

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

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.

User Documentation: **Copyright© 2002 MSC.Software Corporation.** Printed in U.S.A. All Rights Reserved.

This notice shall be marked on any reproduction of this documentation, in whole or in part. Any reproduction or distribution of this document, in whole or in part, without the prior written consent of MSC.Software Corporation is prohibited.

MSC and MSC. are registered trademarks and service marks of MSC.Software Corporation. NASTRAN is a registered trademark of the National Aeronautics and Space Administration. MSC.Nastran is an enhanced proprietary version developed and maintained by MSC.Software Corporation. MSC.Patran is a trademark of MSC.Software Corporation.

All other trademarks are the property of their respective owners.

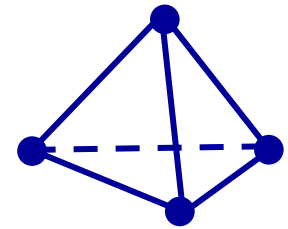
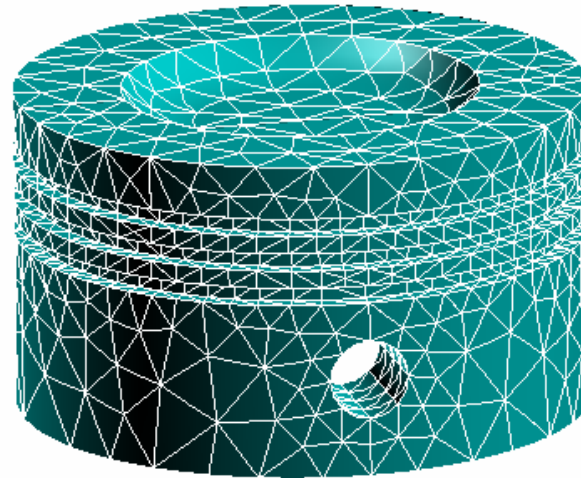
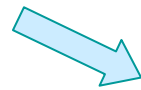
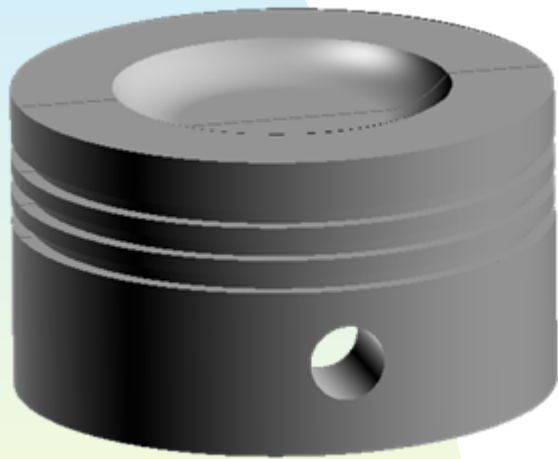
TABLE OF CONTENTS

<u>Workshop</u>	<u>Page</u>
1 FEM Review	1-3
2 Foot Peg	2-3
3 Bicycle Pedal Static Analysis.....	3-3
4 Bicycle Pedal Mesh Refinement and Adaptivity.....	4-3
5 Crank Analysis Using Virtual Parts.....	5-3
6 Rear Rack (Modal) Analysis.....	6-3
7 Seat Post Assembly Analysis.....	7-3
8 Rectangular Section Cantilever Beam.....	8-3
8b Z-Section Cantilever Beam.....	8b-3
9 Stress Concentration for a Stepped Flat Tension Bar.....	9-3
9b Torsion of a Shaft with a Shoulder Fillet.....	9b-3
10 Annular Plate.....	10-3
10b Rectangular Plate Small Concentric Circle Load.....	10b-3
11 Press Fit.....	11-3
12 Flat Plate Column Buckling	12-3
13 Bicycle Fender Surface Meshing.....	13-3
14 Knowledgeware.....	14-3



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 _____ .

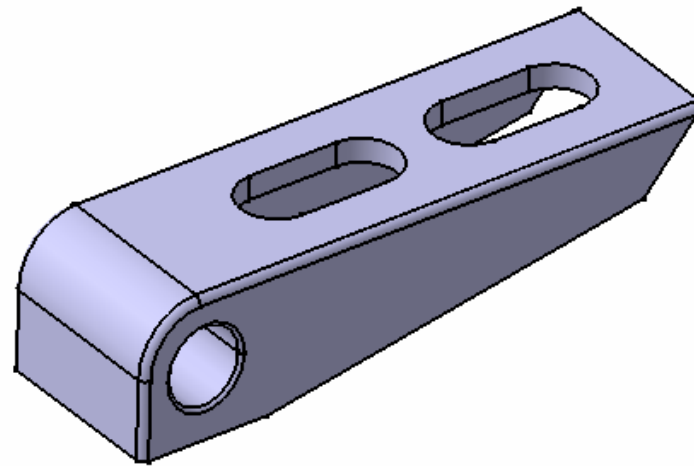
■ Quiz yourself on FEA:

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.

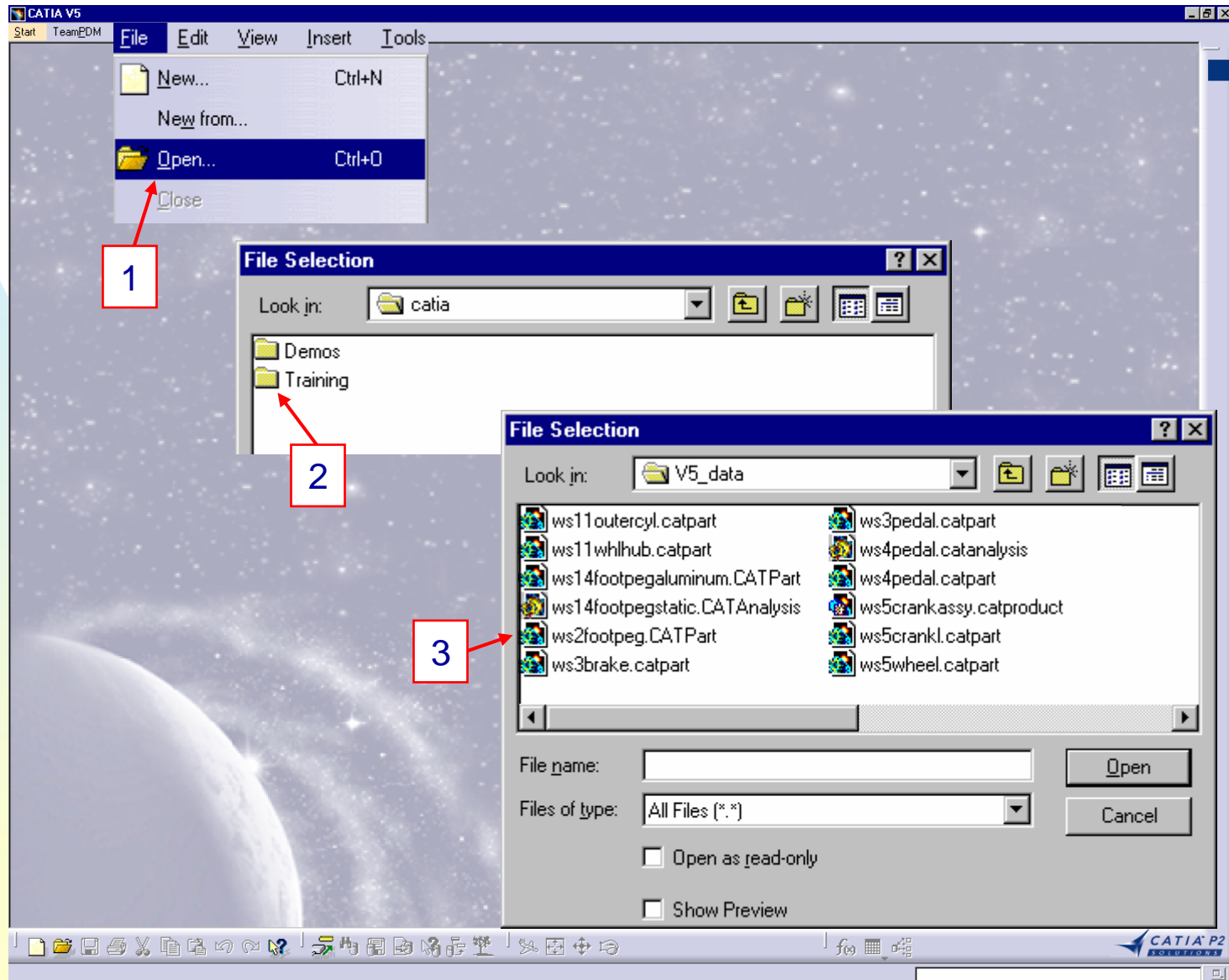
Step 1. Open the part

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 Aluminium.
5. Click Apply Material button...OK.

The screenshot shows the CATIA V5 interface with the 'Foot Peg' part selected in the PartBody tree. The 'Library (ReadOnly)' window is open, showing the 'Metal' tab selected. The 'Aluminium' material is highlighted in the grid. The 'Apply Material' button is highlighted in the bottom right corner of the library window. Red callout boxes with numbers 1 through 5 indicate the steps: 1. Click the Apply Material icon; 2. Select the part; 3. Activate the Metal tab; 4. Select Aluminium; 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.

Material property seen in the specification tree

1

2

3

4

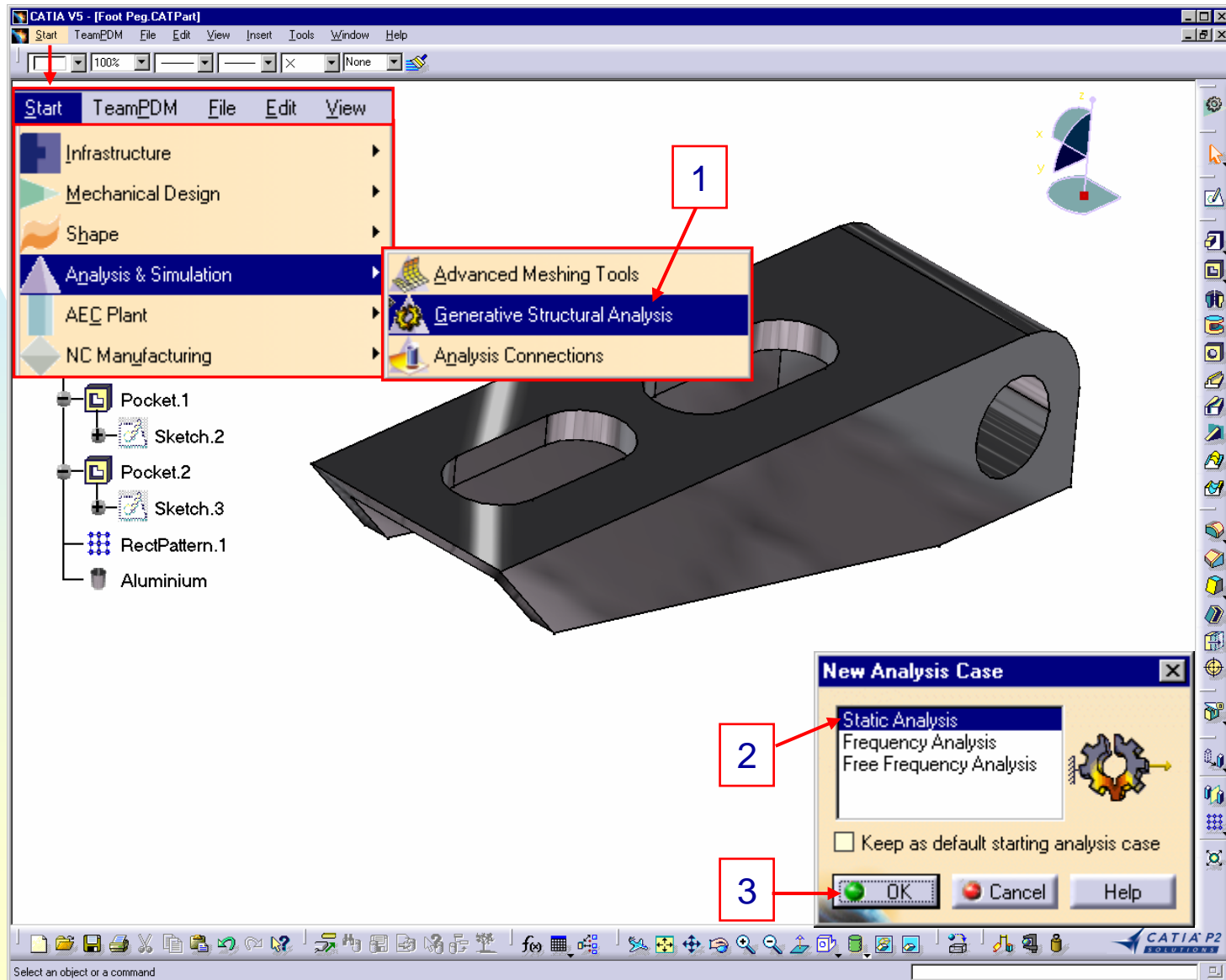
Step 3. Create analysis document

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:

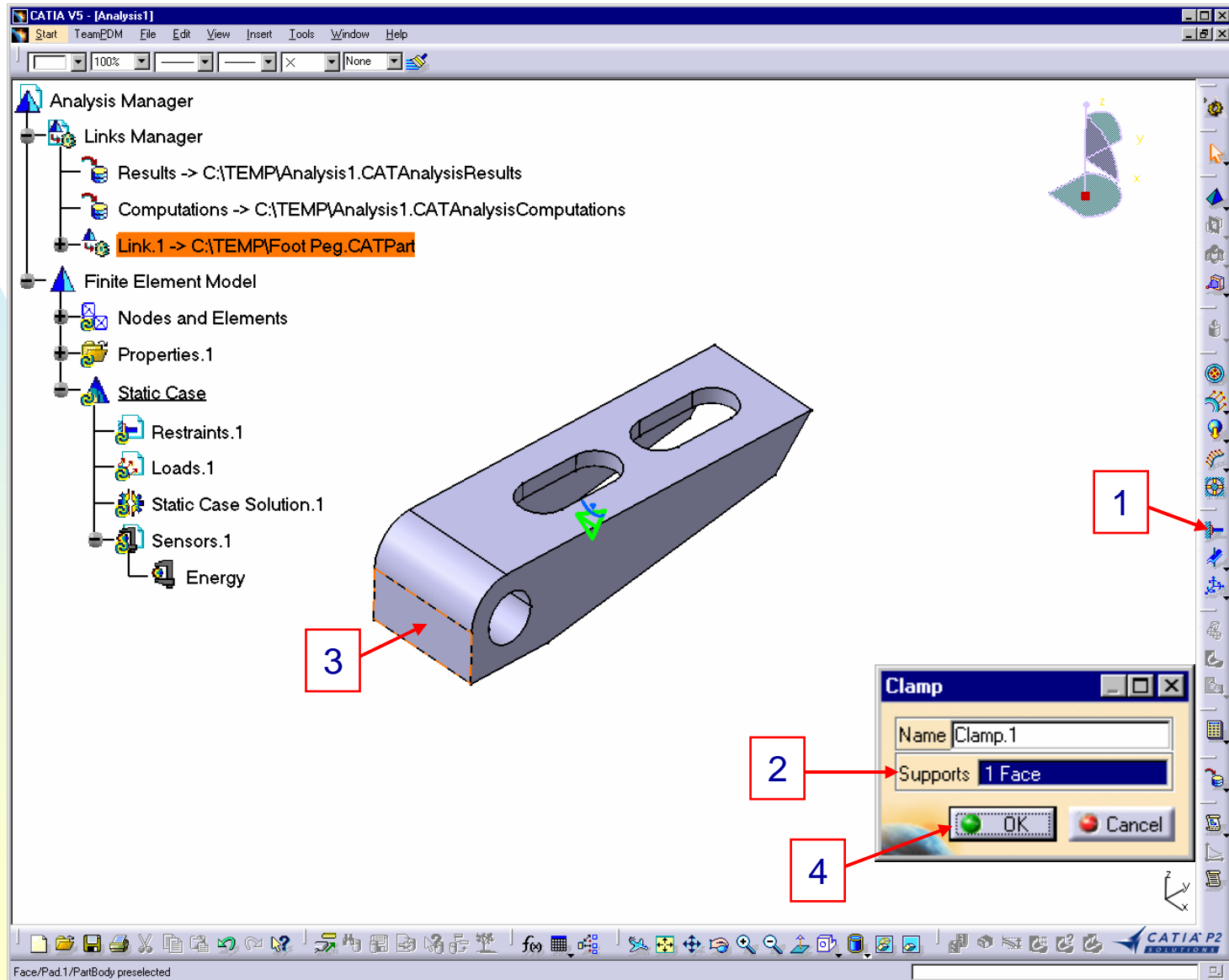
1. Select the clamp icon from the GSA workbench.



2. Be sure Supports field is highlighted.

3. Select geometry to clamp (rear face).

4. Click OK.



The screenshot shows the CATIA V5 interface with the Analysis Manager tree on the left. The tree is expanded to show the 'Static Case' folder, which contains 'Restrains.1', 'Loads.1', 'Static Case Solution.1', and 'Sensors.1'. The 'Restrains.1' folder is highlighted, and the 'Clamp' icon is selected from the GSA workbench. The 'Clamp' dialog box is open, showing 'Name: Clamp.1' and 'Supports: 1 Face'. The 'OK' button is highlighted. The 3D model of the Foot Peg is shown in the center, with a red dashed line indicating the selected rear face. The 'Clamp' icon in the GSA workbench is also highlighted. The 'Supports' field in the dialog box is highlighted. The 'OK' button in the dialog box is highlighted.

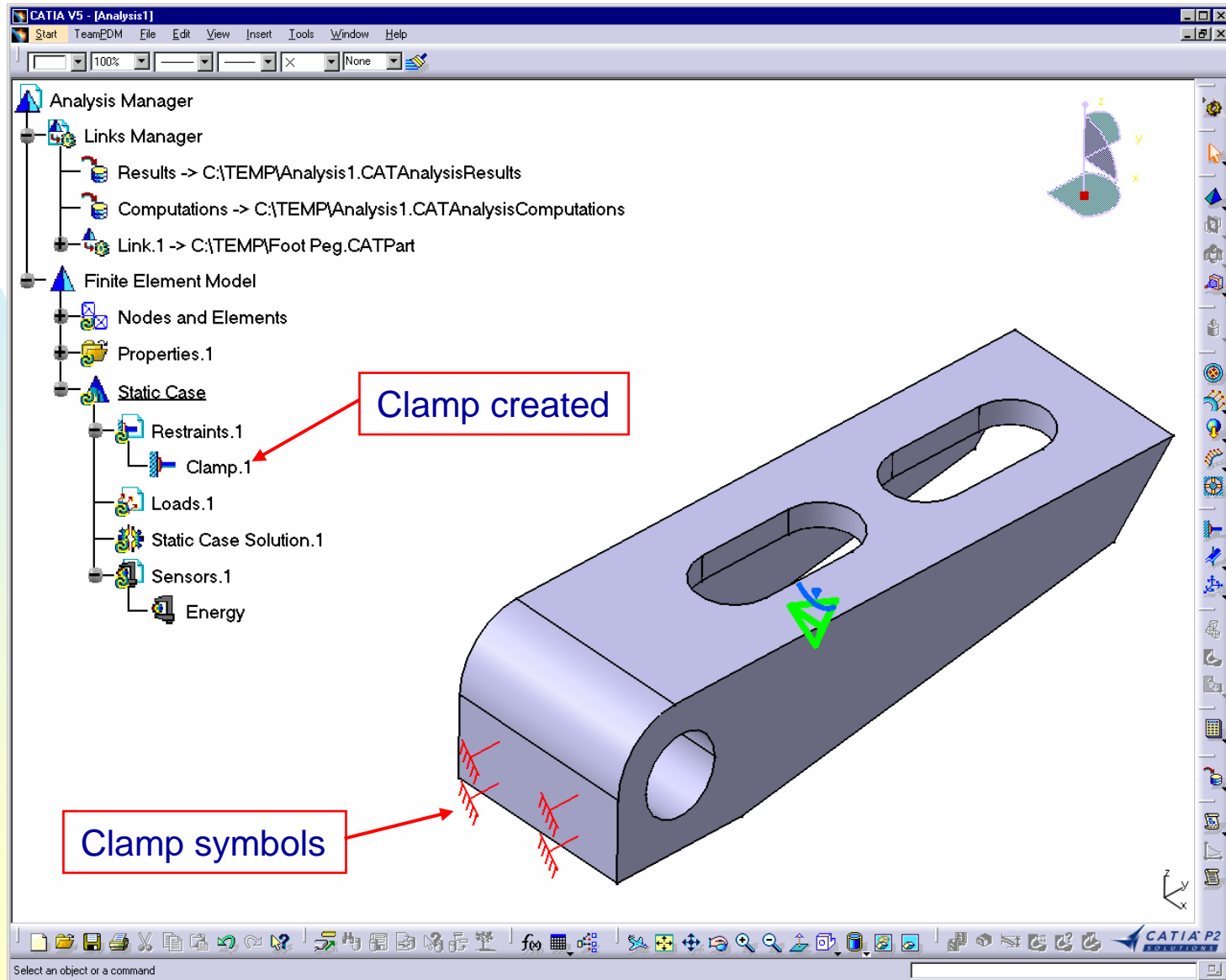
1. Select the clamp icon from the GSA workbench.

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.

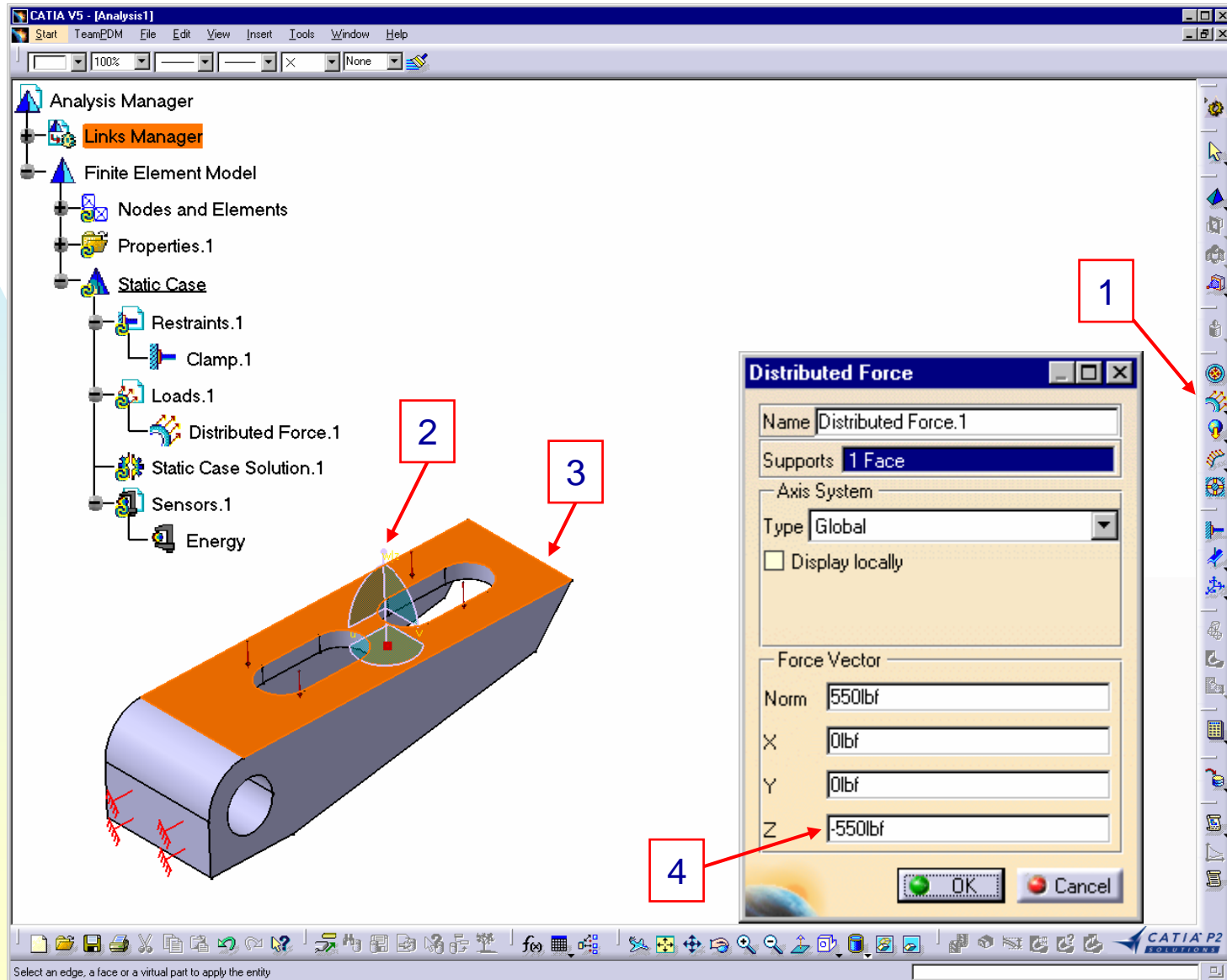
Symbols appear on the part showing the clamp restraint applied to the rear surface of the Foot Peg.

Step 5. Apply load condition

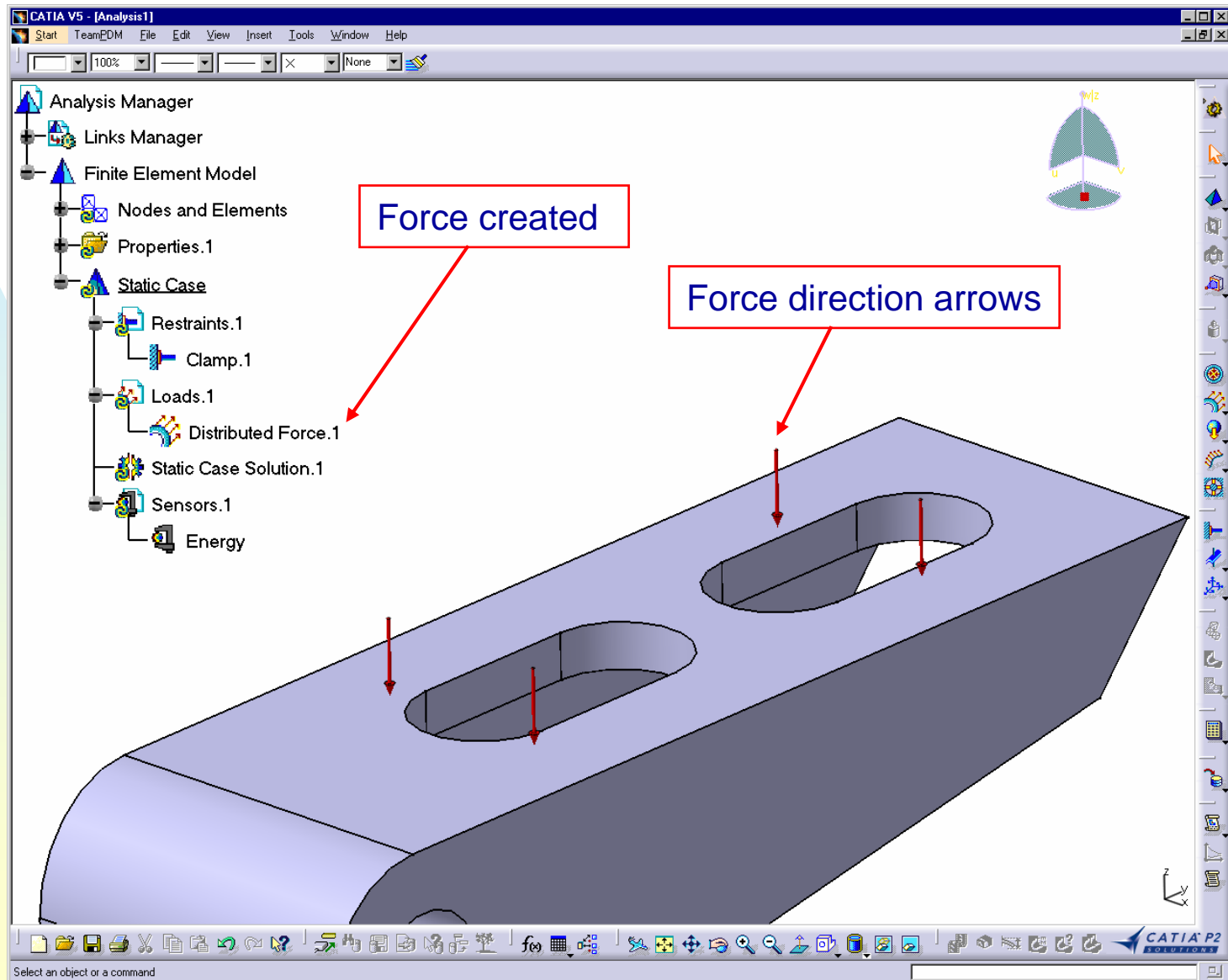
Apply a force load of 550lbf to the Foot Peg top face in a direction normal to the face pushing downward.

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:

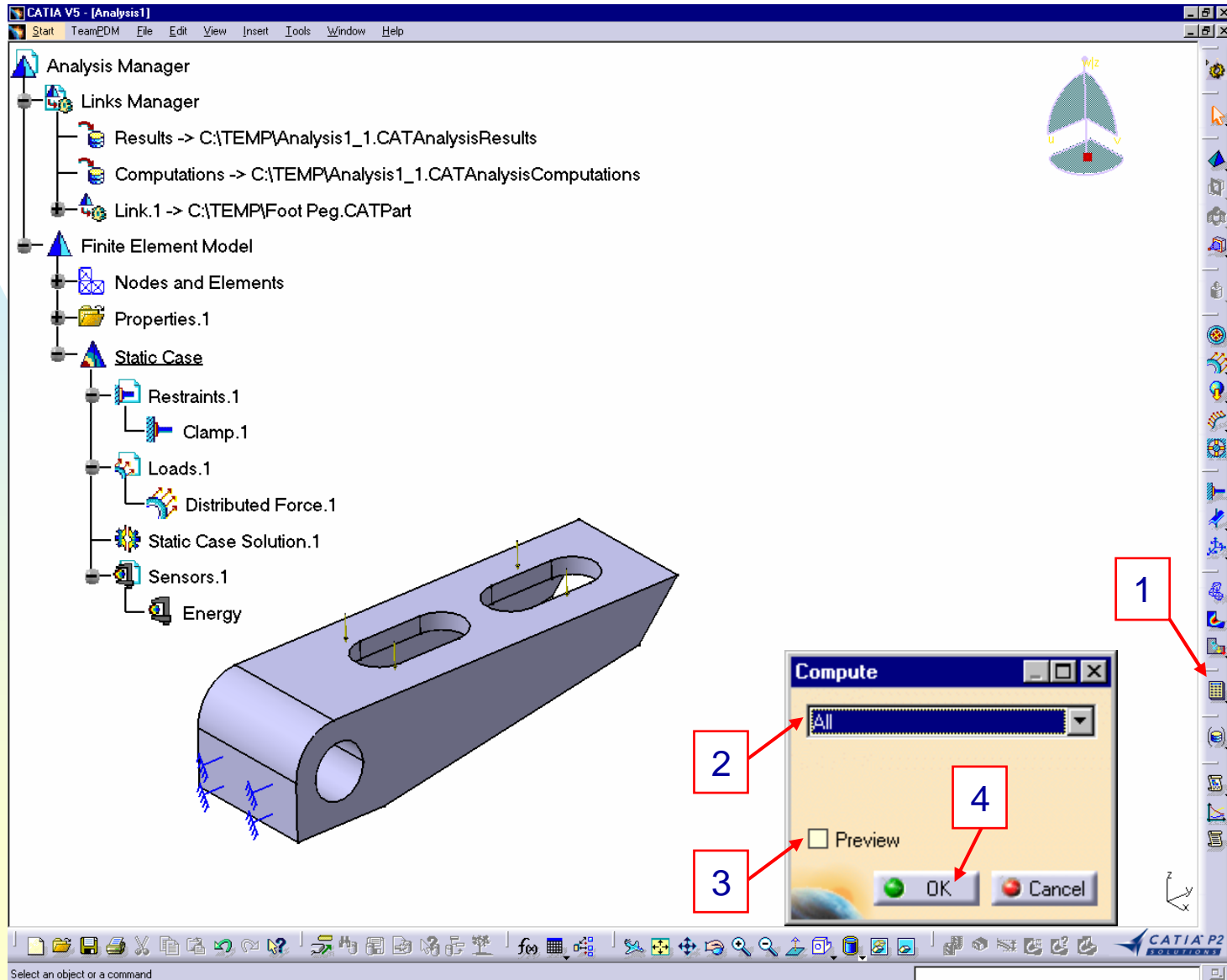
1. Select the Compute icon.



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

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

4. Click OK.



The screenshot displays the CATIA V5 software interface. The Analysis Manager tree on the left shows the following structure:

- Analysis Manager
 - Links Manager
 - Results -> C:\TEMP\Analysis1_1.CATAnalysisResults
 - Computations -> C:\TEMP\Analysis1_1.CATAnalysisComputations
 - Link.1 -> C:\TEMP\Foot Peg.CATPart
 - Finite Element Model
 - Nodes and Elements
 - Properties.1
 - Static Case
 - Restraints.1
 - Clamp.1
 - Loads.1
 - Distributed Force.1
 - Static Case Solution.1
 - Sensors.1
 - Energy

The 3D model of a part is shown in the center, with various restraints and loads applied. The Compute dialog box is open in the bottom right, with the following elements highlighted by red boxes and numbers:

- 1: The Compute icon in the software toolbar.
- 2: The 'All' dropdown menu in the Compute dialog box.
- 3: The 'Preview' checkbox, which is unchecked.
- 4: The 'OK' button in the Compute dialog box.

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.

Von Mises stress image and palette

Image created

1

2

Analysis Manager

- Links Manager
 - Results -> C:\TEMP\Analysis1_1.CATAnalysisResults
 - Computations -> C:\TEMP\Analysis1_1.CATAnalysisComputations
 - Link.1 -> D:\Temp\Foot Peg.CATPart
- Finite Element Model
 - Nodes and Elements
 - Properties.1
 - Static Case
 - Restraints.1
 - Clamp.1
 - Loads.1
 - Distributed Force.1
 - Static Case Solution.1
 - Von Mises Stress (nodal value)
 - Sensors.1

Von Mises Stress (nodal value)
psi

3.4e+003
3.08e+003
2.76e+003
2.44e+003
2.13e+003
1.81e+003
1.49e+003
1.17e+003
855
537
220

On Boundary

CATIA V5 - [Analysis1.CATAnalysis]

Start TeamPDM File Edit View Insert Tools Window Help

Select an object or a command

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.

CATIA V5 - [Analysis1.CATAnalysis]

Start TeamPDM File Edit View Insert Tools Window Help

Analysis Manager

- Links Manager
- Finite Element Model
 - Nodes and Elements
 - Properties.1
 - Static Case
 - Restraints.1
 - Clamp.1
 - Loads.1
 - Distributed Force.1
 - Static Case Solution.1
 - Von Mises Stress (nodal value)
 - Sensors.1

Von Mises Stress (nodal value)

psi

3.4e+003
3.08e+003
2.76e+003
2.44e+003
2.13e+003
1.81e+003
1.49e+003
1.17e+003
855
537
220

On Boundary

Information

Information

Object Name: Von Mises Stress (nodal value)

Display On Boundary Over all the Model

Extrema Values

Min : 219.663 psi

Max : 3396.48 psi

Process List

ELFFProcessElementToNode

Used materials

Foot Peg

Material : Aluminium.1.1

Yield Strength : 13778.589psi

Close

1

2

Max Von Mises stress is lower than material yield strength

From this initial analysis the part will not fail

Step 8. Save analysis document

The screenshot shows the CATIA V5 interface with the File menu open. The Save Management dialog box is displayed, showing a list of files and a 'Save As...' button. Red boxes and arrows indicate the steps:

1. Select 'Save Management...' from the File menu.
2. Highlight 'Analysis1.CATAnalysis' in the list.
3. Click the 'Save As...' button.

