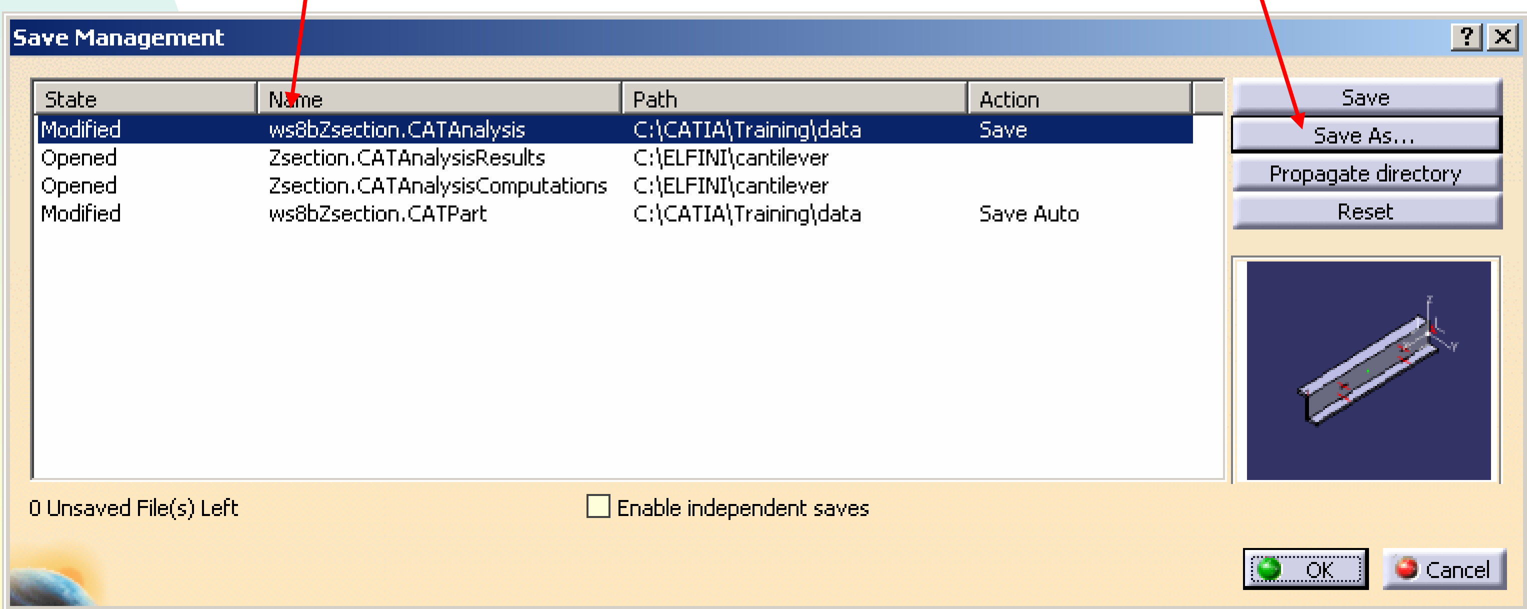
Steps:

1. From the file menu select Save Management.

2. Highlight document you want to save.

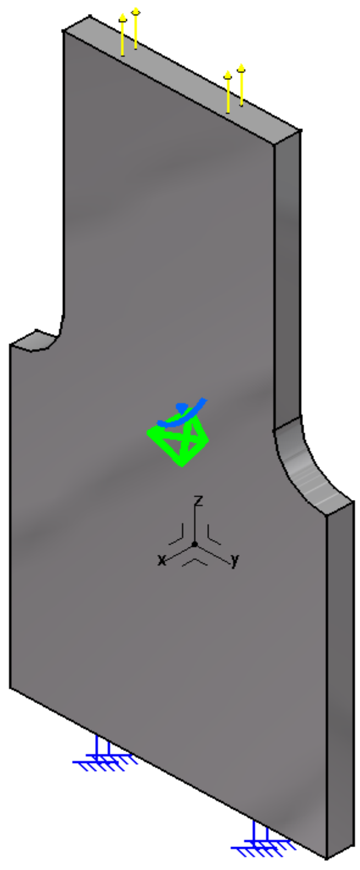
3. Select Save As to specify name and path, select, OK





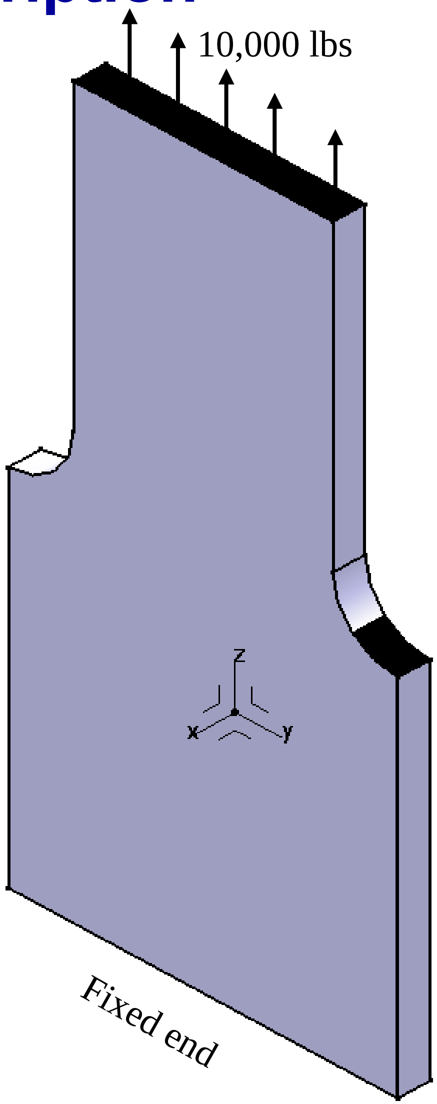
WORKSHOP 9

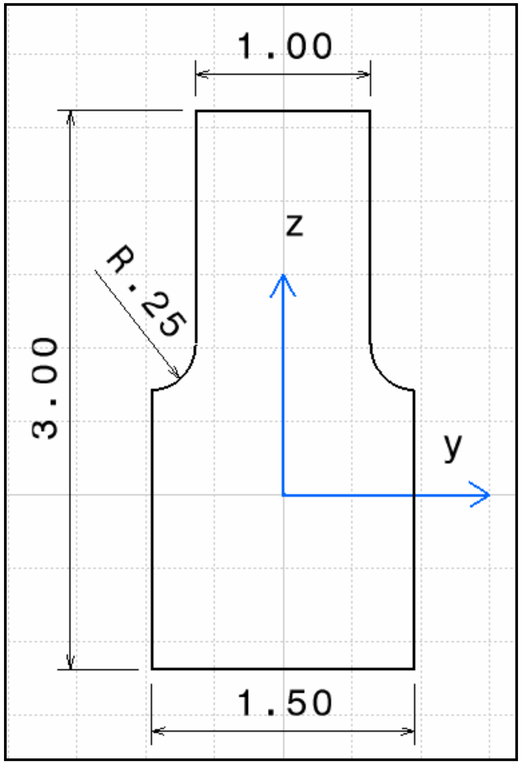
STRESS CONCENTRATION FOR A STEPPED FLAT TENSION BAR



Problem Description

- Load case.





Material: Steel
Young Modulus = 29e6 psi
Poisson Ratio = .266
Density = .284 lb_in3
Yield Strength = 36259 psi

Problem Description

- Approximate axial displacement

$$\text{Axial Deformation} = \delta = \frac{P \cdot L}{E \cdot A} = \frac{10000 \text{ lbs} \cdot 3 \text{ inch}}{29e^{6} \text{ psi} \cdot 0.125 \text{ inch}^{2}} = 0.0083 \text{ inch}$$

- Stress configuration factor and axial stress

$$\text{Stress Configuration Factor} = K_t = \frac{\sigma_{max}}{\sigma_{nom}} \qquad \sigma_{nom} = \frac{P}{t \cdot d}$$

$$\frac{D}{d} = \frac{1.5 \text{ inch}}{1.0 \text{ inch}} = 1.5 \qquad \frac{r}{d} = \frac{0.25 \text{ inch}}{1.0 \text{ inch}} = 0.25 \qquad K_t = 1.74 \text{ (R.E. Peterson 1974 Figure 65)}$$

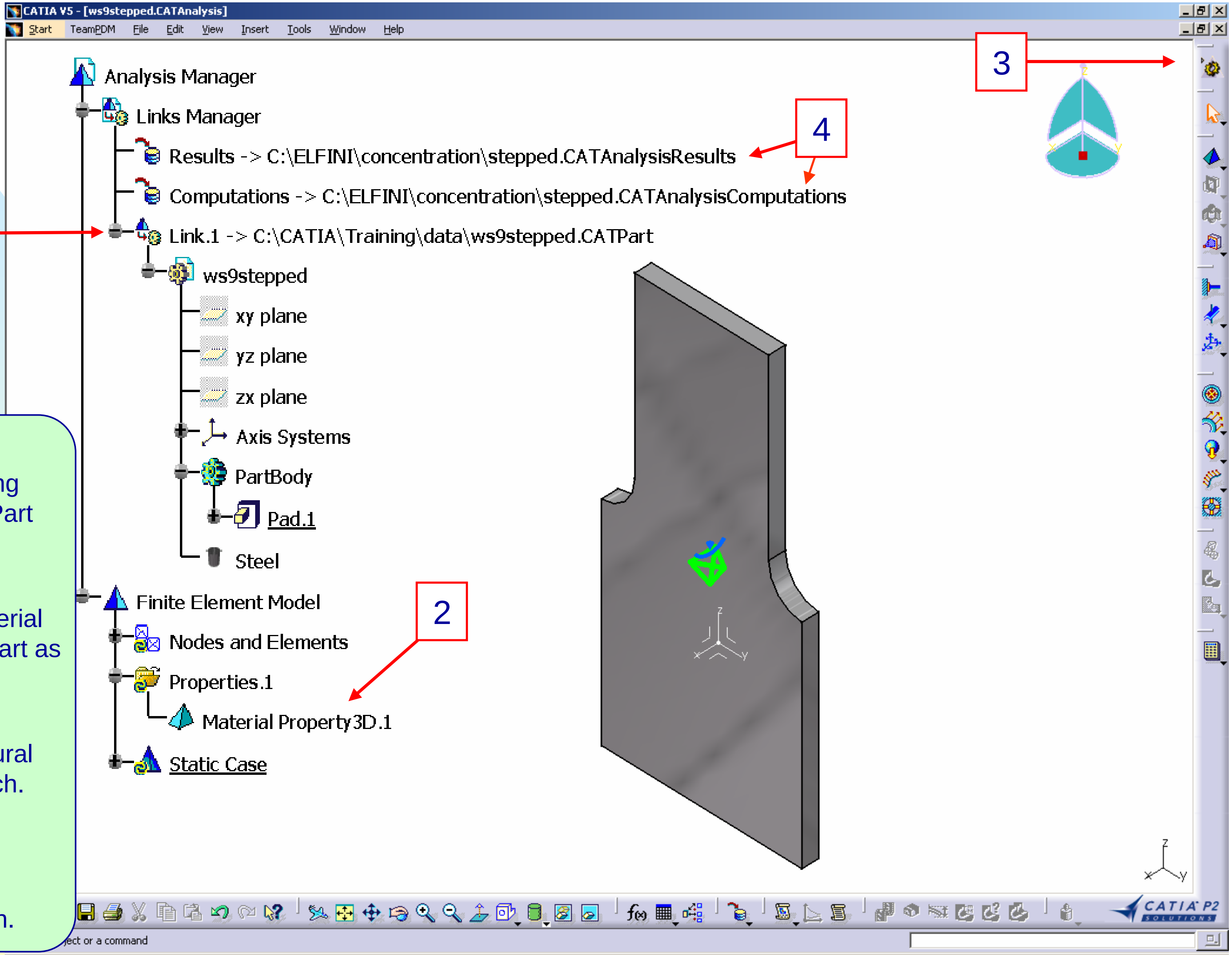
$$\text{Maximum normal tensile stress} = \sigma_{max} = \sigma_{nom} \cdot K_t = \frac{10000 \text{ lbs}}{0.125 \text{ inch} \cdot 1.0 \text{ inch}} \cdot 1.74$$

$$\sigma_{max} = 80000 \text{ psi} \cdot 1.74 = 139200 \text{ psi}$$

Suggested Exercise Steps

1. Create a new CATIA analysis document (.CATAnalysis).
2. Mesh globally with parabolic elements.
3. Apply a clamp restraint.
4. Apply a pressure force.
5. Compute the initial analysis.
6. Check global and local precision (animate deformation, adaptive boxes and extremas).
7. Change mesh size and add local meshing.
8. Compute the precise analysis.
9. Visualize final results.
10. Save the analysis document.

Step 1. Create a new CATIA analysis document



Steps:

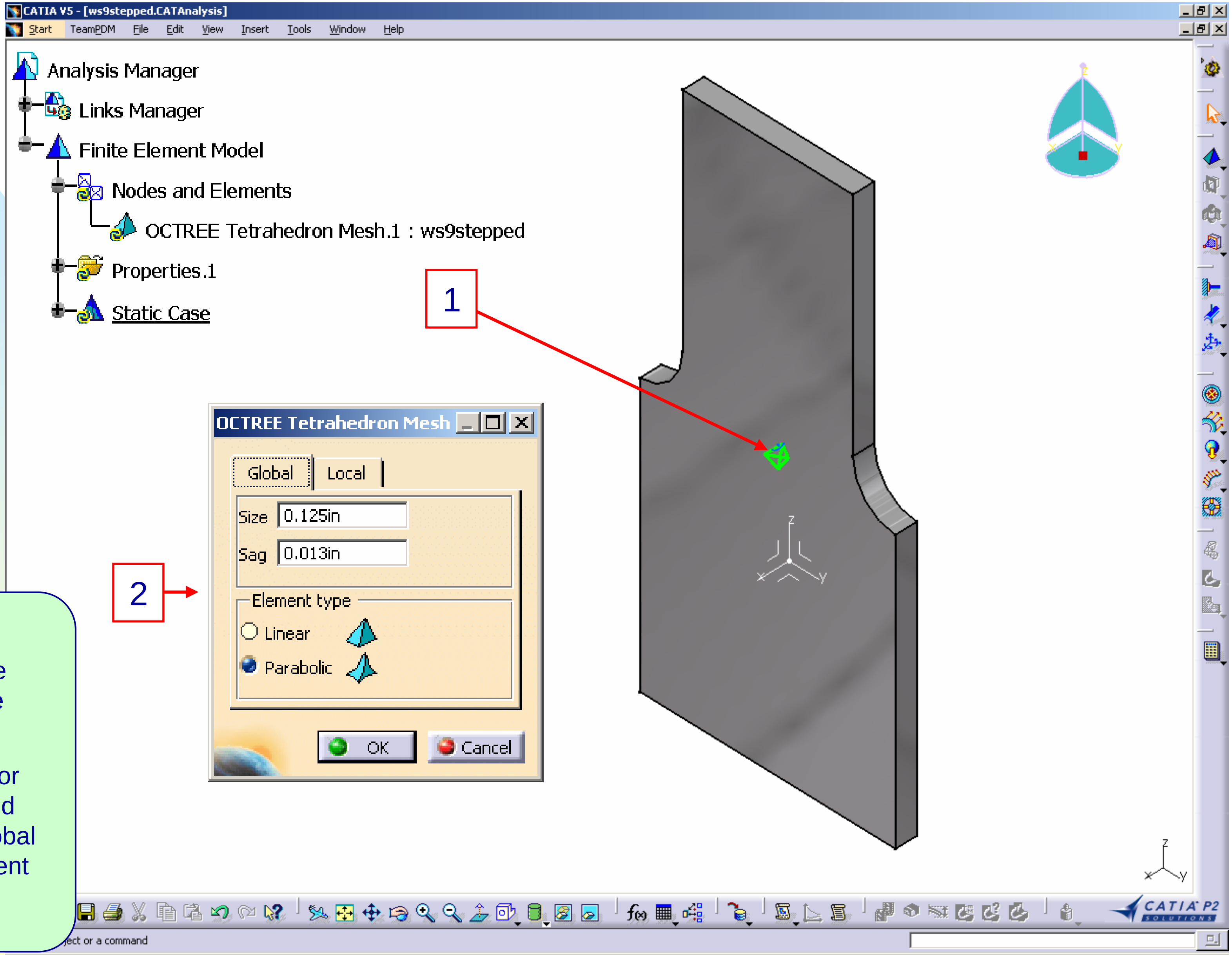
1. Open the existing ws9stepped.CATPart from the training directory.

2. Apply steel material properties to the part as required.

3. Launch the Generative Structural Analysis workbench.

4. Specify the Computations and Results storage locations as shown.

Step 2. Mesh globally with parabolic elements

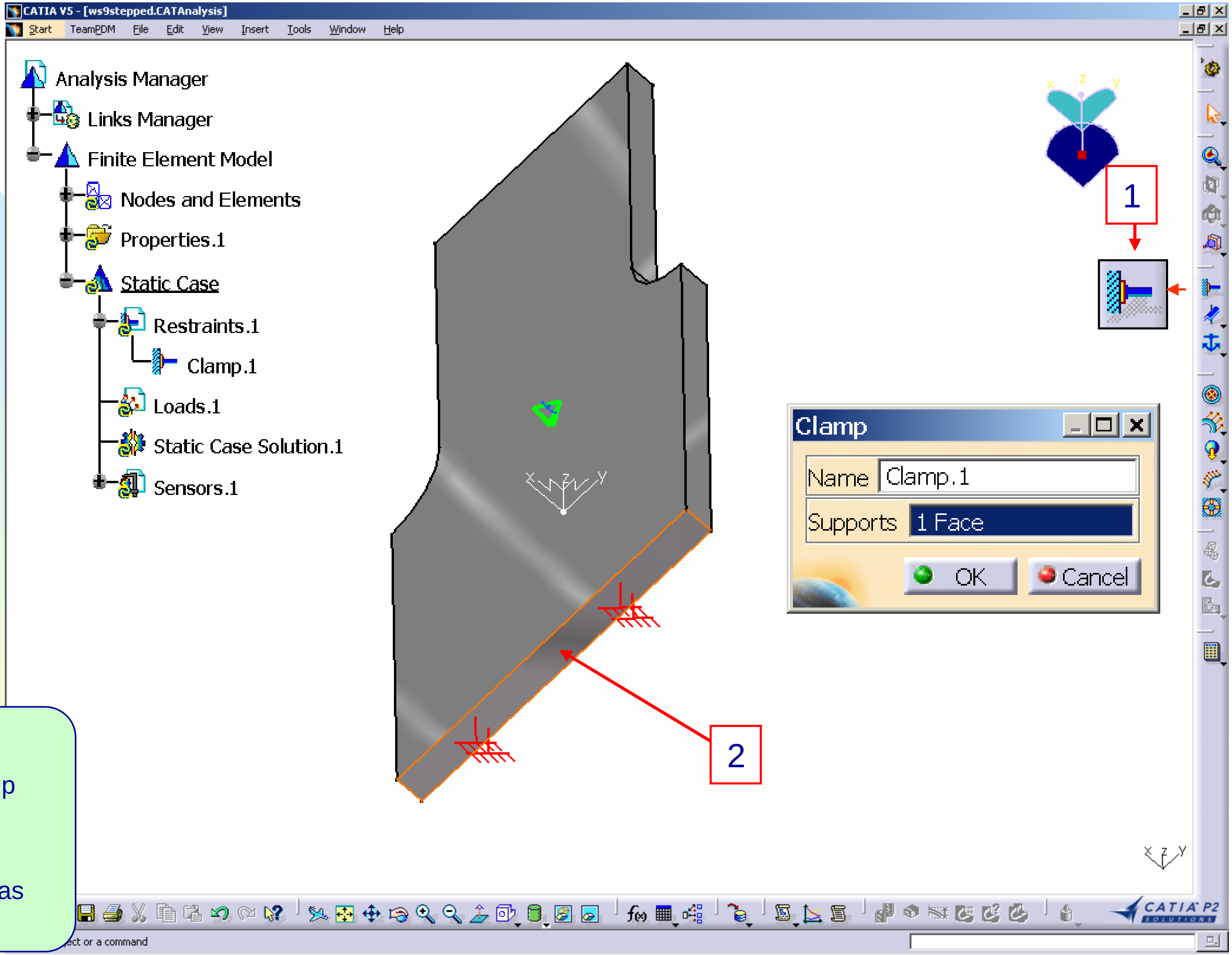


Steps:

1. Double Click the "Mesh" icon on the part.

2. Key in 0.125in for the Global Size and 0.013in for the Global sag, change element type to Parabolic, select OK.

Step 3. Apply a clamp restraint

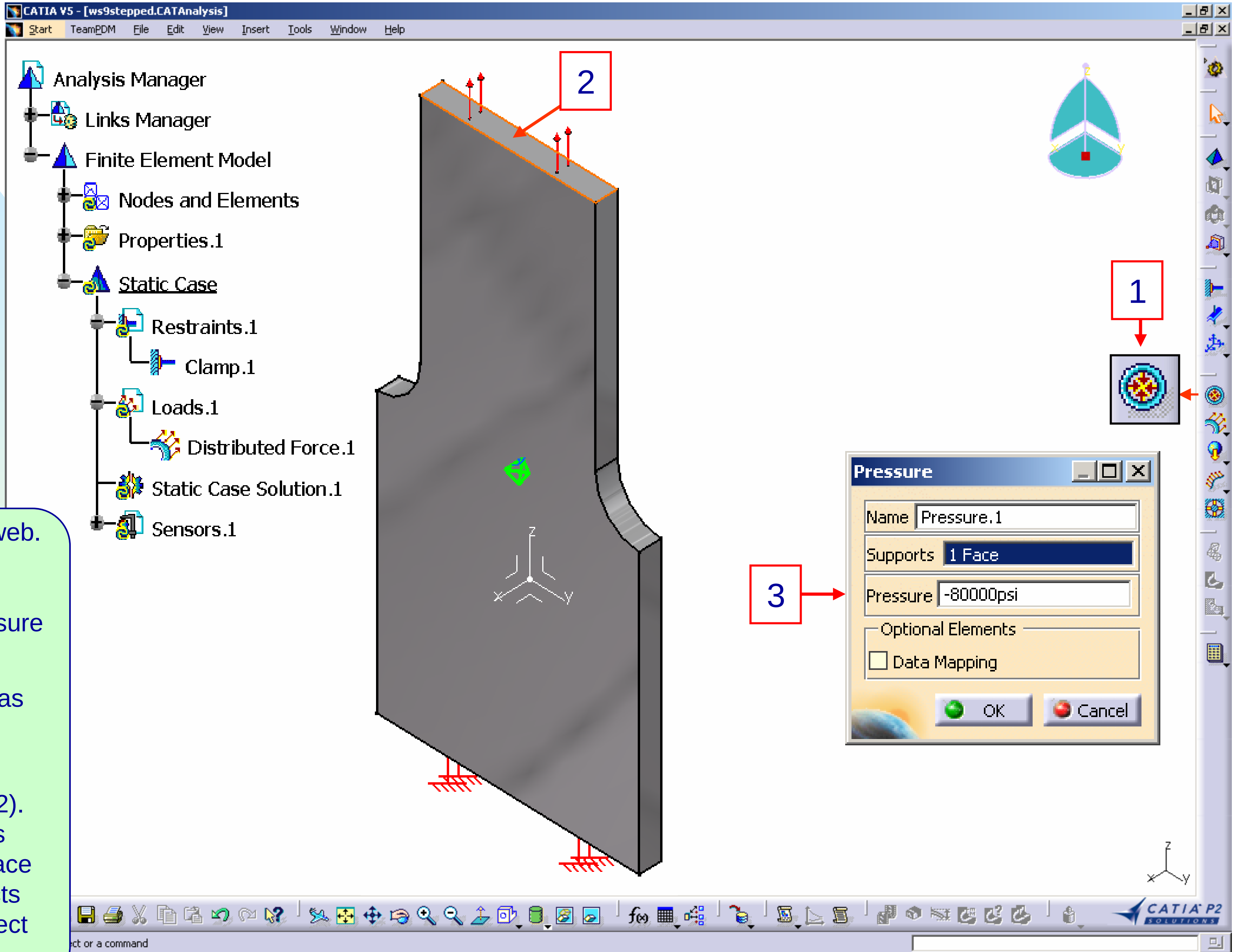


Steps:

1. Select the Clamp Restraint icon.

2. Select the face as shown, select OK.

Step 4. Apply a pressure force



Load only on the web.

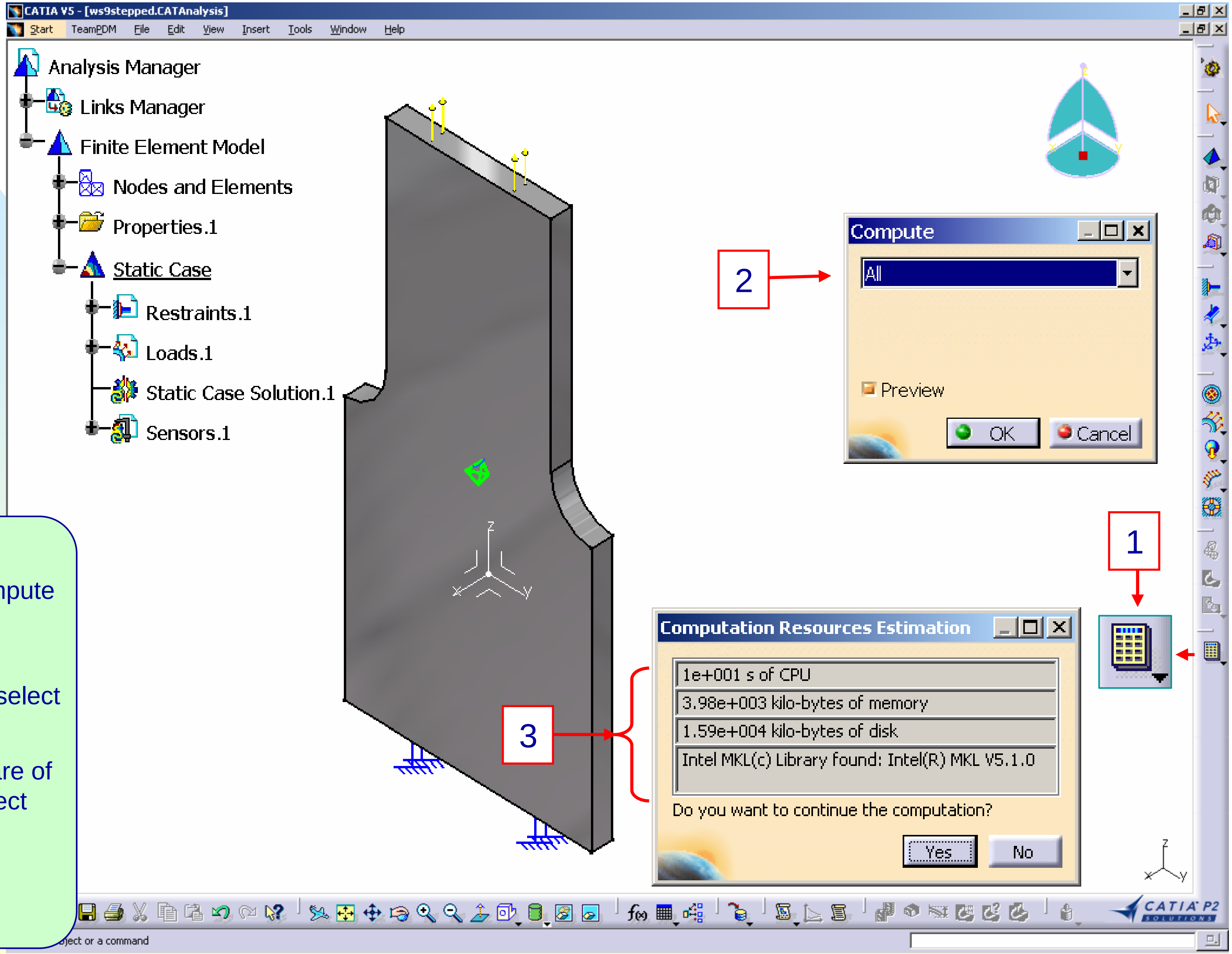
Steps:

1. Select the Pressure icon.

2. Select top face as shown.

3. Enter -80000psi (10000lbs/0.125in2). Pressure is always normal to the surface and negative directs force outward, select OK.

Step 5. Compute the initial analysis



Steps:

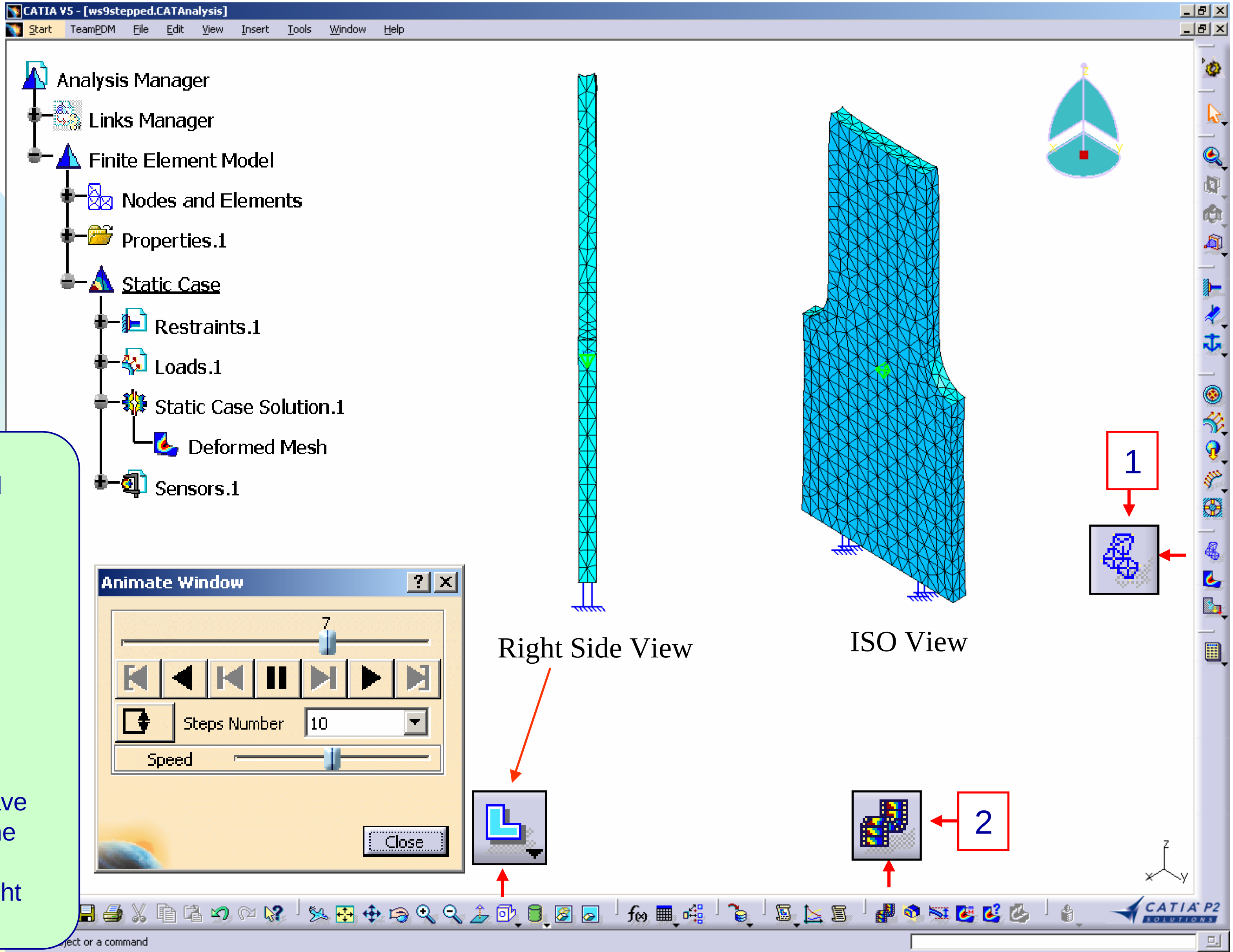
1. Select the Compute icon.

2. Compute All Objects defined, select OK.

3. Always be aware of these values, select Yes.

Save often.

Step 6. Check global and local precision



Visualize the **Deformation** and animate.

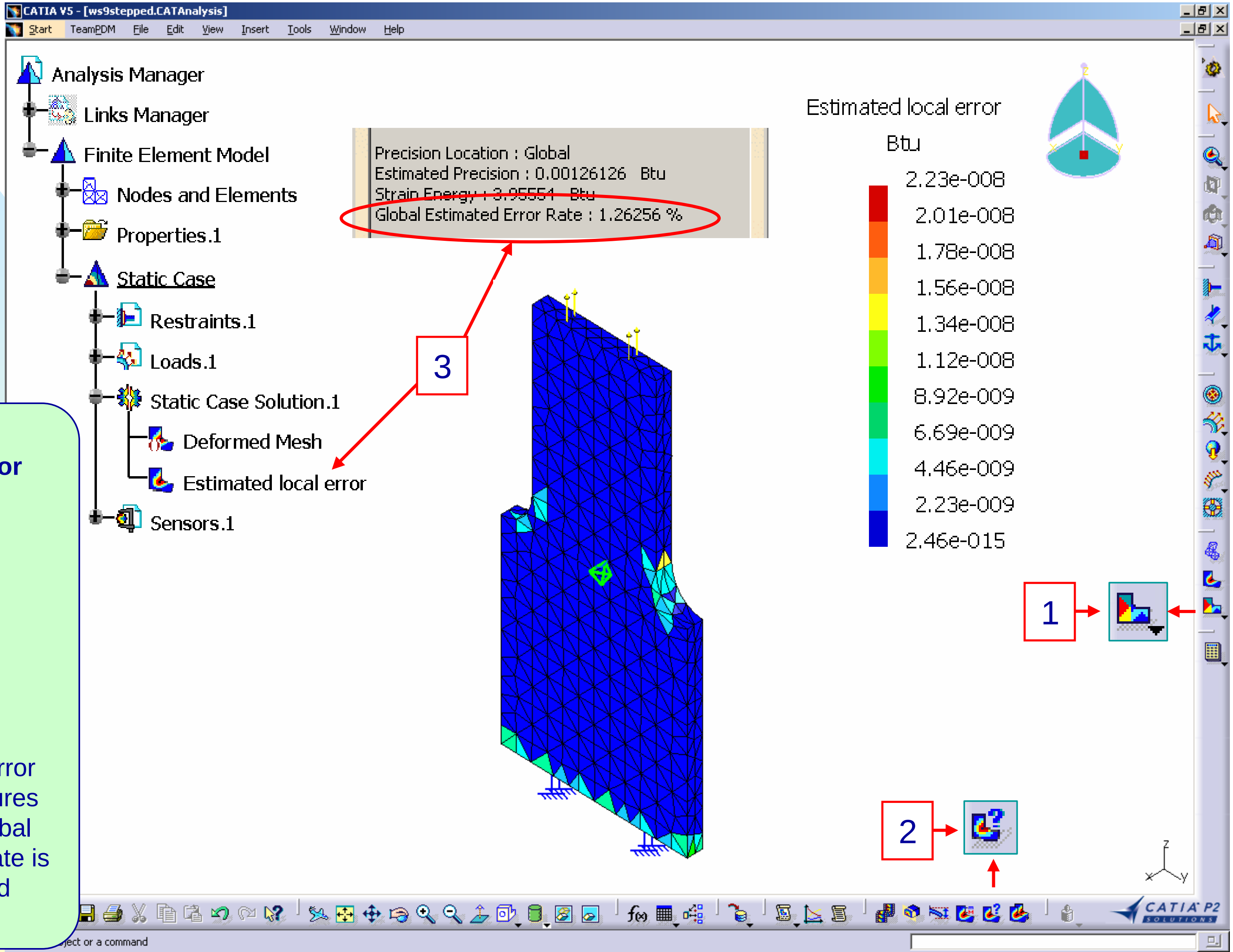
Steps:

1. Select the Deformation icon.

2. Select on the Animate icon.

Verify that you have no deflection in the x-direction by animating the Right side view.

Step 6. Check global and local precision



Visualize the **computation error map.**

Steps:

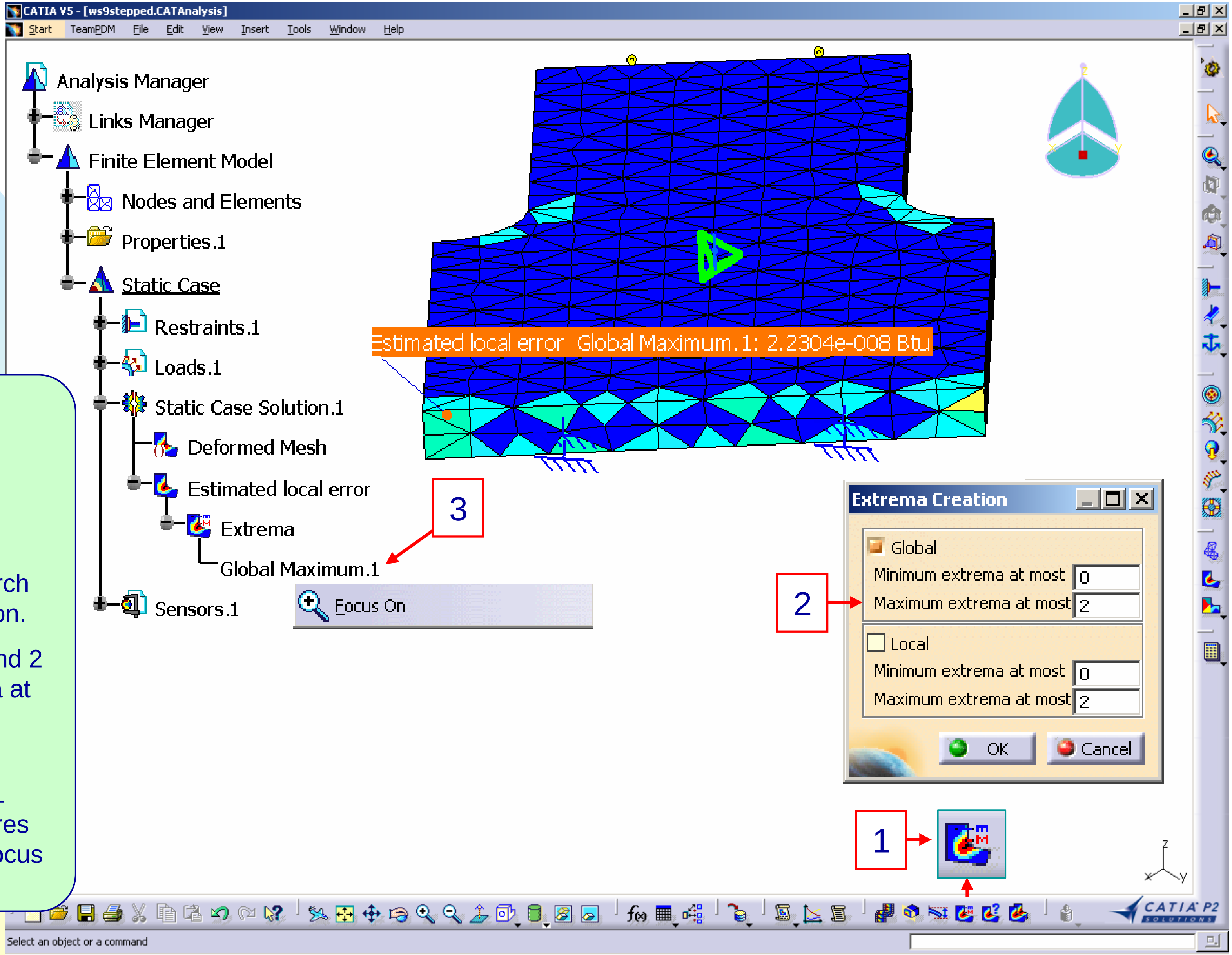
1. Select the Precision icon.

2. Select on the information icon.

3. Select the Estimated local error object in the features tree. Note the global estimated error rate is great (recommend max 20%).

MSC SOFTWARE

Step 6. Check global and local precision



Find the global element with the highest estimated error.

Steps:

1. Select the Search Image Extrema icon.

2. Select Global and 2 maximum extrema at most, select OK.

3. Right click the Global Maximum.1 object in the features tree then select Focus On.

Step 6. Check global and local precision

Determine maximum local error %.

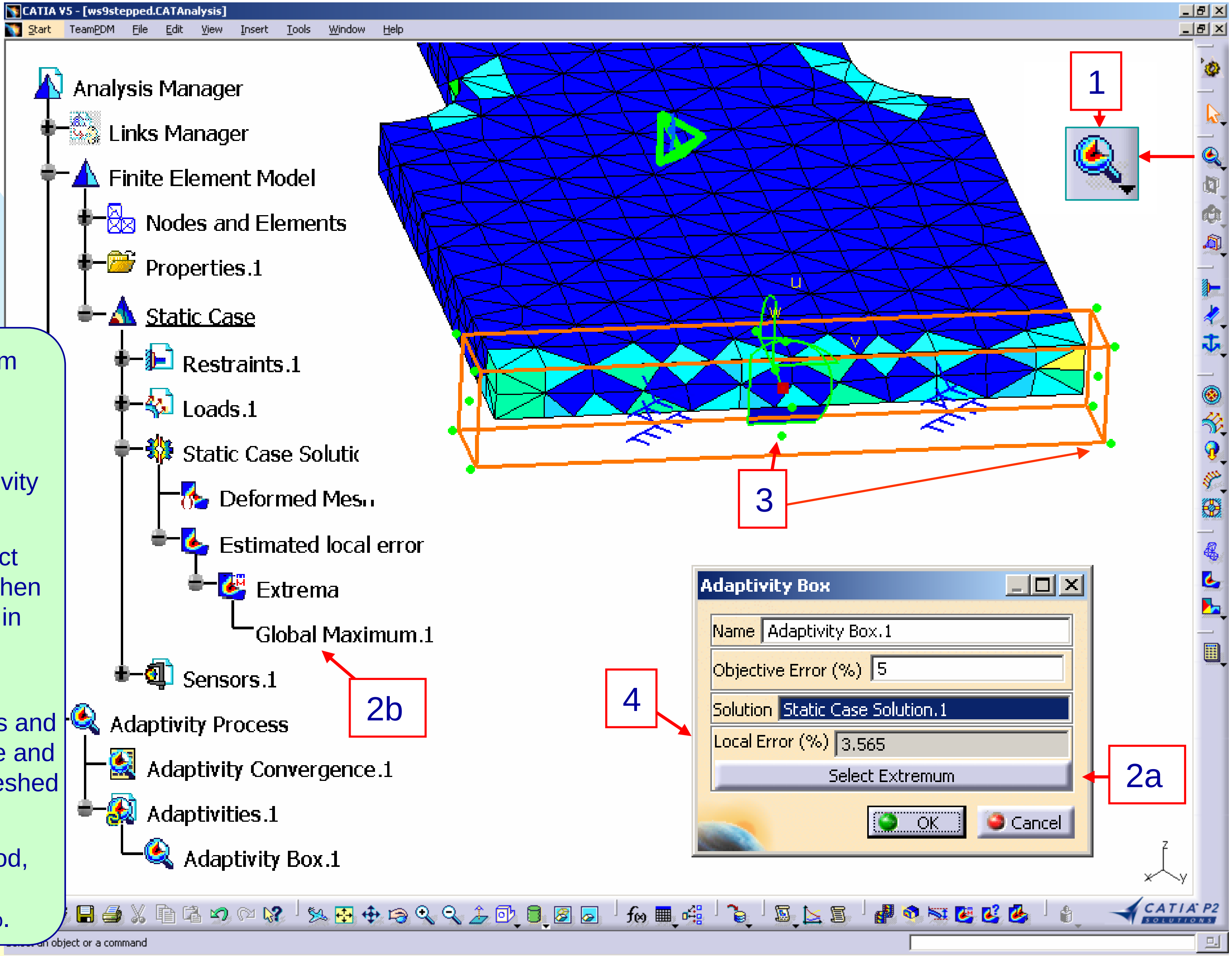
Steps:

1. Select the adaptivity box icon.

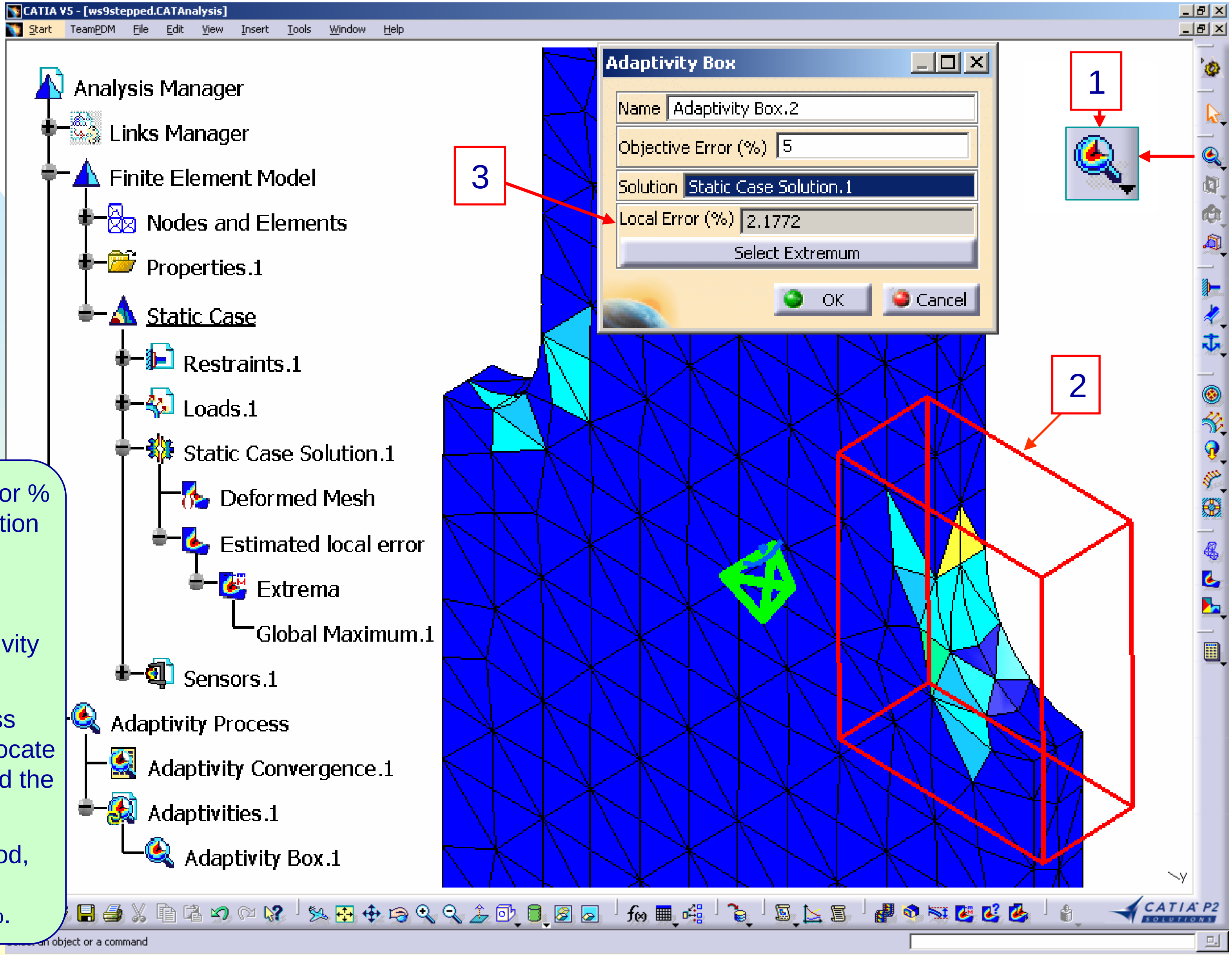
2. Select the "Select Extremum" button then Global Maximum.1 in the features tree to locate box.

3. Use the compass and green dots to locate and size box around meshed areas.

4. Local error is good, well below the recommended 10%.



Step 6. Check global and local precision



Determine local error % at stress concentration area.

Steps:

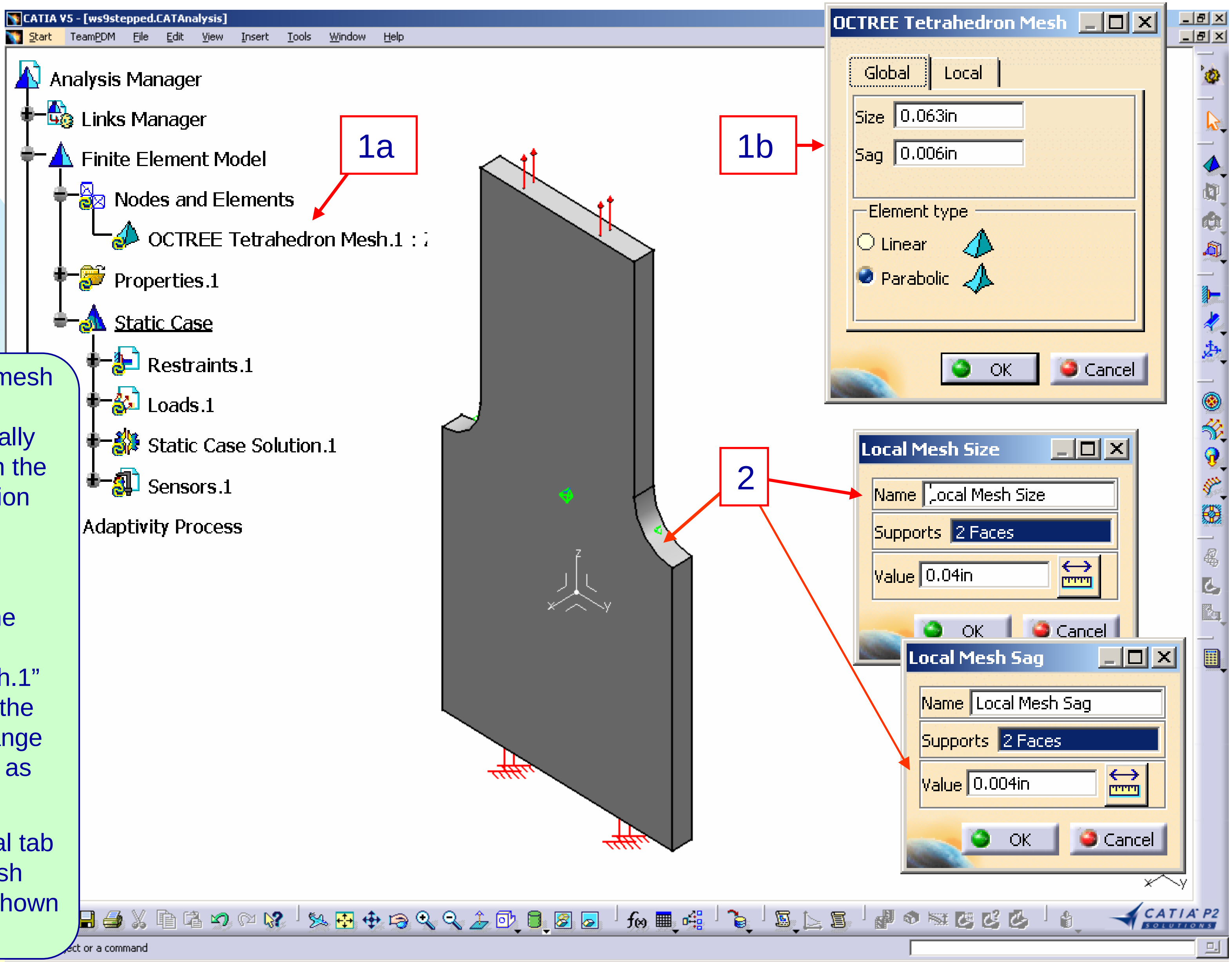
1. Select the adaptivity box icon.

2. Use the compass and green dots to locate and size box around the notch area.

3. Local error is good, well below the recommended 10%.

MSC SOFTWARE

Step 7. Change mesh size and add local meshing



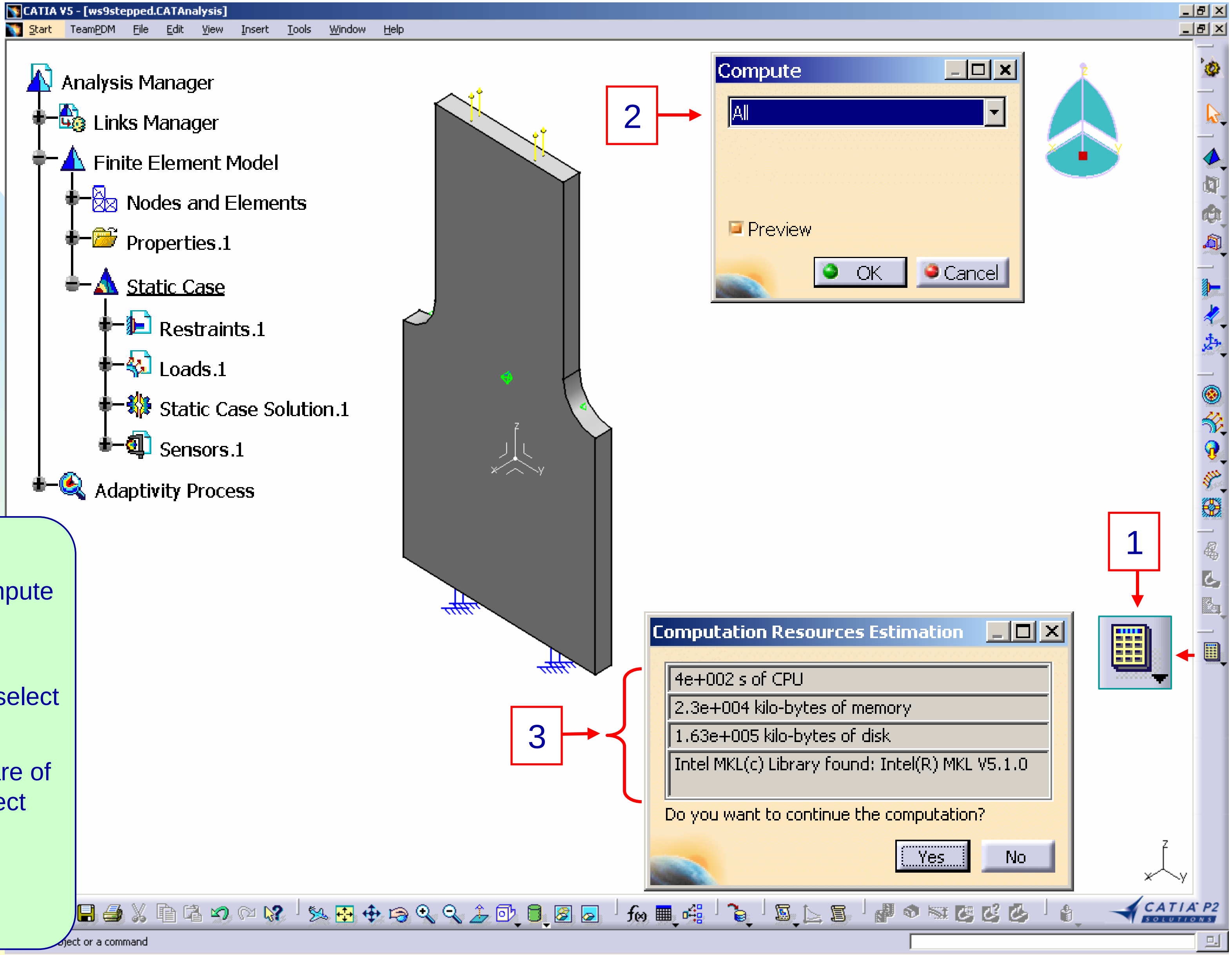
Globally change mesh size to ½ the bar thickness and locally refine the mesh in the stress concentration areas.

Steps:

1. Double Click the “OCTREE Tetrahedron Mesh.1” representation in the features tree, change Global mesh size as shown.

2. Select the Local tab and add local mesh size and sag as shown select OK.

Step 8. Compute the precise analysis



Steps:

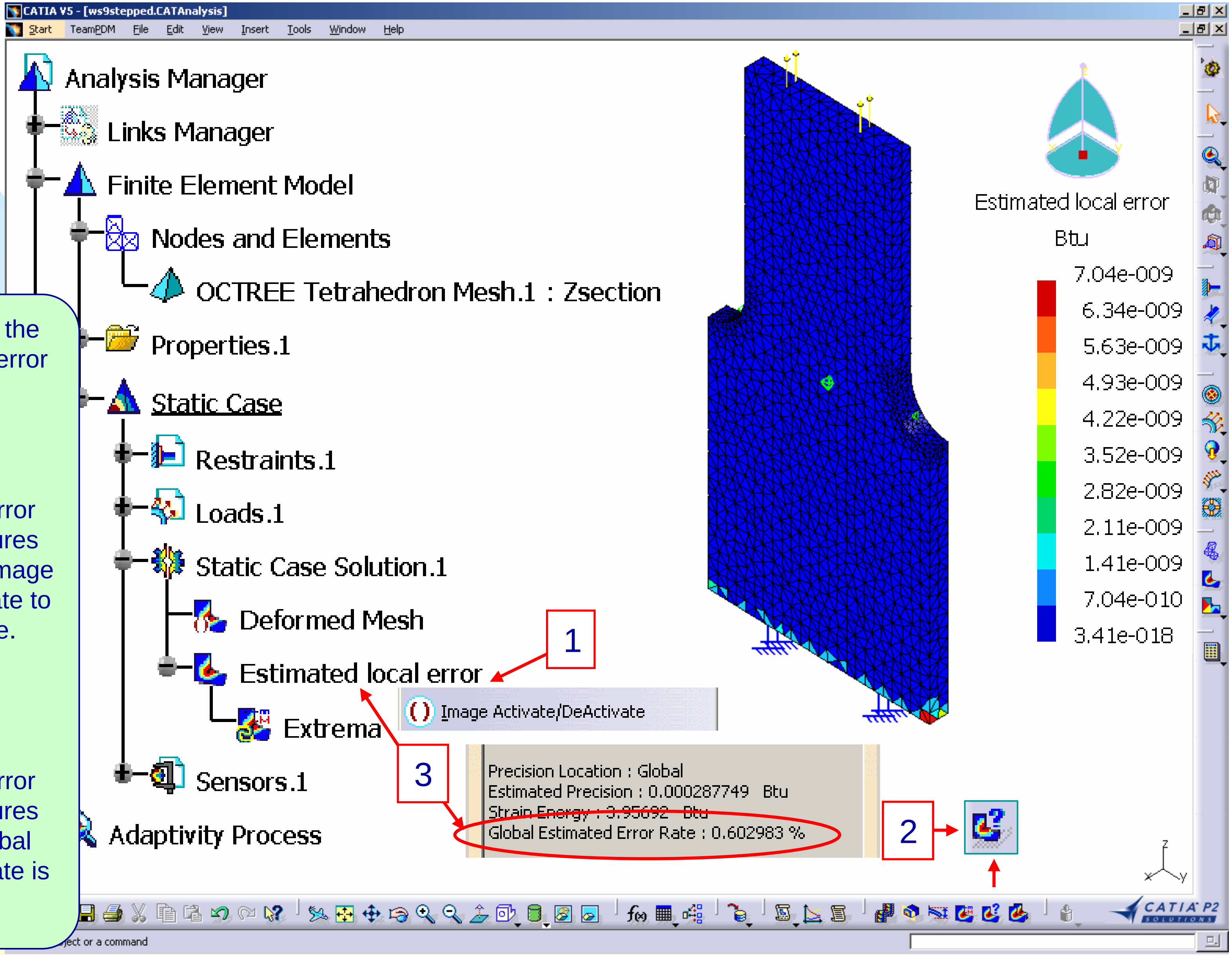
1. Select the Compute icon.

2. Compute All Objects defined, select OK.

3. Always be aware of these values, select Yes.

Save often.

Step 8. Compute the precise analysis



Check how much the global estimated error has improved.

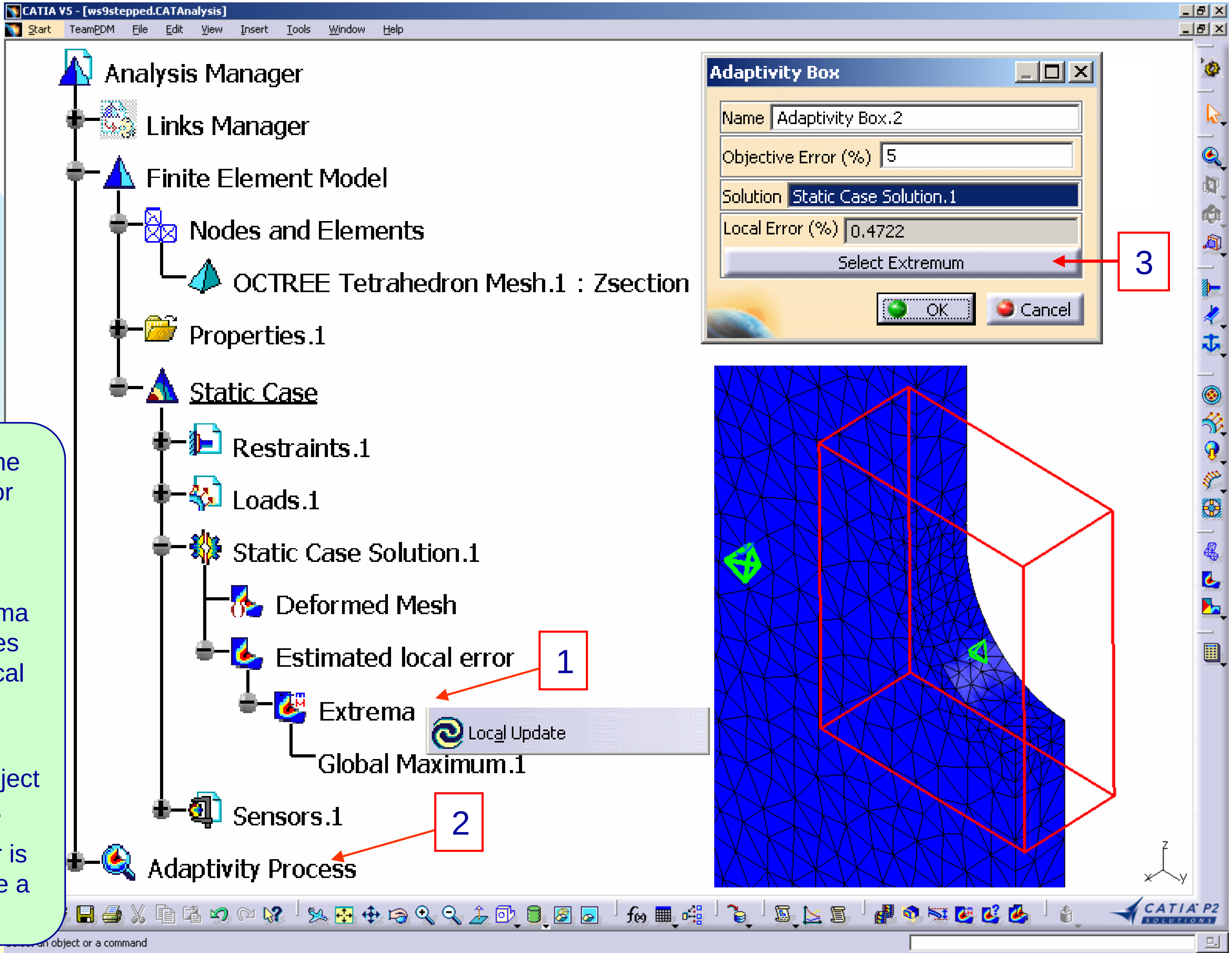
Steps:

1. Right click the Estimated local error object in the features tree then select Image Activate/DeActivate to activate the image.

2. Select on the information icon.

3. Select the Estimated local error object in the features tree. Note the global estimated error rate is very good.

Step 8. Compute the precise analysis



Check how much the local estimated error has improved.

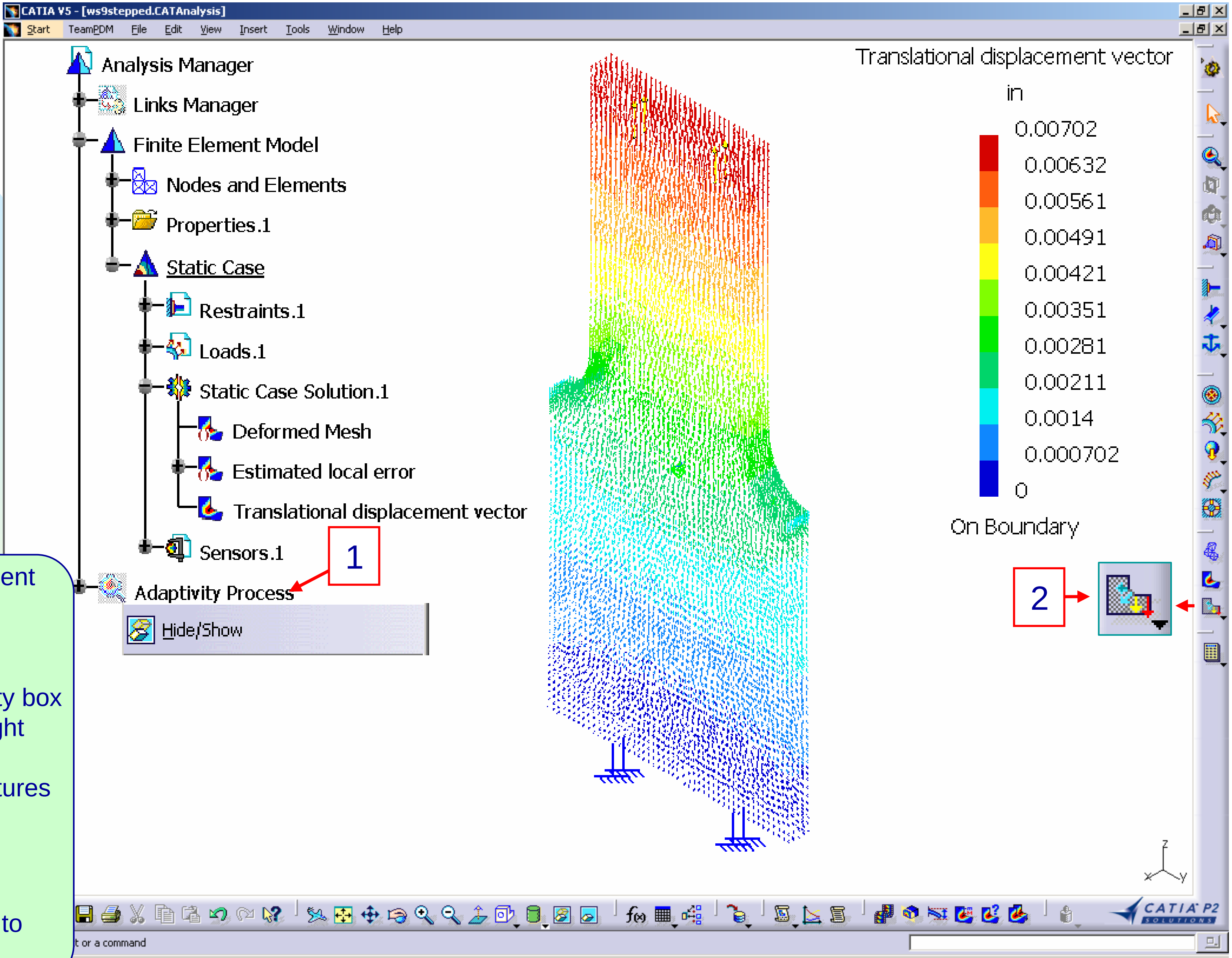
Steps:

1. Right click Extrema object in the features tree then select Local Update.

2. Double click the Adaptivity Box.1 object in the features tree.

3. Since local error is below 10% we have a precise model.

Step 9. Visualize final results



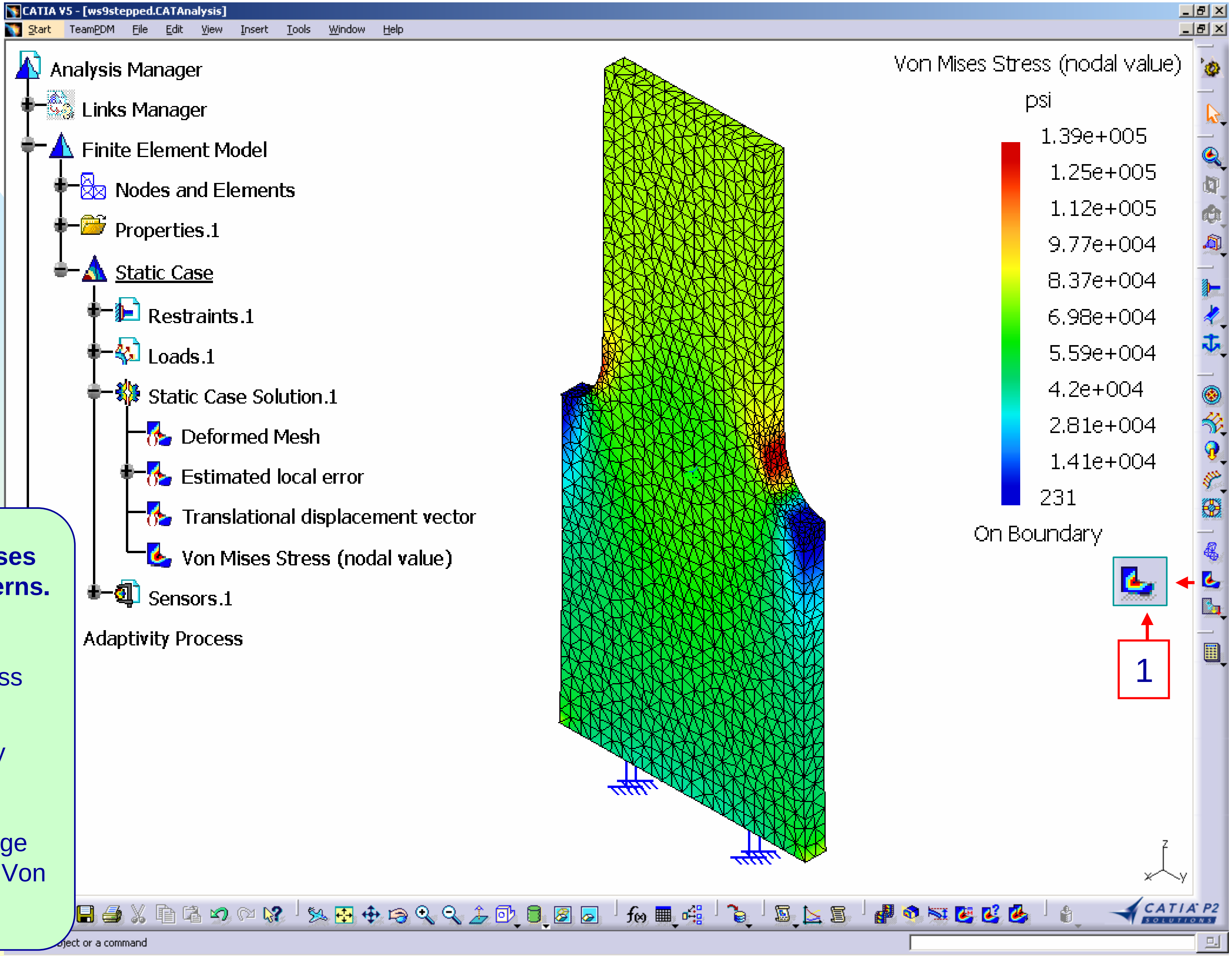
Add the displacement image.

Steps:

1. Put the adaptivity box into no show by right clicking Adaptivity Process in the features tree then select Hide/Show.

2. Select the displacement icon to add this image.

Step 9. Visualize final results



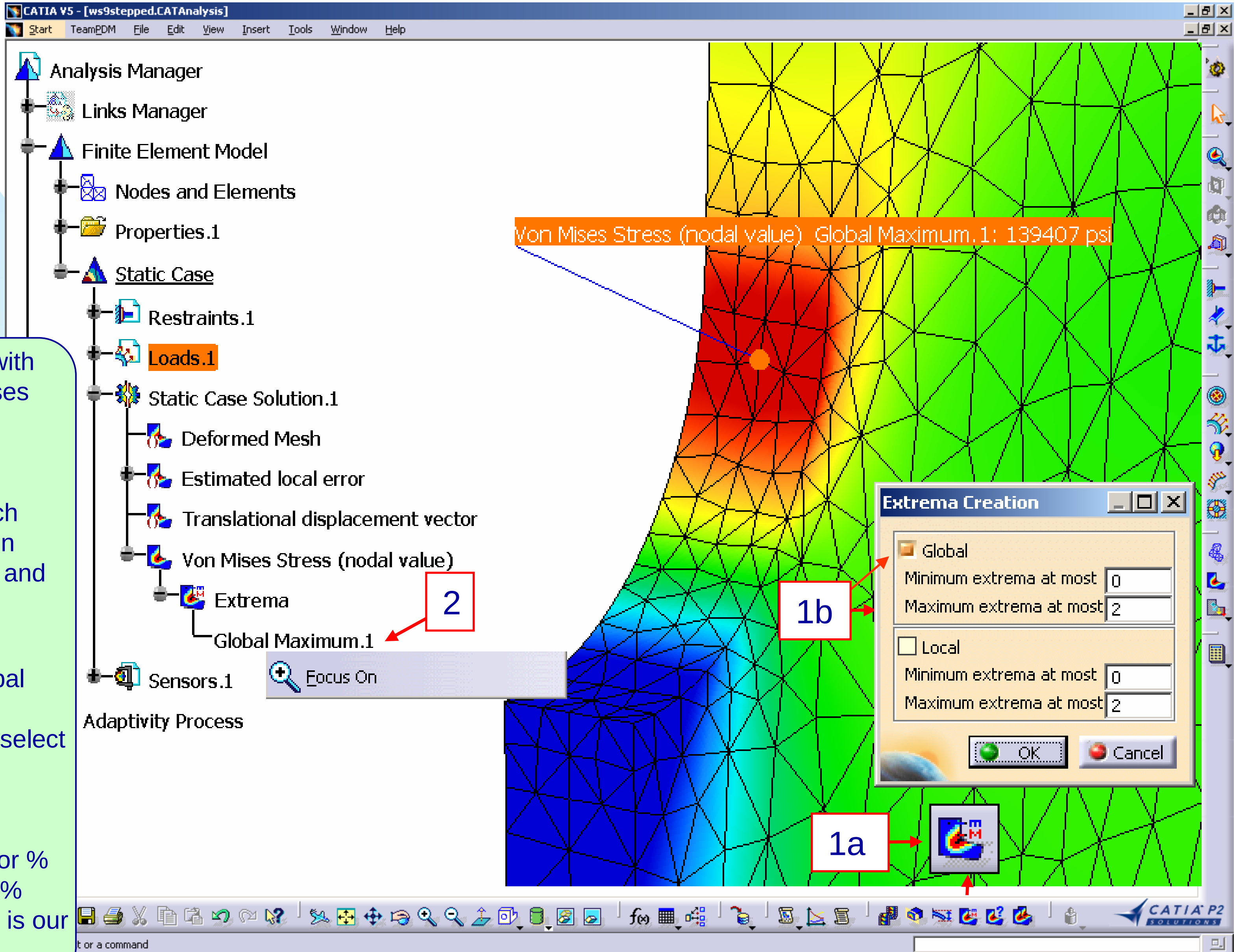
Visualize **Von Mises stress field patterns.**

Steps:

1. Select the Stress Von Mises icon.

This automatically deactivates the Translational displacement image and activates the Von Mises image.

Step 9. Visualize final results



Find the element with maximum Von Mises Stress.

Steps:

1. Select the search image extrema icon then select Global and key in 2 Maximum extrema at most.

2. Right click Global Maximum.1 in the features tree then select Focus On.

Since the local error % in this area is .477% (virtually zero) this is our design stress.

Step 9. Visualize final results

Find exact recommend design stress.

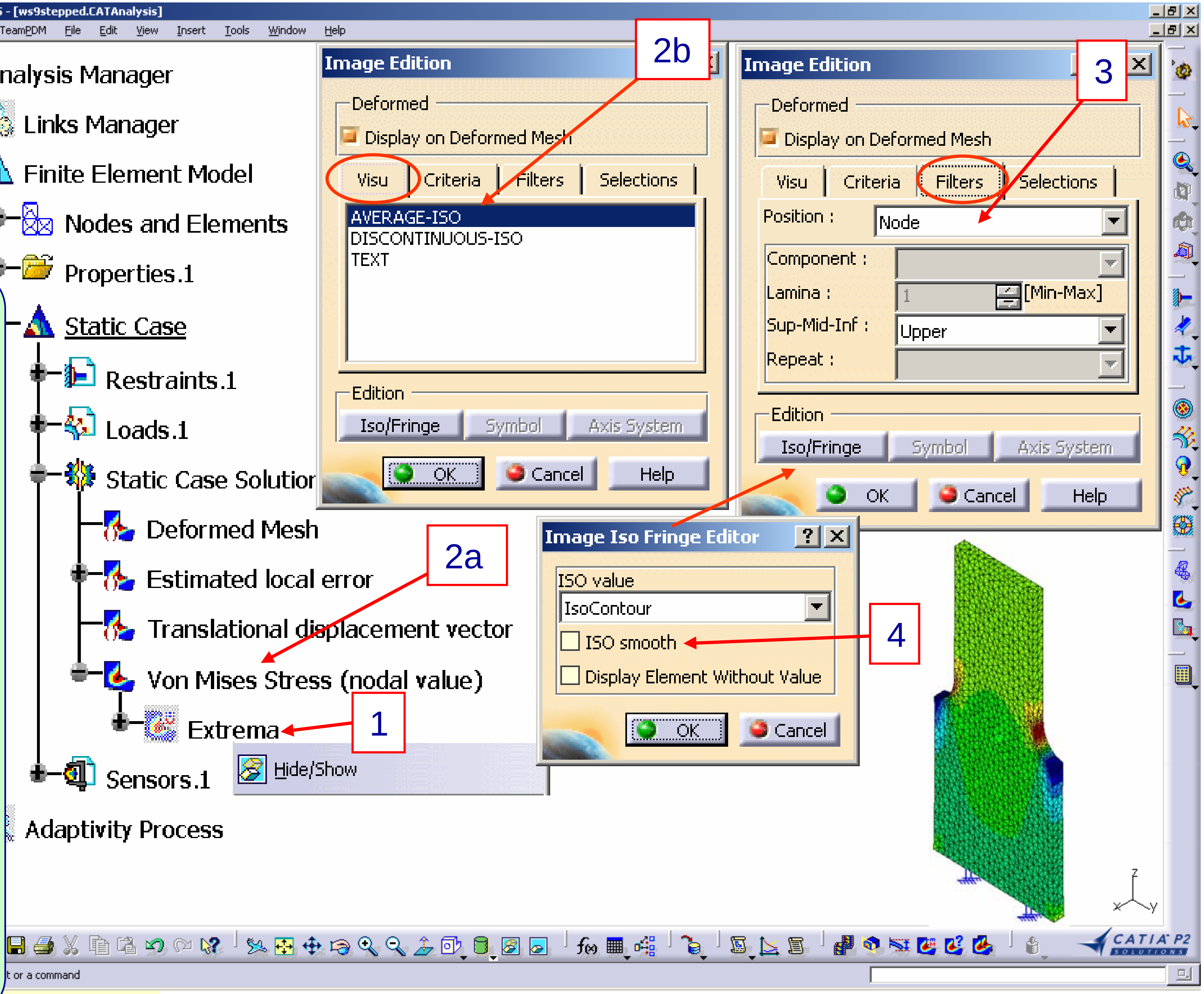
Steps:

1. Right click Extrema in the features tree then select Hide/Show.

2. Double click Von Mises Stress object in the features tree. Note you are looking at stress values averaged across elements.

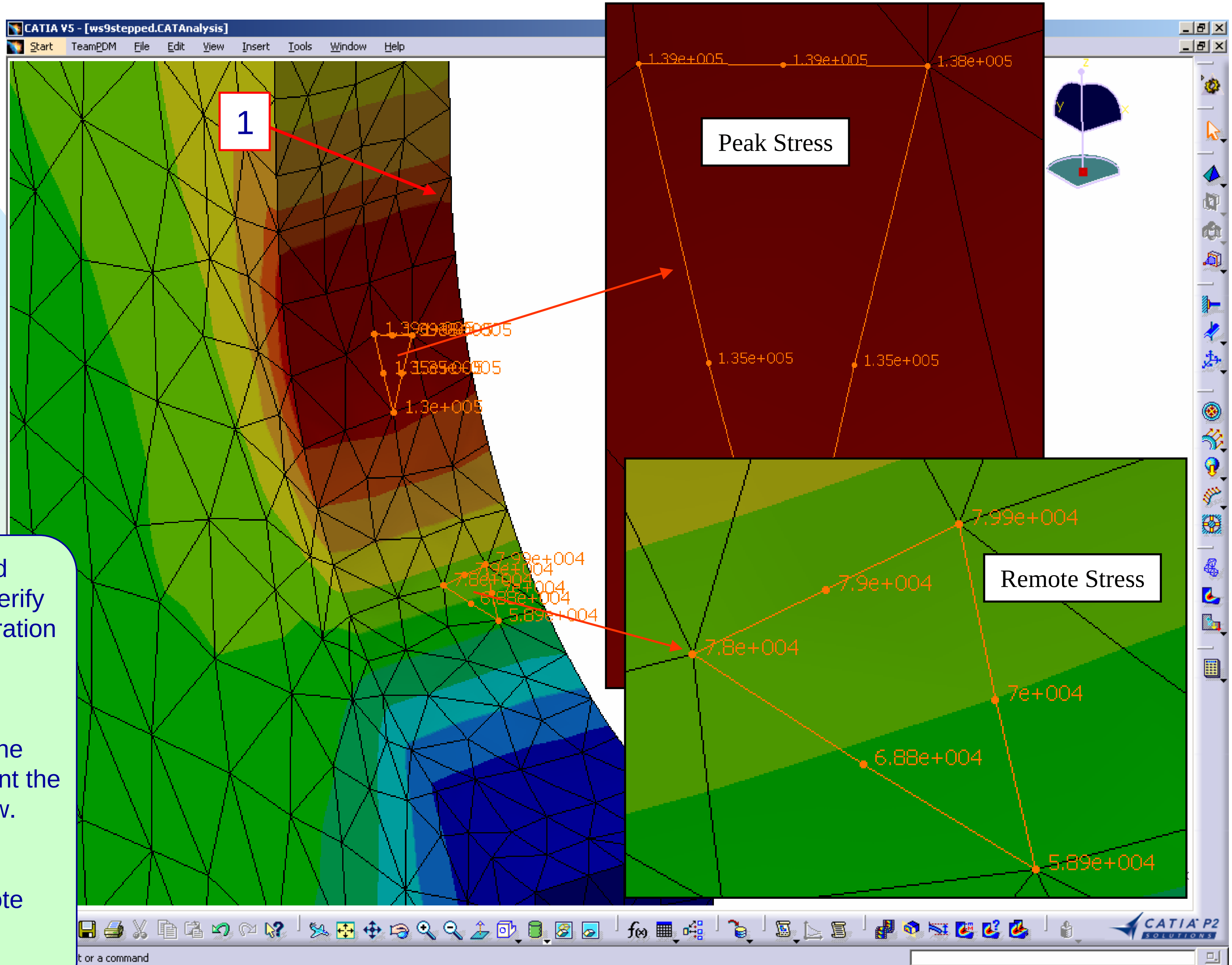
3. Also by selecting the Filters tab notice the stress output is calculated at the nodes.

4. Select Iso/Fringe and select the ISO smooth box to turn it off, select OK twice.



MSC SOFTWARE

Step 9. Visualize final results



Visualize peak and remote stress to verify the stress configuration factor K_t.

Steps:

1. By positioning the cursor on a element the stress values show.

Peak stress/Remote stress = K_t.

1.39e5/7.99e4=1.74

Conclusions

- CATIA V5 GSA workbench is validated for a stepped flat tension bar with shoulder fillets scenario.

	Hand Calculations	.1 inch Parabolic Global Mesh, .01 inch sag
Global % Precision error Local % Precision error	NA NA	0.6 % 0.47 %
Error Estimate	NA	7.04e-9 Btu global
Translational Displacement	0.0083 inch	0.00702 inch
Max Von Mises Stress	139200 psi	139407 psi

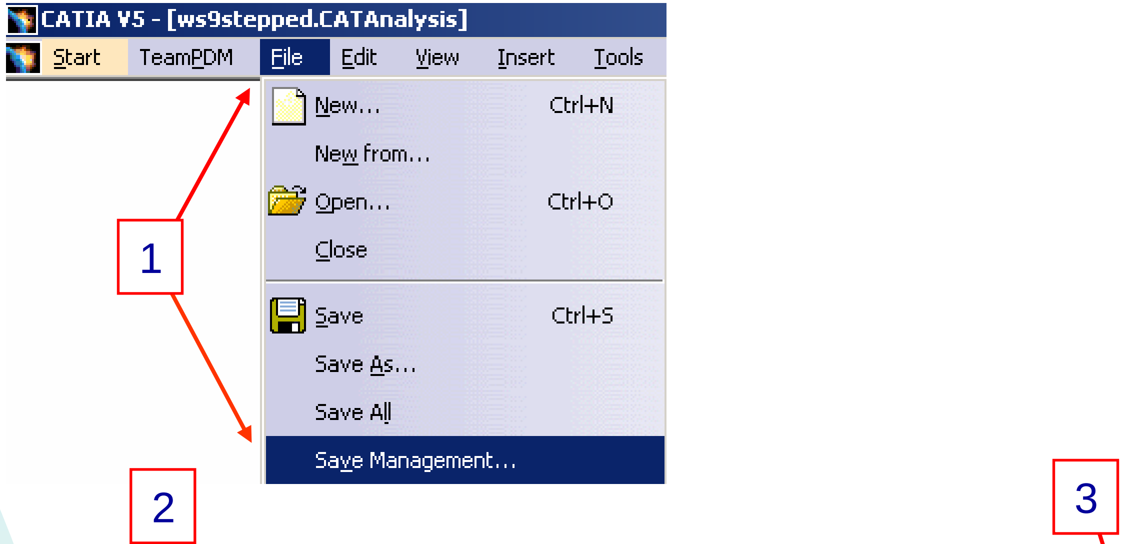
Step 10. Save the analysis document

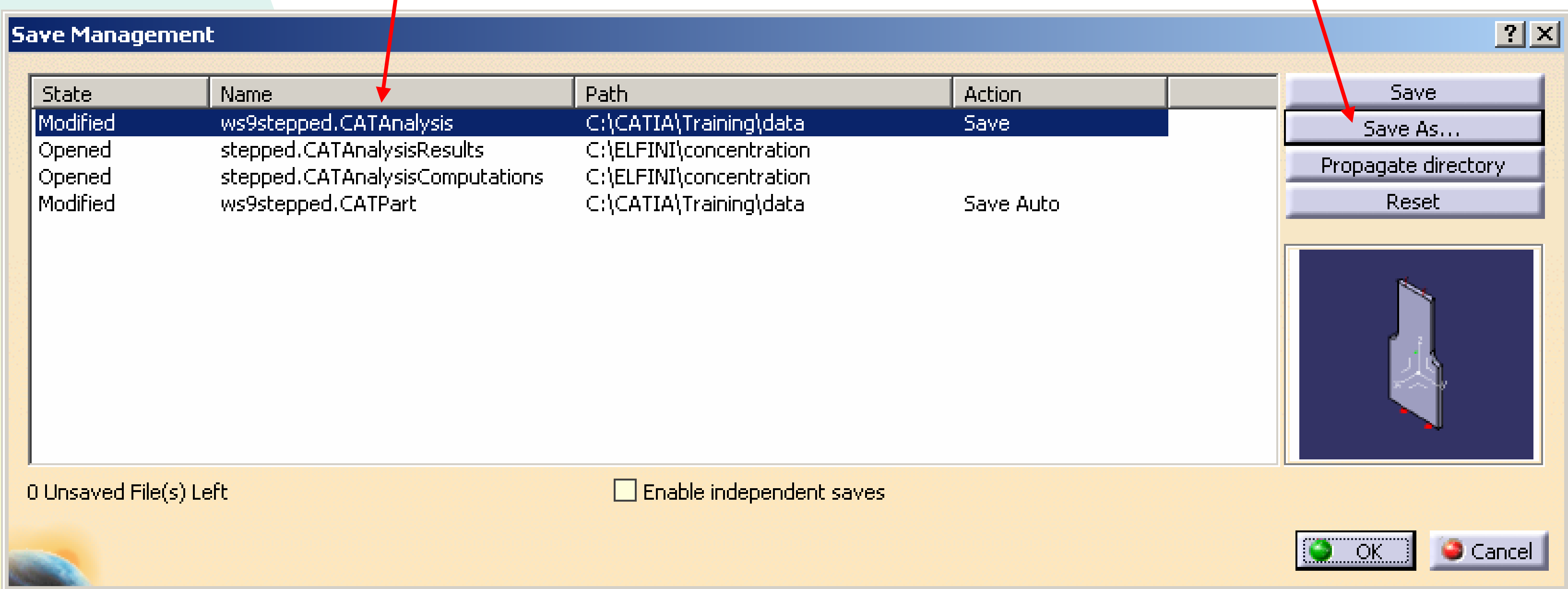
Steps:

1. Select Save Management from the File menu.

2. Highlight document you want to save.

3. Select Save As to specify name and path, select, OK

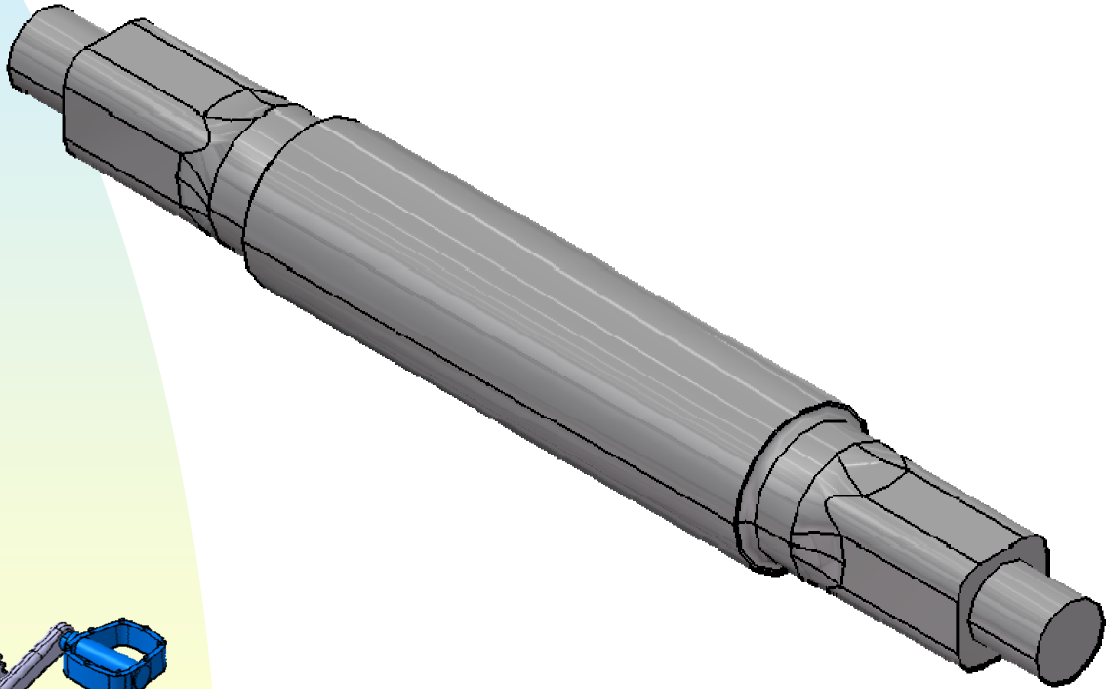


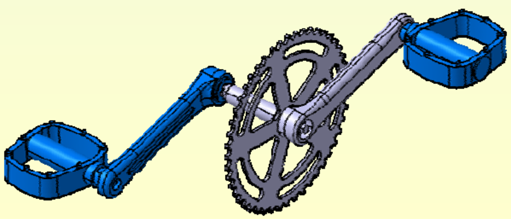


MSC SOFTWARE SIMULATING REALITY

WORKSHOP 9b

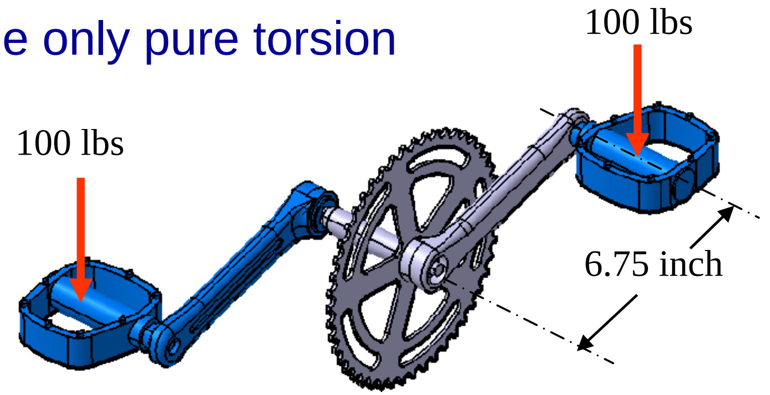
TORSION OF A SHAFT WITH A SHOULDER FILLET

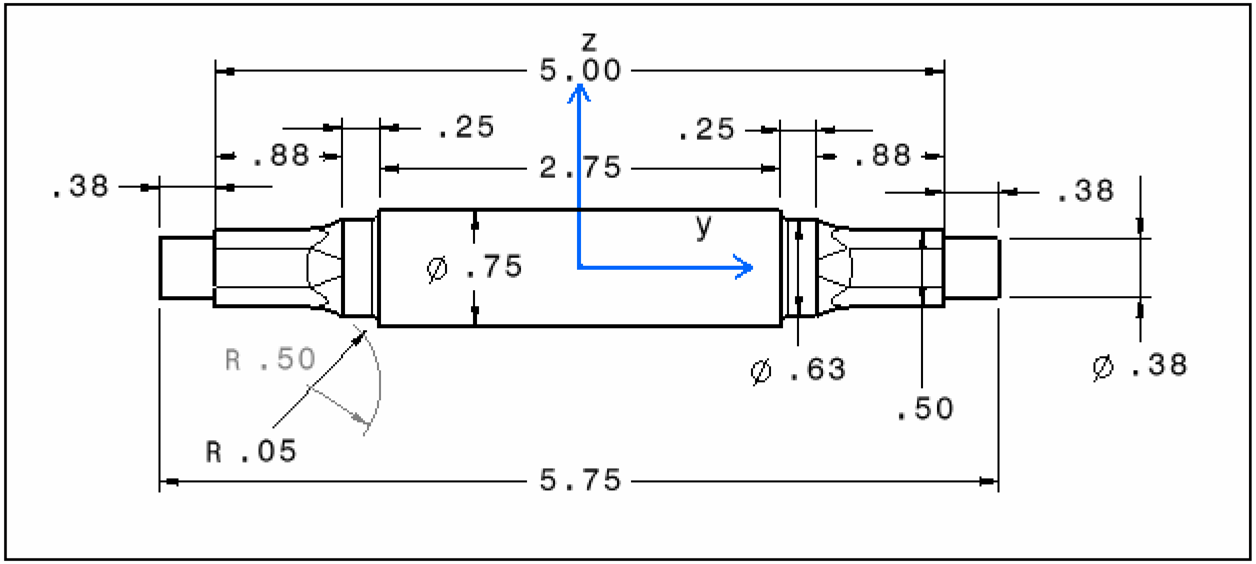




Problem Description

- Load case: Assume only pure torsion





Material: Steel
Young Modulus = 29e6 psi
Modulus of Rigidity = 12e6 psi
Poisson Ratio = .266
Density = .284 lb_in3
Yield Strength = 36259 psi

Hand calculations

- Maximum shear stress at the surface

$$\text{Maximum Shear Stress} = \tau_{max} = \tau_{nom} \cdot K_{ts} \qquad \tau_{nom} = \frac{16 \cdot T}{\pi \cdot d^3} = \frac{16 \cdot 1350 \text{ lb-inch}}{3.14 \cdot 0.63^3} = 27502 \text{ psi}$$

$$K_{ts} = 1.42 \qquad \tau_{max} = 27502 \text{ psi} \cdot 1.42 = 39053 \text{ psi}$$

- Maximum angle of twist

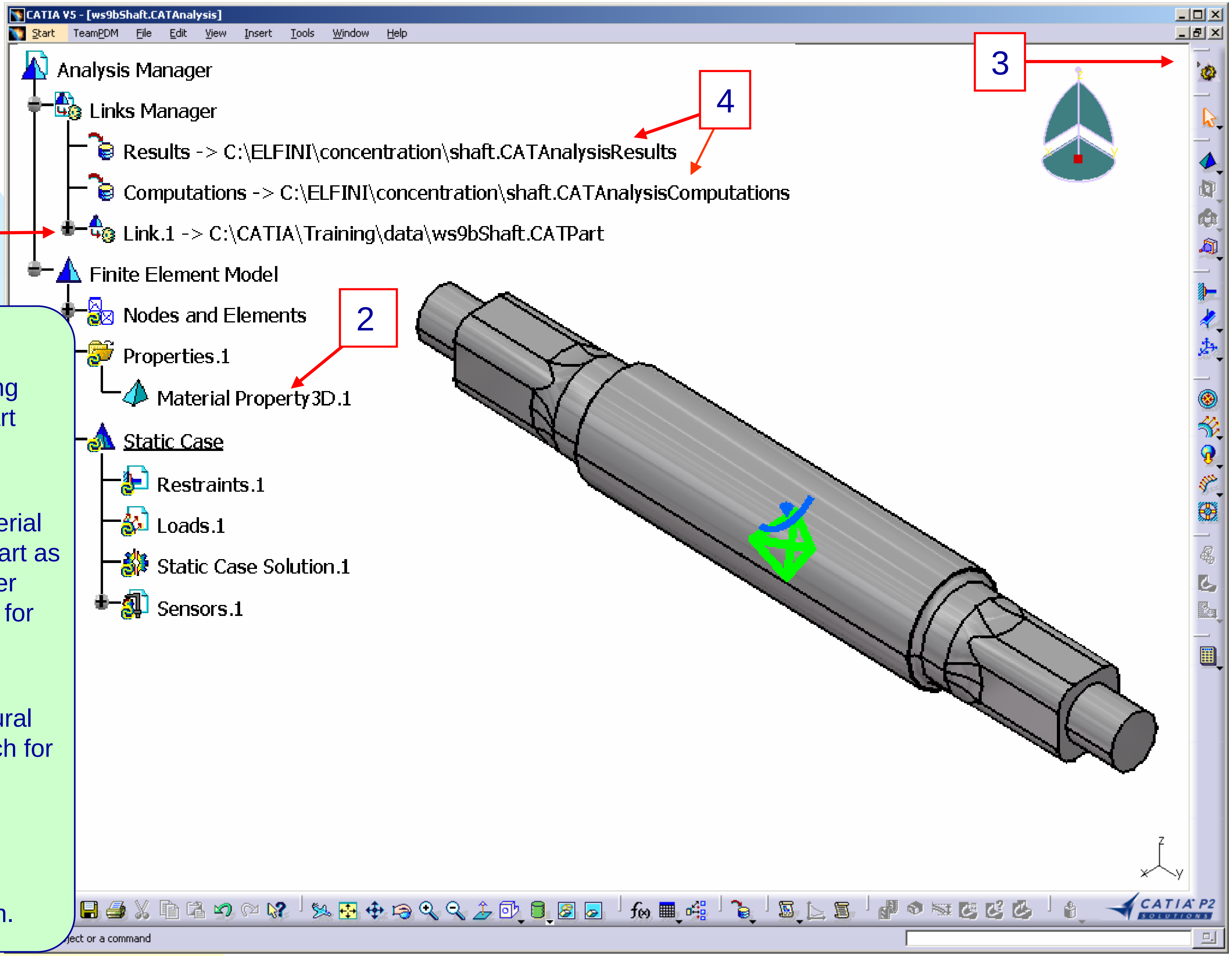
$$\text{Angle of twist (radians)} = \theta = \frac{T \cdot L}{J \cdot G} \qquad J = 2 \cdot I = 2 \cdot \frac{\pi}{4} \cdot R^4 = 2 \cdot \frac{3.14}{4} \cdot 0.375^4 \text{ inch} = 0.031 \text{ inch}^4$$

$$\theta = \frac{1350 \text{ lb-inch} \cdot 5.01 \text{ inch}}{0.031 \text{ inch}^4 \cdot 12e^6 \text{ psi}} = 0.0182 \text{ radians (1.04 degrees)}$$

Suggested Exercise Steps

1. Create a new CATIA analysis document (.CATAnalysis).
2. Mesh globally with linear elements.
3. Apply a clamp restraint.
4. Apply a moment force.
5. Compute the initial analysis.
6. Check global and local precision (animate deformation, adaptive boxes and extremas).
7. Change global and local mesh size.
8. Compute the precise analysis.
9. Visualize final results.
10. Save the analysis document.

Step 1. Create a new CATIA analysis document



Steps:

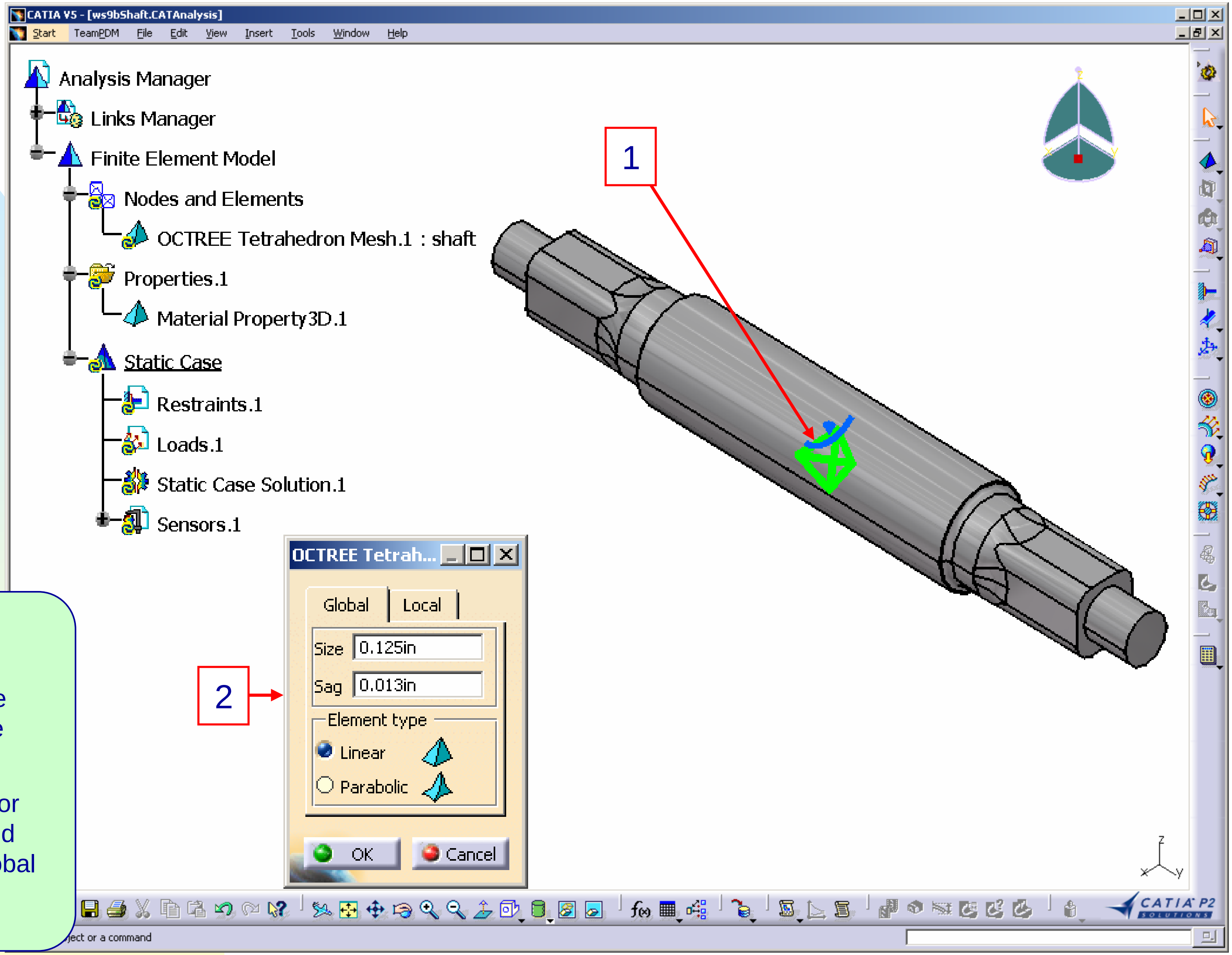
1. Open the existing ws9bShaft.CATPart from the training directory.

2. Apply steel material properties to the part as required (remember modulus of rigidity for torsion).

3. Launch the Generative Structural Analysis workbench for a Static Analysis.

4. Specify the Computations and Results storage locations as shown.

Step 2. Mesh globally with linear elements

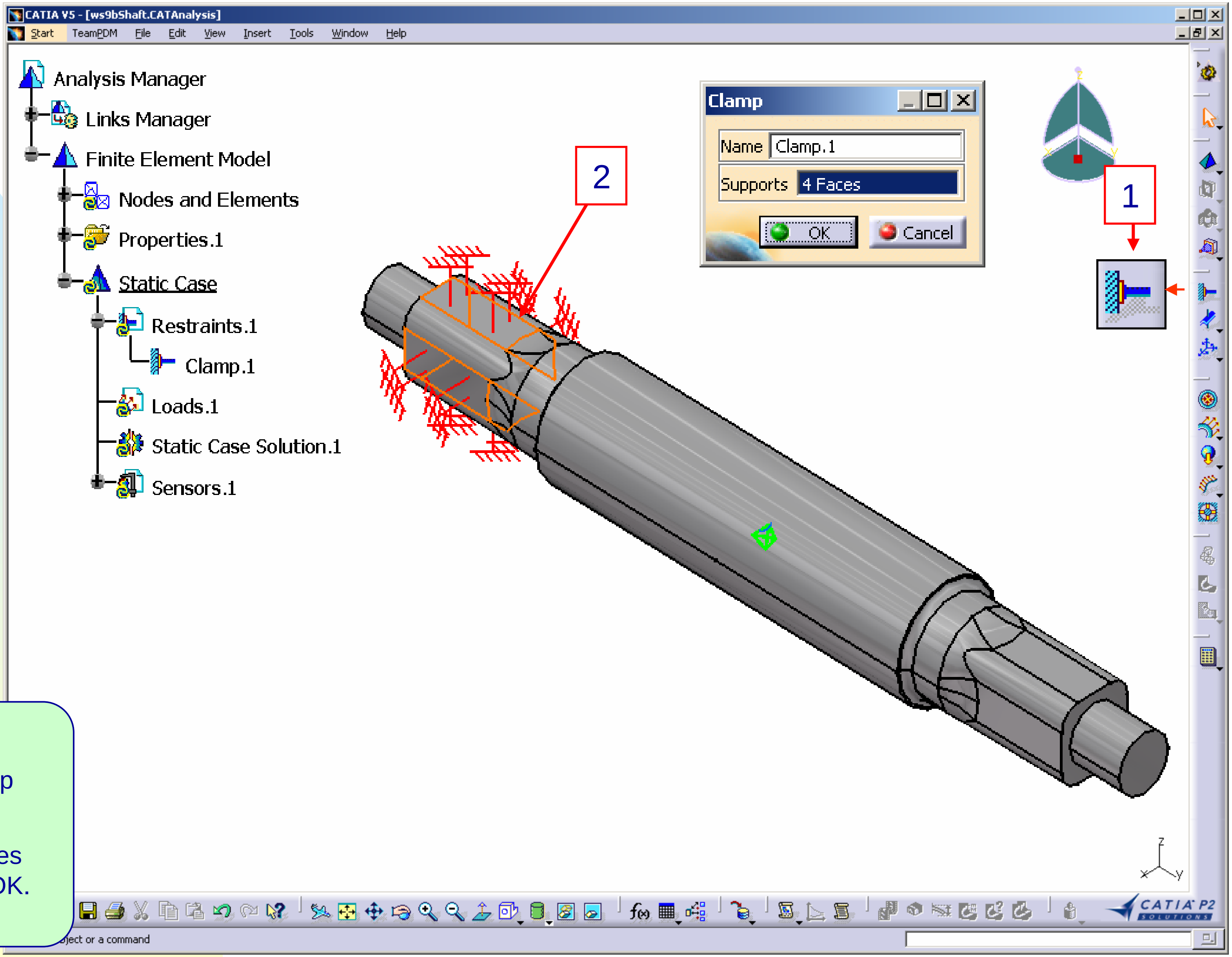


Steps:

1. Double Click the "Mesh" icon on the part.

2. Key in 0.125in for the Global Size and 0.013in for the Global sag, select OK.

Step 3. Apply a clamp restraint

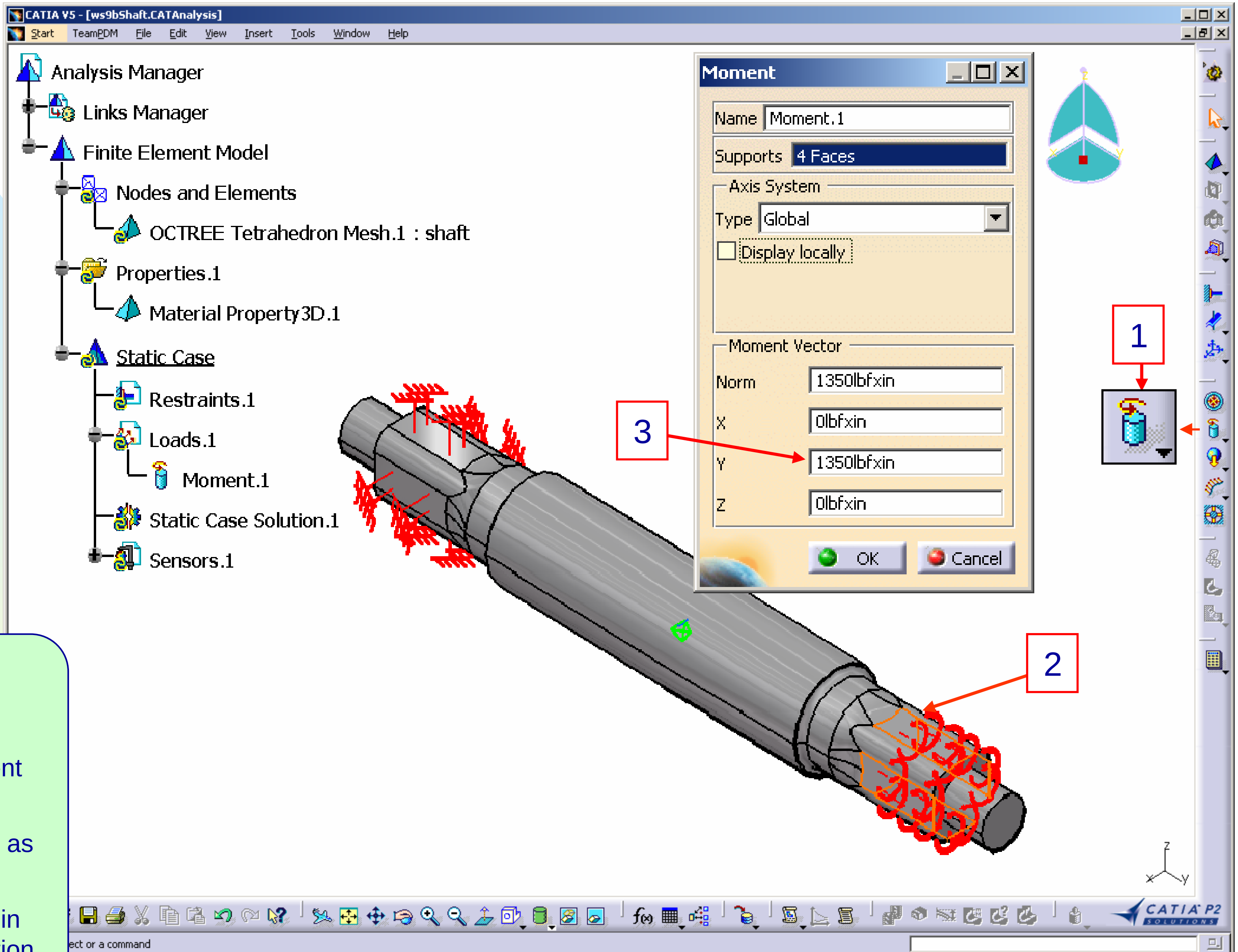


Steps:

1. Select the Clamp restraint icon.

2. Select the 4 faces as shown, select OK.

Step 4. Apply a moment force



Load only one end.

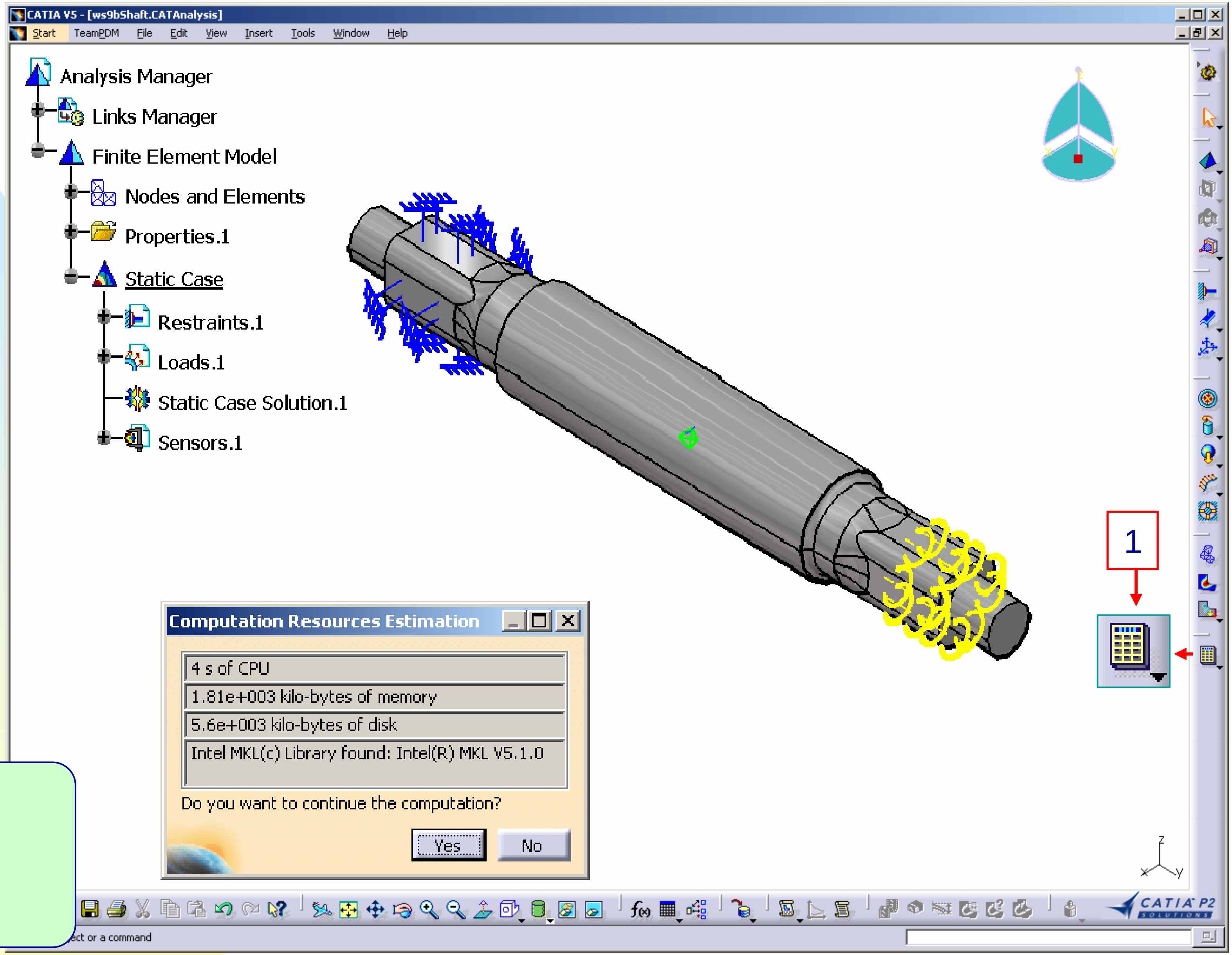
Steps:

1. Select the Moment icon.

2. Select four faces as shown.

3. Enter 1350lbfxin in the positive y-direction.

Step 5. Compute the initial analysis

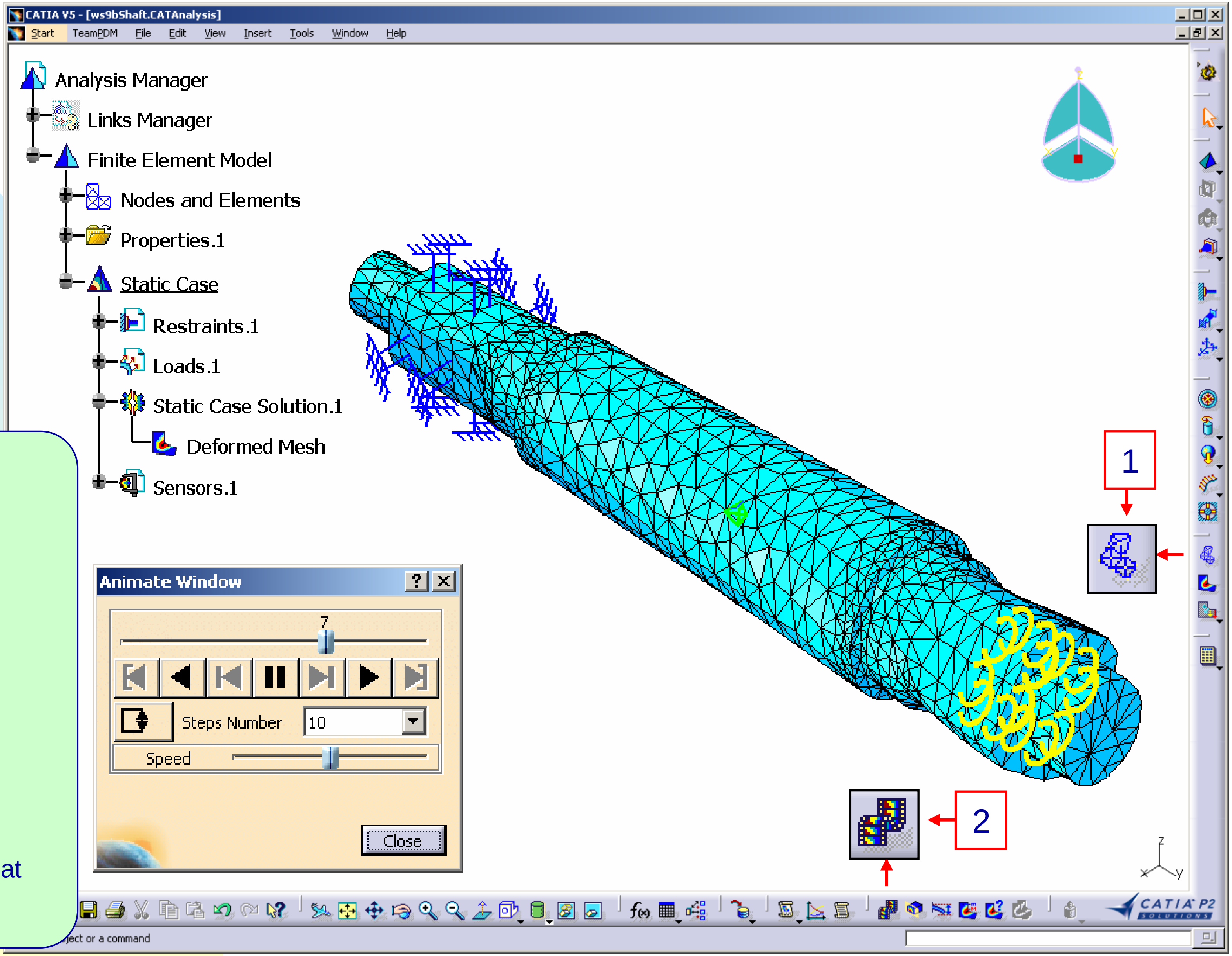


Save first.

Steps:

1. Compute All

Step 6. Check global and local precision



Visualize the **Deformation** and animate.

Steps:

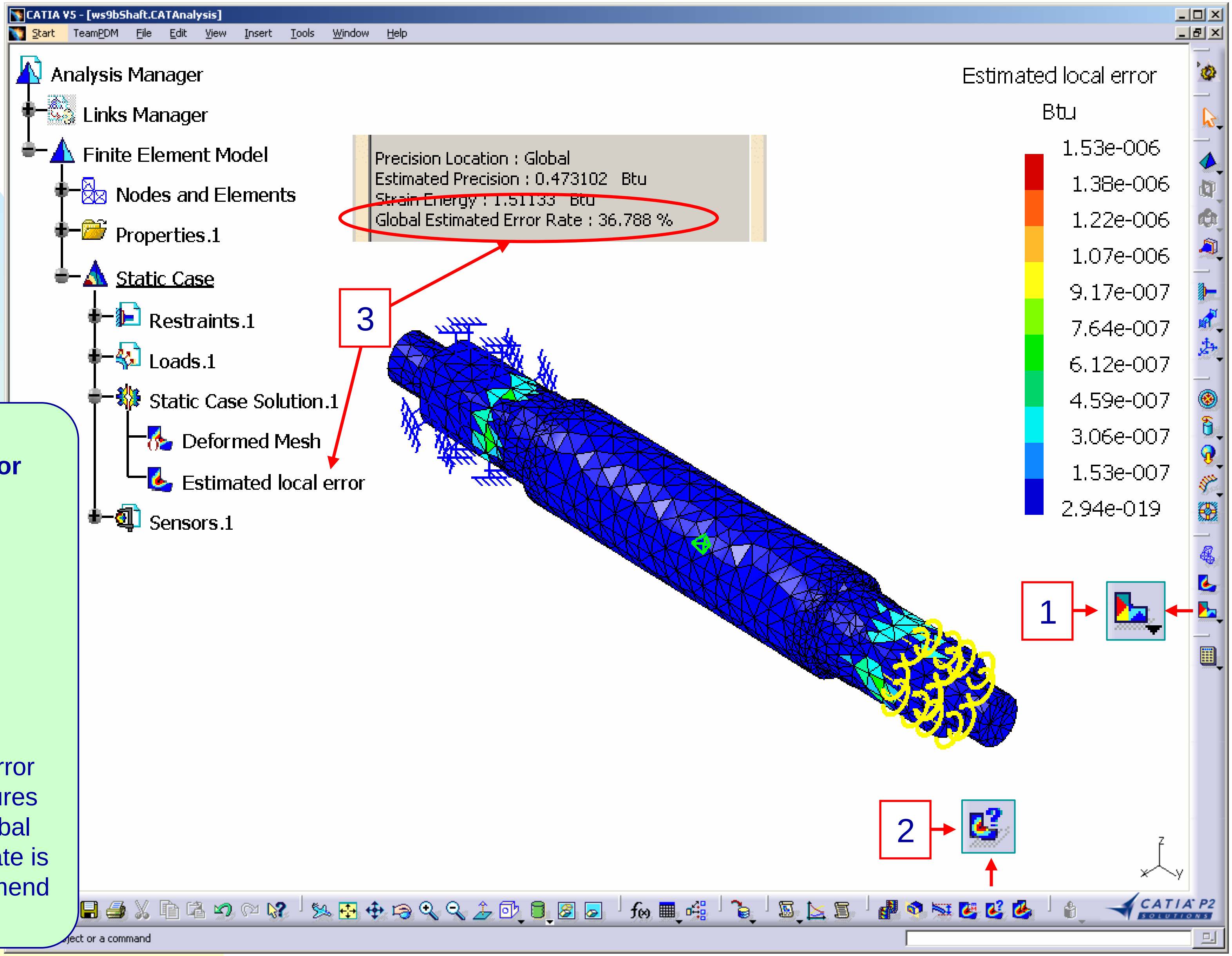
1. Select the Deformation icon.

2. Select on the Animate icon.

Verify that deformation is what you expect.

MSC SOFTWARE
SIMULATING REALITY

Step 6. Check global and local precision



Visualize the **computation error map.**

Steps:

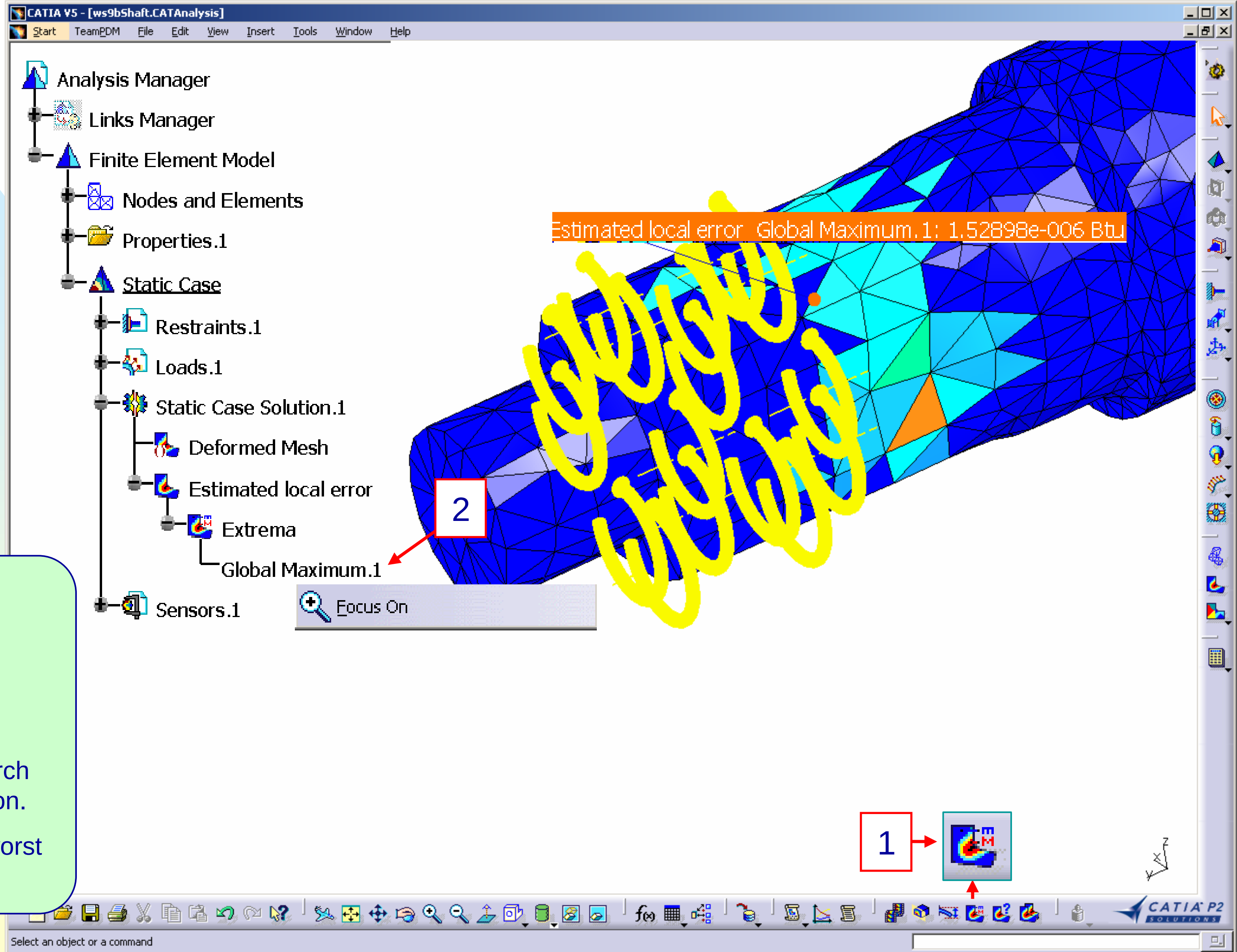
1. Select the Precision icon.

2. Select on the information icon.

3. Select the Estimated local error object in the features tree. Note the global estimated error rate is not good (recommend max 20%).

MSC SOFTWARE

Step 6. Check global and local precision



Find the global element with the highest estimated error.

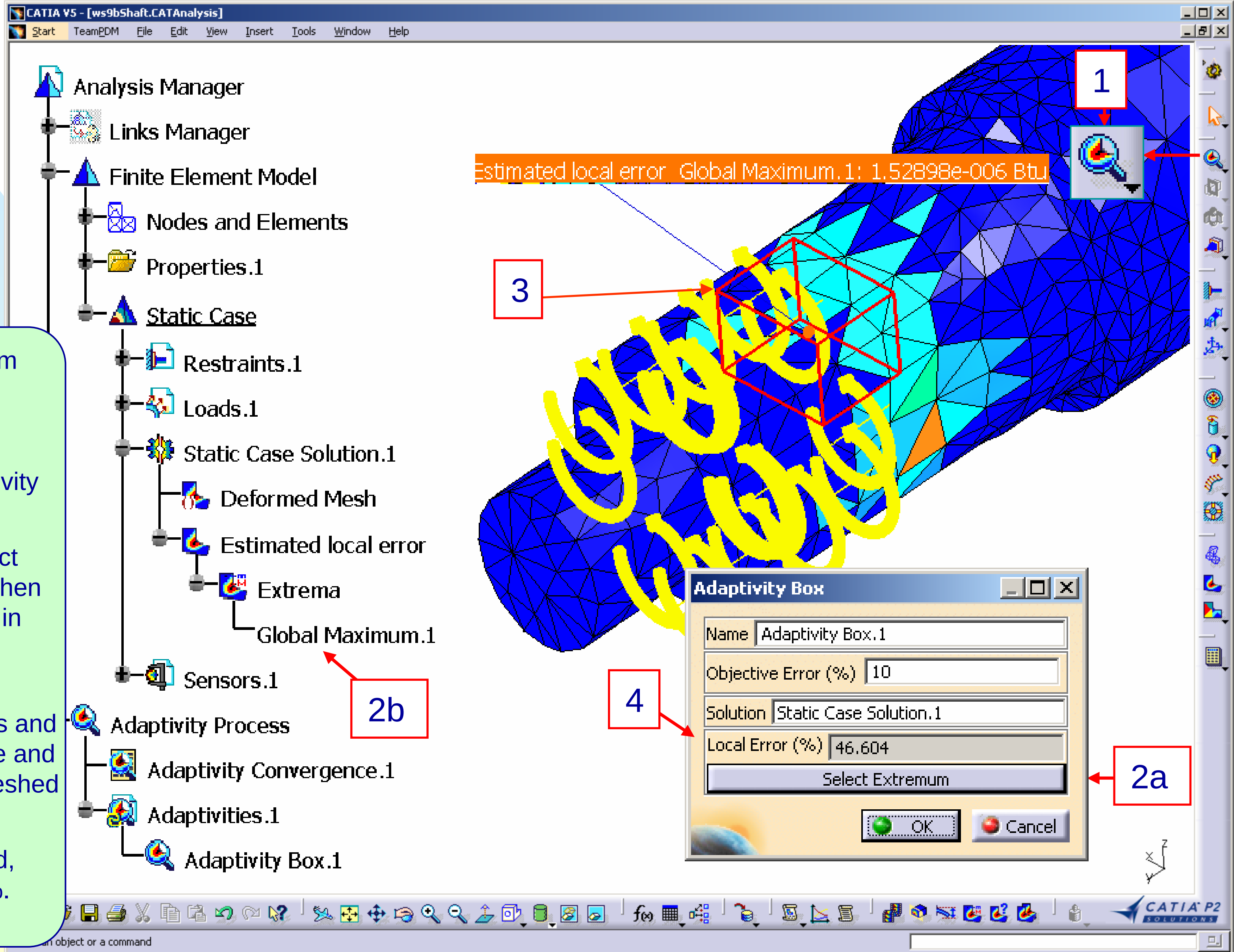
Steps:

1. Select the Search Image Extrema icon.

2. Focus On the worst element.

MSC SOFTWARE

Step 6. Check global and local precision



Determine maximum local error %.

Steps:

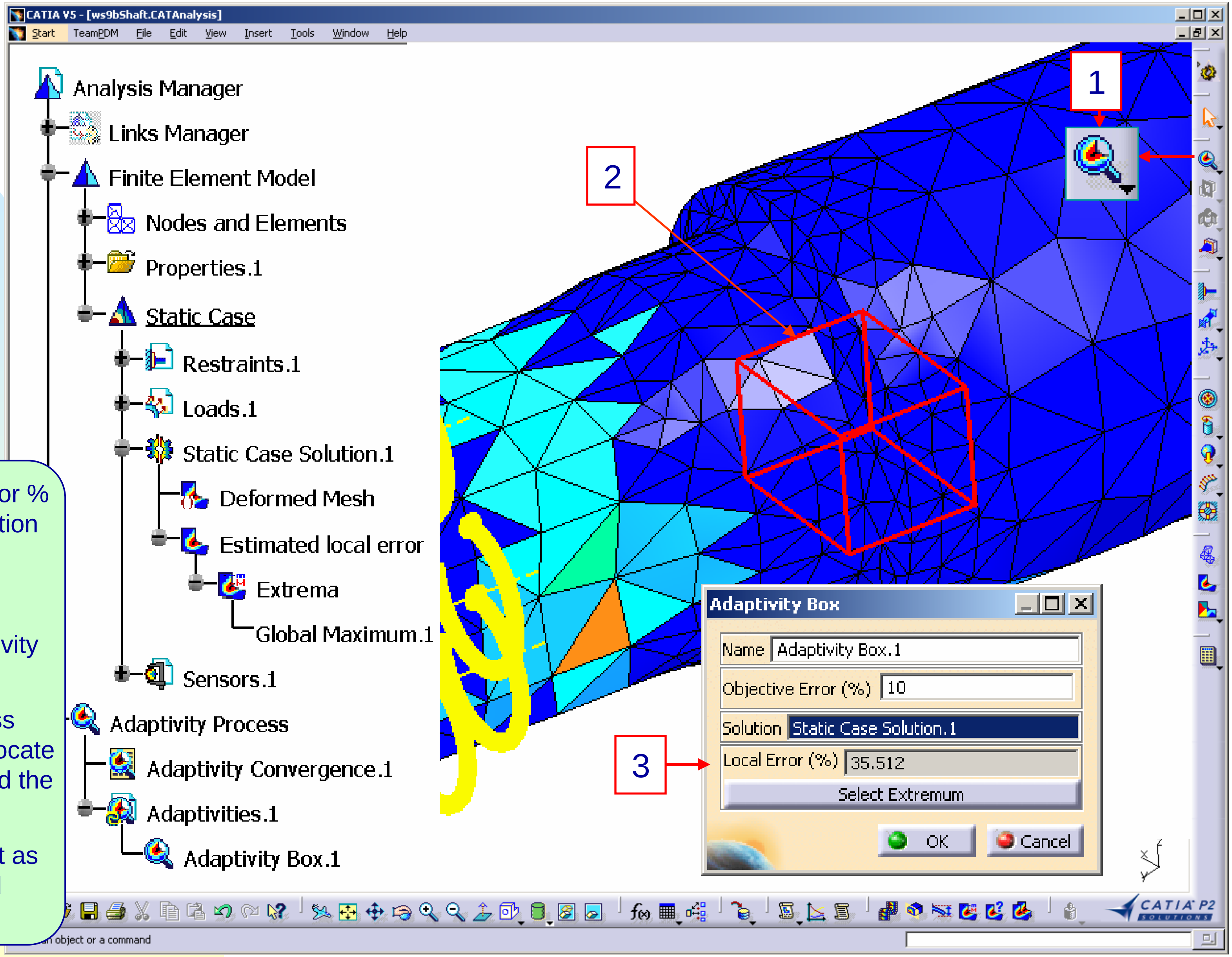
1. Select the adaptivity box icon.

2. Select the "Select Extremum" button then Global Maximum.1 in the features tree to locate box.

3. Use the compass and green dots to locate and size box around meshed areas.

4. Local error is bad, recommended 10%.

Step 6. Check global and local precision



Determine local error % at stress concentration area.

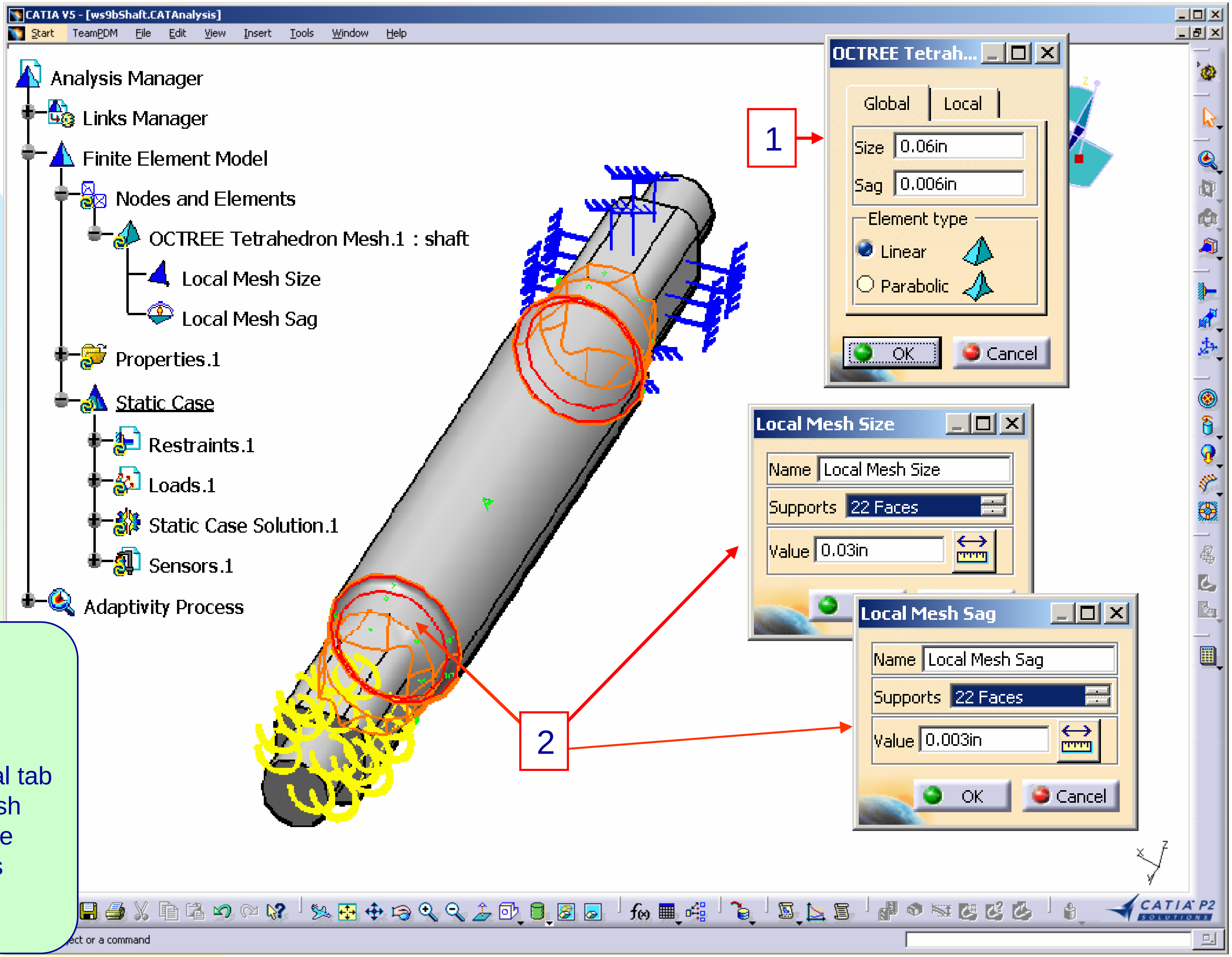
Steps:

1. Select the adaptivity box icon.

2. Use the compass and green dots to locate and size box around the shoulder area.

3. Local error is just as bad, recommended 10%.

Step 7. Change global and local mesh size

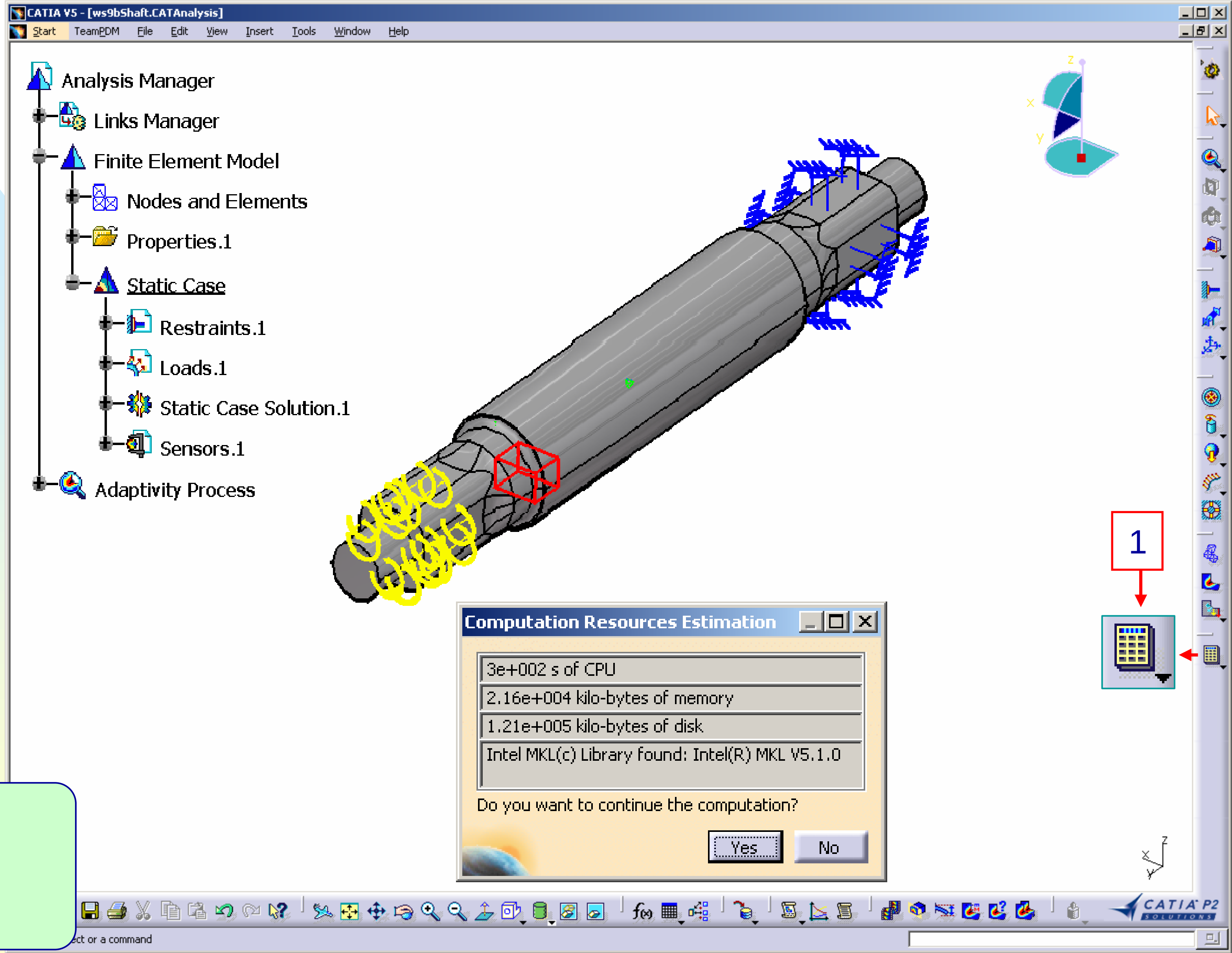


Steps:

1. Change global mesh as shown.

2. Select the Local tab and add local mesh size and sag in the shoulder areas as shown select OK.

Step 8. Compute the precise analysis

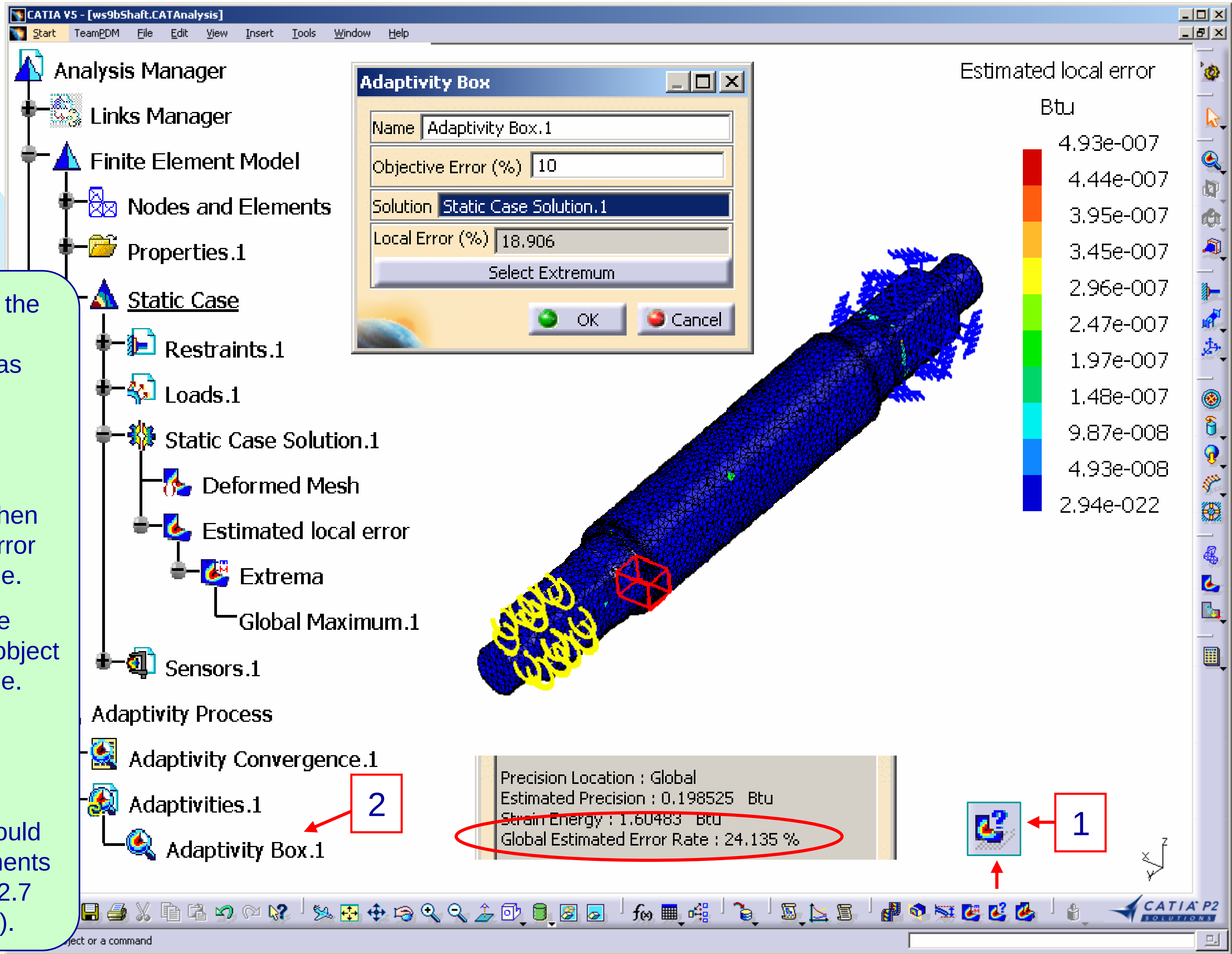


Save first.

Steps:

1. Compute All

Step 8. Compute the precise analysis



Check how much the global and local estimated error has improved.

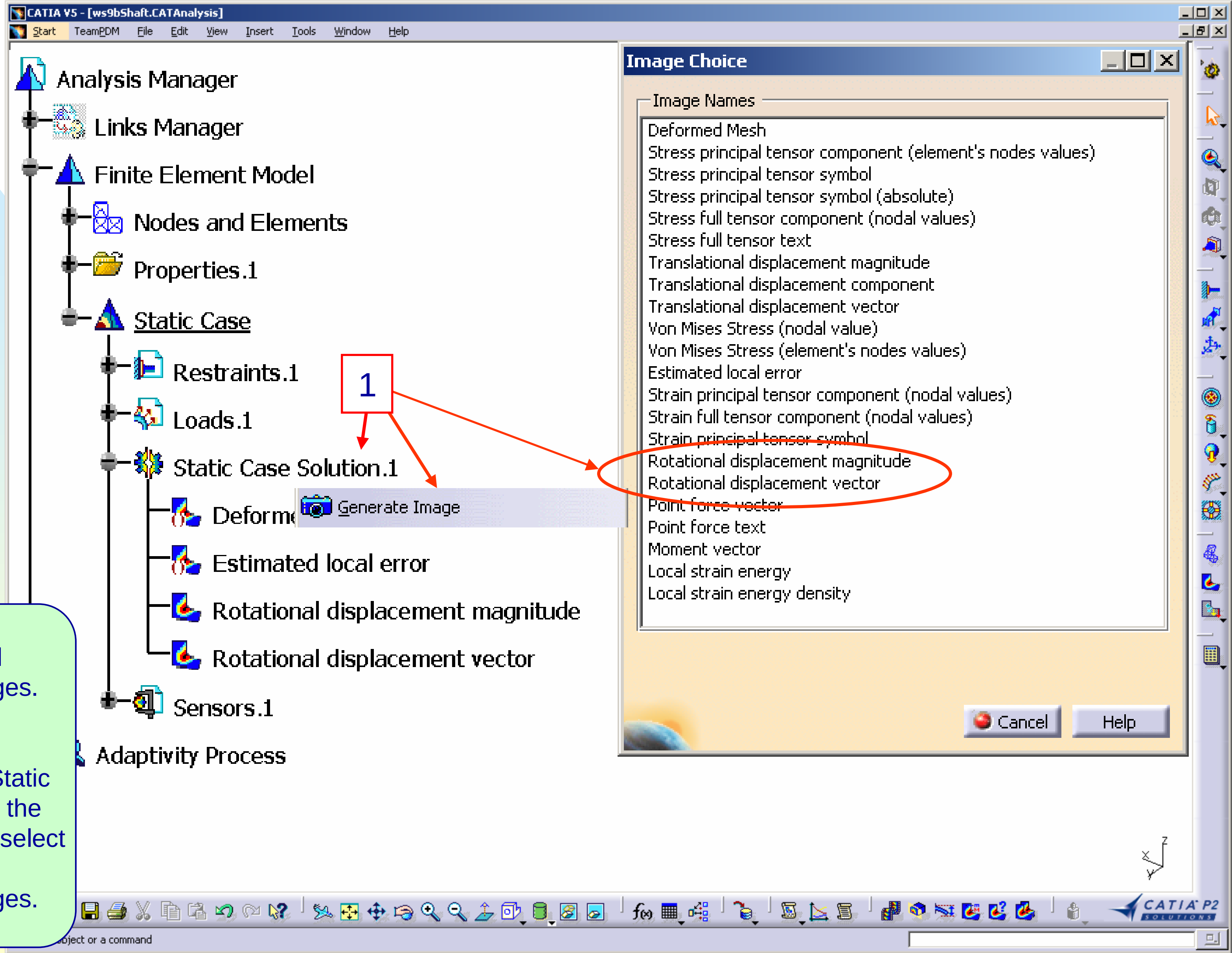
Steps:

1. Select on the information icon then Estimated local error in the features tree.

2. Double click the Adaptivity Box.1 object in the features tree.

Still not per the recommended minimums, you could try parabolic elements if you have time (2.7 hours and 1.9 gig).

Step 9. Visualize final results

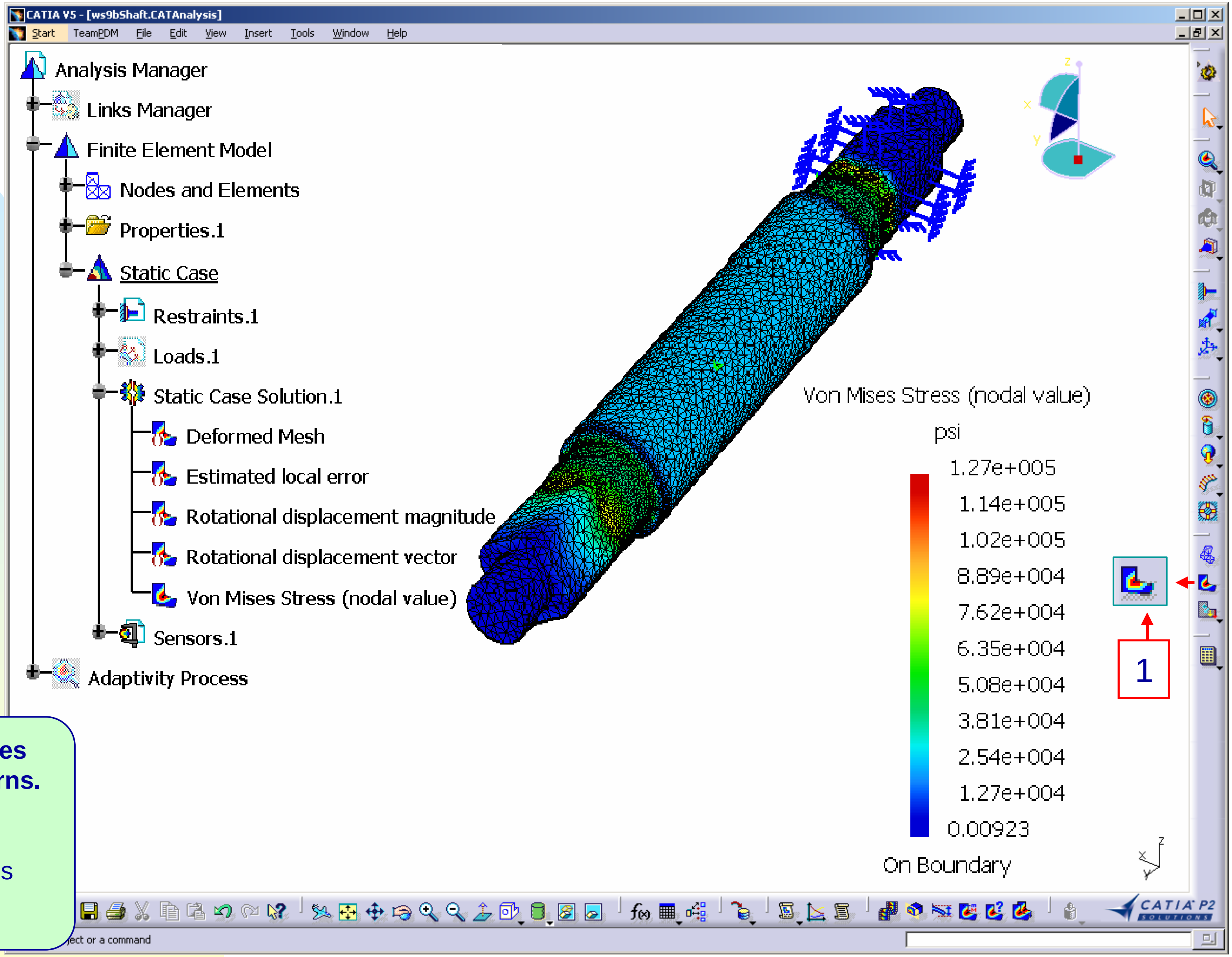


Add the Rotational displacement images.

Steps:

1. Right click the Static Case Solution.1 in the features tree then select the Rotational displacement images.

Step 9. Visualize final results



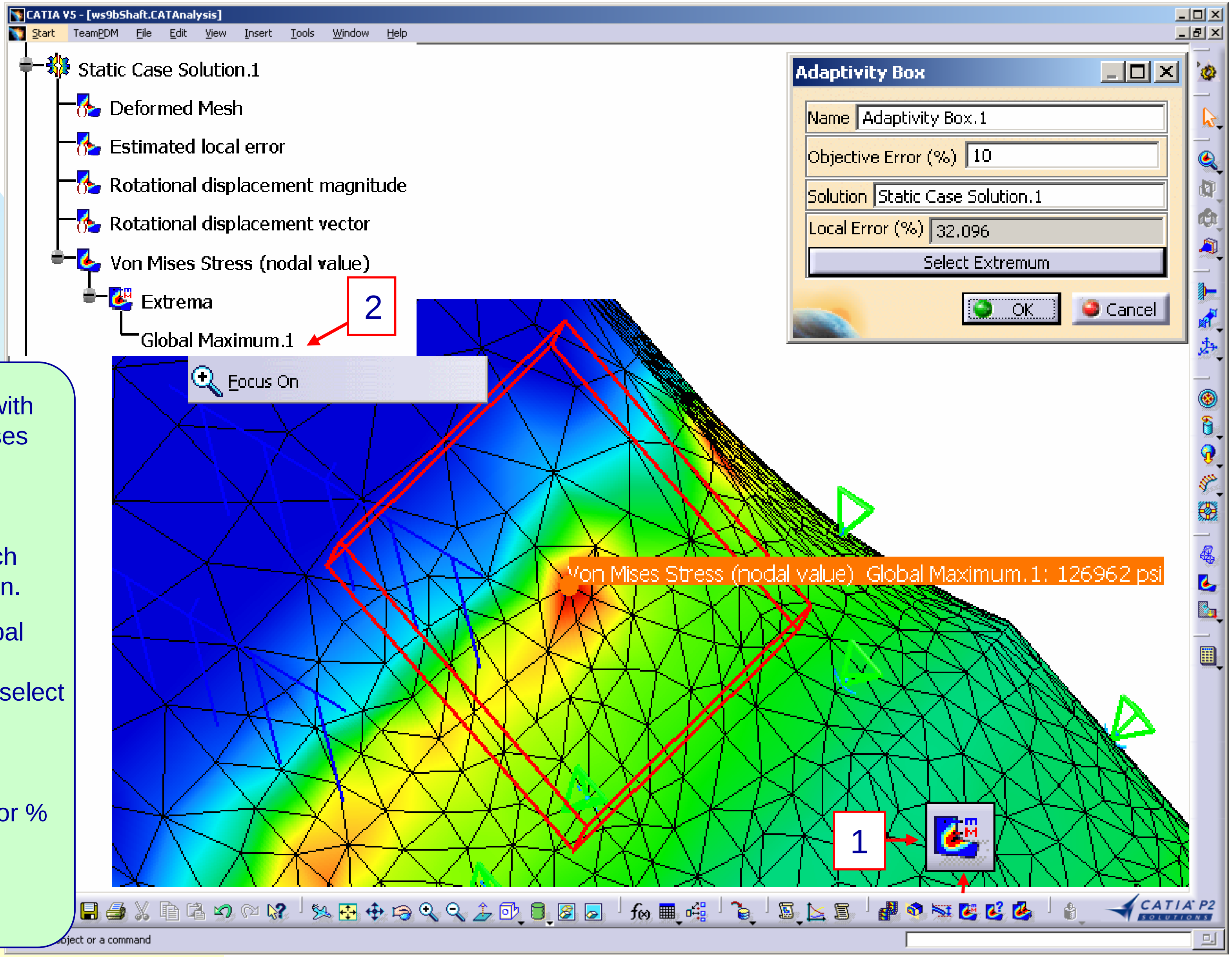
Visualize **Von Mises stress field patterns.**

Steps:

1. Select the Stress Von Mises icon.

MSC SOFTWARE

Step 9. Visualize final results



Find the element with maximum Von Mises Stress.

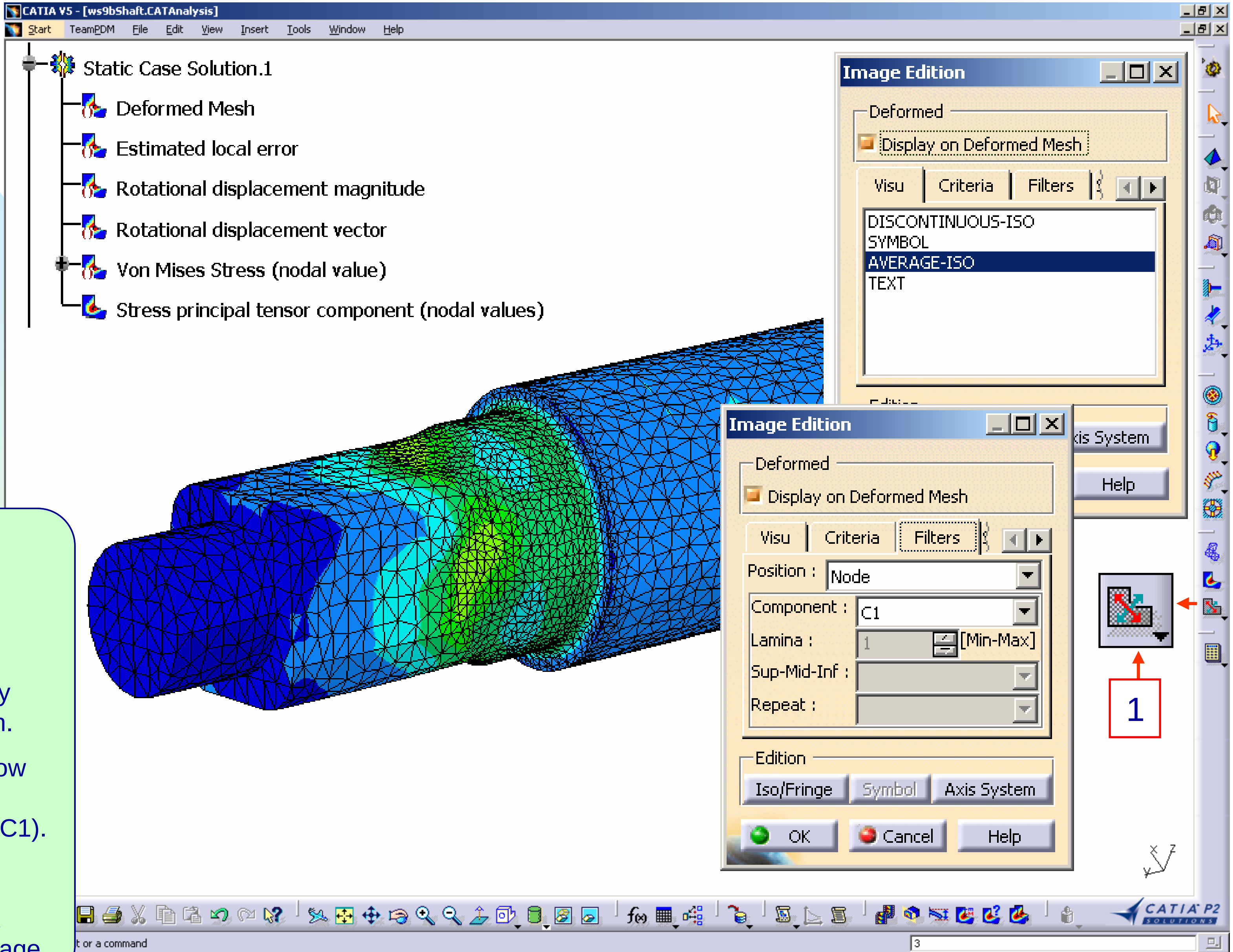
Steps:

1. Select the search image extrema icon.

2. Right click Global Maximum.1 in the features tree then select Focus On.

Since the local error % in this area is 32% design stress is questionable.

Step 9. Visualize final results



Display Principal Stresses to verify maximum shear.

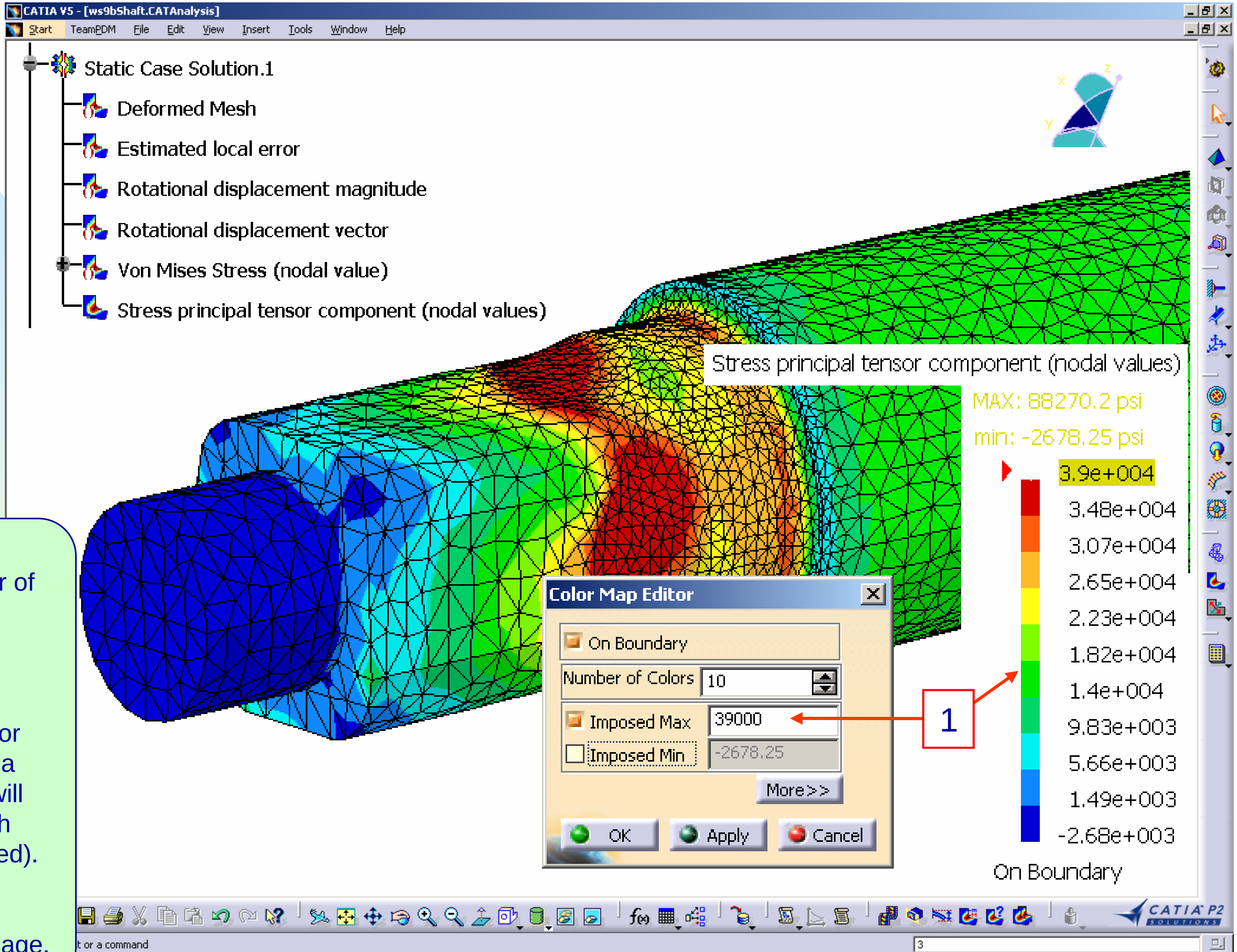
Steps:

1. Select the display principal stress icon.

2. Edit image to show only hoop or shear stress (component C1).

Fine tune your max shear image next page.

Step 9. Visualize final results



Hand calculations indicate max shear of 39,000 psi at the shoulder, so…

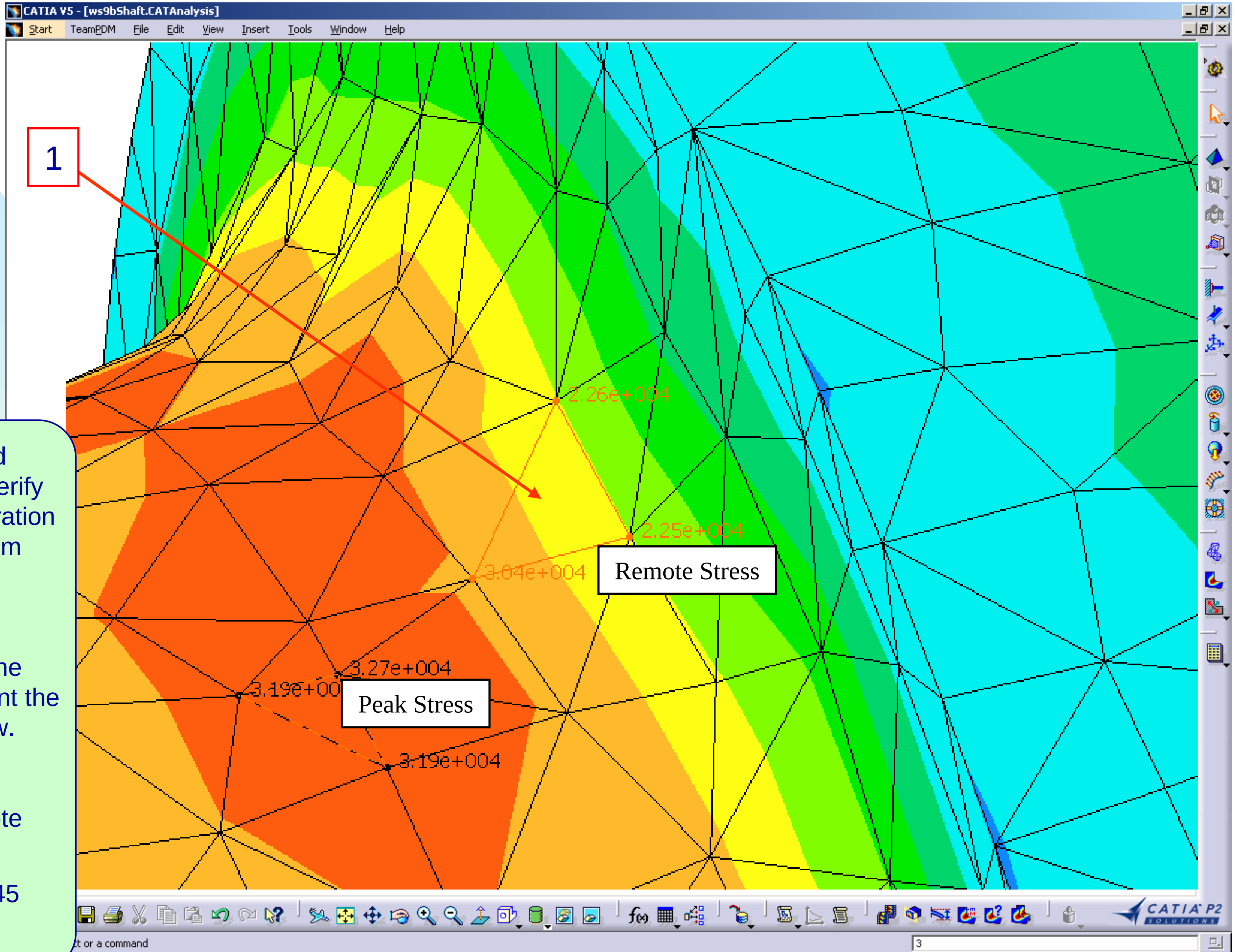
Steps:

1. Double click color pallet and Impose a max 39,000 (this will color elements with 39ksi and higher red).

Dig deeper, next page.