The Save Management dialog box contains the following table:

State	Name	Path	Action
Modified	ws2footpeg.CATPart	D:\Temp	
New	Analysis1.CATAnalysis		
Opened	Analysis1_4.CATAnalysisResults	C:\TEMP	
Opened	Analysis1_4.CATAnalysisComputations	C:\TEMP	

The dialog box also includes buttons for 'Save', 'Save As...', 'Propagate directory', and 'Reset'. A small thumbnail of the 3D model is visible in the bottom right corner of the dialog box.

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

The screenshot shows the CATIA V5 interface with a finite element model of a foot peg. A 'Save As' dialog box is open, and a 'Save Management' dialog box is also open. Red boxes with numbers 4 through 8 indicate the steps to be followed.

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.

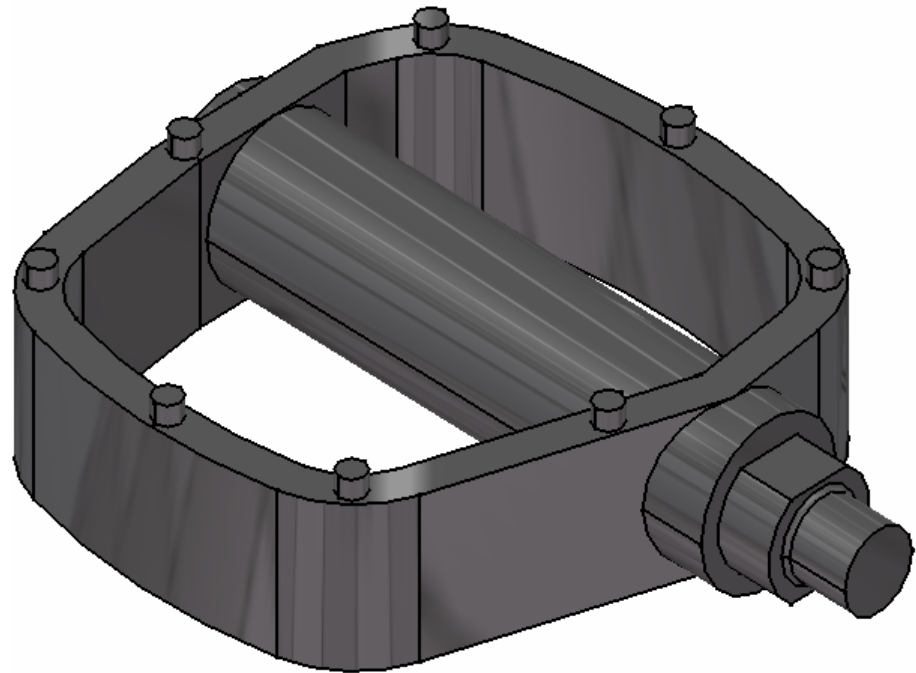
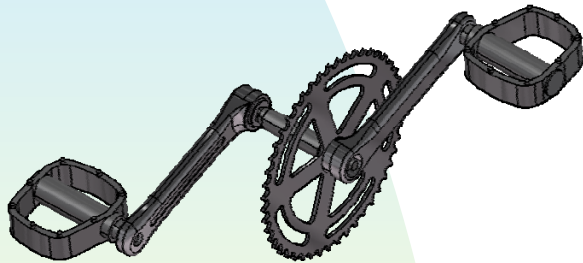
State	Name	Path	Action
Modified	ws2footpeg.CATPart	D:\Temp	
New	Foot Peg Static.CATAnalysis	D:\Temp	Save
Opened	Analysis1_3.CATAnalysisResults	C:\TEMP	
Opened	Analysis1_3.CATAnalysisComputati...	C:\TEMP	

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

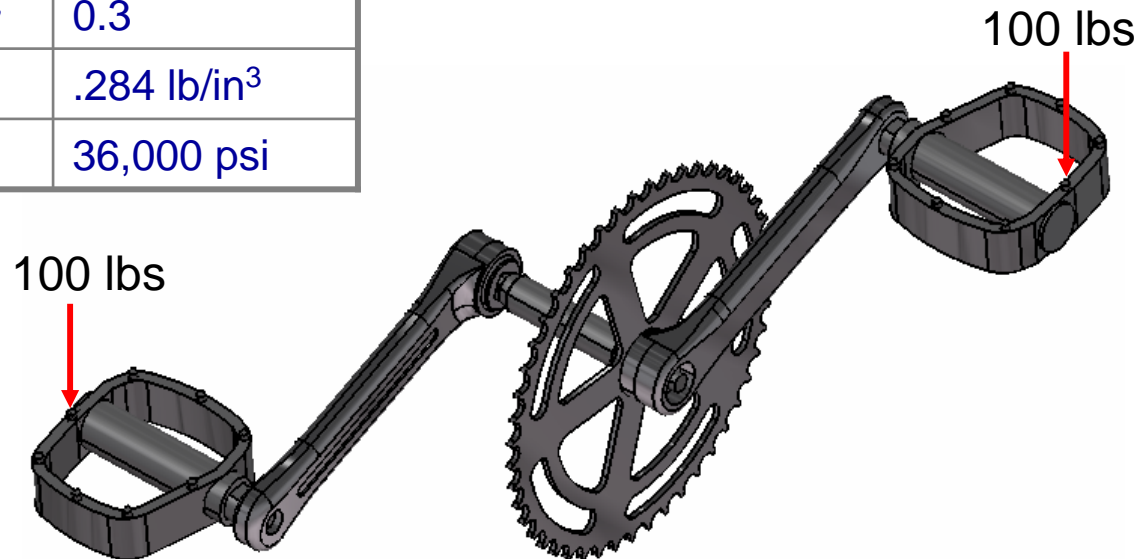
BICYCLE PEDAL STATIC ANALYSIS



■ Problem Description

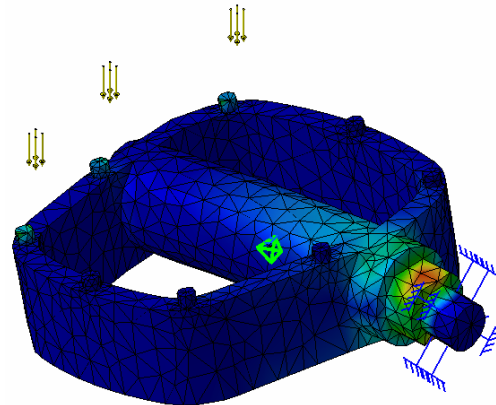
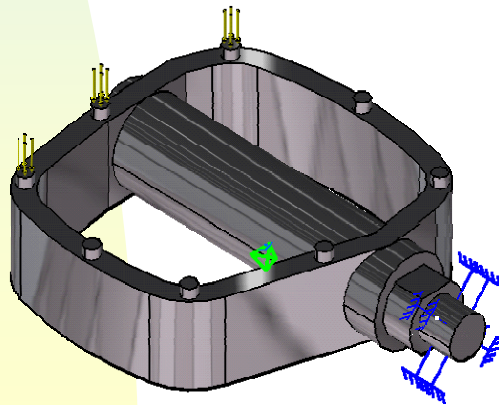
- ◆ Your job will be to analyze various components of a mountain bicycle. We will start with the pedal.
- ◆ Let's assume a 200 lb person riding this bike is standing, balanced evenly on each pedal. Material (Steel) properties as specified below. Use a rough analysis to determine where the high stress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

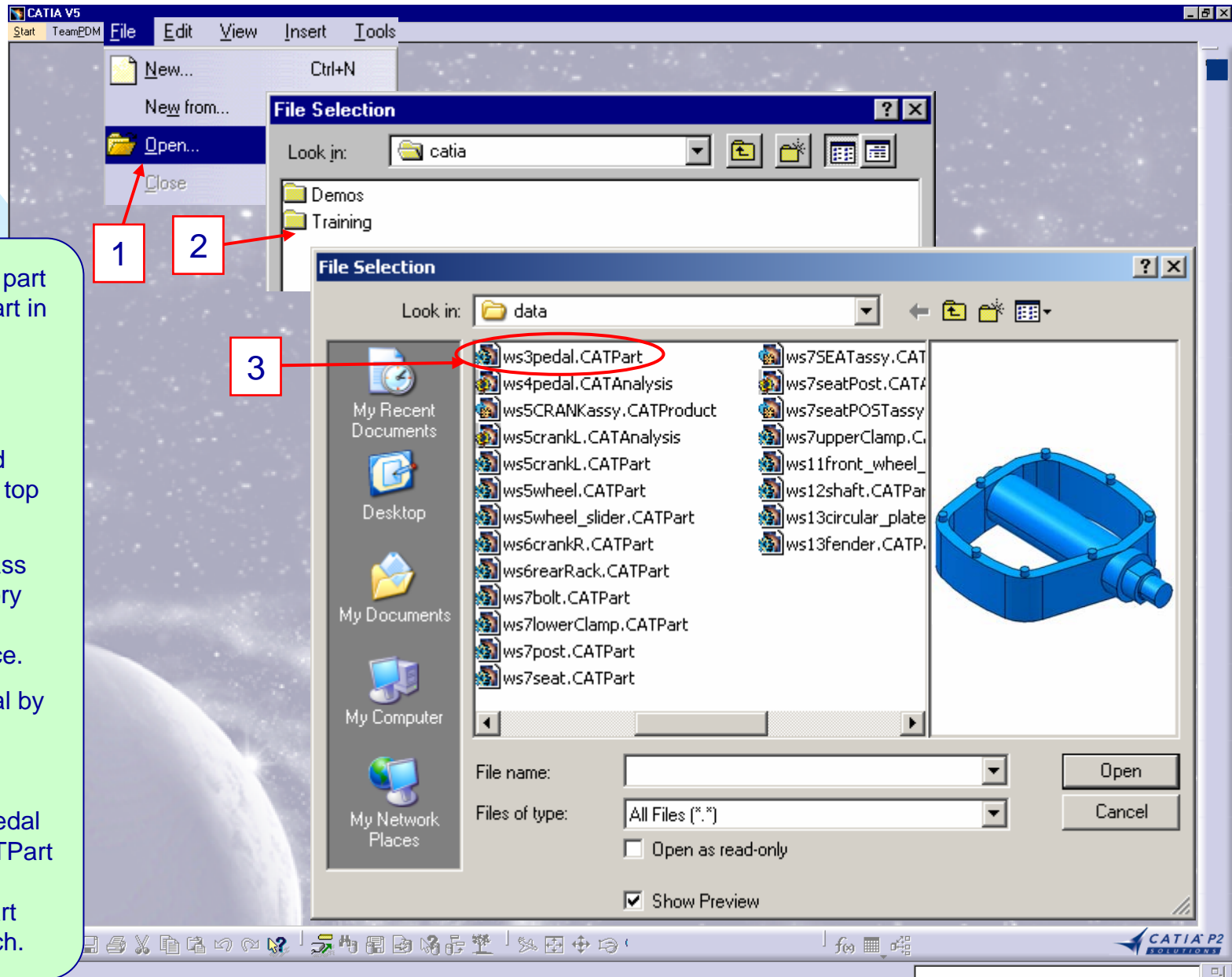


■ Suggested Exercise Steps

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.

Step 2. Apply steel material properties to the part

Tools Window Help

Options...

1

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.

Options

Knowledge Parameters Tolerance Symbols **Units**

Units

Magnitudes	Units	Symbols
Length	Inch	in
Angle	Degree	deg
Time	Second	s
Mass	Pound	lb
Volume	Cubic inch	in3
Density	Pound per cubic inch	lb_in3

Length Inch (in)

Dimensions display

Display trailing zeros

Exponential notation for values greater than 10e+

Exponential notation for values lower than 10e-

Display for the magnitude Length

Same display for read/write numbers and read-only numbers

Decimal places for read/write numbers

Decimal places for read-only numbers

Area	Square inch	in2
Inertia Moment	Pound per square ...	lb_in2
Energy	British thermal unit	Btu
Force	Pound Force	lbf
Inertia	Inch ** 4	in4
Massic flow	Pound per minute	lb_min

Linear Acceleration	Inch per square s...	in_s2
Angular Acceleration	Radian per square...	rad_s2
Strain Energy	Pound force x Inch	lbfxin
Volumic force	Pound Force per c...	lbf_in3
Surfacic mass	Pound per square ...	lb_in2
Speed	Mile per hour	mph

Moment	Pound force x Inch	lbfxin
Pressure	Lb.force per squa...	psi
Angular stiffness	Pound force.inch ...	lbfxin_r
Temperature	Fahrenheit	Fdeg
Linear mass	Pound per inch	lb_in
Linear stiffness	Pound Force per i...	lbf_in

Volumetric flow	US gallon per minute	gal_min
Frequency	Hertz	Hz
Electric power	Watt	W
Voltage	Volt	V
Electric resistance	Ohm	Ohm
Electric intensity	Amper	A

Speed	Mile per hour	mph
Linear feed rate	Inch per minute	in_min
Angular feed rate	Inch per turn	in_turn
Linear spindle speed	Foot per minute	ft_min
Angular spindle speed	Turn per minute	turn_min
Surfacic spindle speed	IN2_MN	in2_min

OK Cancel

3

2

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.

The screenshot displays the CATIA V5 interface for applying material to a part. The features tree on the left shows the 'Pedal' part selected. The 'Library (ReadOnly)' window is open, showing the 'Metal' tab selected. The 'Steel' material is highlighted in the library. The 'Link to file' checkbox is checked, and the 'Apply Material' button is visible. A 3D model of the pedal part is shown at the bottom left, with 'Steel' applied to it.

Construction	Fabrics	Metal	Other	Stone	Wood
Aluminium	Brass	Bronze	Chroma	Copper	
Gold	Iron	Lead	Magnesium	Nickel	
Silver	Steel	Titanium	Tungsten	Uranium	
Yellow Brass	Zinc				

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.

The screenshot displays the CATIA V5 software interface. On the left, the features tree shows a hierarchy: Pedal, xy plane, yz plane, zx plane, Axis Systems, Parameters, PartBody, Open_body.1, and Steel. A red arrow labeled '1' points to 'Steel'. A context menu is open over 'Steel', with a red arrow labeled '2' pointing to the 'Properties' option. The Properties dialog box is open, showing the 'Analysis' tab. A red arrow labeled '3' points to the 'Analysis' tab. The 'Structural Properties' section is visible, with a red arrow labeled '4' pointing to the input fields for Young Modulus (2.901e+007psi), Poisson Ratio (0.3), Density (0.284lb_in3), Thermal Expansion (0.0000117), and Yield Strength (36000psi). A red arrow labeled '5' points to the 'Customized View Parameters' icon in the bottom toolbar. The 3D model of a pedal part is shown in the bottom right corner.

Step 3. Create a new CATIA analysis document

The screenshot displays the CATIA V5 software interface. The 'Start' menu is open, showing the 'Analysis & Simulation' workbench selected. The 'Generative Structural Analysis' option is highlighted. The 'Analysis Manager' tree shows the 'Finite Element Model' structure, including 'Nodes and Elements', 'Properties.1', 'Material Property 3D.1', and 'Static Case'. The 'New Analysis Case' dialog box is open, showing 'Static Analysis' selected. A 3D model of a pedal is shown in the background.

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** location 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.

The screenshot shows the CATIA V5 interface with the following elements:

- Analysis Manager:** A tree view showing the analysis structure:
 - Links Manager
 - Results -> C:\ELFINI\pedal\pedal.CATAnalysisResults
 - Computations -> C:\ELFINI\pedal\pedal.CATAnalysisComputations
 - Link.1 -> C:\CATIA\Training\data\ws3pedal.CATPart
 - Finite Element Model
 - Nodes and Elements
 - Properties.1
 - Static Case
- 3D Model:** A 3D rendering of a pedal component with a green arrow pointing to a specific location on the model.
- Current Storage Location Dialog:** A dialog box with two sections:
 - Results Data: C:\ELFINI\pedal\pedal.CATAnalysisResults (with a 'Modify' button)
 - Computation Data: C:\ELFINI\pedal\pedal.CATAnalysisComputations (with a 'Modify' button)
 - Buttons: OK, Cancel
- Solve... Dialog:** A dialog box with three icons (a folder, a folder with a red arrow, and a folder with a blue arrow) and a '1' in a red box pointing to the first icon.

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.

The screenshot shows the CATIA V5 interface with the Analysis Manager tree on the left. The tree structure is as follows:

- Analysis Manager
 - Links Manager
 - Finite Element Model
 - Nodes and Elements
 - OCTREE Tetrahedron Mesh.1 : Pedal (highlighted with a red box labeled '1')
 - Properties.1
 - Static Case
 - Restraints.1
 - Loads.1
 - Static Case Solution.1
 - Sensors.1
 - Energy

The 3D model on the right shows a grey pedal part with a green tetrahedron mesh applied to its inner surface. A red arrow points from the 'OCTREE Tetrahedron Mesh.1 : Pedal' entry in the tree to the mesh on the part.

The 'OCTREE Tetrahedron Mesh' dialog box is open, showing the following settings:

- Global | Local (selected)
- Size: 0.25in (highlighted with a red box labeled '2')
- Sag: 0.025in (highlighted with a red box labeled '3')
- Element type: Linear (highlighted with a red box labeled '4') and Parabolic
- Buttons: OK, Cancel

Step 5. Apply a clamp restraint

Analysis Manager

- Links Manager
- Finite Element Model
 - Nodes and Elements
 - OCTREE Tetrahed
 - Properties.1
 - Static Case
 - Restraints.1
 - Clamp.1
 - Loads.1
 - Static Case Solutik
 - Sensors.1
 - Energy

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.

Step 6. Apply a distributed force

Analysis Manager

- Links Manager
- Finite Element Model
 - Nodes and Elements
 - Properties.1
 - Static Case
 - Restraints.1
 - Clamp.1
 - Loads.1
 - Distributed Force
 - Static Case Solution.1
 - Sensors.1
 - Energy

Force

Distributed Force

Name: Distributed Force.1

Supports: 3 Faces

Axis System

Type: Global

Display locally

Force Vector

Norm: 100lbf

X: 0lbf

Y: 0lbf

Z: -100lbf

OK Cancel

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.

Preview active

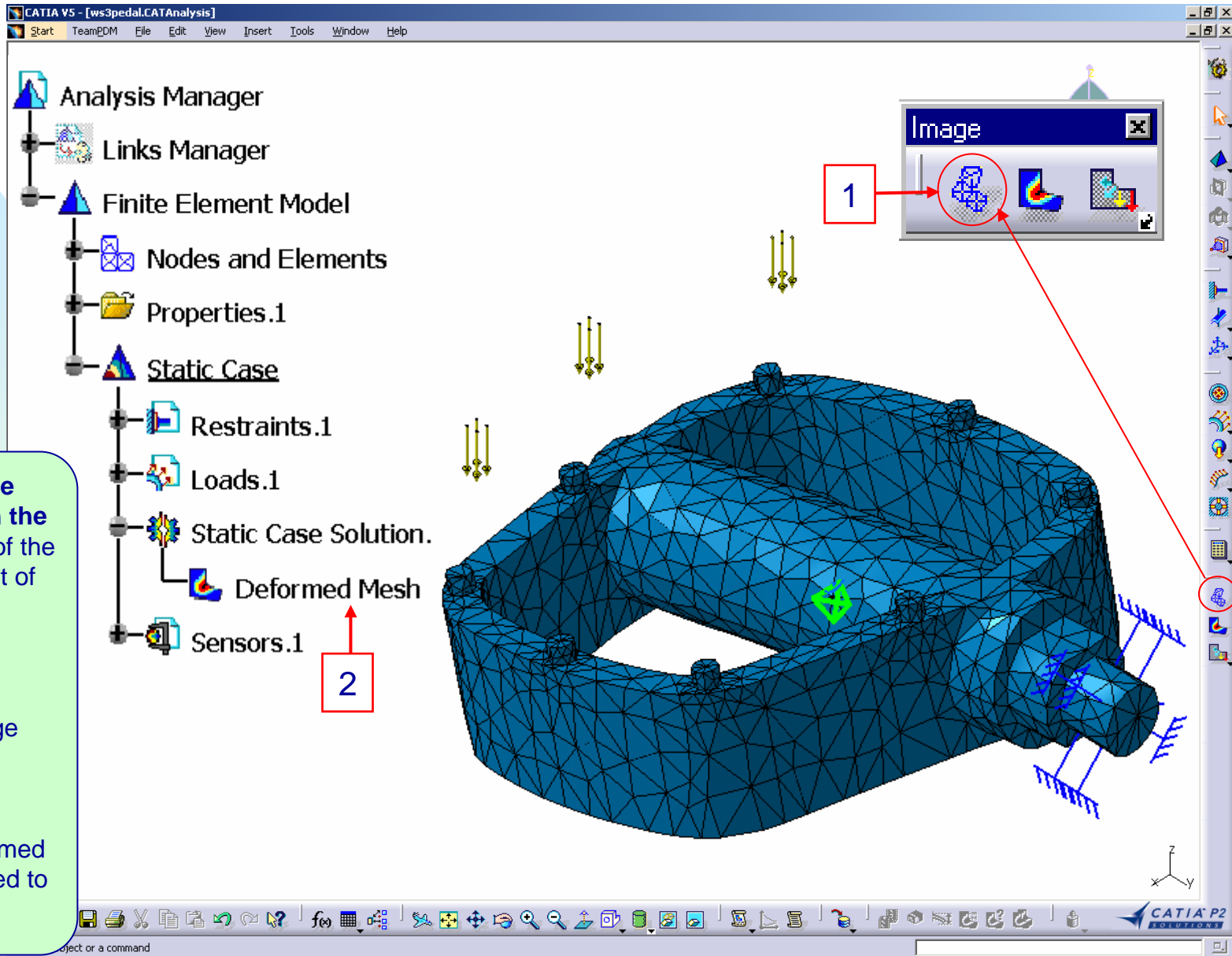
Computation Resources Estimation

1 s of CPU
1.26e+003 kilo-bytes of memory
3.09e+003 kilo-bytes of disk
Intel MKL(c) Library found: Intel(R) MKL V5.1.0

Do you want to continue the computation?

Yes No

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.

Step 8. Visualize the analysis results

CATIA V5 - [ws3pedal.CATAnalysis]

Start TeamPDM File Edit View Insert Tools Window Help

Static Case Solution.1

Deformed Mesh

Von Mises Stress (nodal value)

2

1

Von Mises Stress (nodal value)

psi

2.46e+004

2.21e+004

1.96e+004

1.72e+004

1.47e+004

1.23e+004

9.83e+003

7.38e+003

4.93e+003

2.47e+003

19.5

On Boundary

CATIA V5 P2 SOLUTIONS

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

CATIA V5 - [ws3pedal.CATAnalysis]

Static Case Solution.1

- Deformed Mesh
- Von Mises Stress (nodal value)
- Translational displacement vector

Translational displacement magnitude object Definition...

Translational displacement vector

in

0.00407

0.00367

0.00326

0.00285

0.00244

Image

Image Edition

Deformed

Display on Deformed Mesh

Visu Criteria Filters Select

AVERAGE-ISO

SYMBOL

TEXT

Edition

Iso/Fringe Symbol Axis System

OK Cancel Help

1 2 3 4 5

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.



Step 8. Visualize the analysis results

CATIA V5 - [ws3pedal.CATAnalysis]

Static Case Solution.1

- Deformed Mesh
- Von Mises Stress (nodal value)
- Translational displacement vector

Translational displacement magnitude object ▶ Definition...

Image Edition

Deformed

Display on Deformed Mesh

Visu | Criteria | Filters | Select

AVERAGE-ISO

SYMBOL

TEXT

4

Edition

Iso/Fringe | Symbol | Axis System

OK | Cancel | Help

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

CATIA V5 - [ws3pedal.CATAnalysis]

Static Case Solution.1

- Deformed Mesh
- Von Mises Stress (nodal value)
- Translational displacement vector

Image Edition

Deformed

Display on Deformed Mesh

Visu | Criteria | Filters | Select

AVERAGE-ISO

SYMBOL

TEXT

Edition

Iso/Fringe Symbol Axis System

OK Cancel Help

Image Iso Fringe Editor

ISO value

IsoContour

ISO smooth

Display Element Without Value

OK Cancel

1

2

3

4

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.

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:

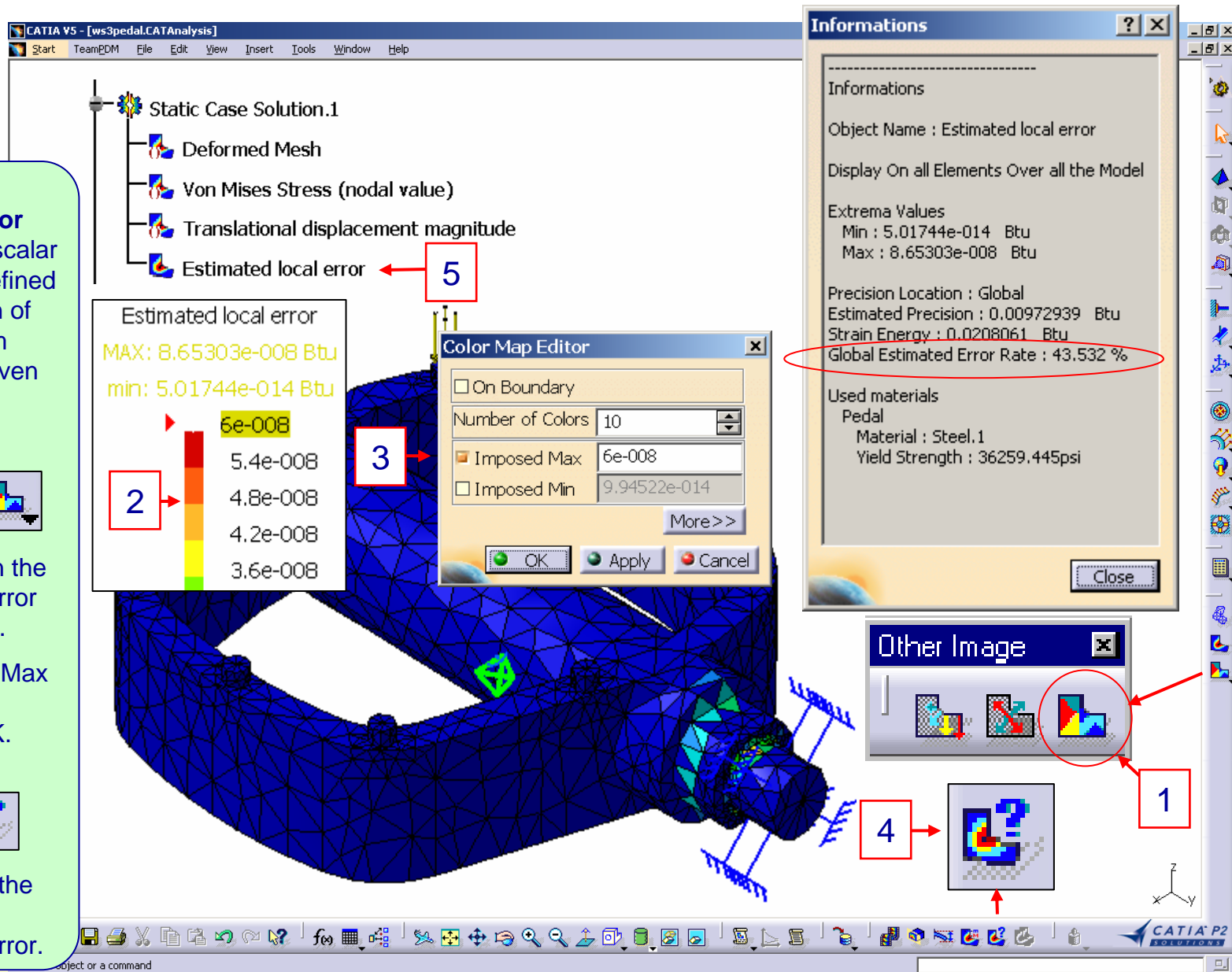
1. Select the  Precision icon.

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.



The screenshot shows the CATIA V5 interface with the following elements:

- Features Tree:** Shows a hierarchy starting with "Static Case Solution.1", followed by "Deformed Mesh", "Von Mises Stress (nodal value)", "Translational displacement magnitude", and "Estimated local error" (highlighted with a red arrow and number 5).
- Color Map Editor:** A dialog box with "Number of Colors" set to 10. The "Imposed Max" checkbox is checked and set to $6e-008$. Other options include "On Boundary" (unchecked), "Imposed Min" (unchecked, set to $9.94522e-014$), and "More>>". Buttons for "OK", "Apply", and "Cancel" are at the bottom.
- Informations Panel:** Displays analysis results for "Estimated local error". It shows "Extrema Values" (Min: $5.01744e-014$ Btu, Max: $8.65303e-008$ Btu) and "Precision Location : Global". The "Global Estimated Error Rate : 43.532 %" is circled in red.
- Other Image Palette:** Located at the bottom right, it contains several icons. The "Precision" icon (a small color map) is circled in red and labeled with a red arrow and number 1. A red arrow and number 4 points to the "Estimated local error" icon in the same palette.
- Color Map Palette:** A vertical color bar on the left of the model, labeled with a red arrow and number 2. It shows a gradient from blue to red. Values are listed: $6e-008$ (red), $5.4e-008$, $4.8e-008$, $4.2e-008$, and $3.6e-008$ (blue). A red arrow and number 3 points to the "Imposed Max" value in the color map editor.

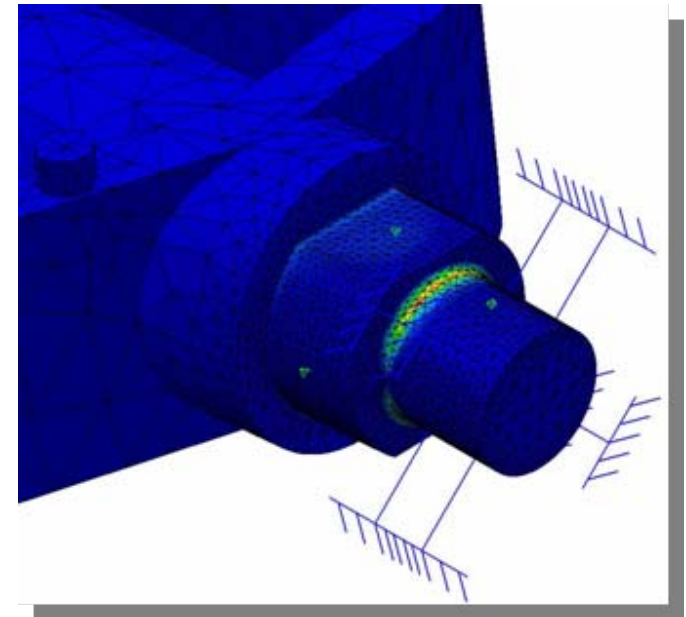
Step 8. Visualize the analysis results

■ 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	43.5 %
Local % Precision error	NA %

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

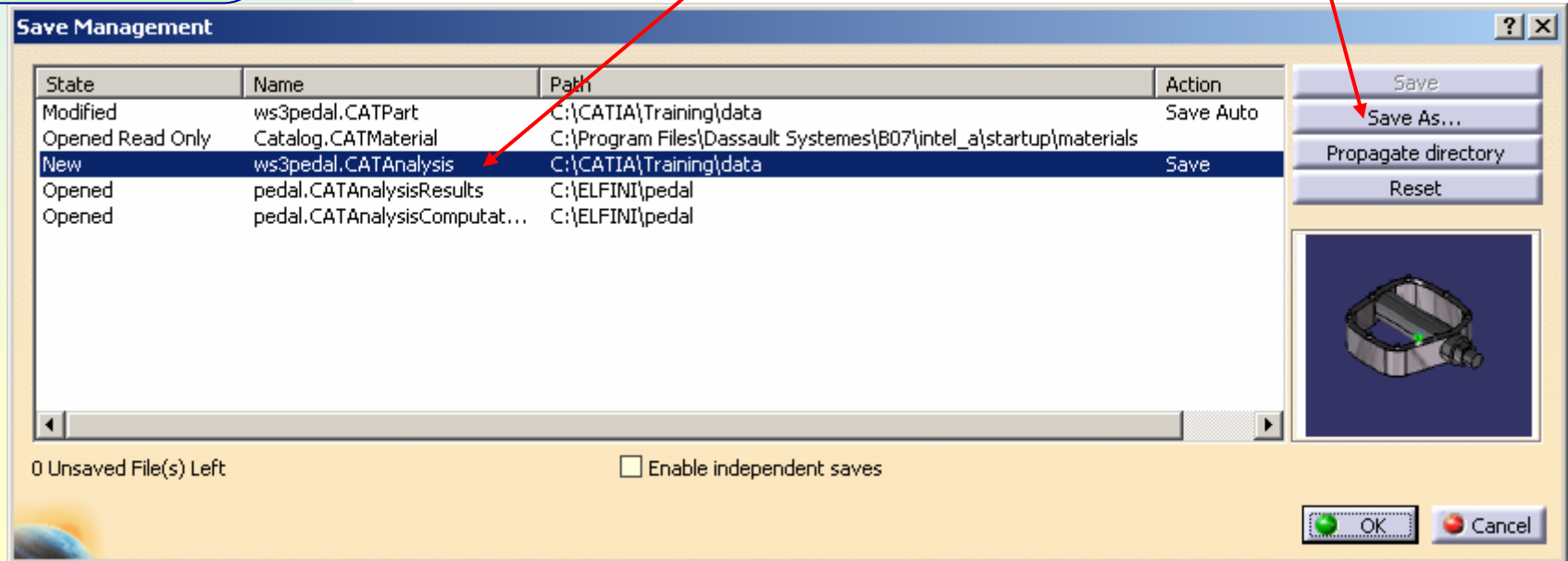
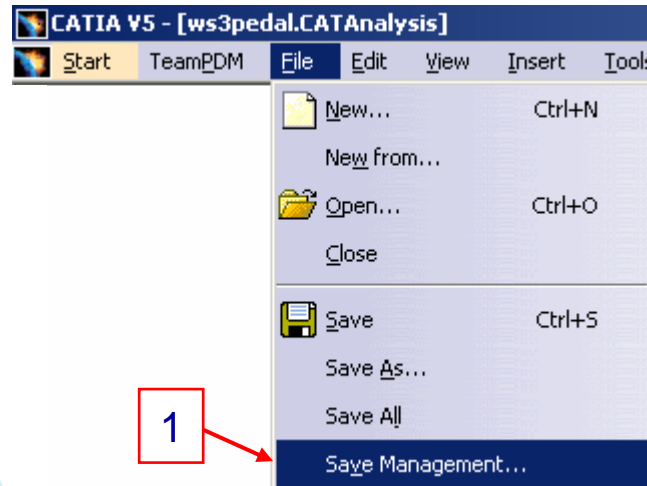


Step 9. Save the analysis document

Steps:

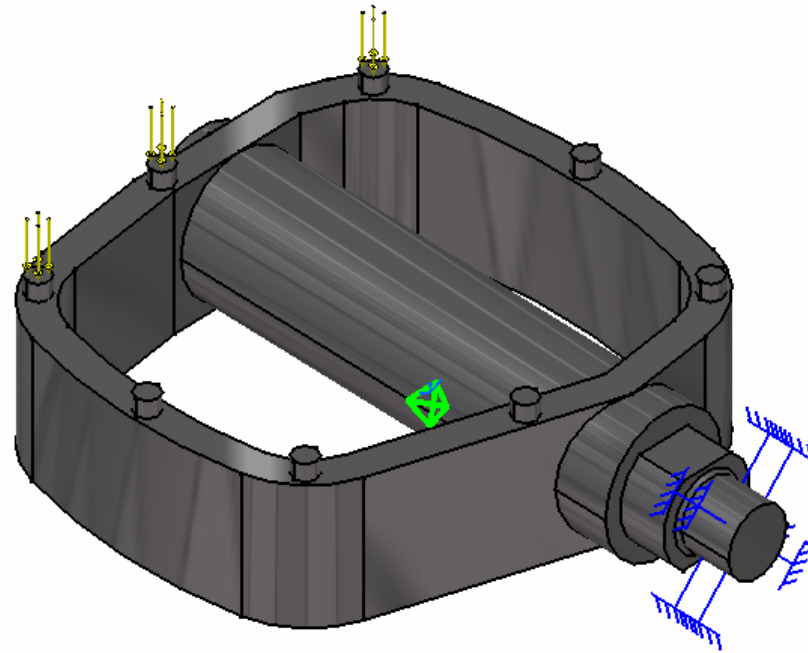
1. From the File menu select Save Management.
2. Select document you want to save.
3. Select Save As to specify name and path, select OK.