MSC SOFTWARE

Step 9. Visualize final results



Visualize peak and remote stress to verify the stress configuration factor K_t =1.42 (from Peterson).

Steps:

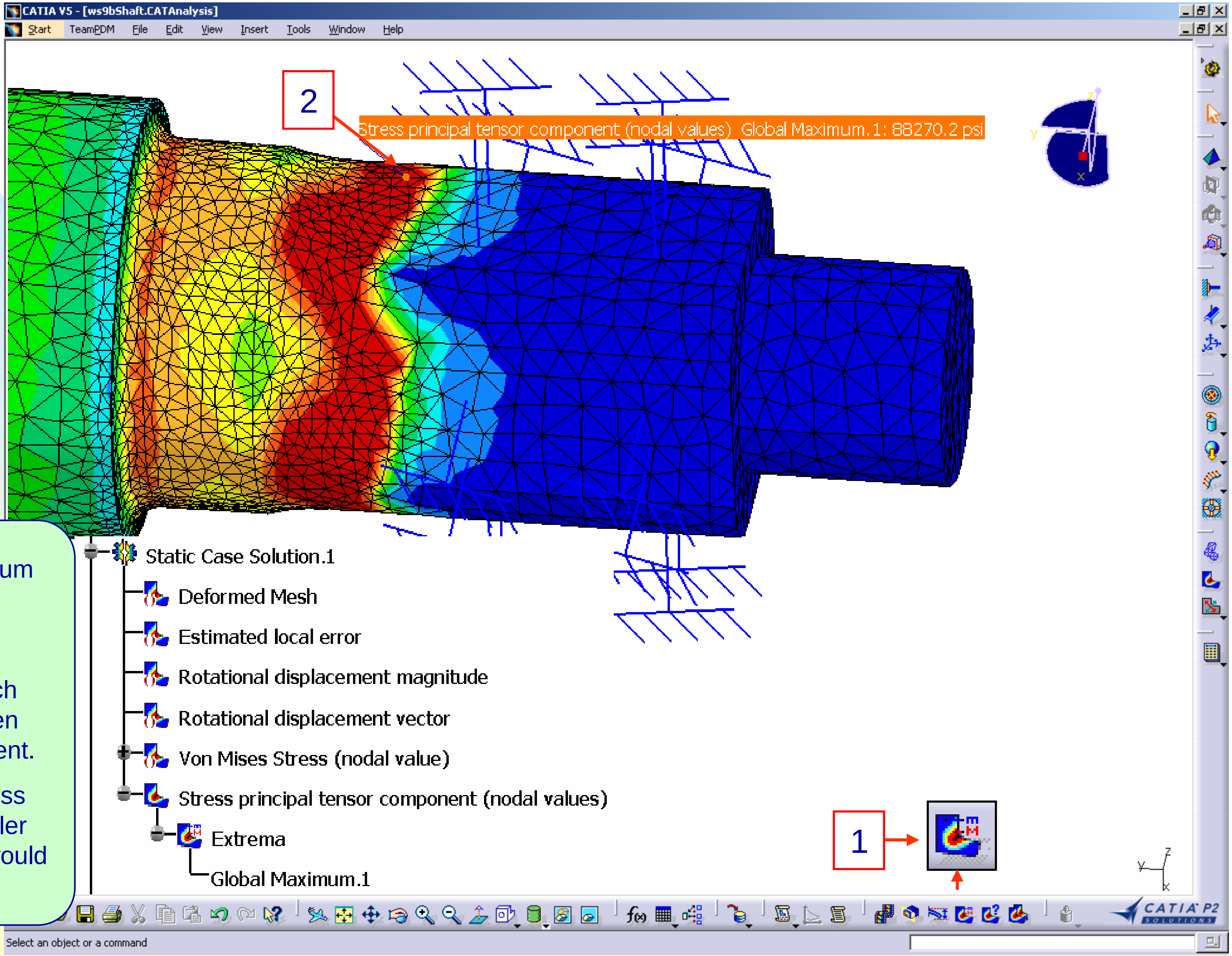
1. By positioning the cursor on a element the stress values show.

Peak stress/Remote stress = K_t.

3.27e4/2.25e4=1.45

We have a match.

Step 9. Visualize final results



Find global maximum shear stress.

Steps:

1. Select the search Image extrema then focus on the element.

2. This design stress occurs at the smaller cross section as would be expected.

Step 9. Visualize final results

Conclusions

- CATIA V5 GSA workbench is validated for a shaft with a shouldered fillet scenario.

	Hand Calculations	.06 inch Linear Global Mesh, .006 inch sag .03 inch Linear Local Mesh, .003 inch sag
Global % Precision error Local % Precision error	NA NA	19 % 24.1 %
Error Estimate	NA	4.93e7 Btu global
Translational Displacement	0.0182 radians	??
Stress Concentration Factor	1.42	1.45
Max Principal Shear Stress	39,053 psi (at shoulder radius)	34,800 – 35,600 psi at shoulder radius (design stress 88,500 psi)

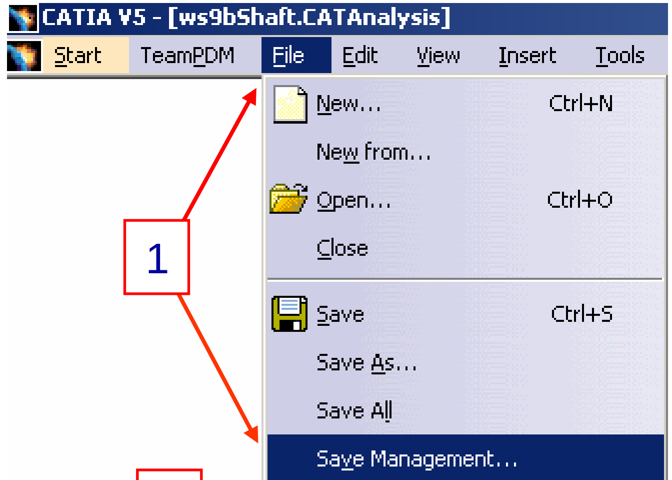
Step 10. Save the analysis document

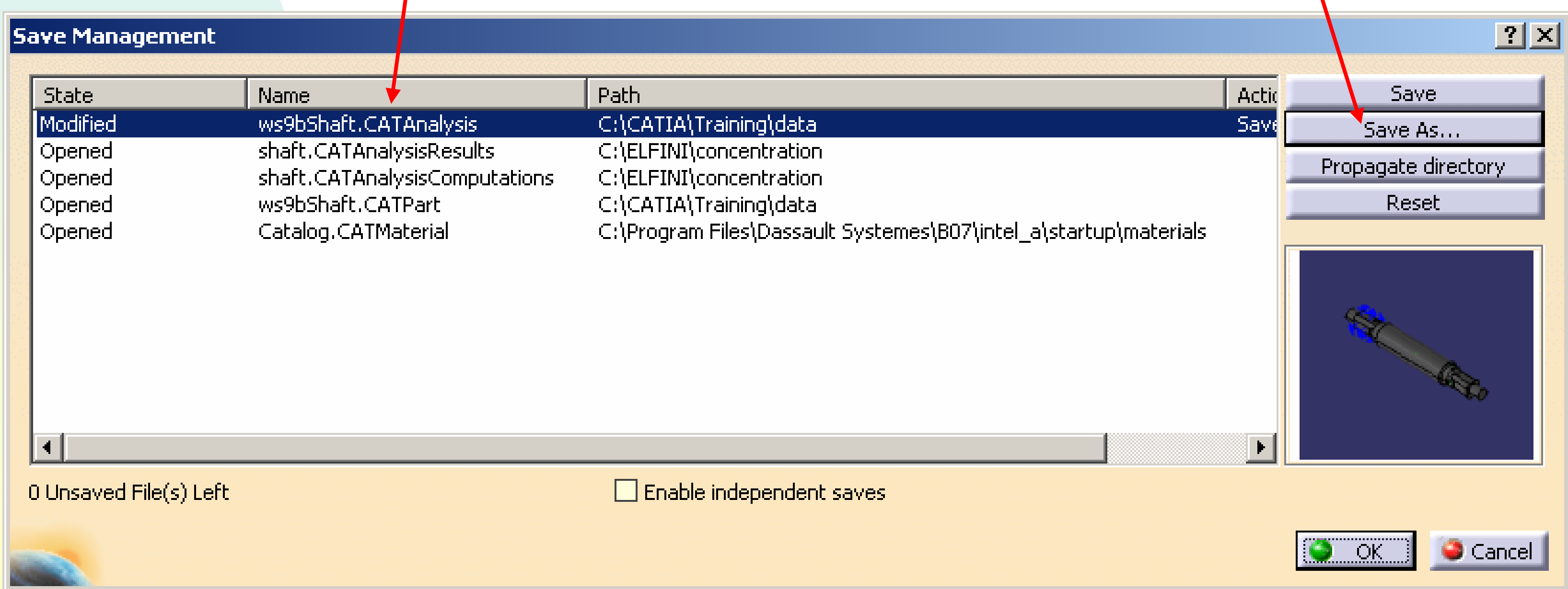
Steps:

1. Select Save Management from the File menu.

2. Highlight document you want to save.

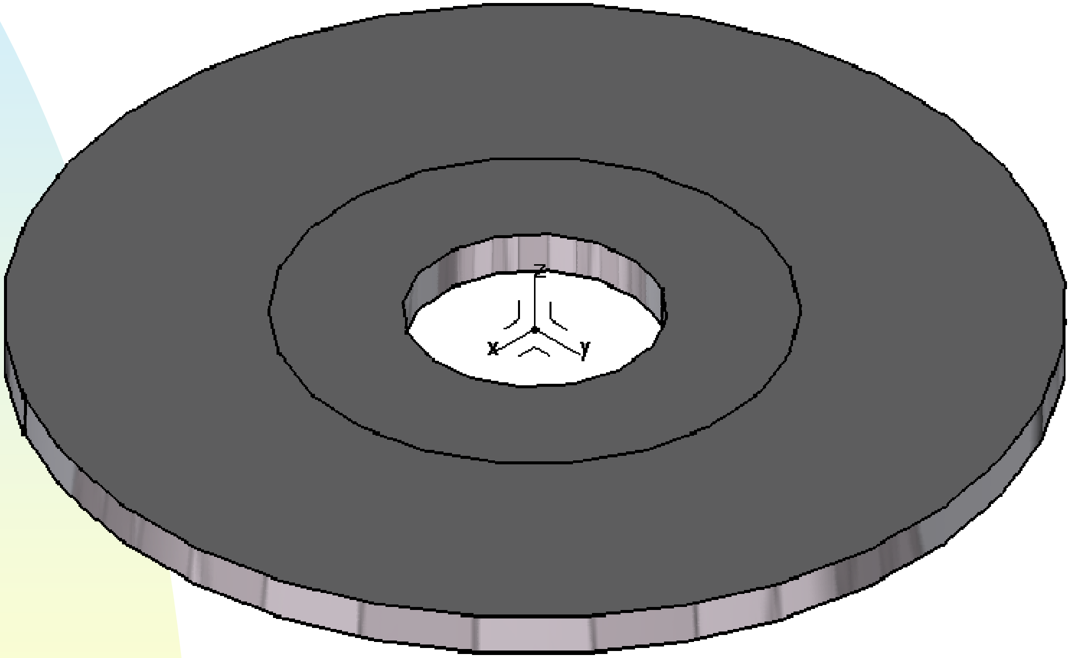
3. Select Save As to specify name and path, select, OK.





WORKSHOP 10

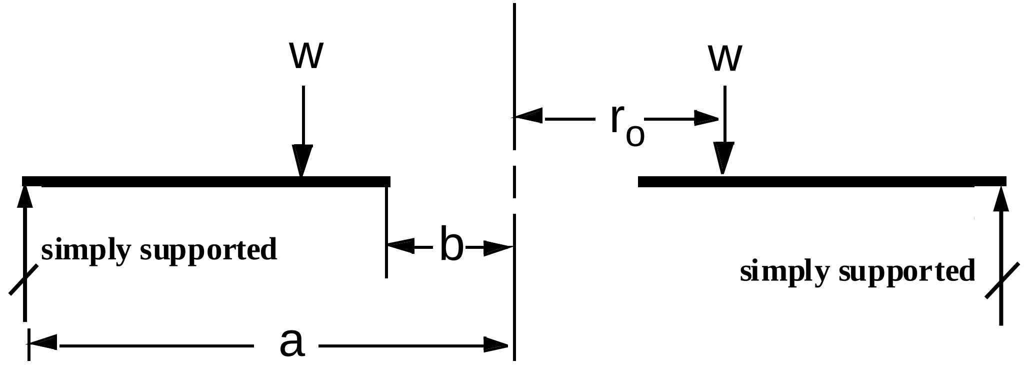
ANNULAR PLATE

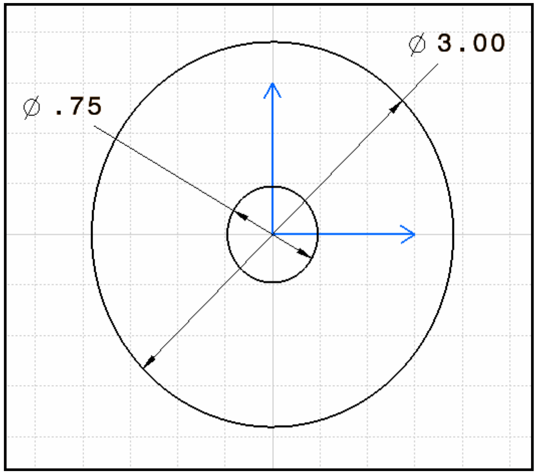


MSC SOFTWARE
SIMULATING REALITY

Problem Description

- Shown below is a 2-D representation of the annular plate shown on the title page. The outer edge of the plate is simply supported and a uniform line load of 85 lb/in is applied a distance r_o from the center of the plate.





Material: Aluminum
Young Modulus = 10e6 psi
Poisson Ratio = .3
Density = .098 lb_in3
Yield Strength = 13778 psi

Design requirements:
Thickness, t = 0.125 inch
Annular Line Load Radius, r_o = 0.75 inch
Line Load, w = 85 lb/inch

MSC SOFTWARE SIMULATING REALITY

Hand calculations

- Displacement:

$$y = \frac{-wa^3}{D}\left(\frac{C_1 L_9}{C_7} - L_3\right)$$

- Plate constant:

$$D = \frac{Et^3}{12\left(1-v^2\right)}$$

- Plate constants dependent on the ratio a/b:

$$C_1 = \frac{1+v}{2}\frac{b}{a}\ln\frac{a}{b} + \frac{1-v}{4}\left(\frac{a}{b} - \frac{b}{a}\right)$$

$$C_7 = \frac{1}{2}\left(1-v^2\right)\left(\frac{a}{b} - \frac{b}{a}\right)$$

- Loading constants dependent upon the ratio a/r_o:

$$L_3 = \frac{r_0}{a}\left\{\left[\left(\frac{r_0}{a}\right)^2 + 1\right]\ln\frac{a}{r_0} + \left(\frac{r_0}{a}\right)^2 - 1\right\}$$

$$L_9 = \frac{r_0}{a}\left\{\frac{1+v}{2}\ln\frac{a}{r_0} + \frac{1-v}{4}\left[1 - \left(\frac{r_0}{a}\right)^2\right]\right\}$$

Hand calculations (cont.)

- Plate constant:

  $D = 1788.576$

- Plate constants dependent on the ratio a/b:

  $C_1 = 0.8815$ $C_7 = 1.7062$

- Loading constants dependent upon the ratio a/r_o:

  $L_3 = 0.0582$ $L_9 = 0.3346$

- Maximum vertical displacement:

  $y = -0.018$

- Maximum bending stress:

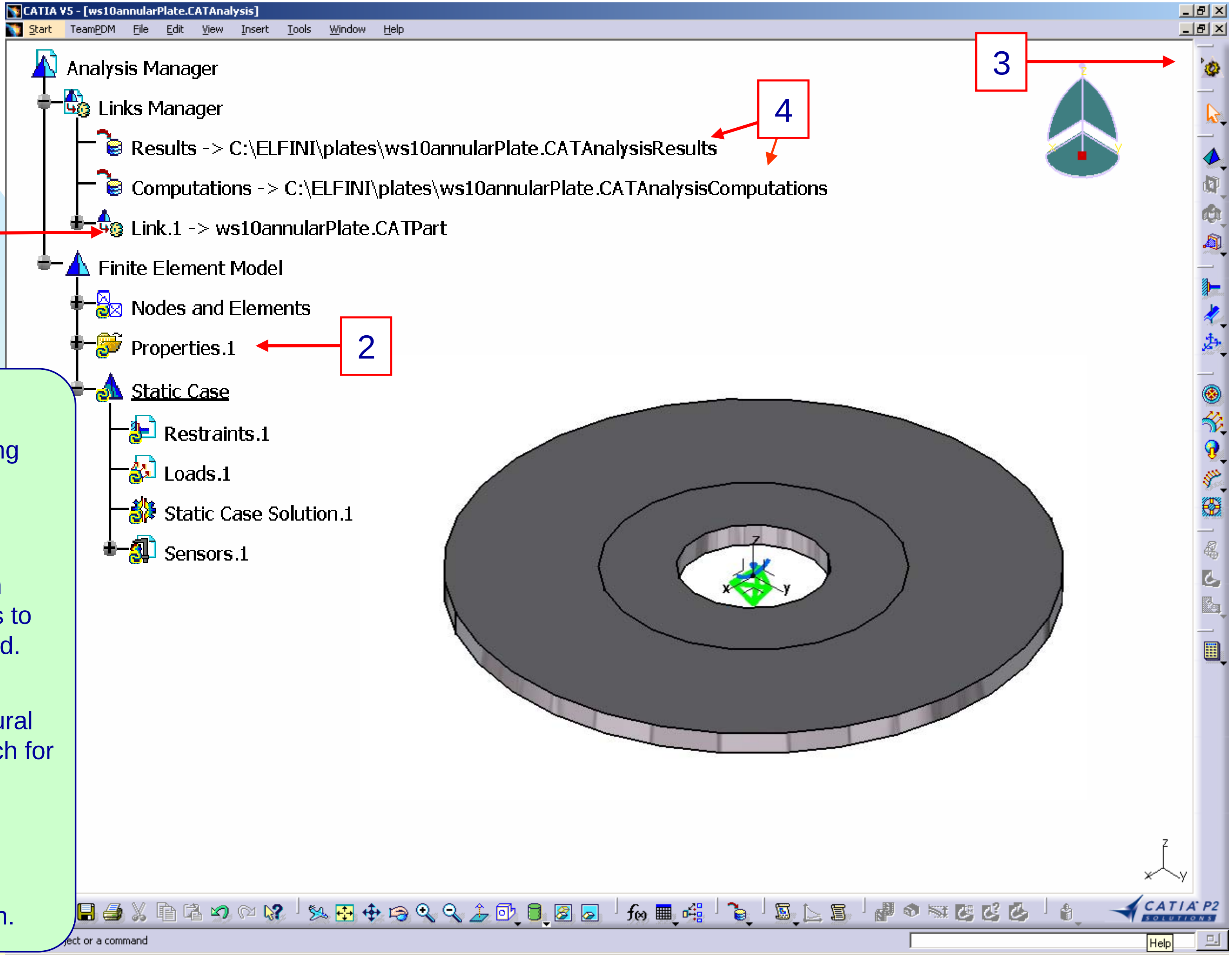
$$\text{Maximum bending stress} = \sigma = \frac{6 \cdot M}{t^2} \qquad M = K_m \cdot w \cdot a = 0.5501 \cdot 85\ \text{lbs/in} \cdot 1.5\ \text{inch} = 70.14\ \text{lbs}$$

$$\sigma = \frac{6 \cdot 70.14\ \text{lbs}}{0.125^2\ \text{inch}} = 26934\ \text{psi}$$

Suggested Exercise Steps

1. Create a new CATIA analysis document (.CATAnalysis).
2. Mesh globally with parabolic elements.
3. Apply an advanced and isostatic restraint (simply supported).
4. Apply a line force density load.
5. Compute the initial analysis.
6. Check global and local precision (animate deformation, adaptive boxes and extremas).
7. Refine the mesh locally.
8. Compute the precise analysis.
9. Visualize final results.
10. Save the analysis document.

Step 1. Create a new CATIA analysis document



Steps:

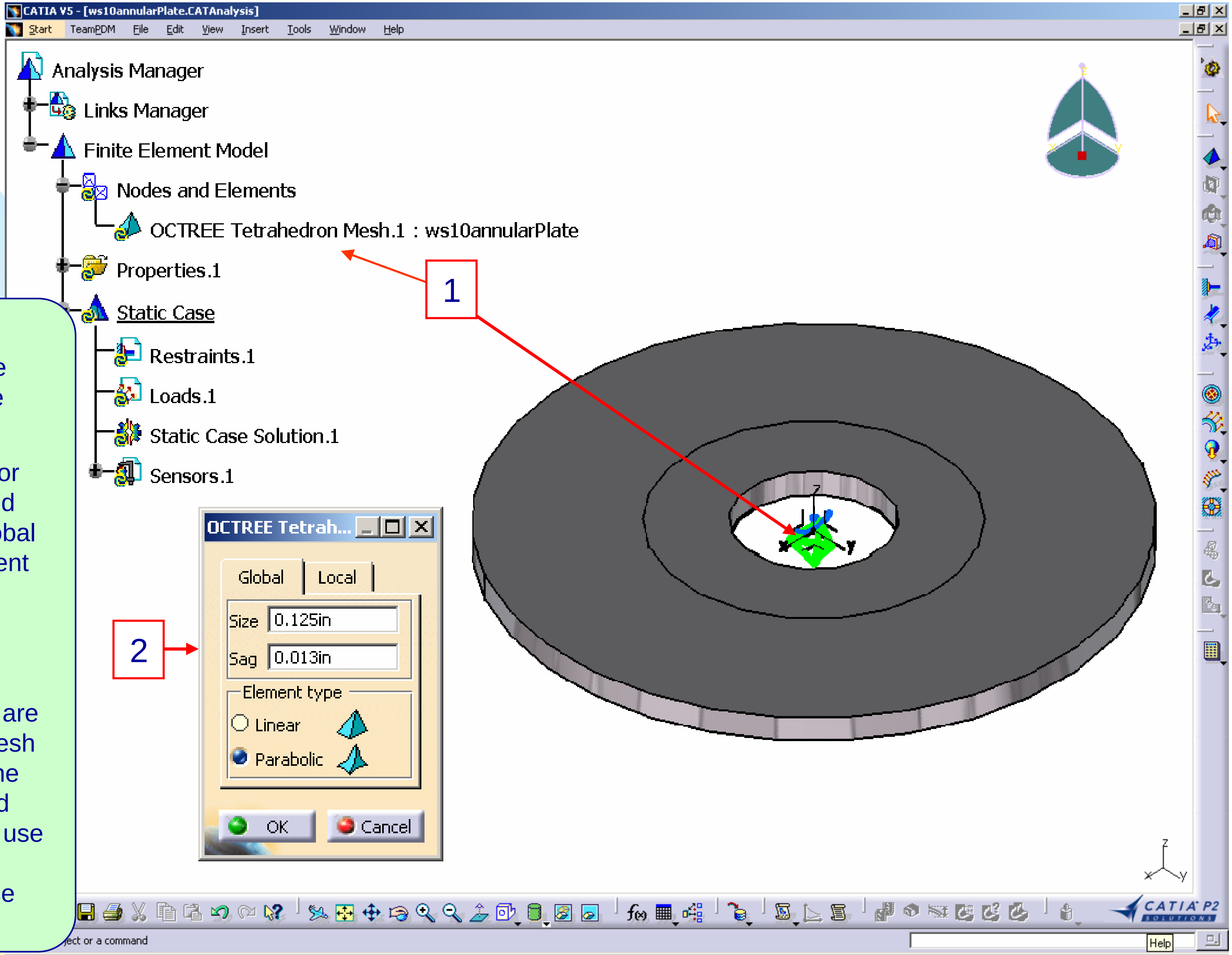
1. Open the existing ws10annularPlate .CATPart from the training directory.

2. Apply aluminum material properties to the part as required.

3. Launch the Generative Structural Analysis workbench for a Static Analysis.

4. Specify the Computations and Results storage locations as shown.

Step 2. Mesh globally with parabolic elements



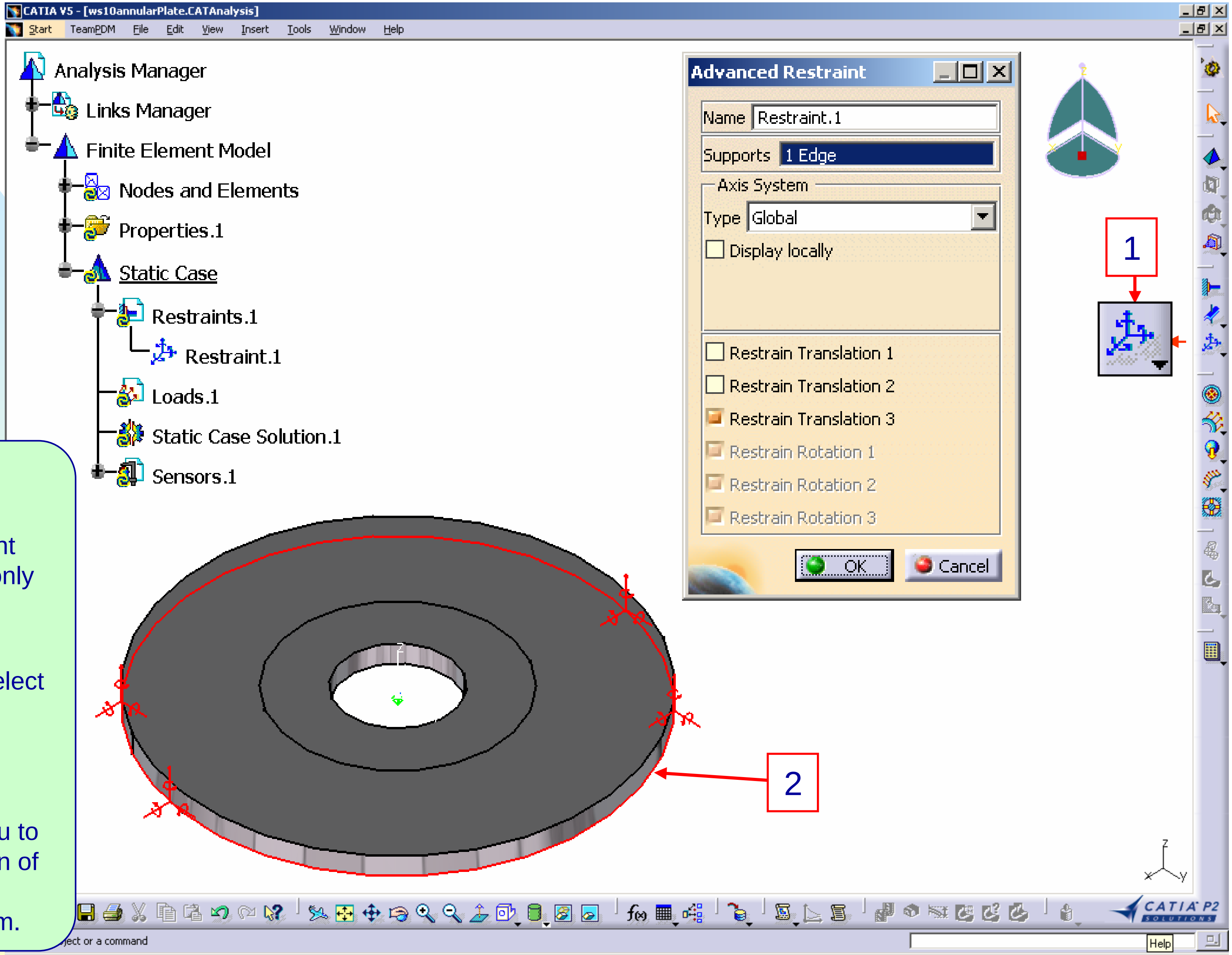
Steps:

1. Double Click the "Mesh" icon on the part.

2. Key in 0.125in for the Global Size and 0.013in for the Global sag, change element type to Parabolic, select OK.

As plates typically are large using one mesh element through the thickness is a good way to start. Then use localized adaptive meshing for precise results.

Step 3. Apply an advanced restraint



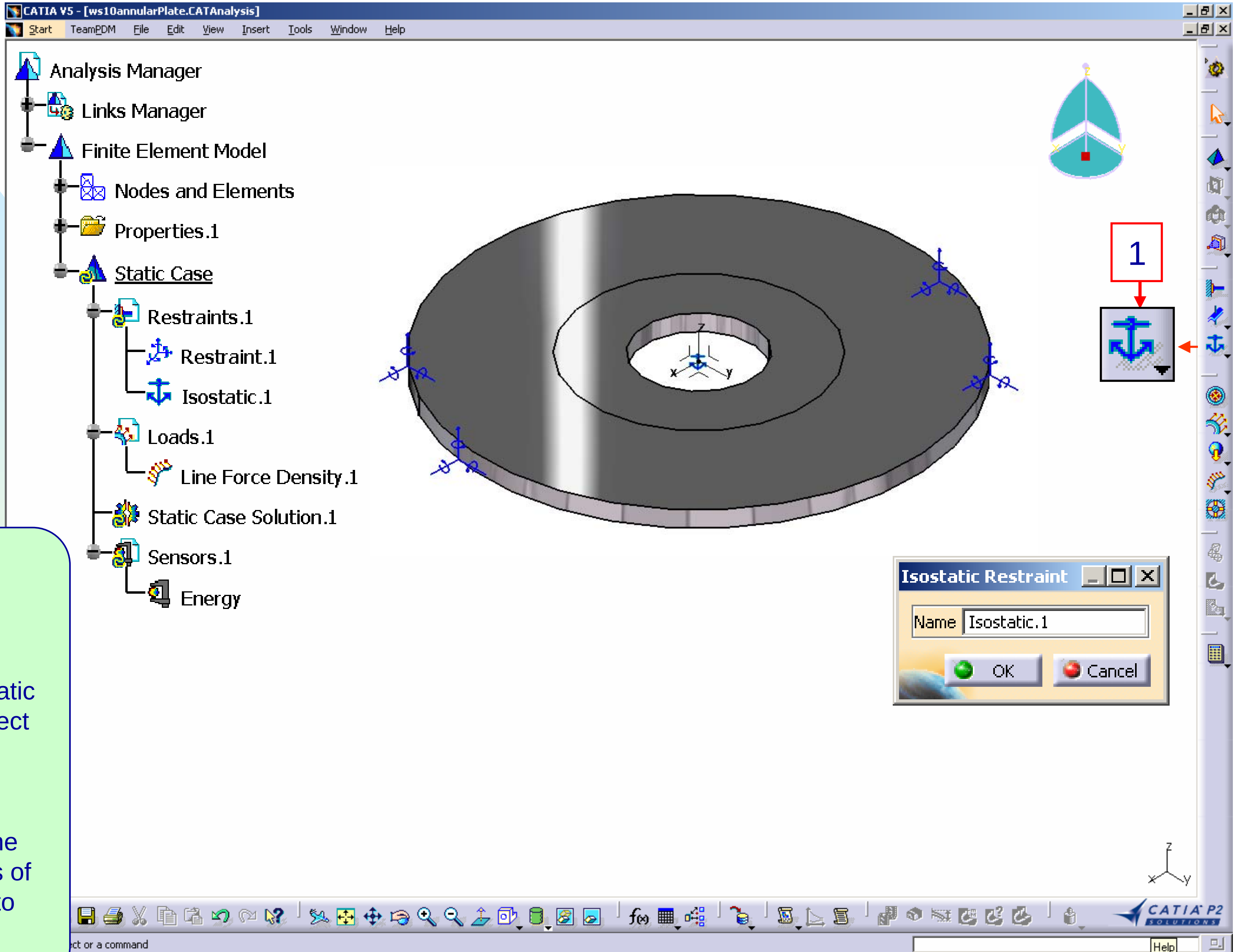
Steps:

1. Select the Advanced Restraint icon and restrain only translation 3.

2. Select the outer edge as shown, select OK.

The advanced restraint allows you to fix any combination of available nodal degrees of freedom.

Step 3. Apply an isostatic restraint



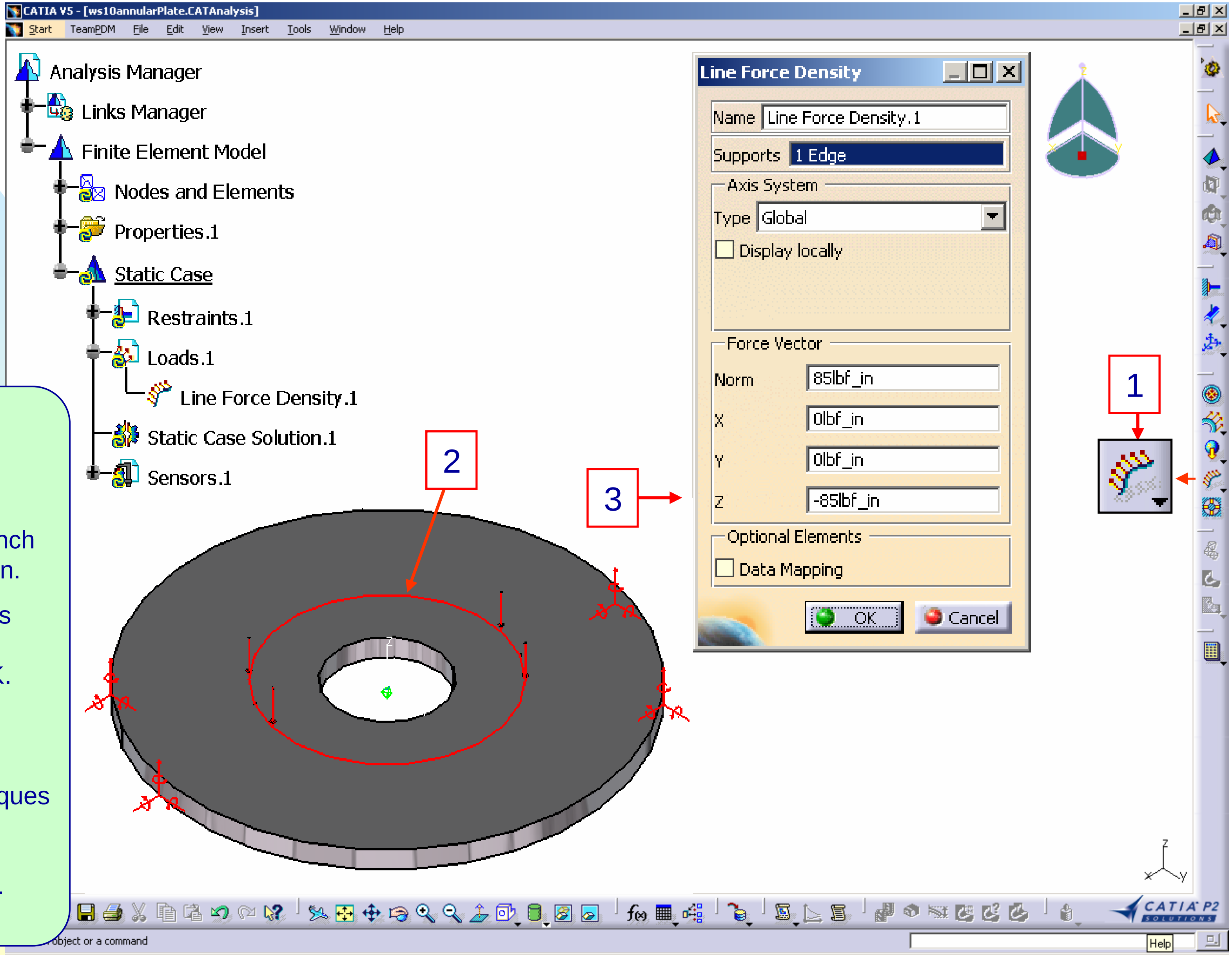
Apply an Isostatic restraint.

Steps:

1. Select the Isostatic Restraint icon, select OK.

This will restrain the remaining degrees of freedom required to make our part statically determinate.

Step 4. Apply a line force density load



Steps:

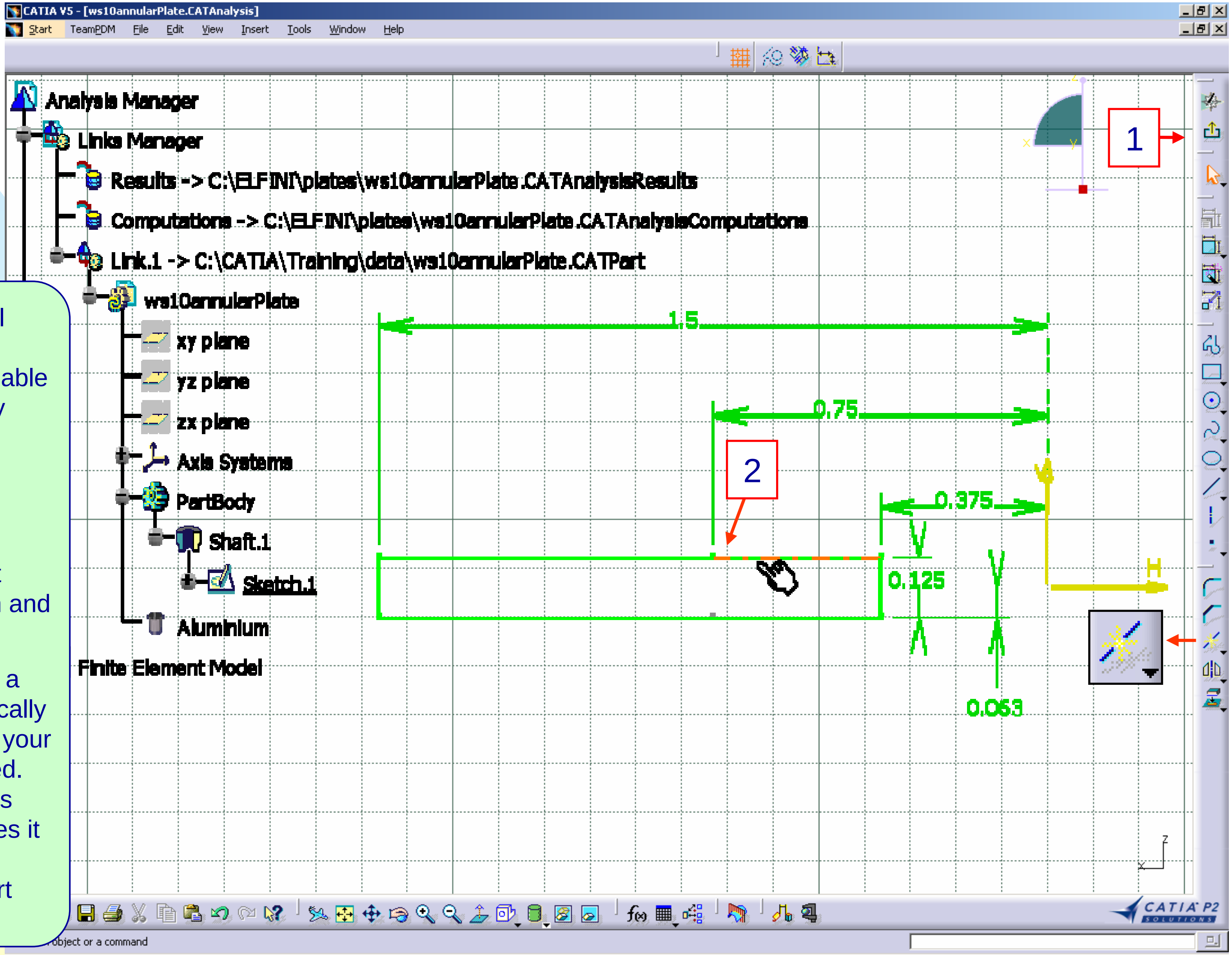
1. Select the Line Force Density icon.

2. Select the 0.75 inch radius line as shown.

3. Enter -85lbf_in as shown in the z-direction, select OK.

Special part construction techniques are necessary to enable Line Force Density application. See next page.

Step 4. Apply a line force density load



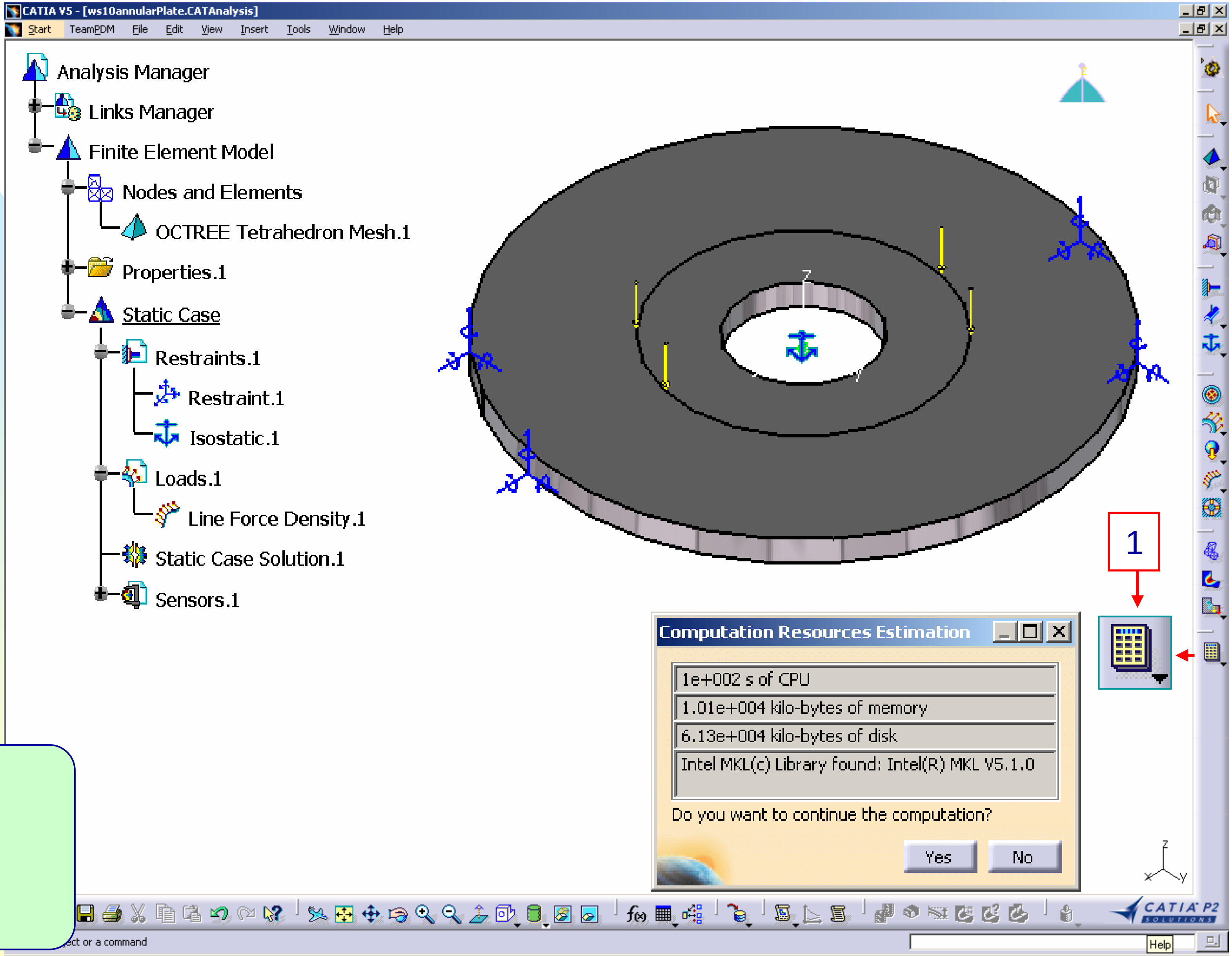
Review the special part construction techniques that enable Line Force Density application.

Steps:

1. Launch the Part Design workbench and enter Sketch.1.

2. This sketch has a line broken specifically at the point where your force will be applied. When this sketch is rotated 360 degrees it gives us the edge element on the part that we select.

Step 5. Compute the initial analysis

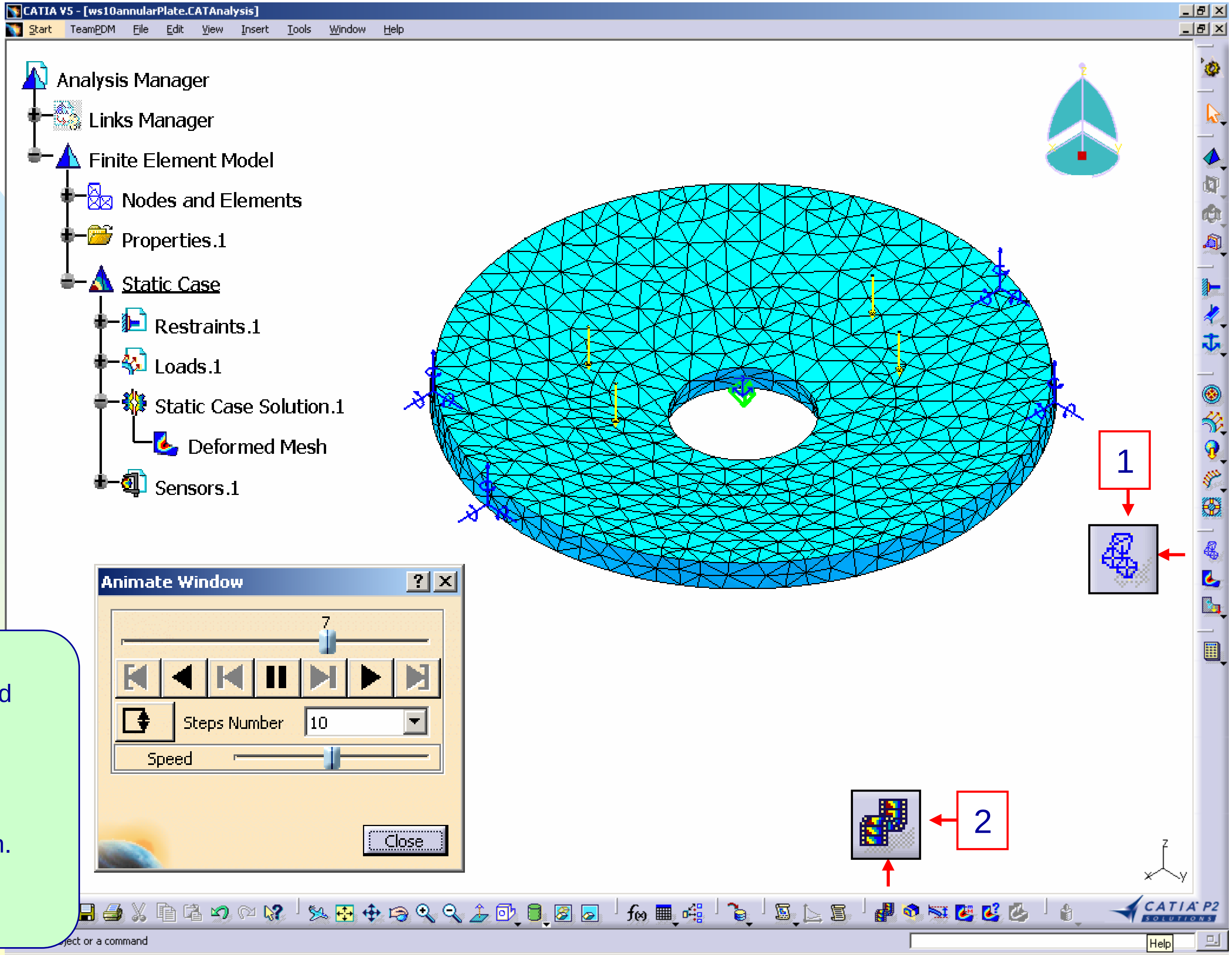


Steps:

1. Compute All

Save often.

Step 6. Check global and local precision



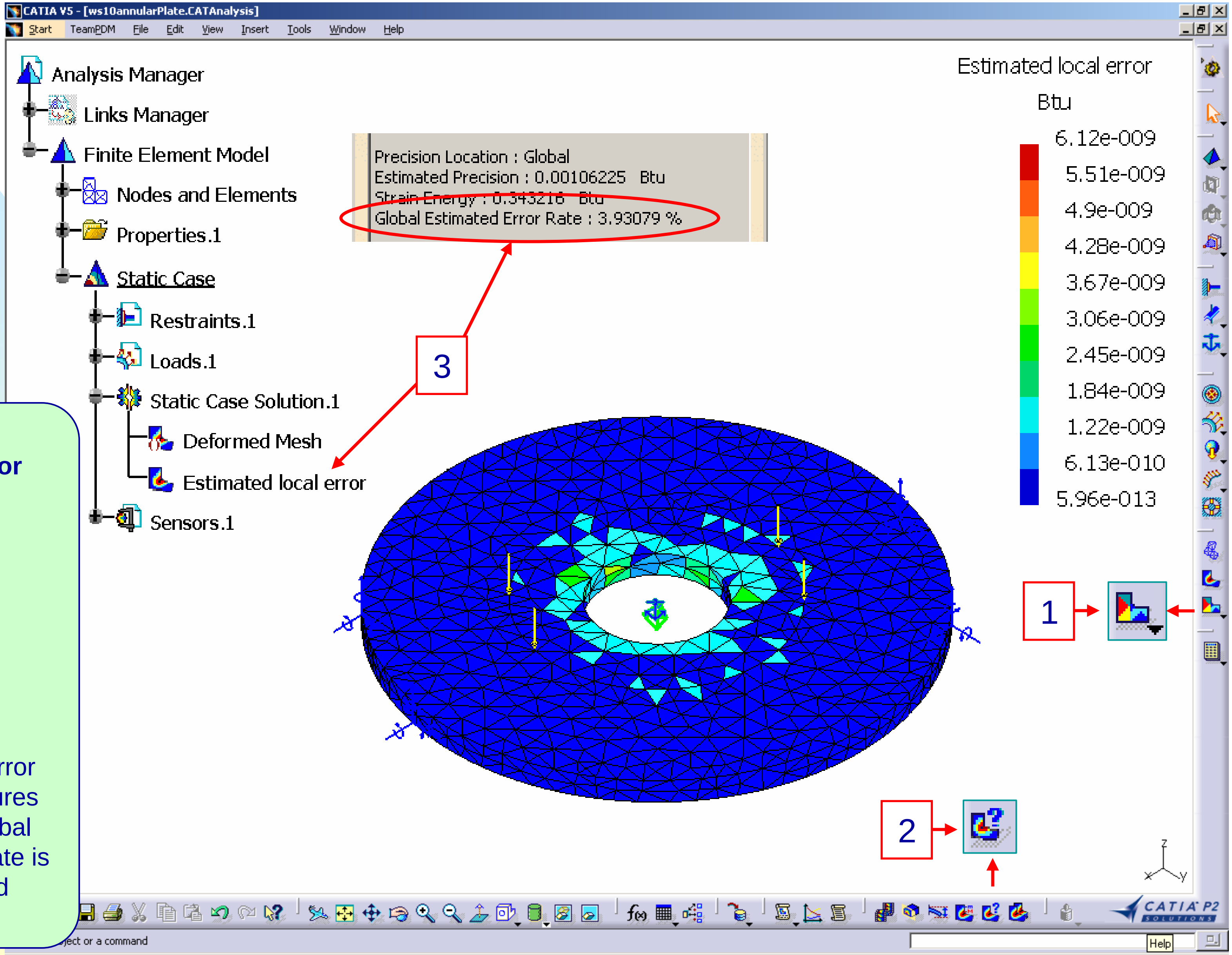
Visualize the **Deformation** and animate.

Steps:

1. Select the Deformation icon.

2. Select on the Animate icon.

Step 6. Check global and local precision



Visualize the **computation error map.**

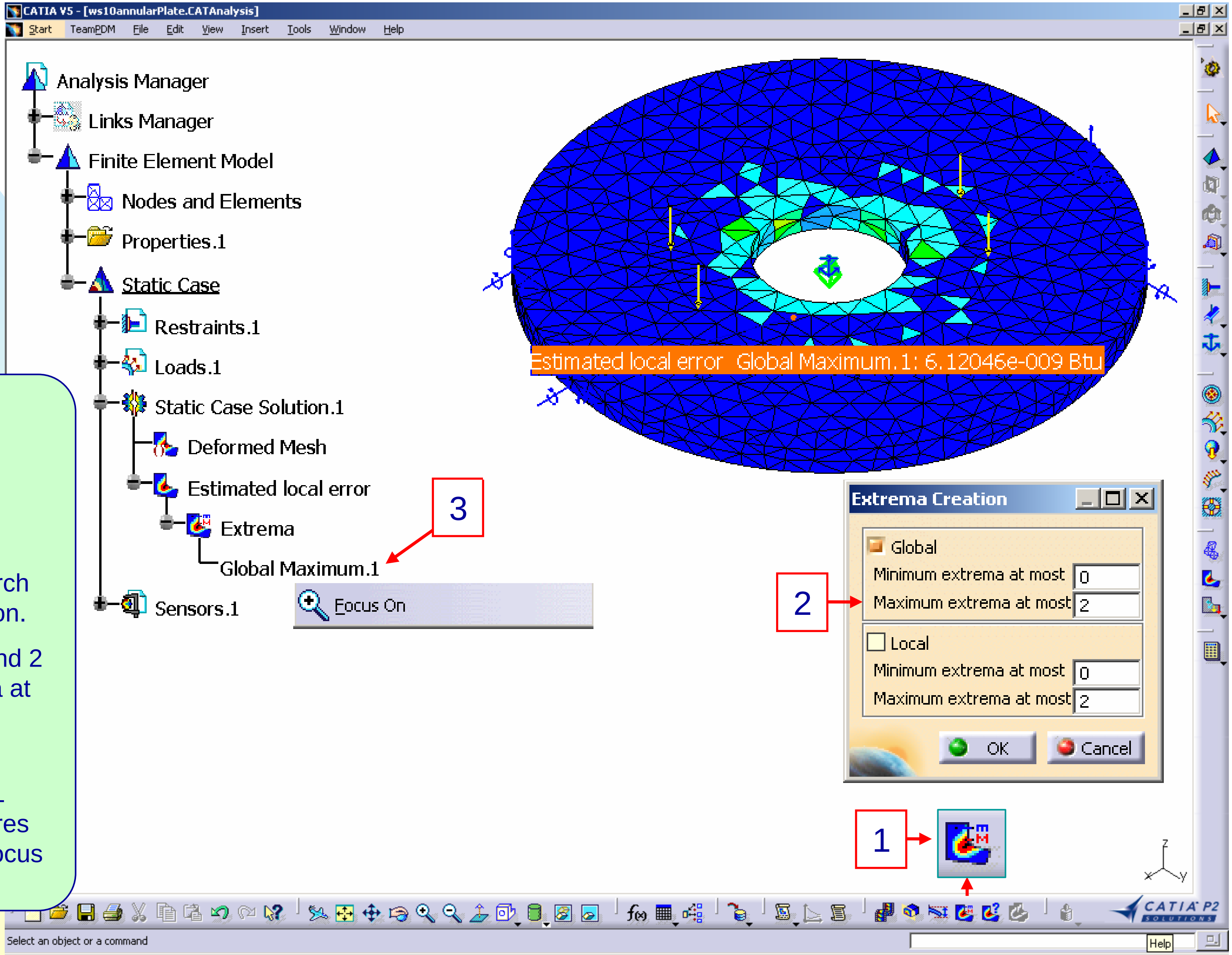
Steps:

1. Select the Precision icon.

2. Select on the information icon.

3. Select the Estimated local error object in the features tree. Note the global estimated error rate is good (recommend max 20%).

Step 6. Check global and local precision



Find the global element with the highest estimated error.

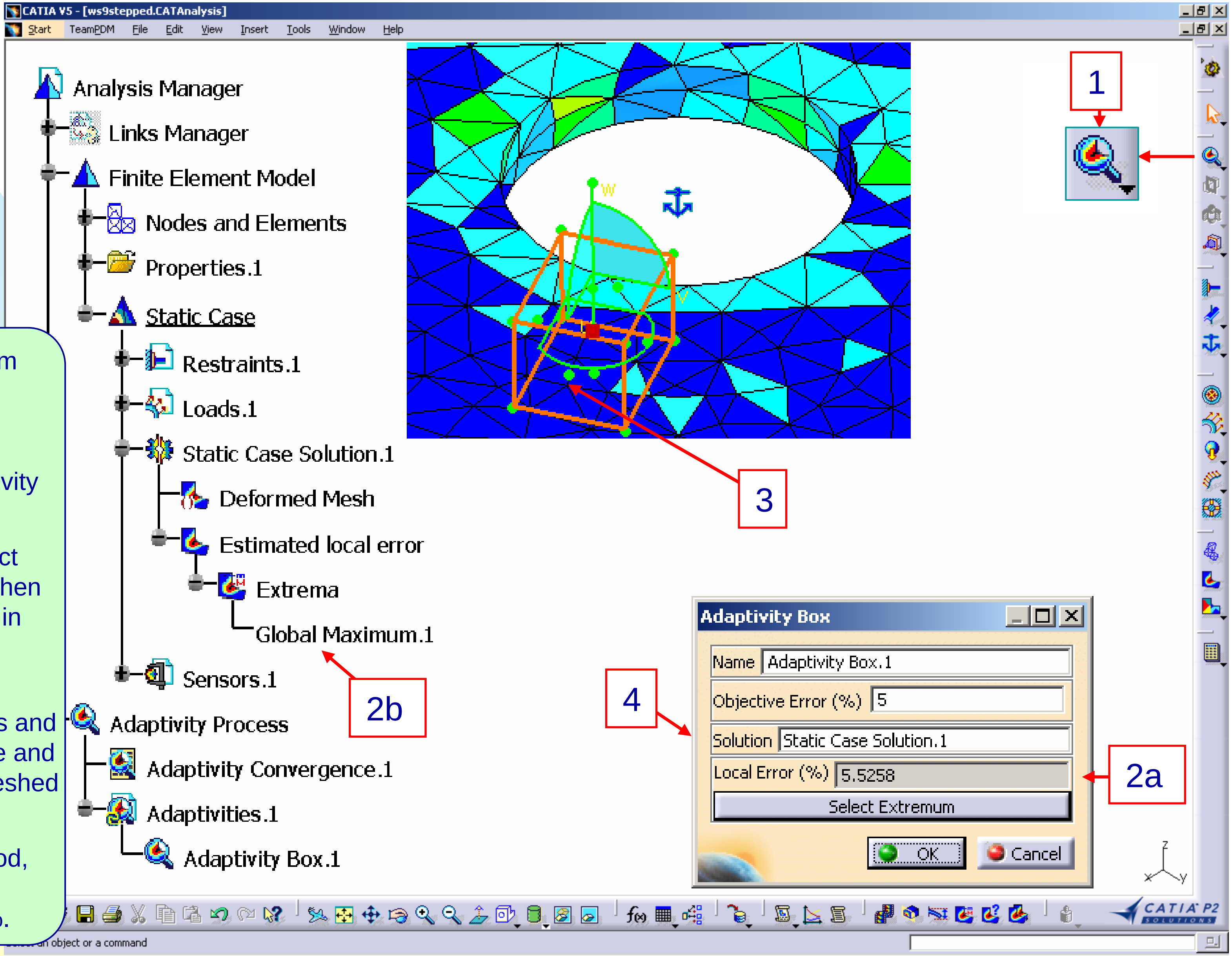
Steps:

1. Select the Search Image Extrema icon.

2. Select Global and 2 maximum extrema at most, select OK.

3. Right click the Global Maximum.1 object in the features tree then select Focus On.

Step 6. Check global and local precision



Determine maximum local error %.

Steps:

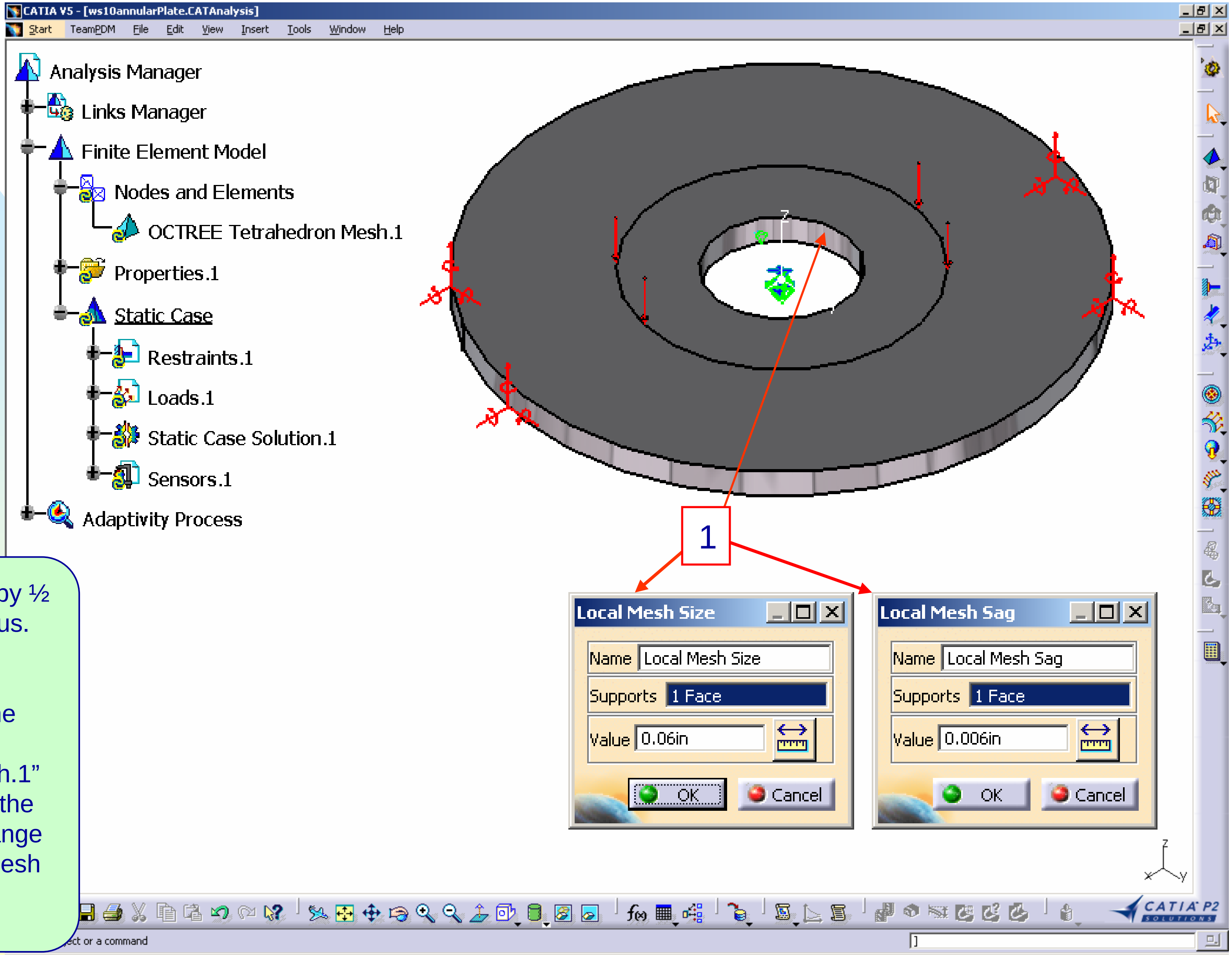
1. Select the adaptivity box icon.

2. Select the "Select Extremum" button then Global Maximum.1 in the features tree to locate box.

3. Use the compass and green dots to locate and size box around meshed areas.

4. Local error is good, well below the recommended 10%.

Step 7. Refine the mesh locally

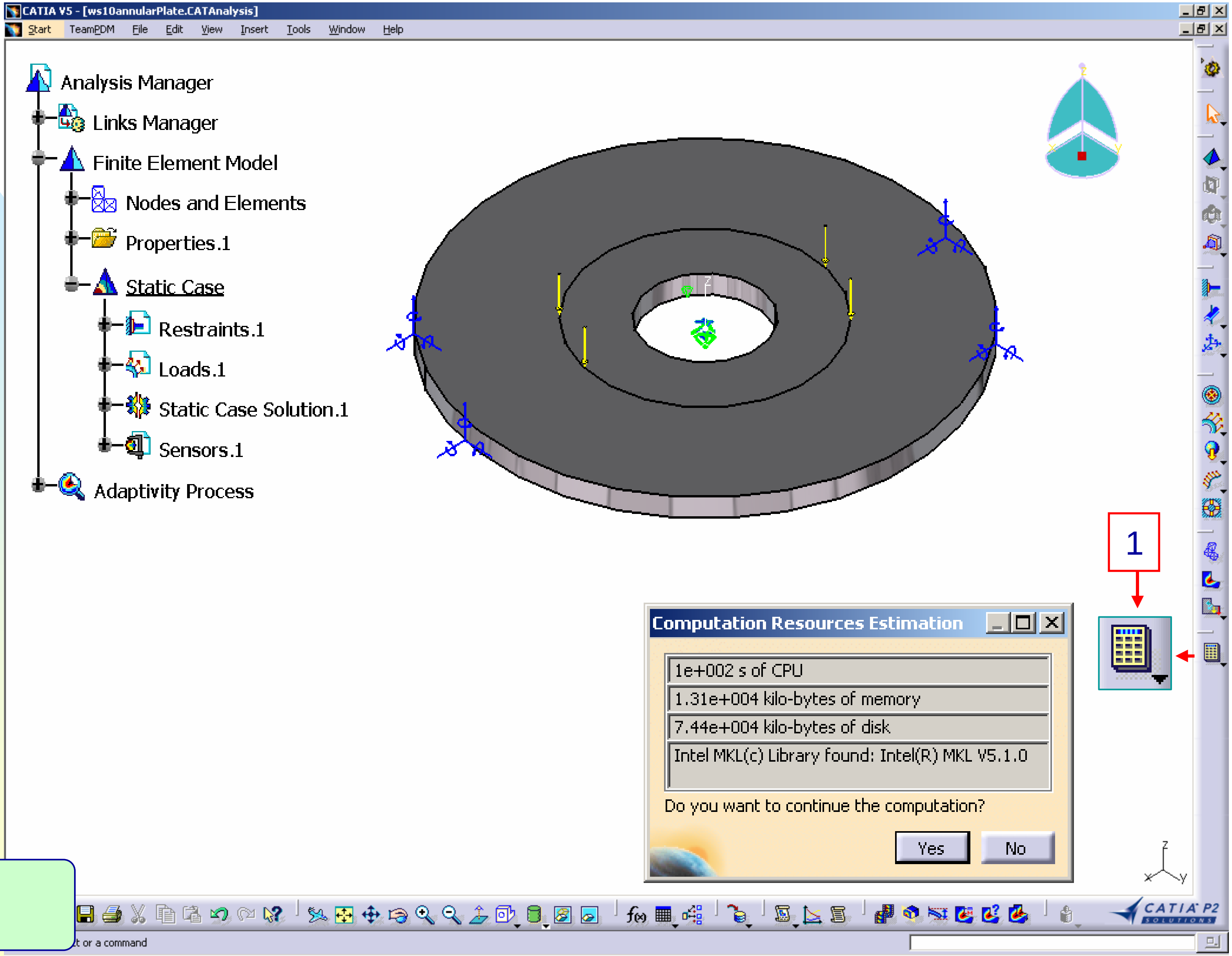


Refine the mesh by ½ on the inside radius.

Steps:

1. Double Click the “OCTREE Tetrahedron Mesh.1” representation in the features tree, change and apply local mesh size and sag as shown.

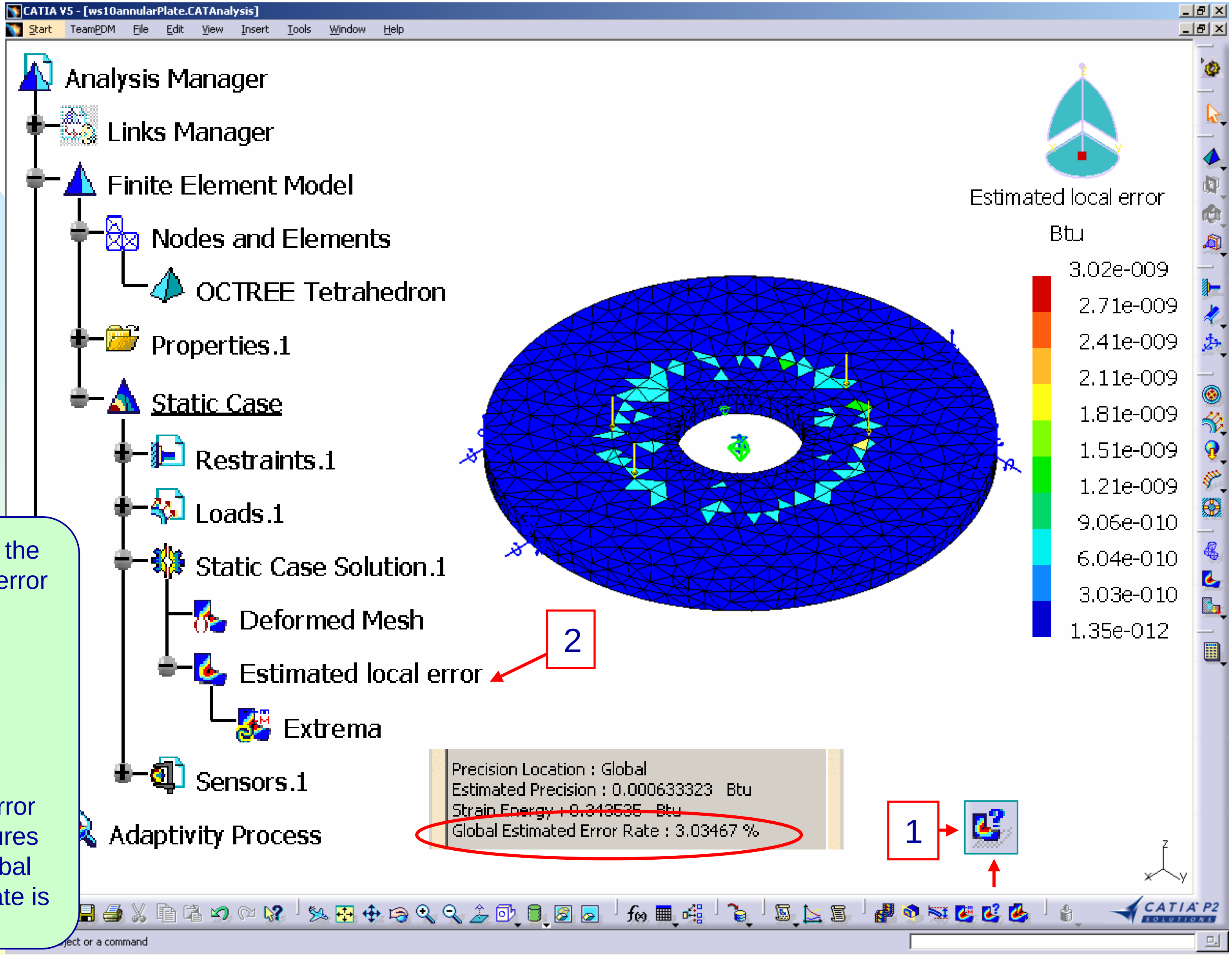
Step 8. Compute the precise analysis



Steps:

1. Compute All.

Step 9. Visualize final results



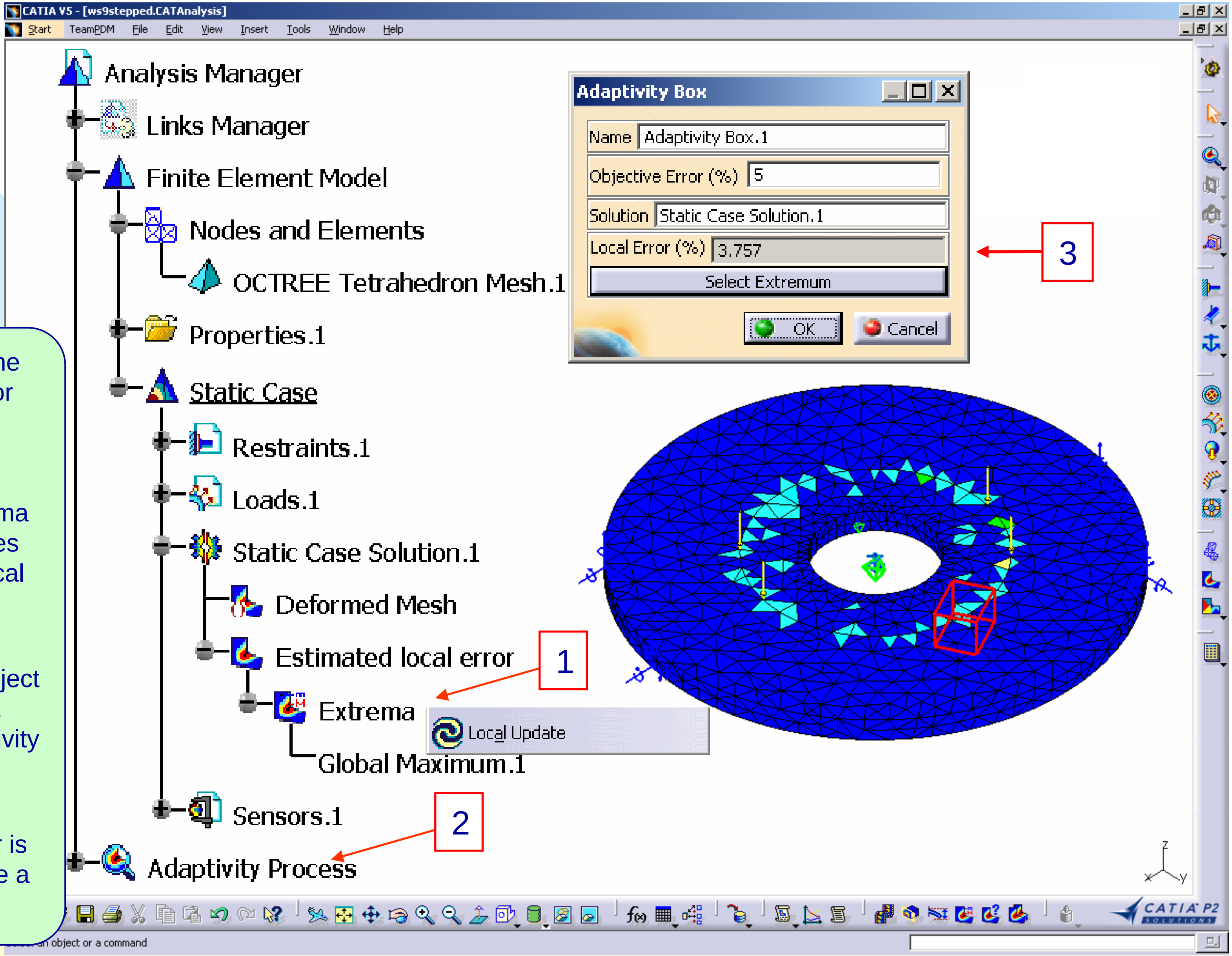
Check how much the global estimated error has improved.

Steps:

1. Select on the information icon.

2. Select the Estimated local error object in the features tree. Note the global estimated error rate is very good.

Step 9. Visualize final results



Check how much the local estimated error has improved.

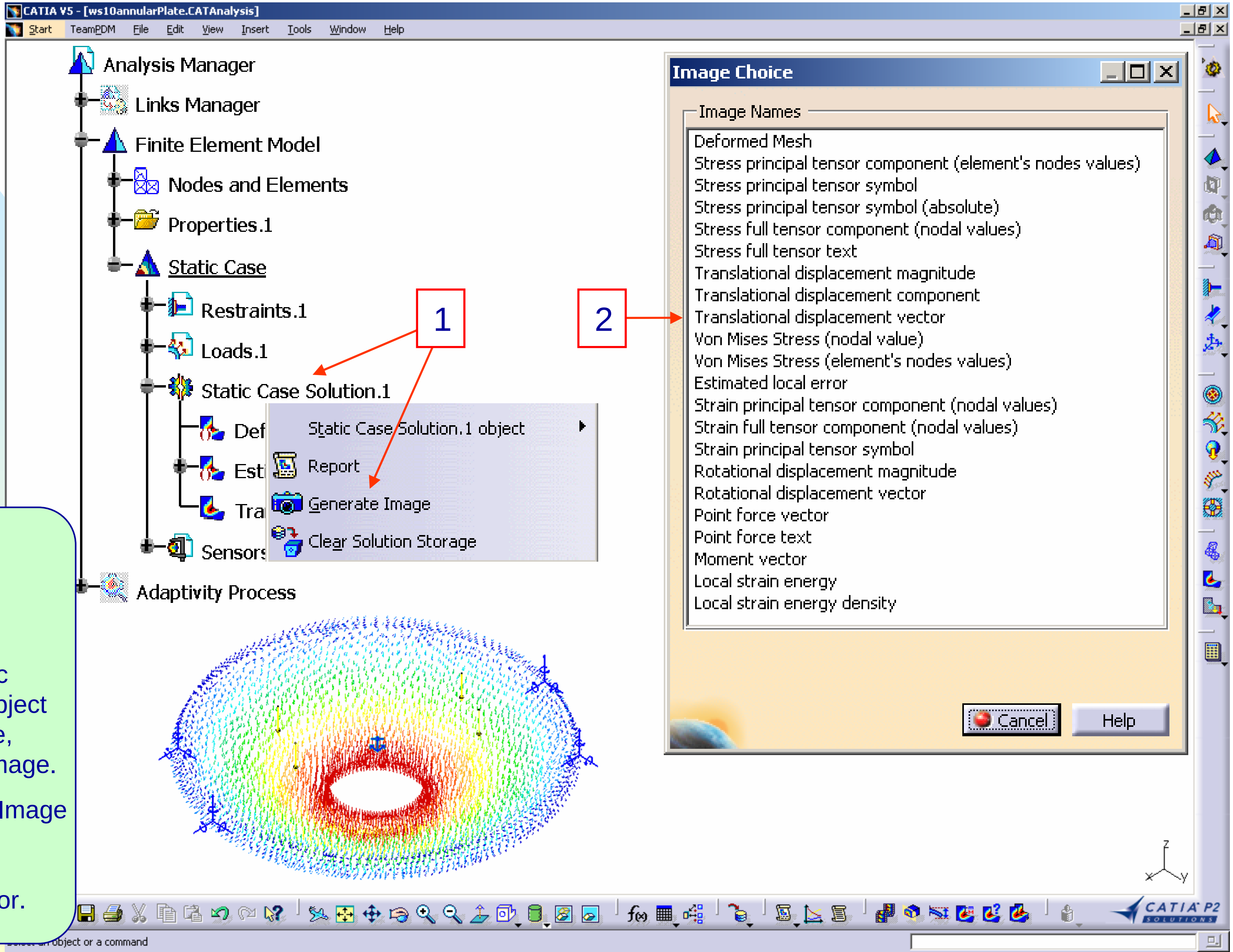
Steps:

1. Right click Extrema object in the features tree then select Local Update.

2. Double click the Adaptivity Box.1 object in the features tree. Relocate the adaptivity box to the new Extrema.

3. Since local error is below 10% we have a precise model.

Step 9. Visualize final results



A new way to add images.

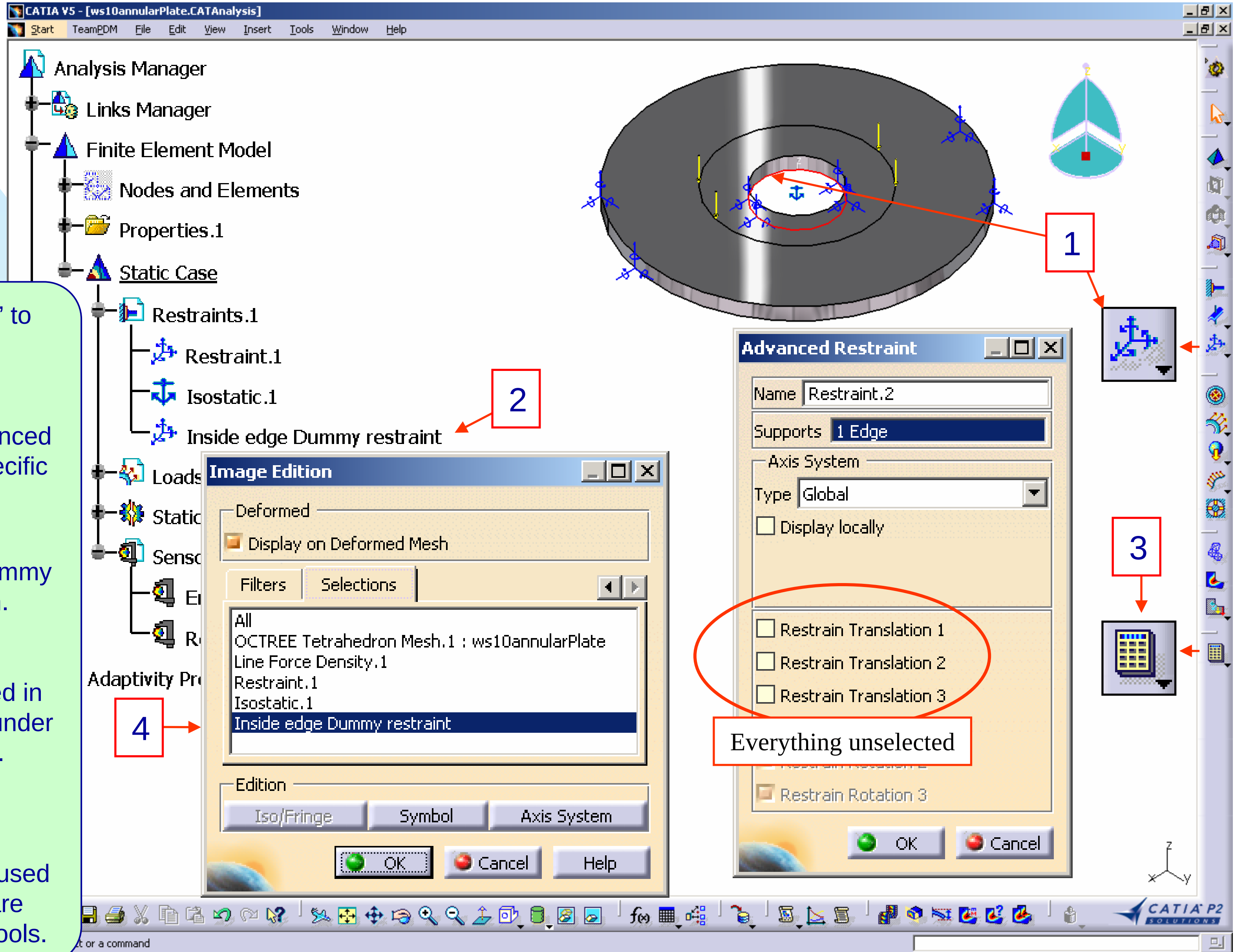
Steps:

1. Right click Static Case Solution.1 object in the features tree, select Generate Image.

2. From the list of Image Choices select the Translational displacement vector.

MSC SOFTWARE

Step 9. Visualize final results



"Trick of the trade" to filter image data.