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?

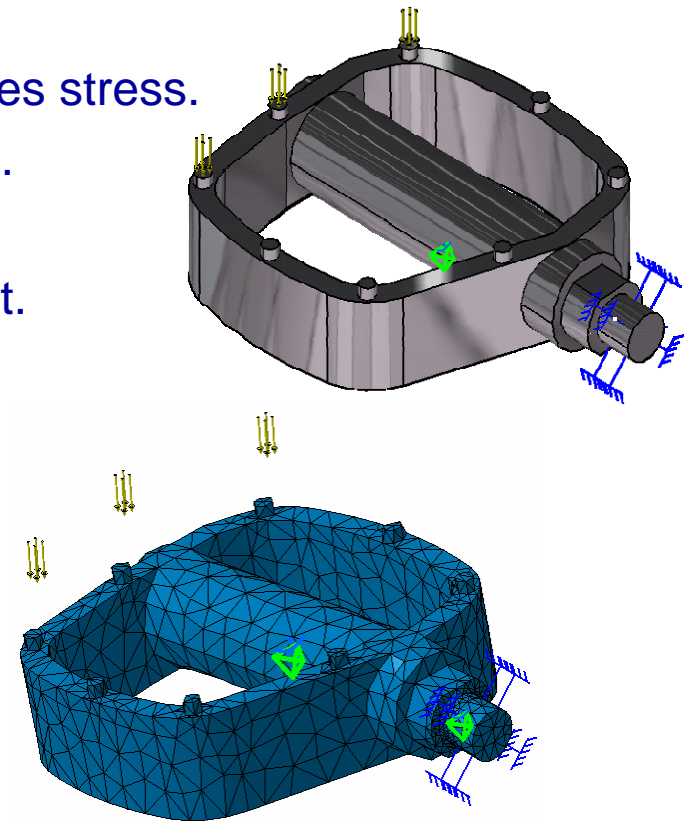
Steel ASTM A36

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.



Step 1. Open the existing CATIA analysis

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

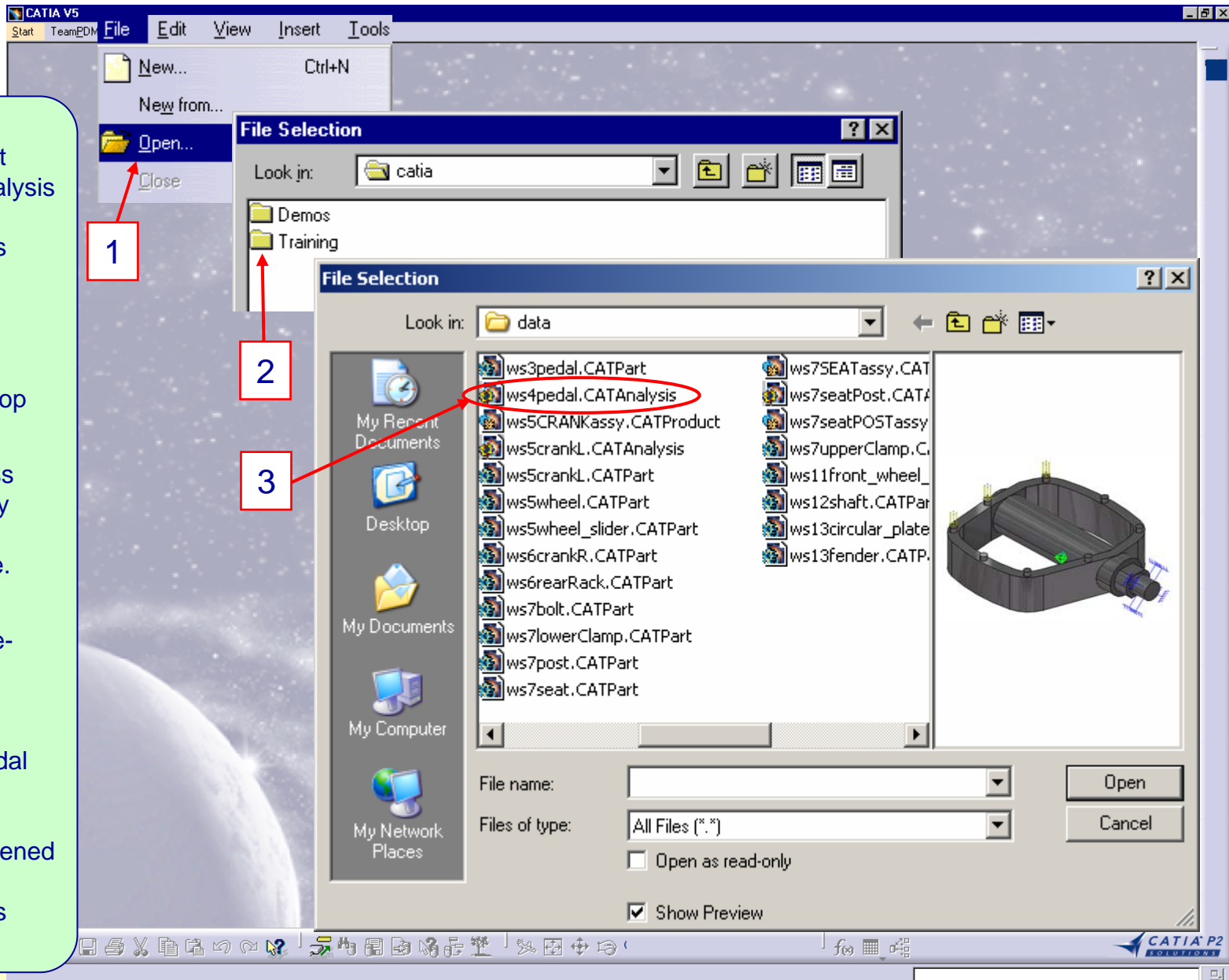
Steps:

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

Analysis Manager

- Links Manager
- Finite Element Model
 - Nodes and Elements
 - OCTREE Tetrahedron Mesh.1 : Pedal
 - Properties.1
 - Static Case
 - Restraints.1
 - Clamp.1
 - Loads.1
 - Distributed Force.1
 - Static Case Solution.1
 - Deformed Mesh
 - Von Mises Stress (nodal value)
 - Translational displacement magnitude
 - Estimated local error
 - Sensors.1
 - Energy

OCTREE Tetrah...

Global | Local

Size: 0.25in

Sag: 0.025in

Element type

Linear

Parabolic

OK Cancel

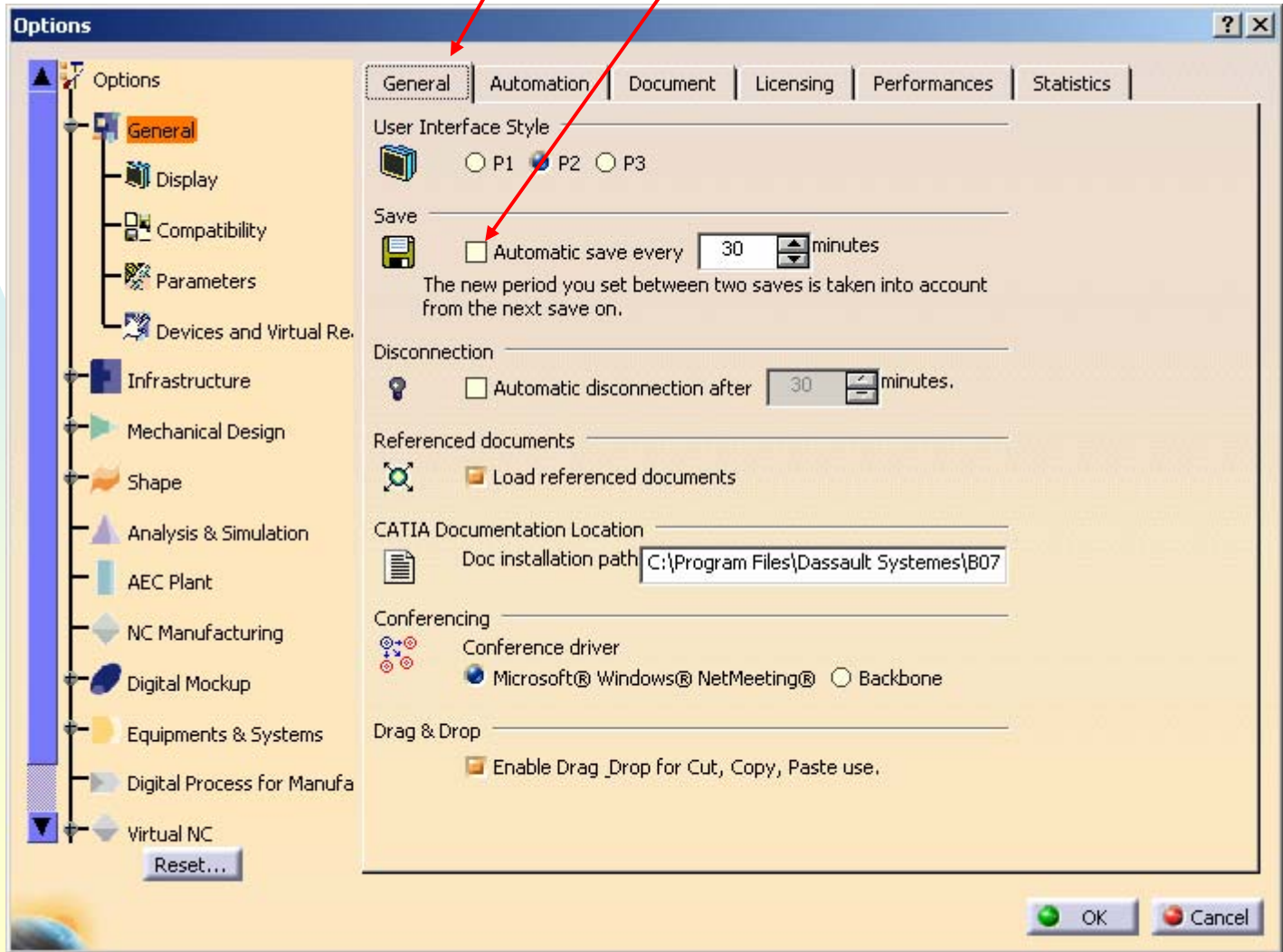
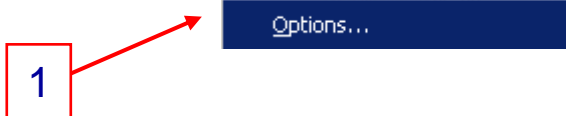
	.25" Linear Mesh
Max Von Mises	24.6 ksi
Translational Displacement	..00407 inch
Error Estimate	8.65e-8
Global % Precision error	43.5 %
Local % Precision error	NA %

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.

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.


The screenshot shows the CATIA V5 software interface with the following components:

- Tree View (Left):** Analysis Manager, Links Manager, Finite Element Model, Nodes and Elements (OCTREE Tetrahedron Mesh.1 : Pedal), Properties.1, Static Case, Restraints.1 (Clamp.1), Loads.1 (Distributed Force.1), Static Case Solution.1 (Deformed Mesh, Von Mises Stress (nodal value), Translational displacement magnitude, Estimated local error), Sensors.1 (Energy).
- 3D Model (Center):** A detailed view of a pedal component with a mesh applied. A green arrow points to a specific element on the model.
- Dialog Boxes:**
 - Element Type:** A dialog box with two radio buttons: 'Linear' (unselected) and 'Parabolic' (selected). It has 'OK' and 'Cancel' buttons. A red box labeled '2' points to the 'Parabolic' option.
 - Mesh Specification:** A dialog box with several icons representing different meshing options. A red box labeled '1' points to the 'Change Element Type' icon.
- Toolbar (Right):** A vertical toolbar with various icons for meshing and analysis.
- Status Bar (Bottom):** Shows 'object or a command' and the CATIA V5 logo.

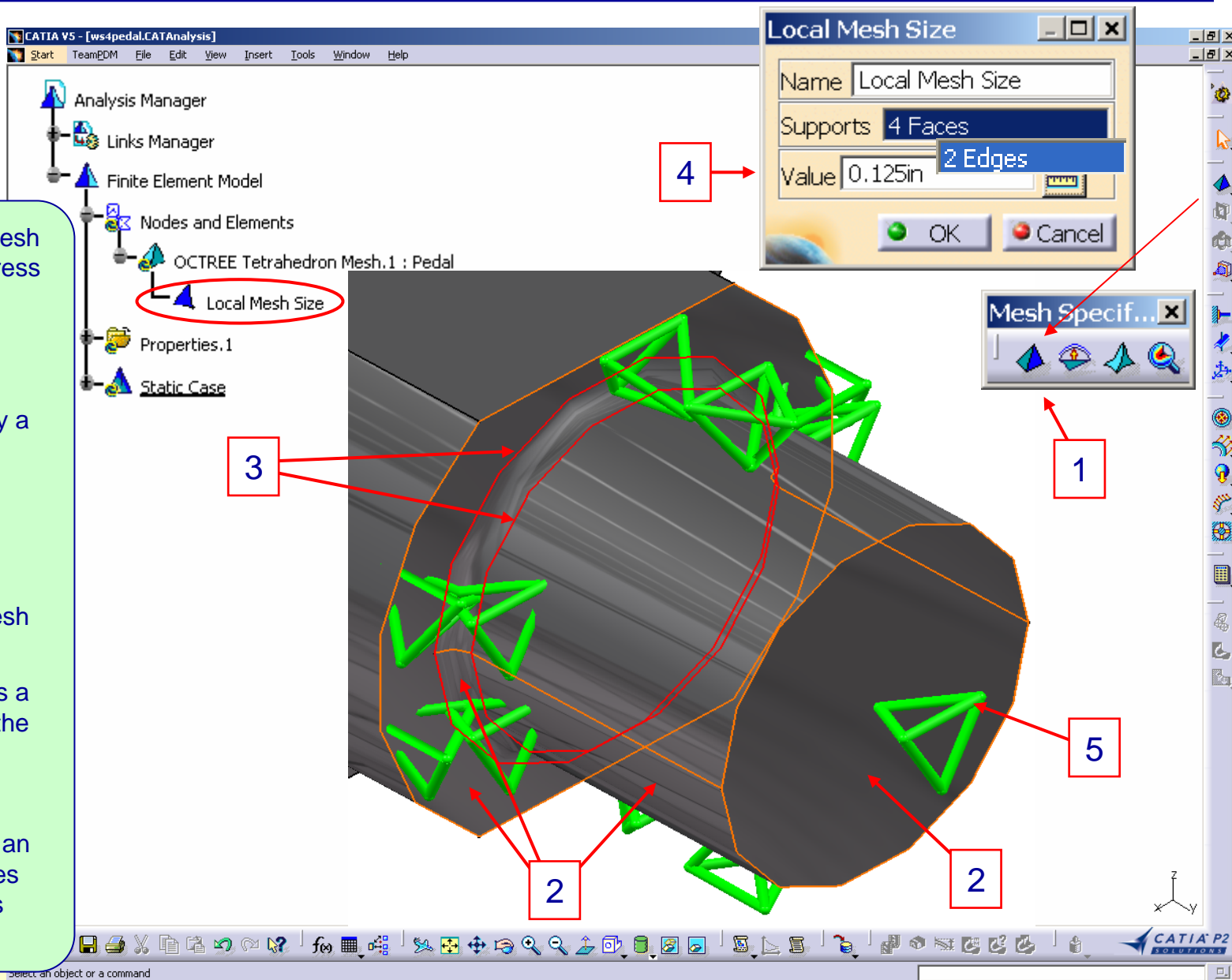
Step 2. Change mesh to parabolic and add local meshing

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).



Step 2. Change mesh to parabolic and add local meshing

CATIA V5 - [ws4pedal.CATAnalysis]

Analysis Manager
Links Manager
Finite Element Model
Nodes and Elements
OCTREE Tetrahedron Mesh
Local Mesh Size
Local Mesh Sag
Properties.1
Static Case

Local Mesh Sag
Name Local Mesh Sag
Supports 4 Faces
Value 0.013in 2 Edges
OK Cancel

Mesh Specific...

1
2
3
4
5

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.

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.

The screenshot shows the CATIA V5 interface with a finite element model of a mechanical part. The left-hand tree view shows the analysis setup, including a Static Case with Restraints, Loads, and a Static Case Solution. The main window displays the model with a green arrow pointing to a specific element. Two dialog boxes are overlaid on the interface:

- Compute Dialog:** Located in the top right, it has a dropdown menu set to 'All', a checked 'Preview' checkbox, and 'OK' and 'Cancel' buttons. A red box labeled '2' points to the dropdown menu, and another red box labeled 'Preview active' points to the 'Preview' checkbox.
- Computation Resources Estimation Dialog:** Located in the bottom center, it displays resource requirements: '1e+002 s of CPU', '1.3e+004 kilo-bytes of memory', and '7.37e+004 kilo-bytes of disk'. It also shows 'Intel MKL(c) Library found: Intel(R) MKL V5.1.0' and asks 'Do you want to continue the computation?' with 'Yes' and 'No' buttons. A red box labeled 'RAM' points to the memory requirement, a red box labeled '1' points to the dialog's title bar, and a red box labeled '3' points to the 'Yes' button.

A red box labeled 'Available disk space required at your specified external storage location.' points to the '7.37e+004 kilo-bytes of disk' requirement.

Available disk space required at your specified external storage location.

RAM

Preview active

1

3

2

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?

CATIA V5 - [ws4pedal.CATAnalysis]

Static Case Solution.1

- Deformed Mesh
- Von Mises Stress (nodal value)
- Extrema**
 - Global Maximum.1
 - Local Maximum.1
 - Local Maximum.2
- Translational displacement
- Estimated local error
- Sensors.1

Von Mises Stress (nodal value) Global Maximum.1: 80384.6 psi

Von Mises Stress (nodal value) Local Maximum.2: 64247.6 psi

Extrema Creation

- Global
 - Minimum extrema at most: 0
 - Maximum extrema at most: 2
- Local
 - Minimum extrema at most: 0
 - Maximum extrema at most: 2

OK Cancel

1

2

3

CATIA V5 SOLUTIONS

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.

CATIA V5 - [ws4pedal.CATAnalysis]

Start TeamPDM File Edit View Insert Tools Window Help

Static Case Solution.1

- Deformed Mesh
- Von Mises Stress (nodal value)
- Extrema
- Translational displacement magnitude
- Estimated local error
- Extrema
 - Global Maximum.1
 - Local Maximum.1
 - Local Maximum.2
- Sensors.1

Extrema Creation

- Global
 - Minimum extrema at most 0
 - Maximum extrema at most 2
- Local
 - Minimum extrema at most 0
 - Maximum extrema at most 2

OK Cancel

Estimated local error Global Maximum.1: 1.44788e-007 Btu

Estimated local error Local Maximum.1: 1.17917e-007 Btu

CATIA P2 SOLUTIONS

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.

CATIA V5 - [ws4pedal.CATAnalysis]

Start TeamPDM File Edit View Insert Tools Window Help

Static Case Solution.1

- Deformed Mesh
- Von Mises Stress (nodal value)
 - Extrema
- Translational displacement magnitude
- Estimated local error **2**
 - Global Maximum.1
 - Local Maximum.1
 - Local Maximum.2
- Sensors.1

Information

Information

Object Name : Estimated local error

Display On all Elements Over all the Model

Extrema Values

Min : 2.22053e-016 Btu

Max : 1.44788e-007 Btu

Precision Location : Global

Estimated Precision : 0.00383491 Btu **4**

Strain Energy : 0.0278214 Btu

Global Estimated Error Rate : 25.3922 % **3**

Used materials

Pedal

Material : Steel.1

Yield Strength : 36259.445psi

Close

1

Select an object or a command

CATIA P2 SOLUTIONS

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.

The screenshot shows the CATIA V5 interface with the following elements:

- Features Tree:** Von Mises Stress (nodal value) -> Extrema -> Global Maximum.1 (highlighted with a red box labeled '4b'). Other items include Local Maximum.1, Local Maximum.2, Translational displacement vector, Estimated local error, Sensors.1, Adaptivity Process, Adaptivity Convergence.1, Adaptivities.1, and Adaptivity Box.1 (highlighted with a red box labeled '4a').
- Adaptivity Box Dialog:** Name: Adaptivity Box.1; Objective Error (%): 10; Solution: Static Case Solution.1; Local Error (%): 42,648; Select Extremum button; OK and Cancel buttons.
- Views:** ISO, Top, and Front views of a mechanical part with a stress distribution. Red boxes labeled '5' show the adaptivity box being manipulated around the maximum stress points.
- Icons:** A red arrow labeled '2' points to the Adaptivity Box icon in the top toolbar. A red arrow labeled '1' points to the Von Mises stress image icon in the bottom toolbar.

Step 8. Adapt and converge

The screenshot displays the CATIA V5 interface with the following components:

- Tree View (Left):** Shows a hierarchy including 'Static Case Solution.1', 'Sensors.1', 'Adaptivity Process', 'Adaptivity Convergence.1', 'Adaptivities.1', and 'Adaptivity Box.1'. A red box labeled '1' highlights the 'Adaptivity Convergence.1' node.
- Adaptivity Convergence Dialog Box (Top Right):** Contains the text 'Name Adaptivity Convergence.1' and 'Iterations Max Number 2'. A red box labeled '3' points to the 'Iterations Max Number' field.
- 3D Model (Center):** A 3D rendering of a pedal assembly with a green 'A' icon on the central shaft.
- Toolbar (Right):** A vertical toolbar with various icons. A red box labeled '2' points to the 'Adaptivity Convergence' icon.
- Status Bar (Bottom):** Shows the text '...1/Adaptivity Process/Analysis Manager selected'.

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

The screenshot displays the CATIA V5 interface for a static analysis of a pedal. The tree view on the left shows the following structure:

- Static Case Solution.1
 - Deformed Mesh
 - Von Mises Stress (nodal value)
 - Translational displacement magnitude
 - Estimated local error (highlighted with a red box and arrow labeled '1')
 - Extrema (highlighted with a red box and arrow labeled '2')
 - Local Update (button)
- Sensors.1
- Adaptivity Process
 - Adaptivity Convergence.1
 - Adaptivities.1
 - Adaptivity Box.1

The 3D model of the pedal is shown with a blue mesh. A green 'A' icon is visible on the model, and a red box labeled '4' points to the 'Local Update' button. The 'Informations' panel on the right provides the following data:

Informations

Object Name : Estimated local error

Display On all Elements Over all the Model

Extrema Values
Min : 8.39503e-016 Btu
Max : 8.20623e-008 Btu

Precision Location : Global
Estimated Precision : 0.00194529 Btu
Strain Energy : 0.0286291 Btu
Global Estimated Error Rate : 18.1267 %

Used materials
Pedal
Material : Steel.1
Yield Strength : 36259.445psi

Step 3 is indicated by a red box labeled '3' pointing to the 'Estimated local error' icon in the bottom toolbar.

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 Adaptivity 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.

The screenshot displays the CATIA V5 interface for an adaptive analysis. The tree view on the left shows the following structure:

- Static Case Solution.1
 - Deformed Mesh
 - Von Mises Stress (nodal value)
 - Translational displacement magnitude
 - Estimated local error
 - Extrema
- Sensors.1
- Adaptivity Process
 - Adaptivity Convergence.1
 - Adaptivities.1
 - Adaptivity Box.1

The 'Adaptivity Box' dialog box is open, showing the following settings:

- Name: Adaptivity Box.1
- Objective Error (%): 10
- Solution: Static Case Solution.1
- Local Error (%): 12.361
- Buttons: Select Extremum, OK, Cancel

The 3D model shows a blue mesh with an orange adaptive box. The local error is visualized with a color scale, and the adaptive box is highlighted in orange. A red box with the number '1' points to 'Adaptivity Box.1' in the tree, and another red box with the number '2' points to the 'Local Error (%)' field in the dialog box.

Step 9. Visualize the adaptive analysis results

2

Static Case Solution.1

- Deformed Mesh
- Von Mises Stress (nodal value)
- Extrema
- Translational displacement magnitude
- Estimated local error

Sensors.1

Adaptivity Process

- Adaptivity Convergence.1
- Adaptivities.1
- Adaptivity Box.1

Von Mises Stress (nodal value)

- Extrema
- Global Maximum.1

Local Update

3

Image Extremum Editor

Show Label

Von Mises Stress (nodal value) Global Maximum. 1: 172252 psi

OK Cancel

1

Von Mises Stress (nodal value) Global Maximum. 1: 172252 psi

Object or a command

CATIA V5 - [ws4pedal.CATAnalysis]

Start TeamPDM File Edit View Insert Tools Window Help

CATIA V5 SOLUTIONS

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

CATIA V5 - [ws4pedal.CATAnalysis]

Analysis Manager
Links Manager
Finite Element Model
Nodes and Elements
Properties. 1
Static Case
Restraints. 1
Loads. 1
Static Case Solution. 1
Deformed Mesh
Von Mises Stress (nodal value)

2

3

1

View section only
Show cutting plane
Close

View section only
Show cutting plane
Close

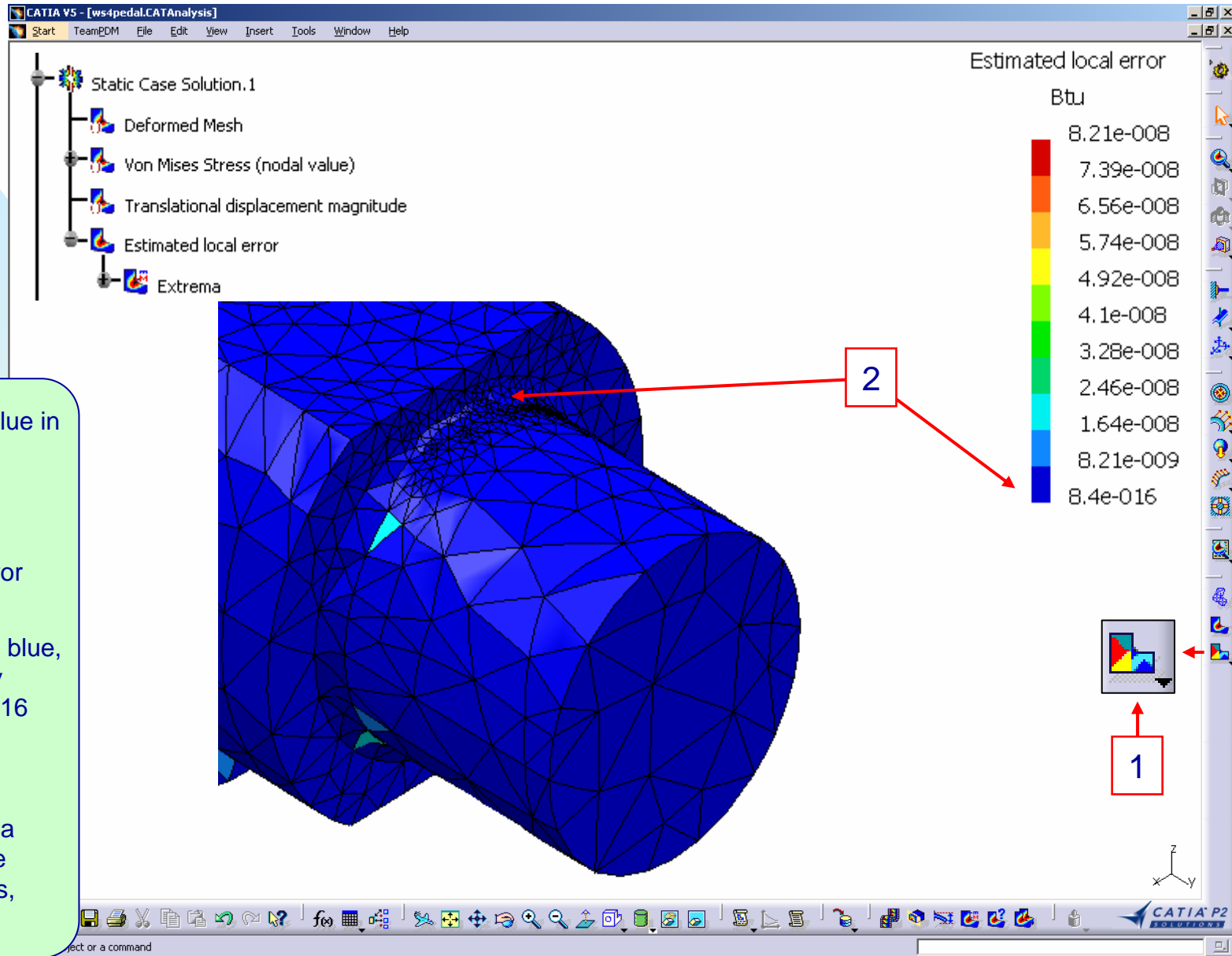
Check cross sectional area stress values.

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.

Reaction Sensor

Available Entities
Clamp.1

Reference Axis
Type: Global (global origin)

OK Cancel

Sensor Creation

Functions
misesmax
dispmax
reaction ← 2
globalerror
dispmaxongroup

OK Cancel

Reaction Tensor

Name: Clamp.1
Axis: Global (global origin)

Force Moment Origin

Parameter	Value
Fx	4.324e-007lbf
Fy	8.021e-007lbf
Fz	100lbf


Norm: 100lbf

Close

Reaction -> Clamp.1 object Definition...

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	43.5 %	18.1 %
Local % Precision error	NA %	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.

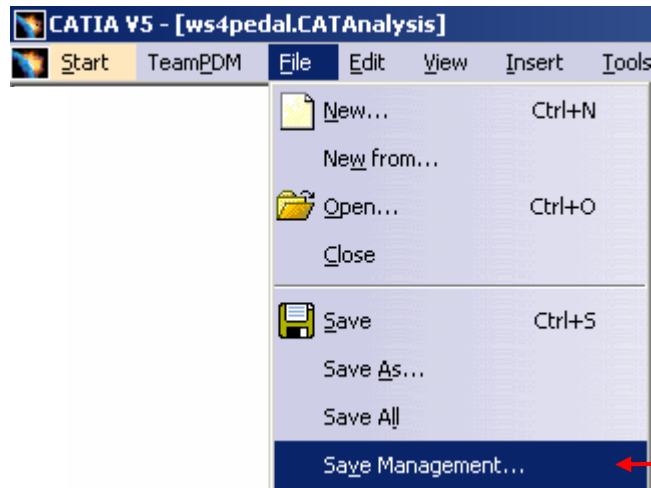
The screenshot shows the CATIA V5 interface with the following elements:

- Tree View:** Analysis Manager, Links Manager, Results, Computations, Link.1, Pedal, Finite Element Model, Nodes and Elements, Properties.1, Static Case, Restraints.1, Loads.1, Static Case Solution.1, Deformed Mesh, Von Mises Stress (nodal value), Translational displacement magnitude, Estimated local error, Sensors.1, Energy, Reaction -> Clamp.1, Adaptivity Process, Adaptivity Convergence.1, Adaptivities.1, Adaptivity Box.1.
- 3D Model:** A 3D model of a pedal with a central shaft and a clamp.
- Reporting options dialog box:** Output directory: C:\ELFINI\pedal; Title of the report: PEDAL CATANALYSIS; Choose the analysis case(s): Static Case; OK and Cancel buttons.
- Analysis Resu... icon:** Located in the bottom toolbar.
- HTML Report Preview:** A preview of the generated report showing the title 'PEDAL.CATANALYSIS', a 'MESH:' section with a table, and an 'ELEMENT TYPE:' section with a table.