Steps:

1. Create an advanced restraint in the specific area with nothing restrained.

2. Rename the dummy restraint as shown.

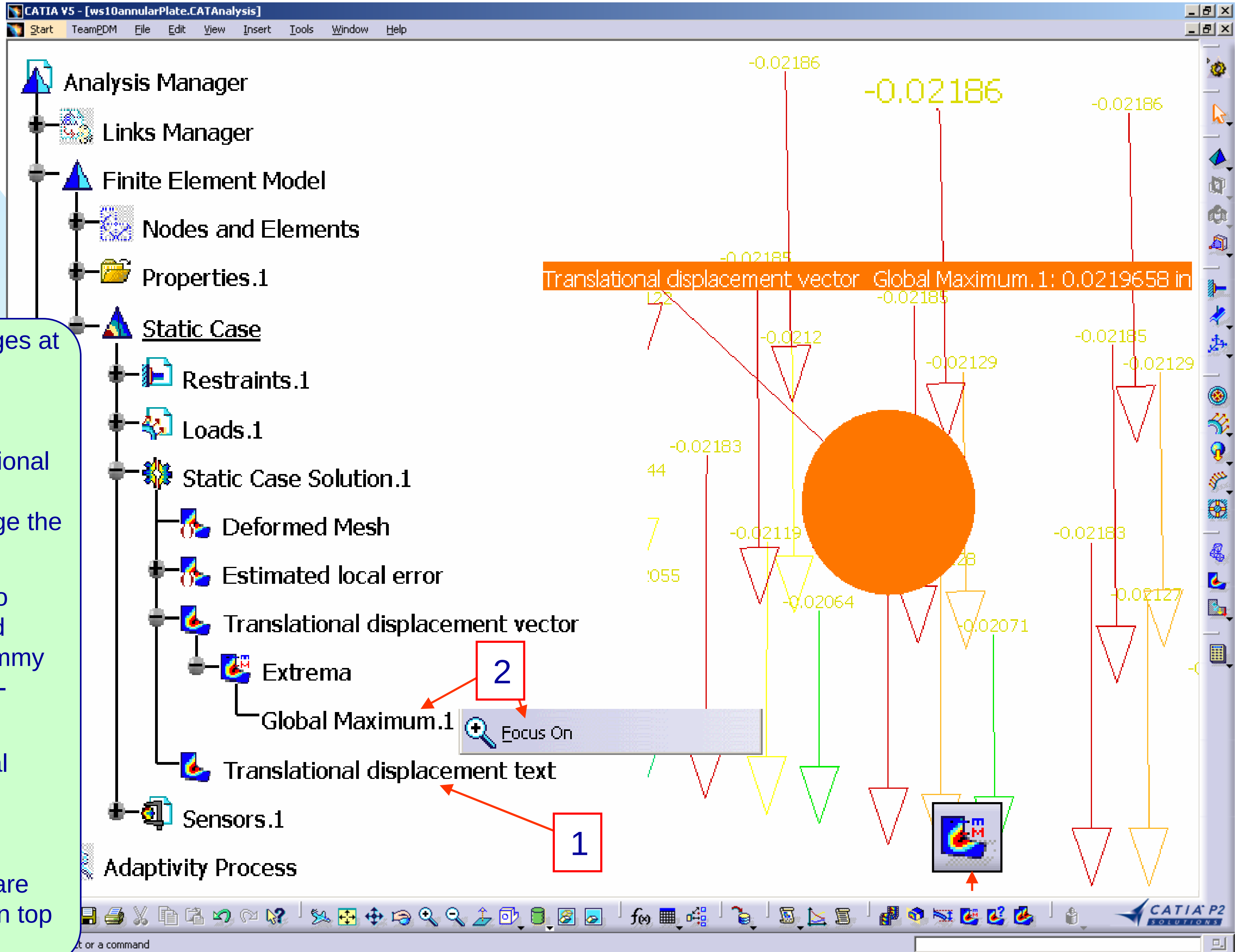
3. Compute All

4. This is then used in the Image Editor under the Selections tab.

Sensors and this technique can be used with knowledgeware and optimization tools.

MSC SOFTWARE SIMULATING REALITY

Step 9. Visualize final results



Visualize two images at once.

Steps:

1. Create an additional Translational displacement image the old way.

Limit the images to show only text and vectors on the dummy restraint and the z-direction.

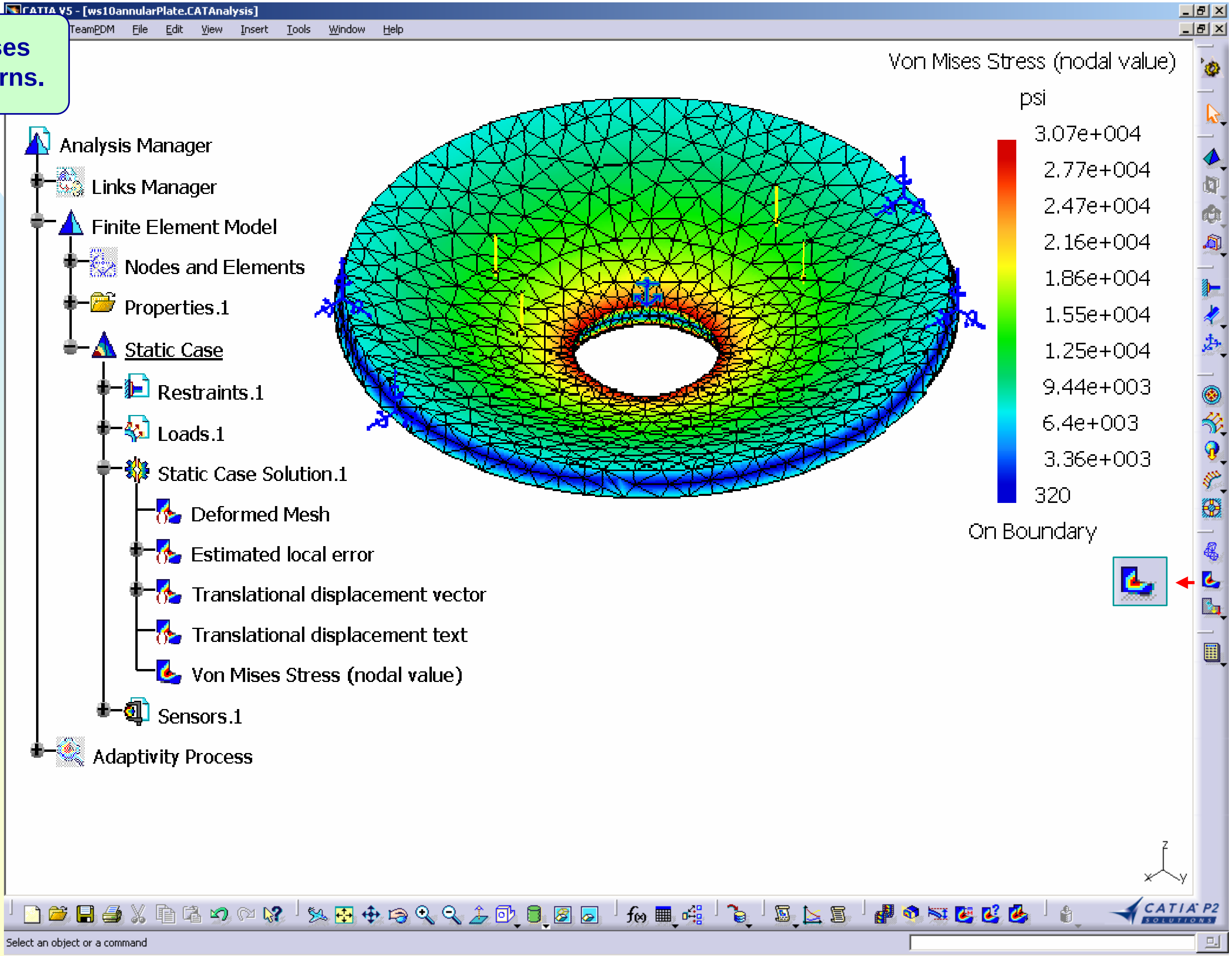
2. Focus on Global Maximum.1.

All active images are seen over-layed on top of each other.

MSC SOFTWARE

Step 9. Visualize final results

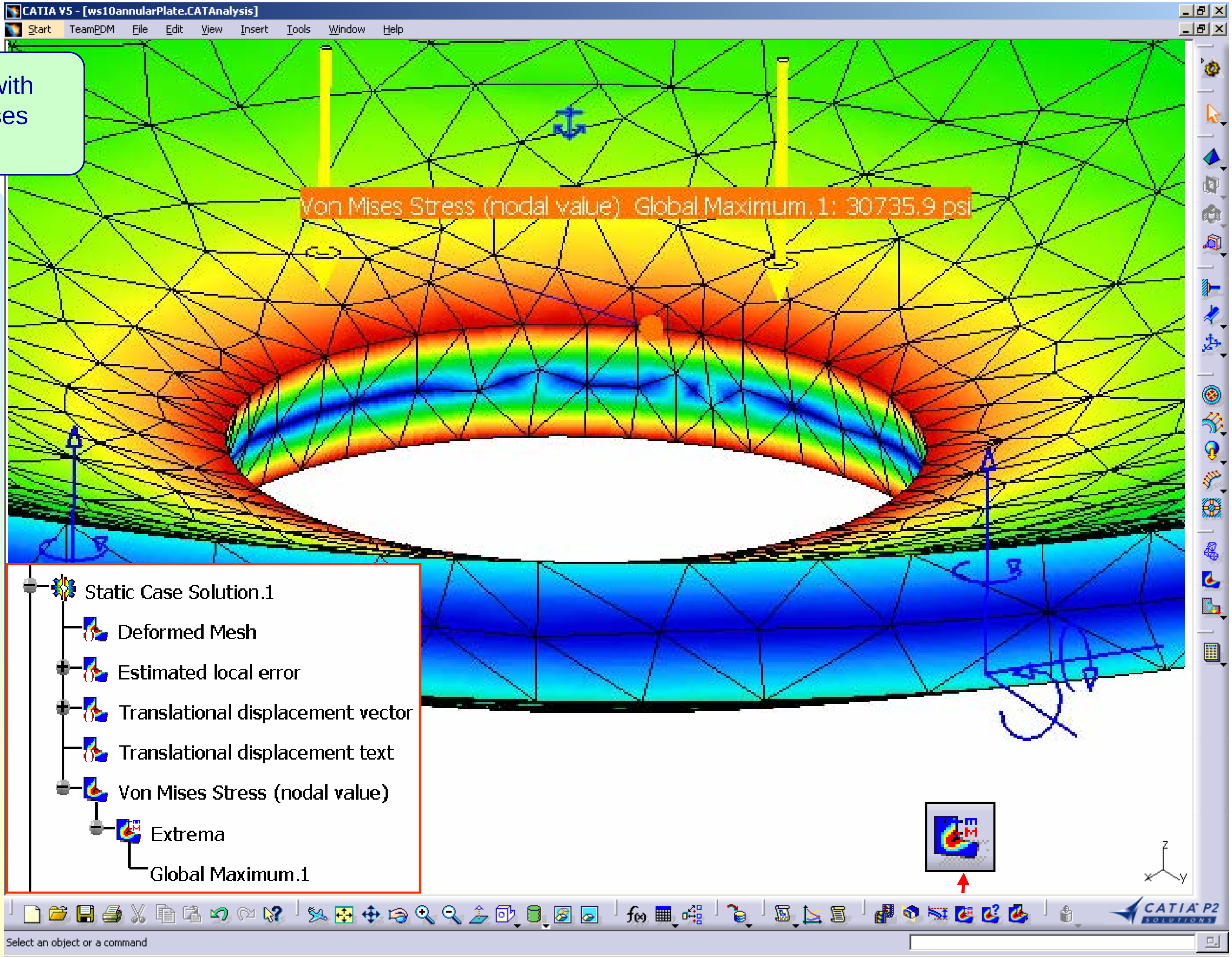
Visualize **Von Mises stress field patterns.**



MSC SOFTWARE

Step 9. Visualize final results

Find the element with maximum Von Mises Stress.



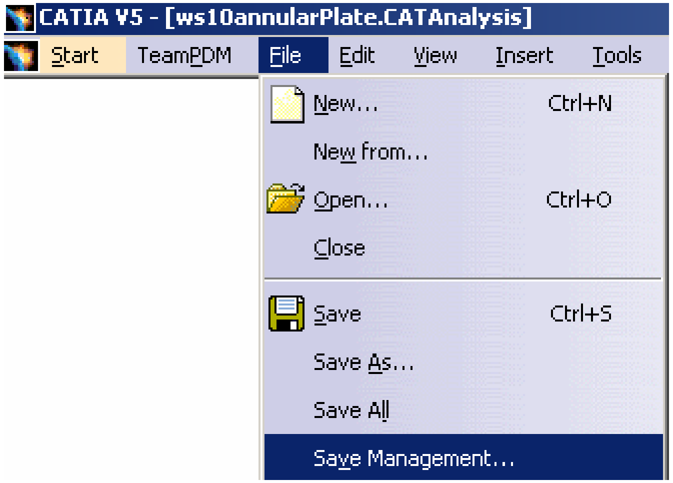
Conclusions

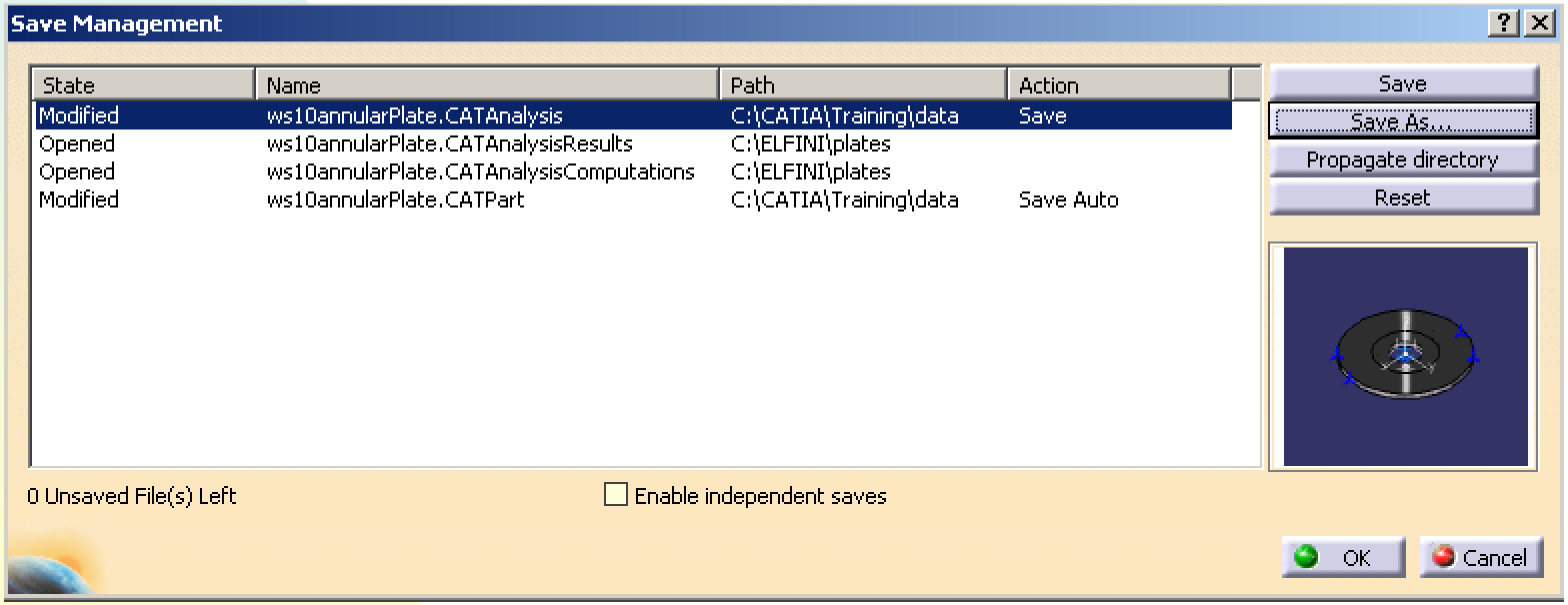
- CATIA V5 GSA workbench is validated for a annular flat circular plate scenario.

	Hand Calculations	.125 inch Parabolic Global Mesh, .013 inch sag .06 inch Local Mesh, .006 inch sag
Global % Precision error Local % Precision error	NA NA	3.03 % 3.76 %
Error Estimate	NA	3.02e-9 Btu global
Translational Displacement	-0.018 inch	-0.021 inch
Max Von Mises Stress	26934 psi	30736 psi

Step 10. Save the analysis document

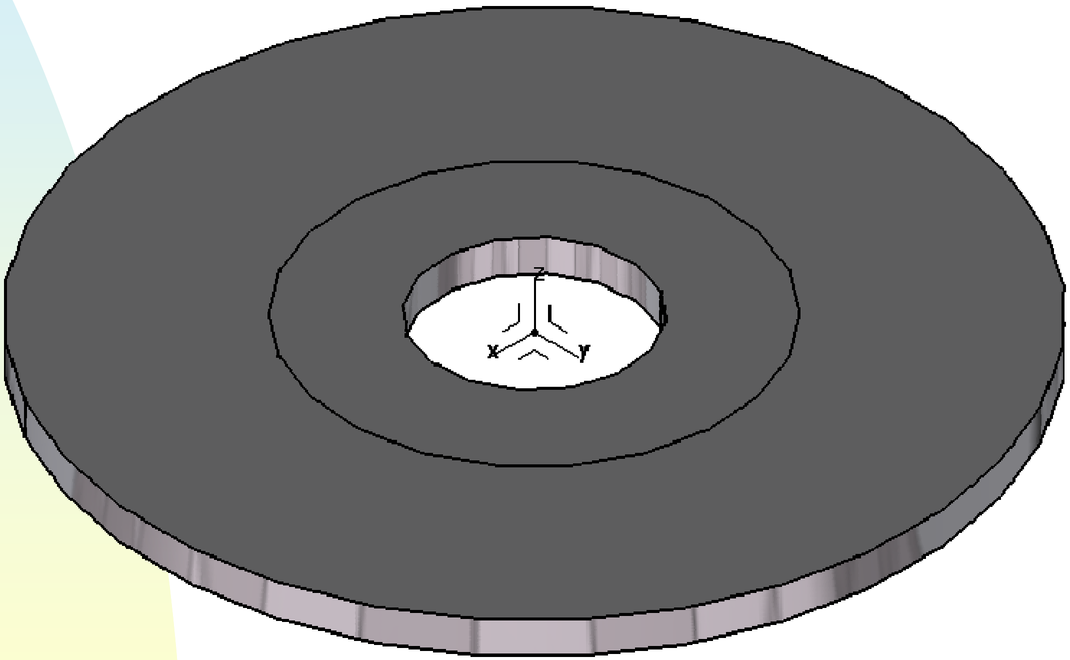
Save your documents





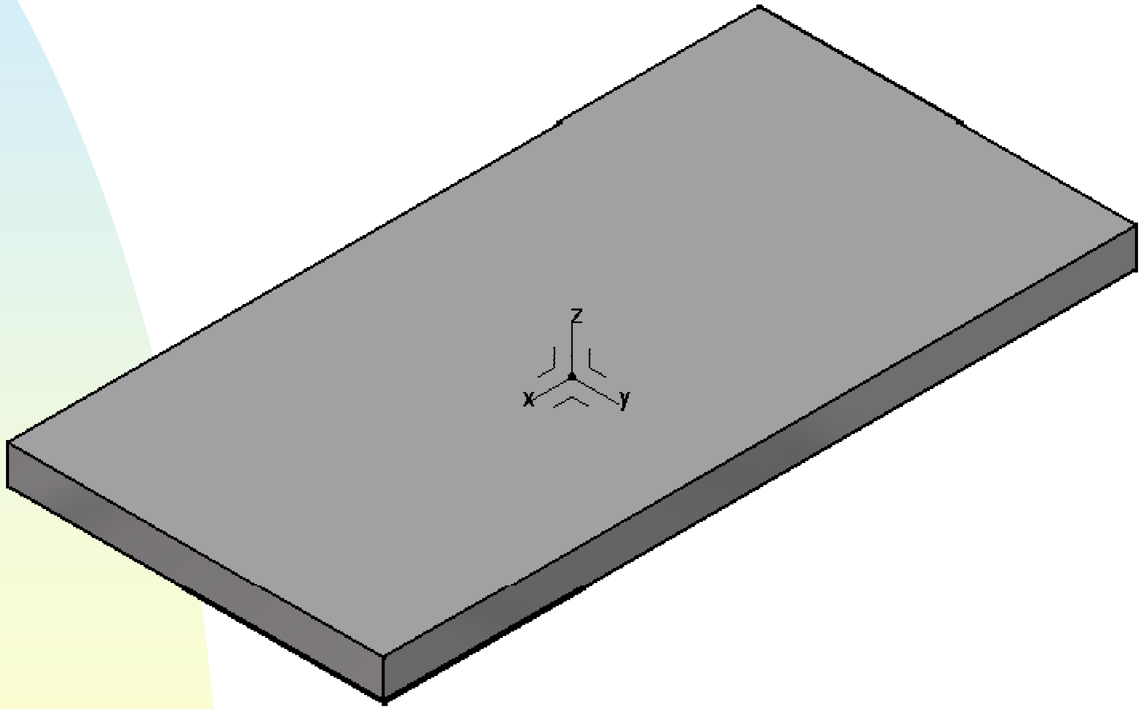
WORKSHOP 10…

OUTER EDGE SIMPLY SUPPORTED, INNER EDGE GUIDED
OUTER EDGE SIMPLY SUPPORTED, INNER SIMPLE SUPPORTED
OUTER EDGE SIMPLY SUPPORTED, INNER EDGE FIXED
OUTER EDGE FIXED, INNER EDGE FREE



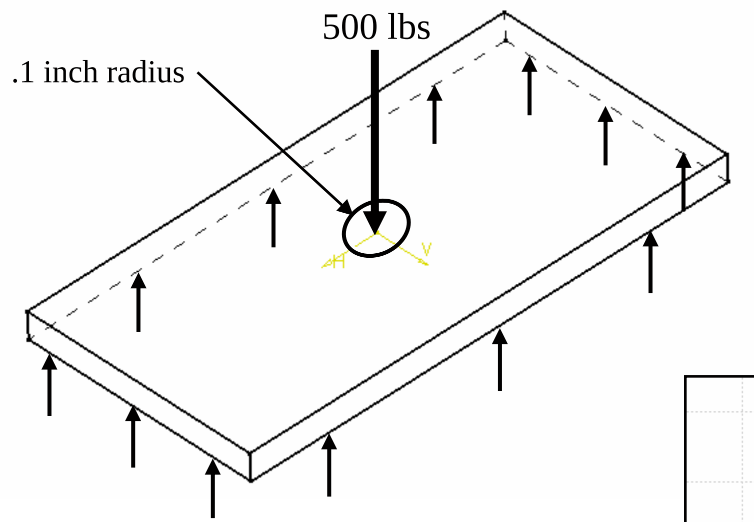
WORKSHOP 10b

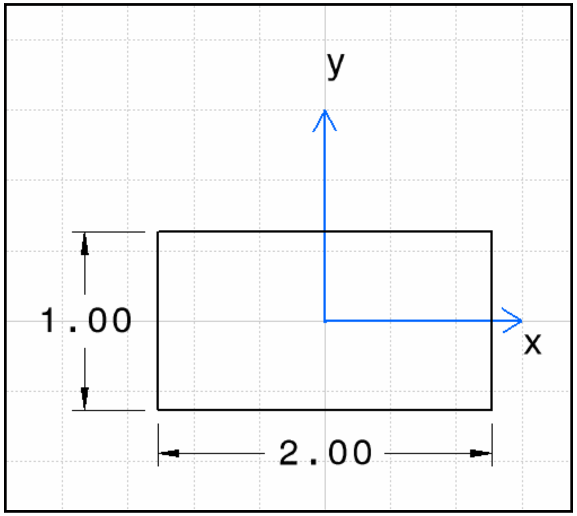
RECTANGULAR PLATE
SMALL CONCENTRIC CIRCLE LOAD



Problem Description

- All edges are simply supported.
- Uniform load over small concentric circle applied at the center.





Material: Aluminum
Young Modulus = 29e6 psi
Poisson Ratio = .3
Density = .283 lb_in3
Yield Strength = 36000 psi

Design requirements:

Thickness, t =	0.1 inch
Radius of contact, r_o =	0.1 inch
Vertical Load, W =	500 lbs

MSC SOFTWARE

Hand calculations

- Maximum Bending Stress:

$$\text{Maximum Bending Stress (at center)} = \sigma_{max} = \frac{3 \cdot W}{2 \cdot \pi \cdot t^2} \cdot \left[(1 + \nu) \cdot \ln \frac{2 \cdot b}{\pi \cdot r_o'} + \beta \right]$$

$$\sigma_{max} = \frac{3 \cdot 500 \text{ lbs}}{2 \cdot \pi \cdot 0.1^2 \text{ inch}} \cdot \left[(1 + 0.3) \cdot \ln \frac{2 \cdot 1.0 \text{ inch}}{\pi \cdot 0.1 \text{ inch}} + 0.958 \right] = 80317 \text{ psi}$$

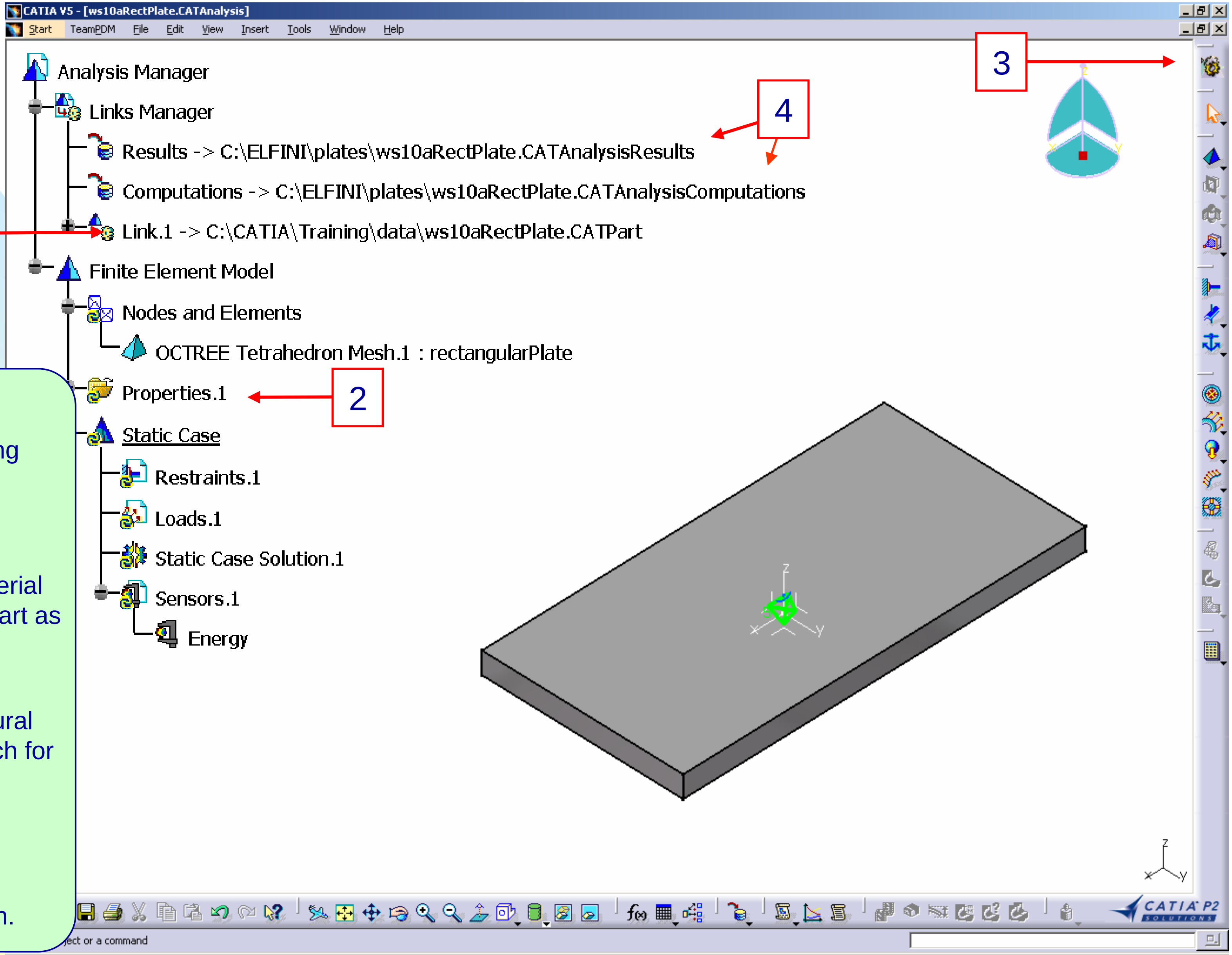
- Maximum Vertical Deflection:

$$\text{Maximum Vertical Deflection} = y = \frac{\alpha \cdot W \cdot b^2}{E \cdot t^3} = \frac{-0.1805 \cdot 500 \text{ lbs} \cdot 1.0^2}{29e^6 \cdot 0.1^2} = 0.00311 \text{ inch}$$

Suggested Exercise Steps

1. Create a new CATIA analysis document (.CATAnalysis).
2. Mesh globally with parabolic elements.
3. Apply an advanced and isostatic restraint (simply supported).
4. Apply a force.
5. Compute the initial analysis.
6. Check global and local precision (animate deformation, adaptive boxes and extremas).
7. Refine the mesh locally with an adaptivity box.
8. Visualize final results.
9. Save the analysis document.

Step 1. Create a new CATIA analysis document



Steps:

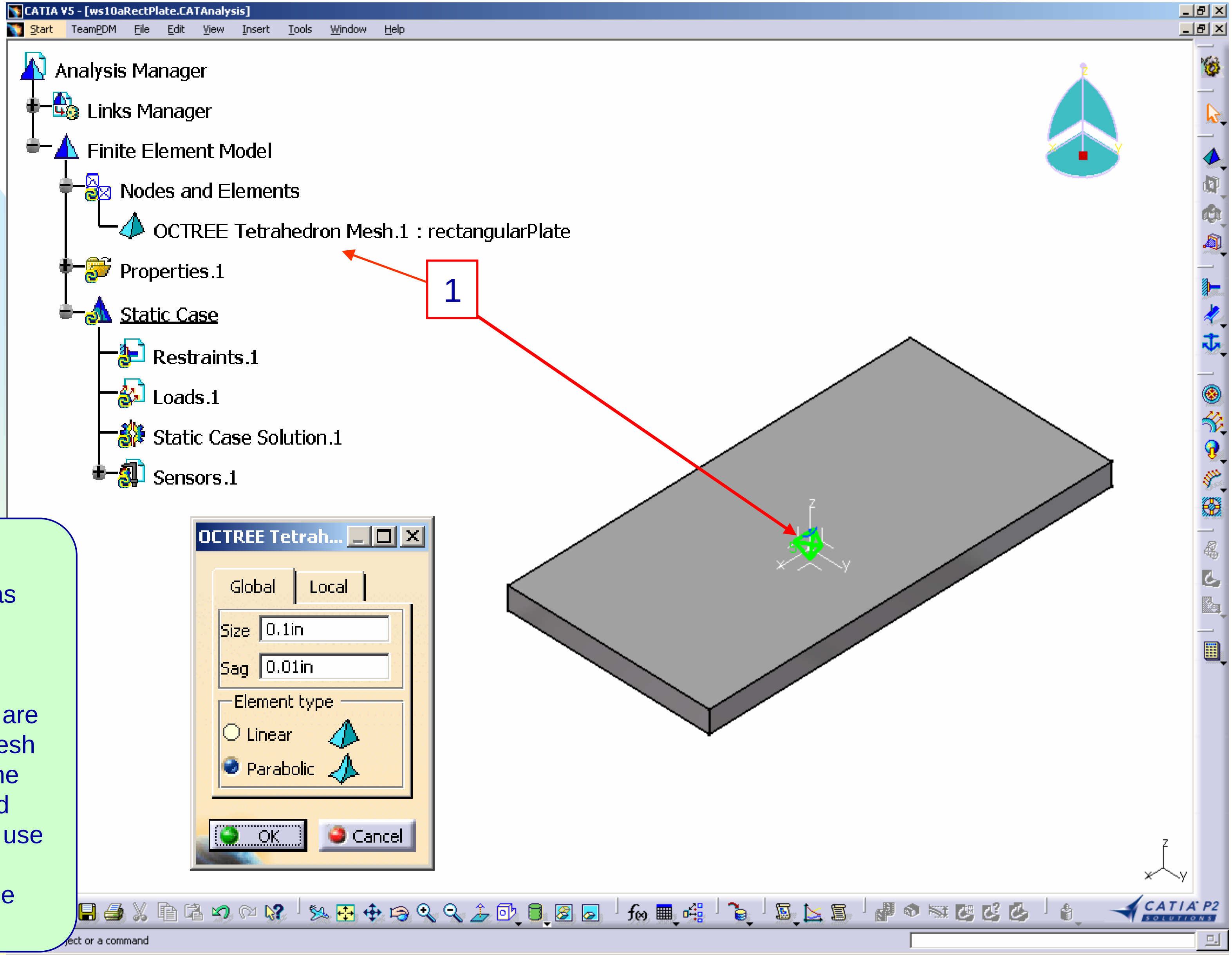
1. Open the existing ws10bRectPlate .CATPart from the training directory.

2. Apply steel material properties to the part as required.

3. Launch the Generative Structural Analysis workbench for a Static Analysis.

4. Specify the Computations and Results storage locations as shown.

Step 2. Mesh globally with parabolic elements

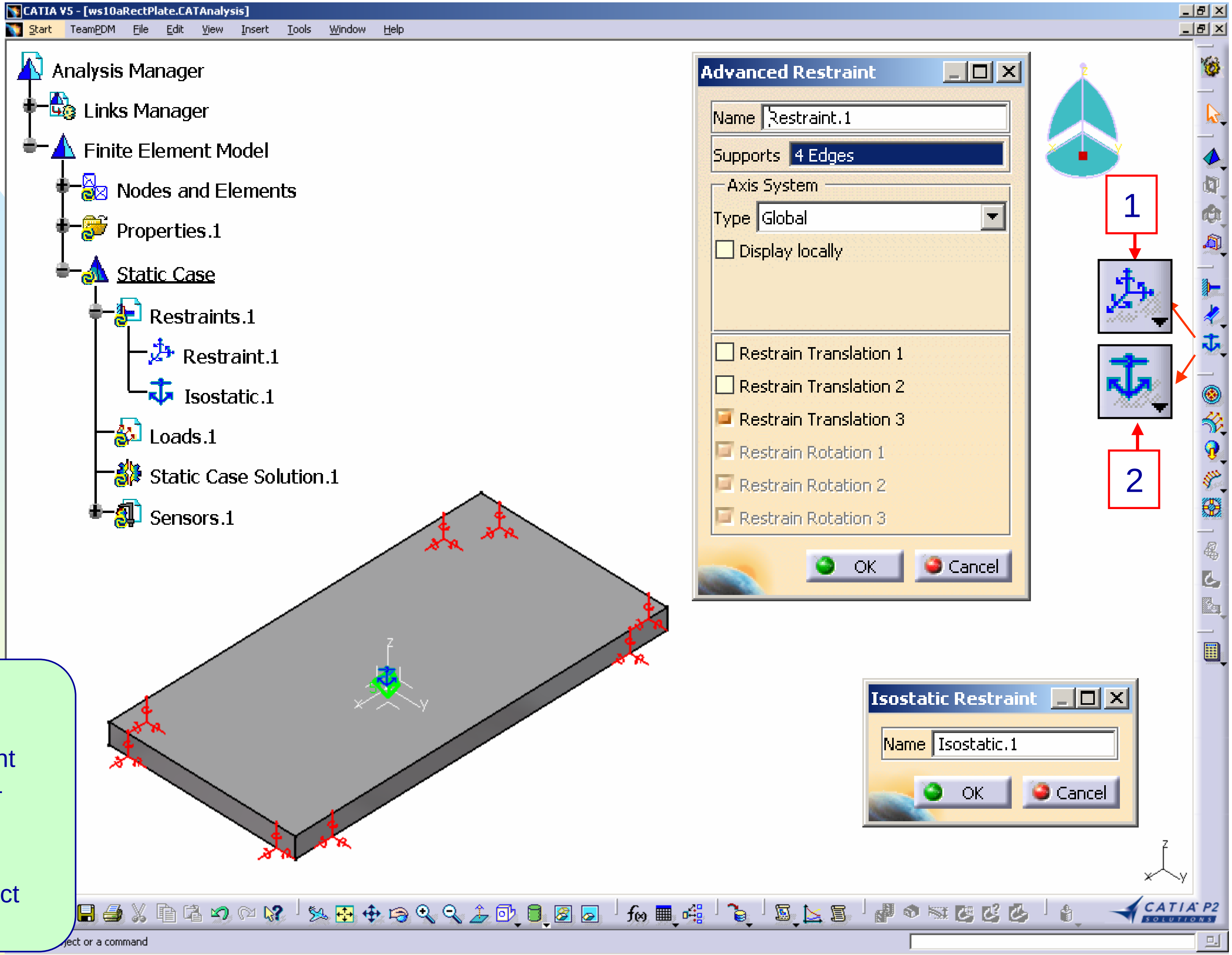


Steps:

1. Globally mesh as shown.

As plates typically are large using one mesh element through the thickness is a good way to start. Then use localized adaptive meshing for precise results.

Step 3. Apply an advanced and isostatic restraint



Steps:

1. Select the Advanced Restraint icon, restrain the 4 bottom edges.

2. Select isostatic restraint icon, select OK.

Step 4. Apply a force

"Trick of the trade" : Special construction techniques are necessary to enable you to apply a force to patterns that do not exist on parts.

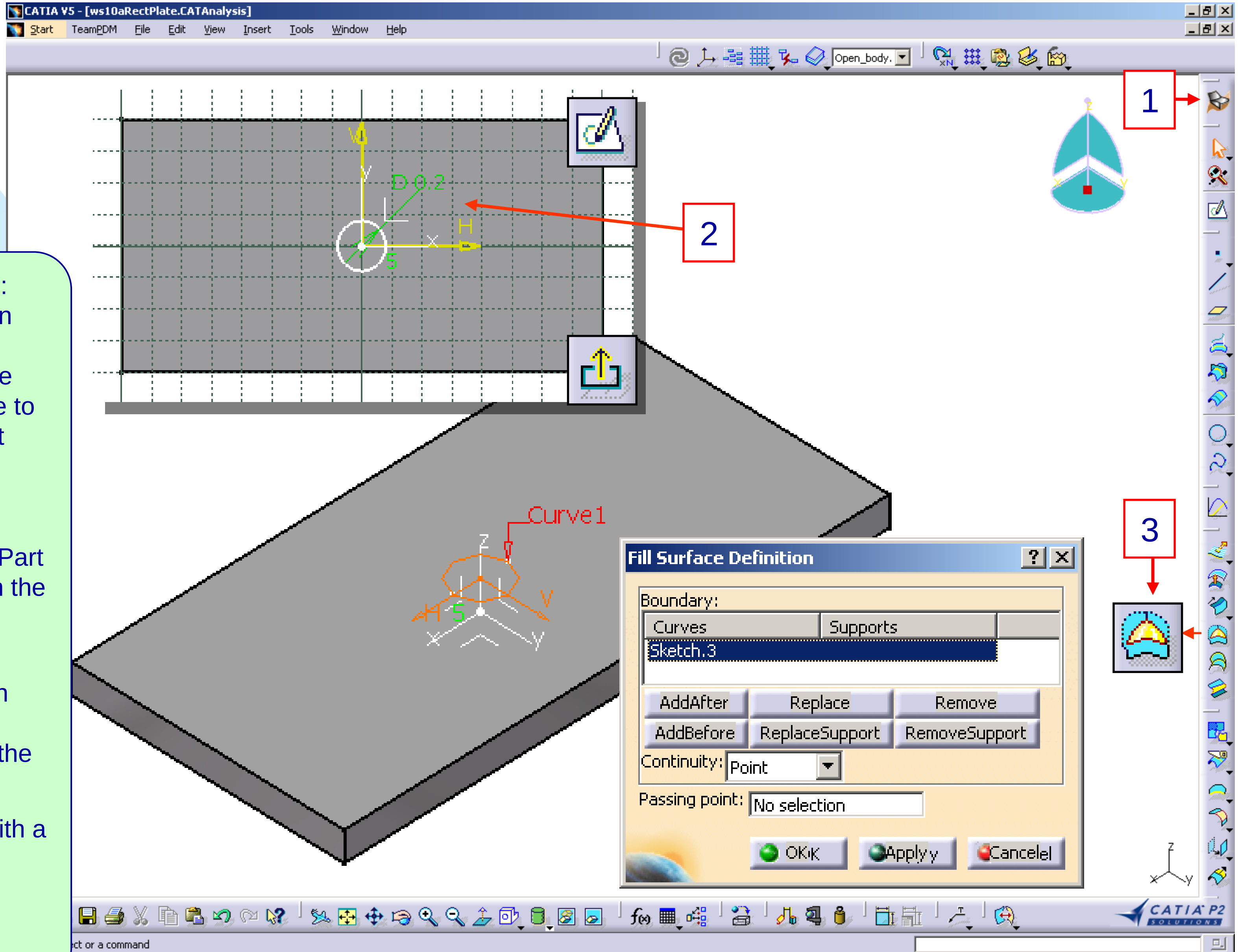
Steps:

1. Make your .CATPart current and Launch the Generative Shape Design workbench.

2. Sketch a 0.2 inch diameter circle centered on top of the plate.

3. Fill this sketch with a surface.

Continue on….



Step 4. Apply a force

Make a solid thickness based on the surface.

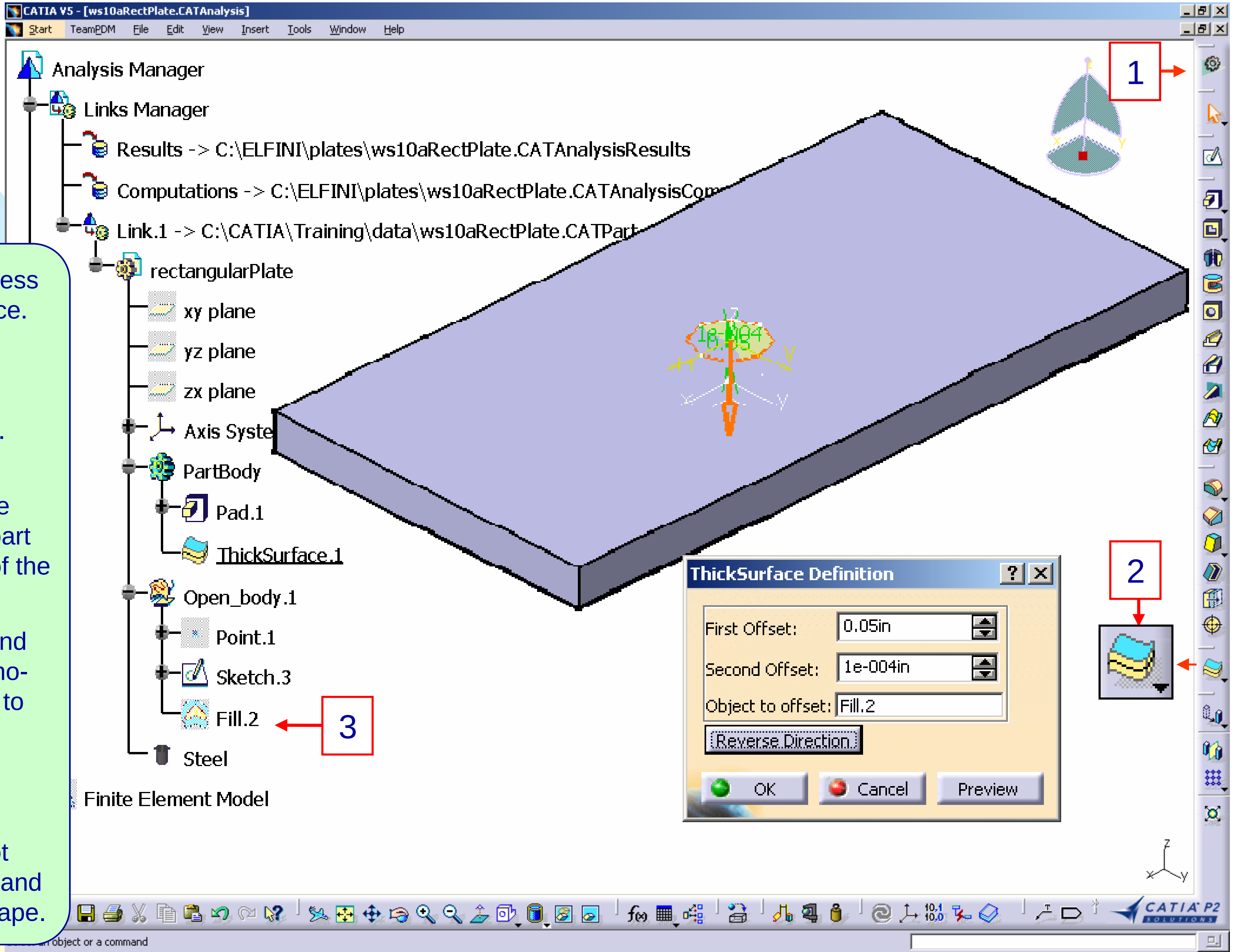
Steps:

1. Go to the Part Design Workbench.

2. Create a "thick" feature 1/2 the plate thickness into the part and a fraction out of the part (0.0001 inch).

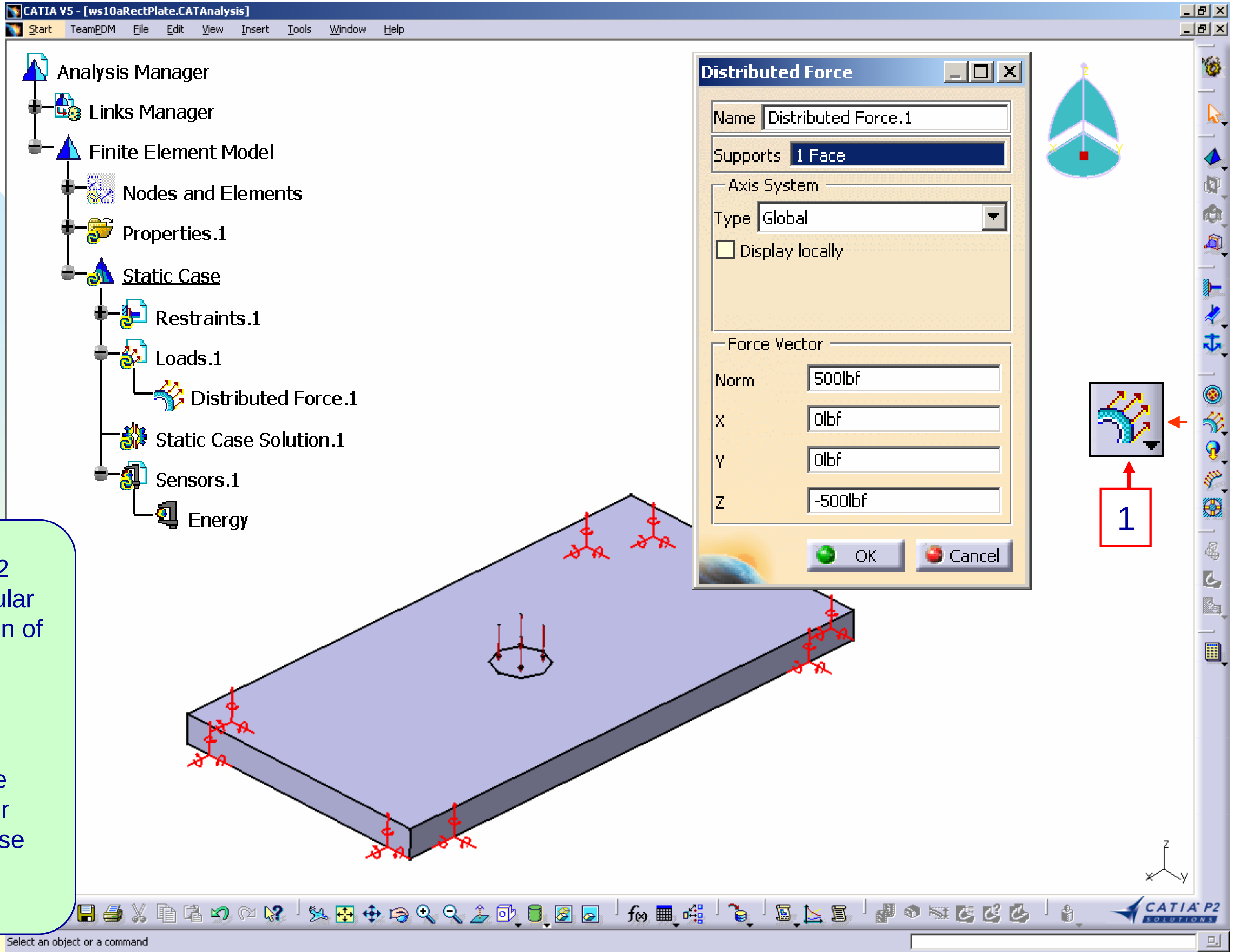
3. Put the Sketch and the Fill features in no-show then go back to the analysis workbench.

This method will not effect stress levels and will work on any shape.



MSC SOFTWARE

Step 4. Apply a force



Now we have a 0.2 inch diameter circular pattern in a location of our choice that is selectable.

Steps:

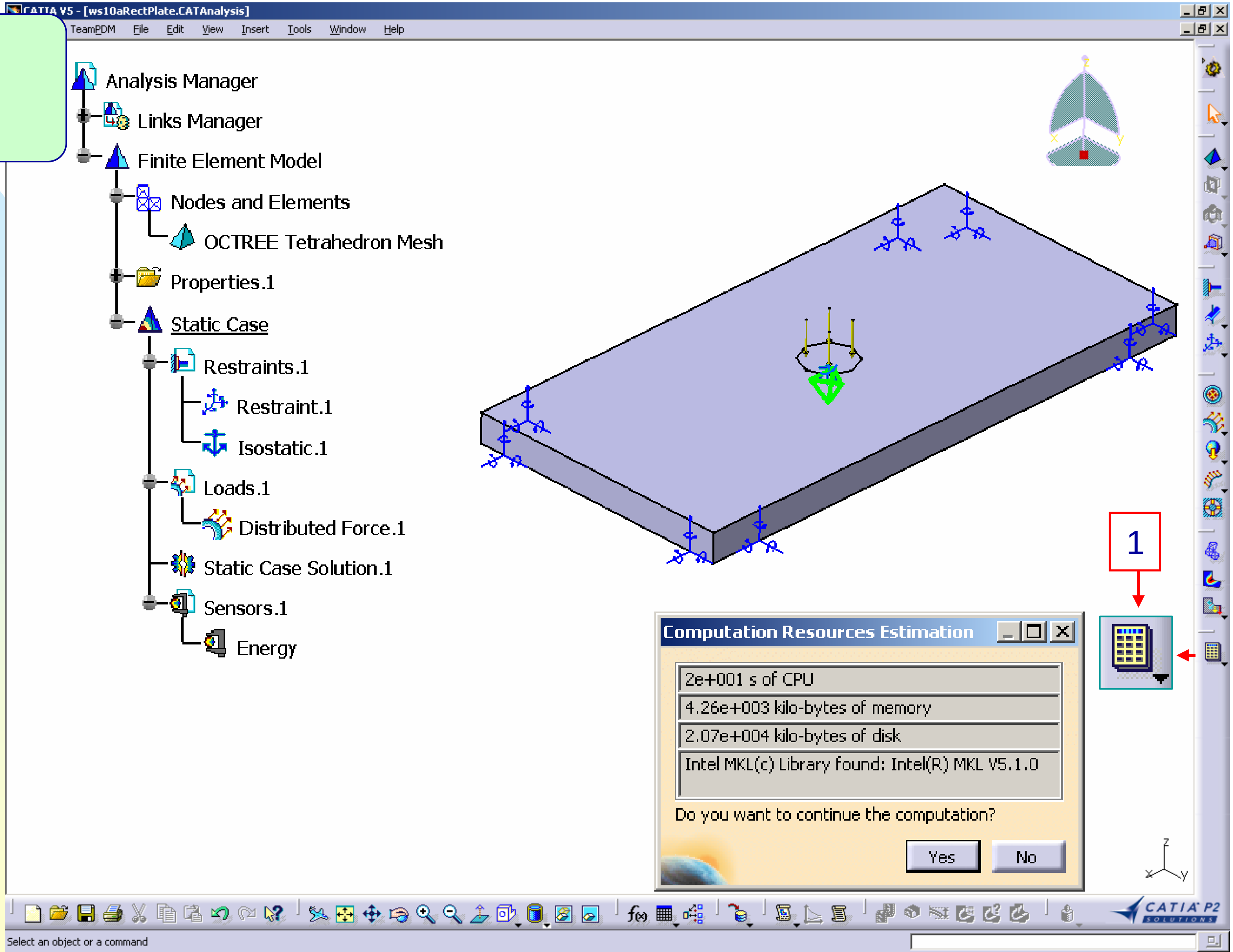
1. Select the Force icon and the center selectable area. Use force magnitude values as shown.

Step 5. Compute the initial analysis

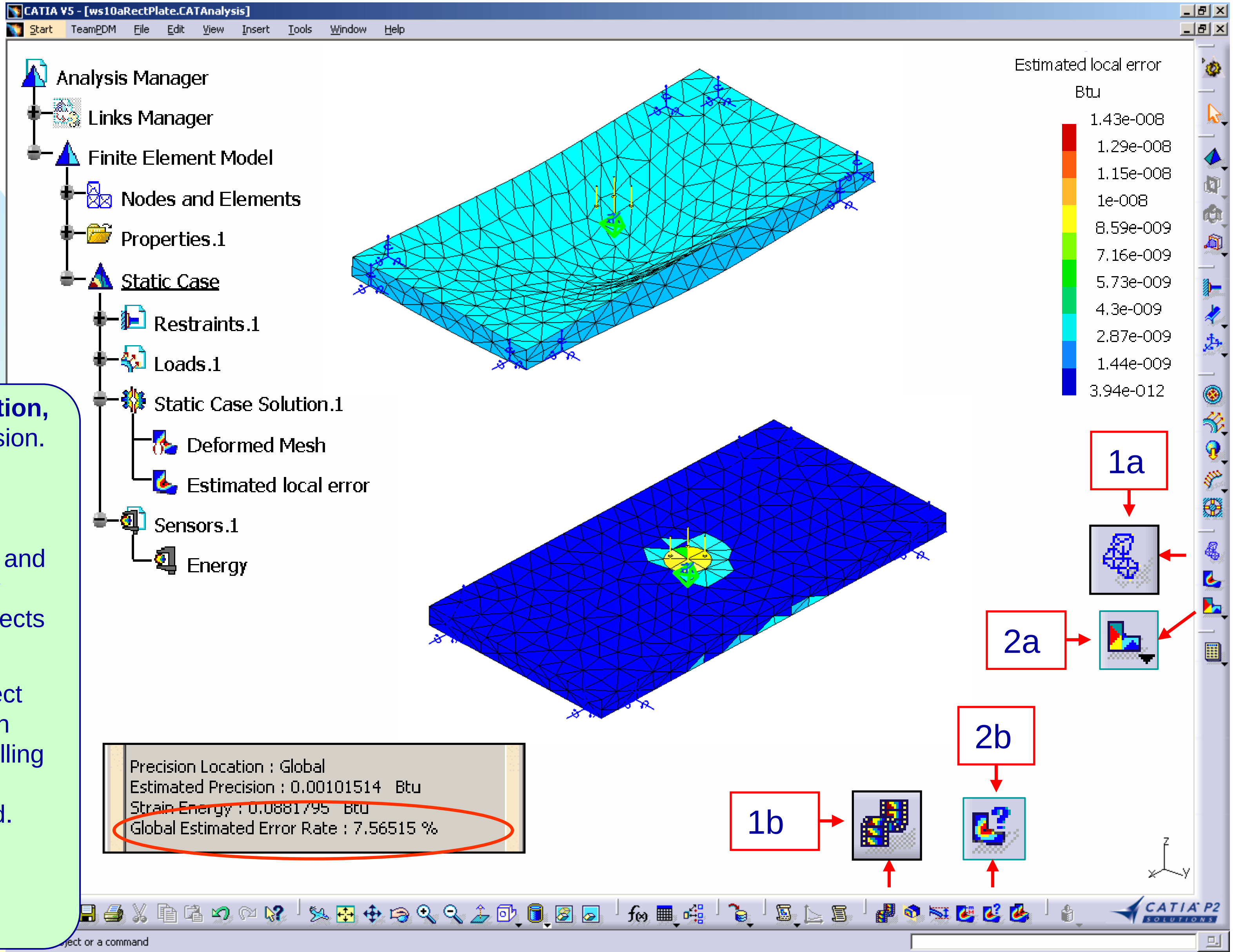
Save first.

Steps:

1. Compute All



Step 6. Check global and local precision



Check **Deformation,** and global precision.

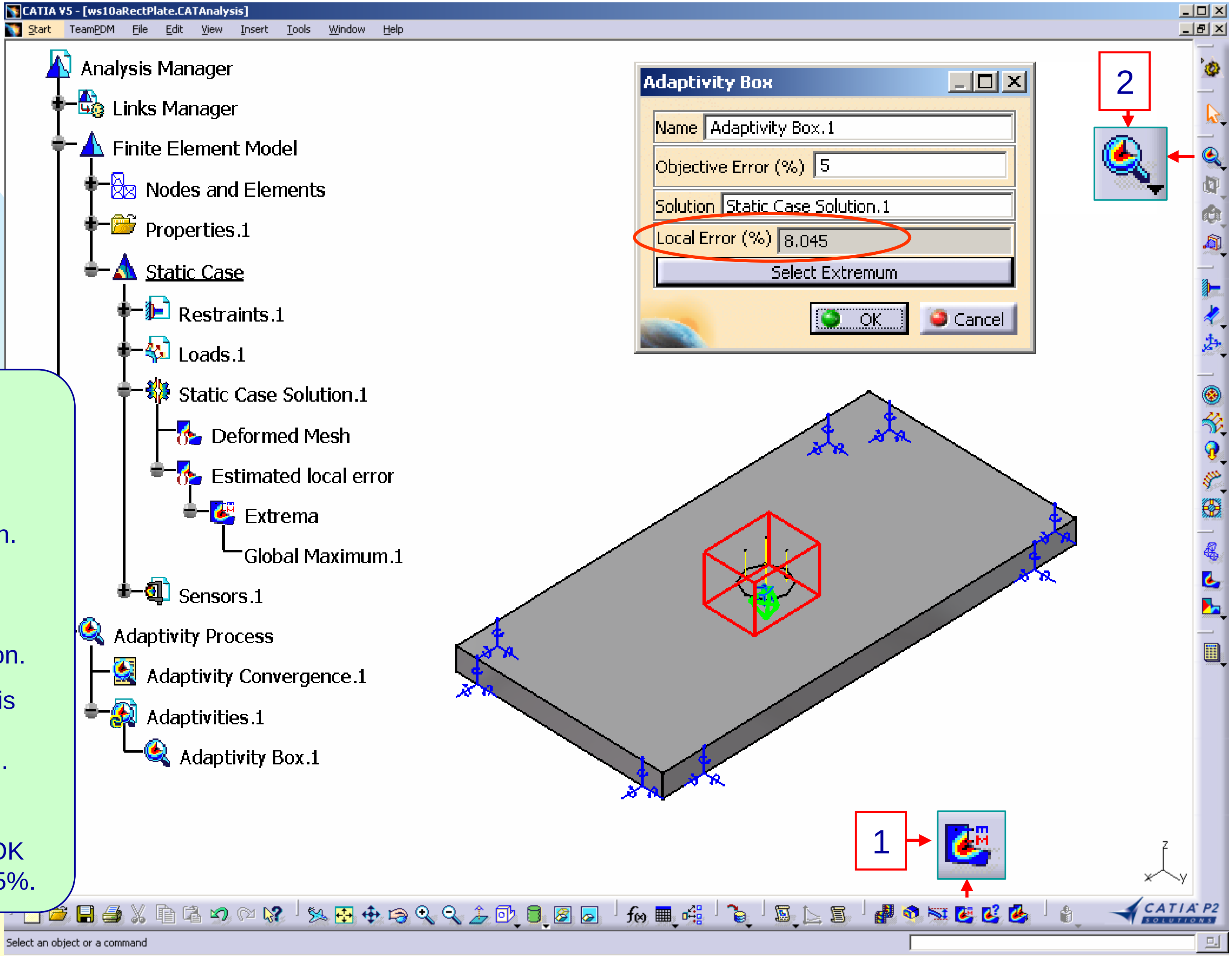
Steps:

1. Create a deformed image and animate to verify your system deflects as expected.

You should expect even deformation and the sides pulling in representing simply supported.

2. Check Global precision (looks good).

Step 6. Check global and local precision



Find the global element with the highest estimated error.

Find local precision.

Steps:

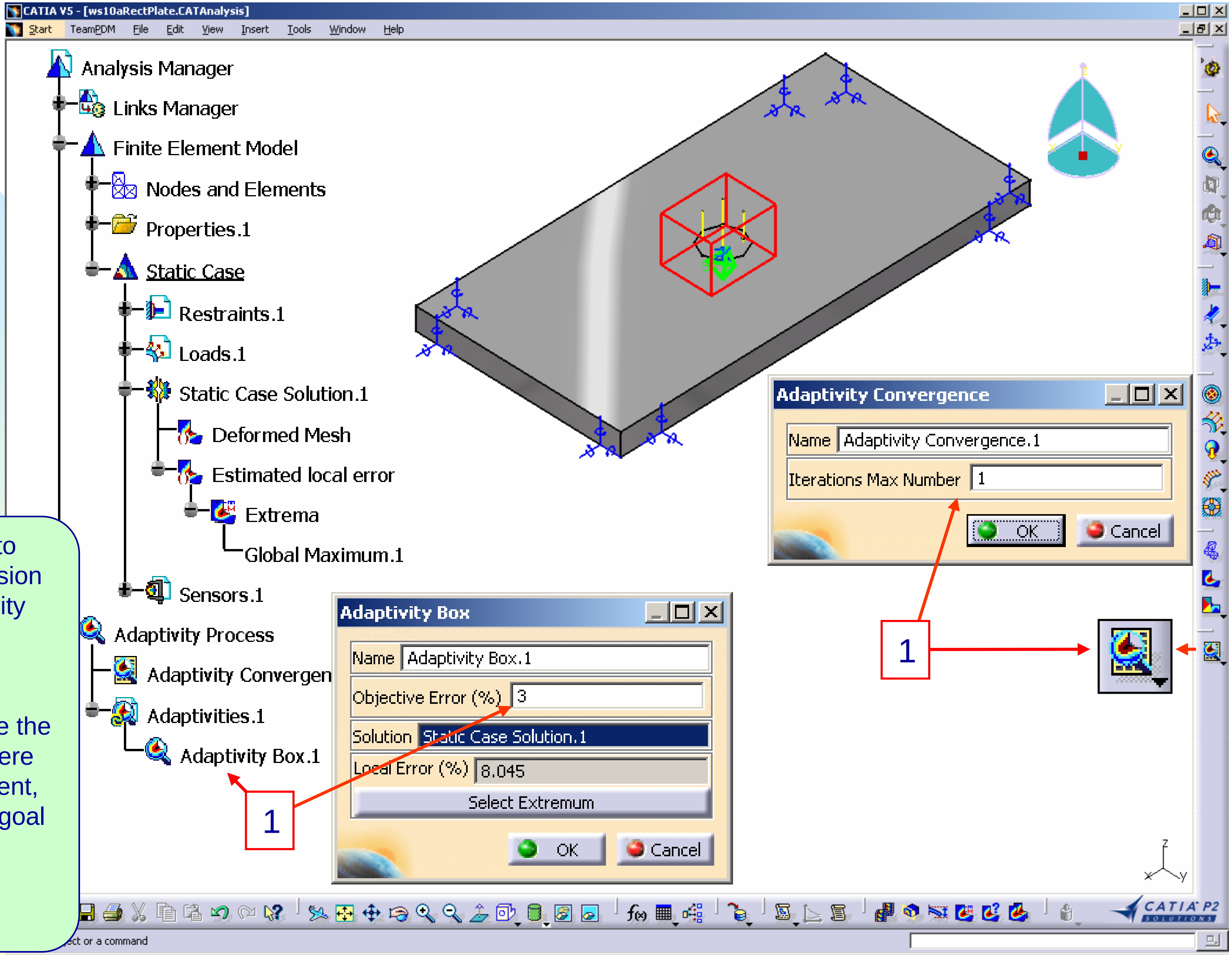
1. Use the Search Image Extrema icon.

2. Local precision is found using the adaptivity box icon.

Local error looks OK but I might prefer 5%.

MSC SOFTWARE

Step 7. Refine the mesh locally



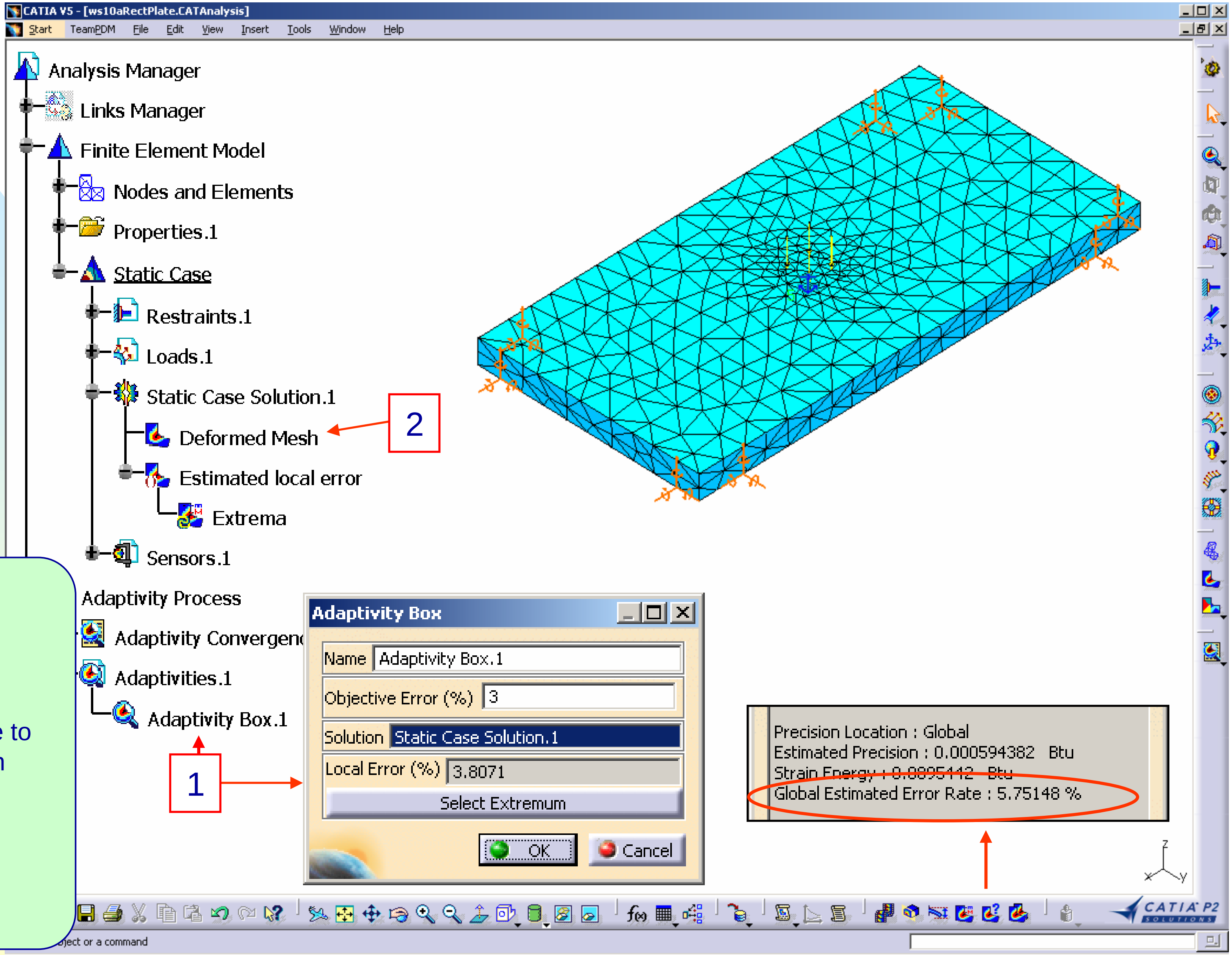
Refine the mesh to achieve 5% precision inside the adaptivity box.

Steps:

1. Locate and size the adaptivity box where you want refinement, as shown. Use a goal of 3% error.

2. Use only one convergence.

Step 8. Visualize final results



Steps:

1. Check local precision again.

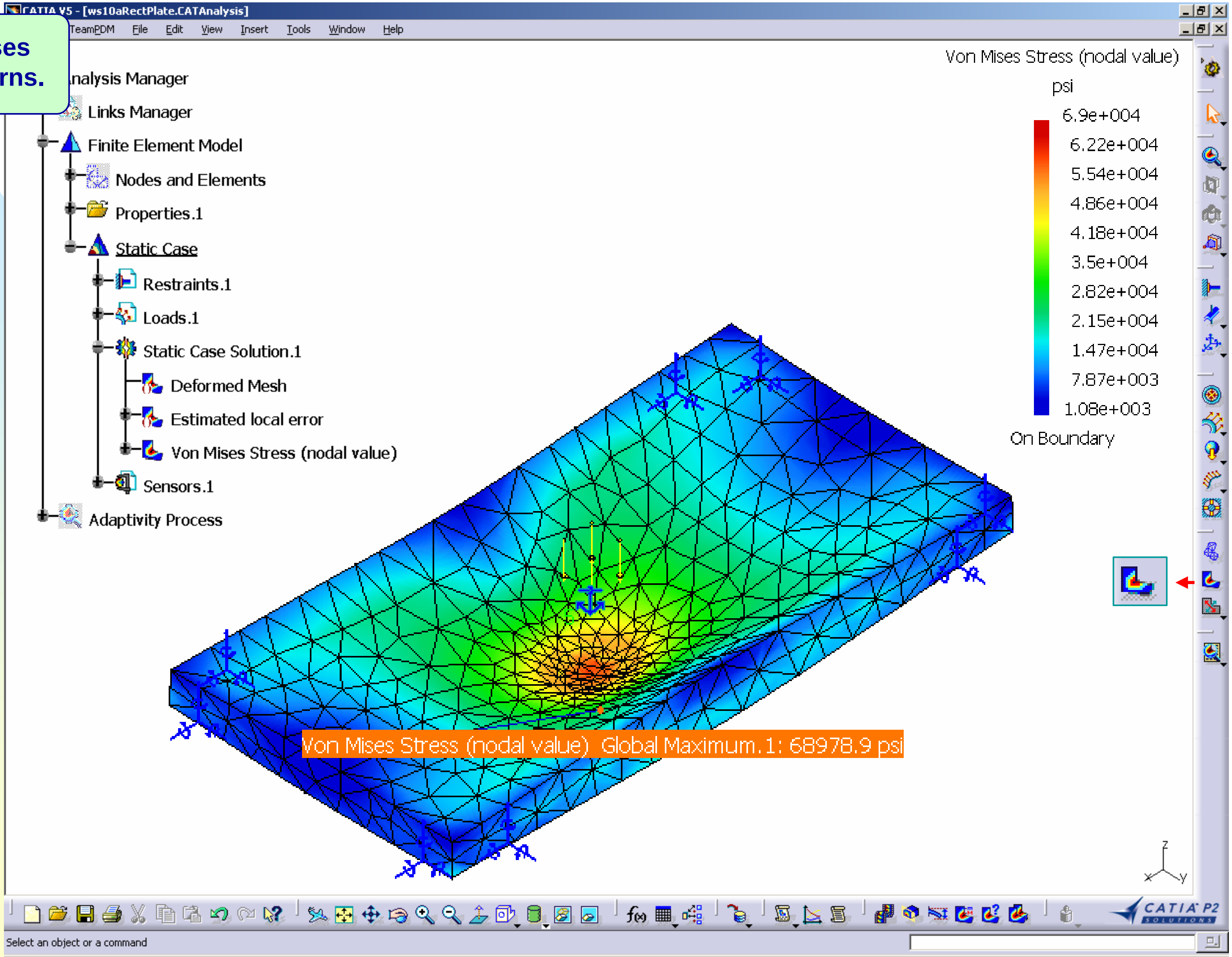
2. Activate the deformation image to see the local mesh refinement.

We now have a precise model.

MSC SOFTWARE

Step 8. Visualize final results

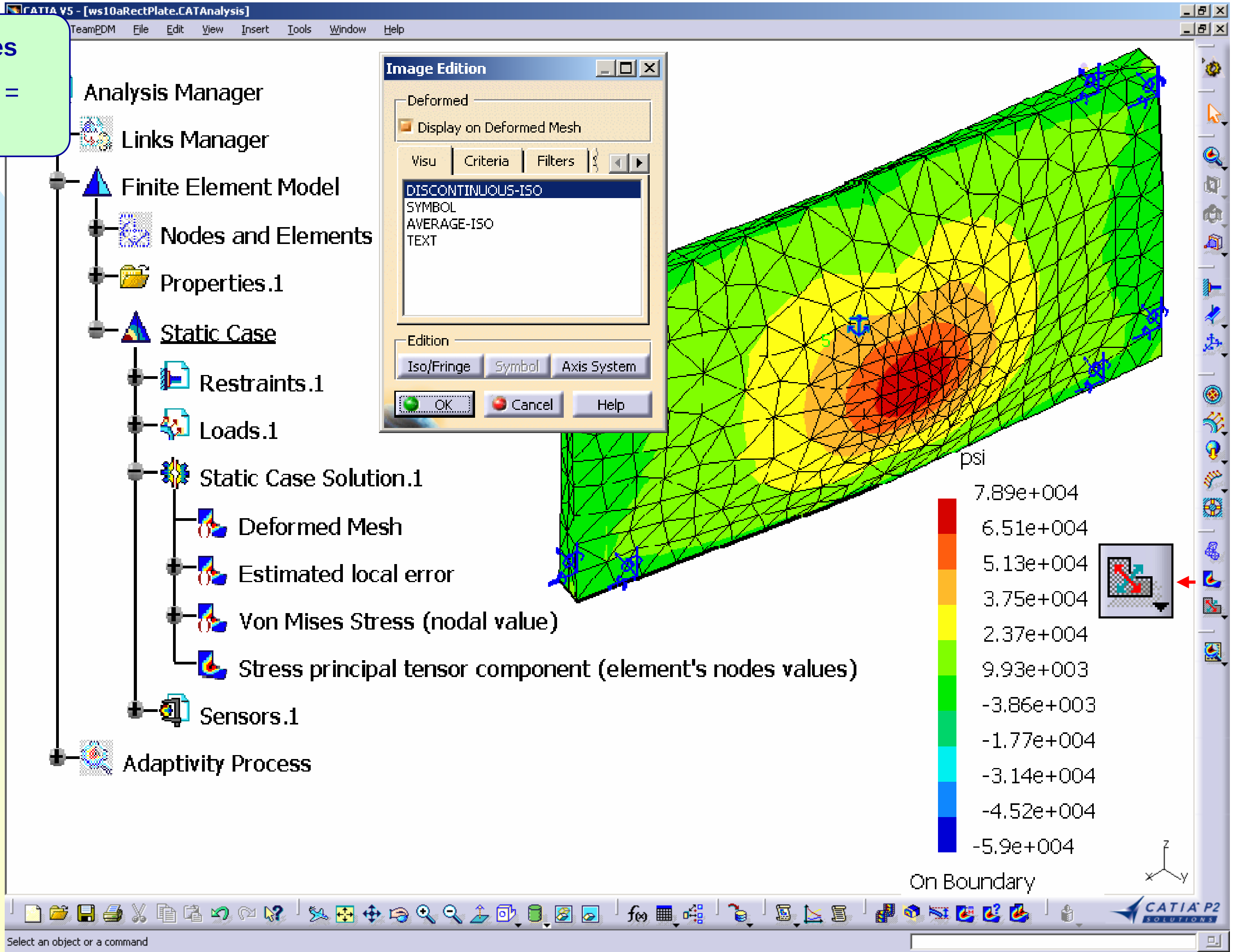
Visualize **Von Mises stress field patterns.**



Step 8. Visualize final results

Principal Stresses

Hand calculations = 80,317 psi

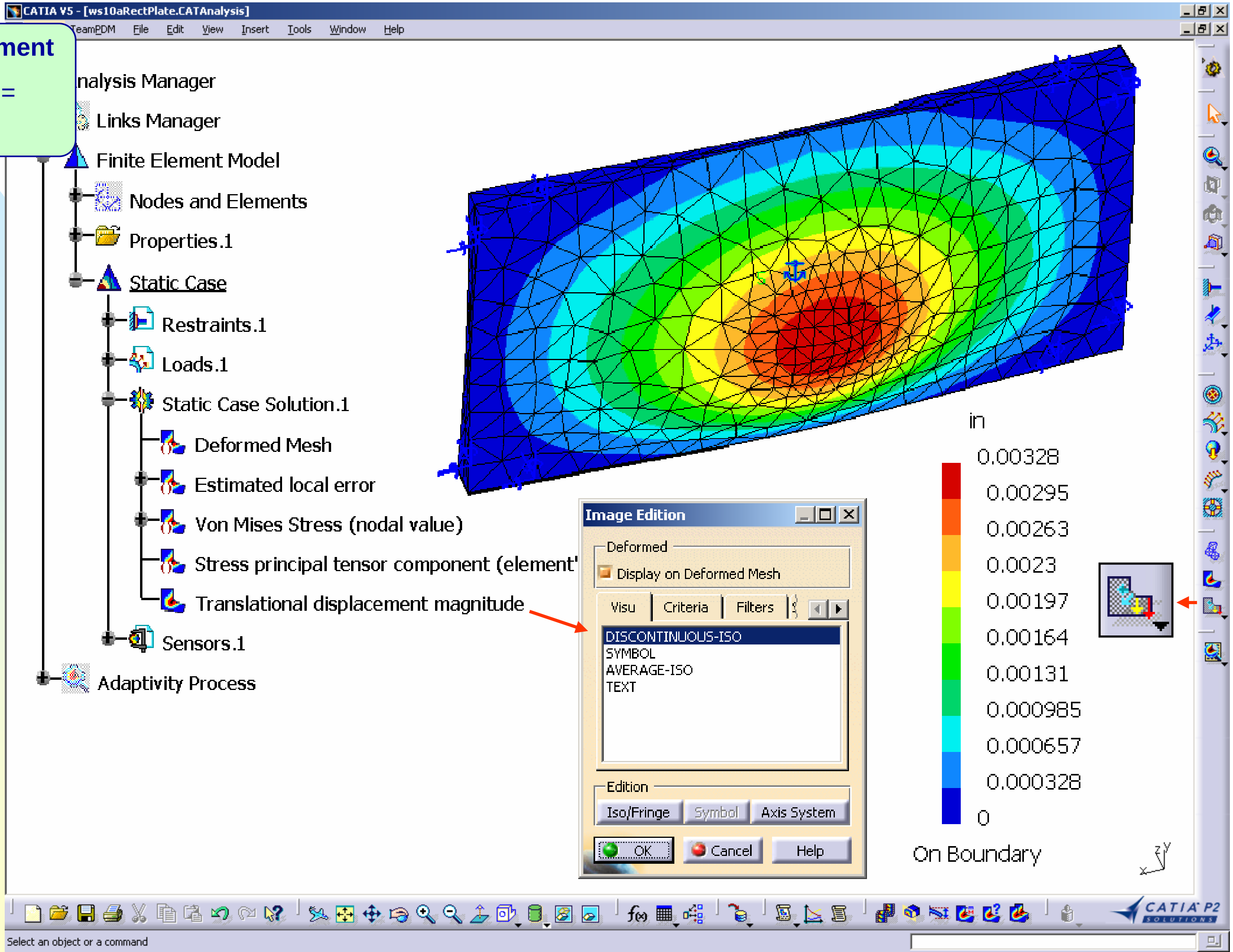


MSC SOFTWARE

Step 8. Visualize final results

Vertical displacement

Hand calculations = 0.003 inches



Step 8. Visualize final results

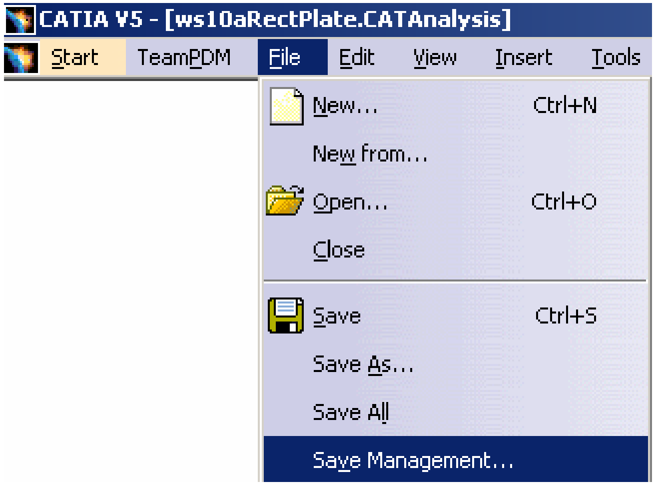
Conclusions

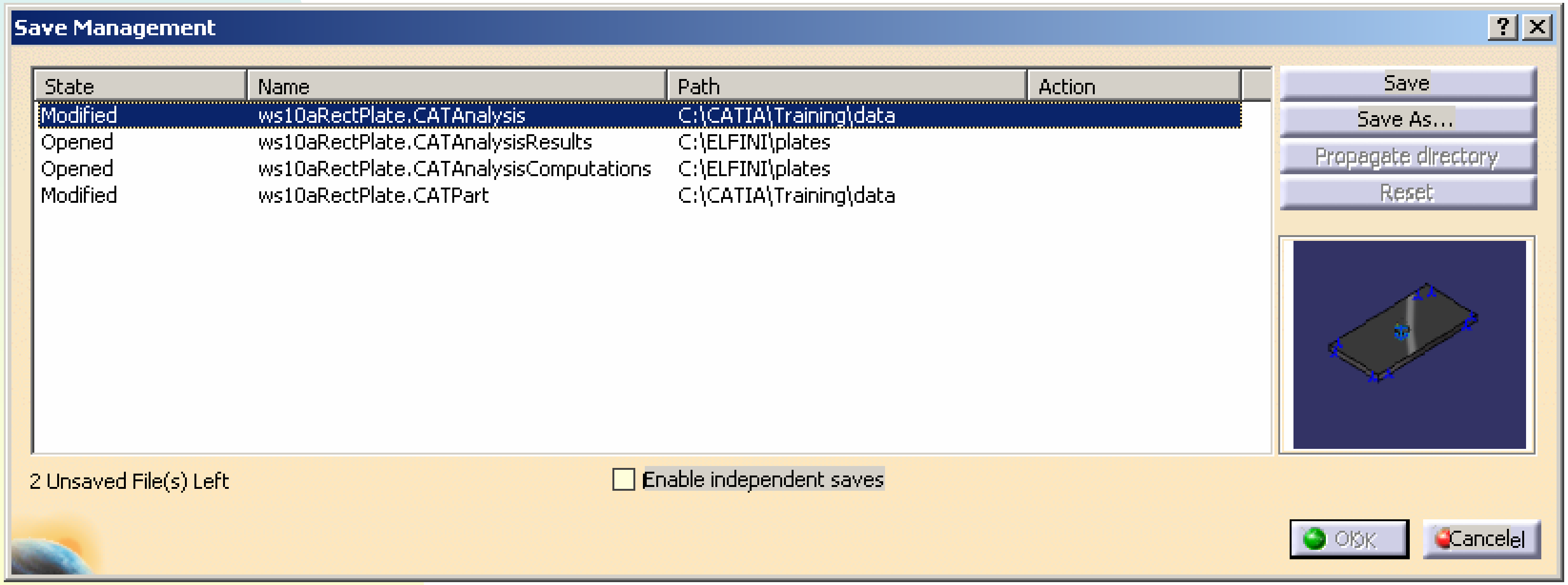
- CATIA V5 GSA workbench is validated for a rectangular flat plate with a uniform load over a small concentric circle scenario.

	Hand Calculations	.125 inch Parabolic Global Mesh, .013 inch sag .06 inch Local Mesh, .006 inch sag
Global % Precision error Local % Precision error	NA NA	5.7 % 3.8 %
Error Estimate	NA	4.93e-9 Btu global
Translational Displacement	-0.003 inch	-0.00328inch
Max Von Mises Stress	80317 psi	78900 psi

Step 9. Save the analysis document

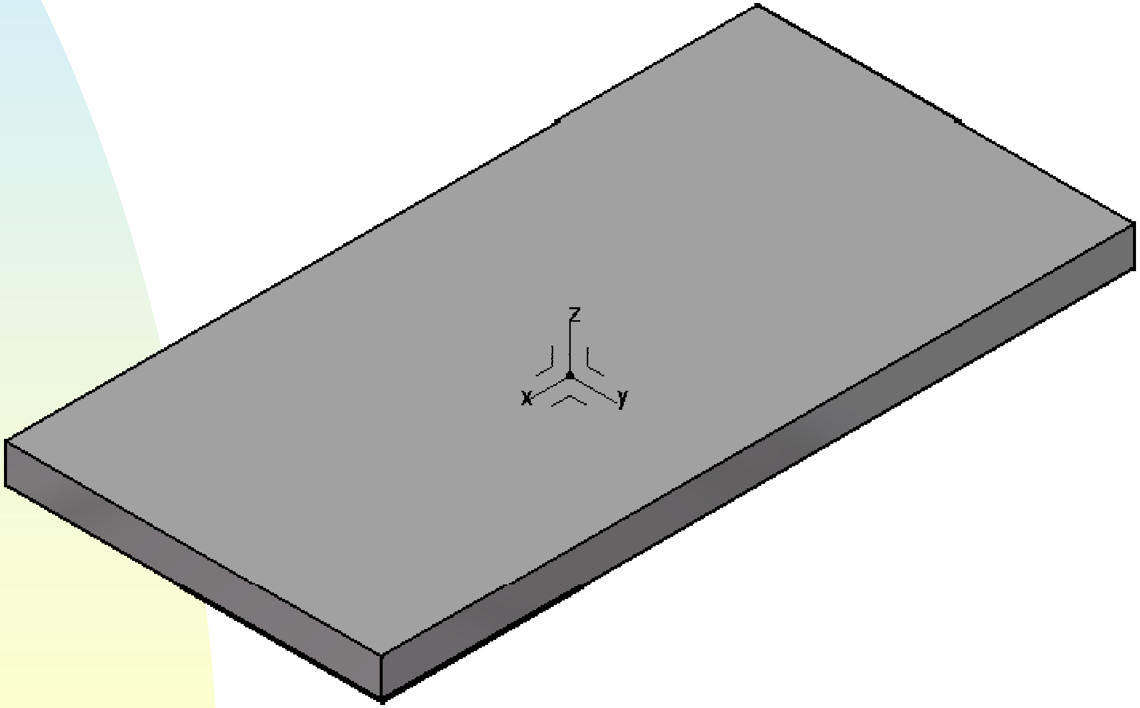
Save your documents





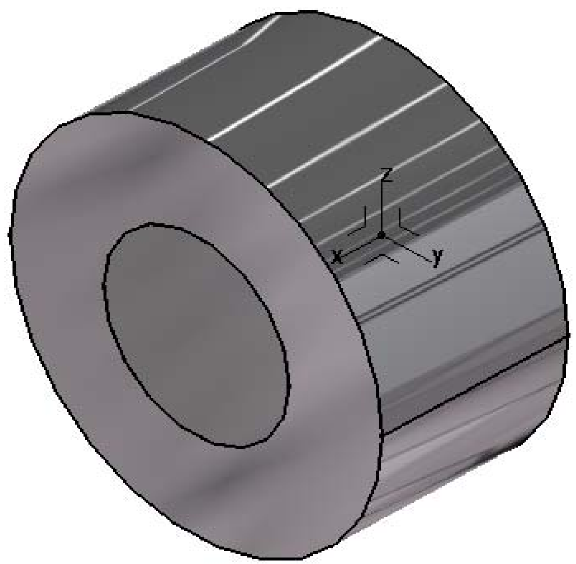
WORKSHOP 10…

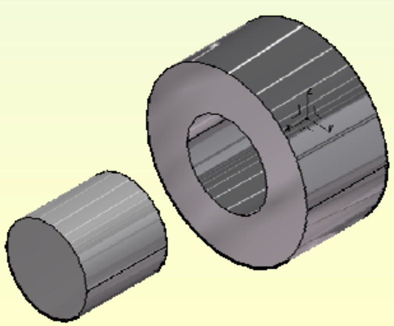
FOUR EDGES FIXED,
TWO EDGES SIMPLY SUPPORTED - TWO EDGE FREE
THREE EDGES FIXED
TWO EDGES FIXED



WORKSHOP 11

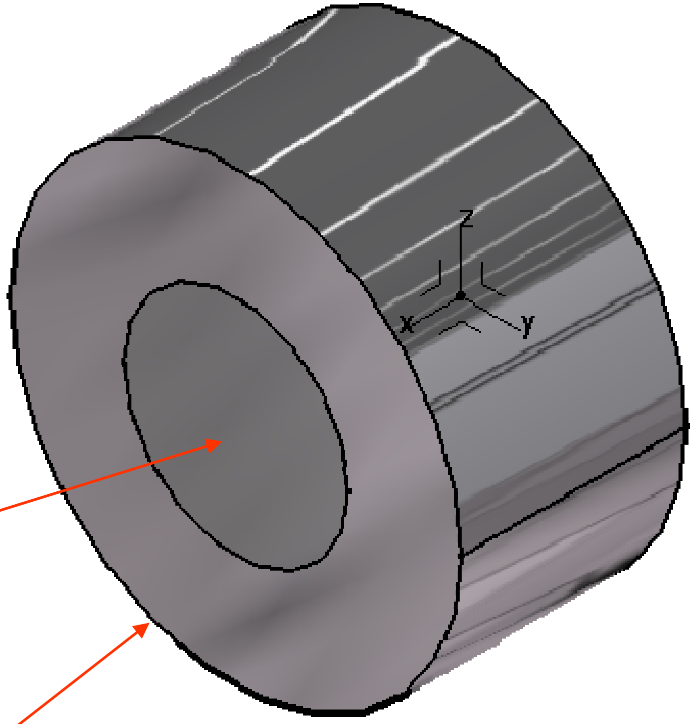
PRESS FIT





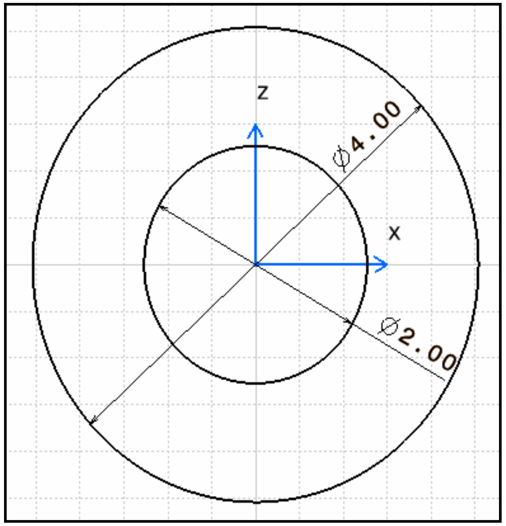
Problem Description

- Determine the contact pressure and the hoop stress for a class FN4 force fit (0.010 inch of interference on the diameter) of a steel plug into an aluminum cylinder.



Plug Material: Steel
Young Modulus = 29e6 psi
Poisson Ratio = .266
Density = .284 lb_in3
Yield Strength = 36259 psi

Outside Cylinder Material: Aluminum
Young Modulus = 10.15e6 psi
Poisson Ratio = .346
Density = .098 lb_in3
Yield Strength = 13778 psi



Note: parts are modeled net size

MSC SOFTWARE

Hand calculations

- Contact pressure due to 0.010 inch press fit on the diameter.

Assume the limit of interference due to a press fit = 0.010 inch on the diameter = δ

$$\delta = \left(\frac{r_{\text{inner cylinder}} \cdot p}{E_{\text{outer cylinder}}}\right) \cdot \left[\frac{(r^2_{\text{outer cyl}} + r^2_{\text{inner cyl}})}{(r^2_{\text{outer cyl}} - r^2_{\text{inner cyl}})} + \nu_{\text{outer}}\right] + \frac{r_{\text{inner cyl}} \cdot p}{E_{\text{inner cylinder}}} \cdot (1 - \nu_{\text{inner}})$$

$$0.010in = \left(\frac{1.0_{\text{inner cylinder}} \cdot p}{10.15e^{6}_{\text{outer cylinder}}}\right) \cdot \left[\frac{(2.0^2_{\text{outer cyl}} + 1.0^2_{\text{inner cyl}})}{(2.0^2_{\text{outer cyl}} - 1.0^2_{\text{inner cyl}})} + 0.346_{\text{outer}}\right] + \frac{1.0_{\text{inner cyl}} \cdot p}{29e^{6}_{\text{inner cylinder}}} \cdot (1 - 0.266_{\text{inner}})$$

$$0.010 = \frac{2.013 \cdot p}{10.15e^{6}} + \frac{30e^{-9} \cdot p}{29e^{6}}$$

$$p = \frac{0.010}{23e^{-8}} = 43478 \text{ psi}$$

- Hoop stress at outside and inside diameters.

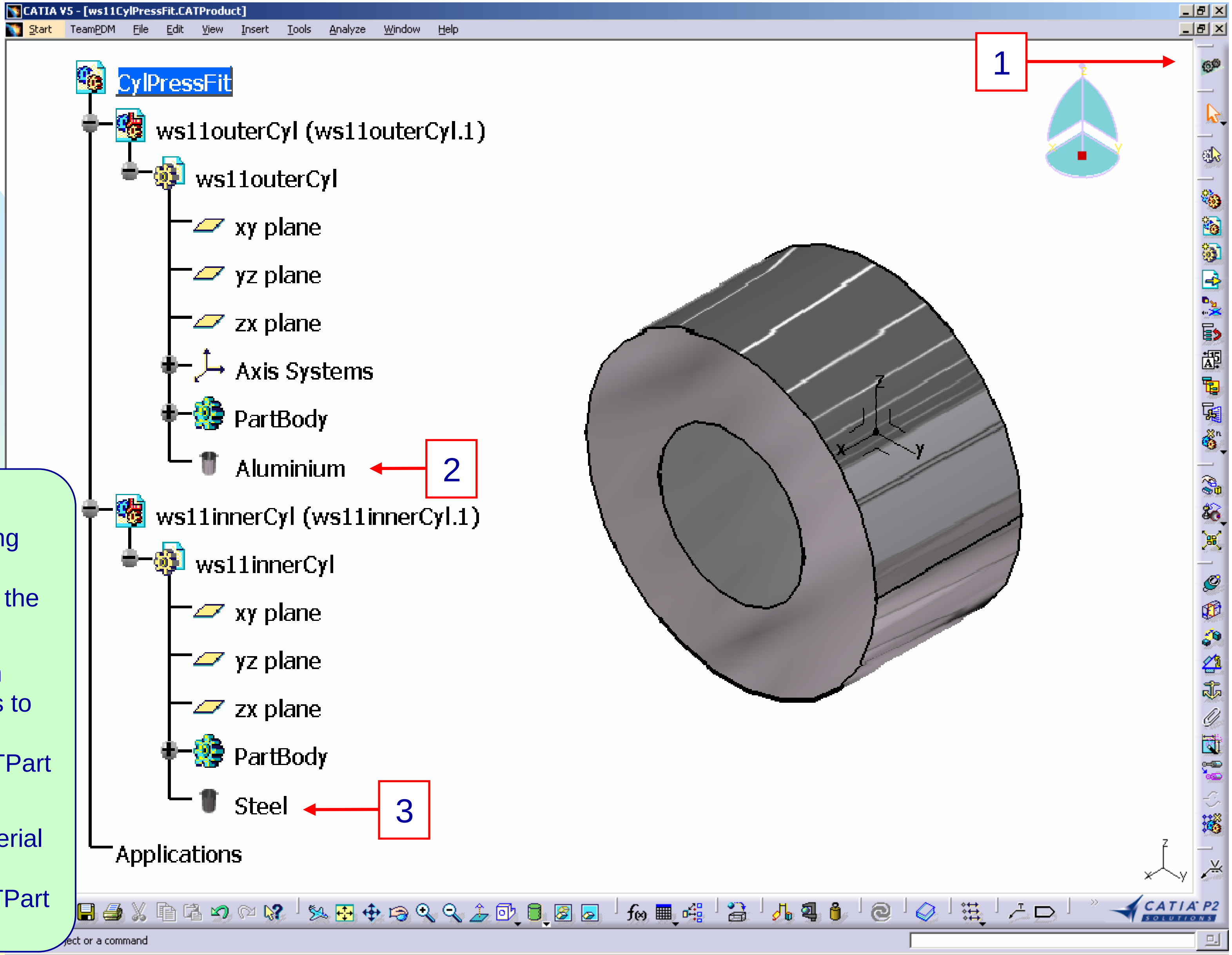
$$\sigma_{hoop} = p \cdot \frac{(r^2_{\text{outer cylinder}} + r^2_{\text{inner cylinder}})}{(r^2_{\text{outer cylinder}} - r^2_{\text{inner cylinder}})}$$

$$\sigma_{\text{hoop}} = 44782psi \cdot \frac{(2.0^2_{\text{outer cyl}} + 1.0^2_{\text{inner cyl}})}{(2.0^2_{\text{outer cyl}} - 1.0^2_{\text{inner cyl}})} = 43478 \cdot 1.667 = 72478 \text{ psi}$$

Suggested Exercise Steps

1. Open a …CATProduct, and specify materials.
2. Create assembly constraints.
3. Create a new CATIA analysis document.
4. Apply analysis properties.
5. Mesh globally and locally.
6. Apply an isostatic restraint.
7. Compute the initial analysis.
8. Check global and local precision (animate deformation, adaptive boxes and extremas).
9. Visualize final results.
10. Save the analysis document.

Step 1. Open …CATProduct and specify material



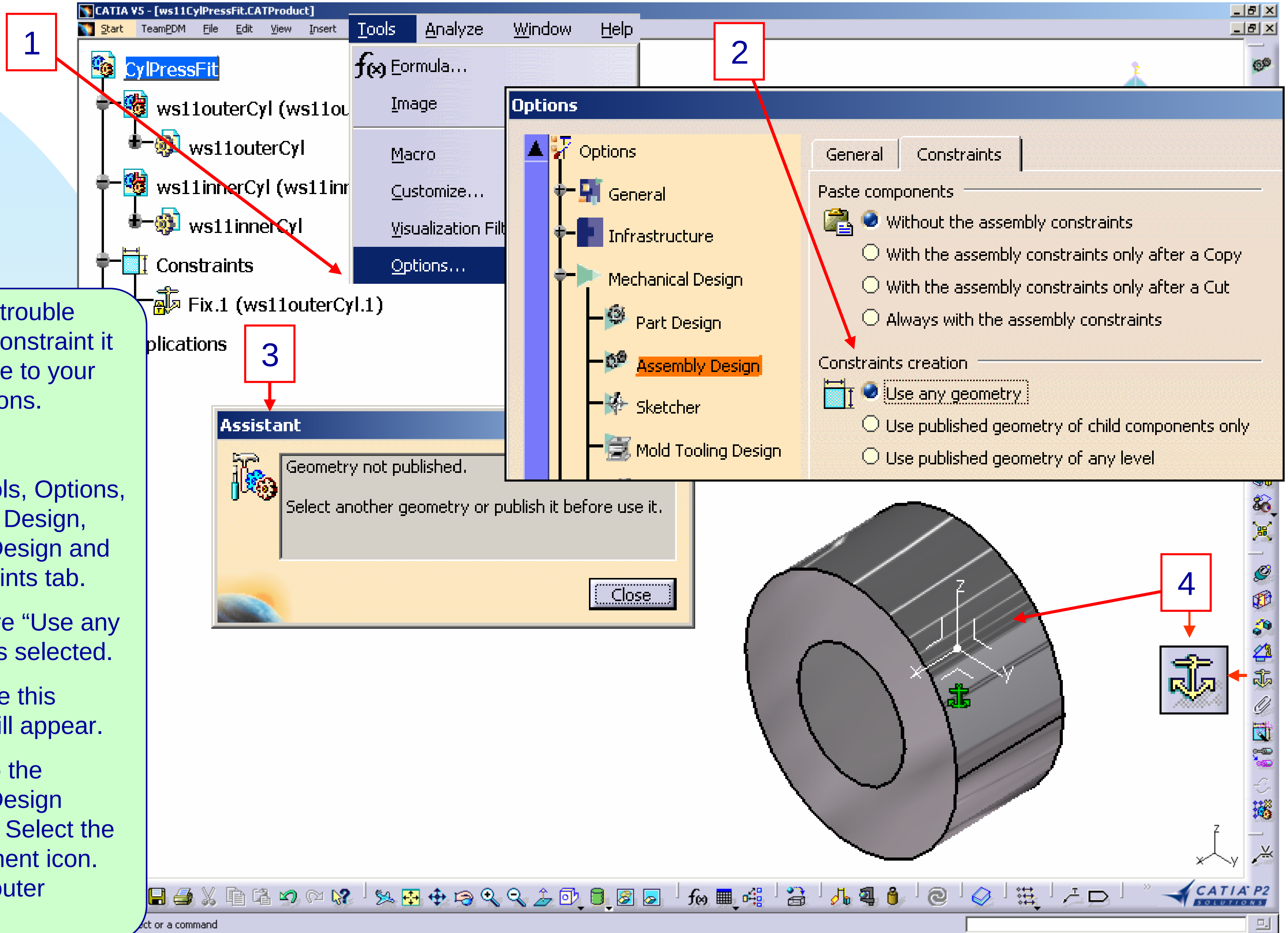
Steps:

1. Open the existing ws11CylPressFit .CATProduct from the training directory.

2. Apply aluminum material properties to the ws11outerCyl.CATPart as required.

3. Apply steel material properties to the ws11innerCyl.CATPart as required.

Step 2. Create assembly constraints



If you have trouble creating a constraint it could be due to your current options.

Steps:

1. Click Tools, Options, Mechanical Design, Assembly Design and the Constraints tab.

2. Make sure “Use any geometry” is selected.

3. Otherwise this Assistant will appear.

4. Return to the Assembly Design workbench. Select the Fix Component icon. Select the outer cylinder.

Step 2. Create assembly constraints

Using the Compass

Steps:

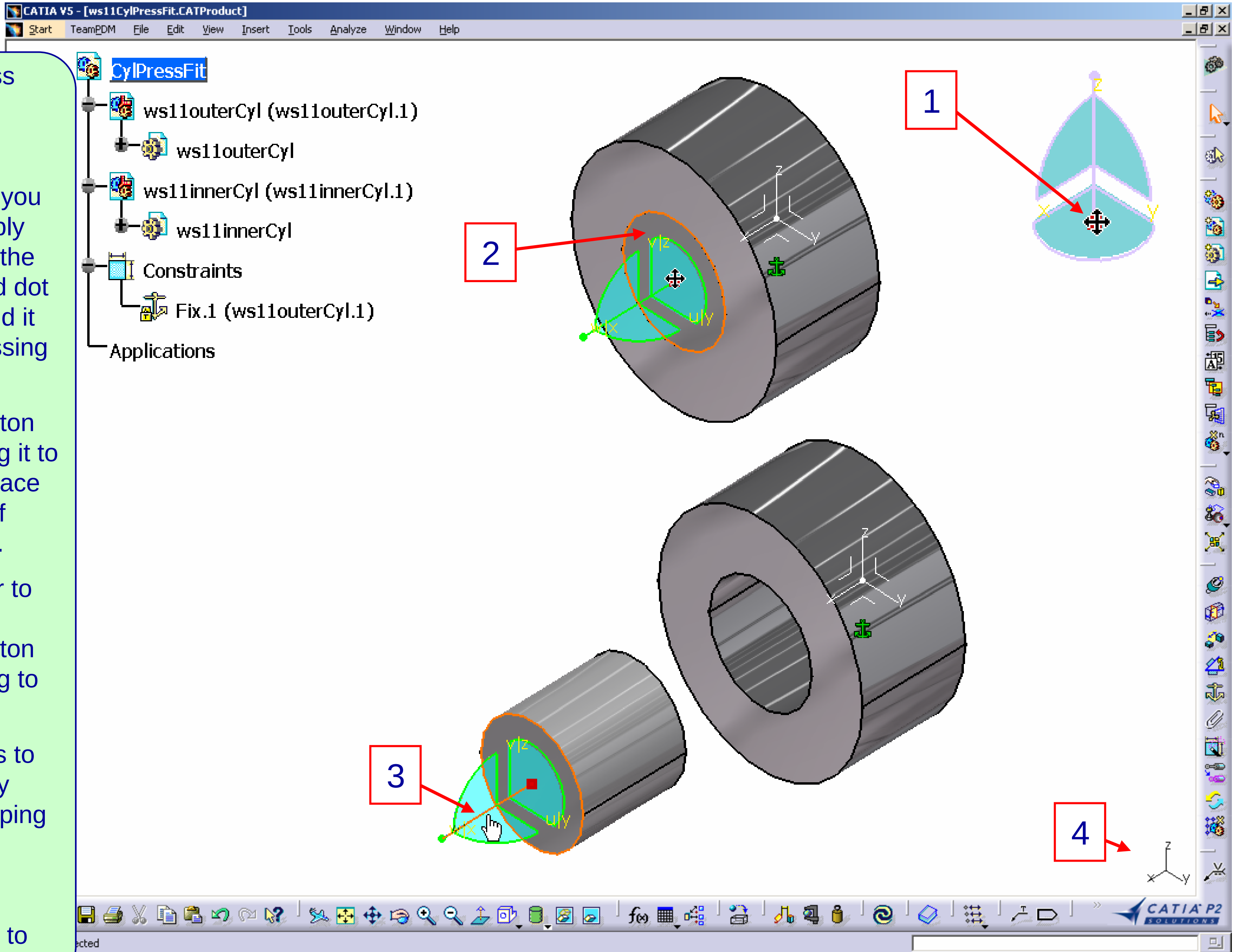
1. The compass is available to assist you in creating assembly constraints. Move the curser onto the red dot of the compass and it will change to crossing arrows.

2. Hold mouse button one down and drag it to the plugs outside face as shown, let go of mouse button one.

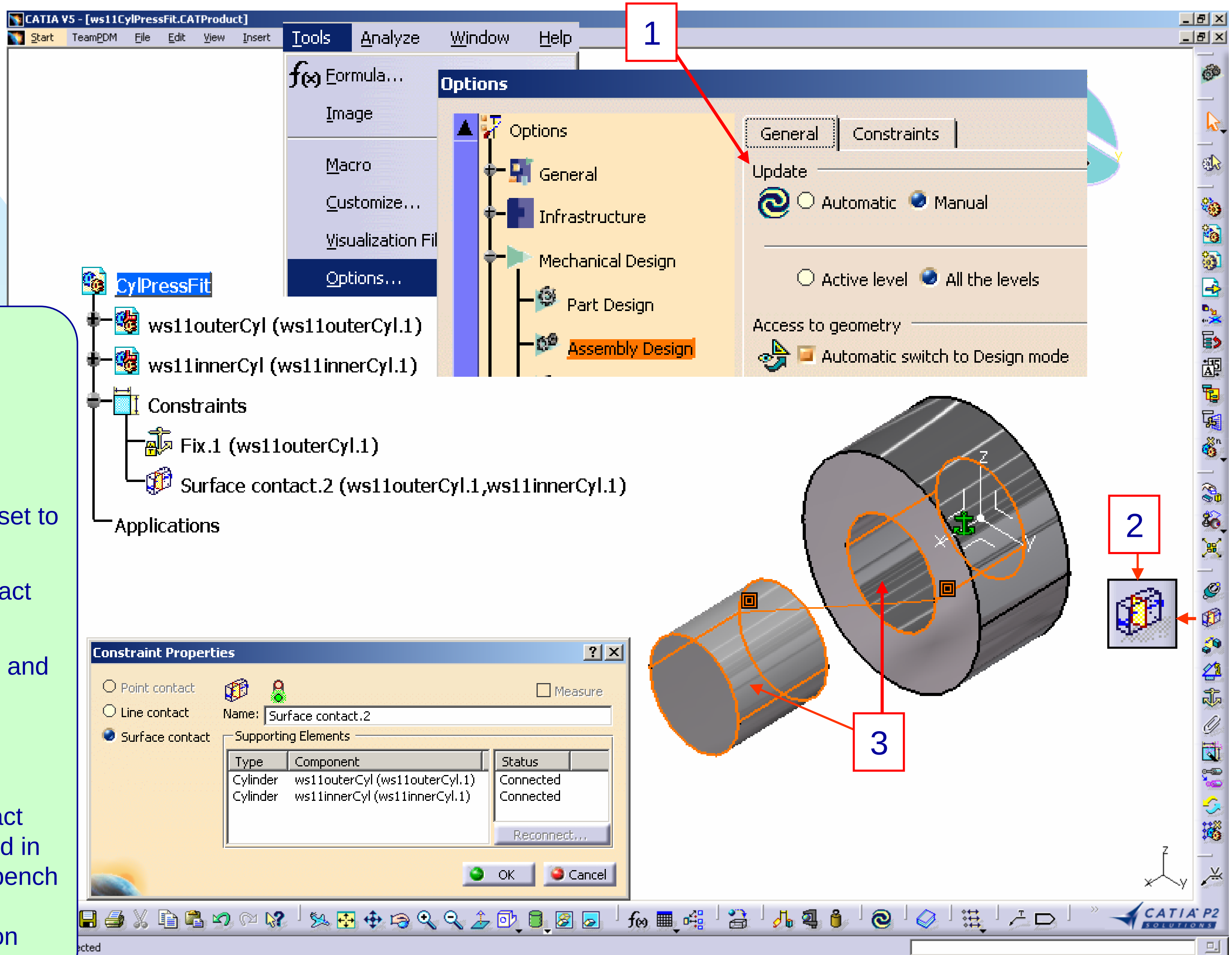
3. Move the curser to the Z vector of the compass, hold button one down and drag to position shown.

4. Return compass to it's original state by dragging and dropping it on the view axis.

We are now ready to apply constraints.



Step 2. Create assembly constraints



Apply a contact constraint.

Steps:

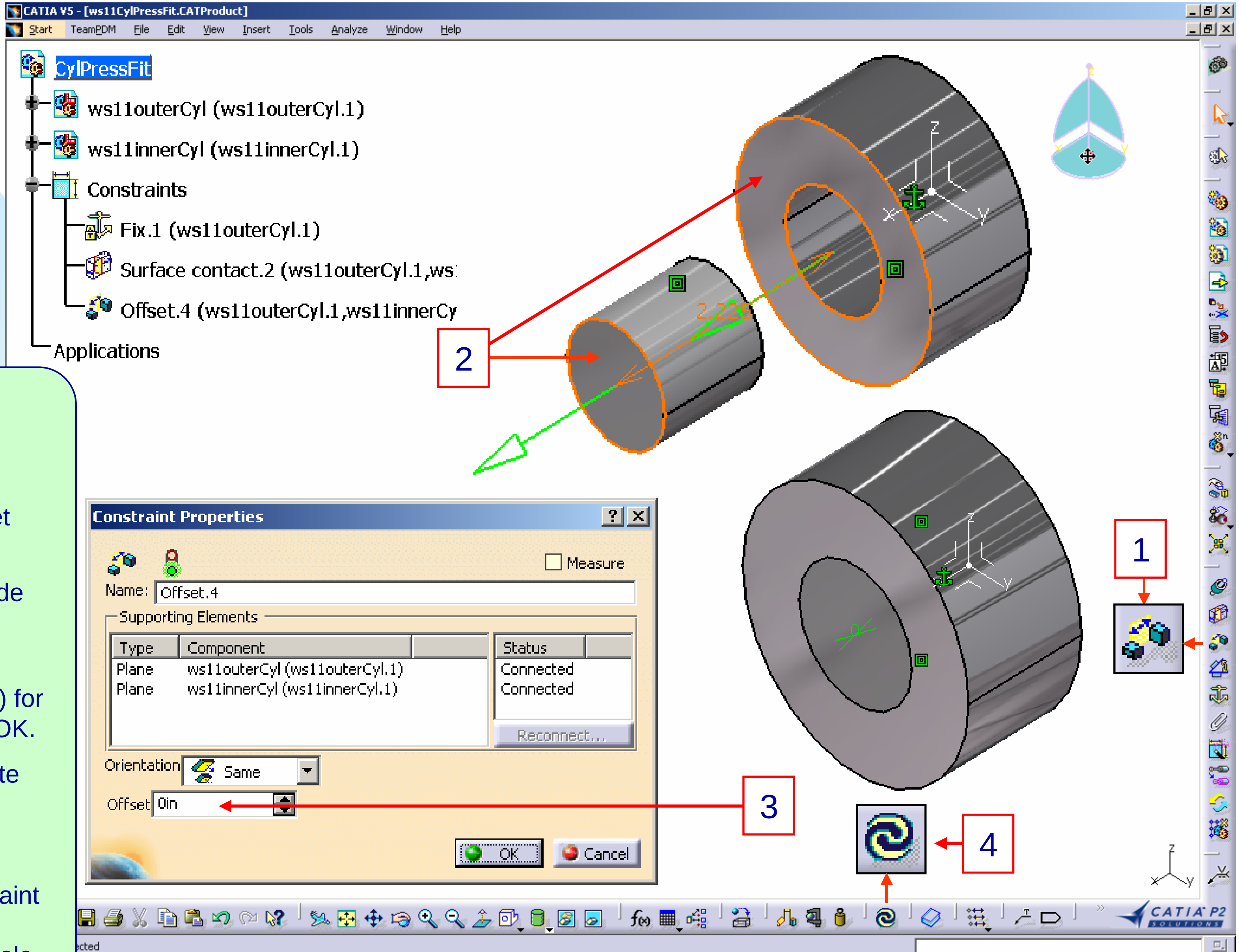
1. Make sure the Assembly Design general update is set to Manual.

2. Select the Contact Constraint icon.

3. Select the inner and outer surfaces as shown. Click OK.

This surface contact feature will be used in the analysis workbench when applying the Contact Connection property.

Step 2. Create assembly constraints



Apply an offset constraint.

Steps:

1. Select the Offset Constraint icon.

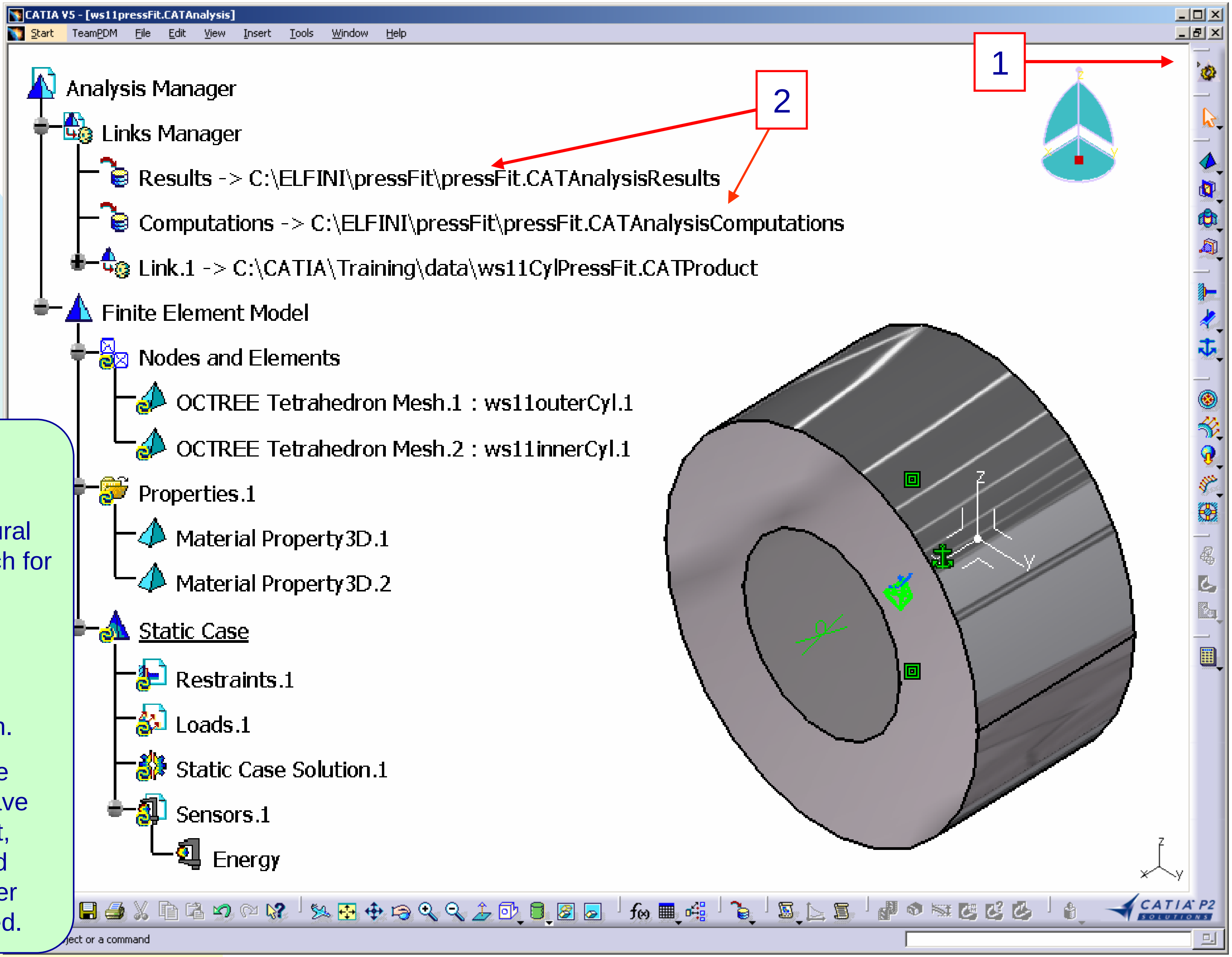
2. Select the outside coplanar faces as shown.

3. Key in 0in (zero) for the Offset, select OK.

4. Select the update icon.

This Offset Constraint keeps the plug positioned in the hole.

Step 3. Create a new CATIA analysis document



Steps:

1. Launch the Generative Structural Analysis workbench for a Static Analysis.

2. Specify the Computations and Results storage locations as shown.

Click File and Save Management to save this new document, confirm names and locations of all other documents involved.

MSC SOFTWARE

Step 4. Apply analysis properties

Simulate a contact connection.

Steps:

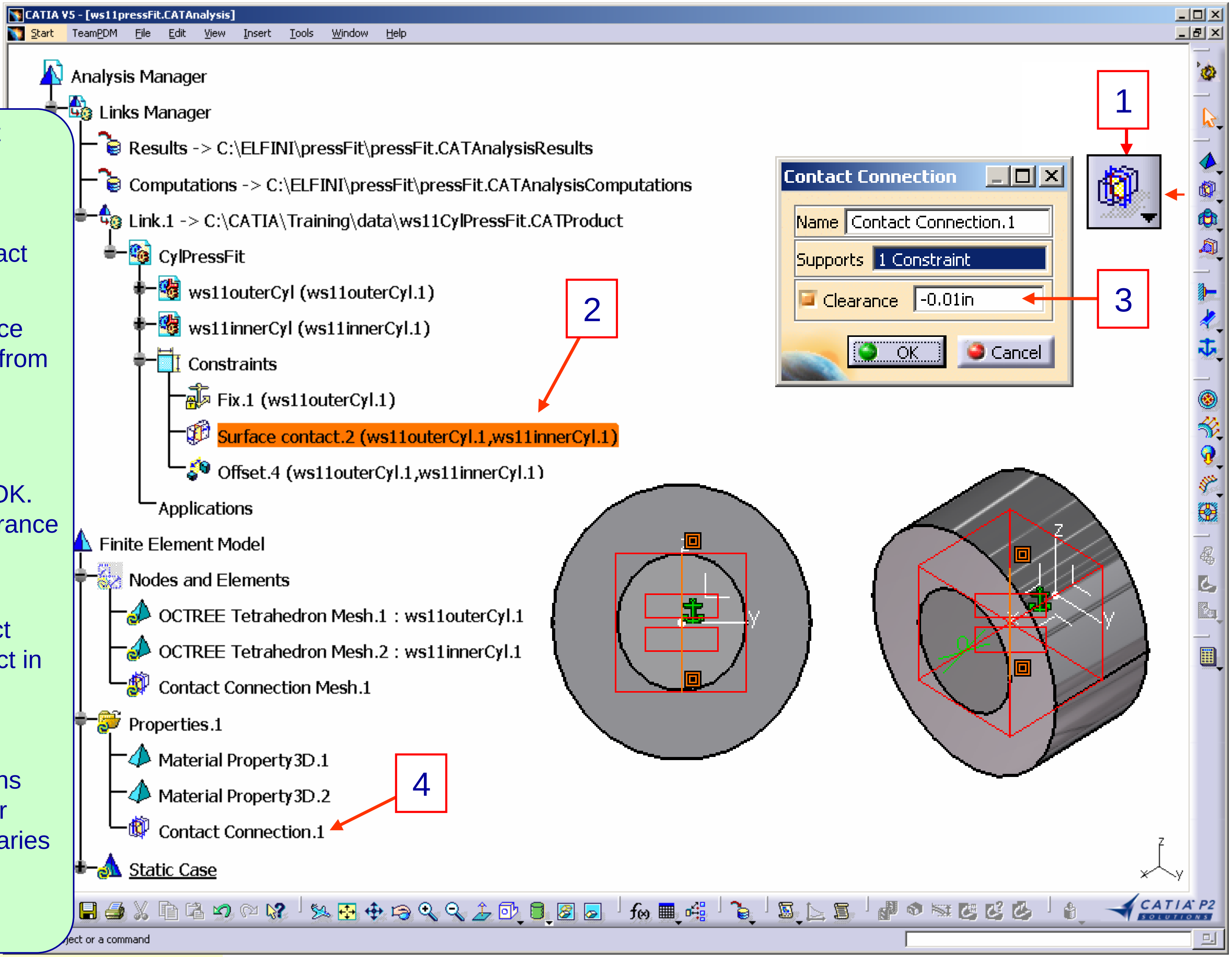
1. Select the Contact Connection icon.

2. Select the surface contact constraint from the features tree.

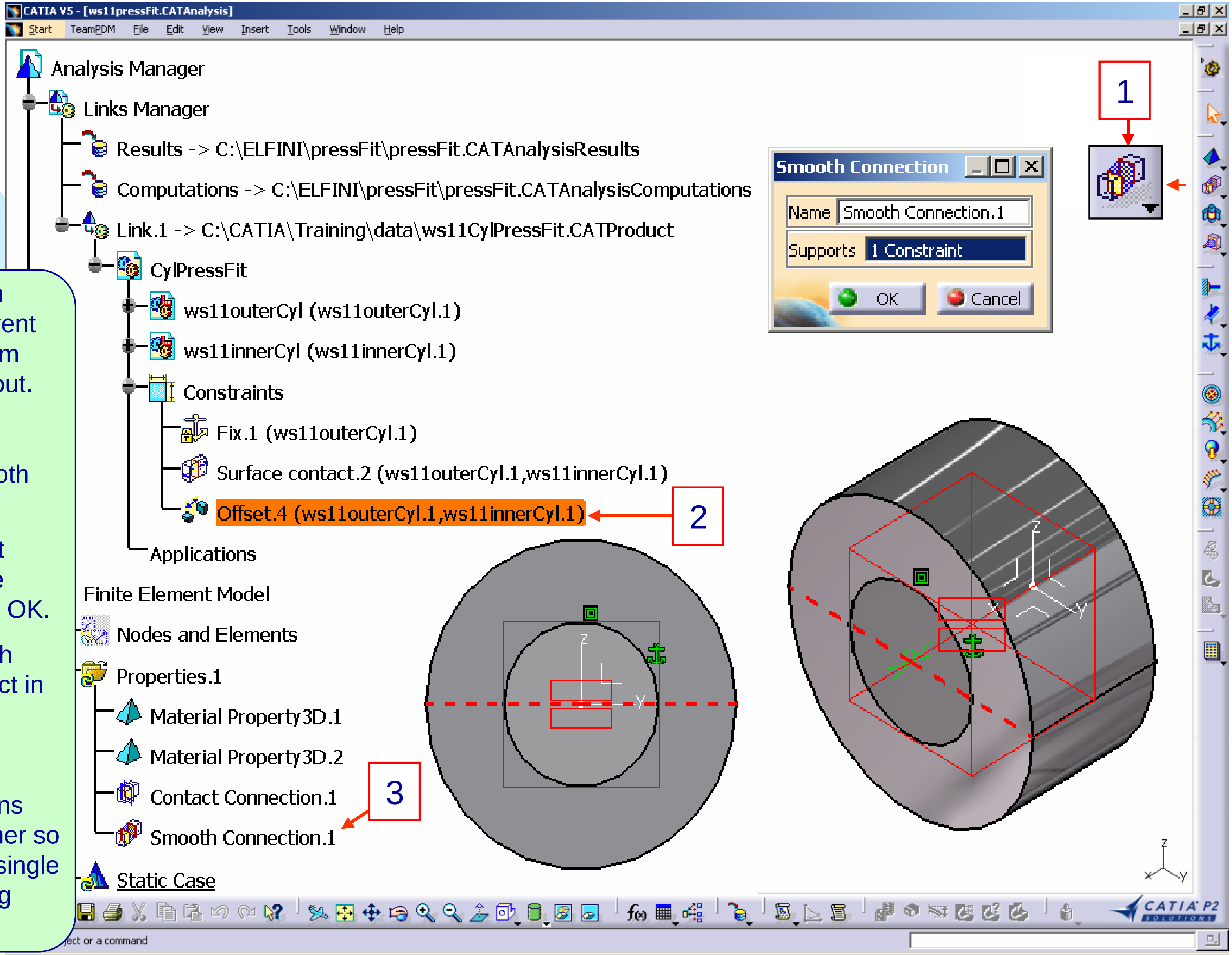
3. Key in negative 0.010in for the clearance, select OK. This negative clearance simulates an interference fit.

4. Note the Contact Connection.1 object in the features tree.

Contact connections define clearance or penetration boundaries between parts but otherwise move arbitrarily.



Step 4. Apply analysis properties



Simulate a smooth connection to prevent the center plug from moving or sliding out.

Steps:

1. Select the Smooth Connection icon.

2. Select the offset constraint from the features tree, click OK.

3. Note the Smooth Connection.1 object in the features tree.

Smooth connections fasten parts together so they behave as a single body while allowing deformation.

Step 5. Mesh globally and locally

The compass is very handy when specifying your mesh on an assembly of parts.

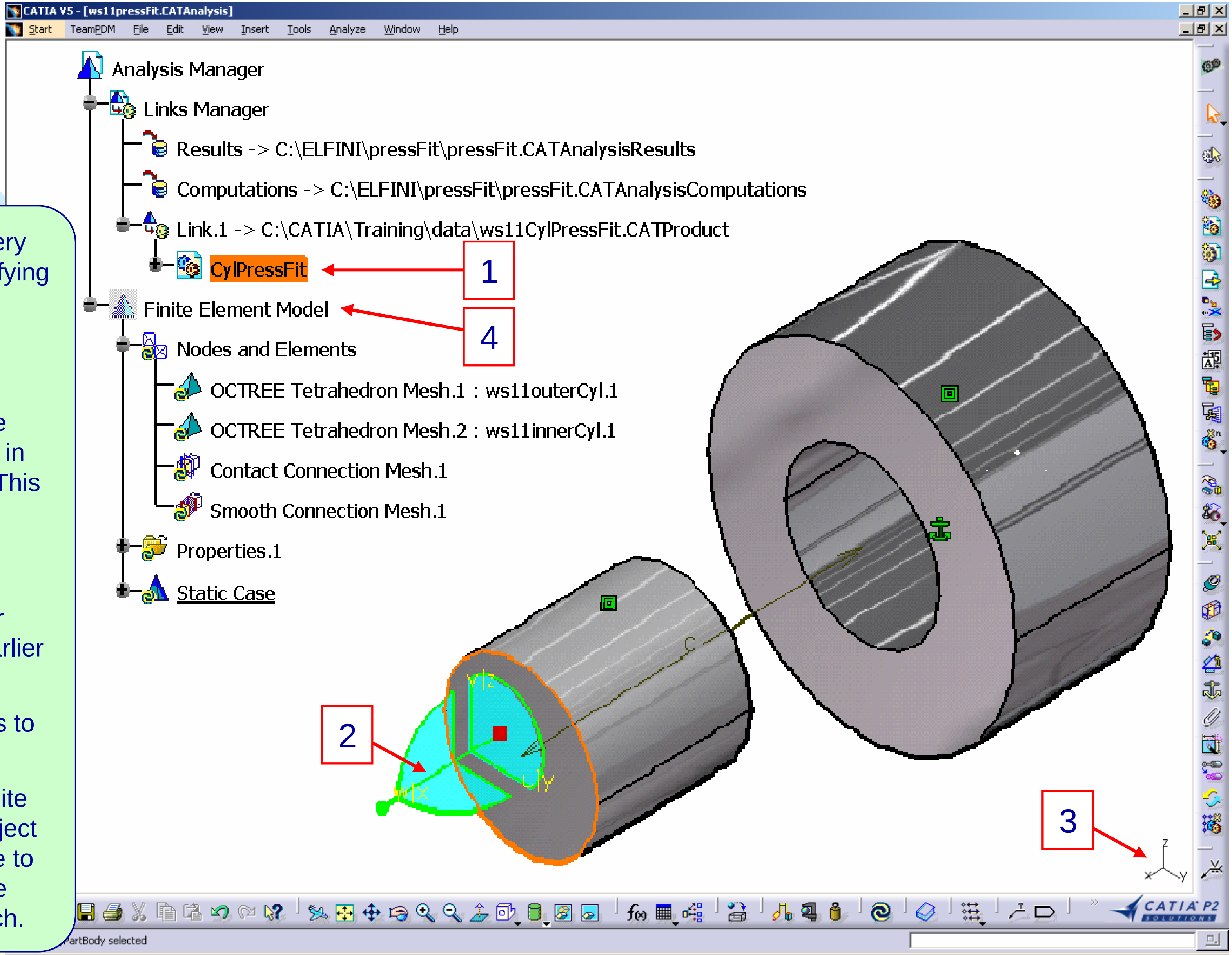
Steps:

1. Double Click the CylPressFit object in the features tree. This takes you into the Assembly Design workbench.

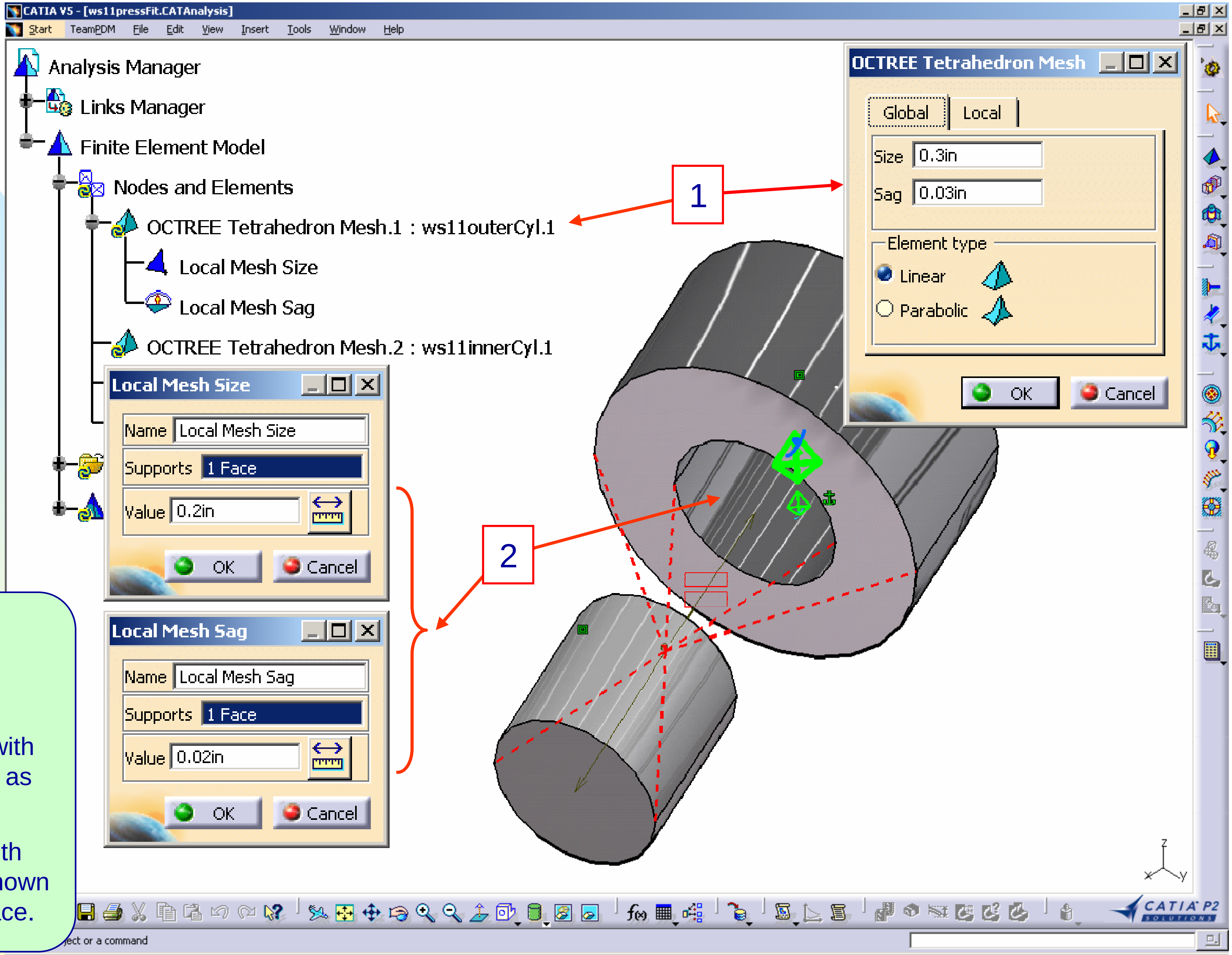
2. Move the center plug like we did earlier as shown.

3. Return compass to it's original state.

4. Double click Finite Element Model object in the features tree to take us back to the Analysis workbench.



Step 5. Mesh globally and locally



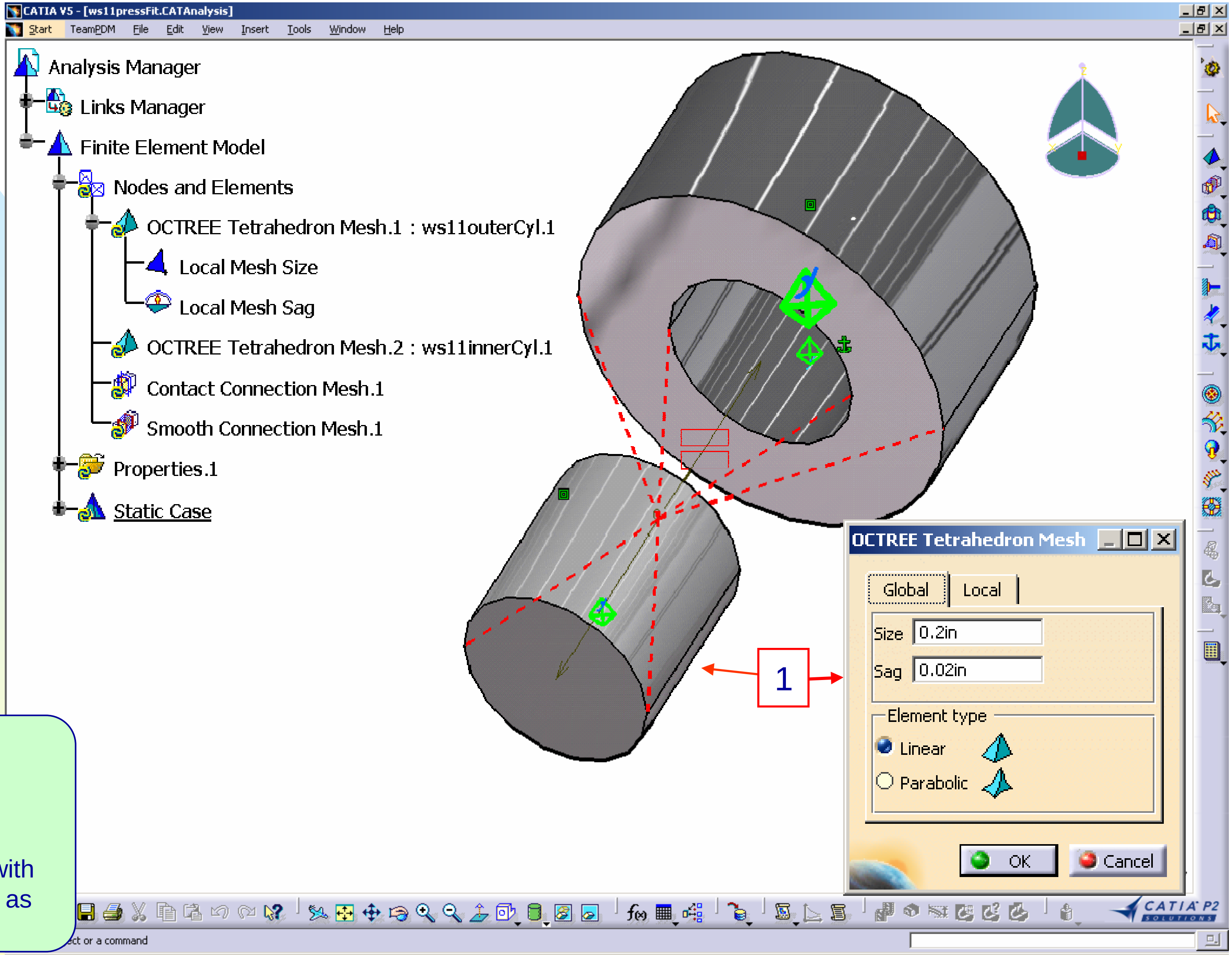
Mesh the outer cylinder.

Steps:

1. Globally mesh with size, sag and type as shown.

2. Locally mesh with size and sag as shown on the inside surface.

Step 5. Mesh globally and locally



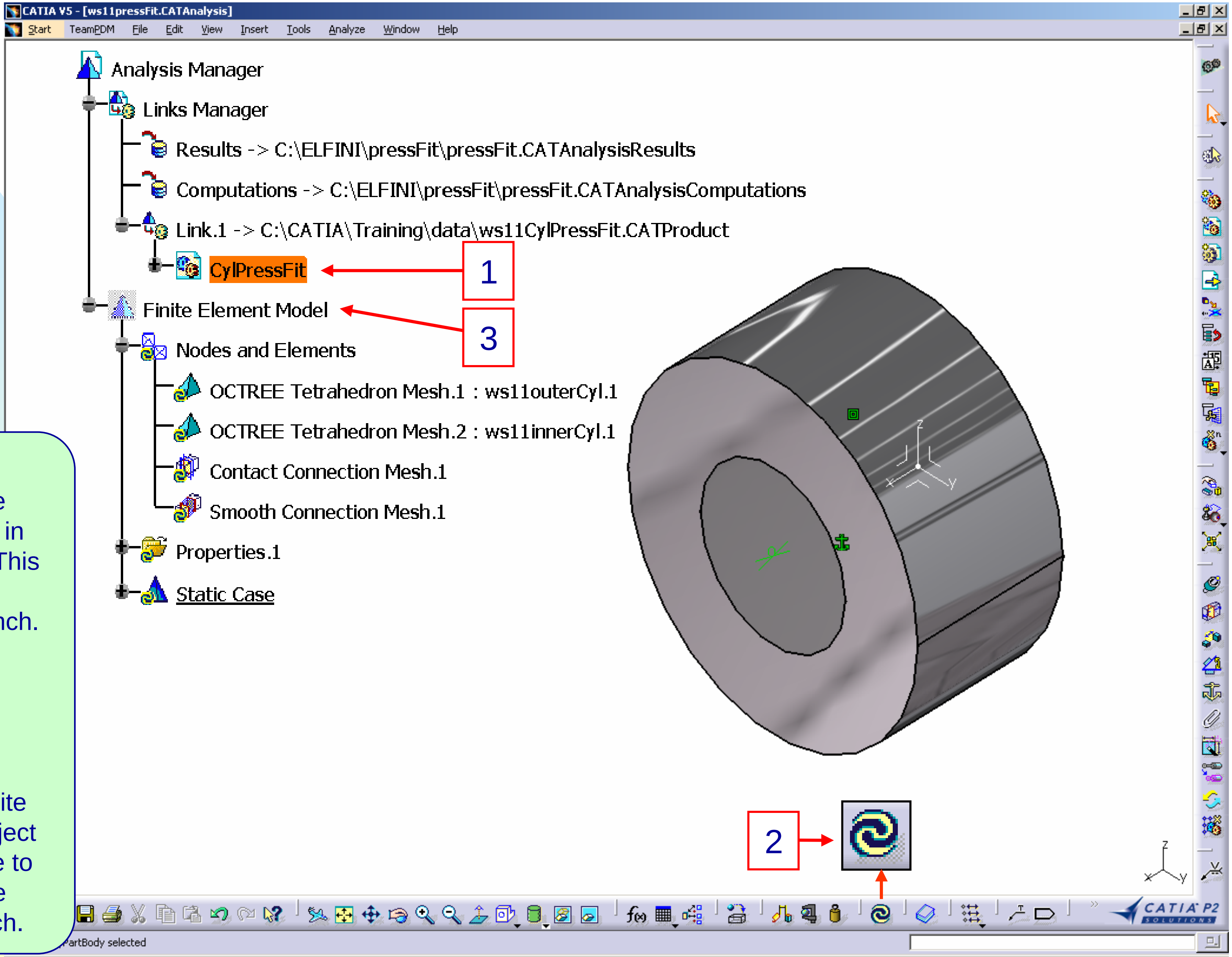
Mesh the inner cylinder.

Steps:

1. Globally mesh with size, sag and type as shown.

MSC SOFTWARE

Step 5. Mesh globally and locally



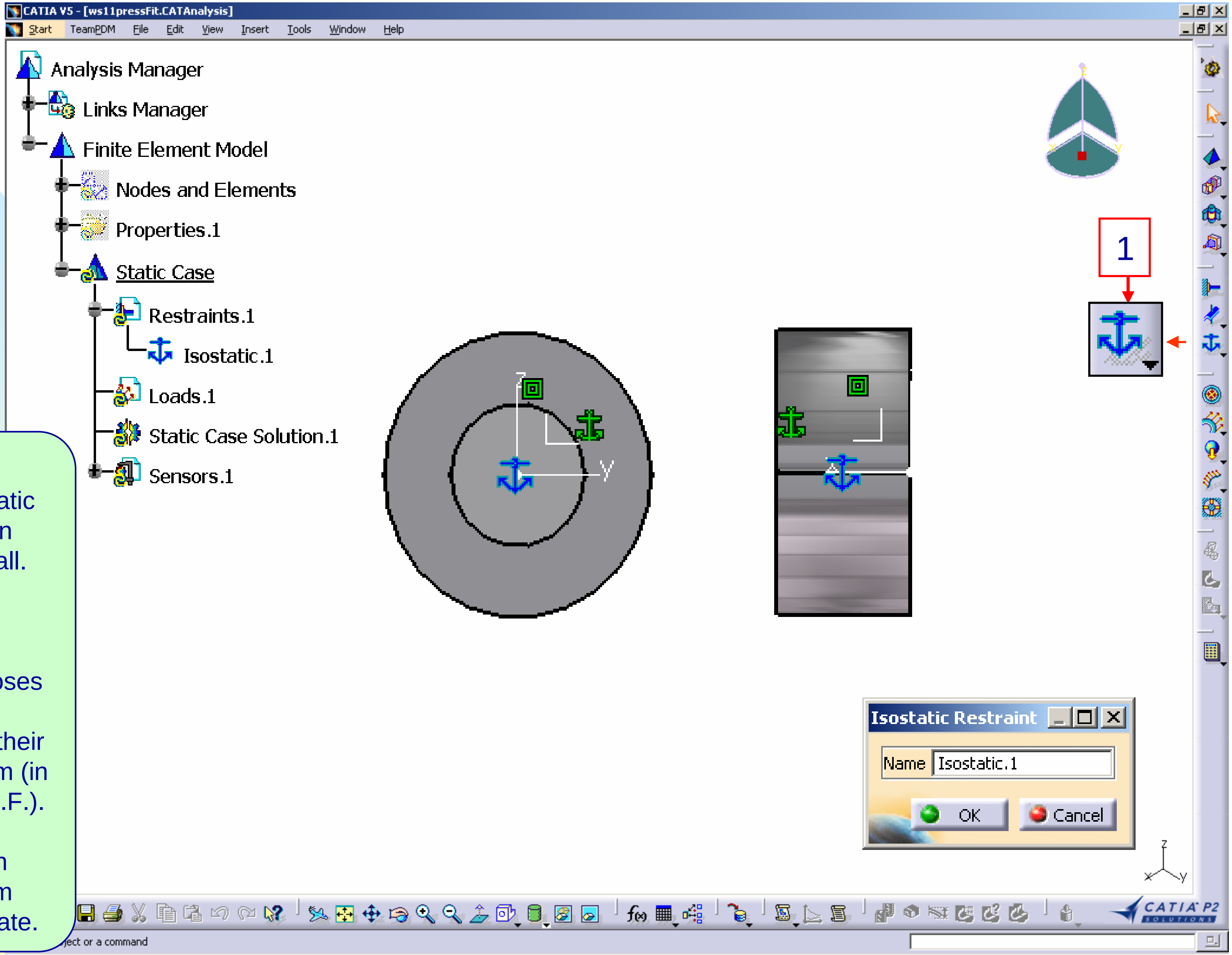
Steps:

1. Double Click the CylPressFit object in the features tree. This takes you into the Assembly workbench.

2. Update the assembly to bring everything back together.

3. Double click Finite Element Model object in the features tree to take us back to the Analysis workbench.

Step 6. Apply an isostatic restraint

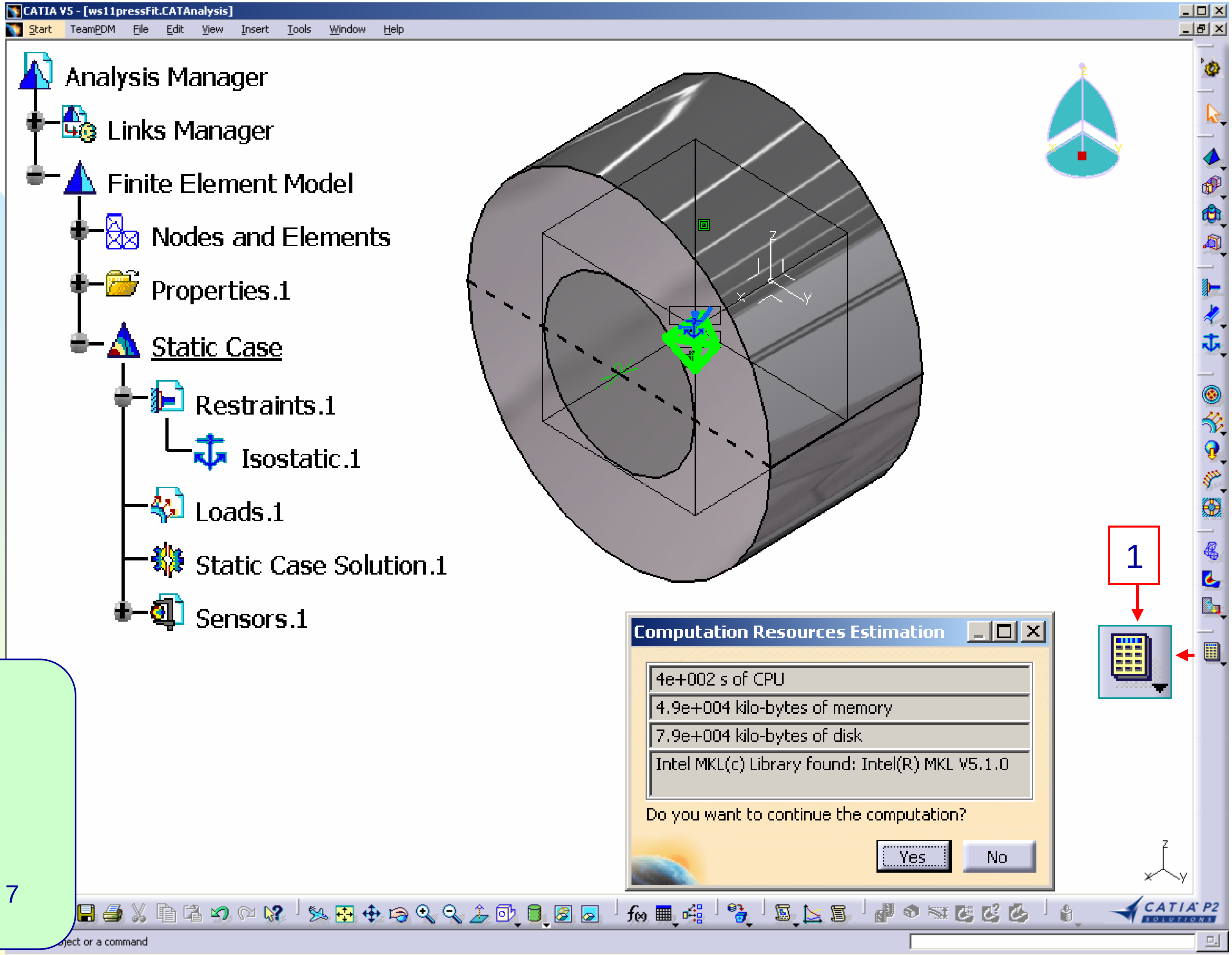


Steps:

1. Select the Isostatic Restraint icon, then select OK. That's all.

The program automatically chooses three points and restrains some of their degrees of freedom (in this case all 6 D.O.F.). The resulting boundary condition makes your system statically determinate.

Step 7. Compute the initial analysis



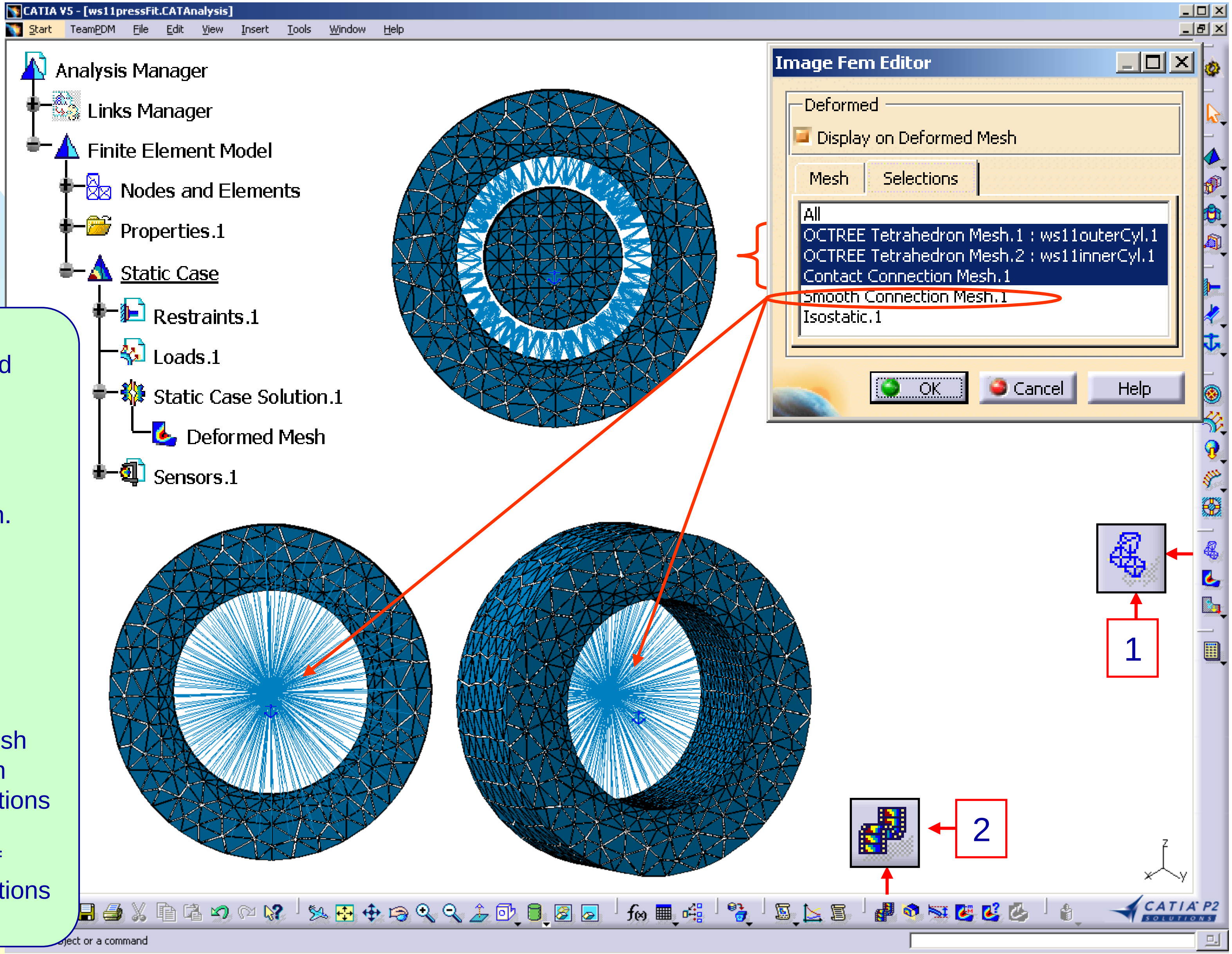
Save all before computing.

Steps:

1. Compute All.

Should take about 7 minutes.

Step 8. Check global and local precision



Visualize the **Deformation** and animate.

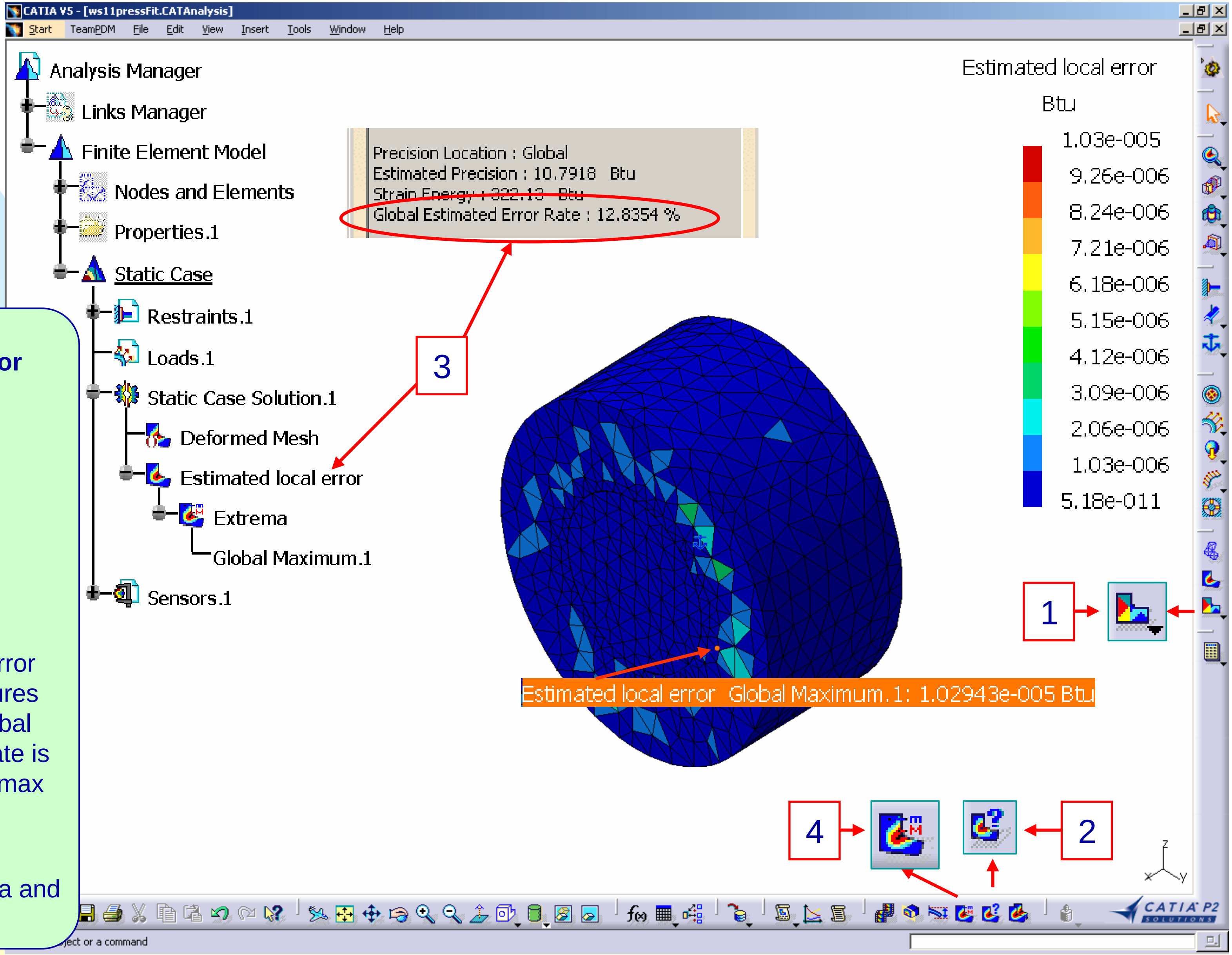
Steps:

1. Select the Deformation icon.

2. Select on the Animate icon to verify expected deformations.

Display each mesh separately and in various combinations to get a better understanding of what the connections create.

Step 8. Check global and local precision



Visualize the **computation error map.**

Steps:

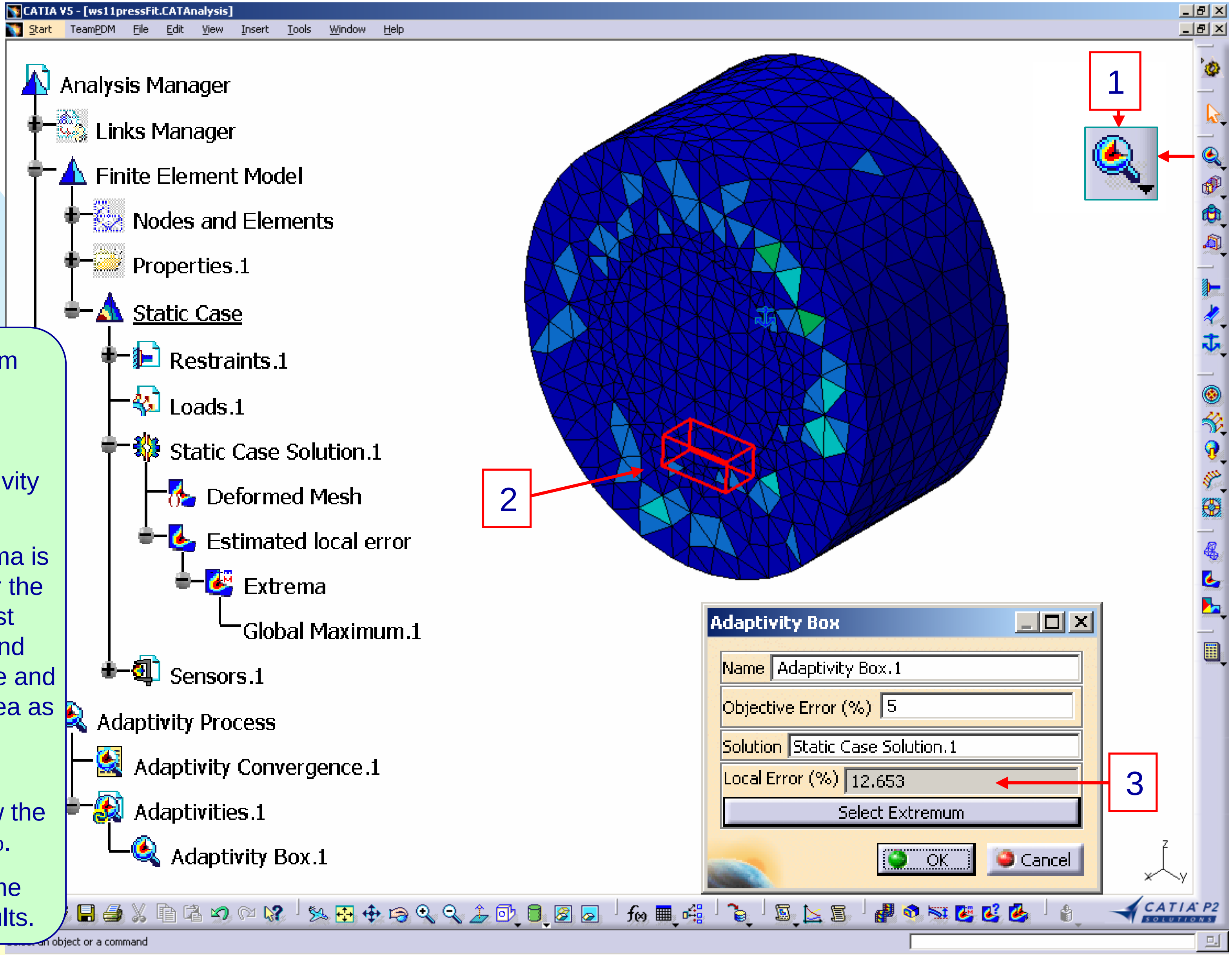
1. Select the Precision icon.

2. Select on the information icon.

3. Select the Estimated local error object in the features tree. Note the global estimated error rate is OK (recommend max 20%).

4. Search for maximum extrema and focus on it.

Step 8. Check global and local precision



Determine maximum local error %.

Steps:

1. Select the adaptivity box icon.

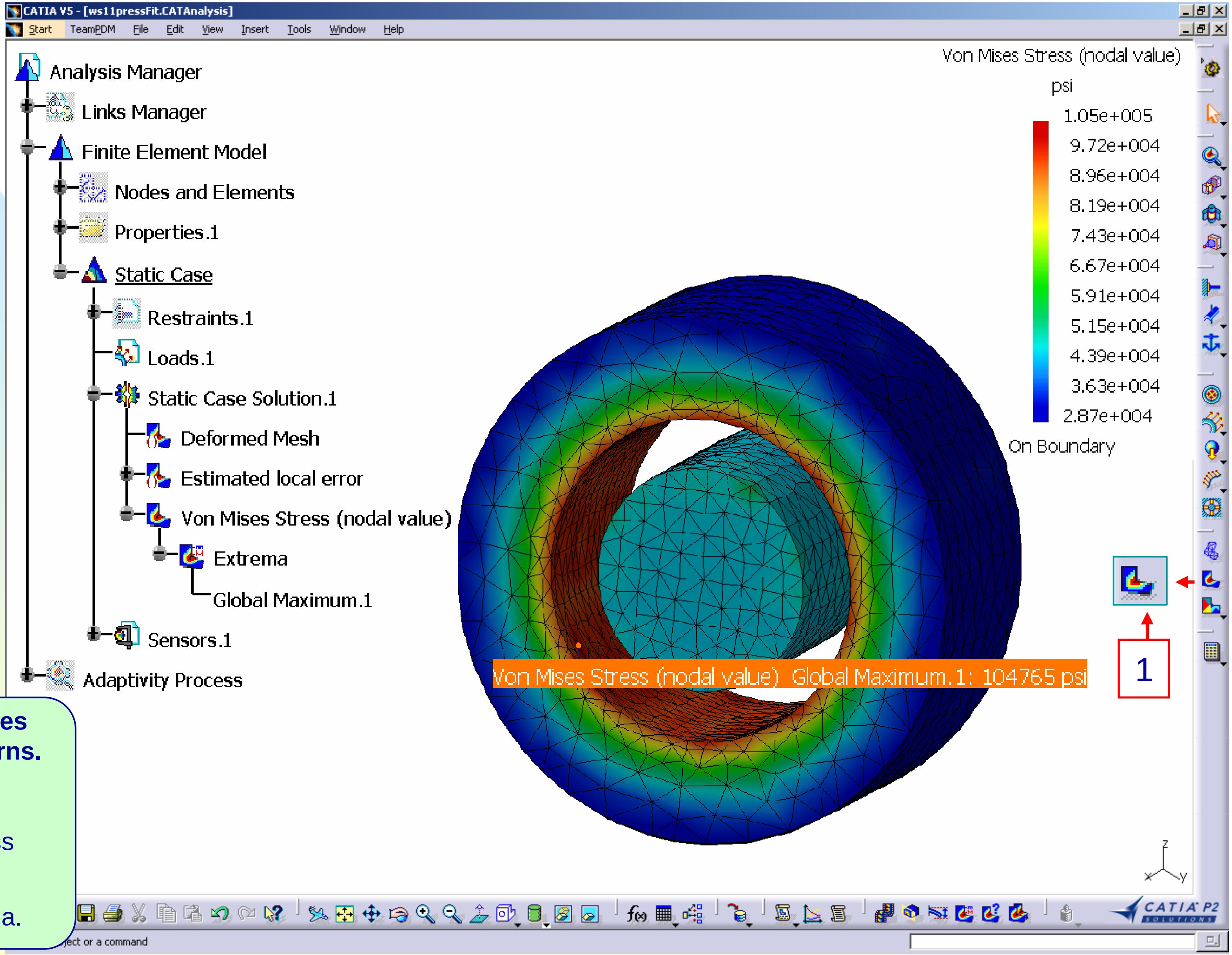
2. Since the extrema is not any where near the press fit location just use the compass and green dots to locate and size box around area as shown.

3. Local error is marginal, not below the recommended 10%.

In the interest of time let's use these results.

MSC SOFTWARE

Step 9. Visualize final results



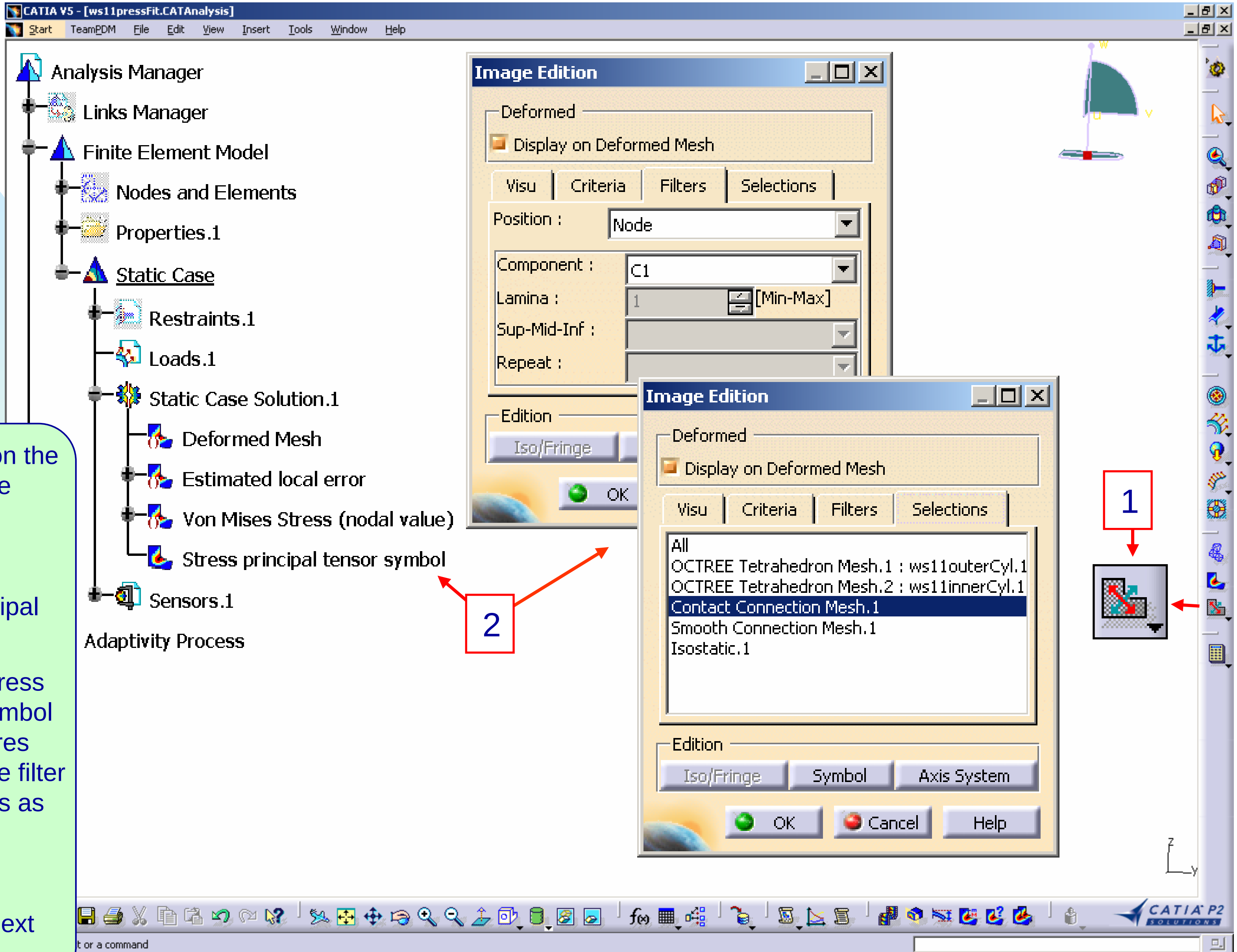
Visualize **Von Mises stress field patterns.**

Steps:

1. Select the Stress Von Mises icon.

2. Find the Extrema.

Step 9. Visualize final results



Find hoop stress on the inner surface of the outer cylinder.