Entity	Size
Nodes	30913
Elements	19879

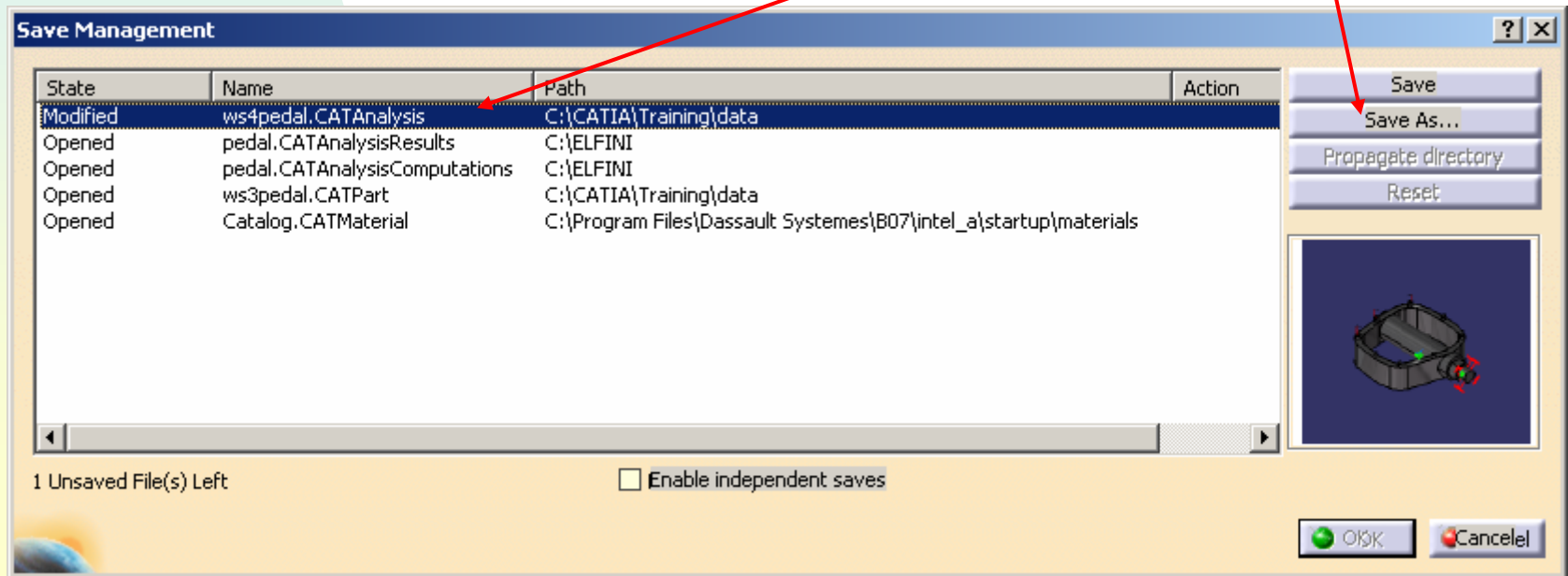
Connectivity	Statistics
TE10	19879 (100.00%)

Step 12. Save the analysis document

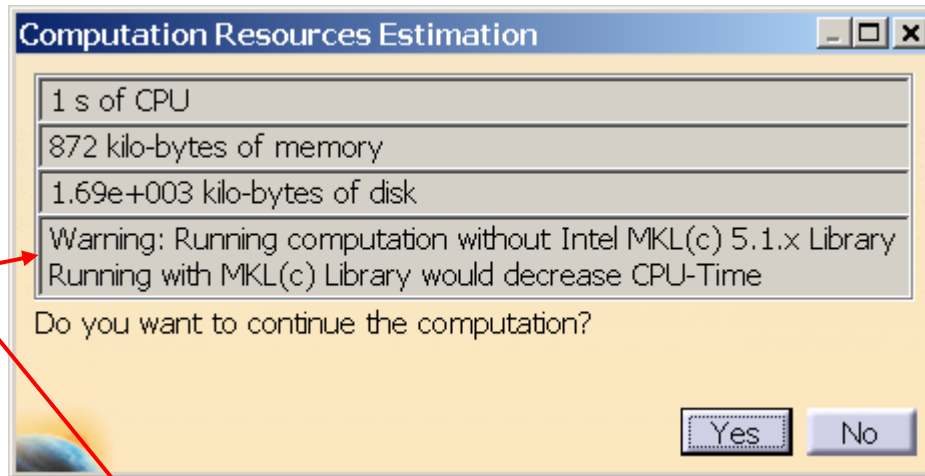


Steps:

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



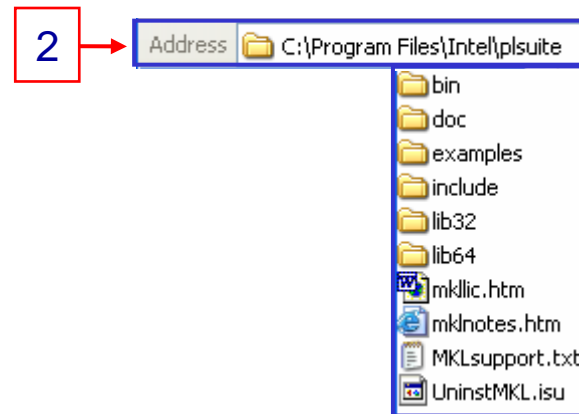
http://intel.com/home/tech/resource_library.htm

Installing the Intel Library to increase computing time.

Steps:

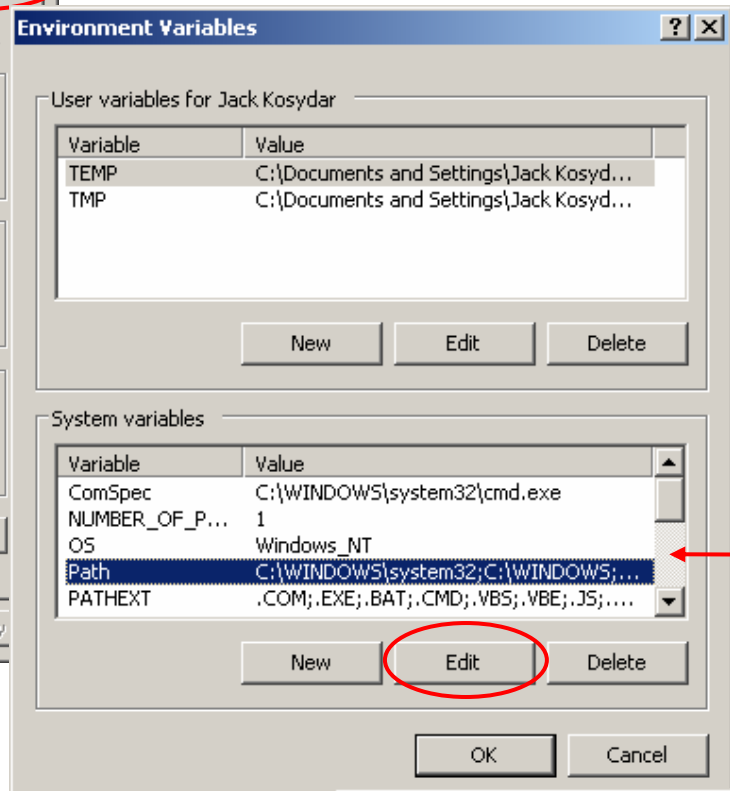
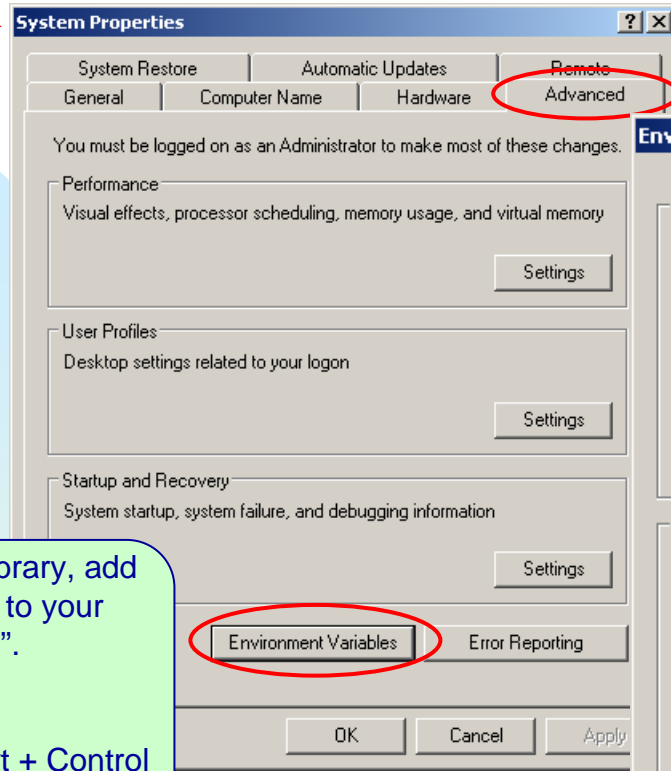
1. This Intel Library should be downloaded and installed.
2. Example location of installed MKL51B.exe.

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



Info Nugget – Running with the Intel MKL Library

1



2

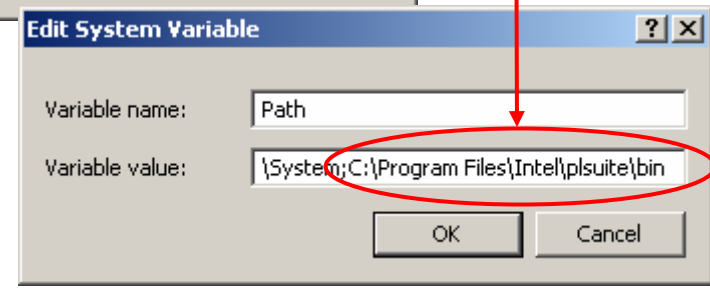
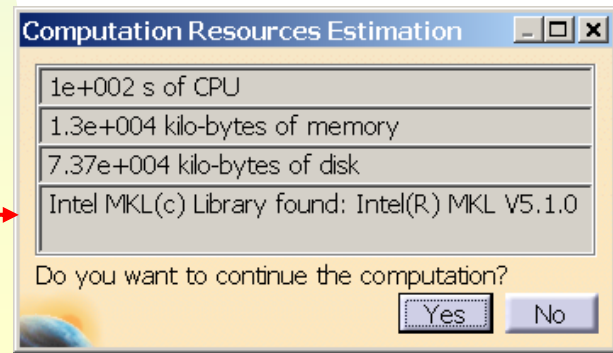
3

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

4



Info Nugget – Paging Space

The image shows a sequence of three Windows dialog boxes. The first is the 'System Properties' window with the 'Advanced' tab selected. The second is the 'Performance Options' window with the 'Advanced' tab selected. The third is the 'Virtual Memory' dialog box for drive C: with 'Custom size' selected and 'Set' button highlighted. Red circles and arrows with numbers 1, 2, and 3 point to the 'Advanced' tab, the 'Change' button, and the 'Set' button respectively.

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.

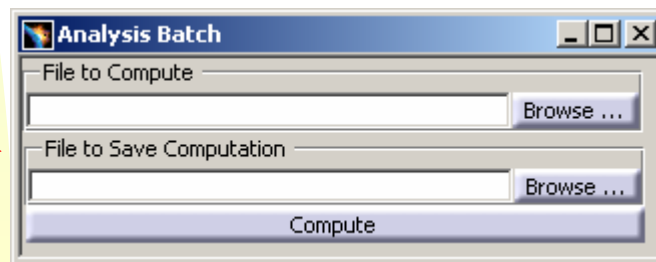
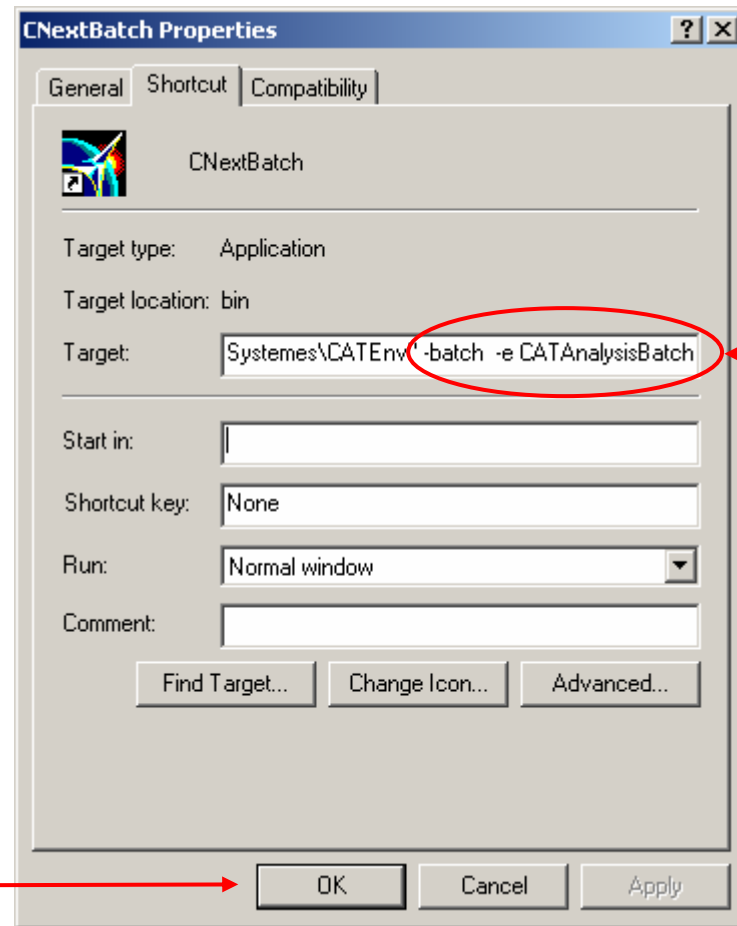
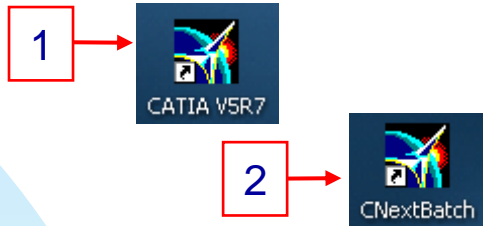
CATIA error note will be "not enough memory"

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



Info Nugget – Material Catalog

State	Name	Path
Opened	ws3pedal.CATAnalysis	C:\CATIA\Training\data
Opened	pedal.CATAnalysisResults	C:\ELFINI\pedal
Opened	pedal.CATAnalysisComputations	C:\ELFINI\pedal
Opened	ws3pedal.CATPart	C:\CATIA\Training\data
Opened	Catalog.CATMaterial	C:\Program Files\Dassault Systemes\B07\intel_a\startup\materials

1

Edit the existing material catalog.

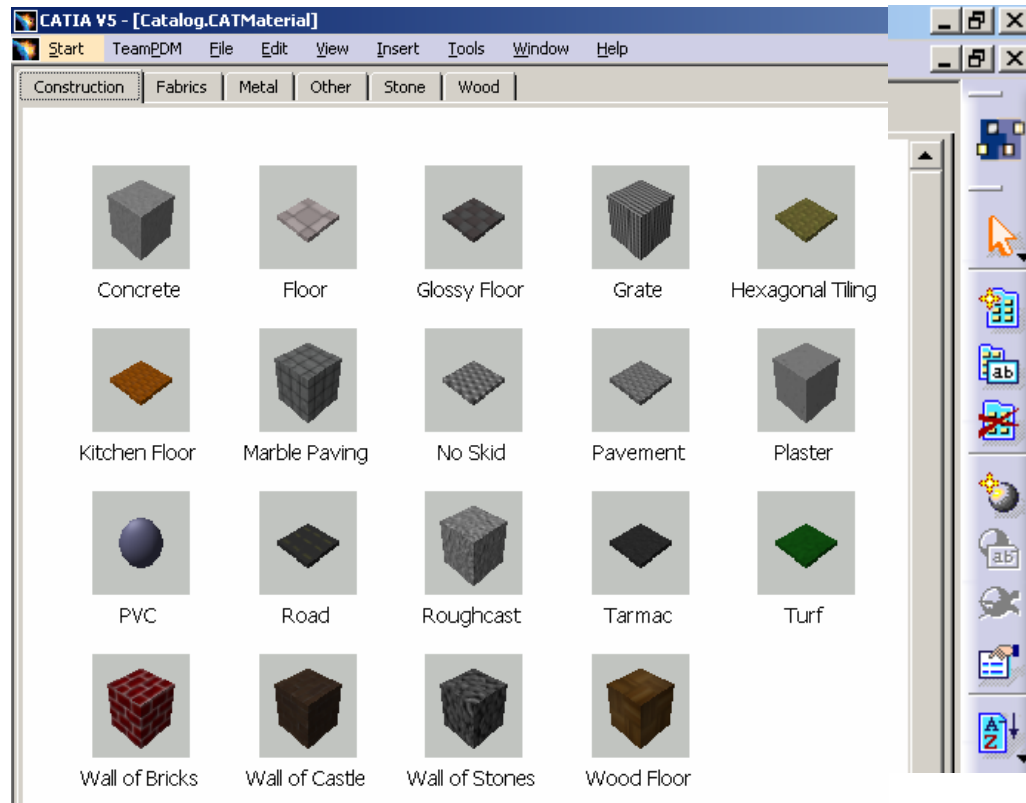


Steps:

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.

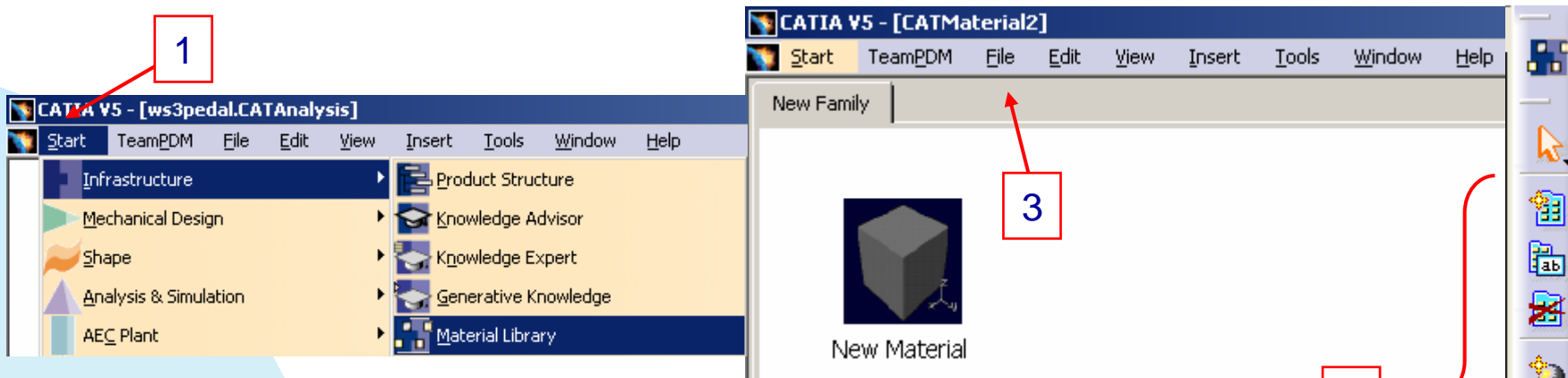
3. Edit this file with the various tools provided and save.



2

3

Info Nugget – Personal Material Catalog



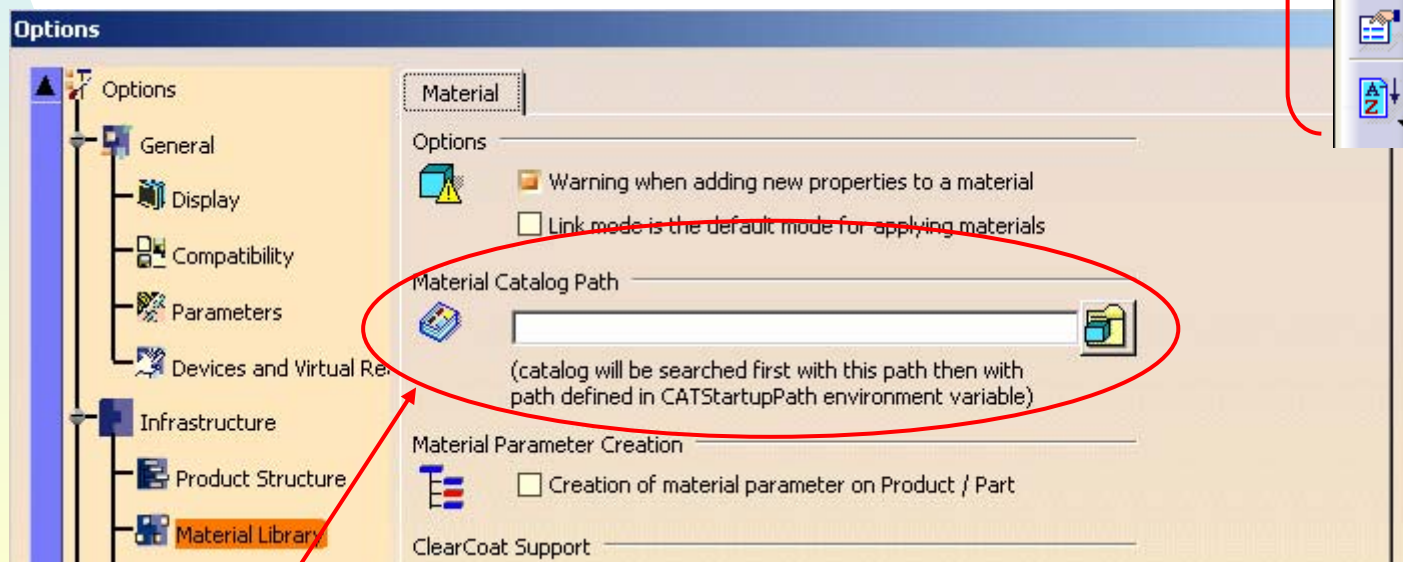
Steps:

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.

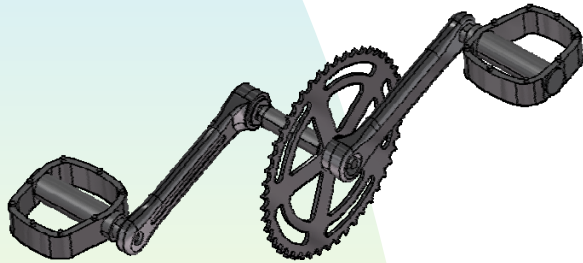


4



WORKSHOP 5

CRANK ANALYSIS USING VIRTUAL PARTS

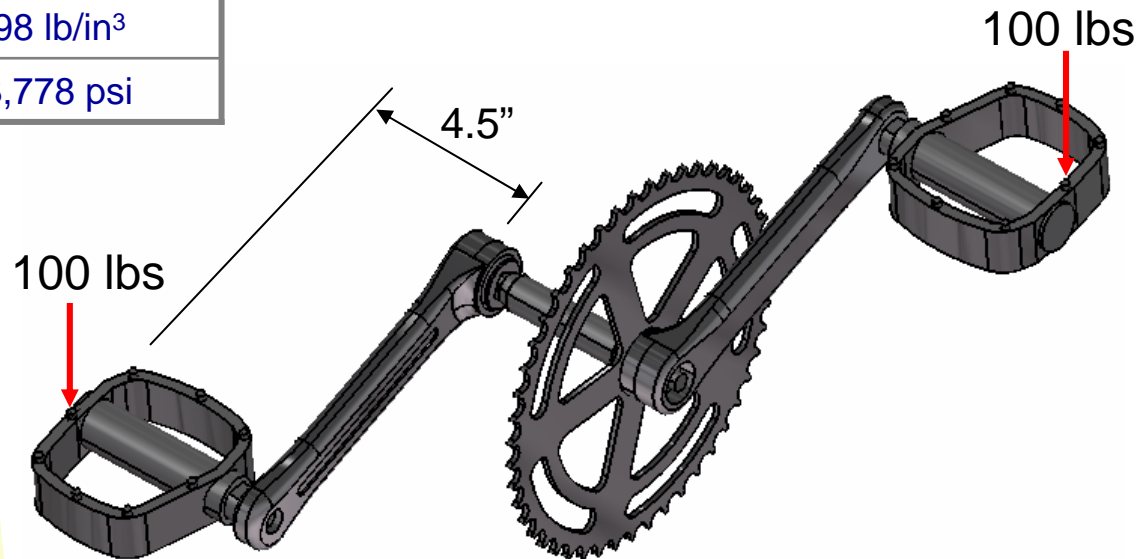


■ Problem Description

- ◆ 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.

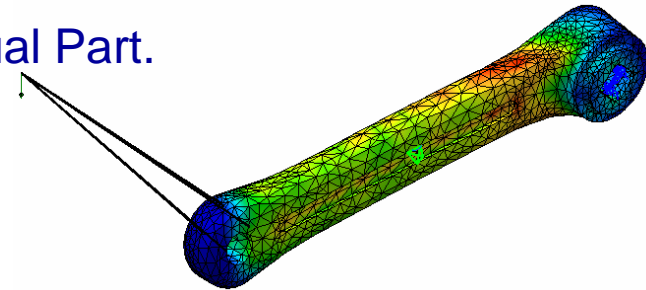
Aluminum

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

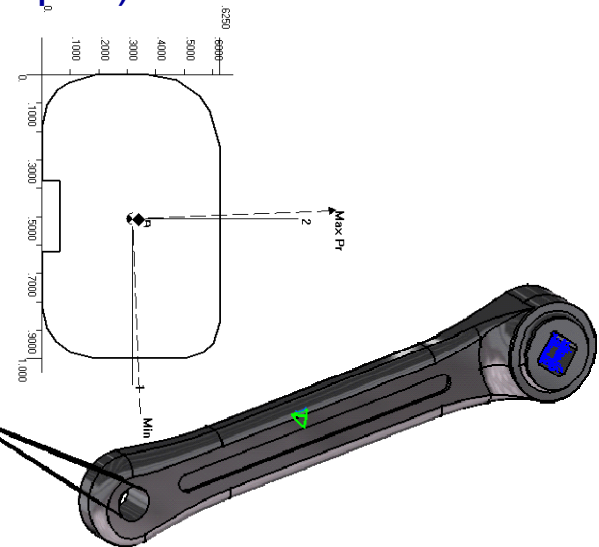
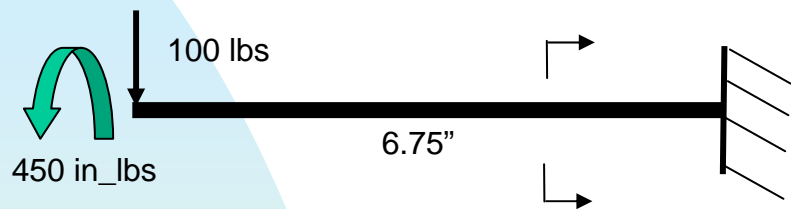
1. Open the existing CATIA part in the Part Design workbench.
2. Apply aluminum material properties to the part.
3. Create a new CATIA analysis document (.CATAnalysis).
4. Mesh globally with linear elements.
5. Apply a clamp restraint.
6. Simulate the pedal using a Smooth Virtual Part.
7. Apply a force 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.



WORKSHOP 5 – BICYCLE CRANK

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

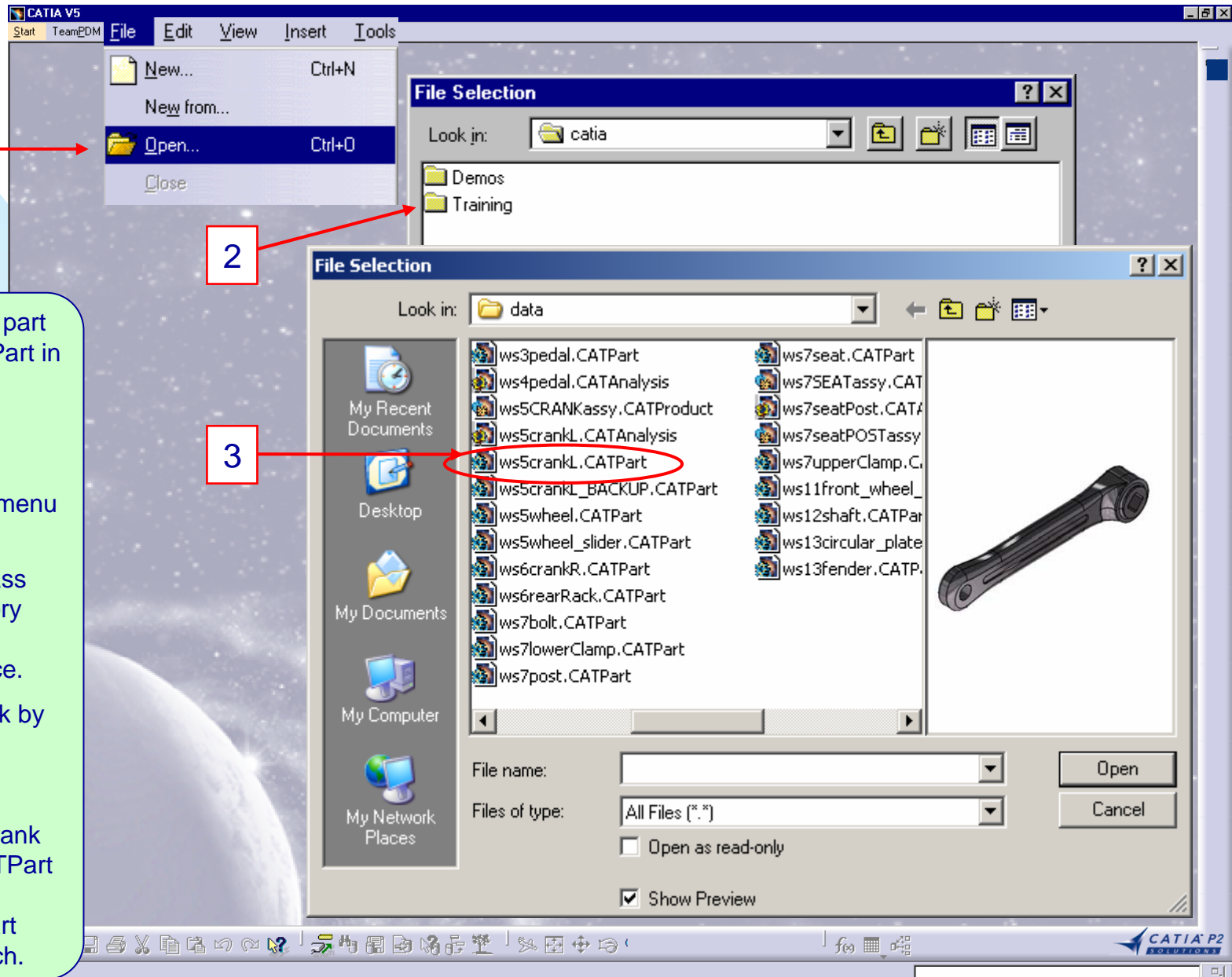
$$\text{Moment} = P \cdot L = 100\text{lbs} \cdot 6.75\text{inches} = 675\text{inlbs}$$

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

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

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

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 Aluminium.
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.

Property	Value
Young Modulus	1.015e+007psi
Poisson Ratio	0.346
Density	0.098lb_in3
Thermal Expansion	0.0000236
Yield Strength	13778.589psi

Step 3. Create a new CATIA analysis document

The screenshot shows the CATIA V5 interface with the 'Start' menu open. The 'Analysis & Simulation' option is selected, and the 'Generative Structural Analysis' sub-option is highlighted. The Analysis Manager tree on the left shows a 'Static Case' with various sub-elements like 'Restrains.1', 'Loads.1', and 'Static Case Solution.1'. The 'New Analysis Case' dialog box is open, showing 'Static Analysis' selected. Red boxes and arrows indicate the following 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.

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.

The screenshot shows the CATIA V5 interface with the Analysis Manager tree on the left. The tree includes: Analysis Manager, Links Manager (with Results, Computations, and Link.1), Finite Element Model, Nodes and Elements, Properties.1 (with Material Property3D.1), Static Case (with Restraints.1, Loads.1, Static Case Solution.1, and Sensors.1), and Energy. A 3D model of a crank is shown in the center. A dialog box titled 'Current Storage Location' is open, showing 'Results Data' and 'Computation Data' fields with their respective paths and 'Modify' buttons. Red boxes and arrows indicate the steps: 1 points to the Storage Location icon in the bottom toolbar; 2 points to the 'Results Data' field; 3 points to the 'Computation Data' field; 4 points to a folder icon in the bottom toolbar; 5 points to the 'Links Manager' section in the tree.

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.

CATIA V5 - [crankL.CATAnalysis]

Start TeamPDM File Edit View Insert Tools Window Help

Analysis Manager

- Links Manager
 - Results -> C:\ELFINI\crankL\crankL.CATAnalysisResults
 - Computations -> C:\ELFINI\crankL\crankL.CATAnalysisComputations
 - Link.1 -> C:\CATIA\Training\data\crankL.CATPart
- Finite Element Model
 - OCTREE Tetrahedron Mesh.1 : CrankL** (1)
 - Properties.1
 - Material Property3D.1
 - Static Case
 - Restraints.1
 - Loads.1
 - Static Case Solution.1
 - Sensors.1
 - Energy

OCTREE Tetrahedron Mesh.1 : CrankL

Global Local

Size 0.25in (2)

Sag 0.025in (3)

Element type

Linear (4)

Parabolic

OK Cancel

object or a command

CATIA P2 SOLUTIONS

Step 5. Apply a clamp restraint

CATIA V5 - [crankL.CATAnalysis]

Start TeamPDM File Edit View Insert Tools Window Help

Analysis Manager

- Links Manager
- Finite Element Model
 - Nodes and Elements
 - Properties.1
 - Static Case
 - Restraints.1
 - Clamp.1
 - Loads.1
 - Static Case Solution.1
 - Sensors.1

1

2

3

Clamp

Name Clamp.1

Supports 4 Faces

OK Cancel

object or a command

Steps:

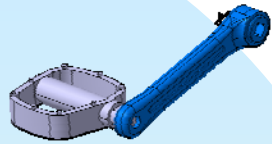
1. Select the Clamp Restraint icon.



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



Point Definition

Point type: Coordinates

X = 6.75in

Y = -4.5in

Z = 0in

Coordinates in absolute axis system

Reference

Point: Point.2

OK Apply Cancel

1

2

3

4

0.937in

4.5 inches

z

y

Object or a command

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).

Step 6. Simulate the pedal using a Smooth Virtual Part

Steps:

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

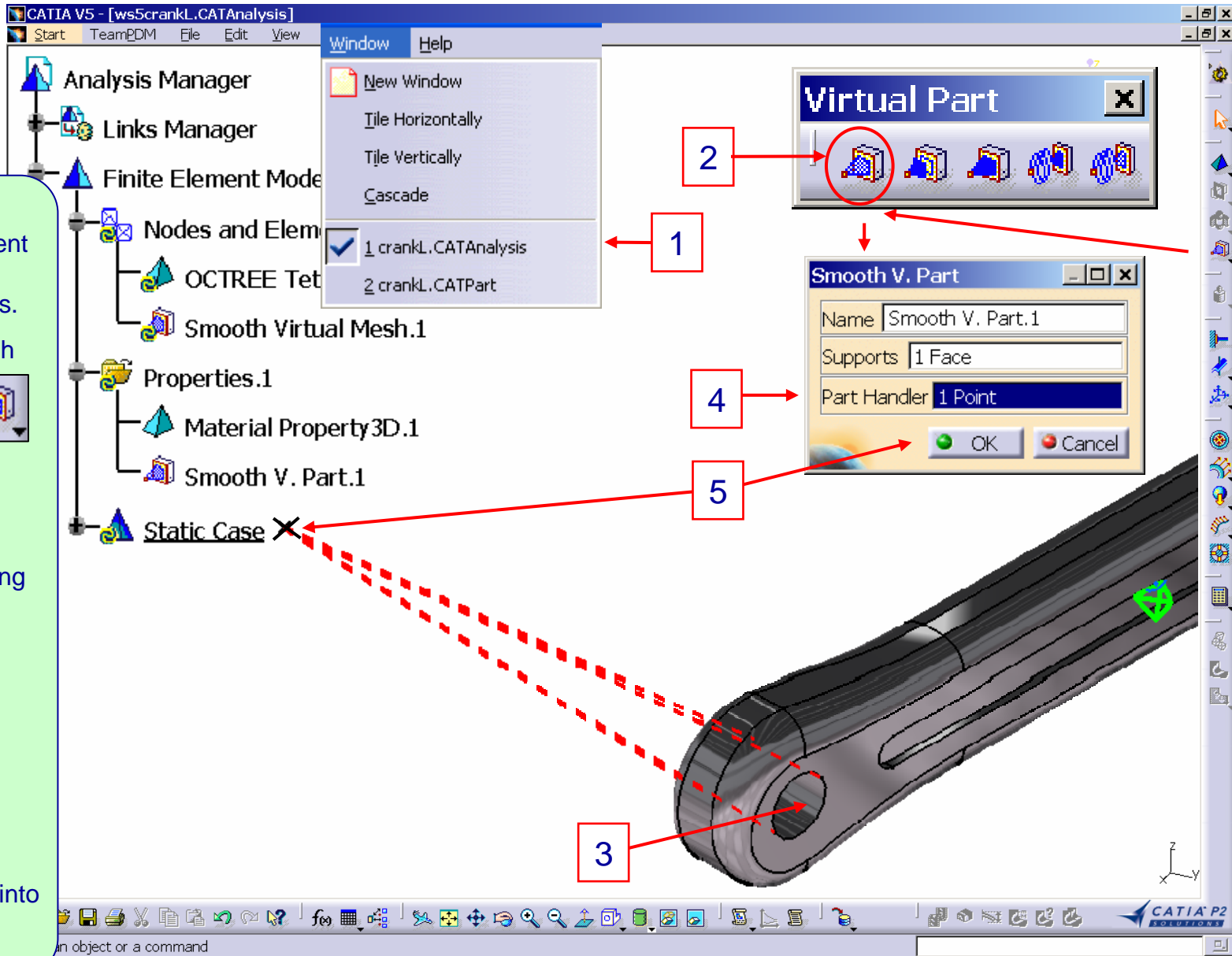
2. Select the smooth virtual part icon.

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.



Step 7. Apply a force to the Smooth Virtual Part

Analysis Manager

- Links Manager
- Finite Element Model
 - Nodes and Elements
 - OCTREE Tetrahedron Mesh.1 : CrankL
 - Smooth Virtual Mesh.1
 - Properties.1
 - Material Property3D.1
 - Smooth V. Part.1
 - Static Case

Distributed Force

Name: Distributed Force.1

Supports: 1 Virtual part

Axis System: Global

Type: Global

Display locally

Force Vector

Norm: 100lbf

X: 0lbf

Y: 0lbf

Z: -100lbf

OK Cancel

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.

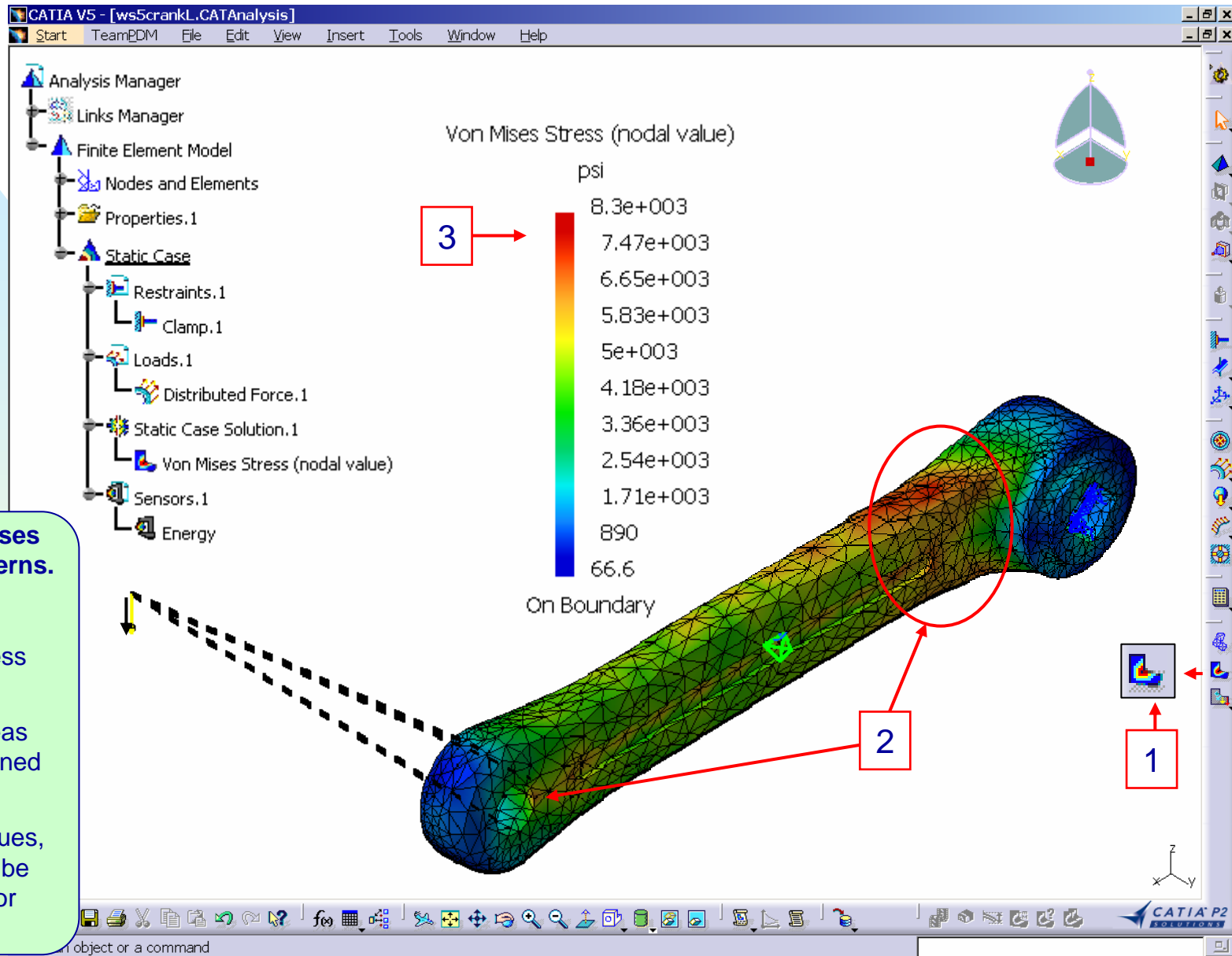
Computation Resources Estimation

7 s of CPU
2.29e+003 kilo-bytes of memory
6.93e+003 kilo-bytes of disk
Intel MKL(c) Library found: Intel(R) MKL V5.1.0

Do you want to continue the computation?

Yes No

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.

CATIA V5 - [ws5crankL.CATAnalysis]

Start TeamPDM File Edit View Insert Tools Window Help

Analysis Manager

- Links Manager
- Finite Element Model
 - Nodes and Elements
 - Properties.1
 - Static Case
 - Restraints.1
 - Clamp.1
 - Loads.1
 - Distributed Force.1
 - Static Case Solution.1
 - Von Mises Stress (nodal value)
 - Estimated local error
 - Sensors.1
 - Energy

Estimated local error
MAX: 1.01394e-006 Btu
min: 5.63623e-013 Btu

Color Map Editor

- On Boundary
- Number of Colors: 12
- Imposed Max: 1e-7
- Imposed Min: 5.63623e-013
- More >>
- OK Apply Cancel

4a

4b

1

2

3

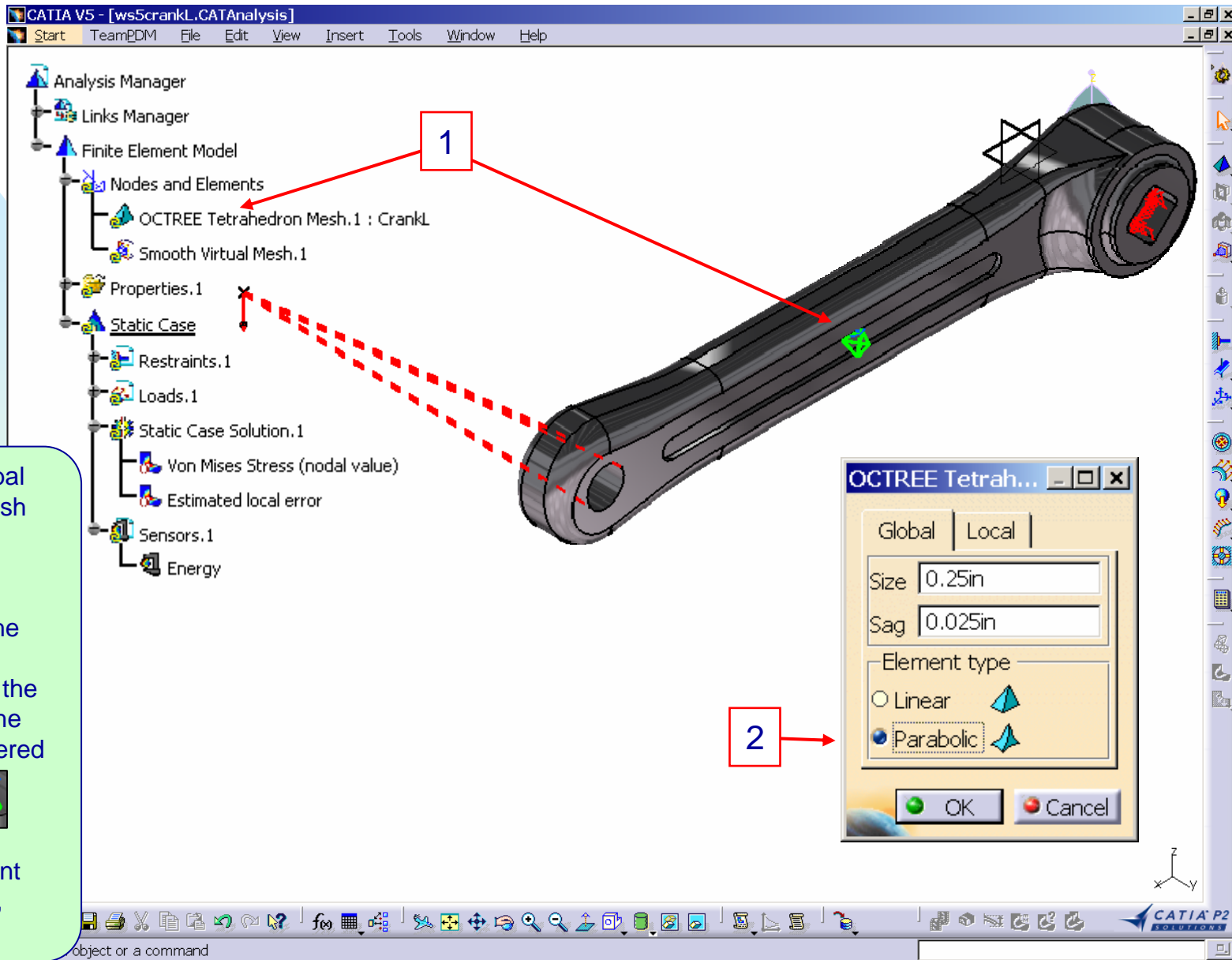
Precision Location : Global
Estimated Precision : 0.171748 Btu
Strain Energy : 0.389259 Btu
Global Estimated Error Rate : 42.5132 %

Estimated local error color map scale:
1e-007
9e-008
8e-008
7e-008
6e-008
5e-008
4e-008
3e-008
2e-008
1e-008
5.64e-013

object or a command

CATIA P2 SOLUTIONS

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.

Step 10. Change mesh to parabolic and add local meshing

1

2

3

Faces

Edge

OCTREE Tetrahedron Mesh.1: CrankL

Local Mesh Size

Local Mesh Sag

Smooth Virtual Mesh.1

Properties.1

Static Case

Global Local

Available specs :

Local size

Local sag

Add

OK Cancel

Local Mesh Size

Name Local Mesh Size

Supports 9 Faces

Value 0.125in 3 Edges

OK Cancel

Select an object or a command

CATIA P2 SOLUTIONS

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

CATIA V5 - [ws5crankL.CATAnalysis]

Start TeamPDM File Edit View Insert Tools Window Help

Analysis Manager

- Links Manager
- Finite Element Model
 - Nodes and Elements
 - OCTREE Tetrahedron Mesh.1 : CrankL
 - Local Mesh Size
 - Local Mesh Sag
 - Smooth Virtual Mesh.1
 - Properties.1
 - Static Case

Faces

Edge

OCTREE Tetrahedron Mesh.1 : CrankL

Global Local

Available specs :

- Local size
- Local sag

Add

OK Cancel

Local Mesh Sag

Name Local Mesh Sag

Supports 9 Faces

Value 0.013in 3 Edges

OK Cancel

object or a command

CATIA P2 SOLUTIONS

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.

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.

The screenshot shows the CATIA V5 interface with the following elements:

- Analysis Manager:** Shows the hierarchy: Links Manager, Finite Element Model, Nodes and Elements, OCTREE Tetrahedron Mesh.1 : CrankL (with sub-items: Local Mesh Size, Local Mesh Sag, Local Mesh Size, Local Mesh Sag), Smooth Virtual Mesh.1, Properties.1, and Static Case.
- Local Mesh Size Dialog:** Shows 'Local size' selected in the 'Available specs' list. The 'Value' field contains '0.125in' and '1 Face' is selected in the 'Supports' list. The 'OK' button is highlighted.
- Local Mesh Sag Dialog:** Shows 'Local sag' selected in the 'Available specs' list. The 'Value' field contains '0.013in' and '1 Face' is selected in the 'Supports' list. The 'OK' button is highlighted.
- Mesh Visualization:** A cylindrical part is shown with a mesh. A green triangle highlights a specific face on the mesh, corresponding to the '1 Face' selection in the dialog boxes.

Select an object or a command

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.

Computation Resources Estimation

6e+002 s of CPU
3.11e+004 kilo-bytes of memory
2.18e+005 kilo-bytes of disk
Intel MKL(c) Library found: Intel(R) MKL V5.1.0

Do you want to continue the computation?

Yes No

Step 12. Visualize extremas

Von Mises Stress (nodal value)

psi

1.53e+004
1.38e+004
1.23e+004
1.07e+004
9.2e+003
7.66e+003
6.13e+003
4.6e+003
3.07e+003
1.54e+003
9.64

On Boundary

Von Mises Stress (nodal value) Global Maximum.1: 15319.8 psi

1

2

3

Extrema Creation

Global

Minimum extrema at most 0

Maximum extrema at most 2

Local

Minimum extrema at most 0

Maximum extrema at most 2

OK Cancel

Select an object or a command

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

CATIA V5 - [ws5crankL.CATAnalysis]

Start TeamPDM File Edit View Insert Tools Window Help

Analysis Manager
Links Manager
Finite Element Model
Nodes and Elements
Properties.1
Static Case
Restrains.1
Loads.1
Static Case Solution.1
Von Mises Stress (nodal value)
Estimated local error
Extrema
Global Maximum.1
Sensors.1

Estimated local error
MAX: 5.73092e-008 Btu
min: 1.26656e-015 Btu

1e-008
9e-009
8e-009
7e-009
6e-009
5e-009
4e-009
3e-009
2e-009
1e-009
1.27e-015

Color Map Editor
 On Boundary
Number of Colors 10
 Imposed Max 1e-008
 Imposed Min 5.63623e-013
More >>
OK Apply Cancel

4

Estimated local error Global Maximum, 1: 5.73092e-008 Btu

Extrema Creation
 Global
Minimum extrema at most 0
Maximum extrema at most 2
 Local
Minimum extrema at most 0
Maximum extrema at most 2
OK Cancel

3

1

2

Select an object or a command

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.



The screenshot shows the CATIA V5 interface with the following elements:

- Features Tree:** Shows the hierarchy from Analysis Manager down to Adaptivity Process, including Adaptivity Convergence.1, Adaptivities.1, and two Adaptivity Boxes (Adaptivity Box.1 and Adaptivity Box.2).
- Adaptivity Box Dialogs:** Two dialog boxes are open. The first, 'Adaptivity Box.1', shows Objective Error (%) at 5, Solution 'Static Case Solution.1', and Local Error (%) at 7.8915. The second, 'Adaptivity Box.2', shows Objective Error (%) at 5, Solution 'Static Case Solution.1', and Local Error (%) at 3.7461. Both have 'Select Extremum', 'OK', and 'Cancel' buttons.
- 3D Model:** A blue meshed part is shown with two red wireframe boxes (Adaptivity Box.1 and Adaptivity Box.2) and one orange wireframe box. Green dots are visible on the mesh.
- Informations Panel:** Located at the bottom left, it displays: Precision Location : Global, Estimated Precision : 0.00558142 Btu, Strain Energy : 8.519379 Btu, and Global Estimated Error Rate : 7.31057 %.
- Numbered Callouts:** Red boxes with numbers 1, 2, 3, and 4 point to the information icon, the adaptivity box icon, the 'Select Extremum' button, and the 'Adaptivity Box.1' dialog box respectively.

Step 14. Visualize final results

CATIA V5 - [ws5crankL.CATAnalysis]

Start TeamPDM File Edit View Insert Tools Window Help

Analysis Manager

- Links Manager
- Finite Element Model
 - Nodes and Elements
 - Properties.1
 - Static Case
 - Restraints.1
 - Loads.1
 - Static Case Solution.1
 - Von Mises Stress (nodal value)
 - Estimated local error
 - Deformed Mesh
 - Sensors.1
 - Adaptivity Process

Animate Window

7

Steps Number 10

Speed

Close

1

2

Visualize exaggerated Deformation.

Steps:

1. Select the Deformation icon.
2. Animate the deformation image.

CATIA P2 SOLUTIONS

Step 14. Visualize final results

CATIA V5 - [ws5crankL.CATAnalysis]

Start TeamPDM File Edit View Insert Tools Window Help

Analysis Manager

Links Manager

Finite Element Model

Nodes and Elements

Properties.1

Static Case

Restraints.1

Loads.1

Static Case Solution.1

Von Mises Stress (nodal value)

Estimated local error

Deformed Mesh

Translational displacement vector

Sensors.1

Adaptivity Process

0.000266 = x
0.00328 = y
-0.0919 = z

Translational displacement vector

in

0.092
0.0828
0.0736
0.0644
0.0552
0.046
0.0368
0.0276
0.0184
0.0092
0

On Boundary

1

object or a command

CATIA P2 SOLUTIONS

Add the displacement image

Steps:

1. Select the displacement icon to add this image.

Step 14. Visualize final results

Von Mises Stress (nodal value) Global Maximum.1: 15319.8 psi

Von Mises Stress (nodal value) psi

1.53e+004
1.38e+004
1.23e+004
1.07e+004
9.2e+003
7.66e+003
6.13e+003
4.6e+003
3.07e+003
1.54e+003
9.64

On Boundary

1

2

Center Graph
Reframe On
Hide/Show
Global Maximum.1 object
Focus On

Analysis Manager
Links Manager
Finite Element Model
Nodes and Elements
Properties.1
Static Case
Restrains.1
Loads.1
Static Case Solution.1
Von Mises Stress (nodal value)
Extrema
Global Maximum.1
Estimated local error
Deformed Mesh
Sensors.1
Adaptivity Process

To Focus On Extremum

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

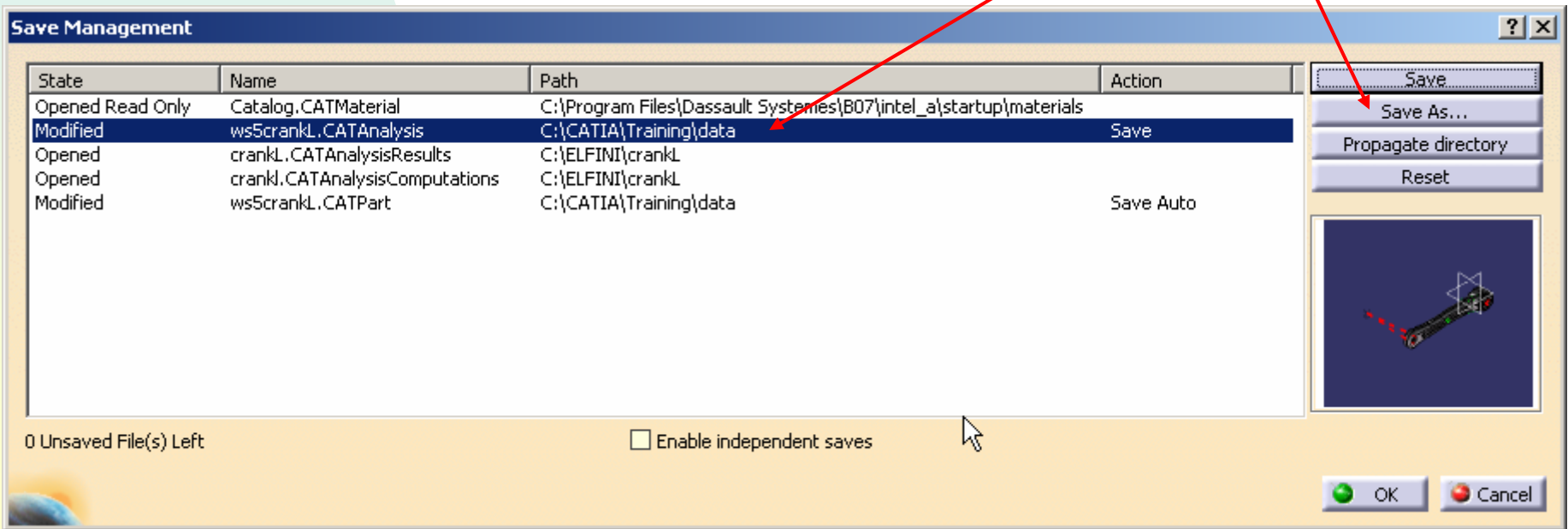
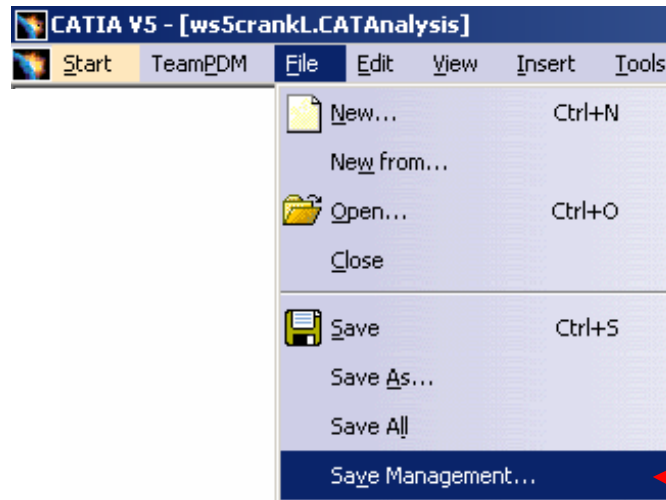
- ◆ New material is required with a yield strength higher than 15.3 ksi.

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	42.5 %	7.3 %
Local % Precision error	NA %	7.9 % and 3.7%

Step 15. Save the analysis document

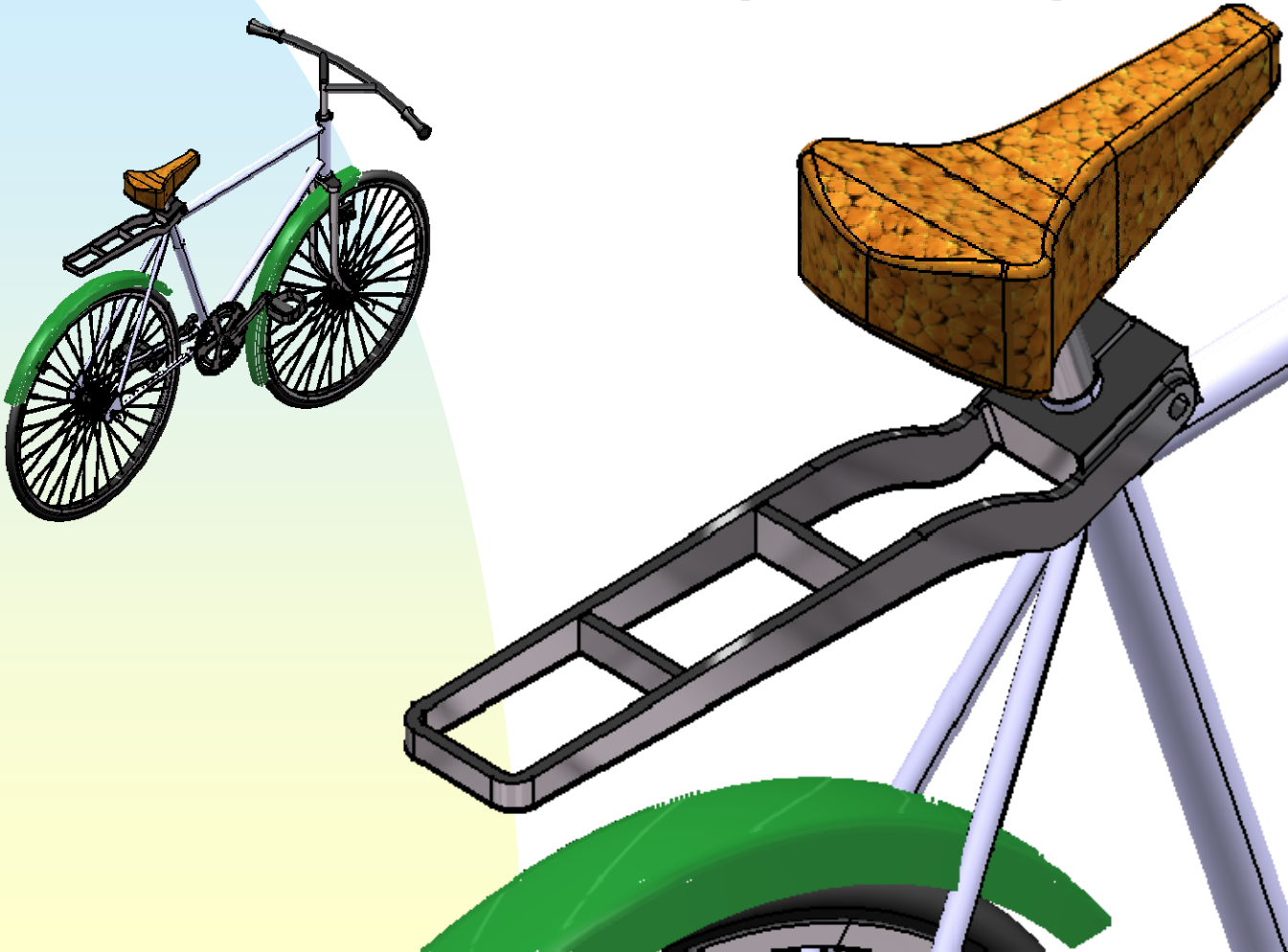
Steps:

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

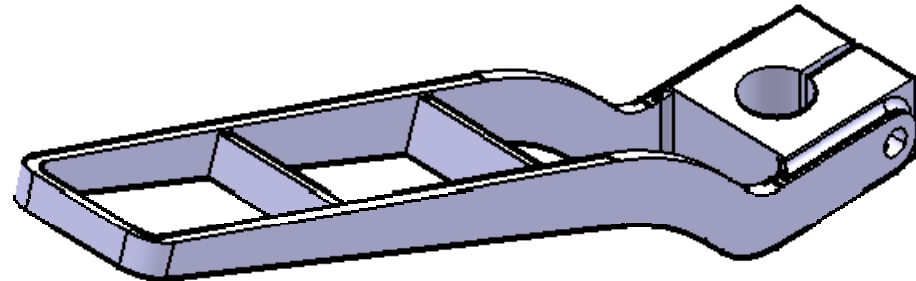


■ Problem Description

- ◆ 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

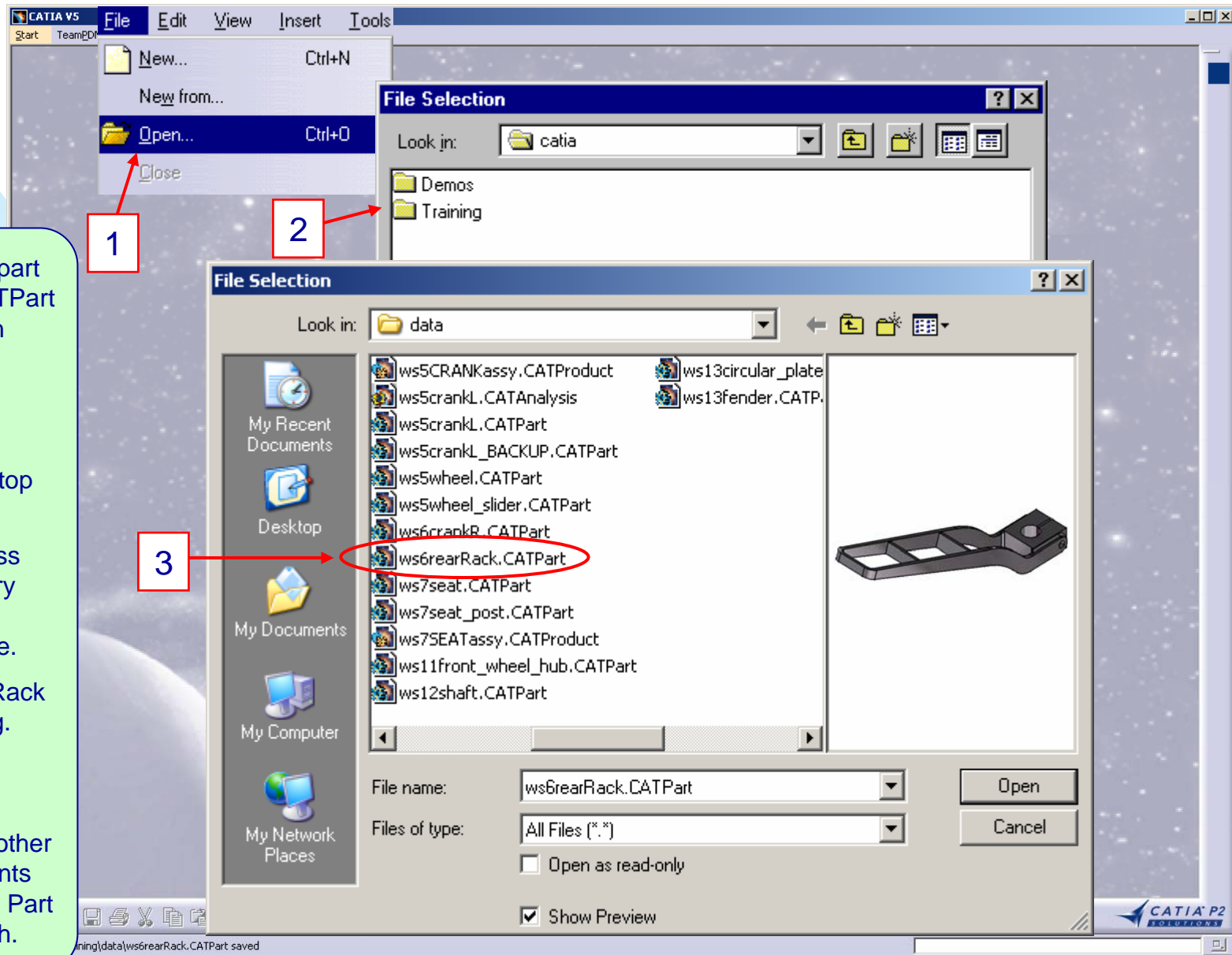
Elastic Modulus, E	10.15E6 psi
Poisson's Ratio, ν	0.346
Density	.098 lb/in ³
Yield Strength	13,778 psi



■ 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

1 → rearRack

- xy plane
- yz plane
- zx plane
- PartBody
- Open_body.1
- Aluminium

2 → [Apply Material Icon]

3 → Metal

4 → Aluminium

5 → OK

6 → Aluminium

Library (ReadOnly)

Construction | Fabrics | **Metal** | Other | Stone | Wood

Aluminium	Brass	Bronze
Gold	Iron	Lead
Silver	Steel	Titanium

Link to file OK Apply Material Close

Part or a command

CATIA V5 - [ws6rearRack.CATPart]

Start TeamPDM File Edit View Insert Tools Window Help

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.

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.

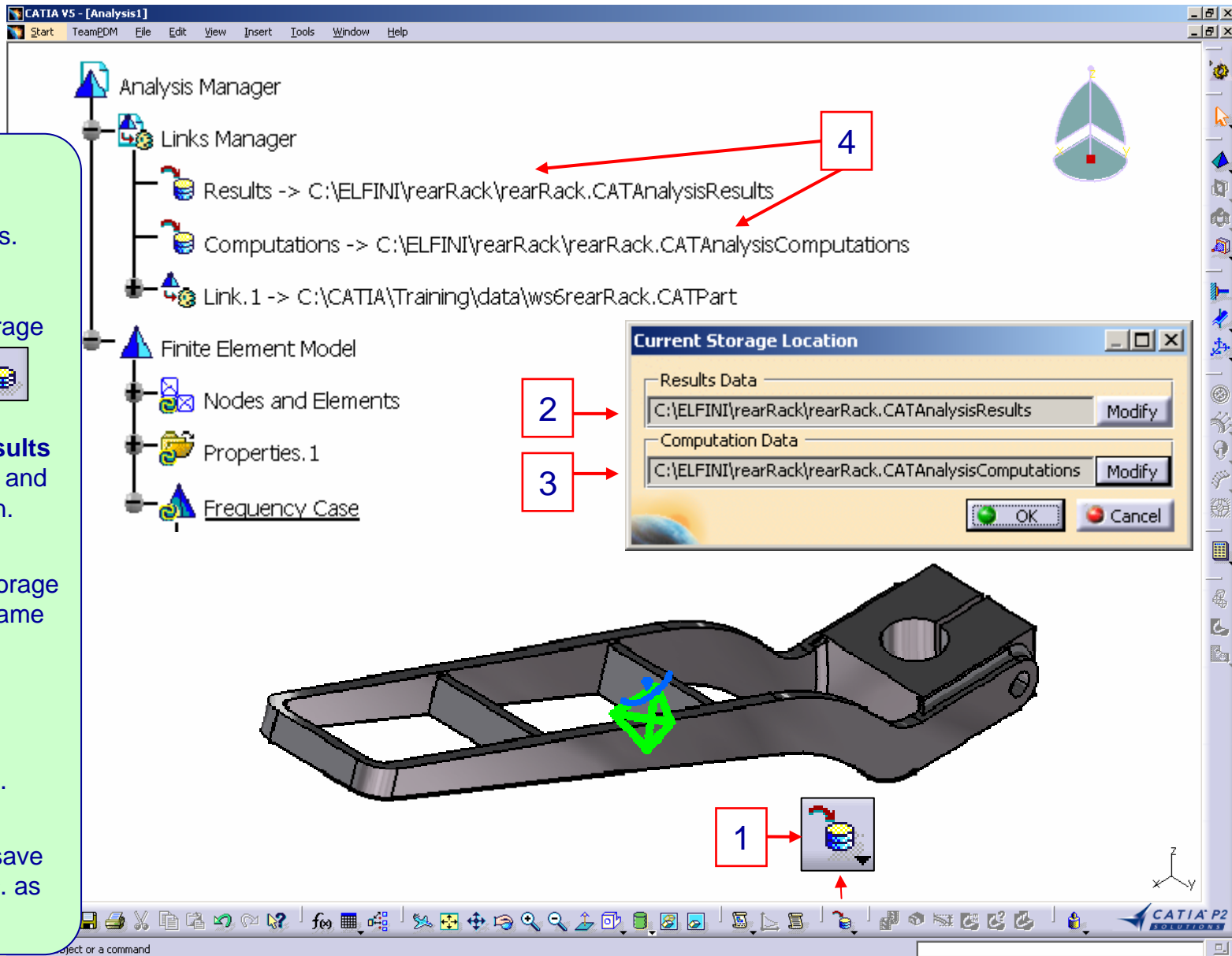


2. Modify the **Results** Storage Location and rename as shown.

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".



The screenshot displays the CATIA V5 software interface. The top menu bar includes 'Start', 'TeamPDM', 'File', 'Edit', 'View', 'Insert', 'Tools', 'Window', and 'Help'. The 'Analysis Manager' tree on the left shows the following structure:

- Analysis Manager
 - Links Manager
 - Results -> C:\ELFINI\rearRack\rearRack.CATAnalysisResults
 - Computations -> C:\ELFINI\rearRack\rearRack.CATAnalysisComputations
 - Link.1 -> C:\CATIA\Training\data\ws6rearRack.CATPart
 - Finite Element Model
 - Nodes and Elements
 - Properties.1
 - Frequency Case

The 'Current Storage Location' dialog box is open, showing the following settings:

- Results Data: C:\ELFINI\rearRack\rearRack.CATAnalysisResults (Modify)
- Computation Data: C:\ELFINI\rearRack\rearRack.CATAnalysisComputations (Modify)
- Buttons: OK, Cancel

Numbered callouts (1-4) indicate the steps: 1 points to the Storage Location icon in the Links Manager; 2 points to the Results Data field; 3 points to the Computation Data field; 4 points to the Results path in the Links Manager tree. A 3D model of a rear rack component is shown at the bottom, with a green arrow pointing to the Storage Location icon in the Links Manager tree.