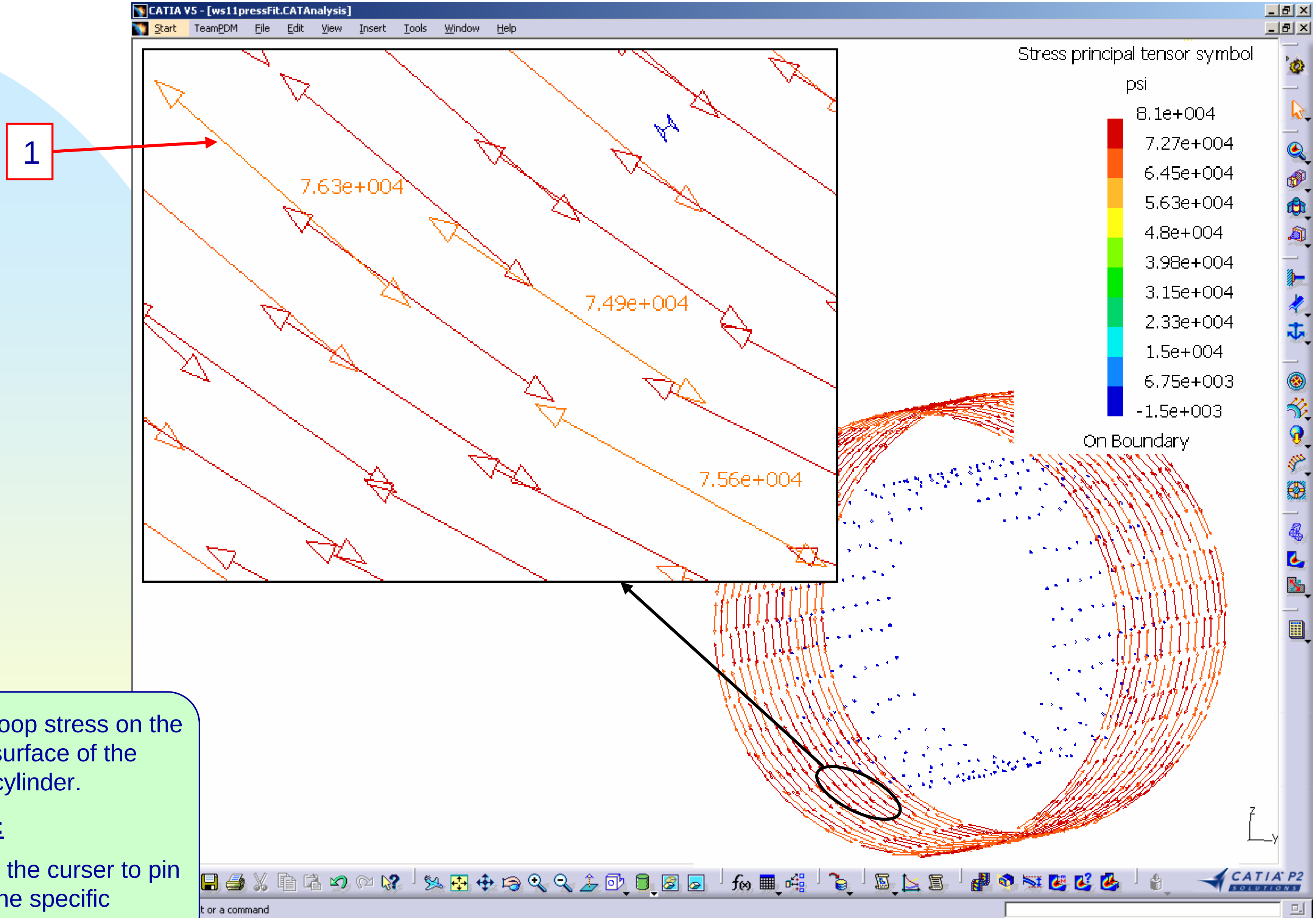
Steps:

1. Select the Principal stress icon.

2. Double click Stress principal tensor symbol object in the features tree and modify the filter and selections tabs as shown.

Image shown on next page.

Step 9. Visualize final results

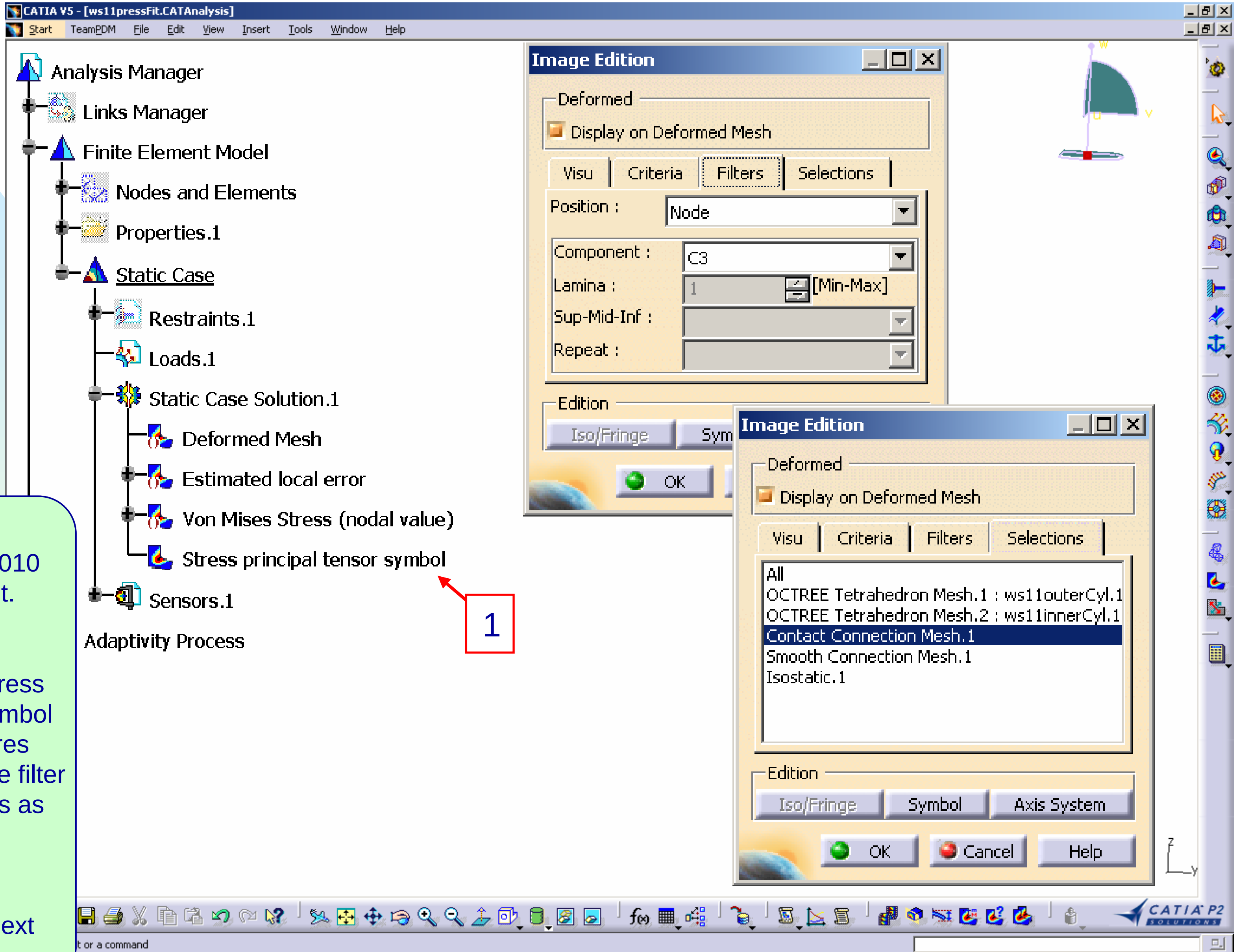


Find hoop stress on the inner surface of the outer cylinder.

Steps:

1. Use the curser to pin point the specific values.

Step 9. Visualize final results



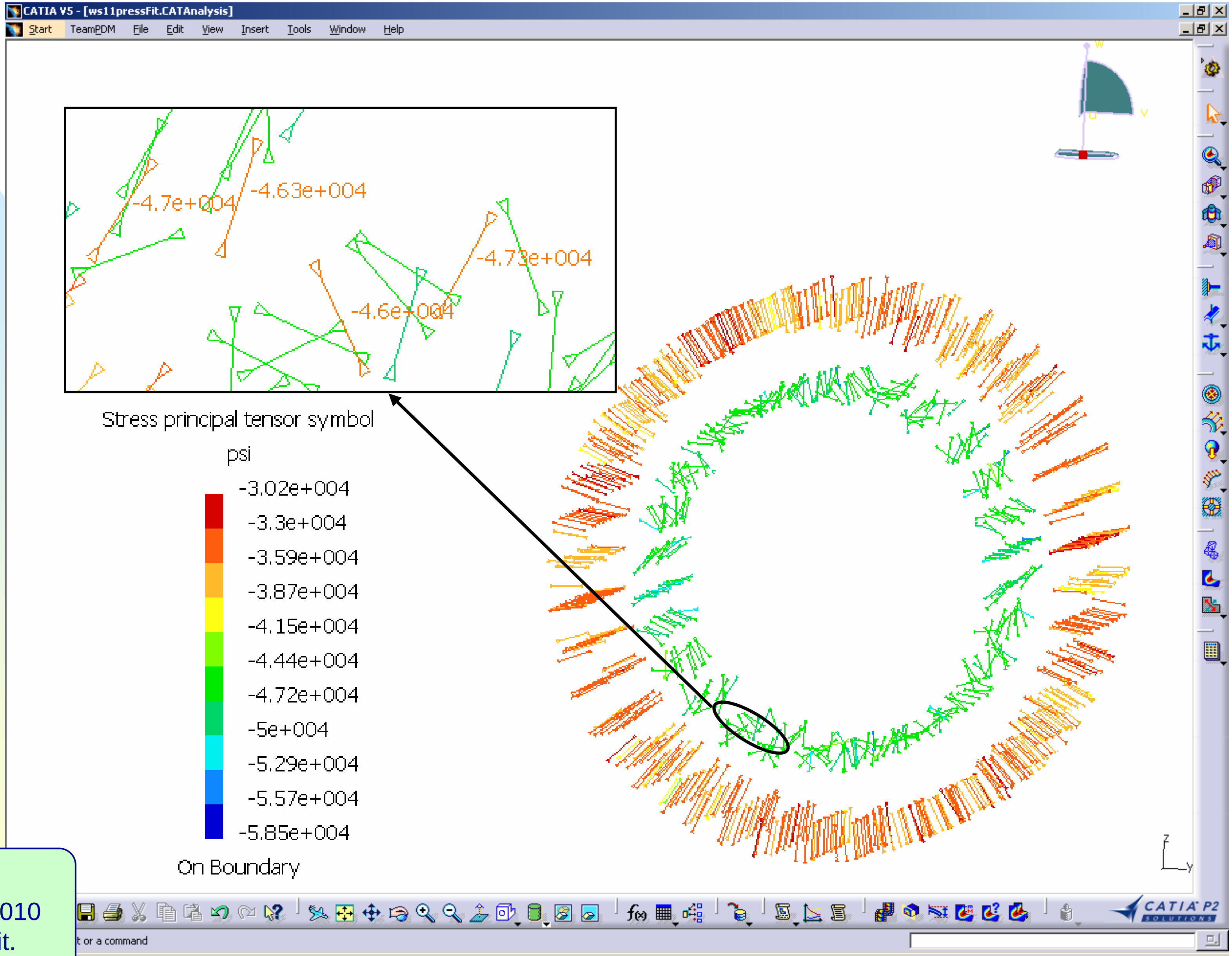
Verify the contact pressure due to 0.010 inch interference fit.

Steps:

1. Double click Stress principal tensor symbol object in the features tree and modify the filter and selections tabs as shown.

Image shown on next page.

Step 9. Visualize final results



Verify the contact pressure due to 0.010 inch interference fit.

Step 9. Visualize final results

Conclusions

- CATIA V5 GSA workbench is validated for a press fit scenario.

	Hand Calculations	various Linear Global Mesh
Global % Precision error **Local % Precision error**	**NA** **NA**	**12.8 %** **12.7 %**
Error Estimate	**NA**	**1.03e-5 Btu global**
Max Von Mises Stress	**NA**	**104,765 psi**
Hoop Stress	**72,478 psi**	**74,900 – 76,300 psi**
Pressure due to 0.010 interference	**43,478 psi**	**46,000 – 47,300 psi**

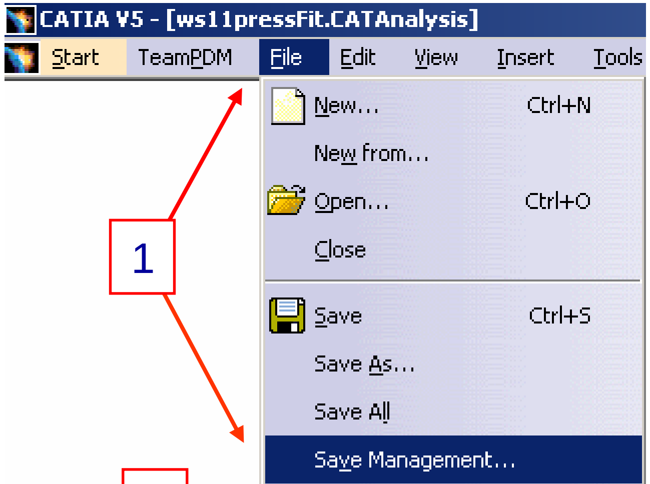
Step 10. Save the analysis document

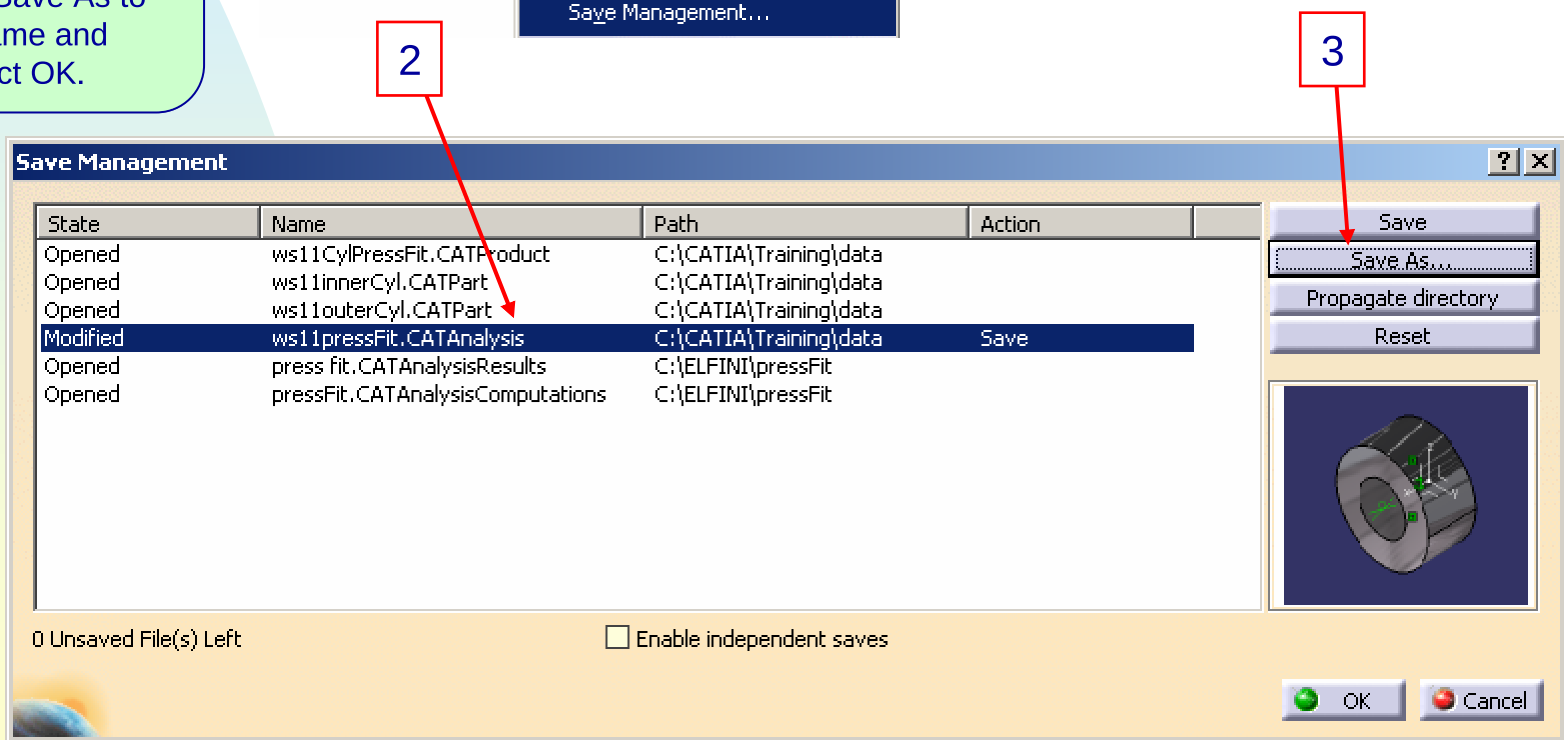
Steps:

1. Select Save Management from the File menu.

2. Highlight document you want to save.

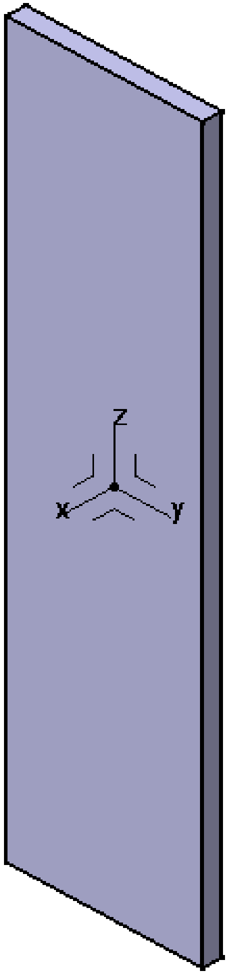
3. Select Save As to specify name and path, select OK.





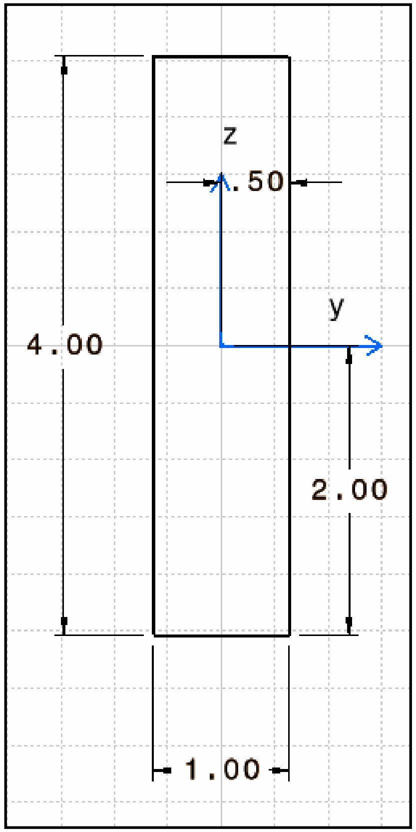
WORKSHOP 12

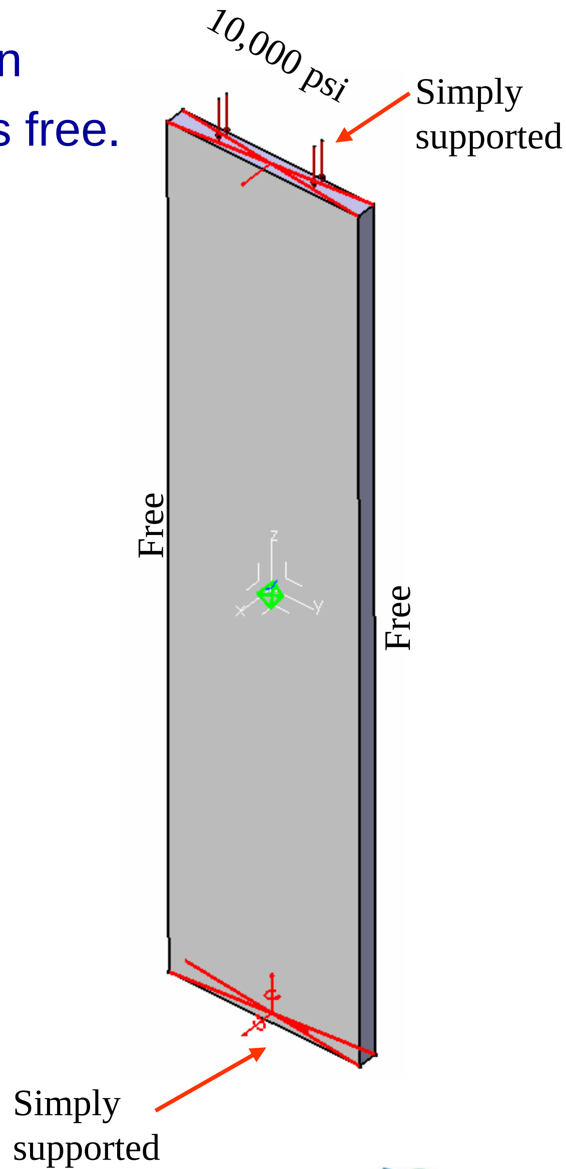
FLAT PLATE COLUMN BUCKLING



Problem Description

- Rectangular plate under uniform edge compression
- Two short edges simply supported, two long edges free.
- Find the critical load when buckling begins.





Material: Aluminum
Modulus of elasticity = 10.15e6 psi
Poisson Ratio = .346
Density = .098 lb_in3
Yield Strength = 13778 psi

Design requirements:

Thickness, t =	0.1 inch
Vertical Load, w =	100 lbs/in

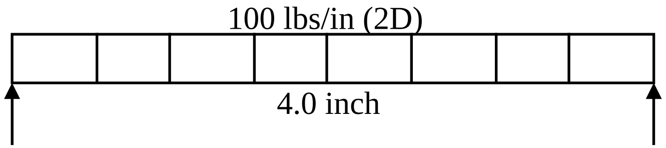
Hand calculations

- Critical load of a long slender column:

$$\text{Critical Load} = P_{cr} = \frac{\pi^2 \cdot E \cdot I_{min}}{L^2} \qquad I_{min} = \frac{b \cdot t^3}{12} = \frac{1.0 \cdot 0.1^3}{12} = 0.00008333\ \text{inch}^4$$

$$P_{cr} = \frac{\pi^2 \cdot 10.15e6\ \text{psi} \cdot 0.00008333\ \text{inch}^4}{4.0^2\ \text{inch}} = 522\ \text{lbs}$$

- Verify model by checking deflection using the standard formula for a simply supported beam at both ends with uniform load over the entire span using a pressure of 100 psi (3D).

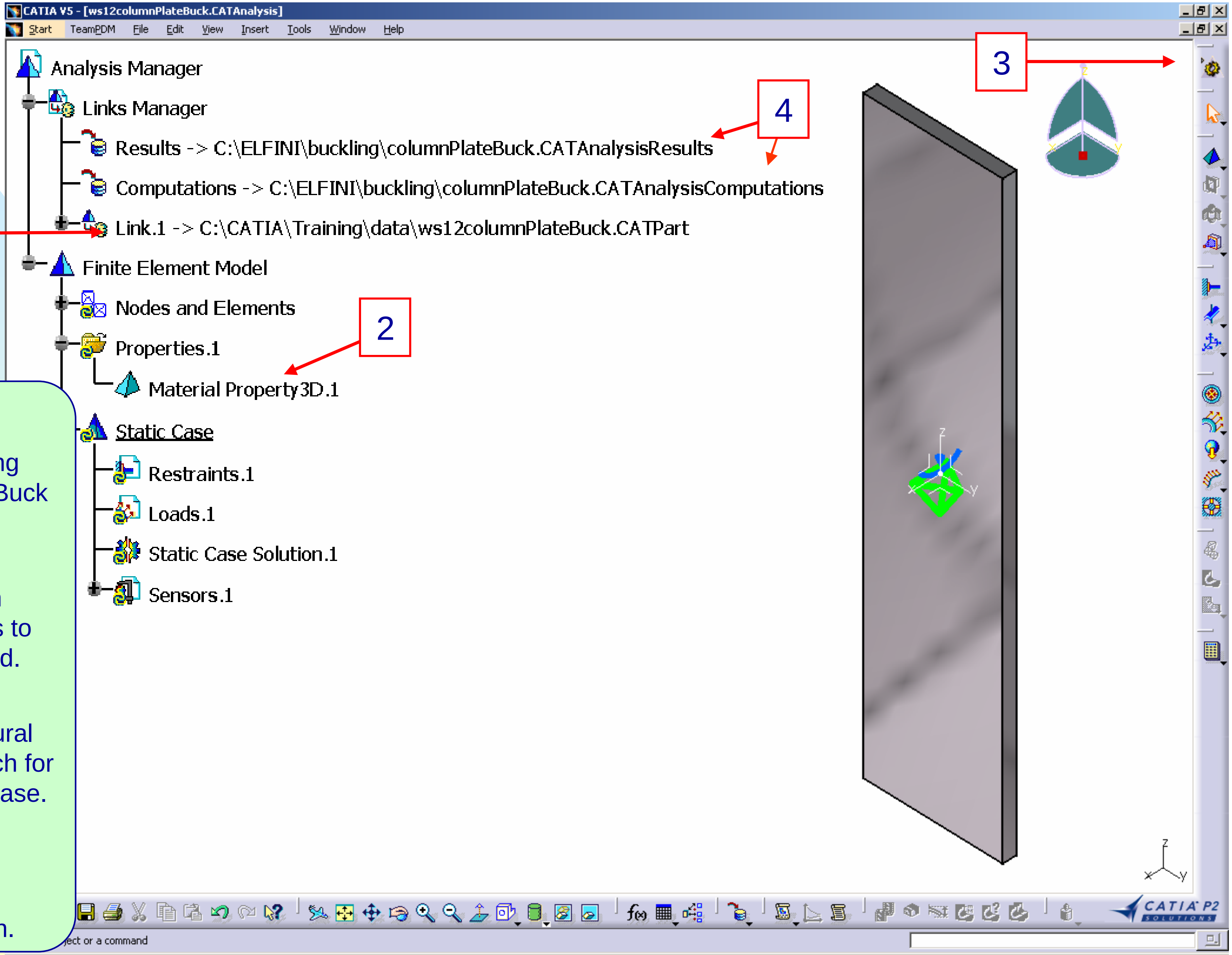


$$\text{Maximum deflection} = \delta = \frac{5 \cdot w \cdot L^4}{384 \cdot E \cdot I_{min}} = \frac{5 \cdot 100\ \text{lbs/inch} \cdot 4.0^4\ \text{inch}}{384 \cdot 10.15e6\ \text{psi} \cdot 0.00008333\ \text{inch}^4} = 0.394\ \text{inch}$$

Suggested Exercise Steps

1. Create a new CATIA analysis document (.CATAnalysis).
2. Mesh globally with parabolic elements.
3. Create virtual parts and apply advanced restraints (simply supported).
4. Apply a force.
5. Insert a Buckling Case.
6. Setup static and buckling parameters.
7. Compute all (the static and buckling analysis).
8. Check global and local precision (animate deformation, adaptive boxes and extremas).
9. Visualize final results.
10. Save the analysis document.

Step 1. Create a new CATIA analysis document



Steps:

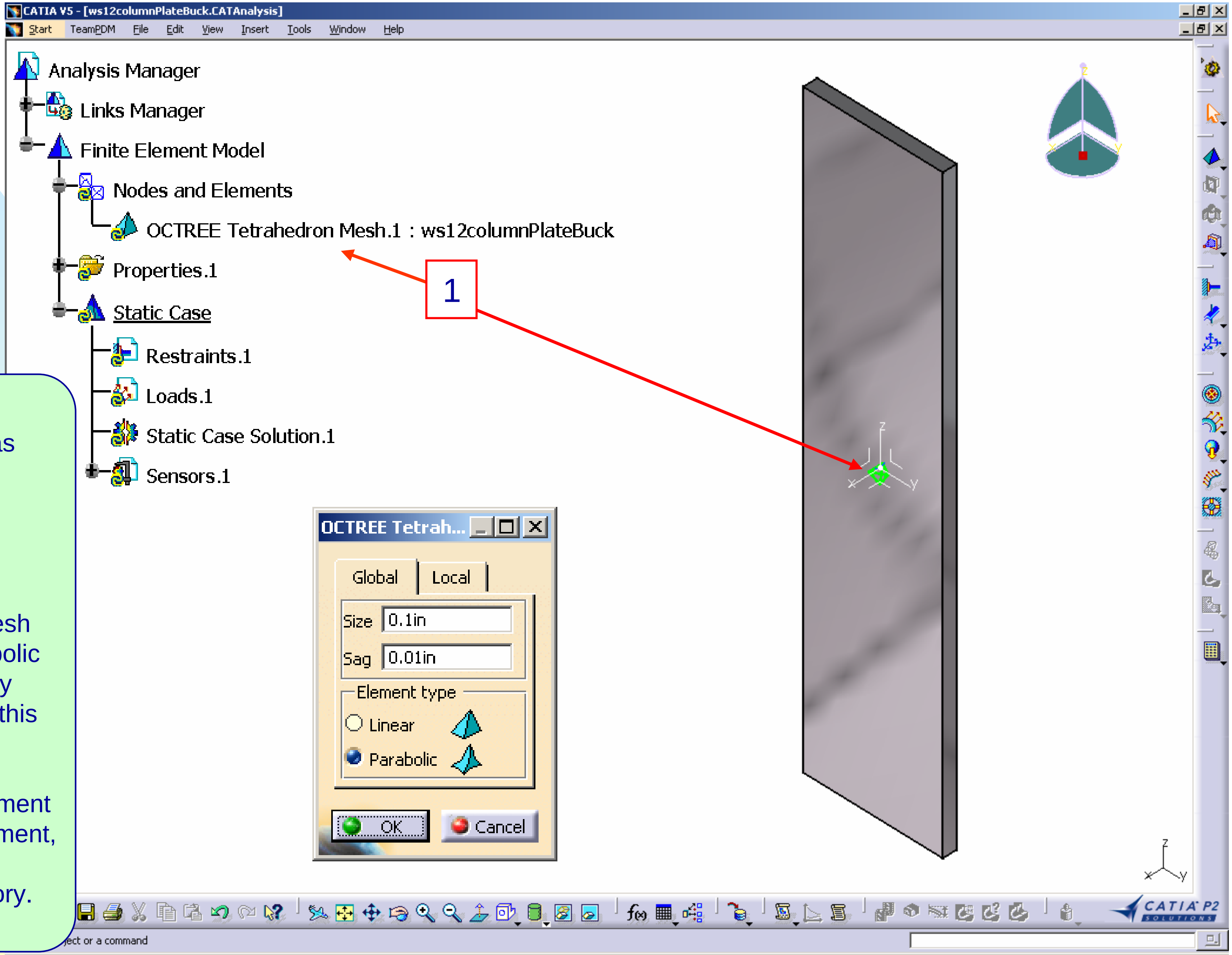
1. Open the existing ws12columnPlateBuck .CATPart from the training directory.

2. Apply aluminum material properties to the part as required.

3. Launch the Generative Structural Analysis workbench for a Static Analysis case.

4. Specify the Computations and Results storage locations as shown.

Step 2. Mesh globally with parabolic elements

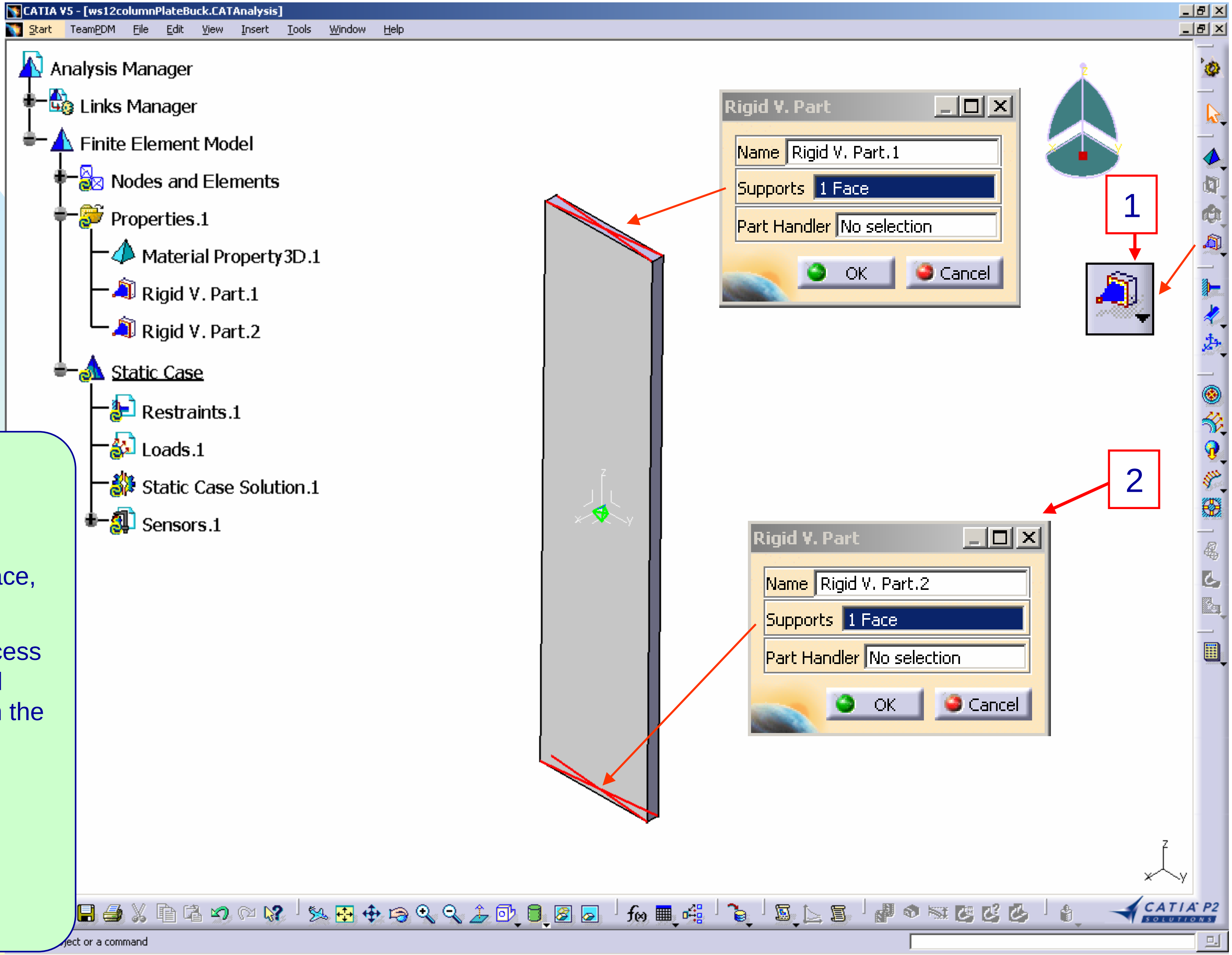


Steps:

1. Globally mesh as shown

Thin gauge sheet problems are very sensitive to the mesh parameters. Parabolic elements are highly recommended for this because they are formulated with a parabolic displacement field within the element, which agrees with basic bending theory.

Step 3. Create virtual parts



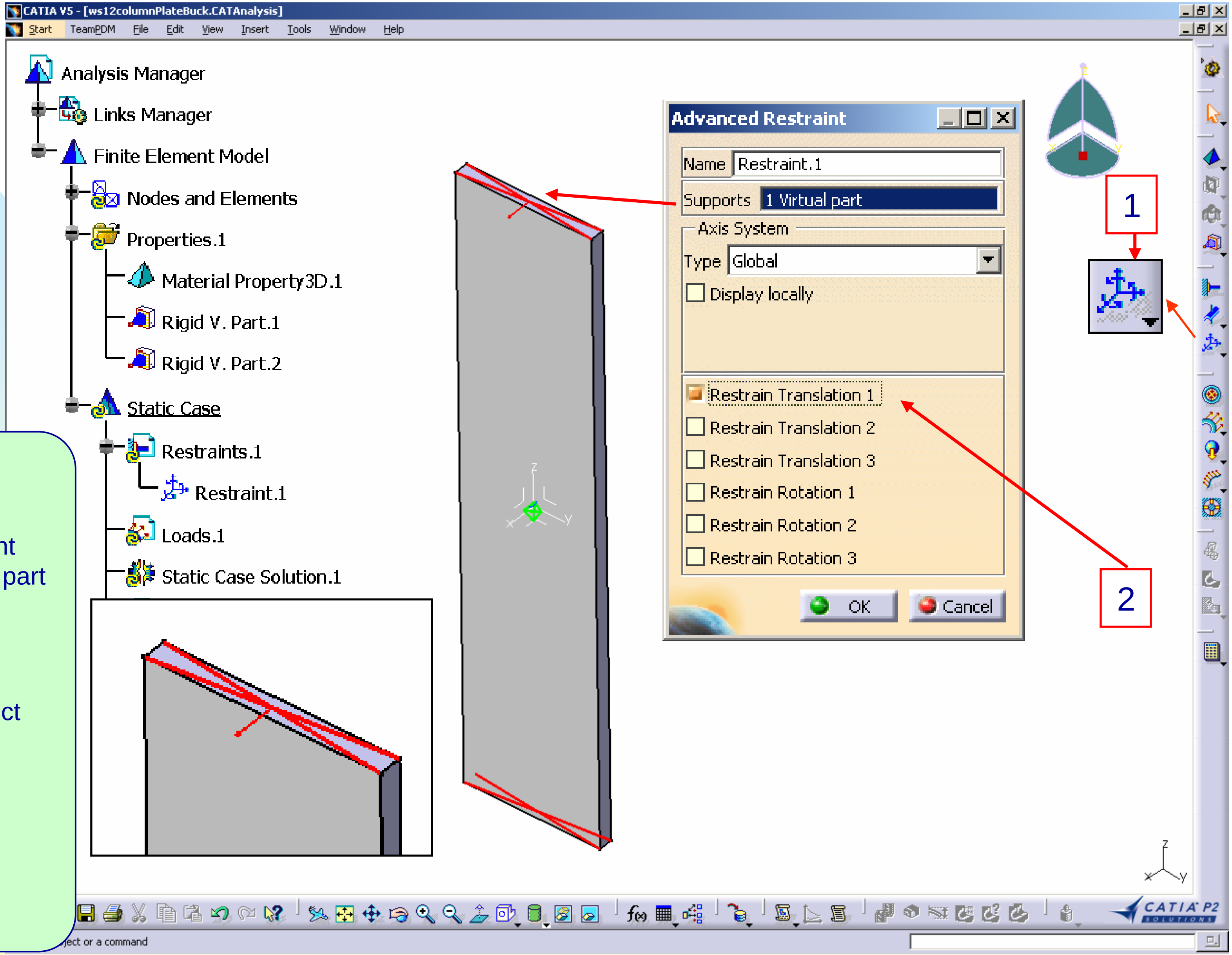
Steps:

1. Select the Rigid Virtual Part icon, select the upper face, select OK.

2. Repeat the process to create a second rigid virtual part on the bottom face.

MSC SOFTWARE

Step 3. Apply advanced restraints

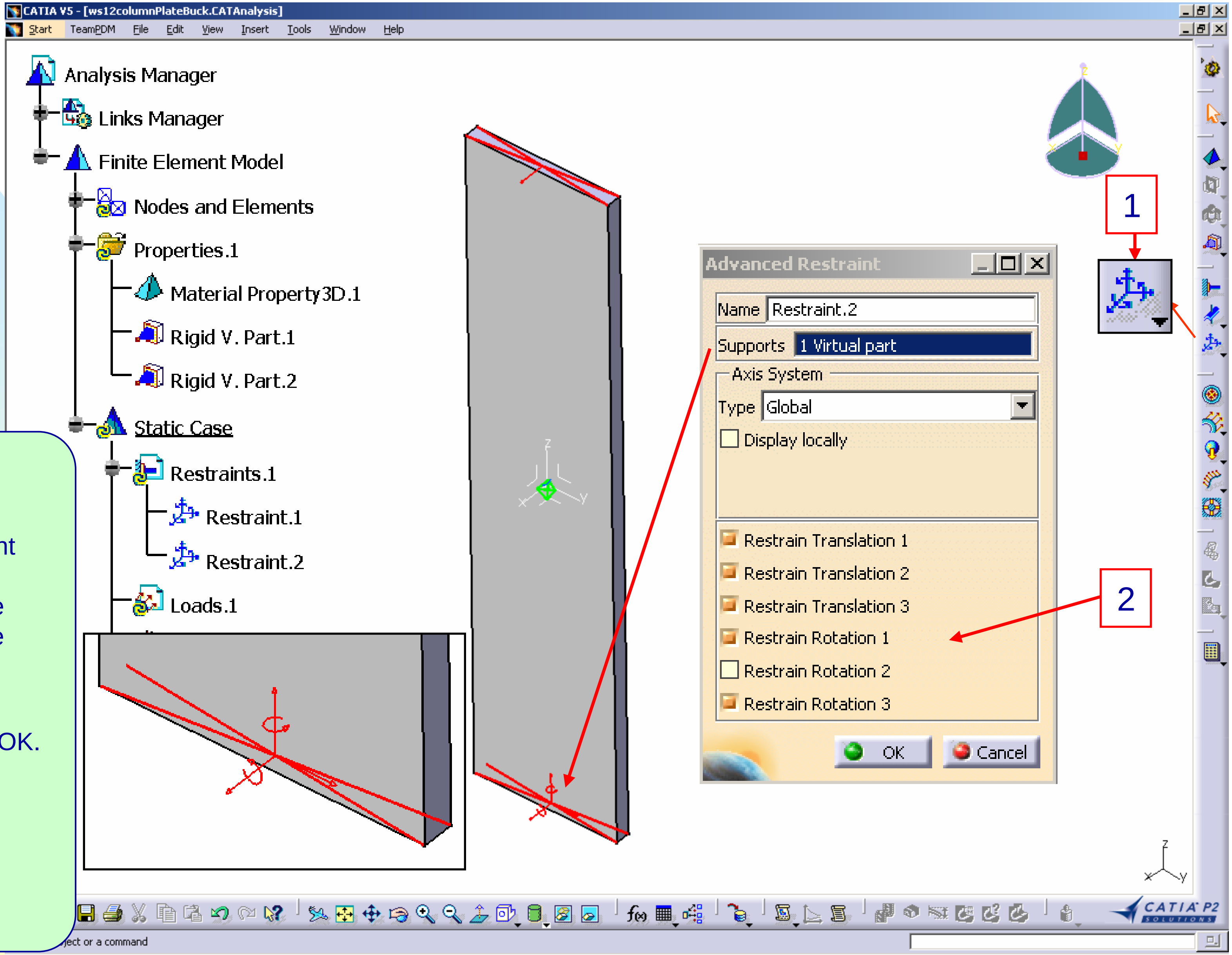


Steps:

1. Select the Advanced Restraint icon, select virtual part 1 at the top of the plate

2. Select Restrain Translation 1, select OK.

Step 3. Apply advanced restraints

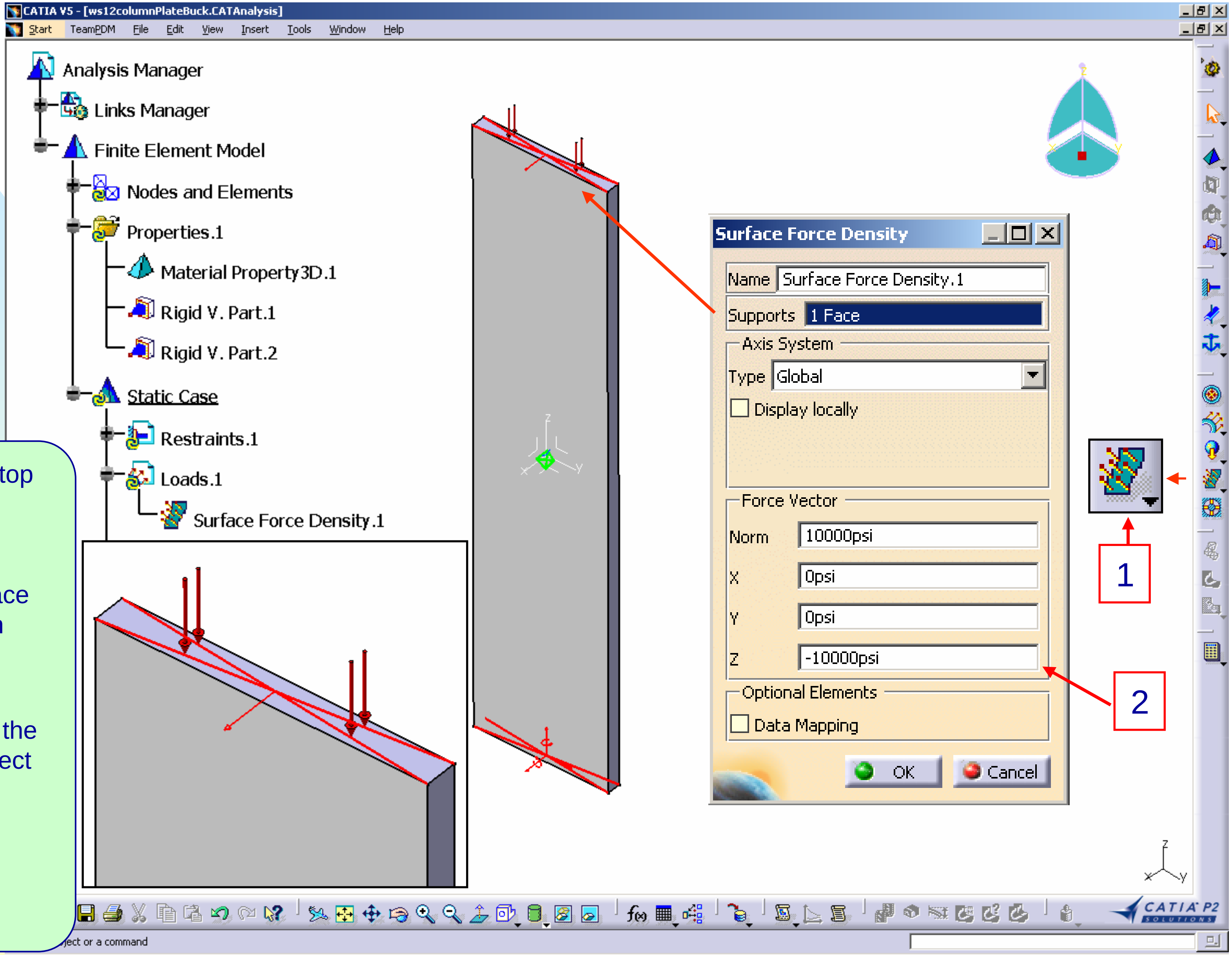


Steps:

1. Select the Advanced Restraint icon again, select virtual part 2 at the bottom of the plate

2. Restrain all directions except Rotation 2, select OK.

Step 4. Apply a force



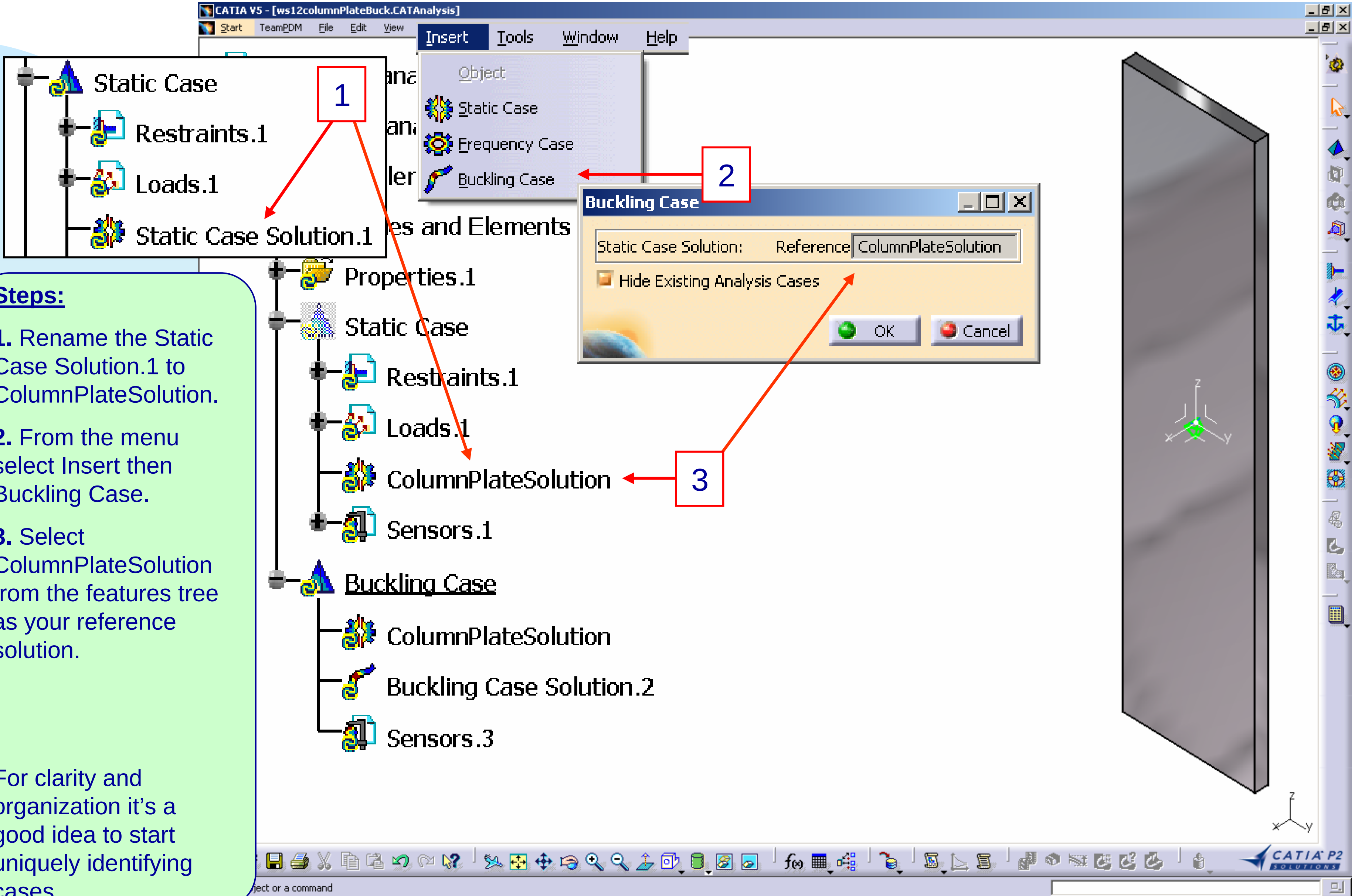
Apply force to the top face

Steps:

1. Select the Surface Force Density icon and select the top face.

2. Enter -10000 in the z direction and select OK.

Step 5. Insert a Buckling Case



Steps:

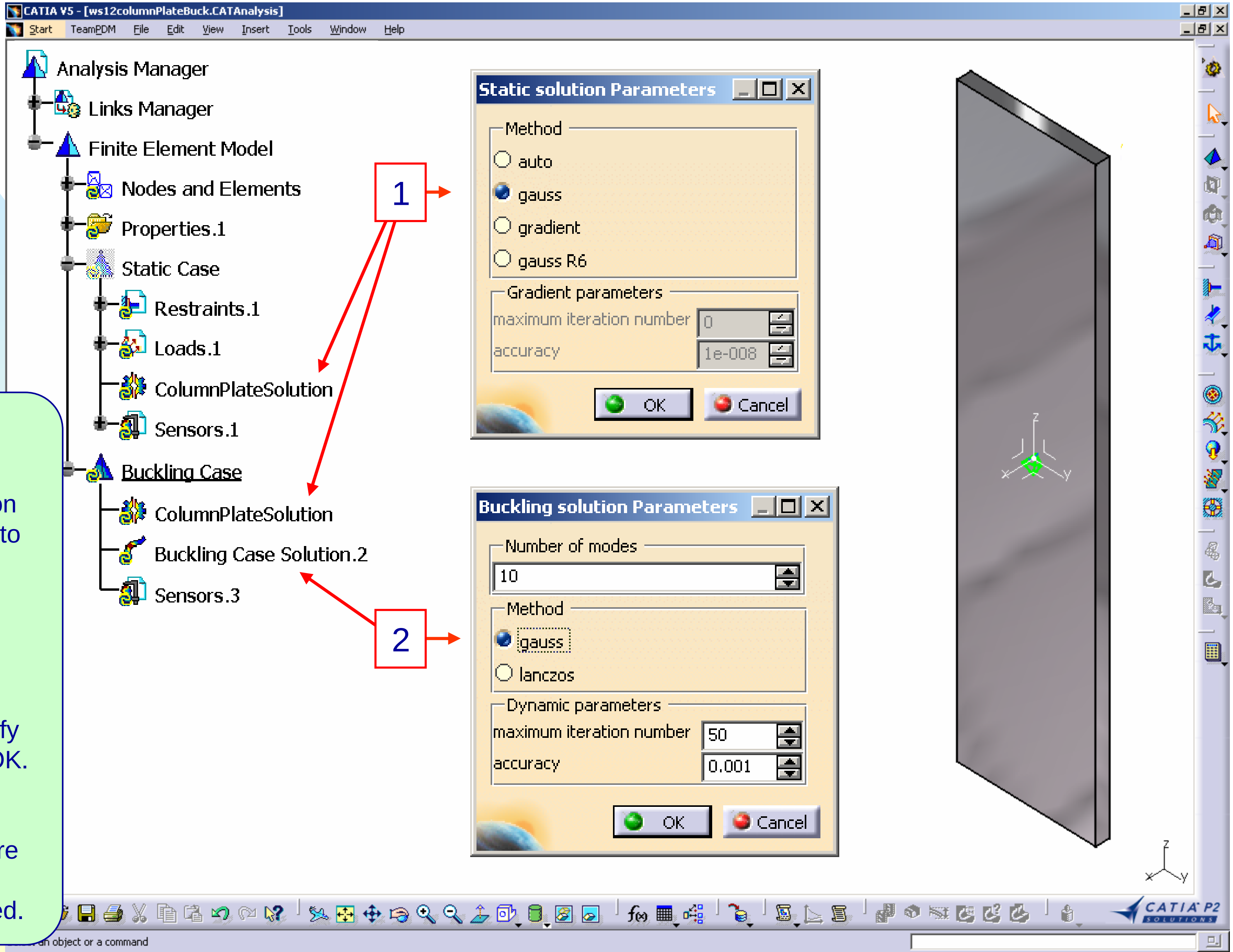
1. Rename the Static Case Solution.1 to ColumnPlateSolution.

2. From the menu select Insert then Buckling Case.

3. Select ColumnPlateSolution from the features tree as your reference solution.

For clarity and organization it's a good idea to start uniquely identifying cases.

Step 6. Setup static and buckling parameters



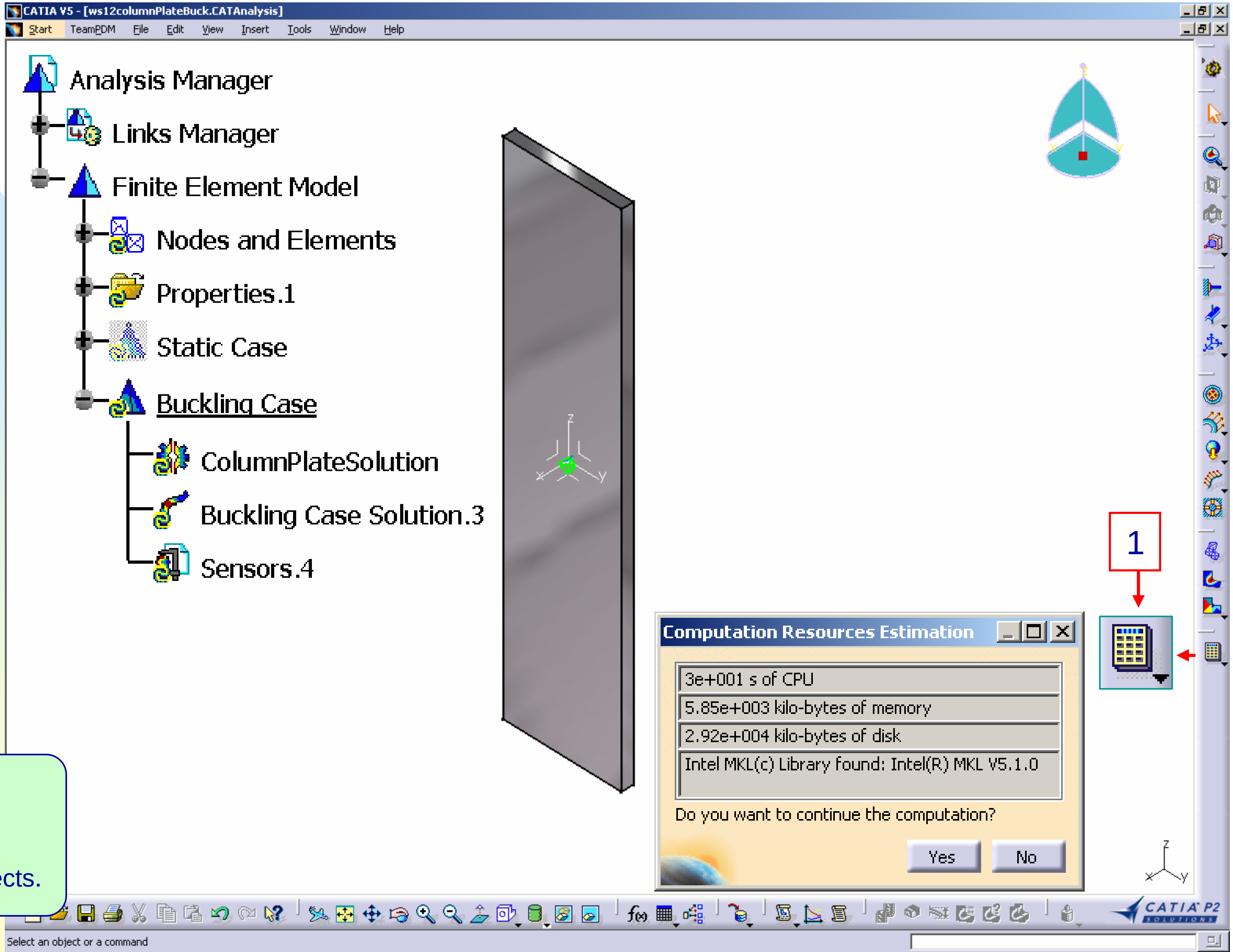
Steps:

1. Double click ColumnPlateSolution in the features tree to verify parameters. Click OK.

2. Double click Buckling Case Solution in the features tree to verify parameters. Click OK.

You should be aware of what calculation methods will be used.

Step 7. Compute all



Save first.

Steps:

1. Compute all objects.

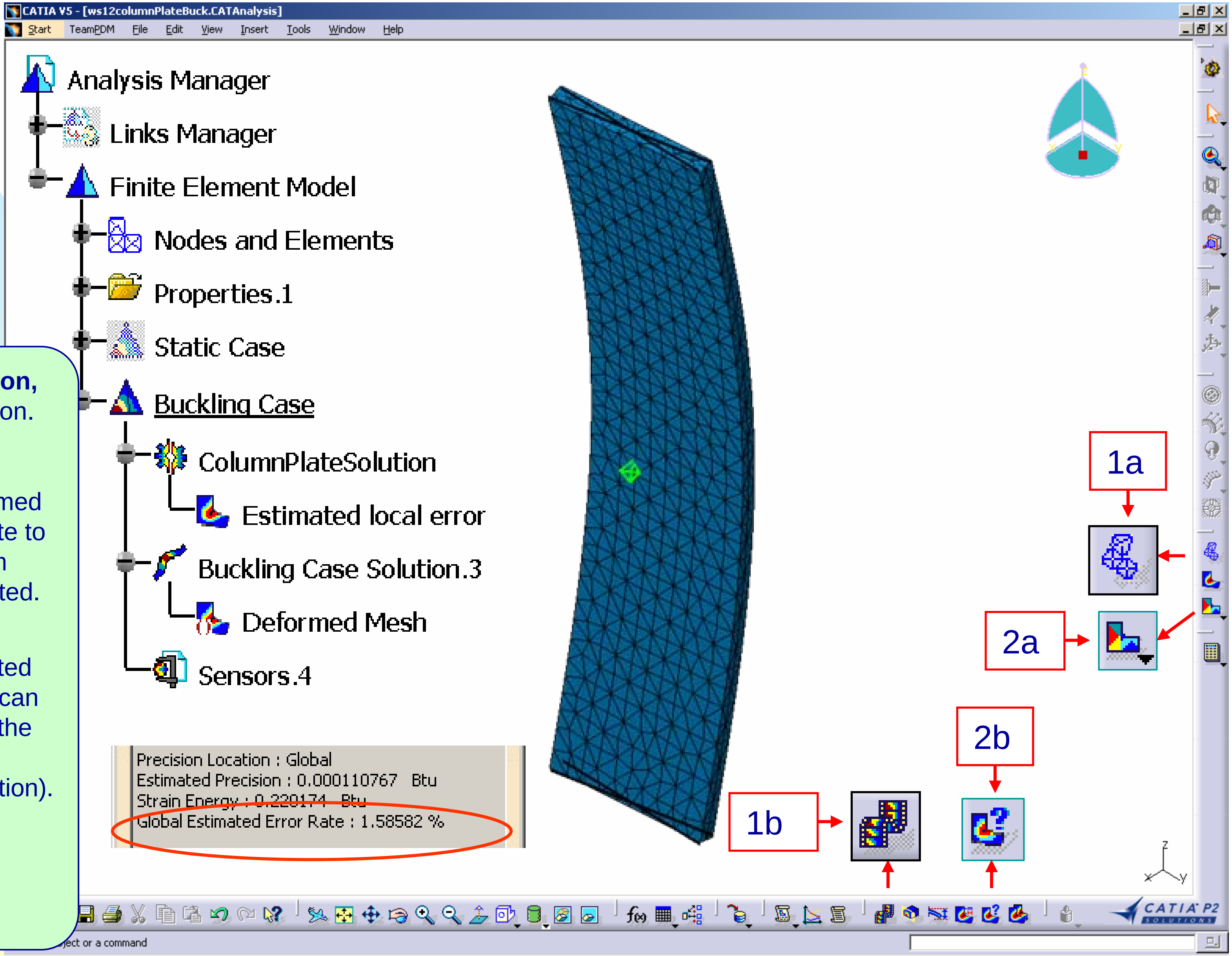
Step 8. Check global and local precision

Check **Deformation,** and global precision.

Steps:

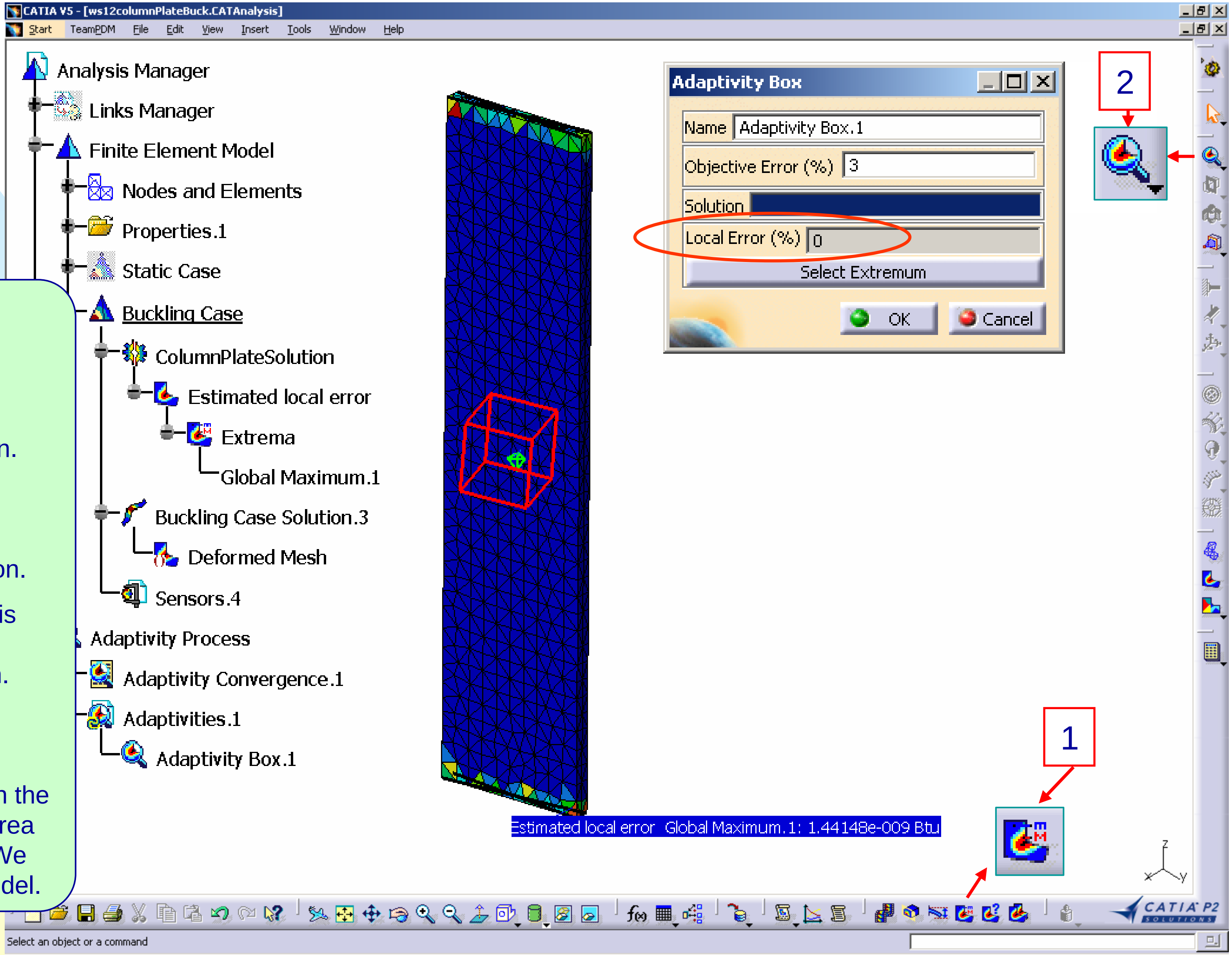
1. Create a deformed image and animate to verify your system deflects as expected.

2. Check Global precision (Estimated local error image can only be added to the Static ColumnPlateSolution).



MSC SOFTWARE

Step 8. Check global and local precision



Find the global element with the highest estimated error.

Find local precision.

Steps:

1. Use the Search Image Extrema icon.

2. Local precision is found using the adaptivity box icon.

Local error shows energy balanced in the plate center, our area of most concern. We have a precise model.

Step 9. Visualize final results

Find the critical load when this plate will fail.

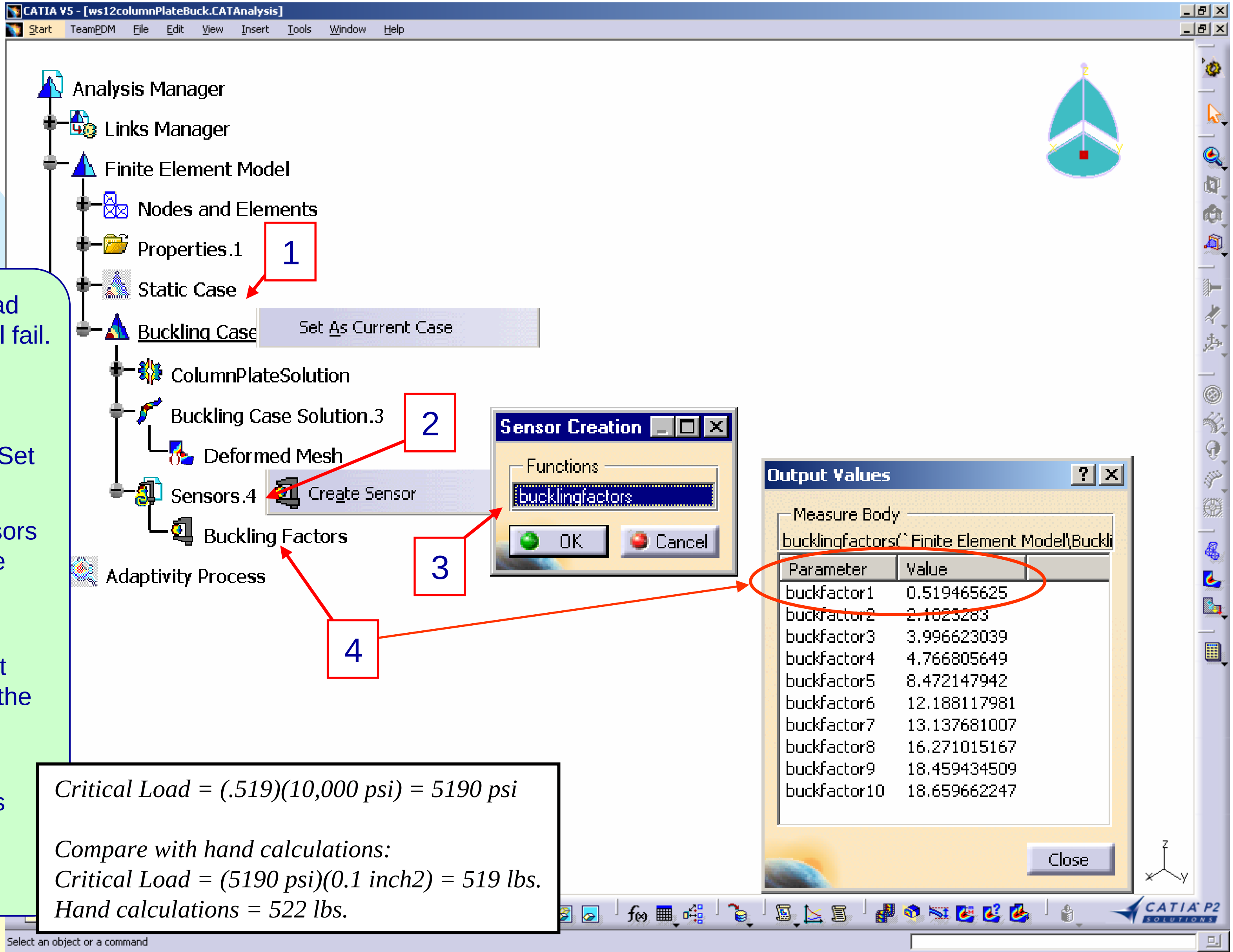
Steps

1. Make sure the Buckling Case is “Set As Current Case”.

2. Right click Sensors in the features tree then select Create Sensor.

3. Click to highlight bucklingfactors in the Sensor Creation window. Click OK.

4. Double click this sensor “Buckling Factors” in the features tree.



Critical Load = (.519)(10,000 psi) = 5190 psi

Compare with hand calculations:
Critical Load = (5190 psi)(0.1 inch2) = 519 lbs.
Hand calculations = 522 lbs.

Step 9. Visualize final results

Add a different Static Case: Beam simply supported at both ends, uniform load over entire area.

Steps

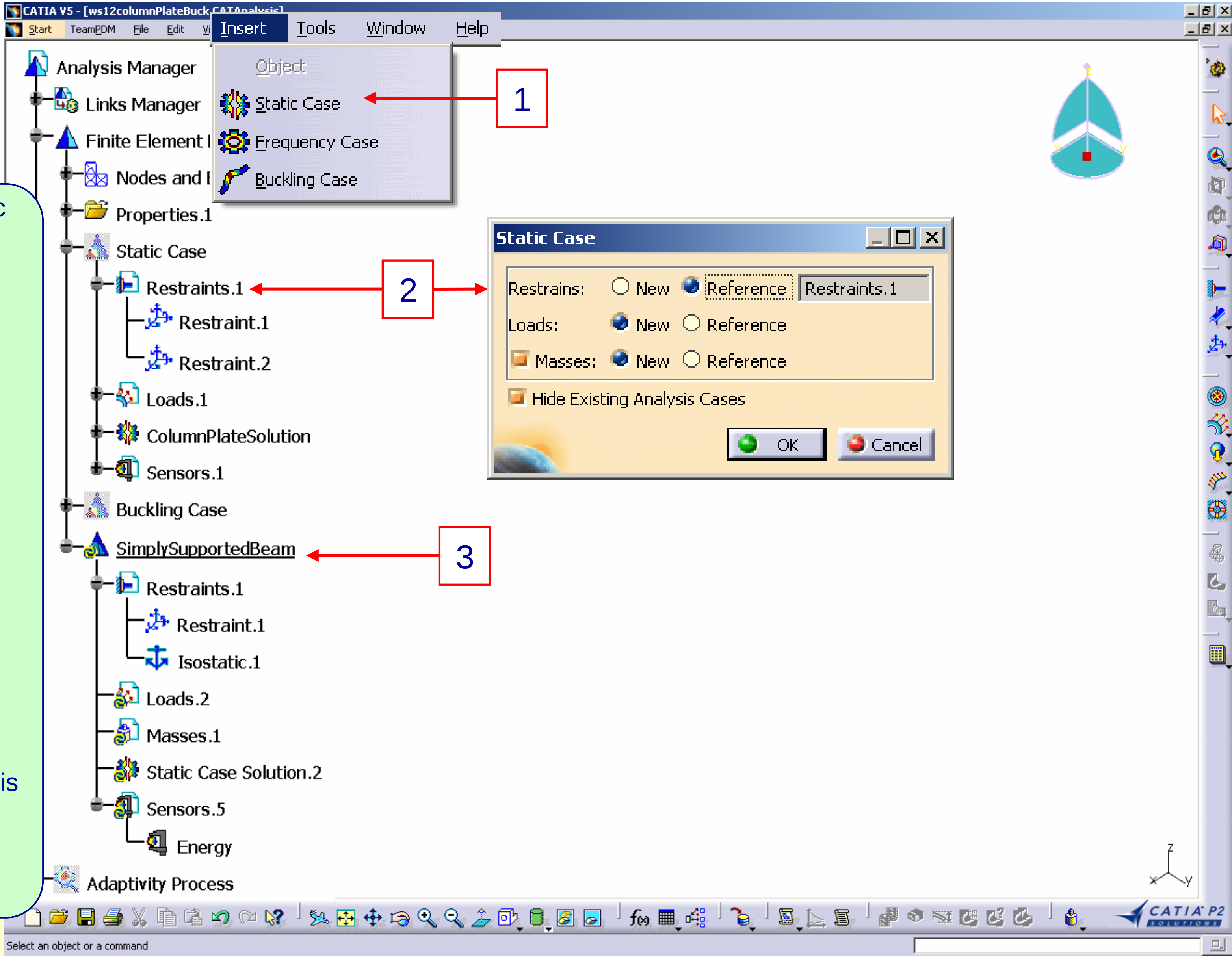
1. From the menu select Insert then Static Case.

2. Select existing restraints to save setup time.

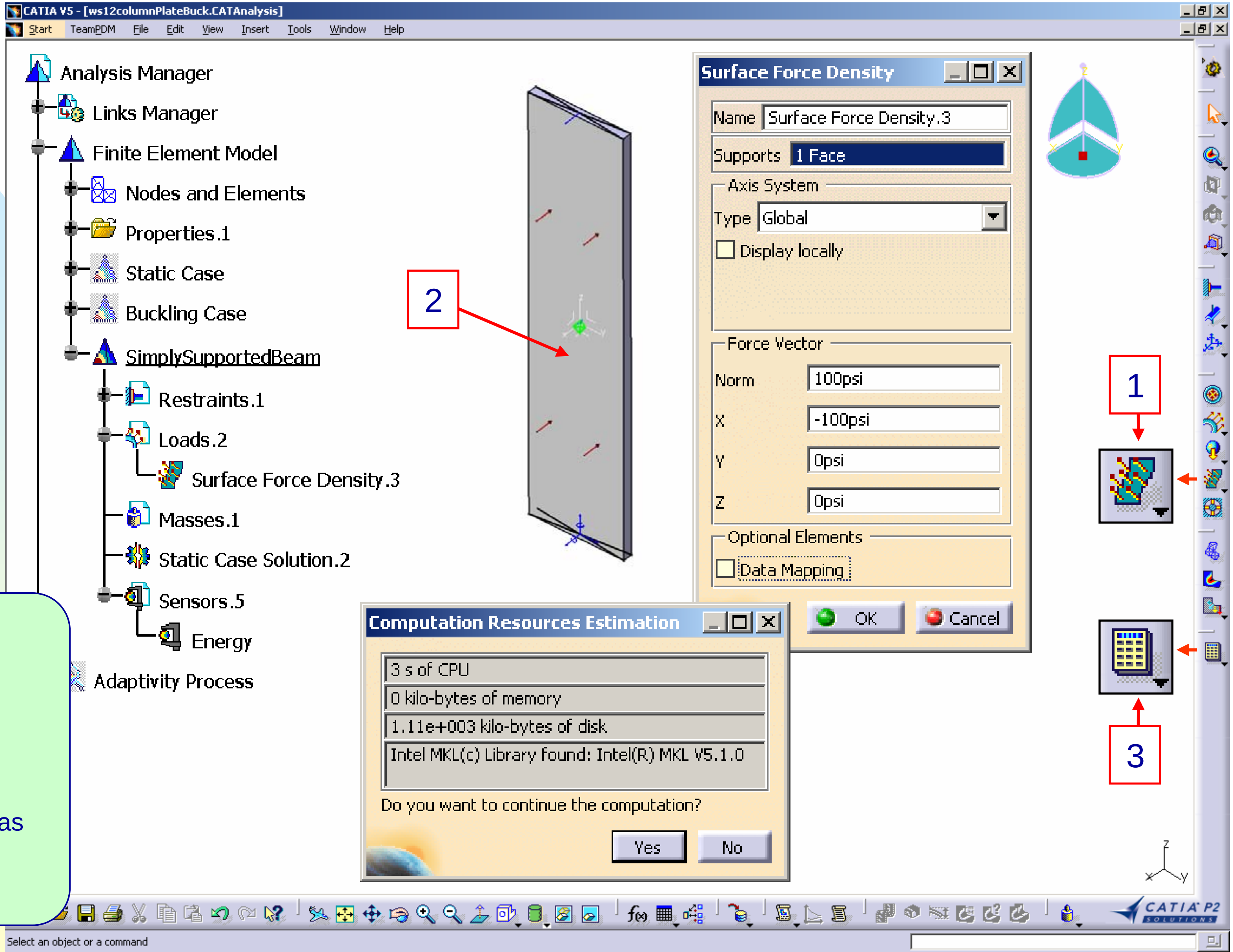
3. Rename Simply Supported Beam.

We are doing this to verify that the beam is deflecting properly.

Also introducing multiple load cases.



Step 9. Visualize final results



Load 100 psi and compute.

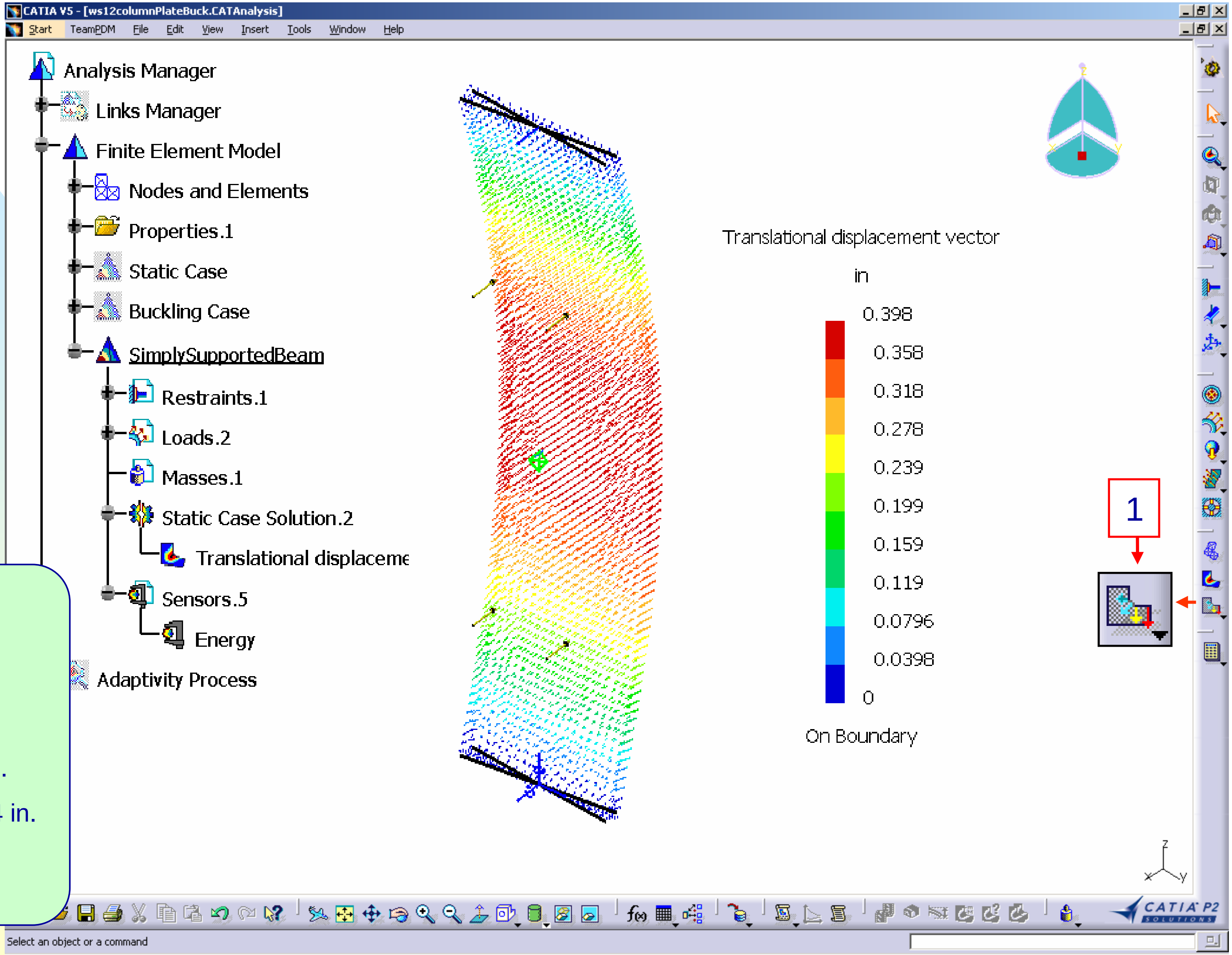
Steps

1. Use the surface force density icon.

2. Select the face as shown.

3. Compute all.

Step 9. Visualize final results



Compare hand calculation displacements.

Steps

1. Select the Displacement icon.

Hand calc's = .394 in.

FEA = .398 in.

Looks good.

MSC SOFTWARE

Step 9. Visualize final results

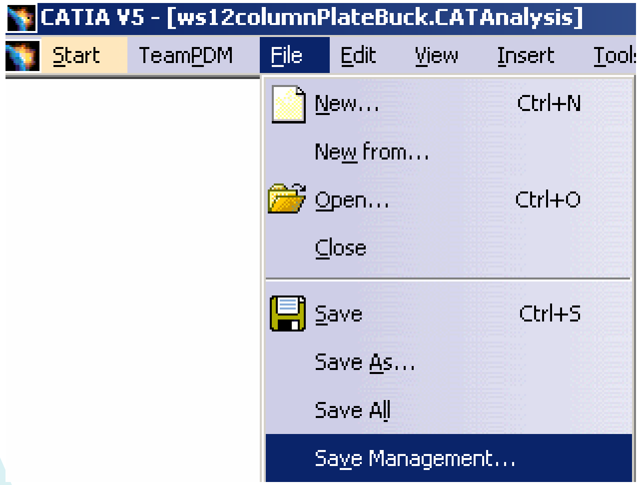
Conclusions

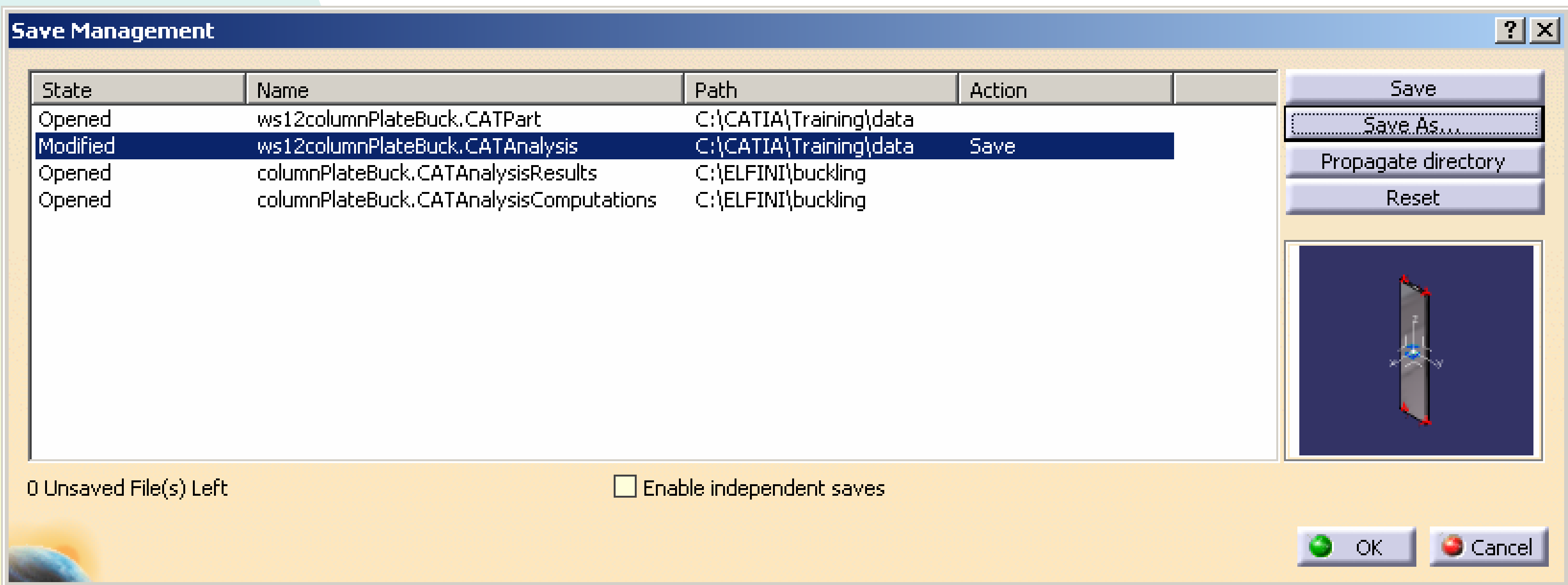
- CATIA V5 GSA workbench is validated for a flat plate column buckling scenario.