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 = $\frac{1}{2}$ the thinnest section.

The screenshot shows the CATIA V5 interface with the following elements:

- Analysis Manager:** Shows the tree structure with 'OCTREE Tetrahedron Mesh.1 : rack' selected.
- Measure Dialogs:** A box pointing to the 'Measure...' icon in the software toolbar.
- Exit Measure:** A box pointing to the 'Exit Measure' icon in the software toolbar.
- OCTREE Tetrahedron Mesh.1 : rack:** A dialog box showing 'Size: 0.987in' and 'Sag: 0.158in'. A context menu is open over the 'Size' field with 'Measure...' and 'Add Range...' options.
- Measure Between:** A dialog box with 'Definition' (Measure type: Between, Reference mode: Any geometry, Target mode: Any geometry) and 'Results' (Selection 1: Line on Pad.4..., Selection 2: Line on Pad.2..., Minimum distance: 0.25in, Angle: 0deg, Components: X 0.236in, Y 0in, Z 0.083in, Reference point: X 9.741in, Y -0.932in, Z 6.353in, Target point: X 9.977in, Y -0.932in, Z 6.436in). It includes 'Keep Measure' and 'Customize...' options.
- Measure command ended:** A dialog box asking 'Do you want to copy the result of this measure in this parameter?' with 'Yes' and 'No' buttons.

Step 4. Pre-process initial finite element mesh

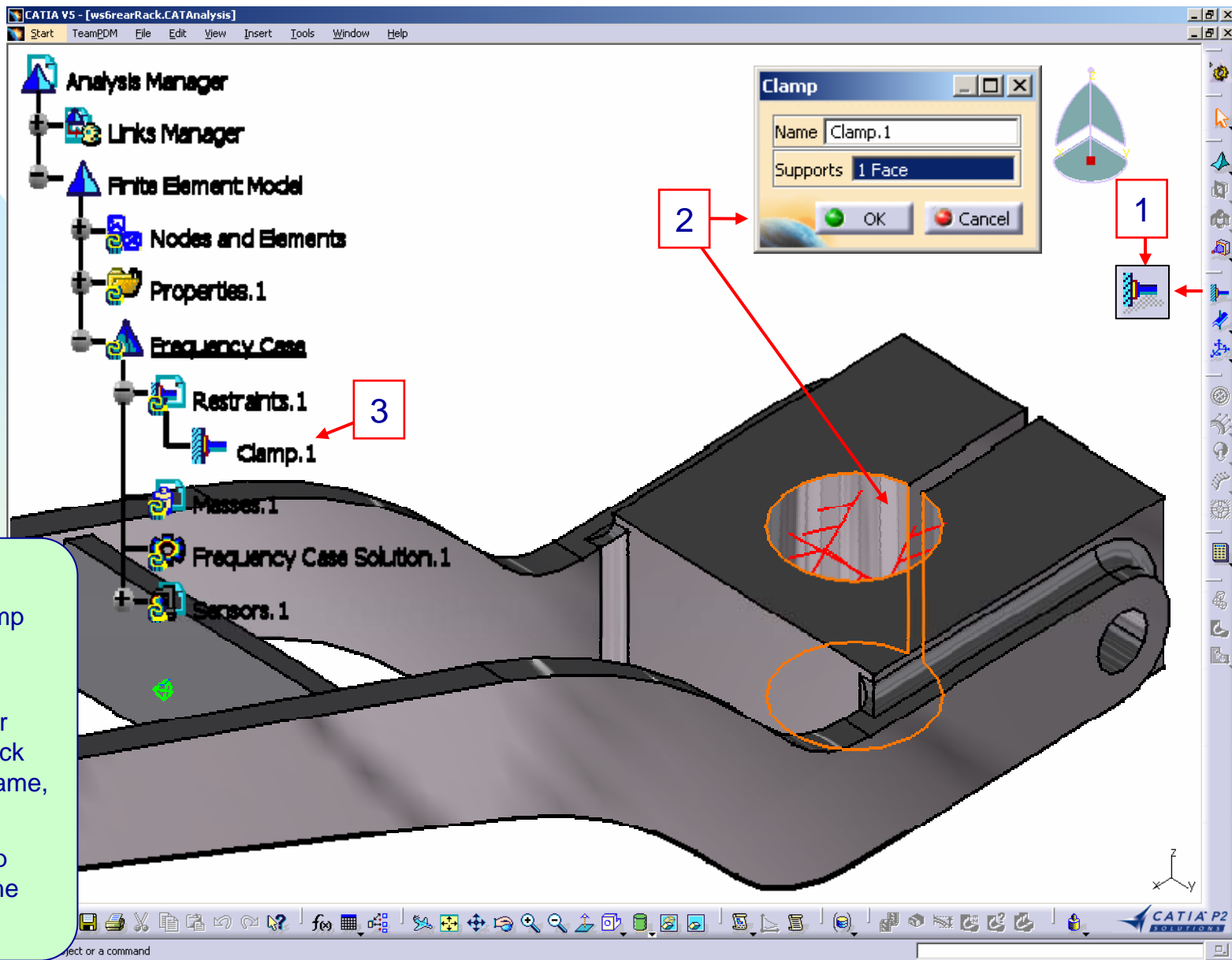
Define the global finite element mesh properties.

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.

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.

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

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, sparse problems (most structural models fit into this category).

Analysis Manager

- Links Manager
- Finite Element Model
 - Nodes and Elements
 - Properties.1
 - Frequency Case
 - Restraints.1
 - Masses.1
 - Frequency Case Solution.1
 - Sensors.1
 - Frequency 1

Frequency solution Param...

Number of modes: 5

Method:
 gauss
 lanczos

Dynamic parameters:
maximum iteration number: 50
accuracy: 0.001

OK Cancel

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 pattern from a natural frequency.

The screenshot shows the CATIA V5 interface with the following elements:

- Analysis Manager:** A tree view on the left showing the model structure, including 'Frequency Case', 'Restraints.1', 'Masses.1', 'Frequency Case Solution.1', and 'Sensors.1'.
- Image Edition Dialog:** A dialog box with tabs for 'Frequencies', 'Visu', 'Criteria', 'Filters', and 'Selections'. The 'Visu' tab is selected, and 'AVERAGE-ISO' is highlighted in the list. The 'Edition' section has buttons for 'Iso/Fringe', 'Symbol', and 'Axis System'.
- Image Iso Fringe Editor Dialog:** A dialog box with an 'ISO value' dropdown set to 'IsoContour', a checked 'ISO smooth' checkbox, and an unchecked 'Display Element Without Value' checkbox. It has 'OK' and 'Cancel' buttons.
- 3D Model:** A 3D visualization of a mechanical part with a color-coded displacement magnitude. A legend on the right shows a scale from 0 to 6.08 inches, with 'On Boundary' at the bottom.
- Annotations:** Red boxes and arrows labeled 1, 2, 3, 4a, and 4b indicate the steps described in the text.

Step 8. Visualize the analysis results

Analysis Manager

- Links Manager
- Finite Element Model
 - Nodes and Elements
 - Properties.1
 - Frequency Case
 - Restraints.1
 - Masses.1
 - Frequency Case Solution.1
 - Translational displacement magnitude
 - Sensors.1

Translational displacement Magnitude

- Mode 1 Primary Bending
6.08 inch
- Mode 2 Primary Bending
10.1 inch
- Mode 3 Torsion
8.28 inch
- Mode 4 Secondary Bending
13.3 inch
- Mode 5 Secondary Bending
15.8 inch

Image Edition

Deformed

Display on Deformed Mesh

Frequencies Visu Criter

Number of modes	Frequency (Hz)
1	9.47206
2	9.70803
3	31.663
4	40.501
5	61.3551

Edition

Iso/Fringe Symbol Axis System

OK Cancel Help

1

2

3

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.

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.

The screenshot shows the CATIA V5 interface with the following elements:

- Analysis Manager Tree:** A tree view on the left showing the hierarchy: Analysis Manager > Links Manager > Finite Element Model > Nodes and Elements > Properties.1 > Frequency Case > Restraints.1 > Masses.1 > Frequency Case Solution.1 > Translational displacement magnitude (highlighted with a red box and arrow labeled '1') > Sensors.1.
- 3D Model:** A 3D visualization of a gear rack with a color gradient from red to blue, indicating displacement magnitude.
- Frequencies Dialog:** A dialog box titled 'Frequencies' with a table of modes. A red box and arrow labeled '4' points to the table.
- Animate Window Dialog:** A dialog box titled 'Animate Window' with playback controls. A red box and arrow labeled '2' points to the 'Animate an Analysis Image' icon in the toolbar. A red box and arrow labeled '3' points to the 'Current Occurrence' field. A red box and arrow labeled '5' points to the playback controls.

Number of modes	Frequency (Hz)
1	9.47206
2	9.70803
3	31.663
4	40.501
5	61.3551

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.

Analysis Manager

- Links Manager
- Finite Element Model
 - Nodes and Elements
 - Properties. 1
 - Frequency Case
 - Restraints. 1
 - Masses. 1
 - Frequency Case Solution. 1
 - Translational displacement magnitude
 - Sensors. 1

Image Edition

Deformed

Display on Deformed Mesh

Frequencies Visu Criter

Number of modes	Frequency (Hz)
1	9.47206
2	9.70803
3	31.663
4	40.501
5	61.3551

Edition

Iso/Fringe Symbol Axis System

OK Cancel Help

Extrema Creation

Global

Minimum extrema at most 0

Maximum extrema at most 2

Local

Minimum extrema at most 0

Maximum extrema at most 2

OK Cancel



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.

The screenshot shows the CATIA V5 interface with the 'REAR RACK' analysis results. A red box with the number '4' points to the results area. The 'MESH' section contains a table:

Entity	Size
Nodes	97374
Elements	59000

The 'STRUCTURE COMPUTATION' section lists the following data:

- Number of nodes : 97374
- Number of elements : 59000
- Number of D.O.F. : 292122
- Number of Contact Elements : 0
- Number of Kinematic relations : 0

A red oval highlights the 'Number of D.O.F.' value, with a callout box stating 'If parabolic elements were used'. The 'Reporting options' dialog box is open, showing the following fields:

- Output directory : C:\ELFINI\rearRack
- Title of the report : Rear Bike Rack
- Choose the analysis case(s) : Frequency Case

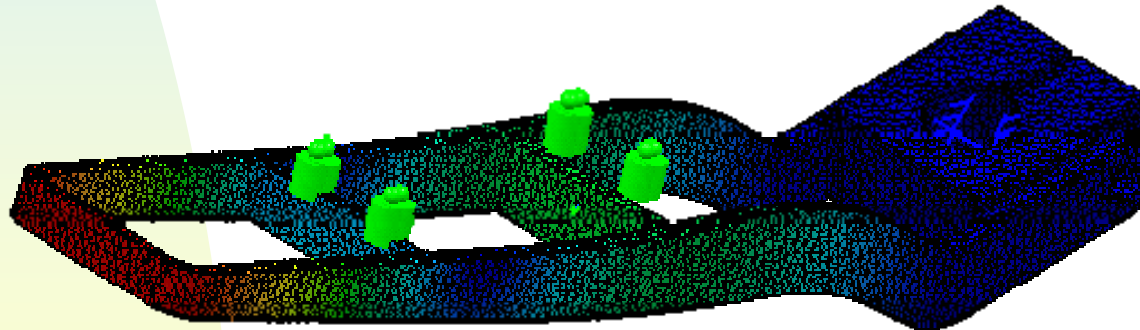
Red boxes with numbers 1, 2, and 3 point to the Basic Analysis Report icon, the Output directory field, and the Title of the report field, respectively. The 'OK' button is also highlighted with a red box.

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.

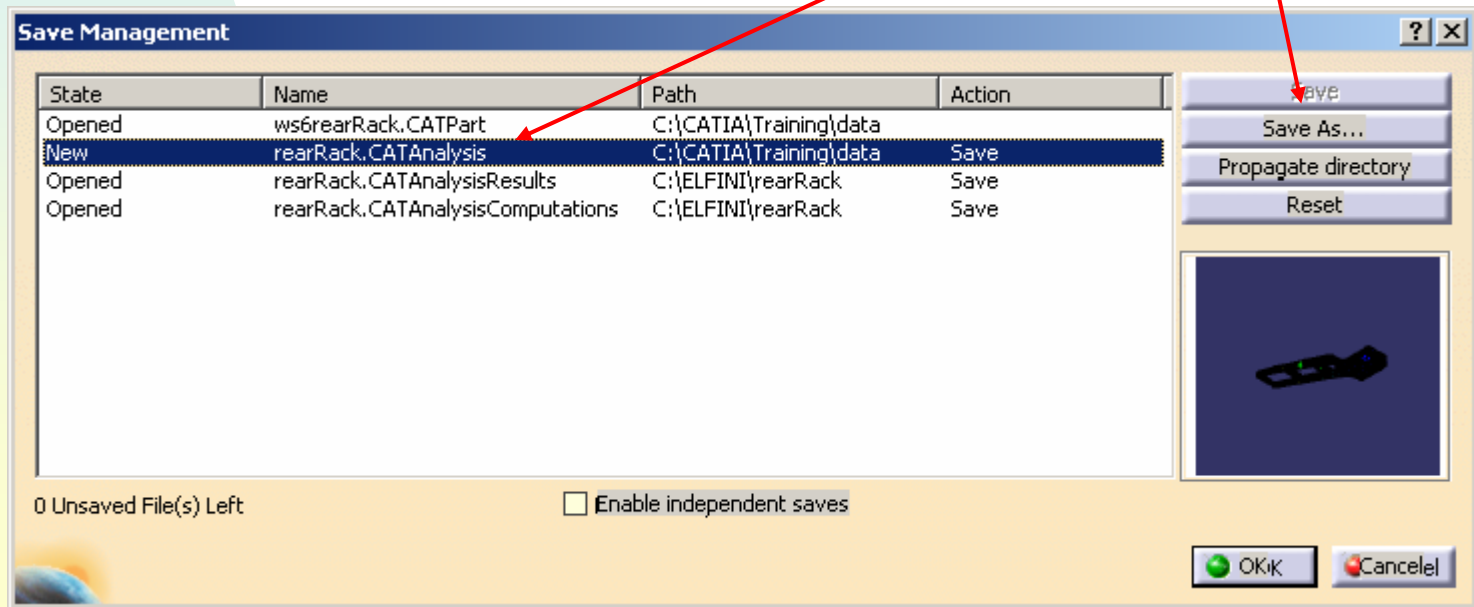
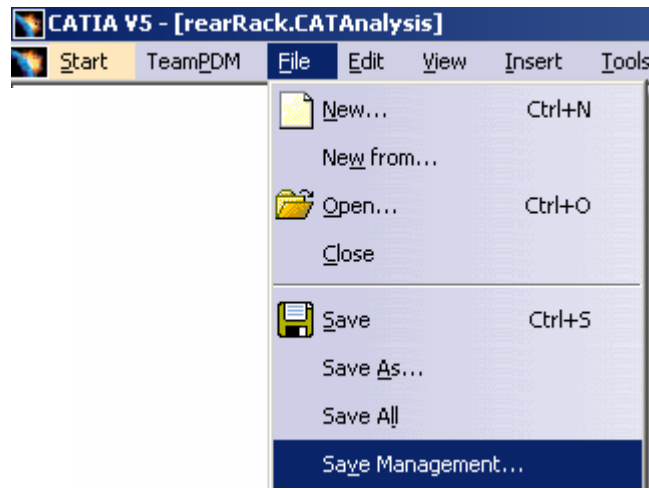
Mode Number	Bike Frequency Hz (cycles/sec)	Rack Frequency Hz Parabolic Elements
1	95	9.47
2	100	9.71
3	110	31.66
4	120	40.50
5	135	61.36



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



WORKSHOP 7 – SEAT POST

■ Problem Description

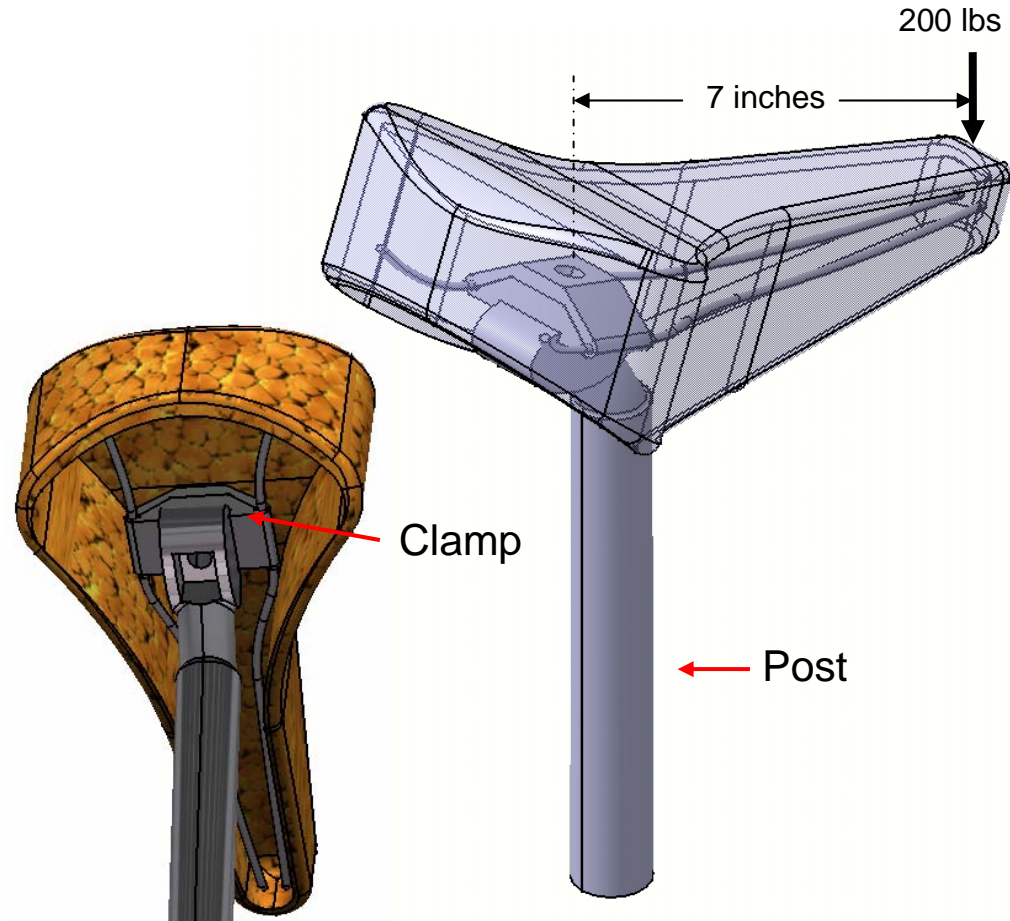
- ◆ The sales department has informed engineering that the seat post keeps breaking.
- ◆ Perform an assembled static analysis to determine why and recommend a solution. Be conservative by using a design case of 200 lbs forward on the seat.

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.



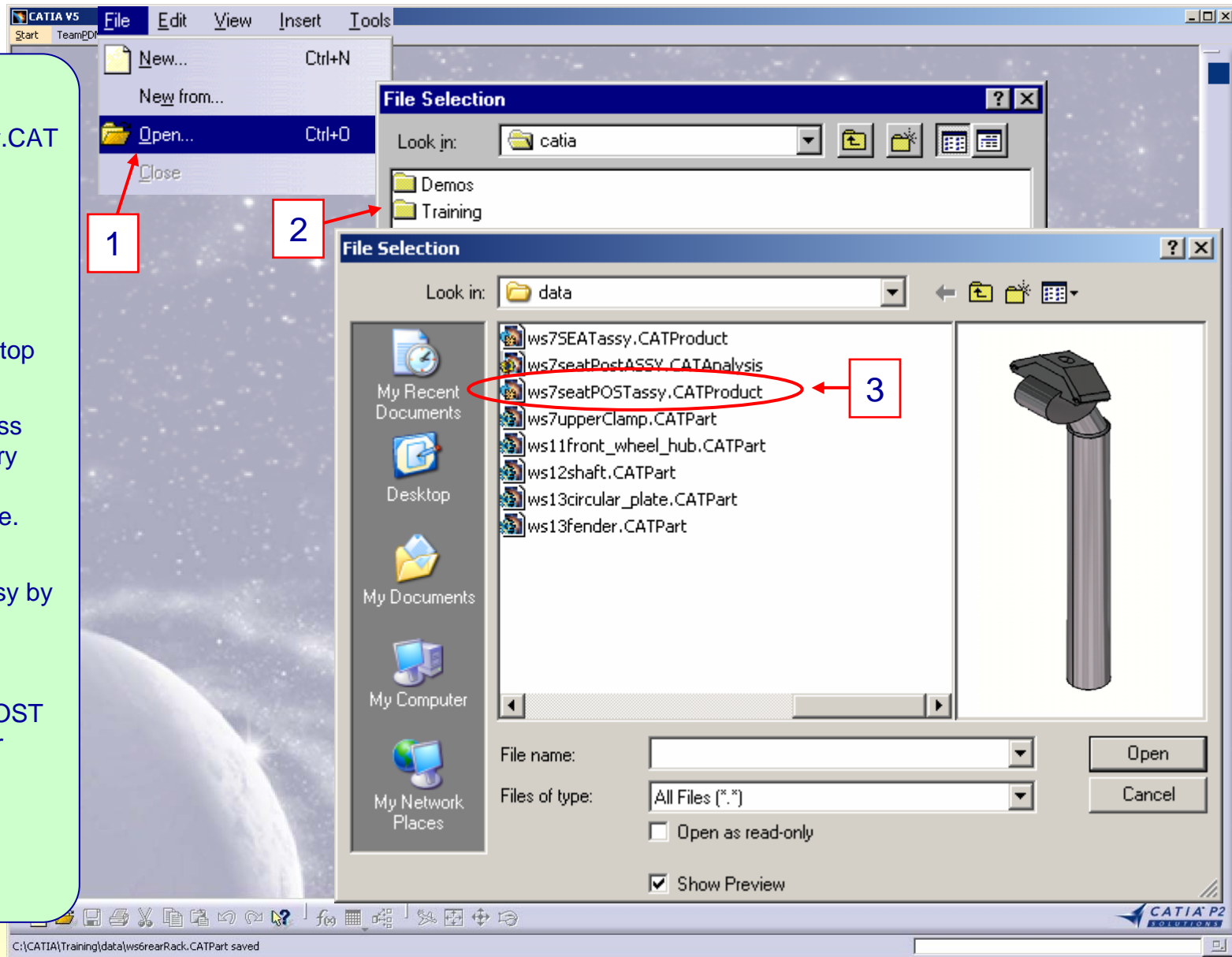
Step 1. Open the existing CATIA product (assembly)

Open the CATIA product ws7seatPostassy.CATProduct 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.

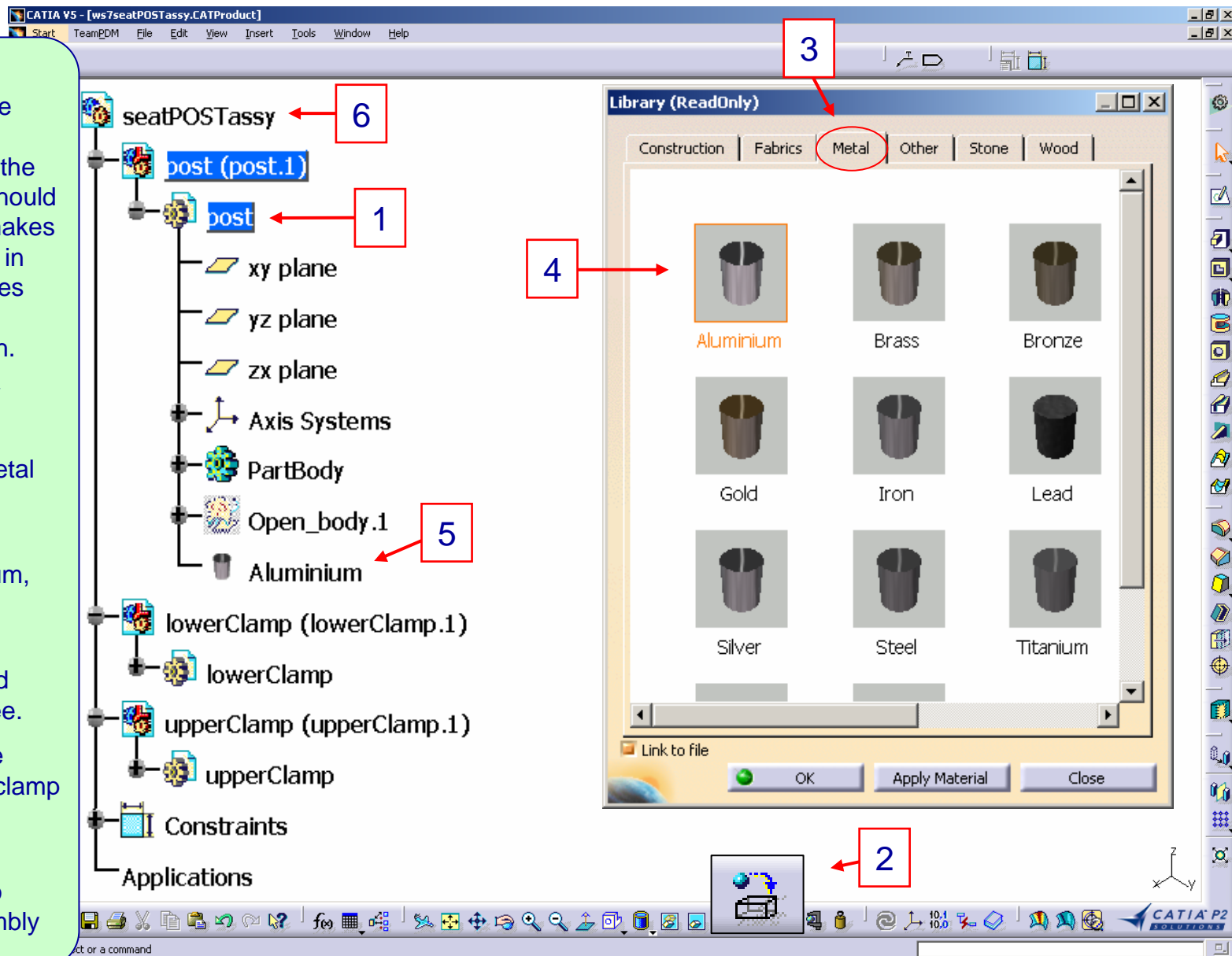
By default, the POST assy and all other CATProduct documents are opened in the Assembly Design workbench.



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.
 6. Double click seatPOSTassy to access the assembly workbench.
- Apply Steel to the lower and upper clamp parts.



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.

The screenshot shows the CATIA V5 interface with the following elements:

- Features Tree (Left):**
 - seatPOSTassy (highlighted in blue, labeled 2)
 - post (post.1)
 - post
 - lowerClamp (lowerClamp.1)
 - lowerClamp
 - upperClamp (upperClamp.1)
 - upperClamp
 - Constraints
 - Fix.1 (post.1) (highlighted in orange, labeled 5)
 - Coincidence.93 (post.1, lowerClamp.1)
 - Surface contact.94 (post.1, lowerClamp.1)
 - Offset.98 (post.1, lowerClamp.1)
 - Coincidence.99 (lowerClamp.1, upperClamp.1)
 - Surface contact.100 (lowerClamp.1, upperClamp.1)
 - Surface contact.101 (lowerClamp.1, upperClamp.1)
 - Applications
- Constraints Dialog Box (Center):**
 - Buttons for: Coincidence Constraint, Contact Constraint, Offset Constraint, Angle Constraint, Fix Component, Fix Together.
- 3D Model (Right):**
 - Shows a mechanical part with assembly constraints applied.
 - Label 4 points to the model.
- Annotations:**
 - Label 1 points to the Assembly Design Workbench icon in the top right.
 - Label 3 points to the 'seatPOSTassy' icon in the features tree.
 - Label 4 points to the 3D model.
 - Label 5 points to the 'Fix.1 (post.1)' constraint in the tree.

Step 3. Examine and verify assembly constraints

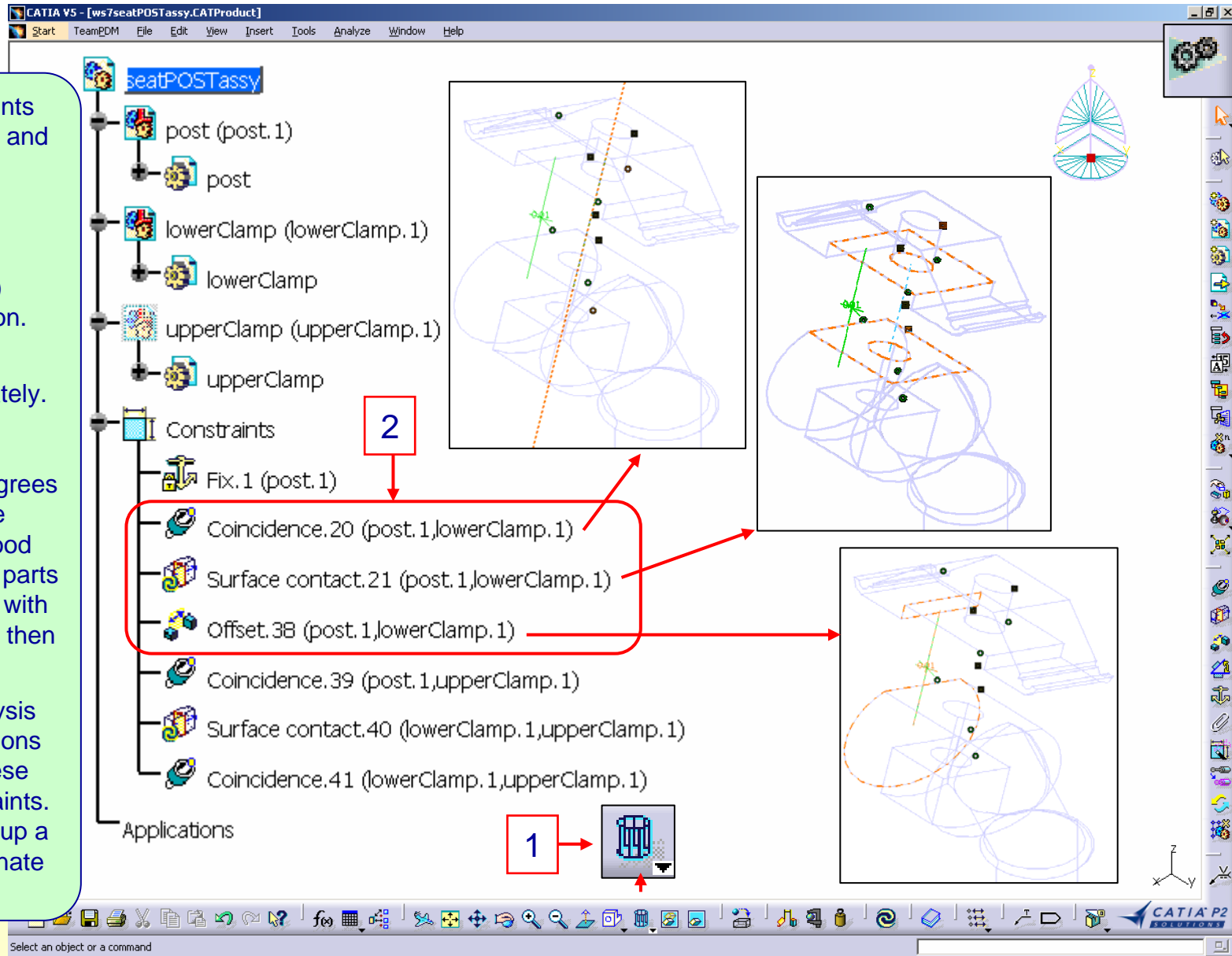
Examine constraints between the post and lower clamp.

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

CATIA V5 - [ws7seatPOSTassy.CATProduct]

Start TeamPDM File Edit View Insert Tools Analyze Window Help

seatPOSTassy

- post (post.1)
 - post
- lowerClamp (lowerClamp.1)
 - lowerClamp
- upperClamp (upperClamp.1)
 - upperClamp
- Constraints
 - Fix.1 (post.1)
 - Coincidence.93 (post.1,lowerClamp.1)
 - Surface contact.94 (post.1,lowerClamp.1)
 - Offset.98 (post.1,lowerClamp.1)
 - Coincidence.99 (lowerClamp.1,upperClamp.1)**
 - Surface contact.100 (lowerClamp.1,upperClamp.1)
 - Surface contact.101 (lowerClamp.1,upperClamp.1)
- Applications

1

Examine upper clamp constraints.

Steps:

1. Highlight each constraint separately.

Select an object or a command

CATIA V5 P2 SOLUTIONS

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.

Analysis Manager

- Links Manager
 - Results -> C:\ELFINI\seatPost\Analysis1_1.CATAnalysisResults
 - Computations -> C:\ELFINI\seatPost\Analysis1_1.CATAnalysisComputations
 - Link.1 -> C:\CATIA\Training\data\ws7seatPOSTassy.CATProduct
- Finite Element Model
 - Nodes and Elements
 - Properties.1
 - Static Case
 - Restraints.1
 - Loads.1
 - Static Case Solution.1
 - Sensors.1

Start TeamPDM File Edit View

- Infrastructure
- Mechanical Design
- Shape
- Analysis & Simulation
 - Advanced Meshing Tools
 - Generative Structural Analysis**
 - Analysis Connections
- AEC Plant
- NC Manufacturing

New Analysis Case

- Static Analysis
- Frequency Analysis
- Free Frequency Analysis

Keep as default starting analysis case

OK Cancel Help

Step 4. Create an assembly static analysis document

The screenshot shows the CATIA V5 interface with the Analysis Manager tree on the left. The tree includes Analysis Manager, Links Manager, Finite Element Model, Nodes and Elements, Properties.1, Static Case, Restraints.1, Loads.1, Static Case Solution.1, and Sensors.1. The Links Manager shows Results, Computations, and Link.1. The Current Storage Location dialog is open, showing Results Data and Computation Data fields with their respective paths and Modify buttons. The Save Management dialog is also open, showing a table of files and their paths. Red boxes and arrows indicate the steps: 1 points to the Save Management icon in the toolbar, 2 points to the Results Data field, 3 points to the Computation Data field, and 4 points to the Save Management dialog.

Current Storage Location

Field	Path	Action
Results Data	C:\ELFINI\seatPost\seatPost.CATAnalysisResults	Modify
Computation Data	C:\ELFINI\seatPost\seatPost.CATAnalysisComputations	Modify

Save Management

State	Name	Path	Action
Opened Read Only	Catalog.CATMaterial	C:\Program Files\Dassault Systemes\B07\intel_a\startup\materials	
Opened	ws7seatPOSTassy.CATProduct	C:\CATIA\Training\data	
Opened	ws7upperClamp.CATPart	C:\CATIA\Training\data	
Opened	ws7lowerClamp.CATPart	C:\CATIA\Training\data	
Opened	ws7post.CATPart	C:\CATIA\Training\data	
Opened	ws7seatPostASSY.CATAnalysis	C:\CATIA\Training\data	
Opened	Analysis1_1.CATAnalysisResults	C:\ELFINI\seatPost	
Opened	Analysis1_1.CATAnalysisComputations	C:\ELFINI\seatPost	

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.

The screenshot displays the CATIA V5 interface for finite element analysis. The Analysis Manager tree on the left shows the following structure:

- Analysis Manager
 - Links Manager
 - Finite Element Model
 - Nodes and Elements
 - OCTREE Tetrahedron Mesh.1 : post.1
 - OCTREE Tetrahedron Mesh.2 : lowerClamp.1
 - OCTREE Tetrahedron Mesh.3 : upperClamp.1
 - Properties.1
 - Static Case
 - Restraints.1
 - Loads.1
 - Static Case Solution.1
 - Sensors.1

Two 'OCTREE Tetrah...' dialog boxes are open on the right. The top dialog is for 'post.1' with the following settings:

- Global | Local
- Size: 0.125in
- Sag: 0.013in
- Element type: Linear (selected), Parabolic
- Buttons: OK, Cancel

The bottom dialog is for 'upperClamp.1' with the following settings:

- Global | Local
- Size: 0.063in
- Sag: 0.006in
- Element type: Linear (selected), Parabolic
- Buttons: OK, Cancel

Red arrows and boxes indicate the steps: 1 points to the mesh entries in the tree, 2 points to the 'post.1' dialog, and 3 points to the 'upperClamp.1' dialog.

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.

The screenshot displays the CATIA V5 software interface for a simulation analysis. The main window shows the assembly tree for 'seatPOSTassy'. The 'Constraints' folder is expanded, showing various constraints including 'Surface contact.94 (post.1,lowerClamp.1)', which is highlighted with a red box and labeled '3'. A 'Connection' dialog box is open, showing the 'Fastened Connection' icon selected, with a red box and label '2' pointing to it. Below the dialog, a 'Fastened Connection' property dialog is shown with 'Name: Fastened Connection.1' and 'Supports: 1 Constraint'. In the 'Applications' folder, 'Fastened Connection Mesh.1' is highlighted with a red box and label '4'. Two 3D mesh visualizations are shown at the bottom right, illustrating the mesh connection between parts. The bottom status bar indicates 'Select an object or a command'.

Step 7. Apply a clamp restraint

seatPOSTassy

- post (post.1)**
- post
- lowerClamp (lowerClamp.1)**
- lowerClamp
- upperClamp (upperClamp.1)**
- upperClamp

Constraints

- Fix.1 (post.1)**
- Coincidence.93 (post.1,lowerClamp.1)
- Surface contact.94 (post.1,lowerClamp.1)
- Offset.98 (post.1,lowerClamp.1)
- Coincidence.99 (lowerClamp.1,upperClamp.1)
- Surface contact.100 (lowerClamp.1,upperClamp.1)
- Surface contact.101 (lowerClamp.1,upperClamp.1)

Applications

Window

- New Window
- Tile Horizontally
- Tile Vertically
- Cascade
- 1 ws7seatPOS..Product**
- 2 ws7seatPos..Analysis

1

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.

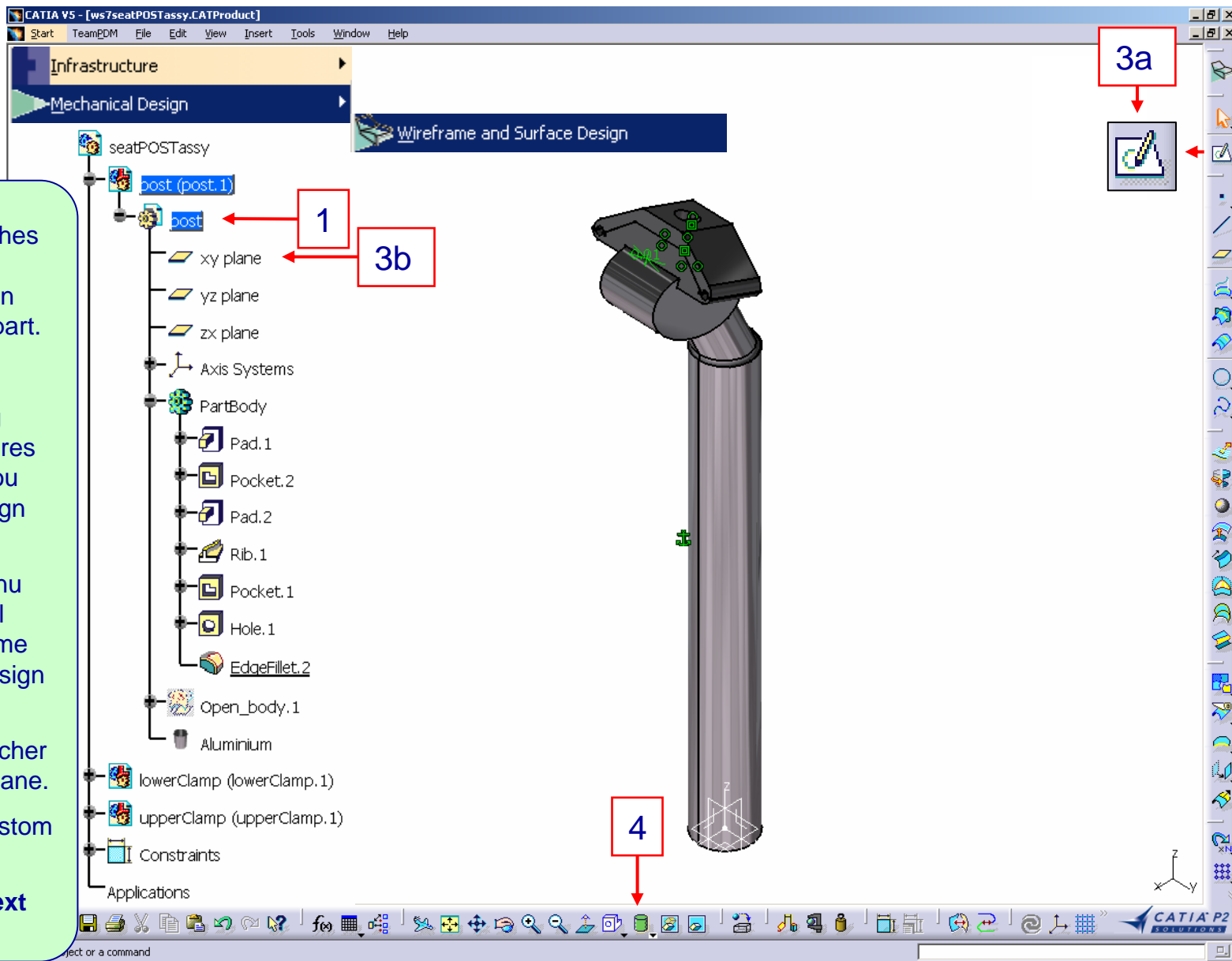
This changes your active document and launches you into the Assembly Design Workbench.

4 inches

Clamping requirement

Step 7. Apply a clamp restraint

2



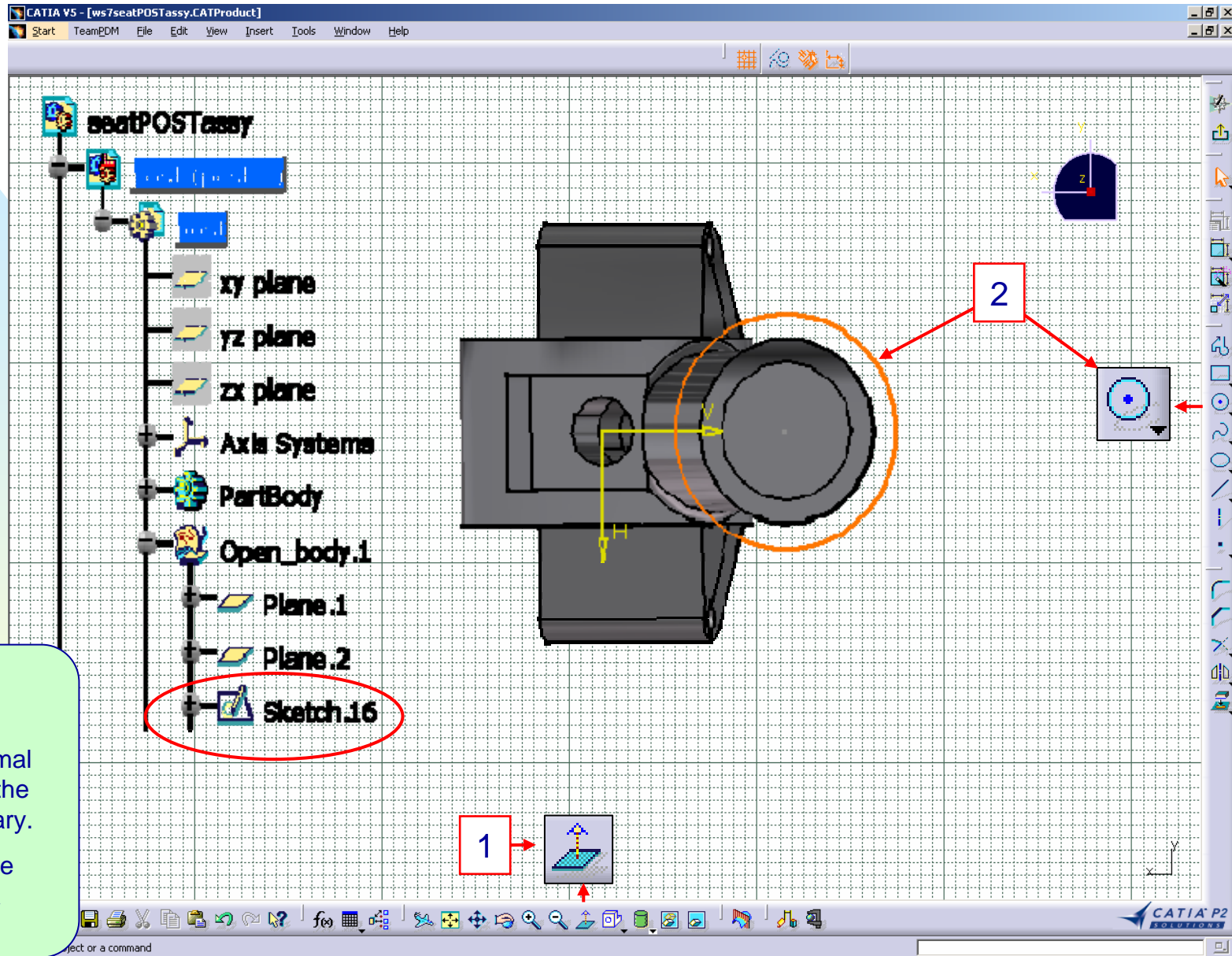
We will create a surface that matches your clamping requirements, then “sew” it onto the part.

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

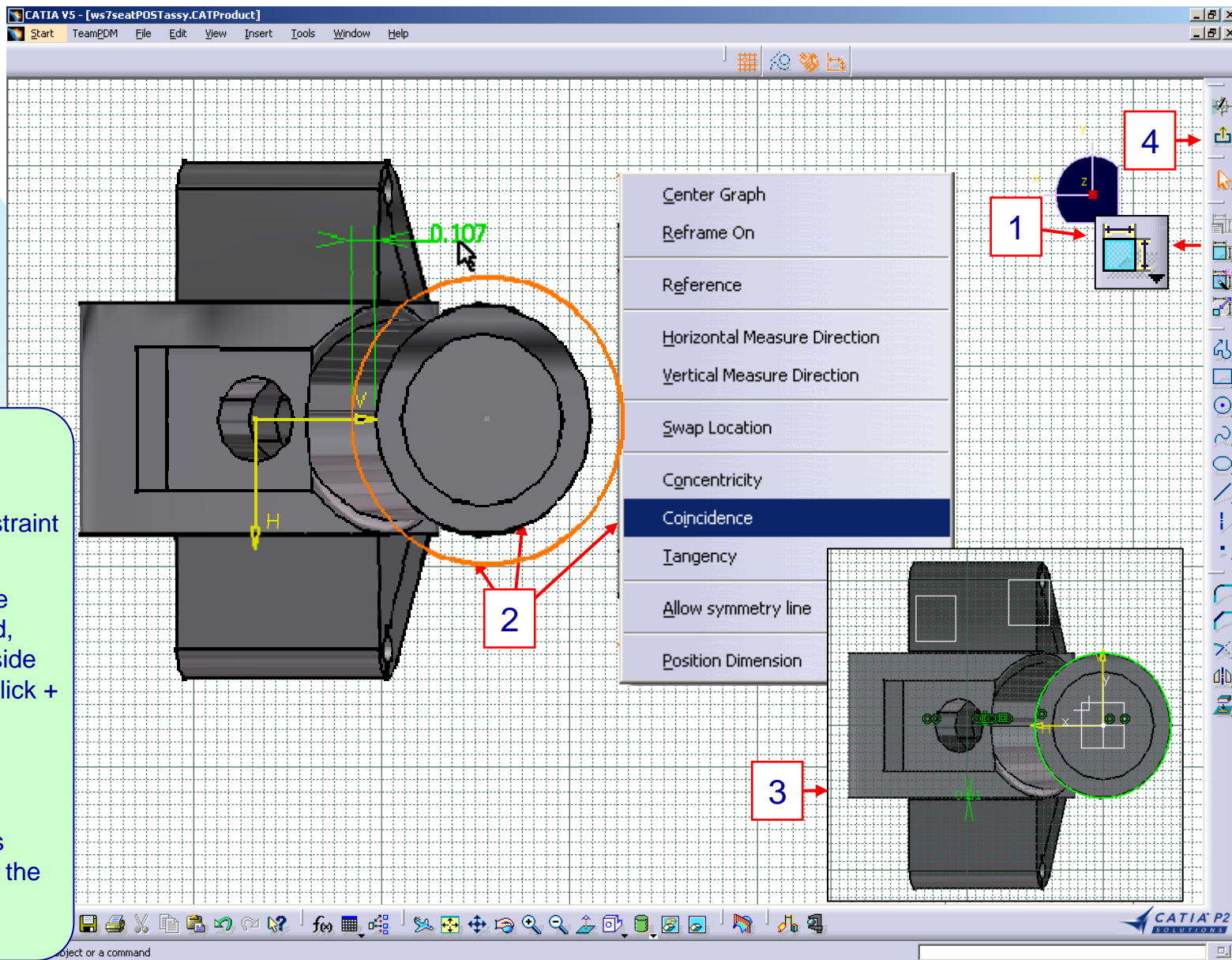


Sketch a shape.

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

seatPOSTassy

- post (post.1)
- post
 - xy plane
 - yz plane
 - zx plane
 - Axis Systems
 - PartBody
 - Open_body.1
 - Plane.1
 - Plane.2
 - Sketch.16
 - Extrude.1
 - Aluminium
 - lowerClamp (lowerClamp.1)
 - upperClamp (upperClamp.1)
 - Constraints
 - Applications

1

2a

2b

3

Extruded Surface Definition

Profile: Sketch.16

Direction: xy plane

Extrusion Limits

Limit 1: 4in

Limit 2: 0in

Reverse Direction

OK Apply Cancel

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

1

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.


The screenshot displays the CATIA V5 software interface. The top menu bar includes 'Start', 'TeamPDM', 'File', 'Edit', 'View', 'Insert', 'Tools', 'Window', and 'Help'. The 'Infrastructure' and 'Mechanical Design' workbenches are visible. The tree view on the left shows the assembly structure, with 'Extrude.1' highlighted. The main view shows a 3D model of a post with a clamp. Two inset diagrams illustrate the 'SewSurface' operation: 'incorrect' shows arrows pointing outwards, and 'Correct' shows arrows pointing inwards. A 'SewSurface Definition' dialog box is open, showing 'Object to sew: Extrude.1'. A 'Container4' dialog box is also open, with a red circle around a specific icon. Red callout boxes labeled '1', '2a', '2b', and '3' point to various elements in the interface.

Step 7. Apply a clamp restraint

The screenshot shows the CATIA V5 interface with the following elements:

- Features Tree (Left):** Analysis Manager, Links Manager, Finite Element Model, Nodes and Elements, Properties.1, Static Case, Restraints.1 (containing Clamp.1), Loads.1, Static Case Solution.1, Sensors.1.
- 3D Model (Center):** A shaft with a clamp. A red box labeled '1' points to the clamp icon in the toolbar. A red box labeled '2' points to the shaft's surface. A dimension line indicates a 4-inch length for the clamp. A red box labeled '3' points to the 'Clamp.1' object in the features tree.
- Clamp Dialog Box (Right):** Name: Clamp.1, Supports: 1 Face, OK, Cancel.

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

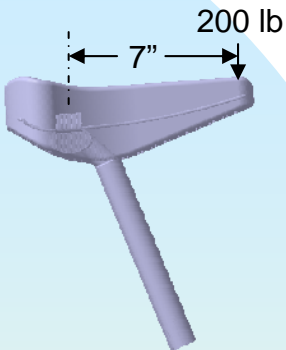


Image to show
The -y moment

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.

CATIA V5 - [ws7seatPostASSY.CATAnalysis]

Analysis Manager

- Links Manager
- Finite Element Model
- Nodes and Elements
- Properties.1
- Static Case
 - Restraints.1
 - Clamp.1
 - Loads.1
 - Moment.1
 - Static Case Solution.1
 - Sensors.1

Moment

Name: Moment.1

Supports: 4 Faces

Axis System: [Default]

Type: Global

Display locally

Moment Vector

Norm: 1400lbf·in

X: 0lbf·in

Y: -1400lbf·in

Z: 0lbf·in

OK Cancel

Force

1

2

2

4

3

CATIA V5 P2 SOLUTIONS

Step 9. Compute the analysis

Static solution Parameters

Method

auto

gauss

gradient

gauss R6

Gradient parameters

maximum iteration number: 0

accuracy: 1e-008

Compute

All

Preview

Computation Resources Estimation

5e+001 s of CPU

9.93e+003 kilo-bytes of memory

4.71e+004 kilo-bytes of disk

Intel MKL(c) Library found: Intel(R) MKL V5.1.0

Do you want to continue the computation?

Error

Singularity detected (relative pivot too small)

Static Case Solution.1

Restraints.1

Clamp.1

Loads.1

Moment.1

Sensors.1

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

CATIA V5 - [ws7seatPostASSY.CATAnalysis]

Start TeamPDM File Edit View Insert Tools Window Help

Analysis Manager

- Links Manager
- Finite Element Model
 - Nodes and Elements
 - Properties.1
 - Static Case
 - Restraints.1
 - Loads.1
 - Static Case Solution.1
 - Deformed Mesh
 - Sensors.1

2

1

3

Image Activate/DeActivate

CATIA V5 SOLUTIONS

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.

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.

The screenshot displays the CATIA V5 interface for defining a fastener connection. The tree view on the left shows the assembly structure, including 'seatPOSTassy', 'post (post.1)', 'lowerClamp (lowerClamp.1)', 'upperClamp (upperClamp.1)', and 'Constraints'. The 'Fastened Connection' dialog box is open, showing 'Name: Fastened Connection.2' and 'Supports: 1 Constraint'. The 3D model views at the bottom show the meshed seat assembly with red boxes highlighting the fastener locations.

Step 9. Compute the analysis

Show the Deformed Mesh.

Steps:

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.

The screenshot displays the CATIA V5 software interface for a finite element analysis. The left-hand side features a tree view with the following structure:

- Analysis Manager
 - Links Manager
 - Finite Element Model
 - Nodes and Elements
 - Properties.1
 - Static Case
 - Restrains.1
 - Clamp.1
 - Loads.1
 - Moment.1
 - Static Case Solution.1
 - Deformed Mesh
 - Sensors.1
 - Energy

The central 3D model shows a blue mesh of a hand with yellow curved arrows indicating applied loads. A 'Compute' dialog box is open on the right, with 'All' selected in the dropdown menu and 'OK' button highlighted. A 'Computation Resources Estimation' dialog box is also open, showing the following data:

9e+001 s of CPU
1.5e+004 kilo-bytes of memory
6.07e+004 kilo-bytes of disk
Intel MKL(c) Library found: Intel(R) MKL V5.1.0

The dialog asks: "Do you want to continue the computation?" with 'Yes' and 'No' buttons. The bottom toolbar contains various icons, with the 'Compute' icon (a grid) highlighted by a red box labeled '1'. The 'Deformed Mesh' image in the tree view is highlighted by a red box labeled '5'.

Step 10. Visualize the analysis results

Von Mises Stress (nodal value)

psi

7.28e+004
6.55e+004
5.83e+004
5.1e+004
4.37e+004
3.64e+004
2.91e+004
2.18e+004
1.46e+004
7.28e+003
2.77e-008

On Boundary

Analysis Manager

Links Manager

Finite Element Model

Nodes and Elements

Properties.1

Static Case

Restraints.1

Loads.1

Static Case Solution.1

Deformed Mesh

Von Mises Stress (nodal value)

Extrema

Global Maximum.1

Sensors.1

Extrema Creation

Global

Minimum extrema at most 0

Maximum extrema at most 2

Local

Minimum extrema at most 0

Maximum extrema at most 2

OK Cancel

1

2

3

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.

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.

CATIA V5 - [ws7seatPostASSY.CATAnalysis]

Start TeamPDM File Edit View Insert Tools Window Help

Analysis Manager

- Links Manager
- Finite Element Model
 - Nodes and Elements
 - Properties.1
 - Static Case
 - Restraints.1
 - Loads.1
 - Static Case Solution.1
 - Deformed Mesh
 - Von Mises Stress (nodal value)
 - Extrema
 - Global Maximum.1
 - Sensors.1

Cut Plane Analysis ? X

- View section only
- Show cutting plane

Close

3

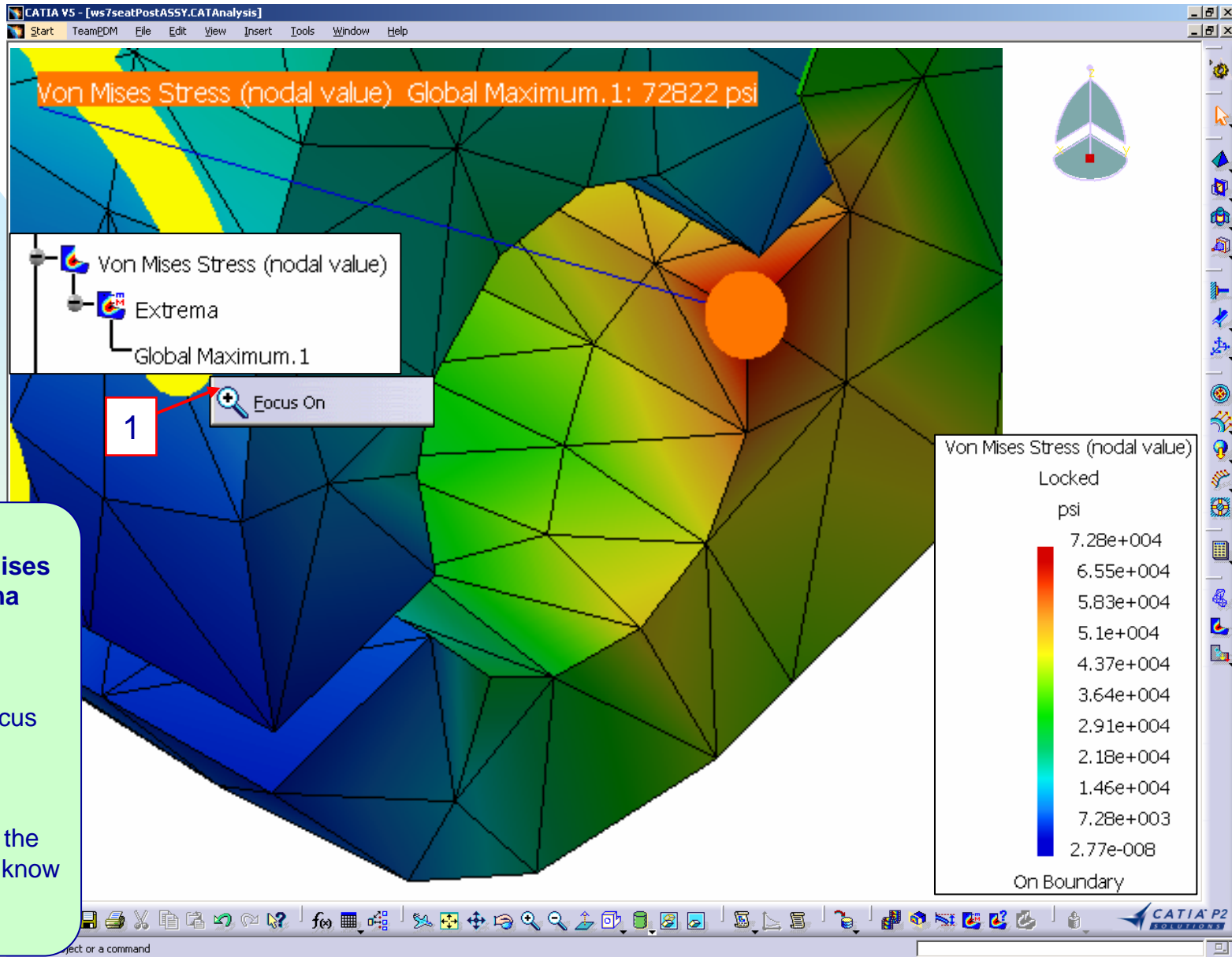
2

1

Project or a command

CATIA P2 SOLUTIONS

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.

Analysis Manager

- Links Manager
- Finite Element Model
 - Nodes and Elements
 - Properties.1
 - Static Case
 - Restraints.1
 - Loads.1
 - Static Case Solution.1
 - Deformed Mesh
 - Von Mises Stress
 - Estimated local error
 - Extrema
 - Global Maximum.1
 - Sensors.1

Extrema Creation

Global

Minimum extrema at most: 0

Maximum extrema at most: 2

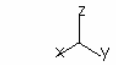
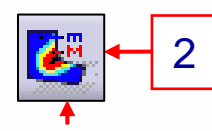
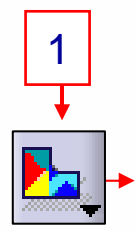
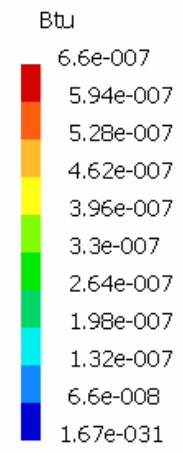
Local

Minimum extrema at most: 0

Maximum extrema at most: 2

OK Cancel

Estimated local error



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.

The screenshot displays the CATIA V5 interface for a finite element analysis. The features tree on the left shows the analysis setup, including a 'Static Case' with 'Restrains.1', 'Loads.1', and 'Static Case Solution.1'. The 3D model shows a blue meshed part with yellow 'E' markers. The 'Informations' dialog box on the right provides details for the 'Estimated local error' feature, including 'Extrema Values' (Min: 1.6746e-031 Btu, Max: 6.60403e-007 Btu) and 'Precision Location : Global'. The 'Global Estimated Error Rate : 29.5527 %' is highlighted with a red circle. A tooltip over the model shows 'Estimated local error Global Maximum.1: 6.60403e-007 Btu'. Three numbered red boxes (1, 2, 3) indicate the steps to find the error rate: 1 points to the 'Informations' icon, 2 points to 'Estimated local error' in the tree, and 3 points to the error rate in the dialog.

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.

The screenshot displays the CATIA P2 Analysis Manager interface. The left-hand tree view shows the analysis structure, including 'Static Case Solution.1', 'Estimated local error', and 'Adaptivity Process'. A red box labeled '1a' points to the 'Estimated local error' icon in the tree. The 'Adaptivity Box' dialog box is open in the center, showing 'Name: Adaptivity Box.1', 'Objective Error (%): 5', 'Solution: Static Case Solution.1', and 'Local Error (%): 44.576'. A red oval highlights the 'Local Error (%)' field, with a red box labeled '3' pointing to it. To the right, a color scale legend for 'Estimated local error' is shown, ranging from 1.67e-031 to 6.6e-007. A red box labeled '1b' points to a magnifying glass icon in the toolbar. Below the legend, three views of the mesh are shown: a top view, a side view, and a bottom view. Red boxes labeled '2' point to specific areas on these views where the adaptivity box was defined. The bottom of the screen shows the CATIA P2 SOLUTIONS toolbar.

Step 10. Visualize the analysis results

CATIA V5 - [ws6rearRack.CATAnalysis]

Analysis Manager

- Links Manager
- Finite Element Model
 - Nodes and Elements
 - Properties.1
 - Static Case
 - Restraints.1
 - Loads.1
 - Static Case Solution.1
 - Deformed Mesh
 - Von Mises Stress (nodal value)
 - Estimated local error
 - Translational displacement magnitude
 - Sensors.1
 - Adaptivity Process

Image Edition

Deformed

Display on Deformed Mesh

Frequencies Visu Criteria Filters Selections

AVERAGE-ISO

SYMBOL

TEXT

3

Edition

Iso/Fringe Symbol Axis System

OK Cancel Help

Translational displacement magnitude

in

0.0766

0.069

0.0613

0.0537

0.046

0.0383

0.0307

0.023

0.0153

0.00766

0

On Boundary

1

2

3

CATIA V5 SOLUTIONS

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 activates 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**.

The screenshot shows the CATIA V5 interface with the following elements:

- Features Tree:** Shows a hierarchy including Analysis Manager, Links Manager, Finite Element Model, Nodes and Elements, Properties.1, Static Case, Restraints.1, Loads.1, Static Case Solution.1, Deformed Mesh, Von Mises Stress (nodal value), Extrema, Estimate, and Translational displacement magnitude. A red box highlights the 'Von Mises Stress (nodal value)' and 'Extrema' nodes, with a red arrow pointing to 'Local Update'.
- 3D Model:** A mechanical part is shown with a color-coded stress distribution. A red box (5a) highlights the 'Global Maximum.1' node in the tree, with a red arrow pointing to a 'Focus On' button.
- Image Extremum Editor (5b):** A dialog box showing 'Von Mises Stress (nodal value) Global Maximum.1: 58012.9 psi' and 'OK'/'Cancel' buttons.
- Image Edition (1):** A dialog box with 'Deformed' checked and 'Display on Deformed Mesh' checked. The 'Selections' tab is active, showing a list of mesh objects. 'OCTREE Tetrahedron Mesh.1 : post.1' is selected. The 'Edition' section has 'Iso/Fringe' selected.

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.

The screenshot displays the CATIA V5 interface for post-analysis visualization. The features tree on the left shows the hierarchy: Analysis Manager, Links Manager, Finite Element Model, Nodes and Elements, Properties.1, Static Case, Restraints.1, Loads.1, Static Case Solution.1, Deformed Mesh, Von Mises Stress (nodal value), Extrema, Estimated local displacement magnitude, Translational displacement magnitude, Von Mises Stress (nodal value), Extrema, and Global Maximum.1. The 3D model shows the lower clamp with a Von Mises stress color map. The 'Image Editor' dialog is open, showing the 'Selections' tab with a list of mesh objects. The 'Image Extremum Editor' dialog is also open, showing the global maximum stress value of 72822 psi.

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.

The screenshot shows the CATIA V5 interface with the following elements:

- Features Tree (Left):** Shows the analysis hierarchy. Key items include 'Static Case Solution.1', 'Deformed Mesh', 'Von Mises Stress (nodal value)', 'Extrema', 'Estimated local', 'Translational displacement magnitude', 'Von Mises Stress (nodal value)', 'Extrema', and 'Global Maximum.1'. Red boxes and arrows highlight 'Local Update' (3), 'Focus On' (4a), and 'Global Maximum.1' (4b).
- 3D Model (Center):** A mesh of the upper clamp with a color-coded stress distribution. A red box labeled '1' points to the 'Von Mises Stress (nodal value)' feature.
- Image Extremum Editor (Top Center):** A dialog box titled 'Image Extremum Editor' with a 'Show Label' checkbox and a text field displaying 'Von Mises Stress (nodal value) Global Maximum.1: 41603.3 psi'. It has 'OK' and 'Cancel' buttons. A red box labeled '4b' points to this dialog.
- Deformed Dialog (Bottom Right):** A dialog box titled 'Deformed' with tabs for 'Visu', 'Criteria', 'Filters', and 'Selections'. The 'Selections' tab is active and circled in red. It lists various mesh objects, with 'OCTREE Tetrahedron Mesh.3 : upperClamp.1' selected. It also has 'OK', 'Cancel', and 'Help' buttons. A red box labeled '2' points to this dialog.

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.

The screenshot shows the CATIA V5 interface for a finite element analysis. The main window displays a 3D model of a cylindrical component with a mesh and stress distribution. The left sidebar shows the Analysis Manager tree with 'Von Mises Stress (nodal value)' selected. The 'Image Edition' dialog box is open, showing the 'Selections' tab with 'Clamp.1' selected. Red arrows and boxes labeled '1', '2a', '2b', and '3' indicate the steps described in the text. An orange bar is visible at the top of the model area.

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.

Sensor Creation

Functions

- misesmax
- dispmx
- reaction
- globalerror
- dispmxongroup

OK Cancel

Reaction Sensor

Available Entities

- Clamp.1
- Fastened Connection.1
- Fastened Connection.2

Reference Axis

Type Global (global origin)

OK Cancel

Reaction Tensor

Name Clamp.1

Axis Global (local origin)

Force Moment Origin

Parameter	Value
Mx	-3.728e-007lbf _x in
My	1400lbf _x in
Mz	9.736e-008lbf _x in

Norm 1400lbf_xin

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.



Reaction Tensor

Parameter	Value
Fx	-276.948lbf
Fy	1.237lbf
Fz	-23.677lbf

Norm: 277.961lbf

Step 11. Frequency (Modal) analysis for the assembly

Frequency Case

Restrains: New Reference **Restrains.1**

Masses: New Reference

Static Case Solution:

Hide Existing Analysis Cases

OK Cancel

Analysis Manager Tree:

- Analysis Manager
- Links Manager
- Finite Element
- Nodes and Elements
- Properties.1
- Static Case
 - Restrains.1
 - Loads.1
 - Static Case Solution.1
 - Sensors.1
- Frequency Case**
 - Restrains.1
 - Masses.1
 - Frequency Case Solution.1
 - Sensors.2
- Adaptivity Process

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.

Step 11. Frequency (Modal) analysis for the assembly

CATIA V5 - [ws7seatPostASSY.CATAnalysis]

Analysis Manager

- Links Manager
- Finite Element Model
 - Nodes and Elements
 - Properties.1
 - Static Case
 - Frequency Case
 - Restraints.1
 - Clamp.1
 - Masses.1
 - Distributed Mass.1
 - Frequency Case Solution.1
 - Sensors.2
 - Adaptivity Process

Distributed Mass dialog box:

- Name: Distributed Mass.1
- Supports: 4 Faces
- Mass: 200lb
- Buttons: OK, Cancel

Steps:

1. Select the mass icon.
2. Select four inner faces as the supports.
3. Key in 200 lb as the mass, select OK.

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.

Select or a command

Step 11. Frequency (Modal) analysis for the assembly

Analysis Manager

- Links Manager
- Finite Element Model
 - Nodes and Elements
 - Properties.1
 - Static Case
 - Frequency Case
 - Restraints.1
 - Clamp.1
 - Masses.1
 - Distributed Mass.1
 - Frequency Case Solution.1
 - Sensors.2
- Adaptivity Process

Frequency solution Param...

Number of modes: 5

Method:
 gauss
 lanczos

Dynamic parameters:
maximum iteration number: 50
accuracy: 0.001

OK Cancel

Compute

All

Preview

OK Cancel

1: Frequency Case Solution.1

2: Number of modes, lanczos

3: Compute icon

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.