	Hand Calculations	.1 inch Parabolic Global Mesh, .01 inch sag
Global % Precision error Local % Precision error	NA NA	1.58 % 0 %
Error Estimate	NA	1.44e-9 Btu global
Critical Load	522 lbs	519 lbs
Model verification using simply supported beam displacement	0.394 inch	0.398 inch

Step 10. Save the analysis document

Save your documents

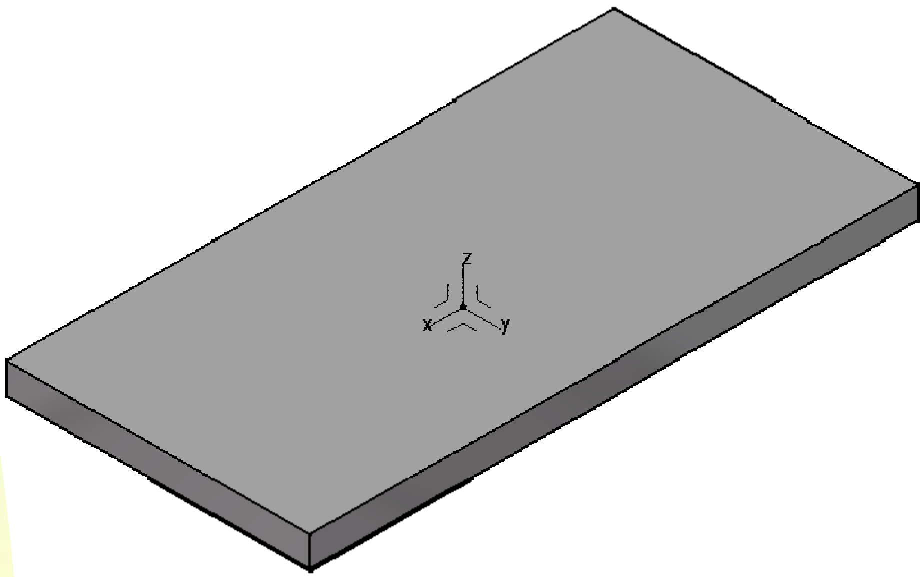




WORKSHOP 12…

FLAT PLATE BUCKLING,
PINNED ALL FOUR EDGES
FIXED ALL FOUR EDGES
PINNED TWO EDGES, FIXED TWO EDGES

CANTILEVER PLATE LATERAL BUCKLING

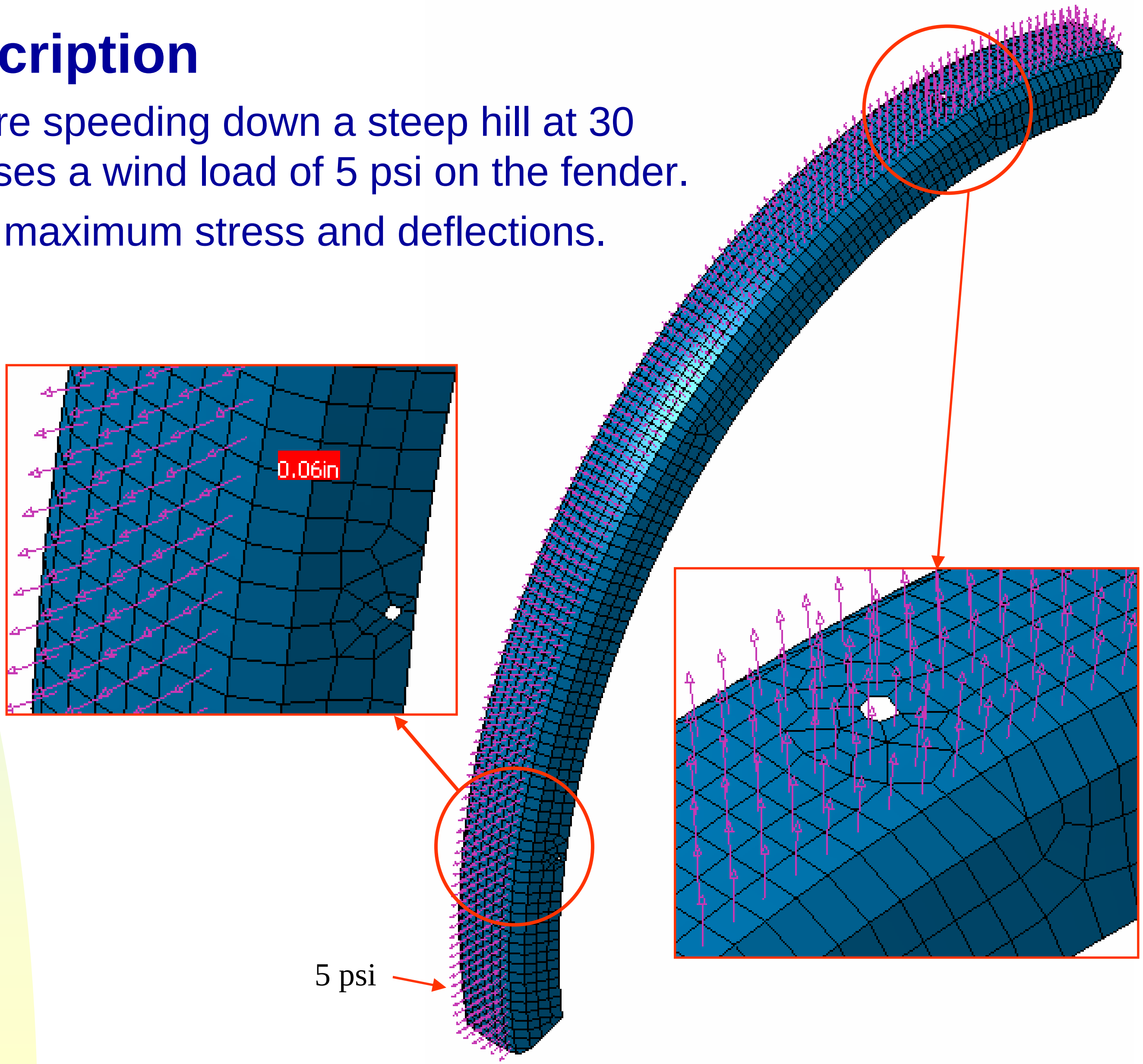


WORKSHOP 13

BICYCLE FENDER SURFACE MESHING

Problem Description

- Assume you are speeding down a steep hill at 30 mph. This causes a wind load of 5 psi on the fender.
- Determine the maximum stress and deflections.



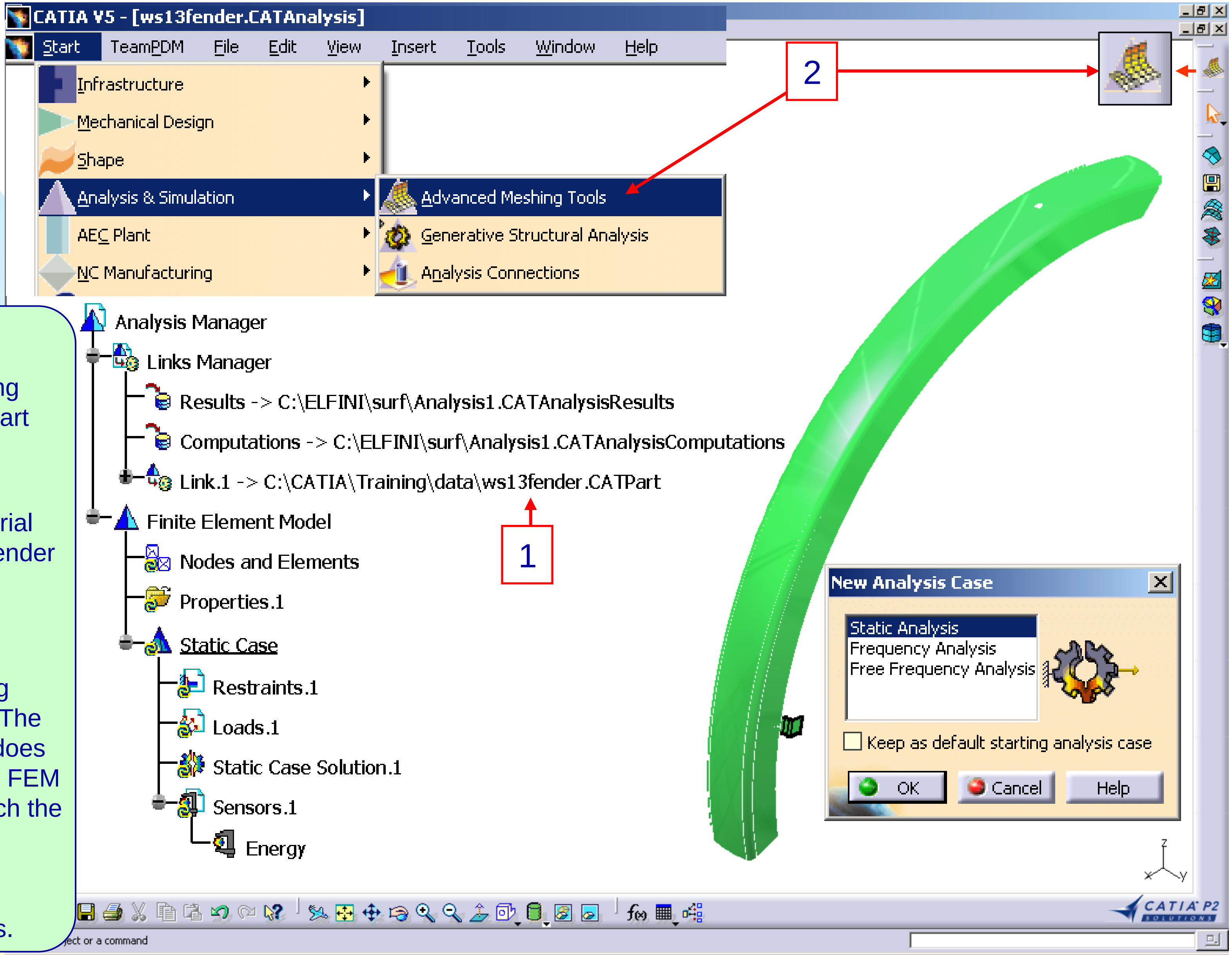
Material: Bright Green Plastic
Modulus of elasticity = 31.9e4 psi
Poisson Ratio = .38
Density = .043 lb_in3

Design requirements:
Thickness, t = 0.06 inch
Wind Load, w = 5 psi

Suggested Exercise Steps

1. Start the Advanced Meshing Tools workbench (static analysis).
2. Specify global surface meshing parameters.
3. Add surface constraints.
4. Impose surface nodes.
5. Mesh the part.
6. Check mesh quality and repair.
7. Start the Generative Structural Analysis workbench.
8. Edit surface thickness.
9. Apply a clamp restraint.
10. Apply a pressure force.
11. Compute all.
12. Check global precision (animate deformation and find extremas).
13. Refine mesh and re-compute.
14. Visualize final results.
15. Save the analysis document.

Step 1. Start the Advanced Meshing Tools workbench



Steps:

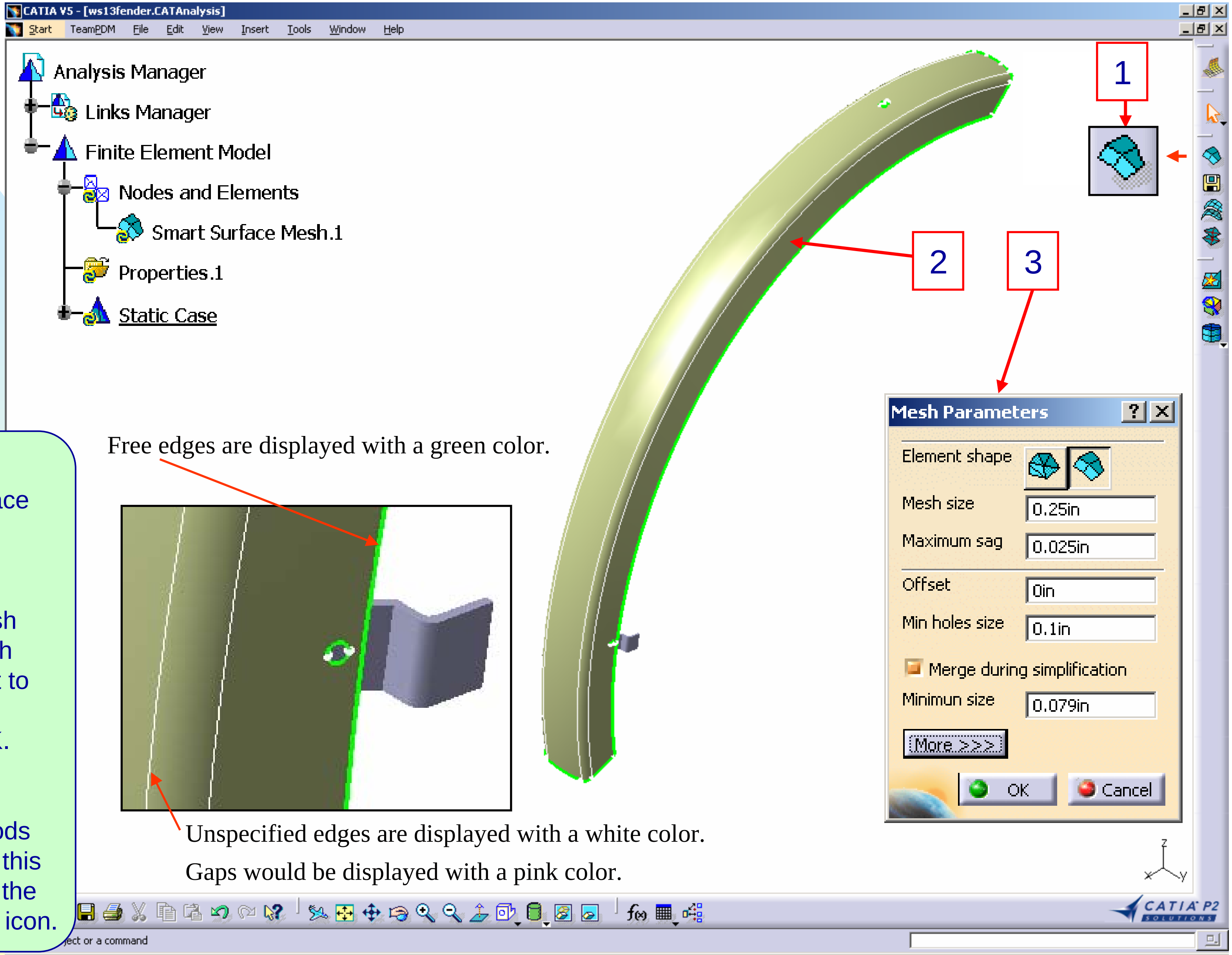
1. Open the existing ws13fender.CATPart from the training directory.

Apply plastic material properties to the fender part as required.

2. Start a Static Analysis with the Advanced meshing tools workbench. The material property does not show up in the FEM tree until you launch the GPS workbench.

Save your analysis.

Step 2. Specify global surface mesh parameters



Steps:

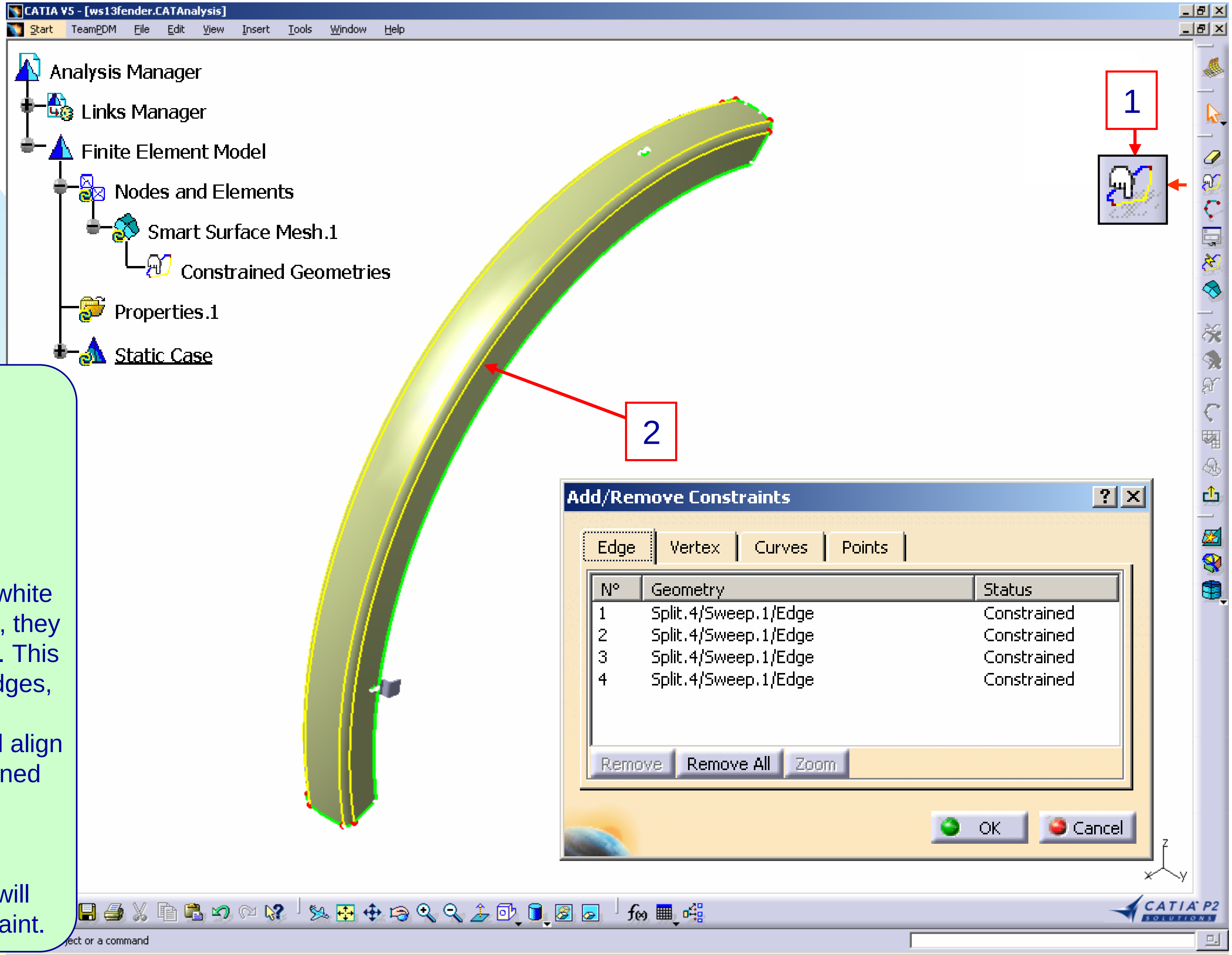
1. Select the Surface Mesher icon.

2. Select the part.

3. Specify the mesh size as shown, with element shape set to frontal quadrangle method, select OK.

Other mesh methods are available after this initial shape using the re-mesh a domain icon.

Step 3. Add surface constraints



Add constraints.

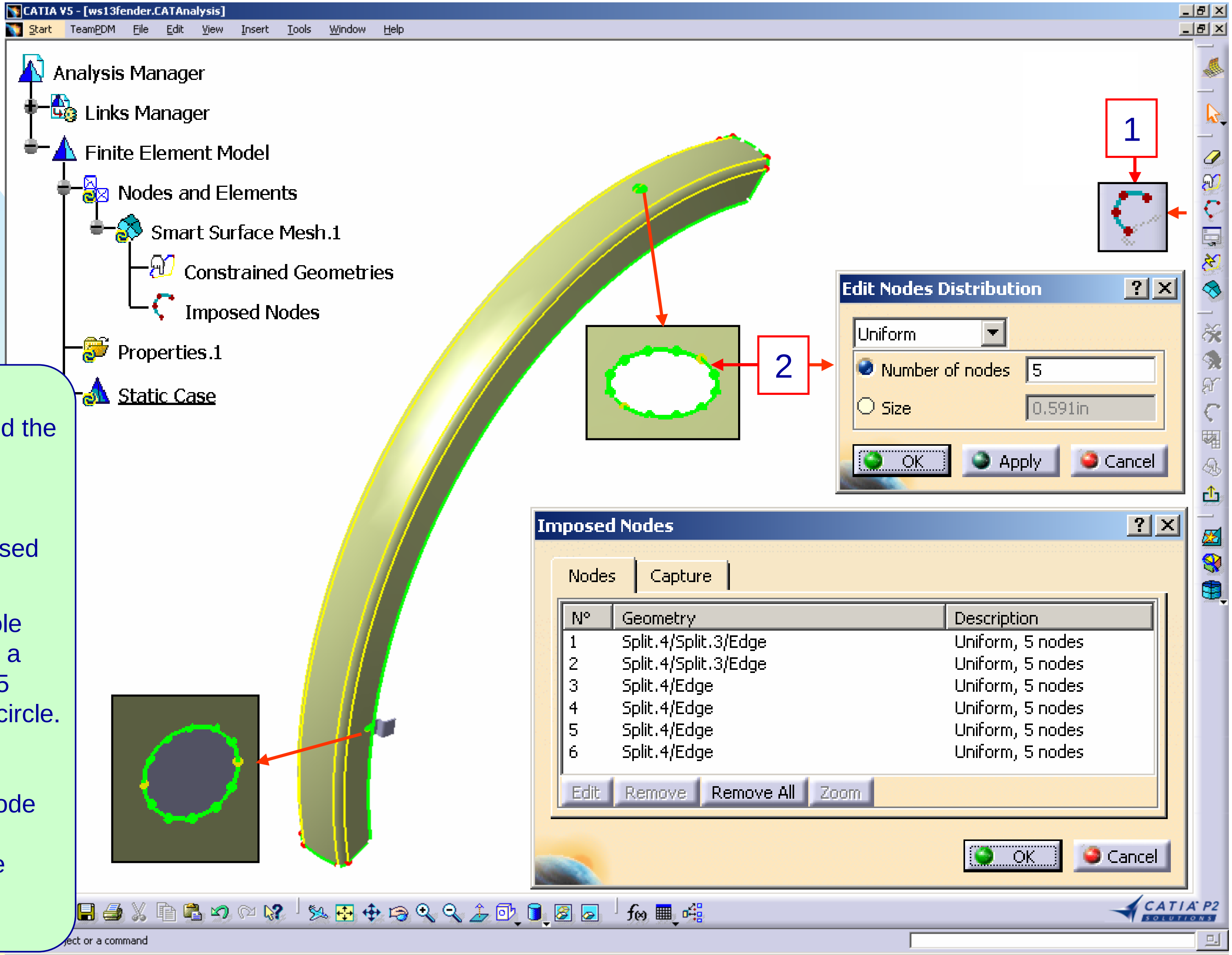
Steps:

1. Select the Add/Remove Constraints icon.

2. Select the four white unspecified edges, they should turn yellow. This “constrains” the edges, meaning the finite element edges will align along this constrained edge.

Selecting it again will remove the constraint.

Step 4. Impose surface nodes



Impose node distributions around the mounting holes.

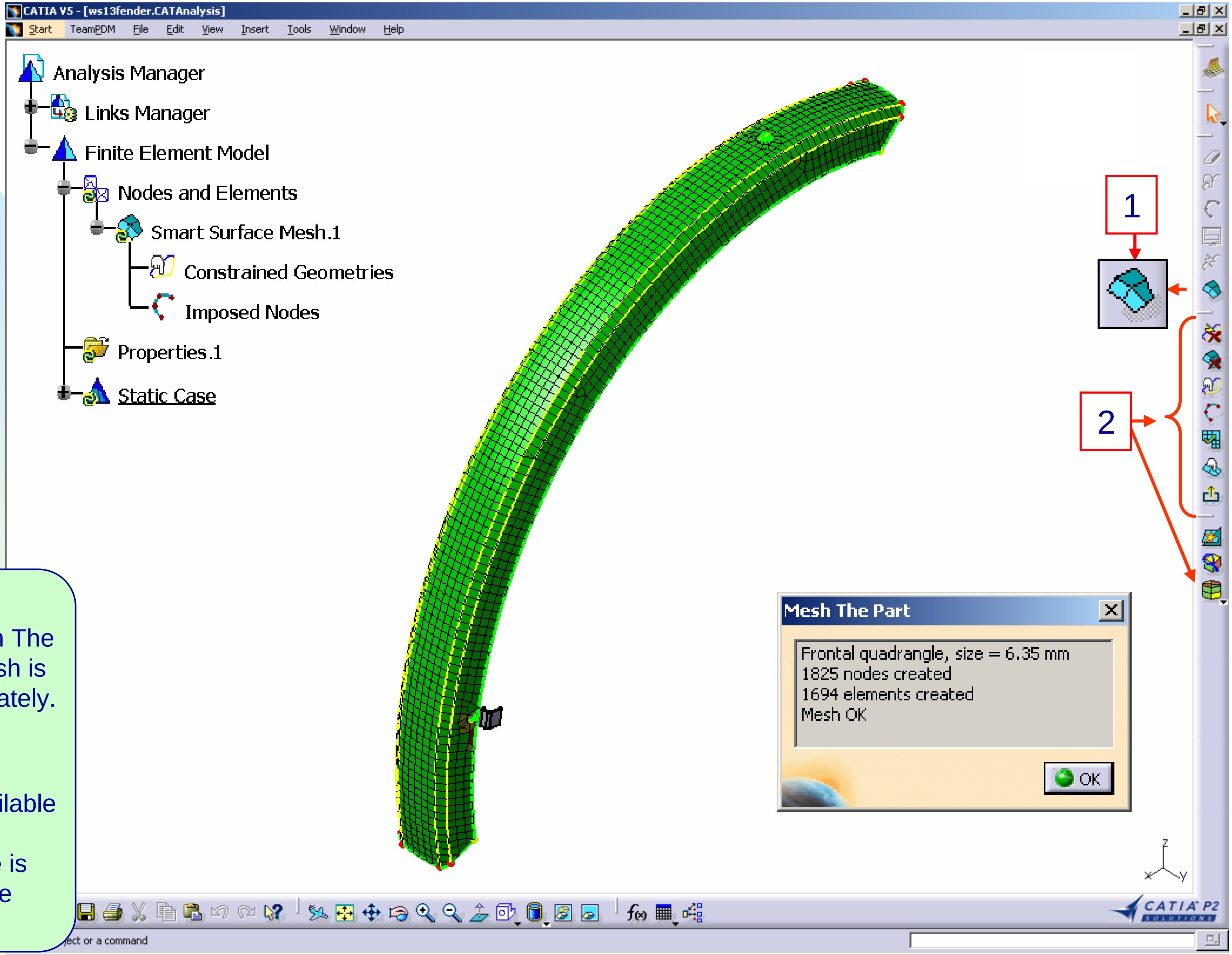
Steps:

1. Select the Imposed Nodes icon.

2. Select all the hole free edges (one at a time) and impose 5 nodes on each ½ circle.

You can specify node distributions on constrained or free edges.

Step 5. Mesh the part



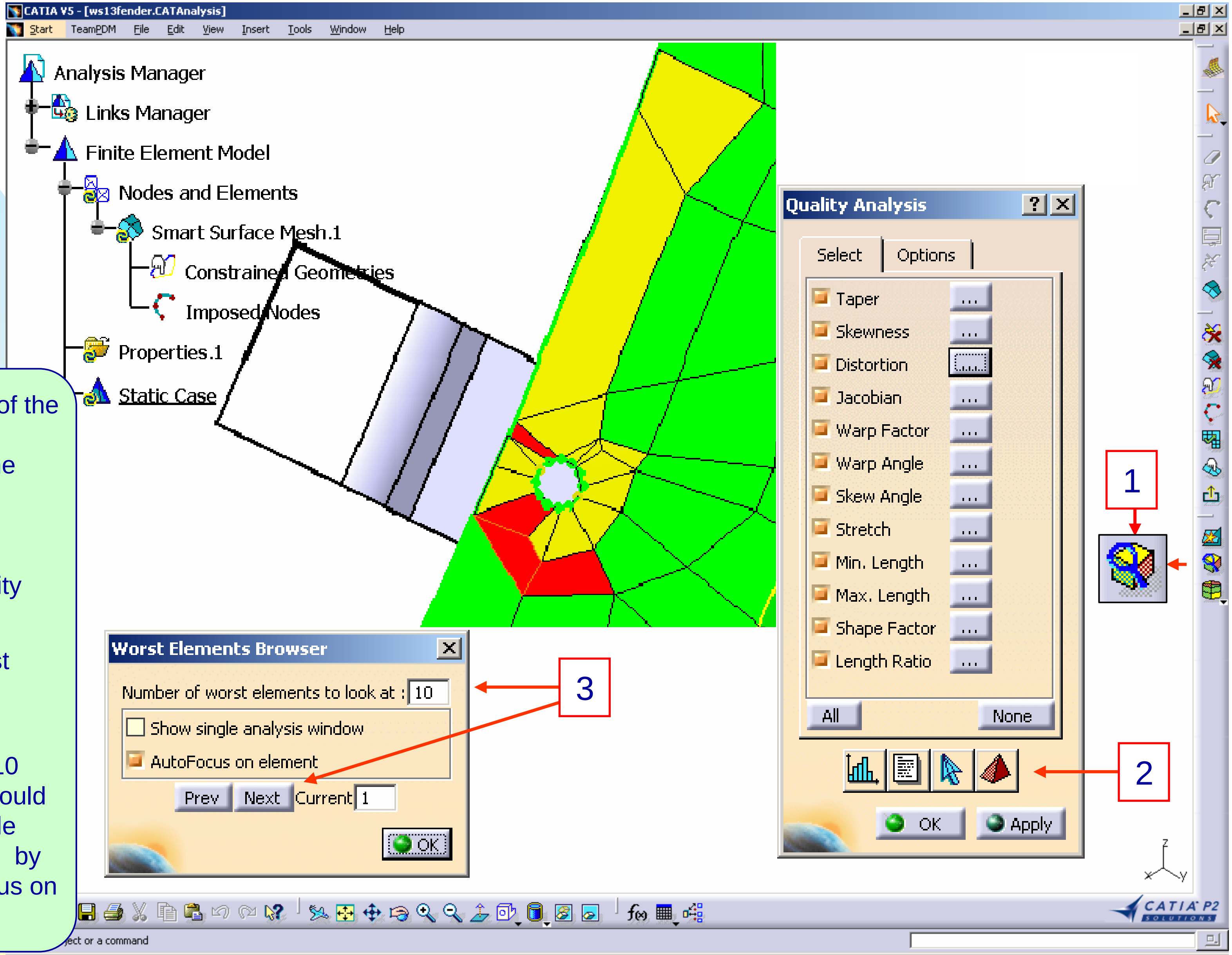
Steps:

1. Select the Mesh The Part icon. The mesh is generated immediately.

2. Notice the modification Tools toolbar is now available and the quality visualization mode is automatically made current.

MSC SOFTWARE

Step 6. Check mesh quality and repair



Check the quality of the finite elements by searching for all the worst elements.

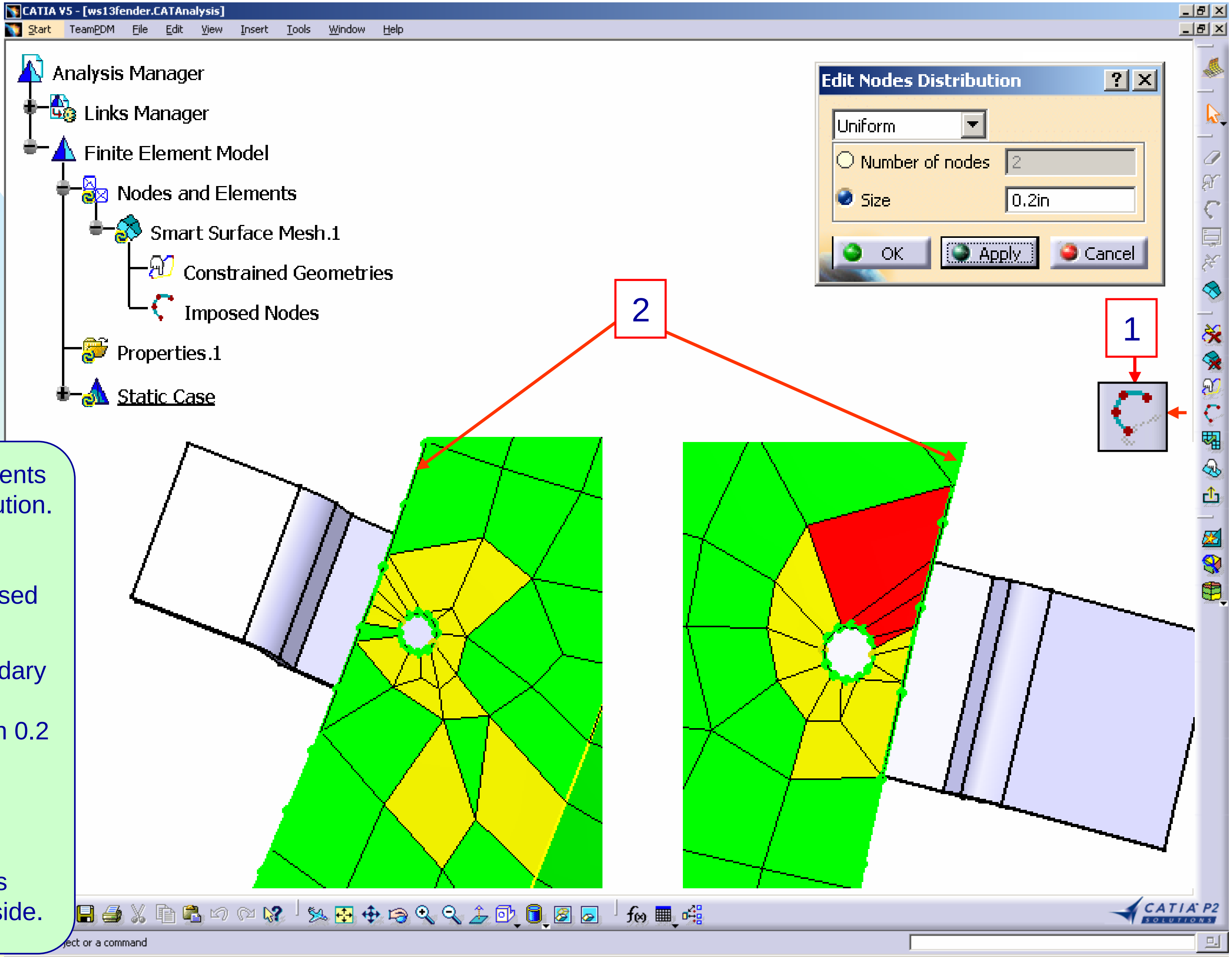
Steps:

1. Select the Quality Analysis icon.

2. Select the Worst elements browser.

3. For this model, searching for the 10 worst elements should find them all. Cycle through the top 10 by selecting AutoFocus on element and Next.

Step 6. Check mesh quality and repair



Fix the worst elements using node distribution.

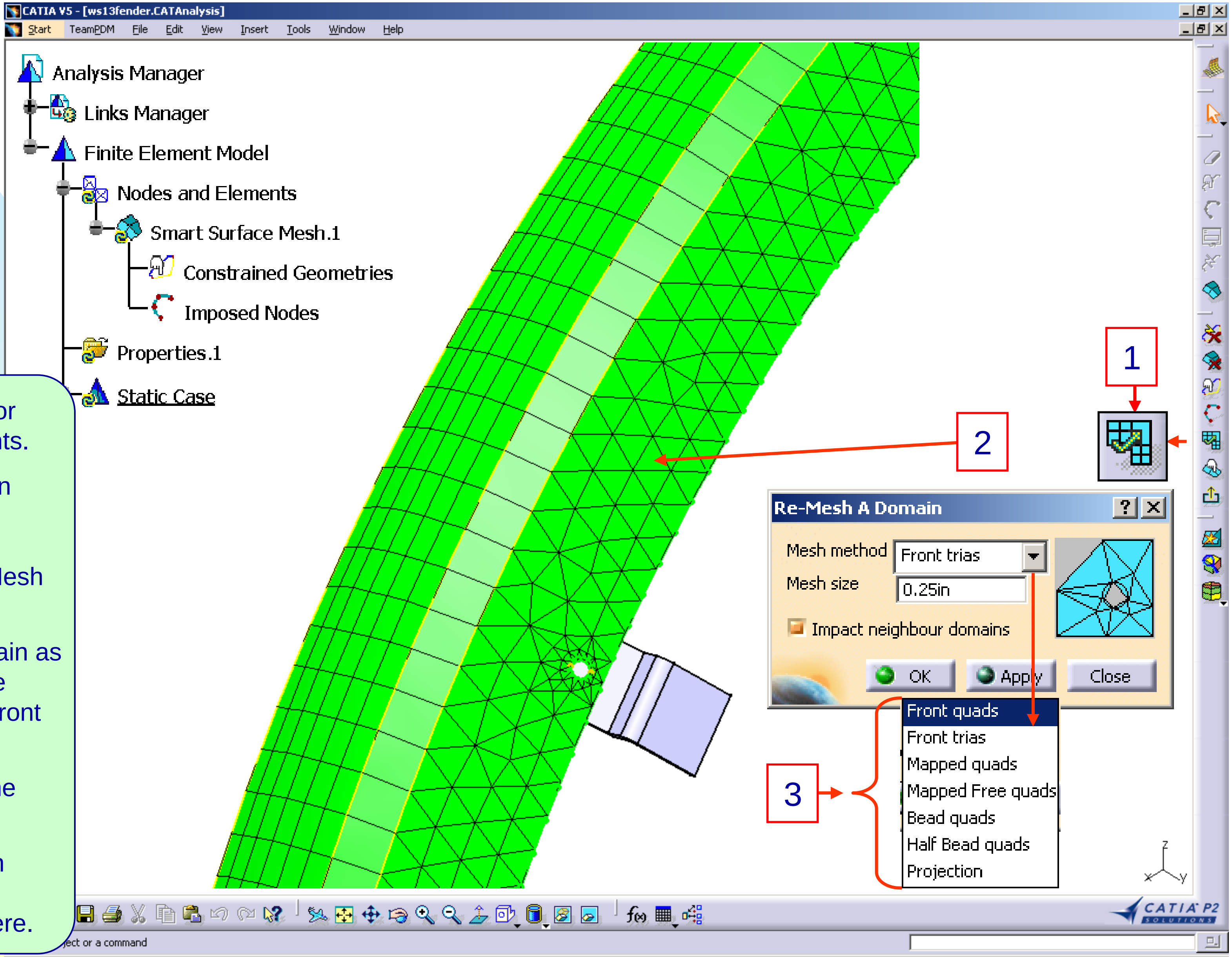
Steps:

1. Select the Imposed Nodes icon.

2. Select the boundary edges as shown (separately), key in 0.2 inch, select OK.

The left side looks good, more work is need on the right side.

Step 6. Check mesh quality and repair



Use another tool for fixing finite elements.

Re-Mesh a Domain

Steps:

1. Select the Re-Mesh a Domain icon.

2. Select the Domain as shown, change the mesh method to Front trias, select OK.

Do the same for the opposite side.

3. Notice the Mesh methods that are available to you here.

Step 6. Check mesh quality and repair

Another tool for fixing bad finite elements.

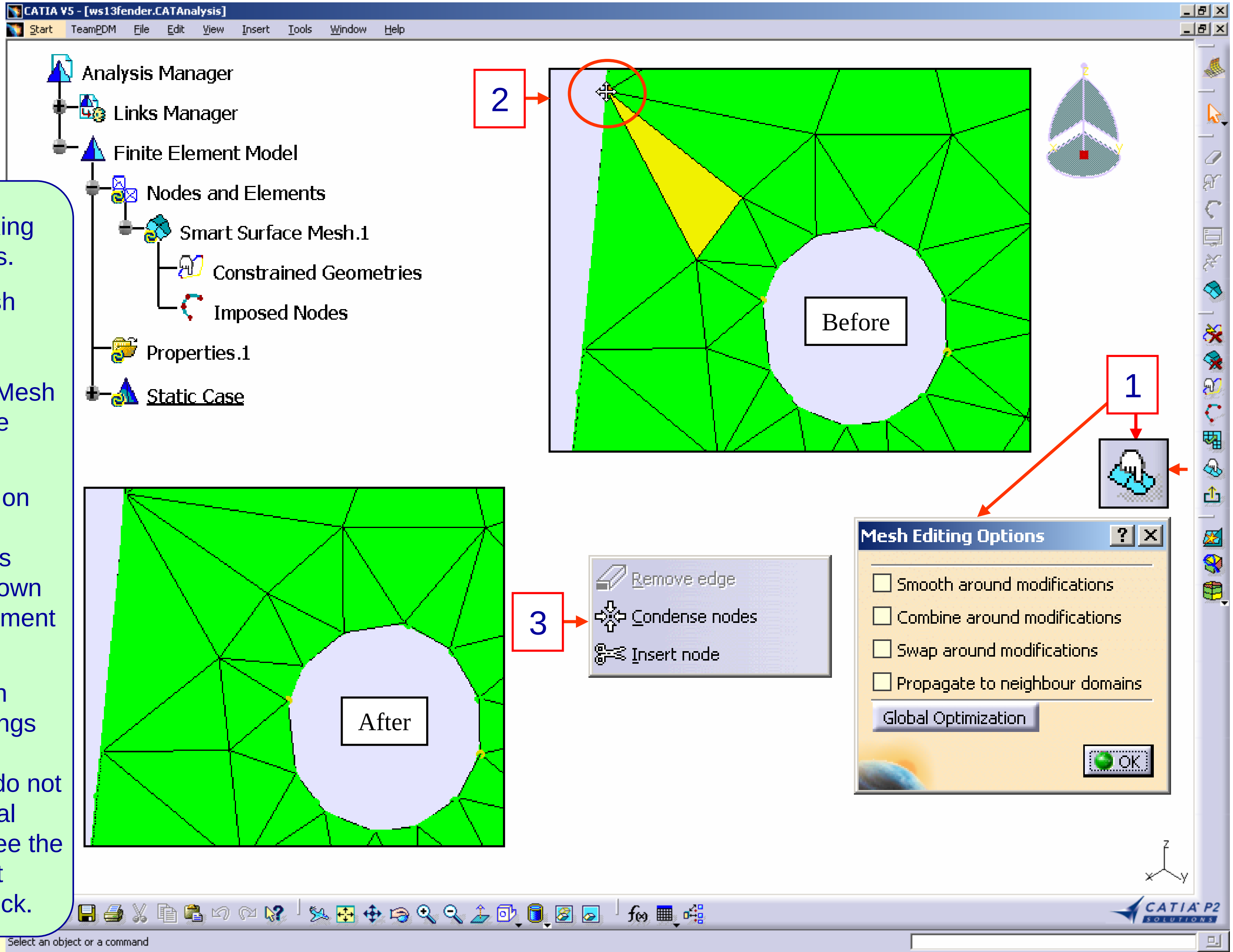
Manually Edit mesh

Steps:

1. Select the Edit Mesh icon, turn off all the options.

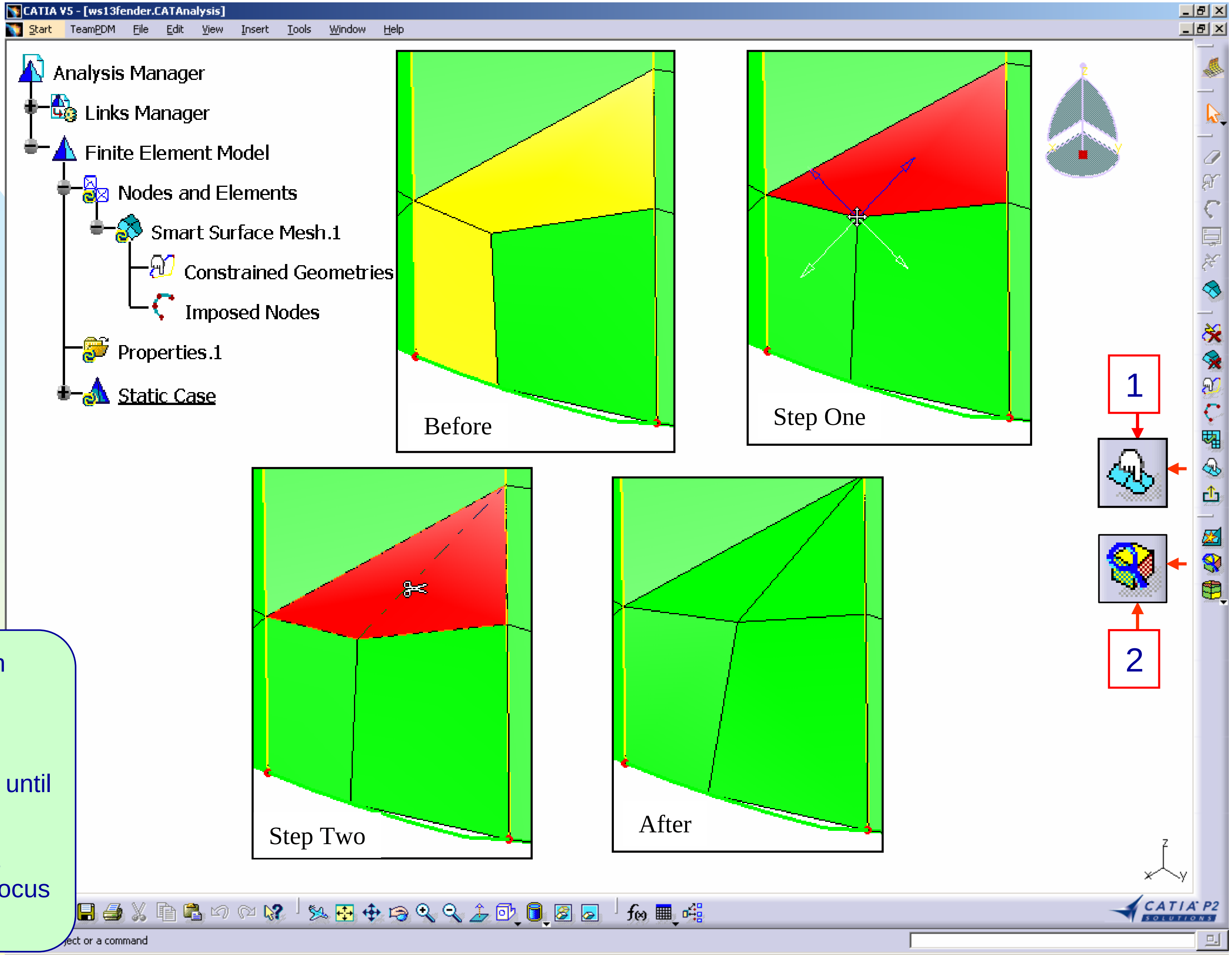
2. Hold the cursor on the node until the symbol changes as shown and drag down slowly until the element turns green.

3. Right clicking on element edges brings up this contextual menu. You really do not need the contextual menu when you see the condense or insert symbol, just left click.



MSC SOFTWARE

Step 6. Check mesh quality and repair



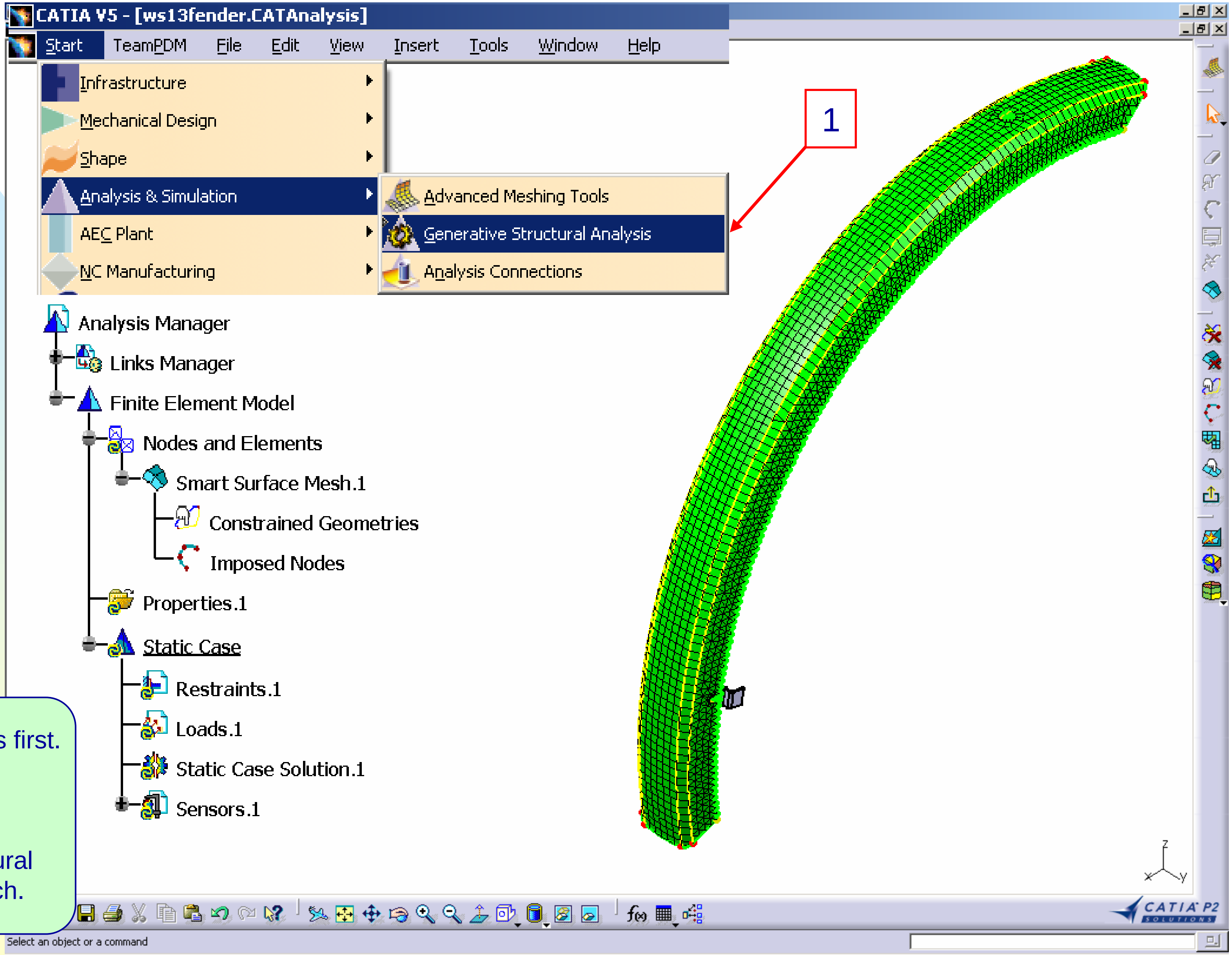
Manually Edit mesh (cont.).

Steps:

1. Edit all elements until you have all green.

2. Use the Element quality browser to focus in on others.

Step 7. Start the GSA workbench

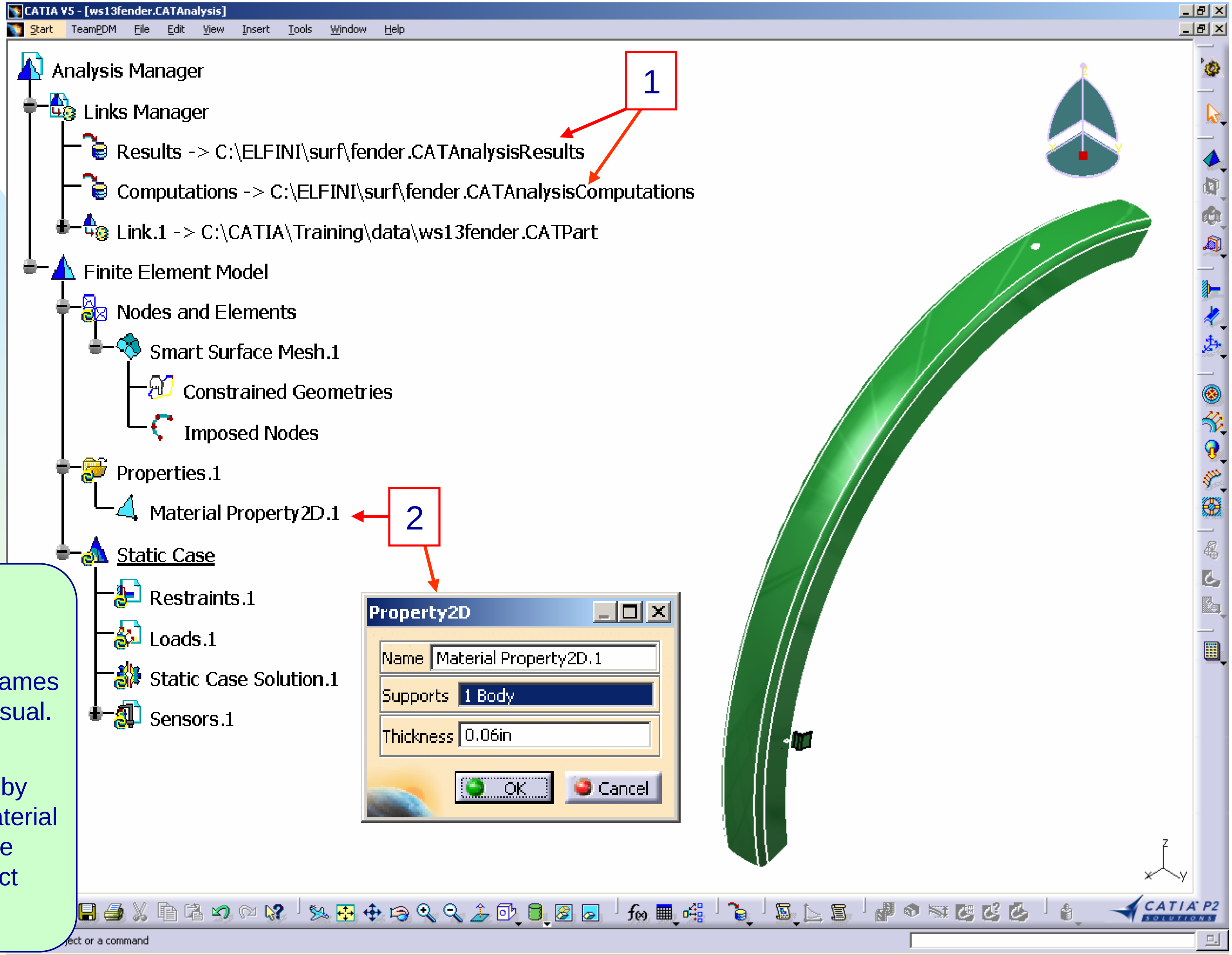


Save your analysis first.

Steps:

1. Launch the Generative Structural Analysis workbench.

Step 8. Edit surface thickness

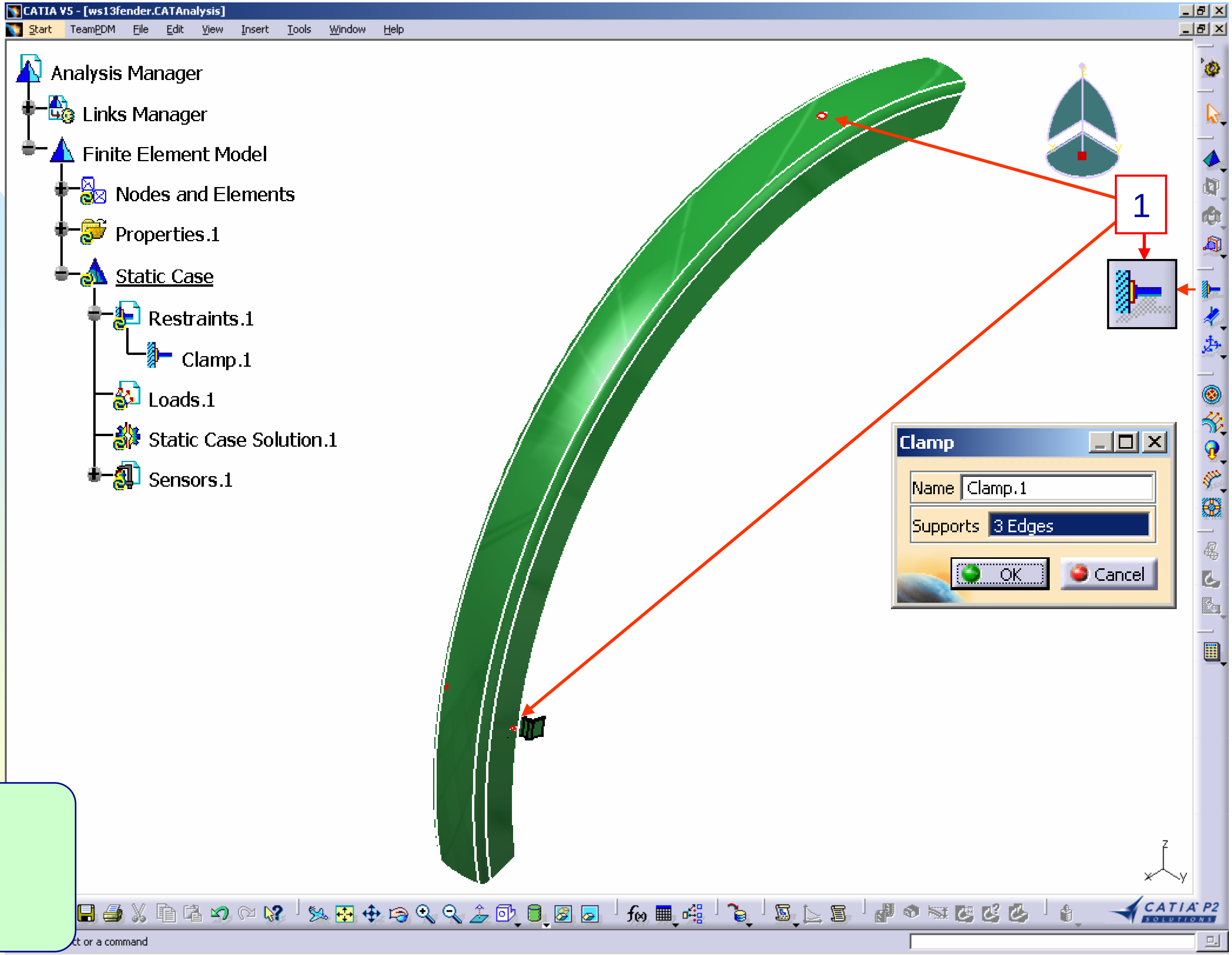


Steps:

1. Set up all your external storage names and locations as usual.

2. Edit the surface thickness (0.06in) by double clicking Material Property2D.1 in the features tree, select OK.

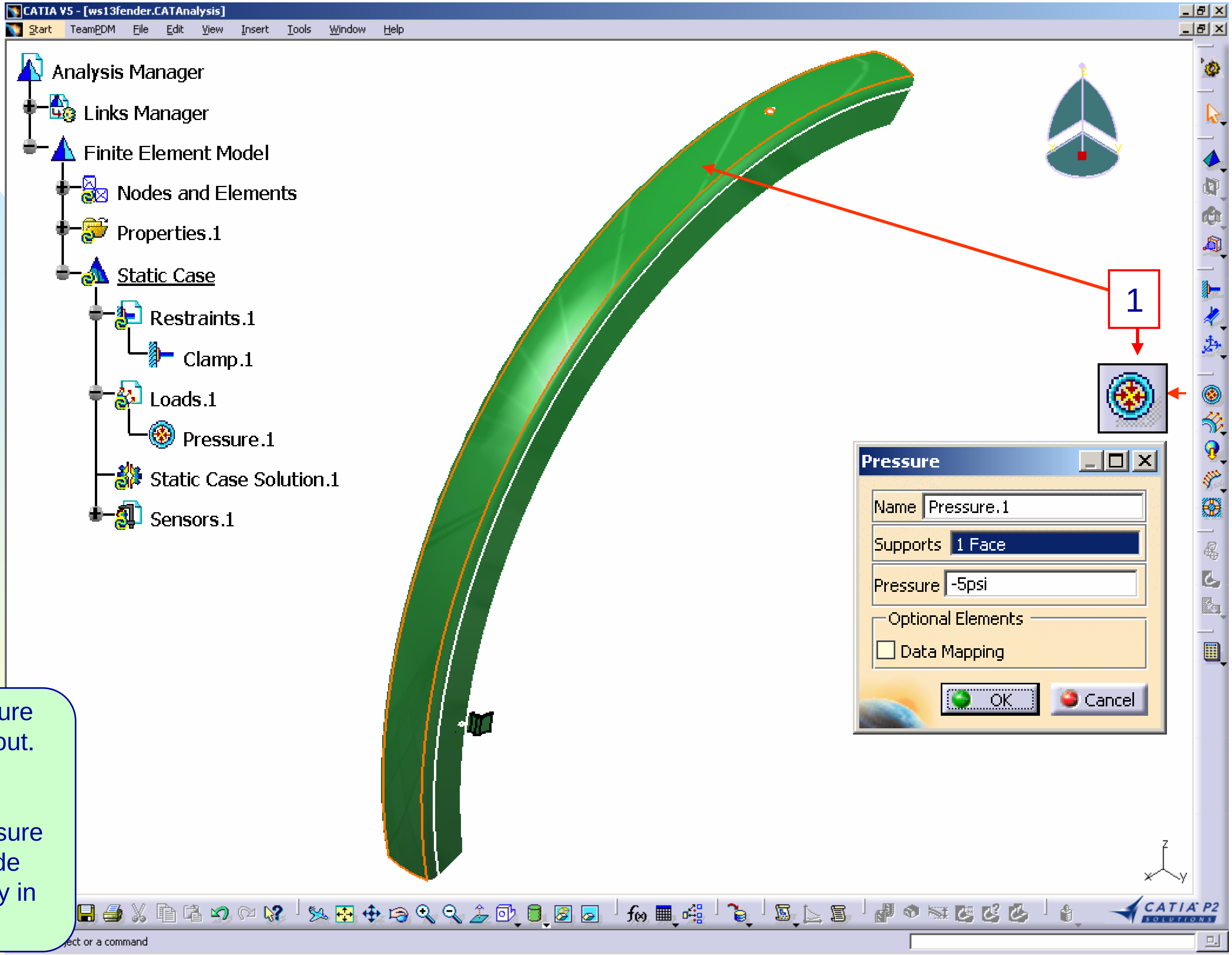
Step 9. Apply a clamp restraint



Steps:

1. Apply a clamp restraint to all the mounting holes.

Step 10. Apply a pressure force

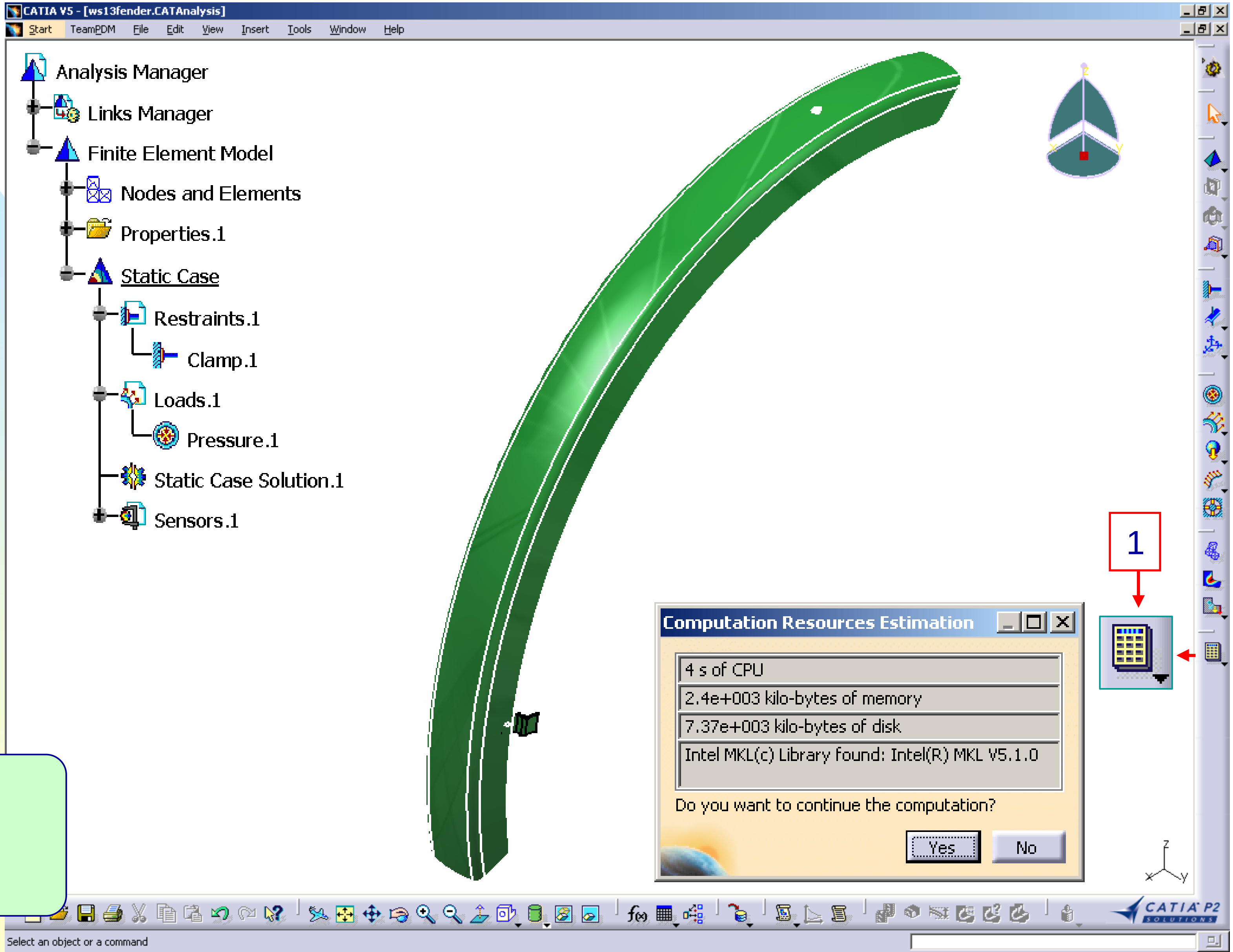


Assume the pressure is from the inside out.

Steps:

1. Select the Pressure icon and the outside face as shown, key in -5psi.

Step 11. Compute all

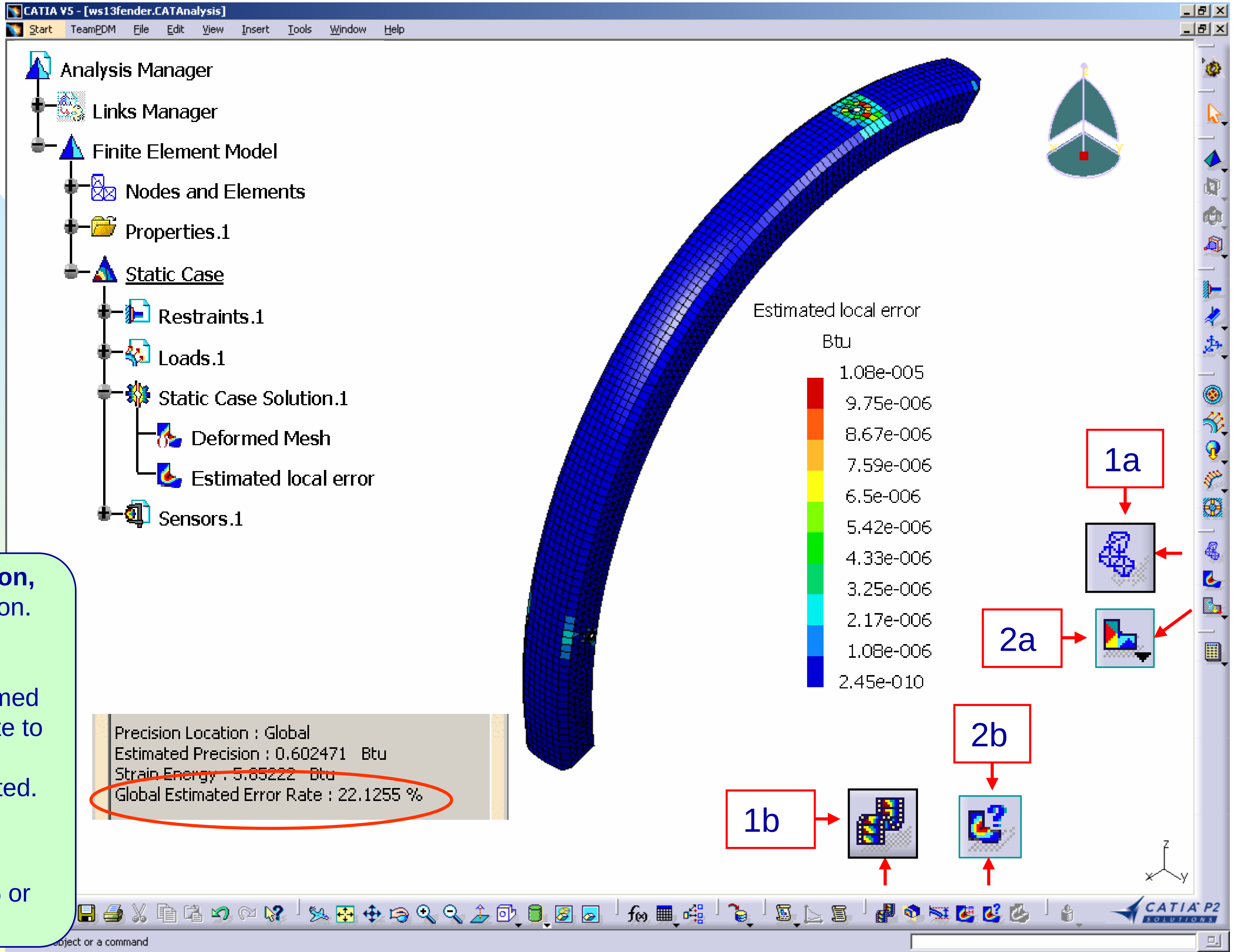


Save first.

Steps:

1. Compute All.

Step 12. Check global precision



Check **Deformation,** and global precision.

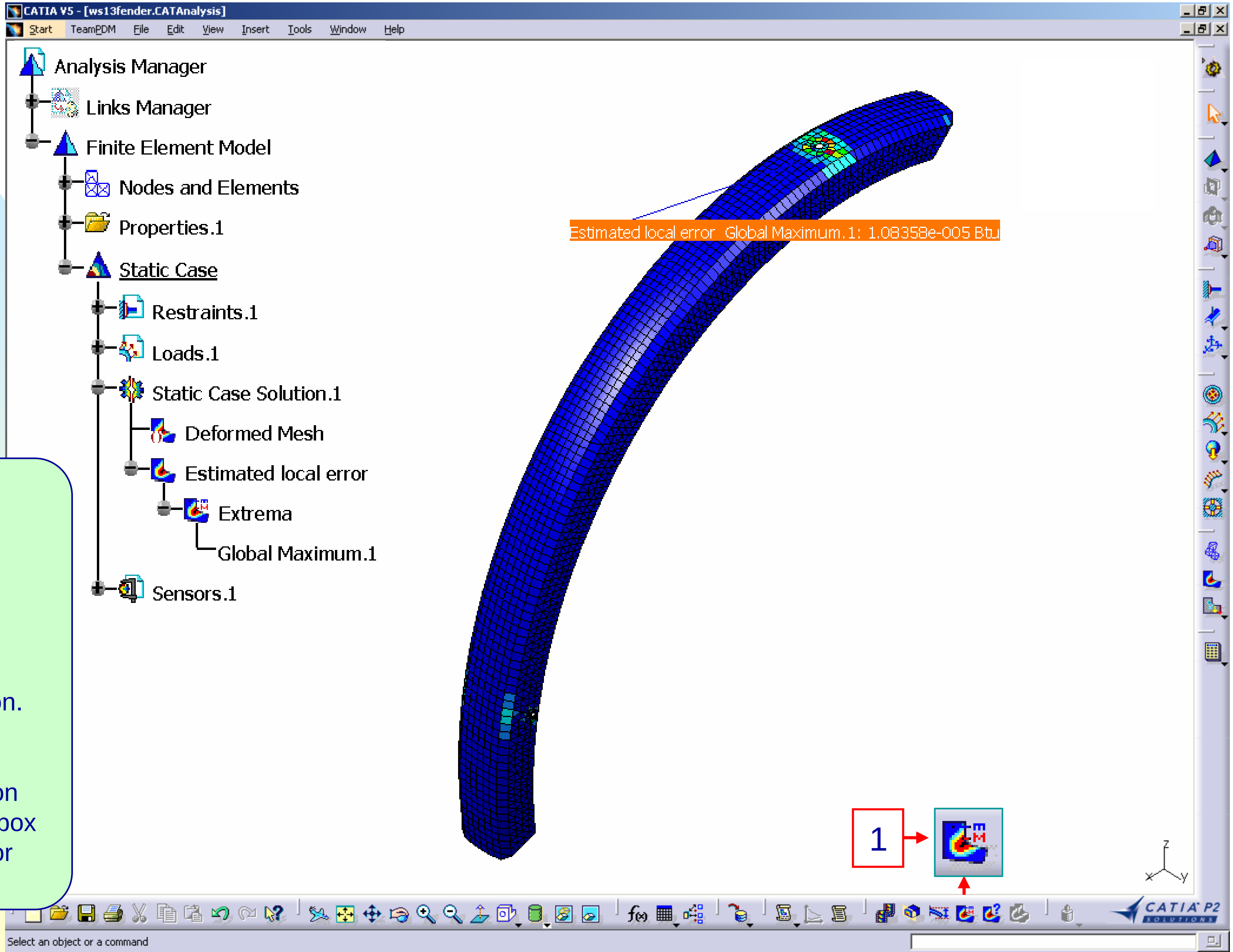
Steps:

1. Create a deformed image and animate to verify your part deflects as expected.

2. Check Global precision. Recommend 20% or less.

MSC SOFTWARE

Step 12. Check global precision



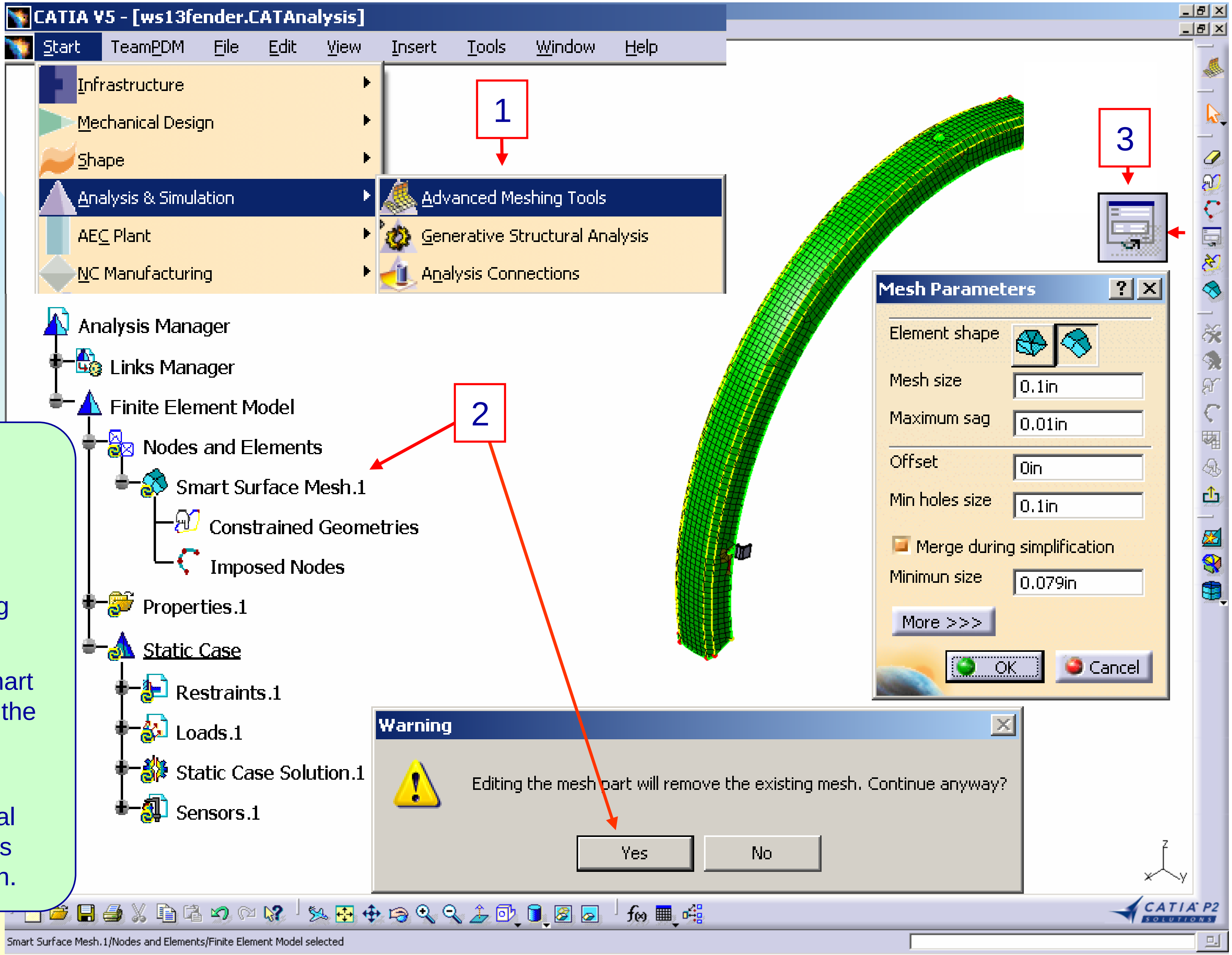
Find the global element with the highest estimated error.

Steps:

1. Use the Search Image Extrema icon.

Note: local precision and the adaptivity box are not available for surface FEM.

Step 13. Refine mesh and re-compute



Refine the global surface mesh.

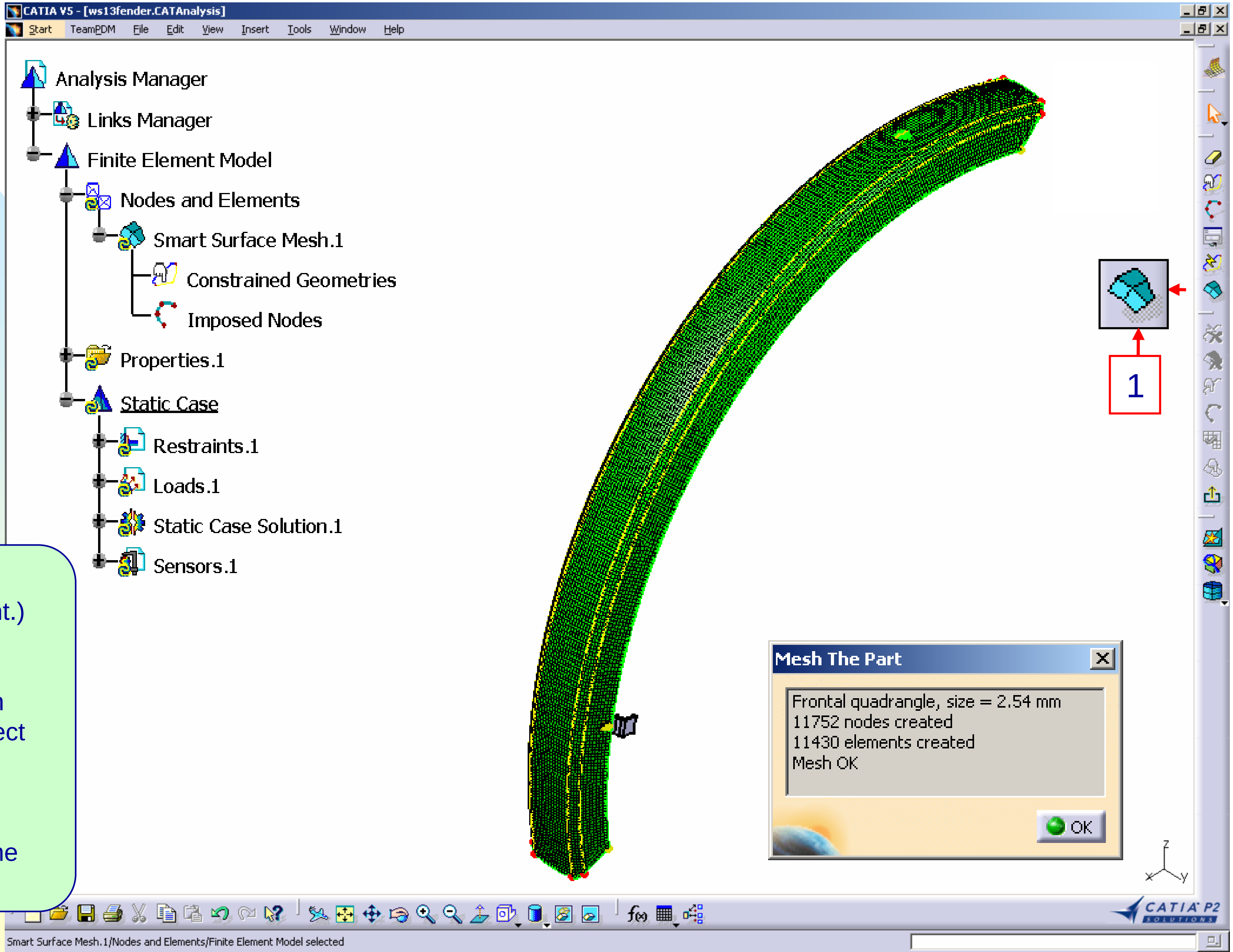
Steps:

1. Launch the Advanced Meshing Tools workbench.

2. Double click Smart surface Mesh.1 in the features tree, then select Yes.

3. Select the Global Meshing Properties icon, edit as shown.

Step 13. Refine mesh and re-compute



Refine the global surface mesh (cont.)

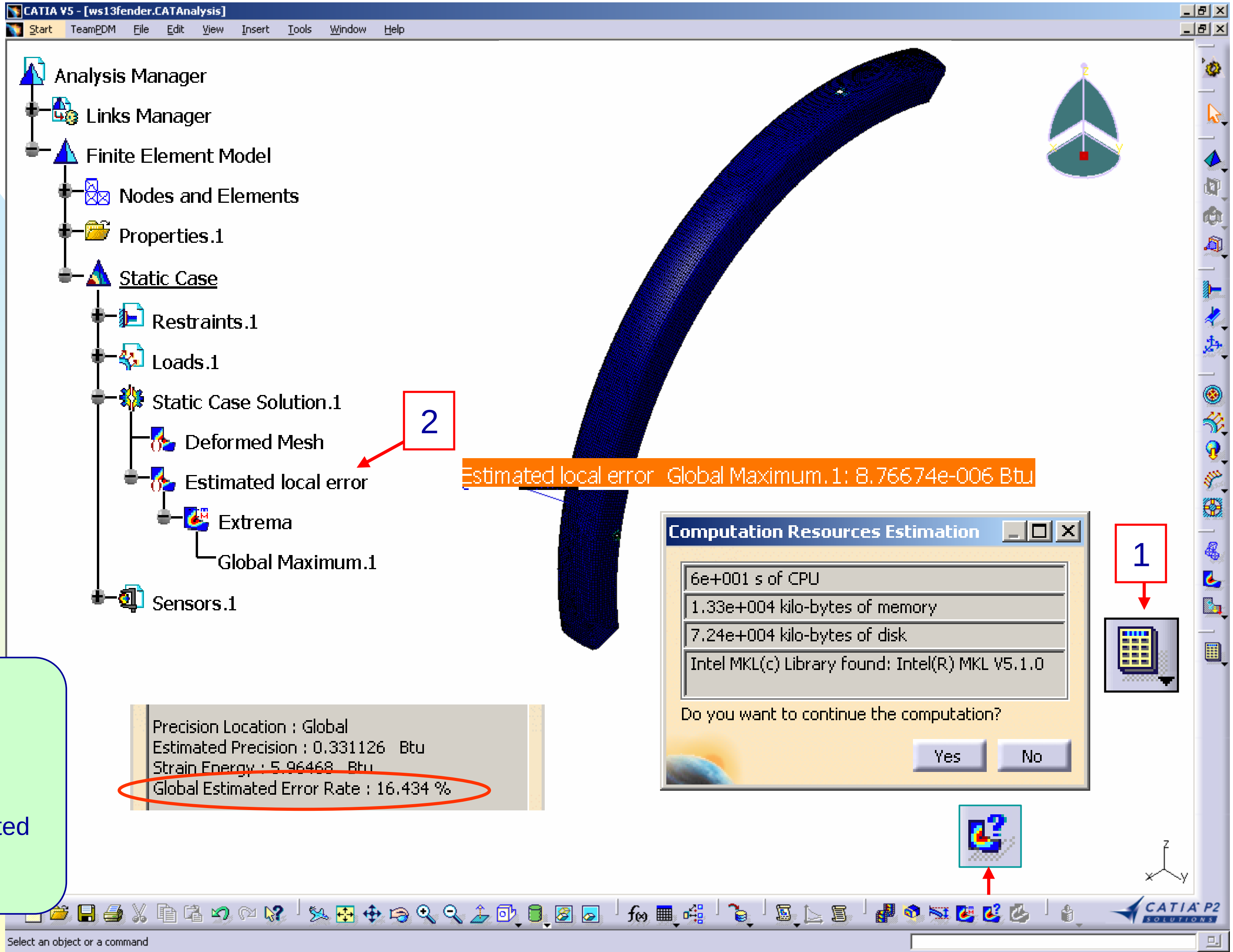
Steps:

1. Select the Mesh The Part icon, select OK.

Then go back to the GSA workbench.

MSC SOFTWARE

Step 14. Visualize final results



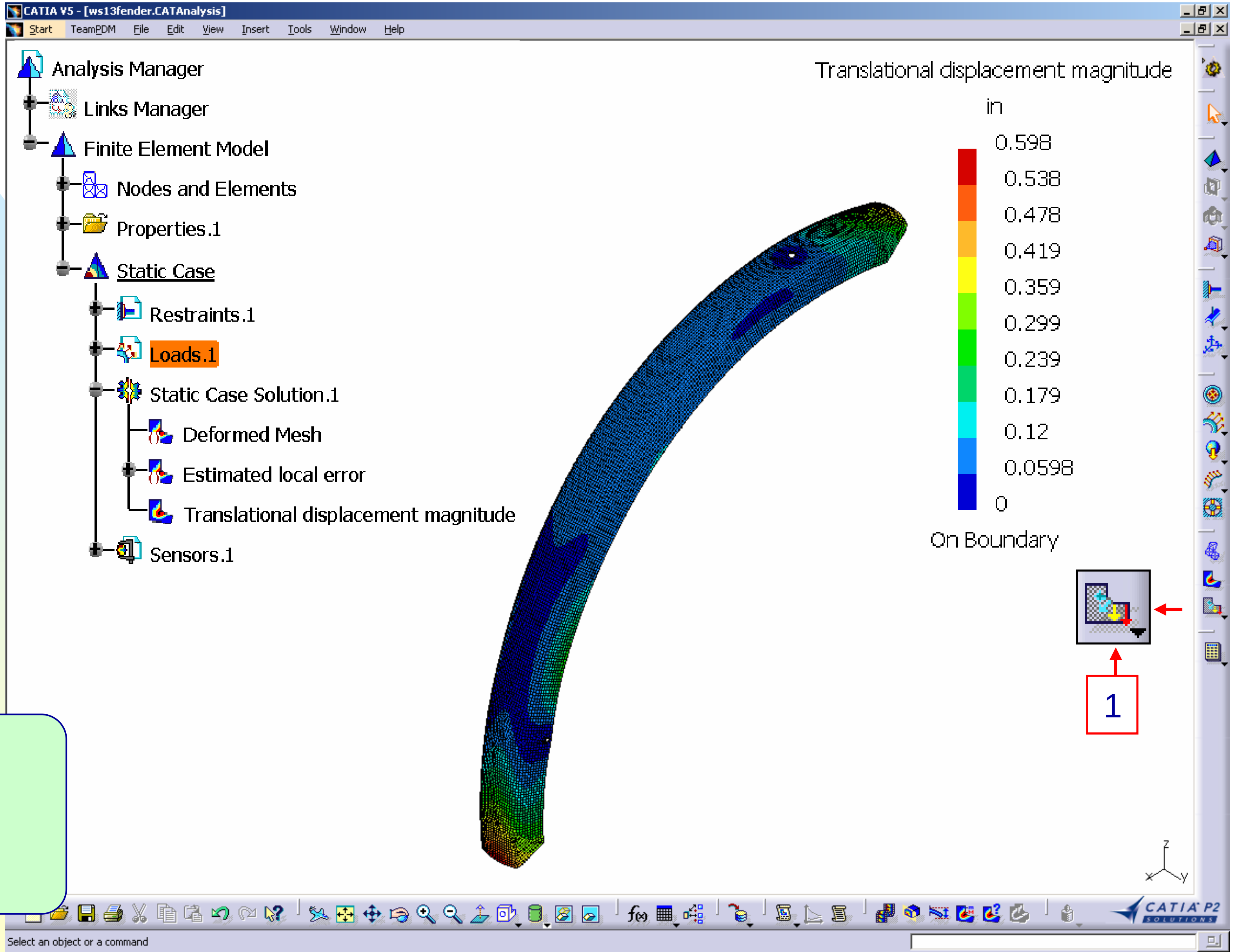
Save first.

Steps

1. Compute All.

2. Find the Estimated Global error again. Good, below 20%.

Step 14. Visualize final results

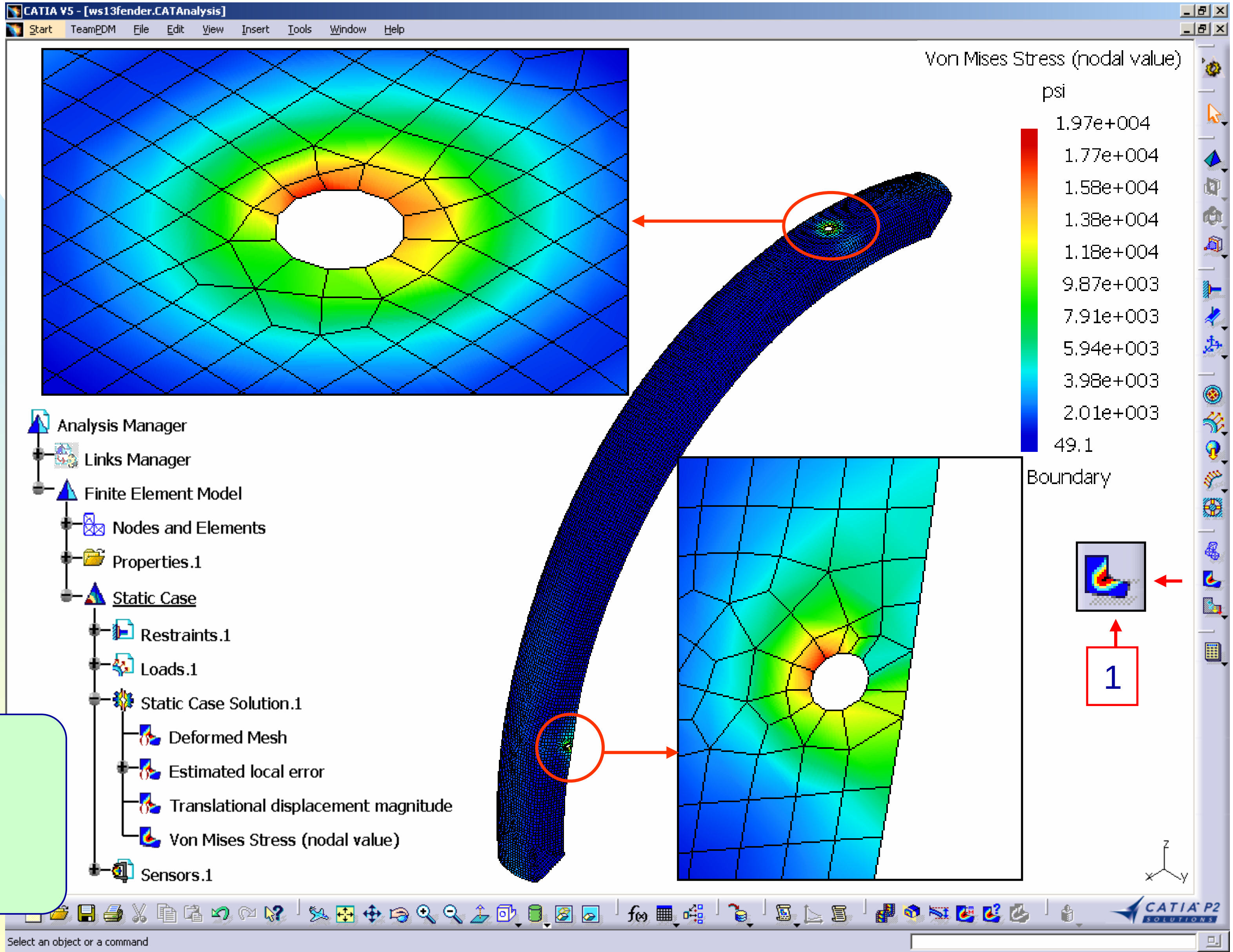


Find the maximum deflection.

Steps

1. Select the displacement icon.

Step 14. Visualize final results



Find the maximum Von Mises Stress.

Steps

1. Select the Von Mises icon.

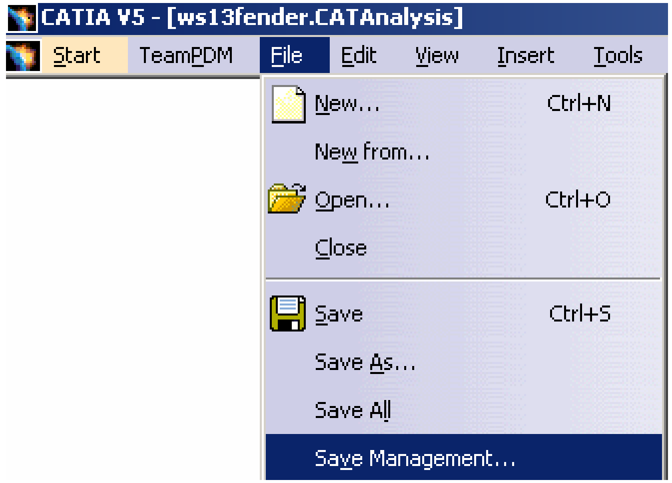
Conclusions

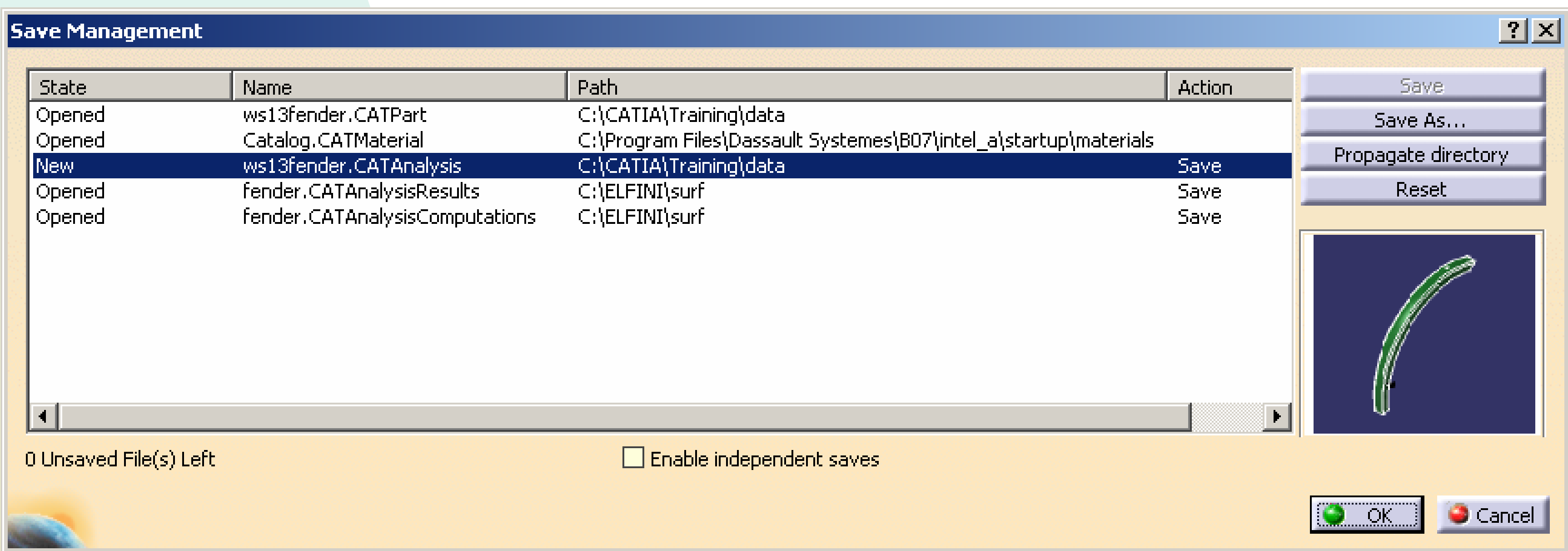
- This fender requires stiffening in the mounting hole areas.

Global % Precision error	**16.4%**
Error Estimate	**8.7e-6 Btu**
Von Mises Stress	**19,700 psi**
Maximum Displacement	**0.598 inch**

Step 15. Save the analysis document

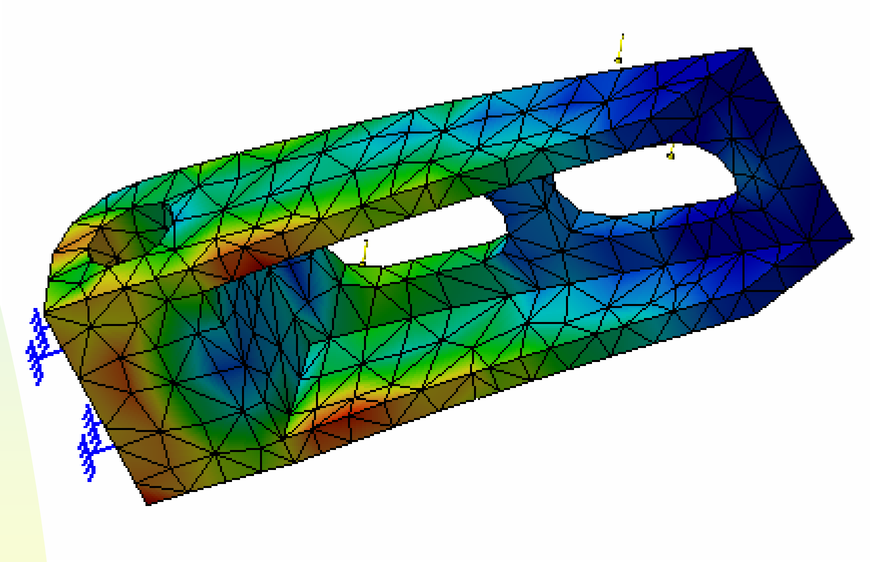
Save your documents





WORKSHOP 14

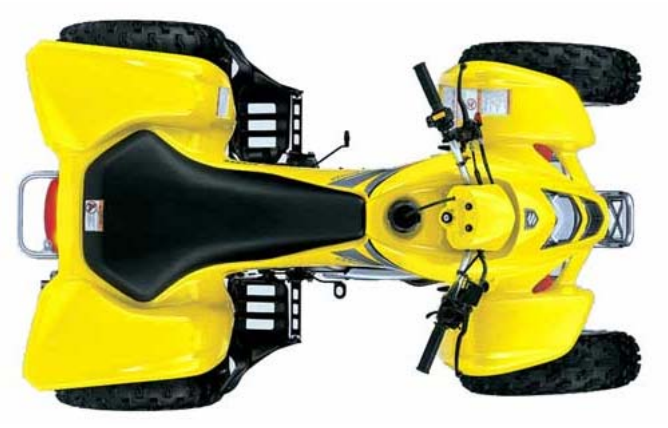
KNOWLEDGEWARE



Problem Description

- The preliminary design of the ATV Foot Peg must be completed as soon as possible and must meet the given structural requirements. The design must not exceed the material yield strength under loading and it must not deform in a manner causing interference with other parts of the vehicle.

- An initial static analysis of the Foot Peg has been completed (Workshop 2). To assist in our design iterations, we need to activate CATIA Knowledgeware capabilities to provide immediate feedback on the critical analysis parameters.



Suggested Exercise Steps

1. Open the existing document for the Foot Peg static analysis.
2. Create analysis sensors for maximum displacement and maximum stress.
3. Create a knowledge rule for maximum displacement.
4. Create a knowledge check for maximum stress.
5. Modify the Foot Peg design to meet requirements.
6. Compute the analysis for the modified design.
7. View results.

Step 1. Open the analysis document

Open the Foot Peg static analysis document from the training directory.

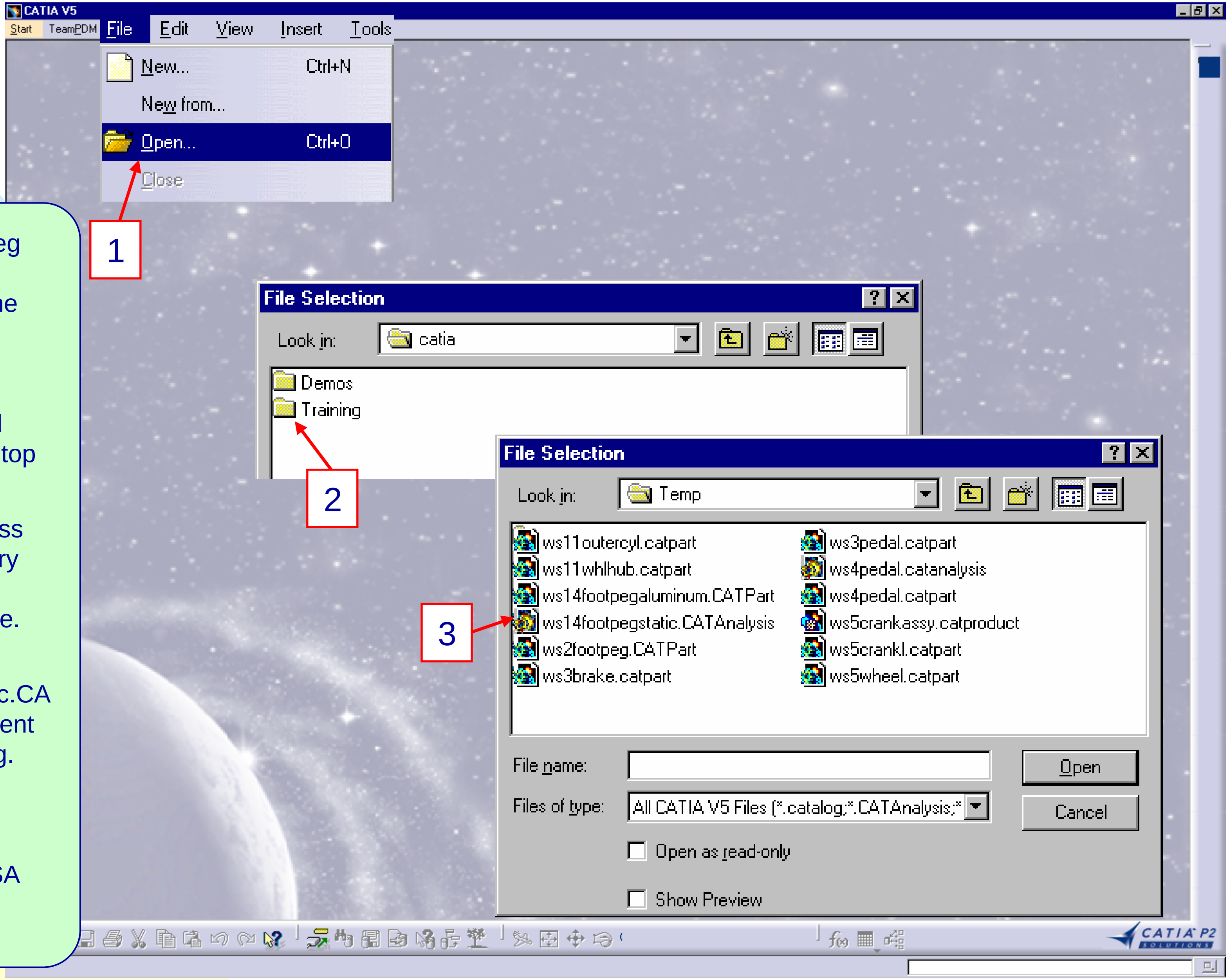
Steps:

1. Select File and Open… from the top pull-down menu.

2. Access the class workshop directory using the typical Windows interface.

3. Open the ws14footpegstatic.CATAnalysis document by double-clicking.

The document is opened in the GSA workbench.



Step 2. Create analysis sensors

Analysis sensors must be created to provide results information to the Knowledgeware application. Create a sensor for maximum displacement and for maximum stress.

Steps:

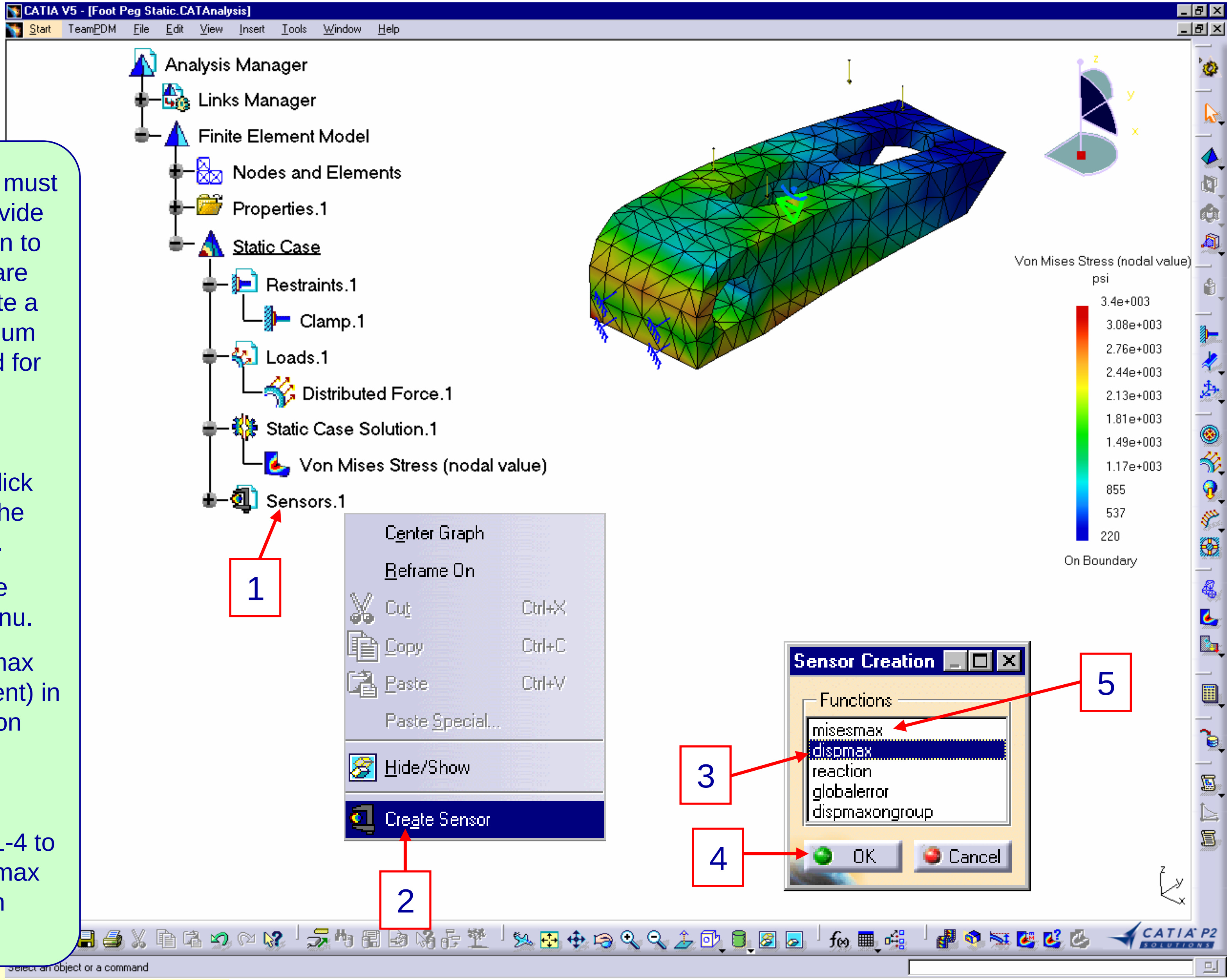
1. Right mouse click on Sensors.1 in the specification tree.

2. Click on Create Sensor in the menu.

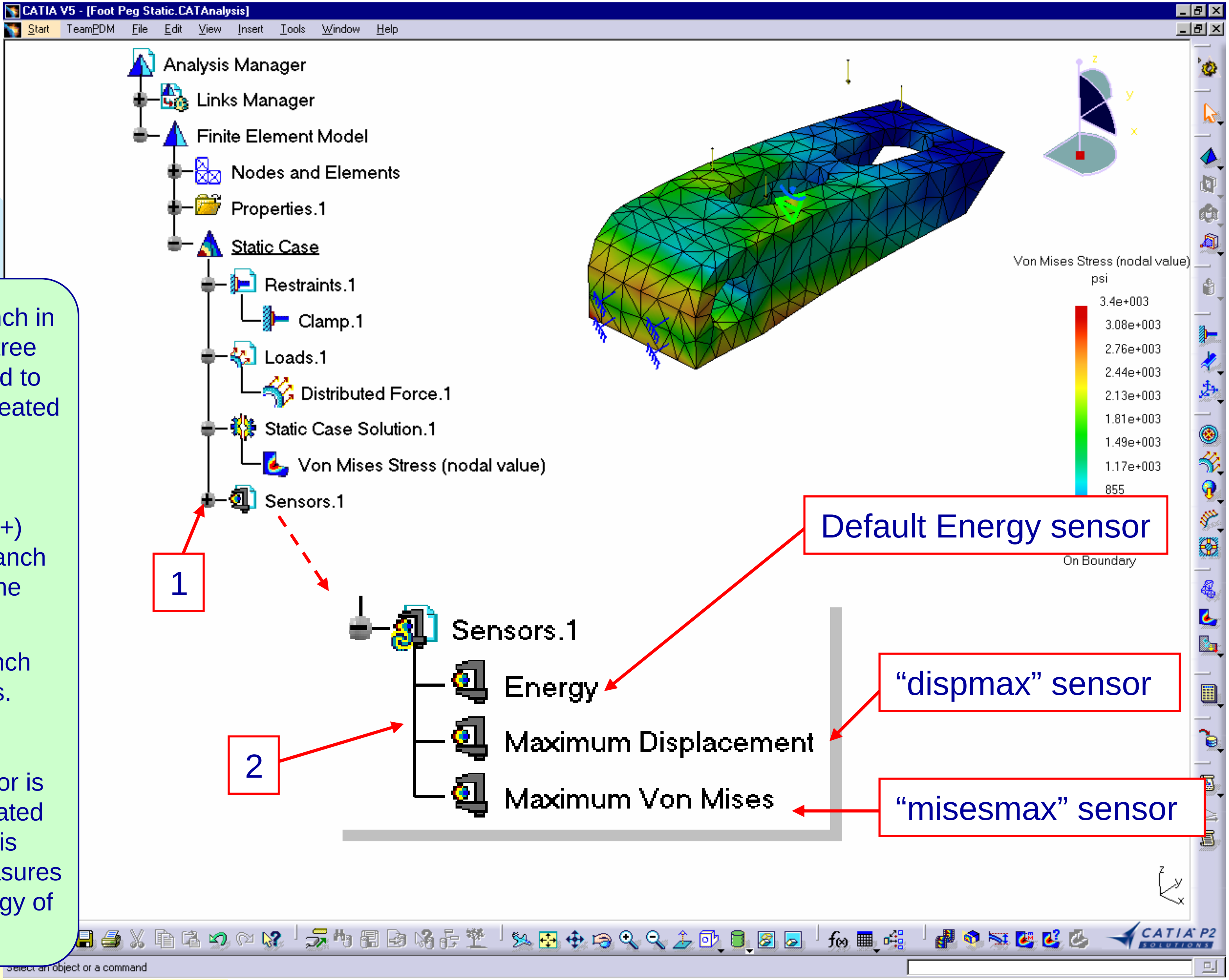
3. Highlight dispmax (max. displacement) in the sensor creation window.

4. Click OK.

5. Repeat steps 1-4 to create the misesmax sensor (max. Von Mises stress).



Step 2. Create analysis sensors



The sensors branch in the specification tree must be expanded to view the newly created sensors.

Steps:

1. Click the plus (+) symbol on the branch node to expand the sensors branch.

2. Expanded branch shows all sensors.

The Energy sensor is automatically created with every analysis document. It measures global strain energy of the structure.

Step 3. Create knowledge rule

Activate the ability to view knowledge rules and checks in the analysis specification tree.

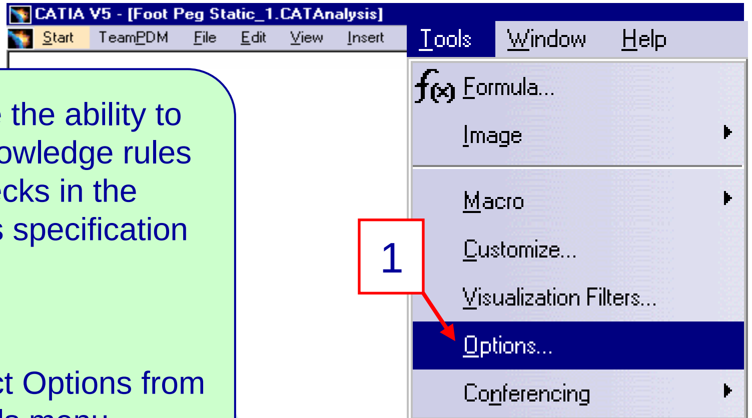
Steps:

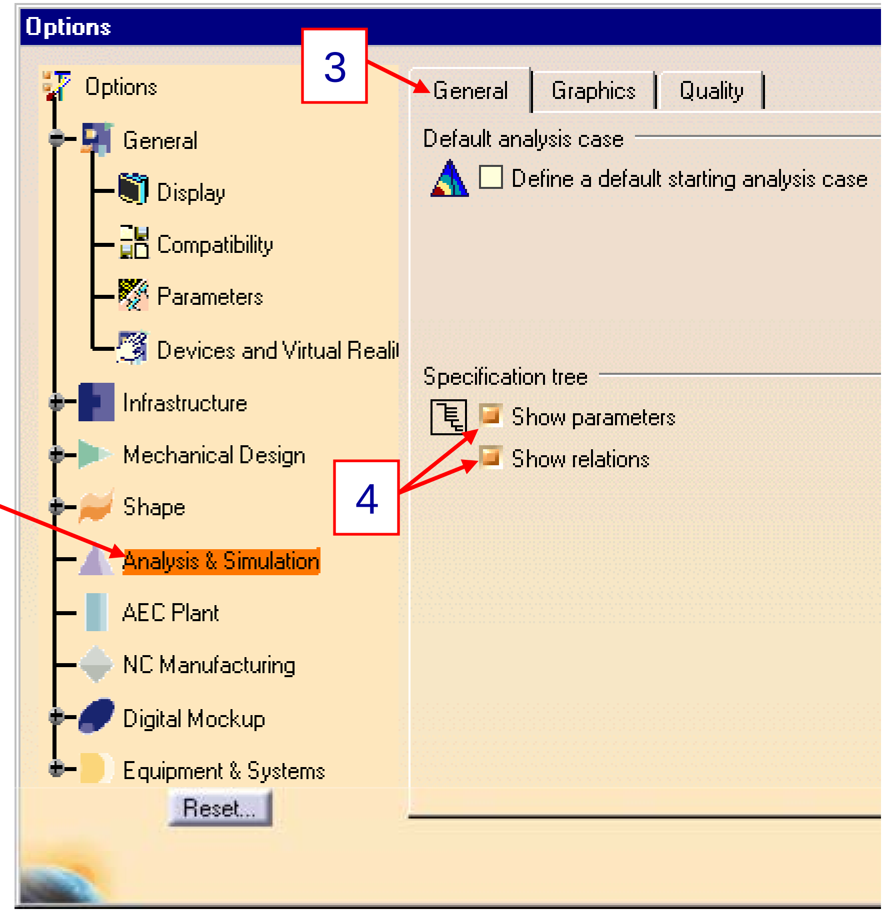
1. Select Options from the Tools menu.

2. Select Analysis & Simulation branch.

3. Select General tab.

4. Activate both boxes to show parameters and relations.





Update the analysis solution if needed.

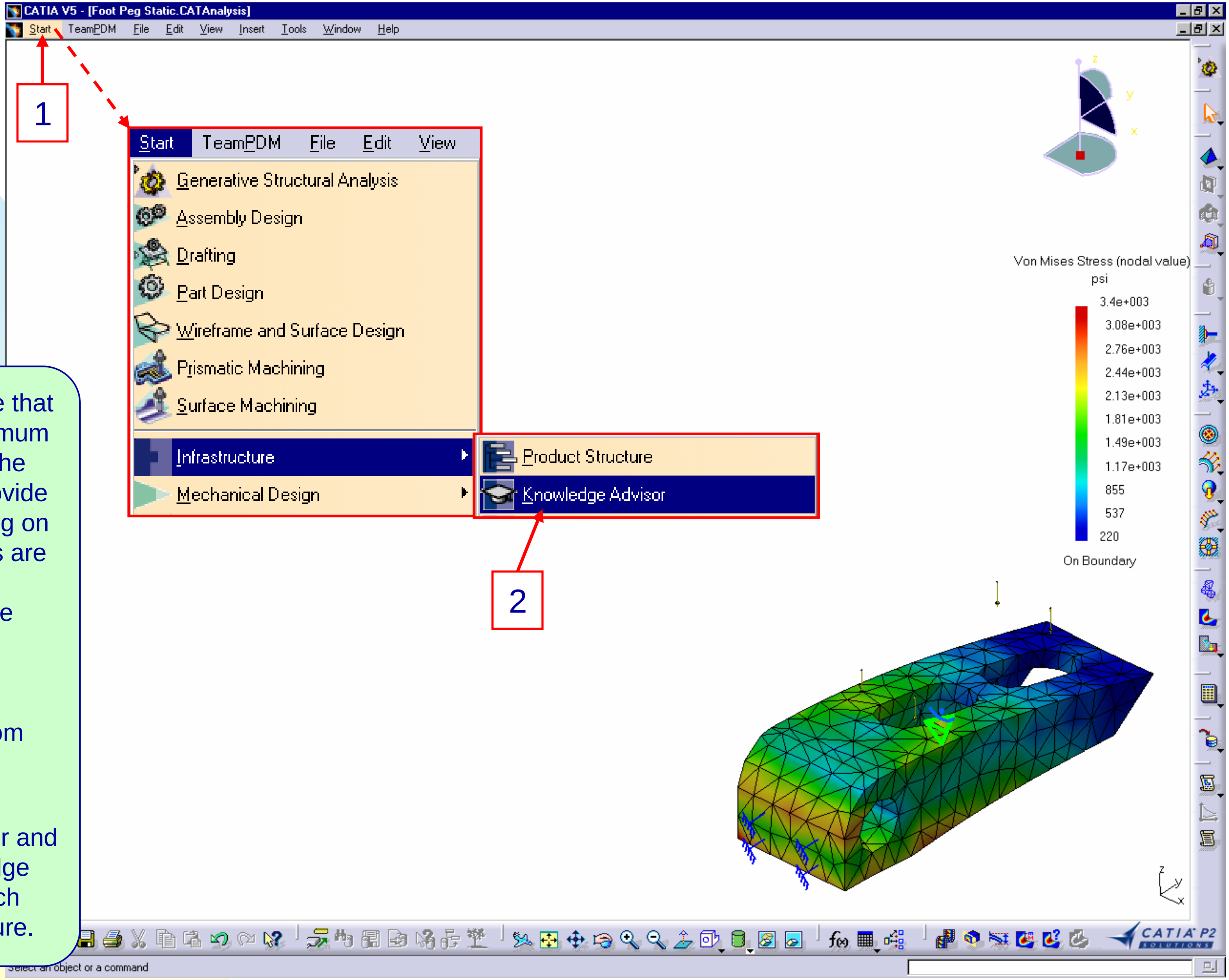
Steps:

1. Check for “update needed” symbol on the Static Case Solution.1

2. Compute to update the analysis results (see Section 3).

MSC SOFTWARE

Step 3. Create knowledge rule



Now create a rule that will monitor maximum displacement of the Foot Peg and provide pop-up messaging on the screen. Rules are created using the CATIA Knowledge Advisor.

Steps:

1. Select Start from the top pull-down menu.

2. Drag the cursor and click the Knowledge Advisor workbench under Infrastructure.

Step 3. Create knowledge rule

The Knowledge Advisor workbench should now be active .

Steps:

1. Click the Rule icon from the Knowledge Advisor workbench.

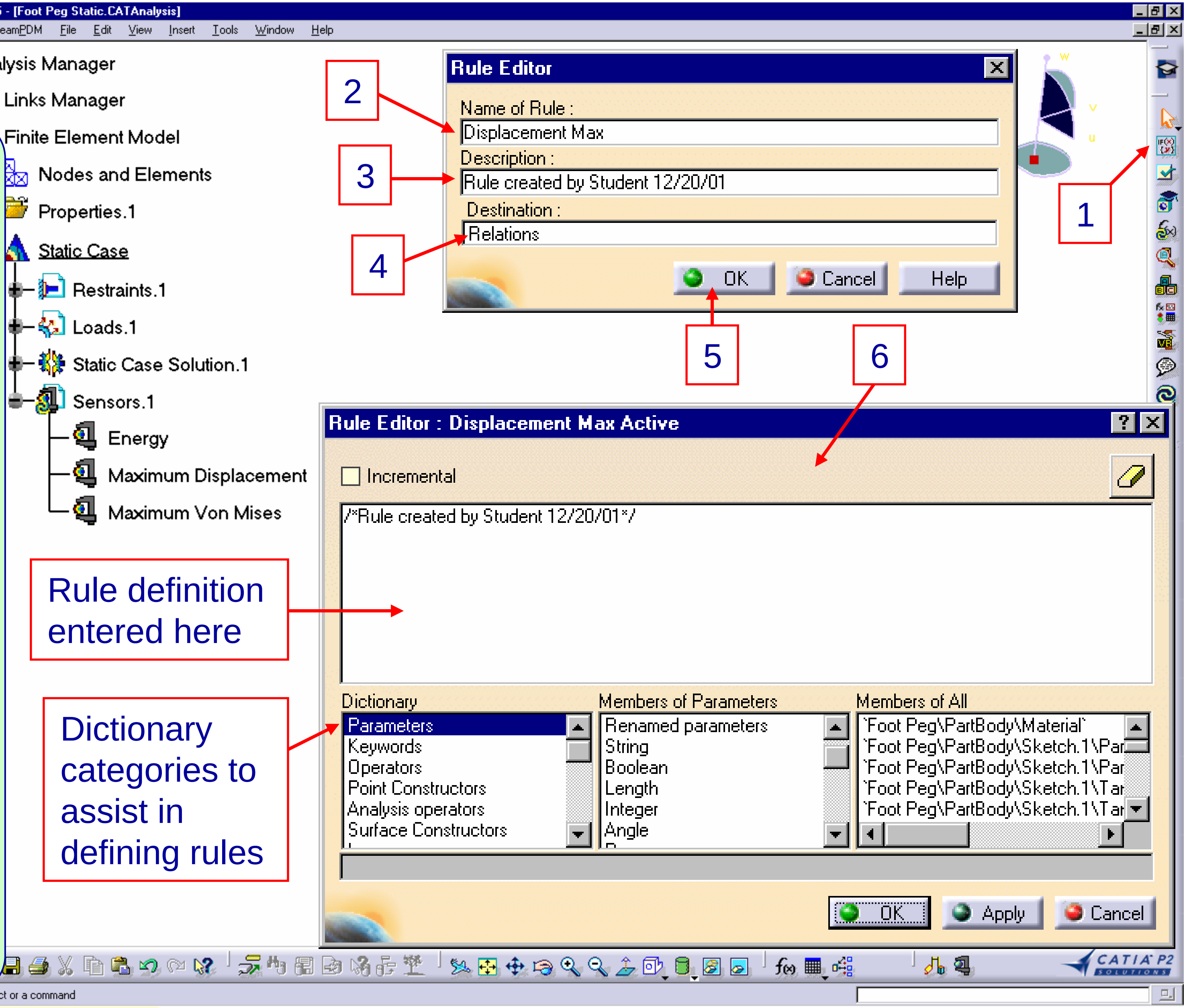
2. Key “Displacement Max” as the name of the rule.

3. Key in a description for the rule or accept the default.

4. The rule will be saved under the Relations category – do not modify.

5. Click OK.

6. Rule Editor window displays the active rule (Displacement Max).



Step 3. Create knowledge rule

Define the rule.

Steps:

1. Click to place the cursor at the end of the 1st line and then hit the Enter key to start a new line.

2. Select Keywords in the Dictionary window.

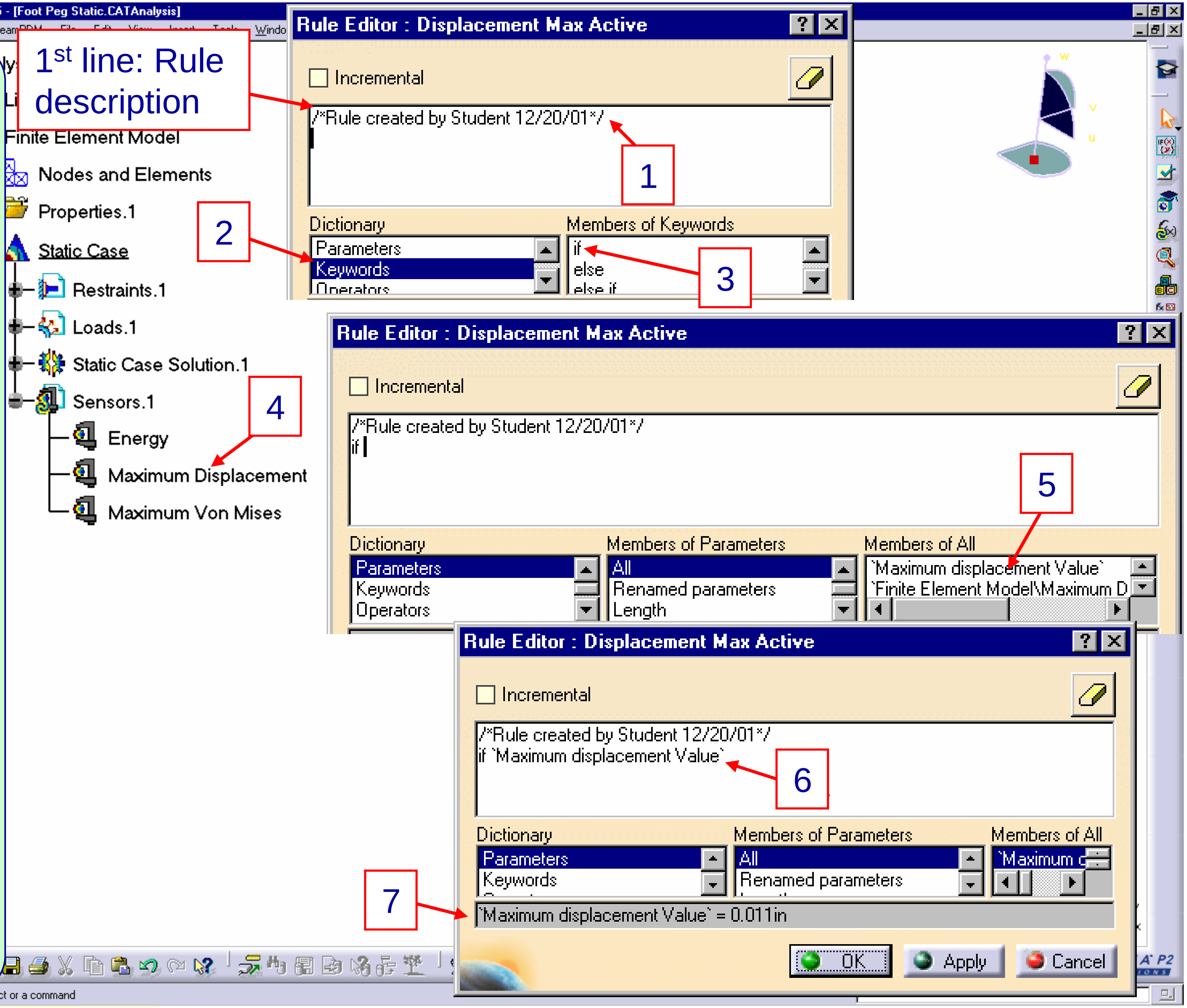
3. Double-click on “if” to begin the line.

4. Single-click the Max Displacement sensor in the tree to list its parameters in the Members of All area.

5. Double-click on ‘Maximum displacement Value’ to add it to the definition.

6. The parameter for the max. displacement value is added.

7. The current value is shown (.011 in).



Step 3. Create knowledge rule

Define the rule (cont.).

Steps:

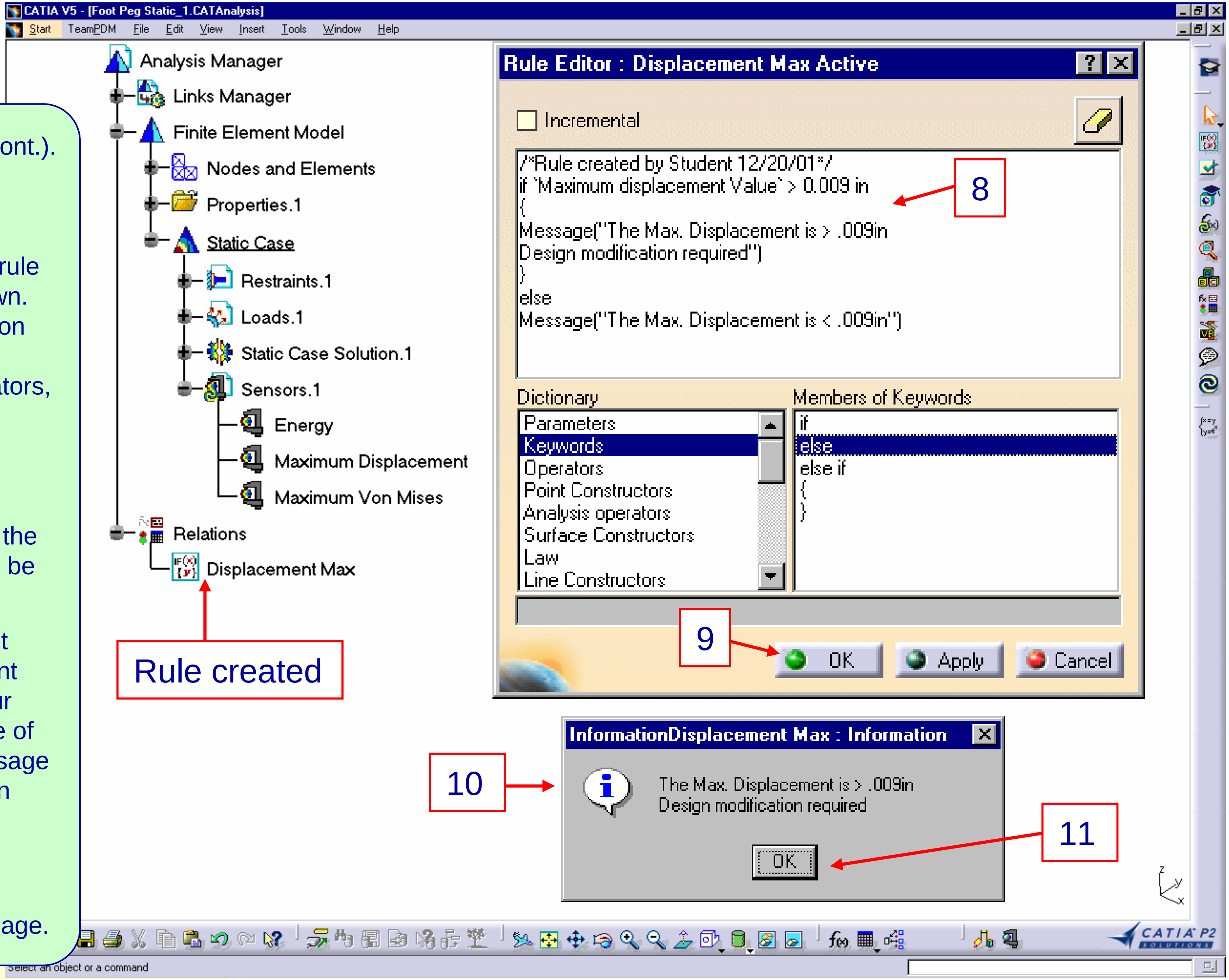
8. Key in the remainder of the rule definition as shown. Dictionary selection can be used for Keywords, Operators, Messages, etc.

9. Click OK when finished.

10. If successful, the rule message will be displayed.

Note: The current max. displacement value exceeds our defined rule value of .009 in. The message suggests a design modification is required.

11. Click OK to dismiss the message.



Step 4. Create knowledge check

Create a check that will monitor maximum Von Mises stress on the Foot Peg so that our design does not exceed the material yield strength.

Steps:

1. Click the Check icon from the Knowledge Advisor workbench.

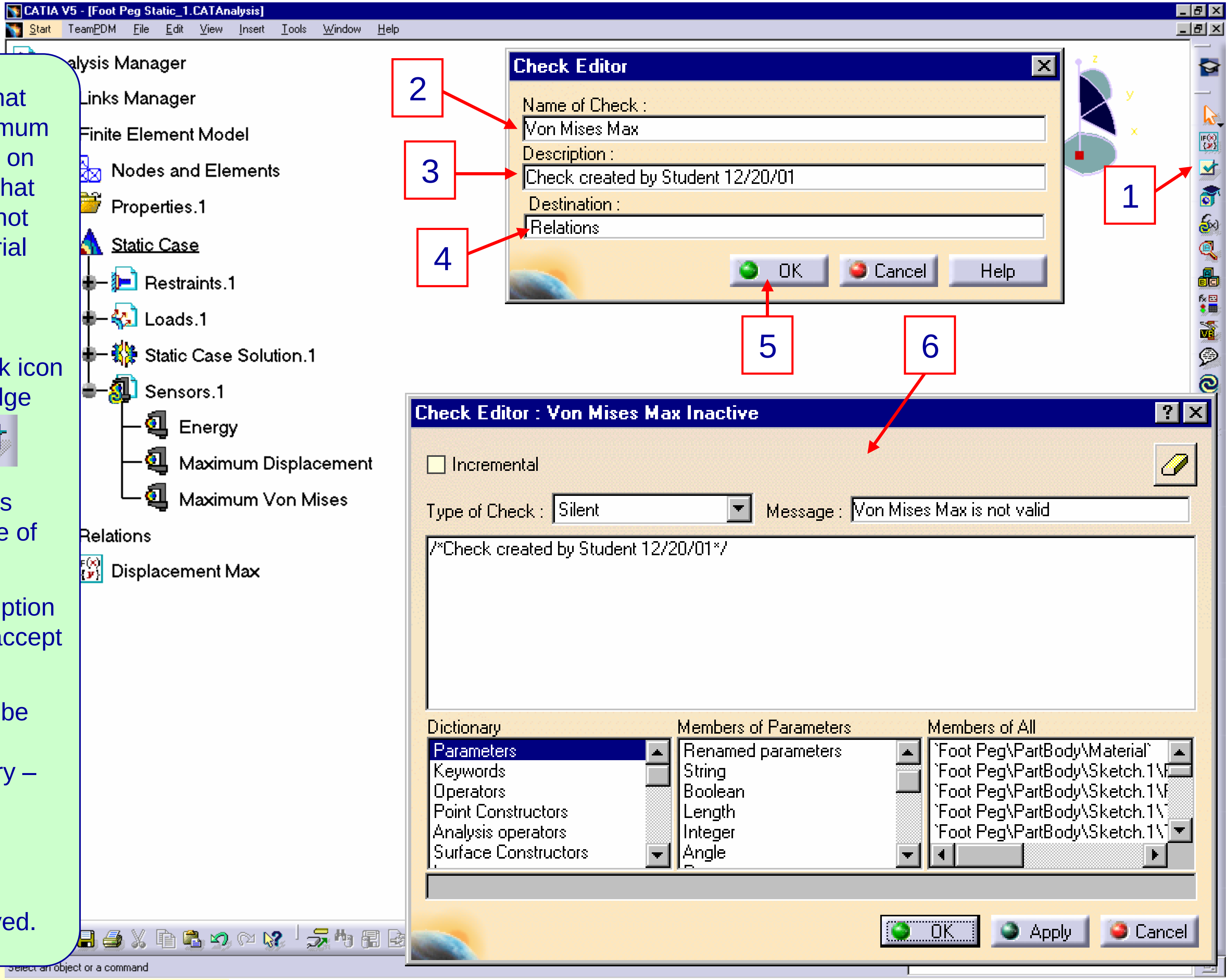
2. Key "Von Mises Max" as the name of the check.

3. Key in a description for the check or accept the default.

4. The check will be saved under the Relations category – do not modify.

5. Click OK.

6. Check Editor window is displayed.



Step 4. Create knowledge check

Define the check.

Steps:

1. Select Warning as the check type.

2. Key warning message as shown.

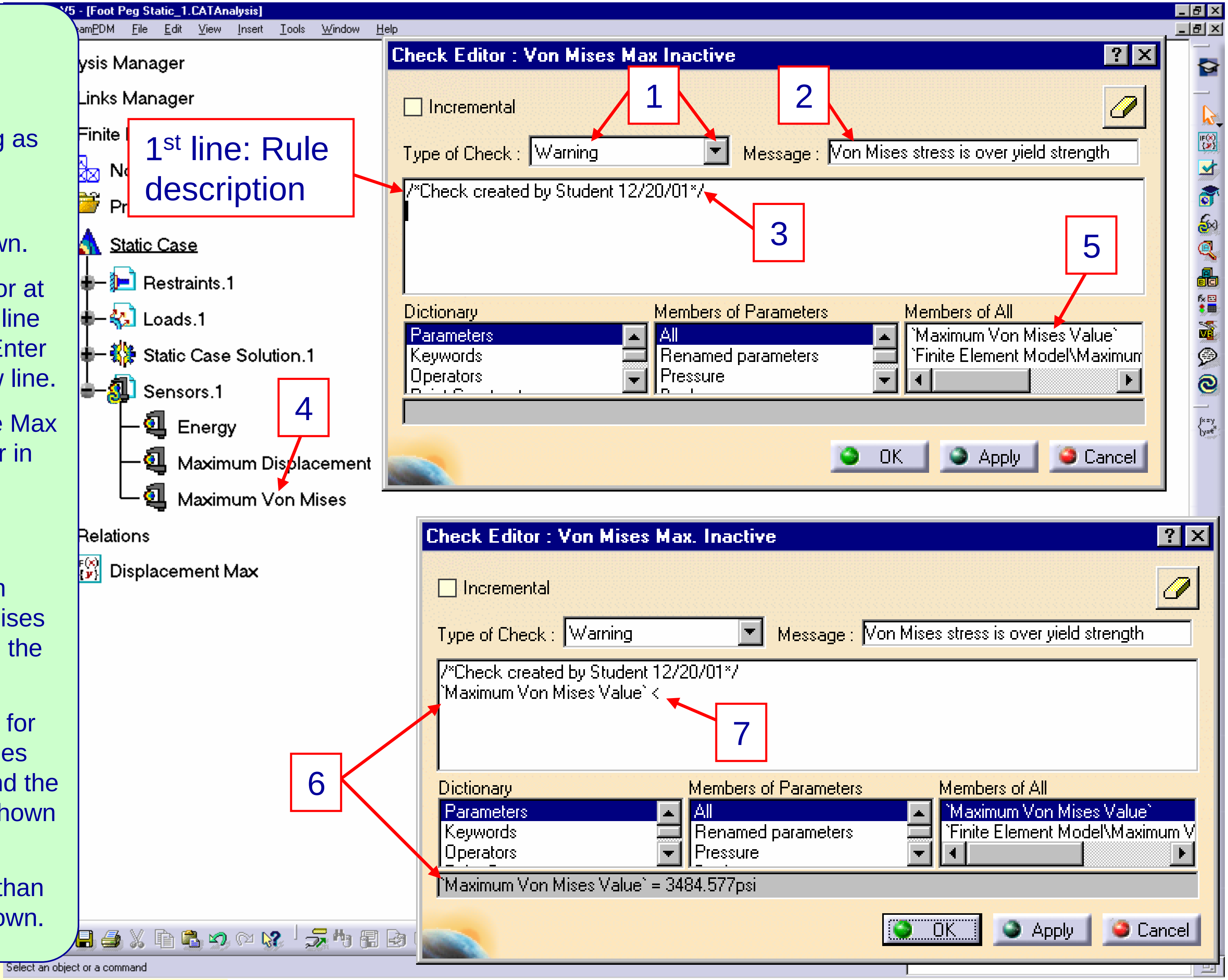
3. Place the cursor at the end of the 1st line and then hit the Enter key to start a new line.

4. Single-click the Max Von Mises sensor in the tree to list its associated parameters.

5. Double-click on 'Maximum Von Mises Value' to add it to the definition.

6. The parameter for the max. Von Mises value is added and the current value is shown (3484.577 psi).

7. Enter the less than symbol (<) as shown.



Step 4. Create knowledge check

Define check (cont.).

Steps:

8. Expand the tree to view the Foot Peg.

9. Single-click the Foot Peg to list associated parameters.

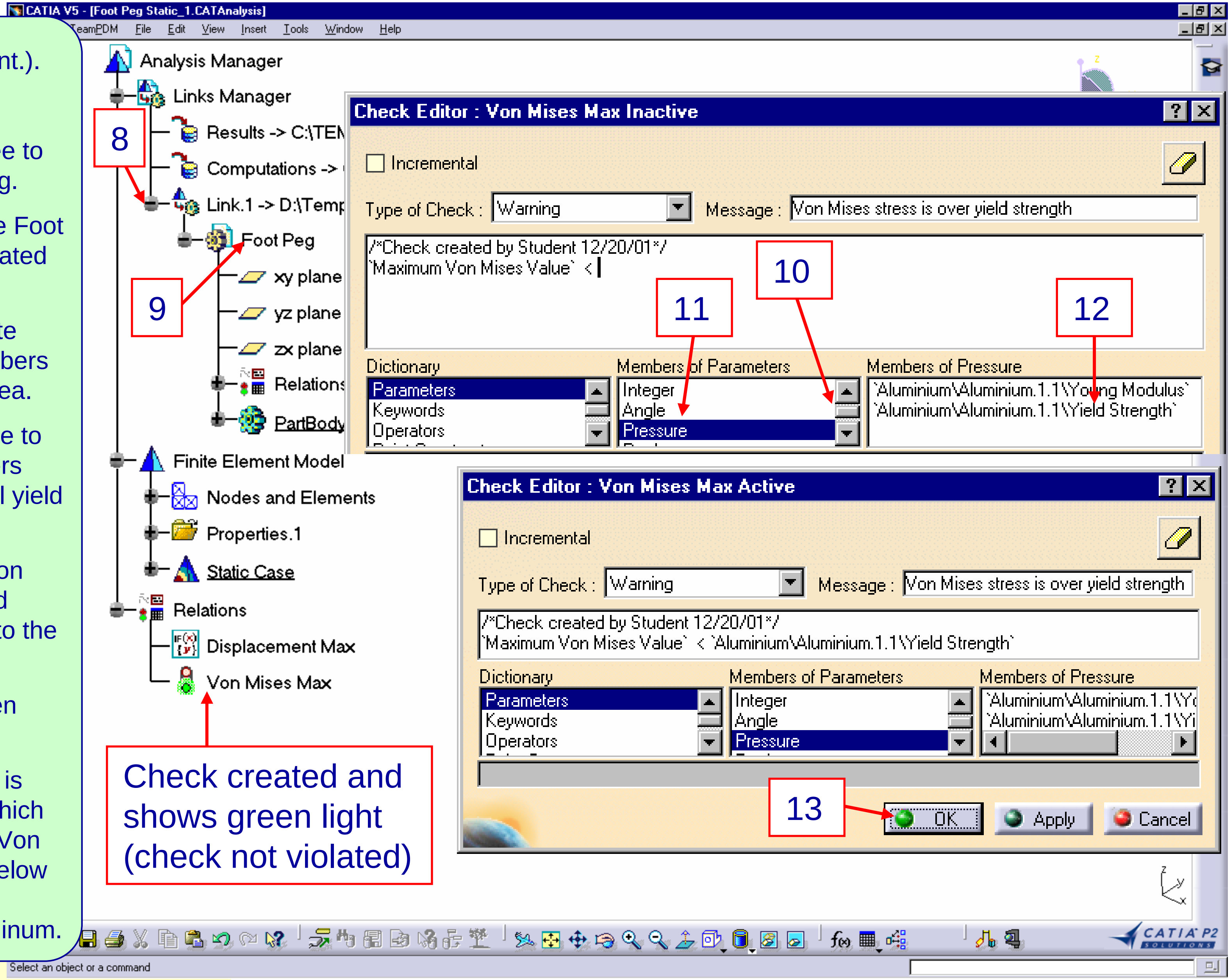
10. Scroll to locate Pressure in Members of Parameters area.

11. Click Pressure to display parameters including material yield strength.

12. Double-click on ‘Aluminum…Yield Strength’ to add to the definition.

13. Click OK when finished.

Note: The check is showing green which means the max. Von Mises stress is below the material yield strength for Aluminum.



Step 5. Modify Foot Peg design

The knowledgeware results indicate the maximum Von Mises stress is acceptable, however the maximum displacement of the Foot Peg is too large.

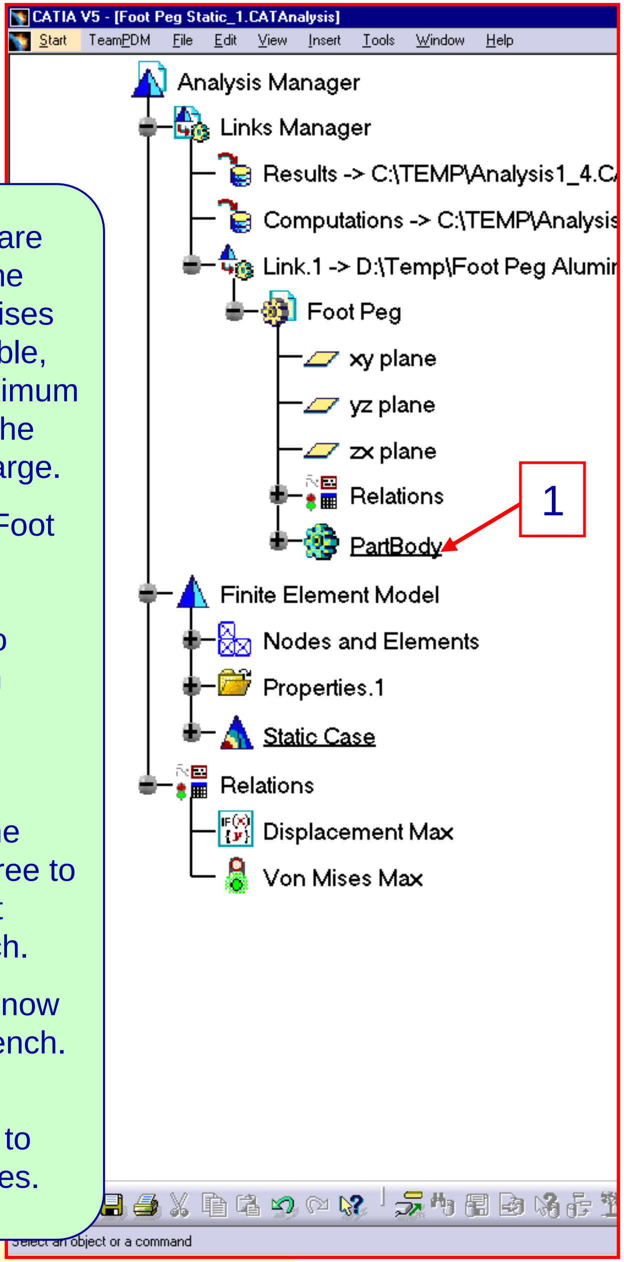
Let's modify the Foot Peg design as suggested by the knowledge rule to reduce maximum displacement.

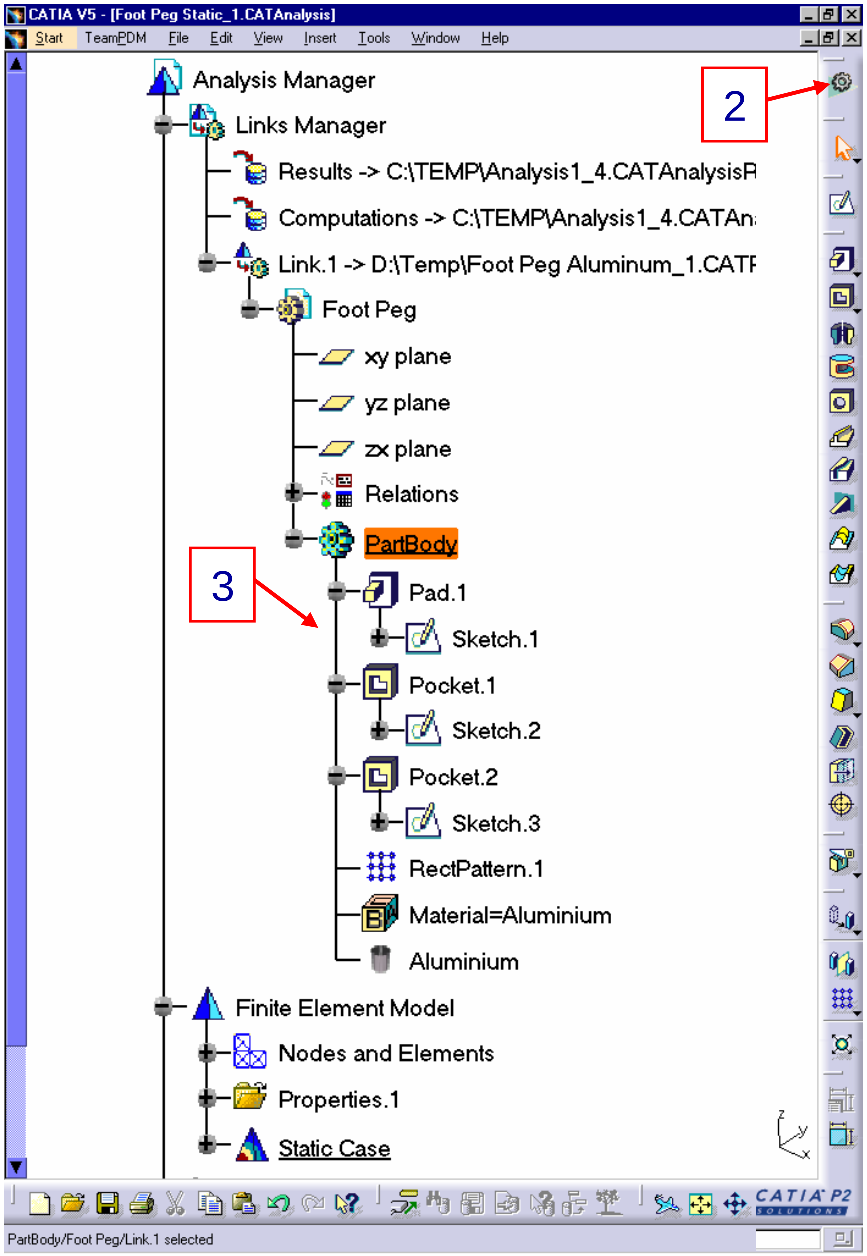
Steps:

1. Double-click the PartBody in the tree to switch to the Part Design workbench.

2. Part Design is now the active workbench.

3. Expand the PartBody branch to show solid features.





Step 5. Modify Foot Peg design

Now in Part Design, reduce the length of the Foot Peg by modifying the corresponding solid feature.

Steps:

1. Double-click Pad.1 in the tree to modify. Pad Definition window is displayed.

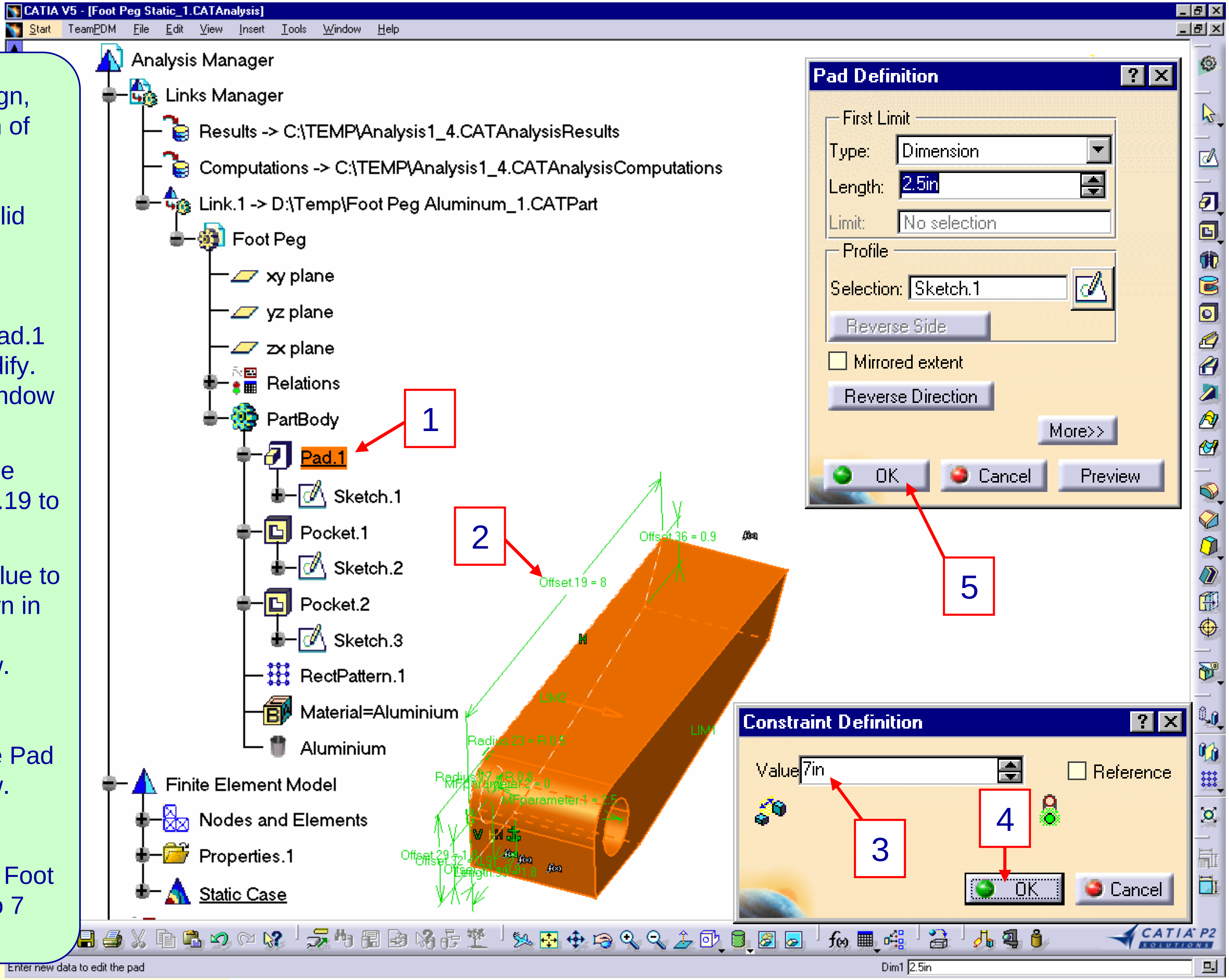
2. Double-click the parameter Offset.19 to modify its value.

3. Change the value to 7 inches as shown in the Constraint Definition window.

4. Click OK.

5. Click OK in the Pad Definition window.

The length of the Foot Peg is reduced to 7 inches.



Step 5. Modify Foot Peg design

After making the change, it is apparent that the outer edge is too thin. Modify the spacing between the top face cutouts.

Steps:

1. Double-click the feature pattern (RectPattern.1) in the tree to modify.

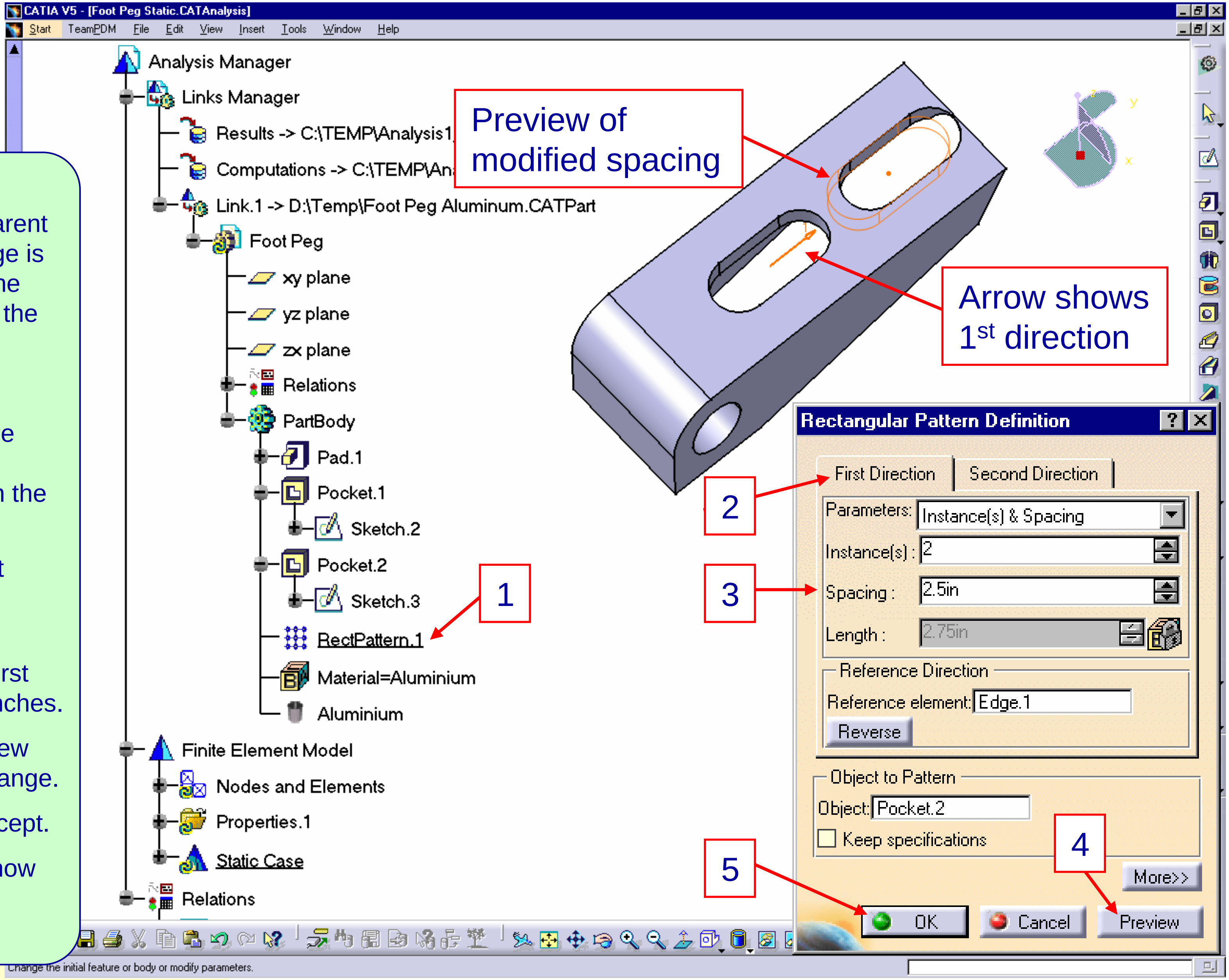
2. Select the First Direction tab.

3. Key in a new spacing for the First Direction of 2.5 inches.

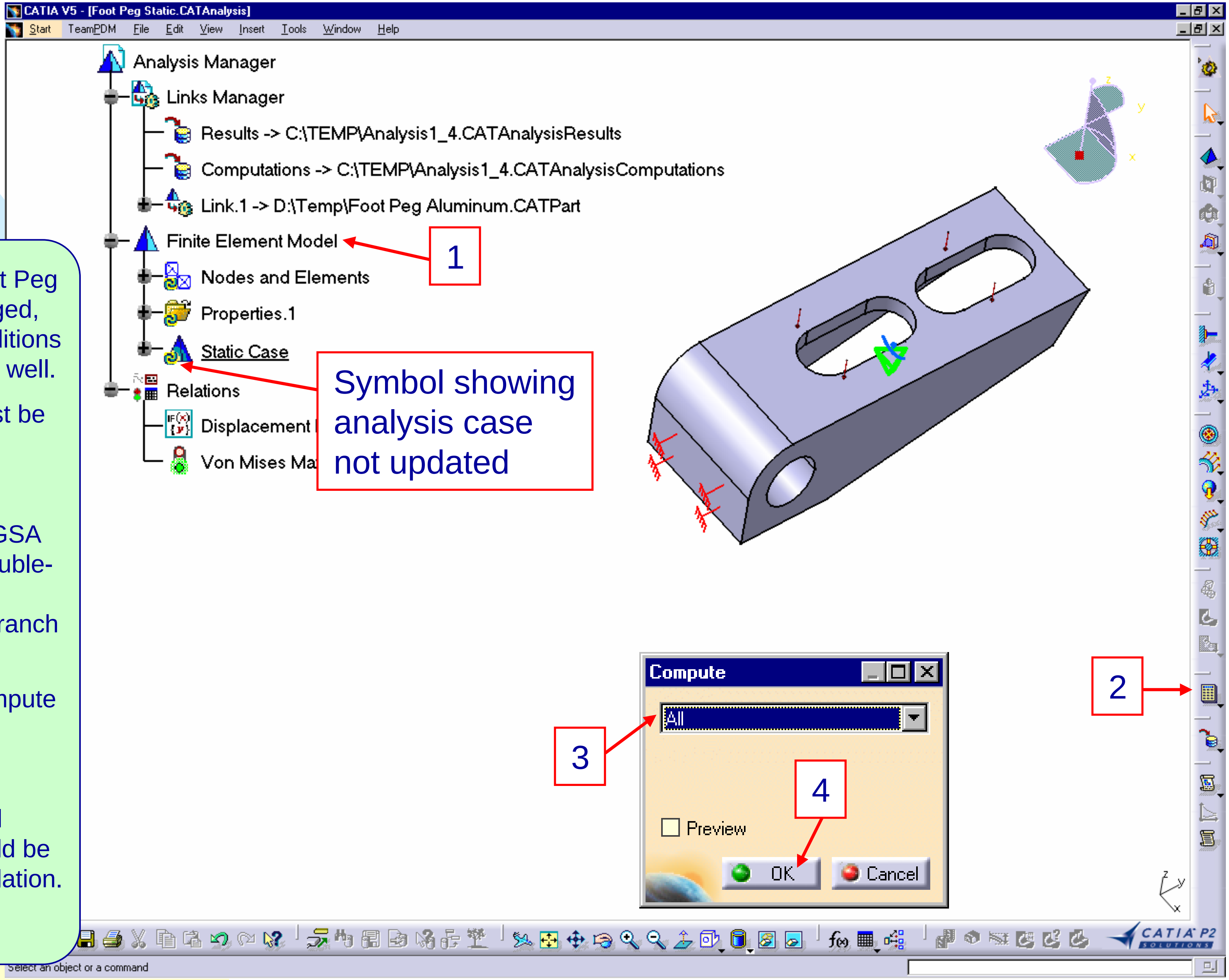
4. Click the Preview button to view change.

5. Click OK to accept.

The cutouts are now positioned closer together.



Step 6. Compute analysis



Now that the Foot Peg design has changed, our analysis conditions have changed as well.

The analysis must be computed again.

Steps:

1. Return to the GSA workbench by double-clicking the Finite Element Model branch in the tree.

2. Select the Compute icon.

3. Specify that All parameters should be used in the calculation.

4. Click OK.

Step 7. View results

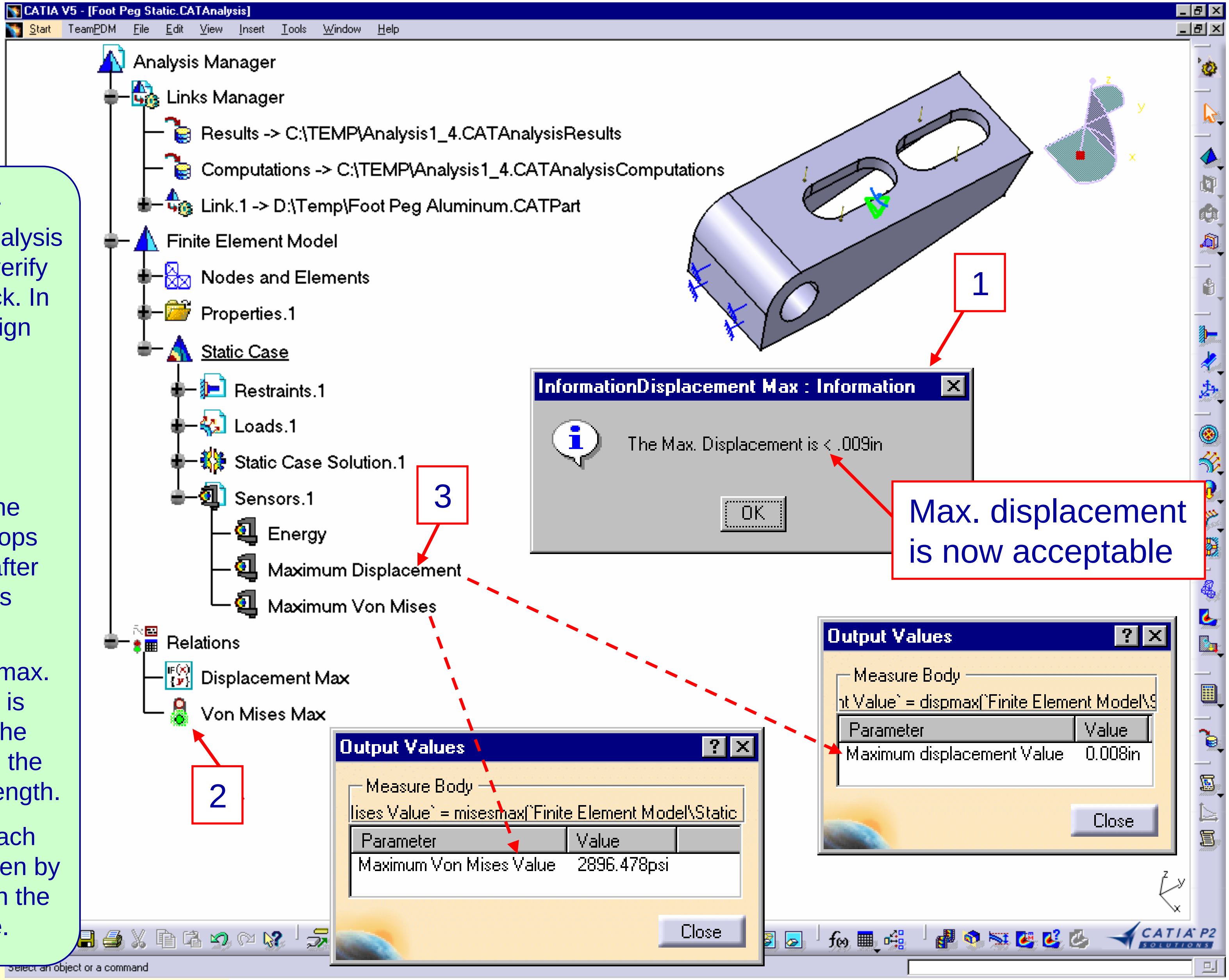
Immediately after computing the analysis solution we can verify our rule and check. In this case our design modifications are successful.

Steps:

1. A message generated from the knowledge rule pops onto the screen after the computation is complete.

2. The check for max. Von Mises stress is green indicating the value is less than the material yield strength.

3. The value of each sensor can be seen by double-clicking on the sensor in the tree.



MSC SOFTWARE