Number of modes	Frequency (Hz)
1	17.4468
2	18.1187
3	47.781
4	104.561
5	117.411

Step 12. Generate a report

Conclusions:

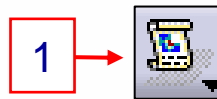
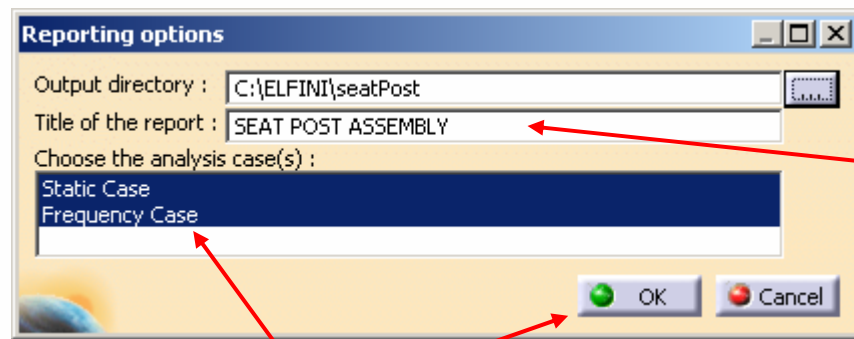
Maximum stress exceeds material yield. Select new material with yield values that exceed the analyzed Von Mises extrema using the parabolic element results.

	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

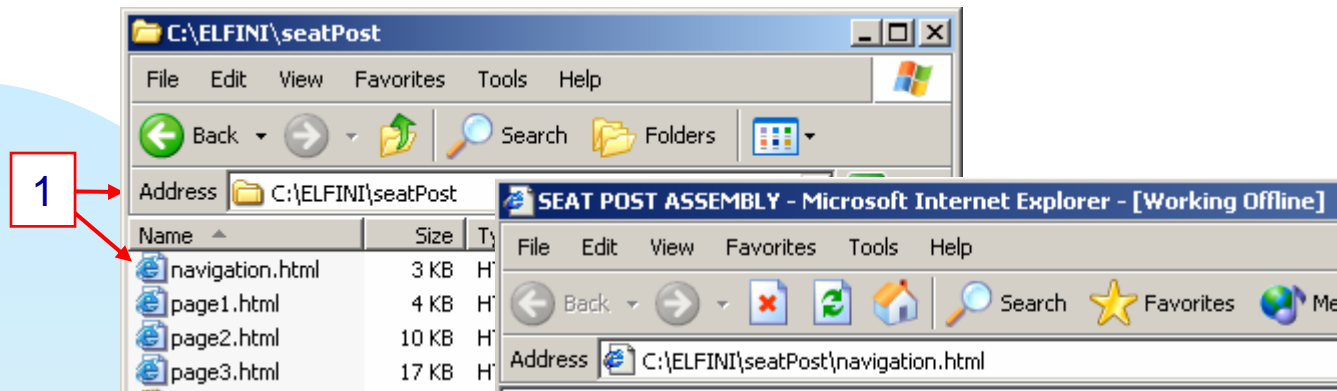
After activating each image at least once, generate a report.

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



2

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.

SEAT POST ASSEMBLY

MESH:

ELEMENT TYPE:

ELEMENT QUALITY:

Properties.1

Static Case

Boundary Conditions

STRUCTURE Computation

RESTRAINT Computation

LOAD Computation

STIFFNESS Computation

Frequency Case

Boundary Conditions

STRUCTURE Computation

RESTRAINT Computation

STRUCTURAL_MASS Computation

MASS Computation

Name: MassSet.4

STIFFNESS Computation

SINGULARITY Computation

CONSTRAINT Computation

NUMBERING Computation

FACTORIZED Computation

FREQUENCY Computation

Deformed Mesh - Mode number : 1 - Mode value : 17.4496Hz

Deformed Mesh - Mode number : 2 - Mode value : 18.0996Hz

3



Step 12. Generate a report

SEAT POST ASSEMBLY - Microsoft Internet Explorer - [Working Offline]

File Edit View Favorites Tools Help

Address <file:///C:/JLF/DN/seatPost/page1.html>

SEAT POST ASSEMBLY

MESH:

Entity	Size
Nodes	13192
Elements	53232

ELEMENT TYPE:

Connectivity	Statistics
SPIDER	506 (0.95%)
TE4	52726 (99.05%)

ELEMENT QUALITY:

Criterion	Good	Poor	Bad	W
Skewness	51875 (98.39%)	848 (1.61%)	3 (0.01%)	0.0

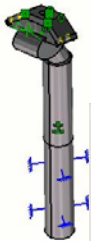
Static Case - Microsoft Internet Explorer - [Working Offline]

File Edit View Favorites Tools Help

Address <file:///C:/JLF/DN/seatPost/page2.html#0>

STATIC CASE

BOUNDARY CONDITIONS



STRUCTURE COM

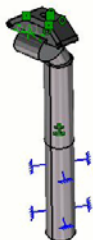
Frequency Case - Microsoft Internet Explorer - [Working Offline]

File Edit View Favorites Tools Help

Address <file:///C:/JLF/DN/seatPost/page3.html#0>

FREQUENCY CASE

BOUNDARY CONDITIONS



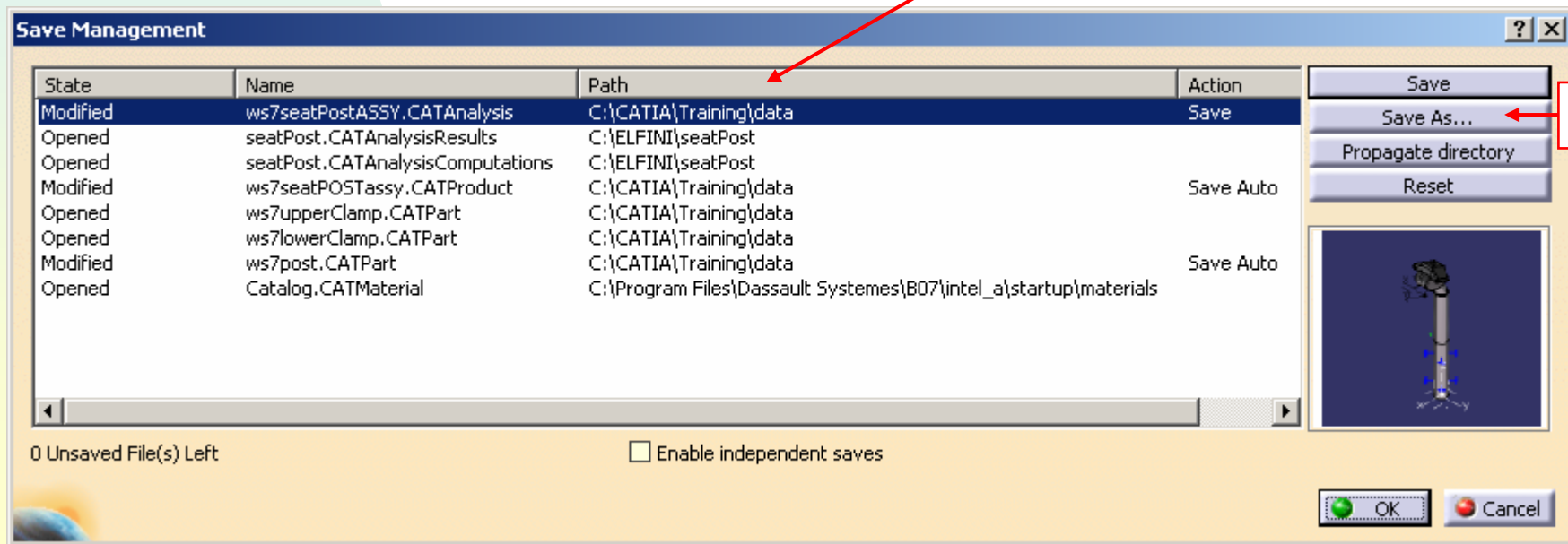
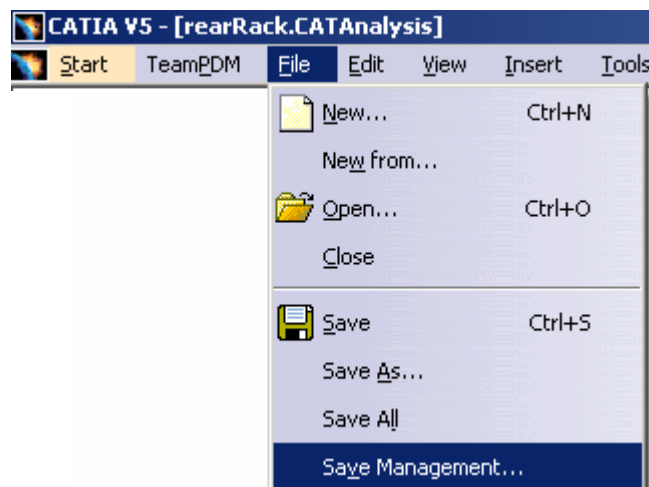
STRUCTURE COMPUTATION

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.



Step 14. Precise Results

Parabolic elements

Informations

Object Name : Estimated local error

Display On all Elements Over all the Model

Extrema Values
Min : 1.60111e-036 Btu
Max : 1.3416e-007 Btu

Precision Location : Global
Estimated Precision : 0.0244247 Btu
Strain Energy : 2.45008 Btu
Global Estimated Error Rate : 7.04255 %

Used materials
post.1
Material : Aluminium.1
Yield Strength : 13778.589psi
lowerClamp.1
Material : Steel.1.1
Yield Strength : 36259.445psi
upperClamp.1
Material : Steel.1.1
Yield Strength : 36259.445psi

Close

Adaptivity Box

Name: Adaptivity Box.1

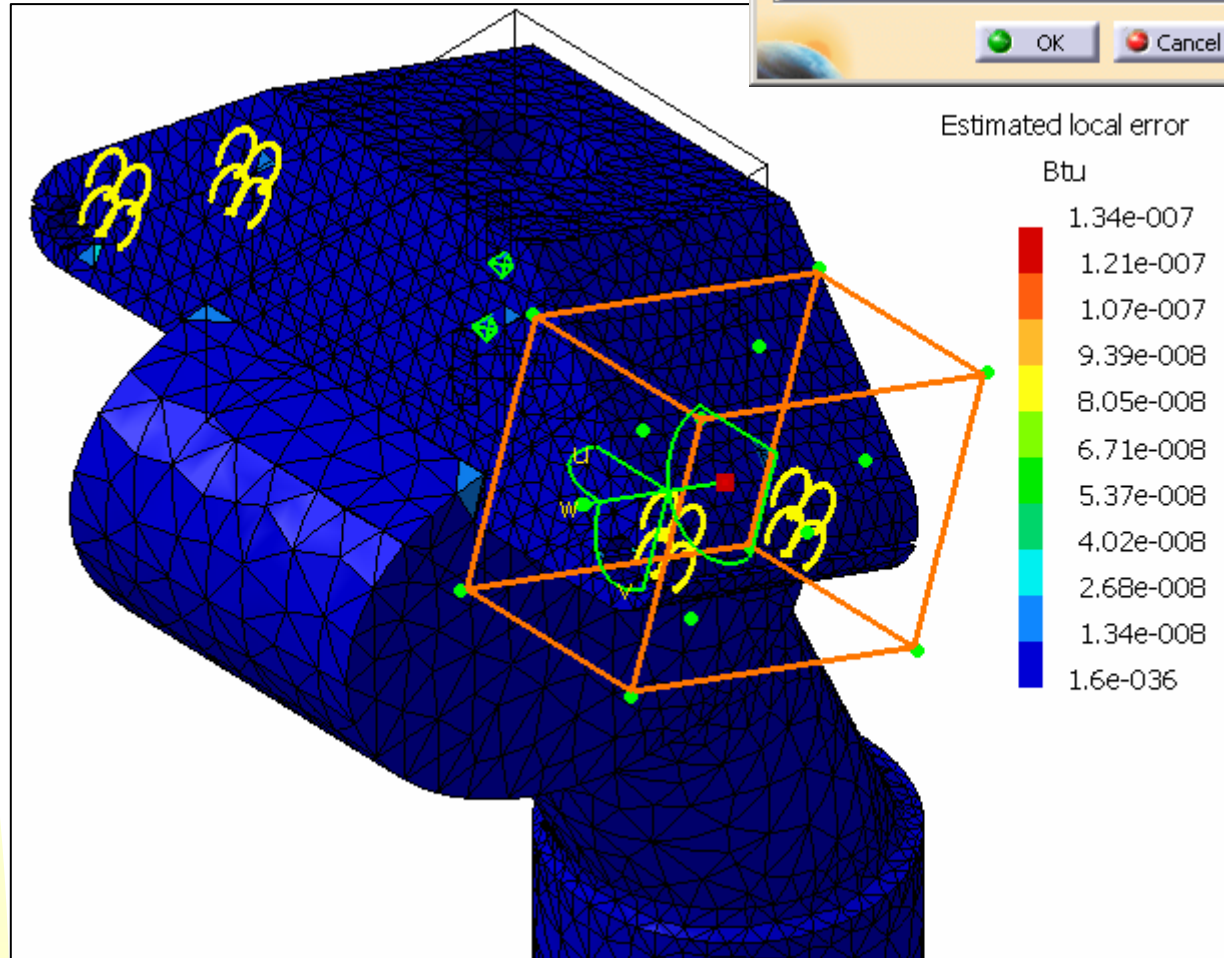
Objective Error (%): 5

Solution: Static Case Solution.1

Local Error (%) 9.7683

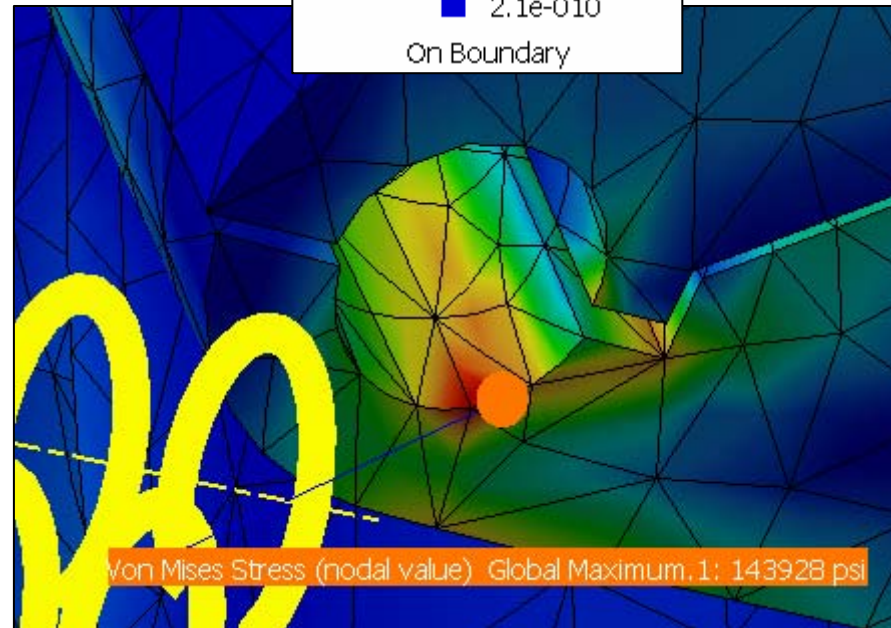
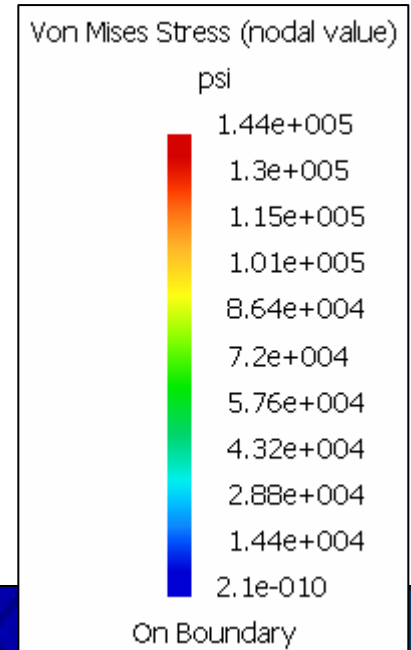
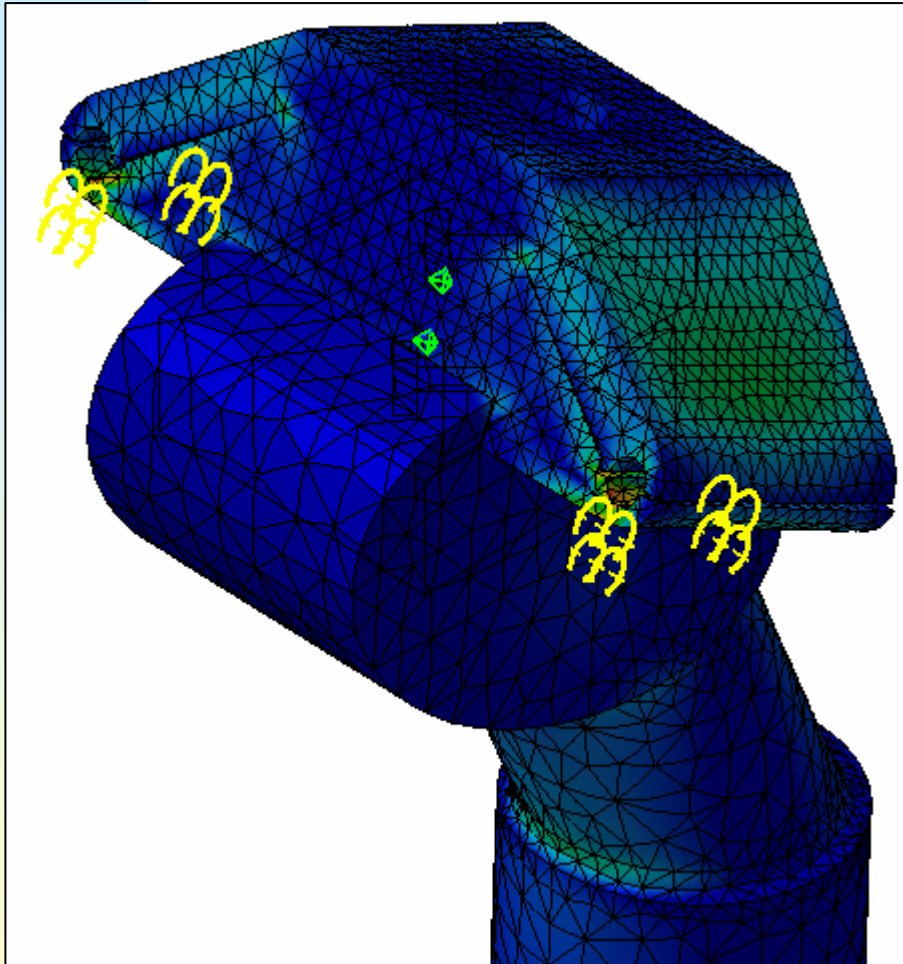
Select Extremum

OK Cancel



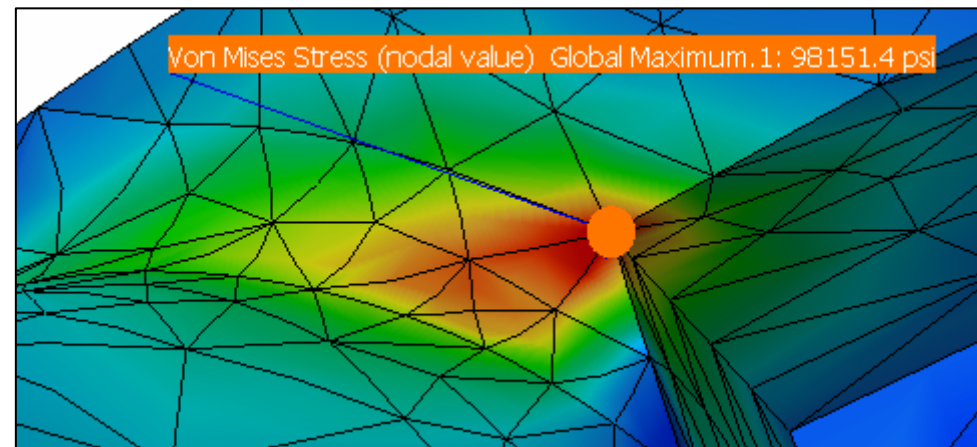
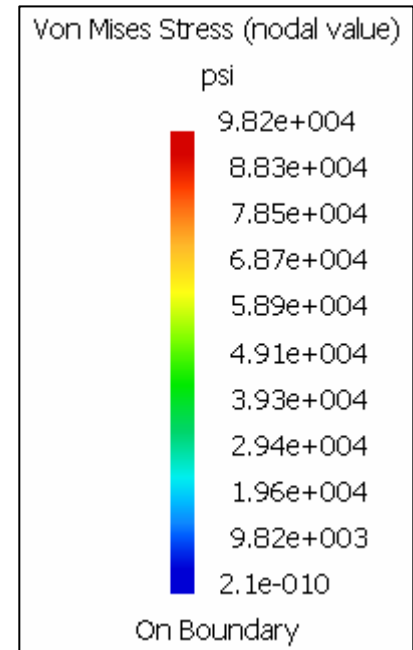
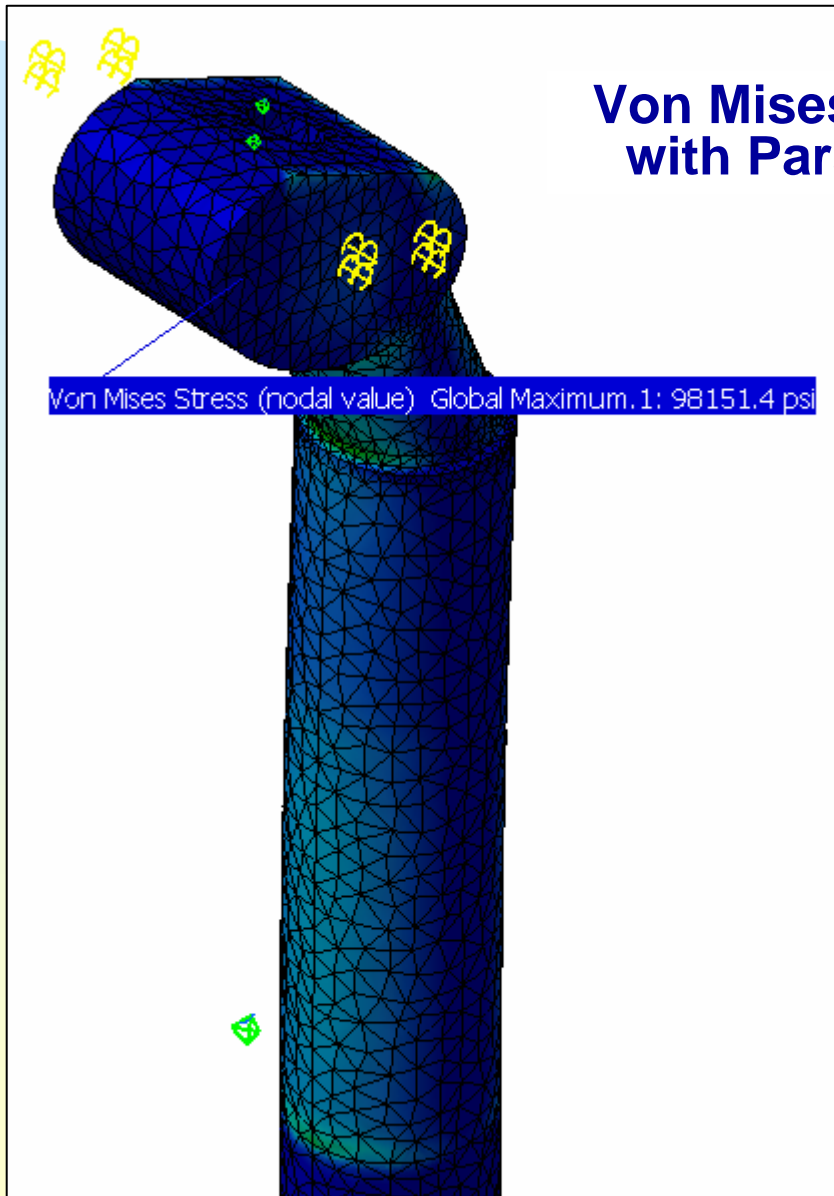
Step 14. Precise Results

Von Mises with Parabolic elements

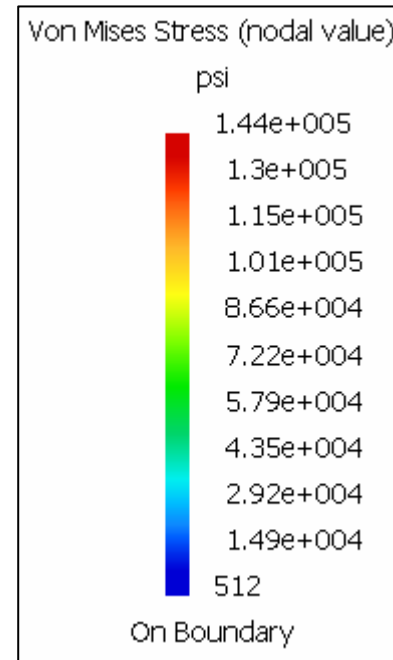
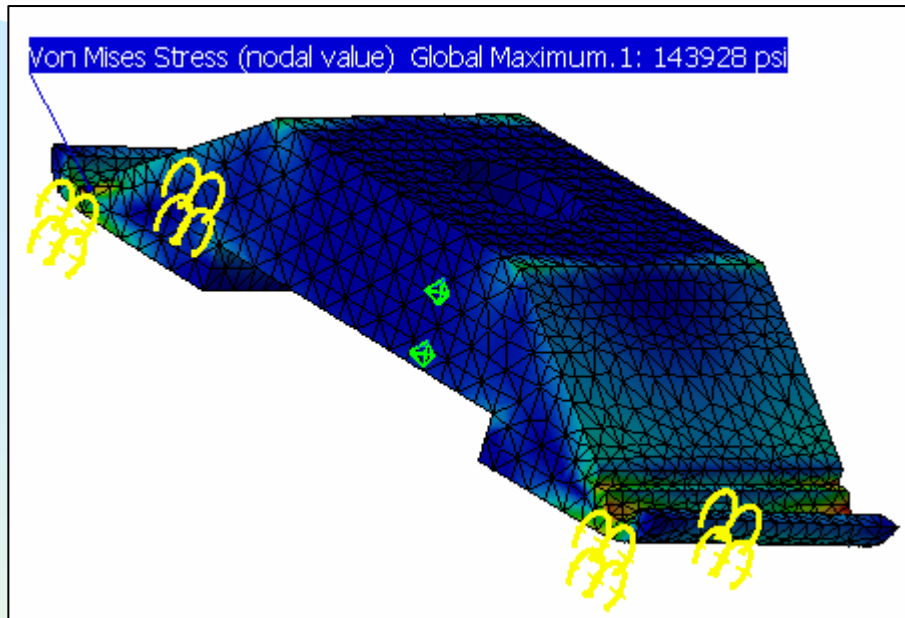


Step 14. Precise Results

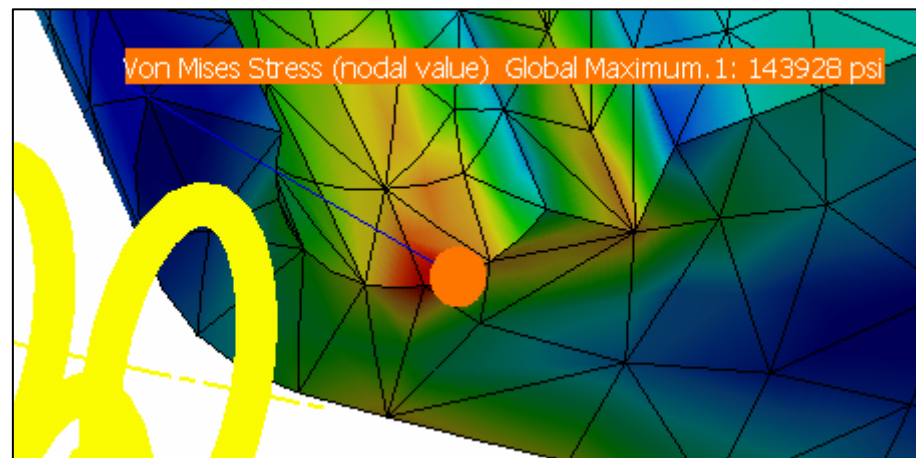
Von Mises "Post" Extrema with Parabolic elements



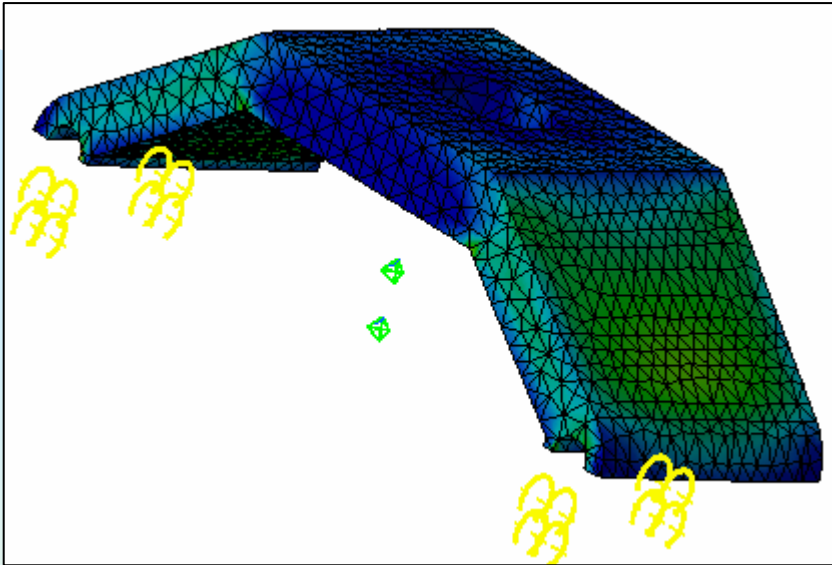
Step 14. Precise Results



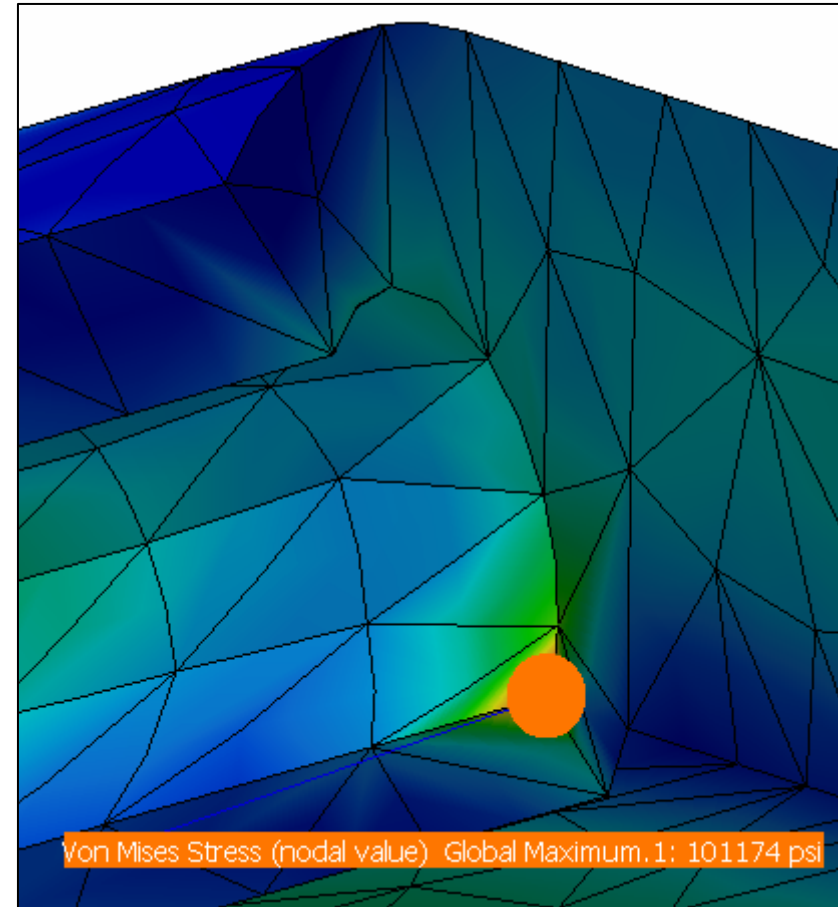
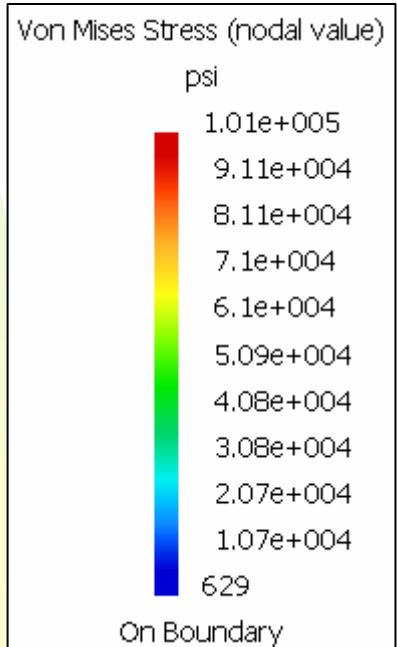
**Von Mises “Lower Clamp”
Extrema with Parabolic
elements**



Step 14. Precise Results



Von Mises "Upper Clamp" Extrema with Parabolic elements

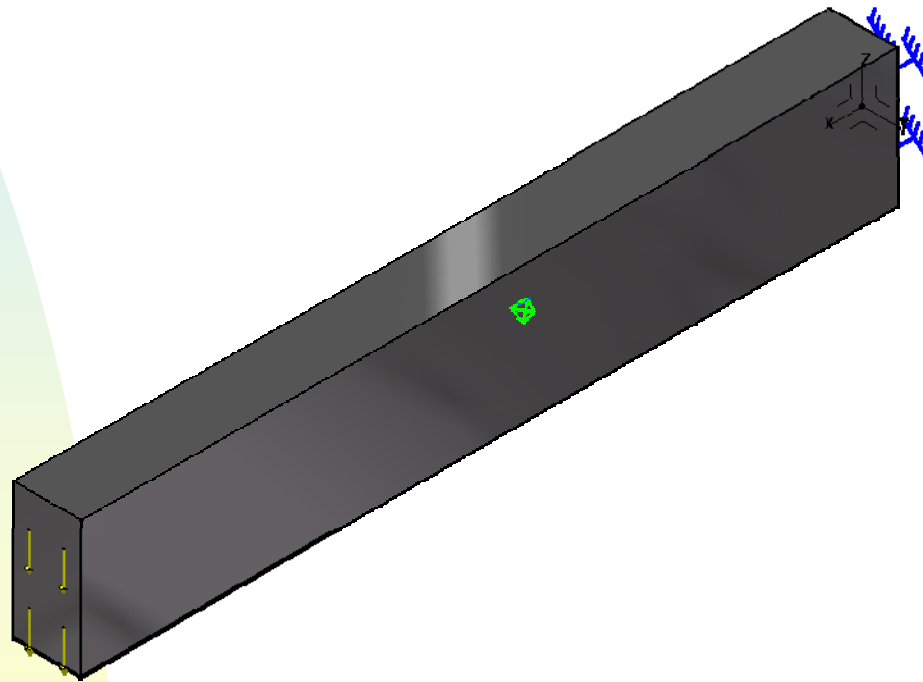


Von Mises Stress (nodal value) Global Maximum.1: 101174 psi



WORKSHOP 8

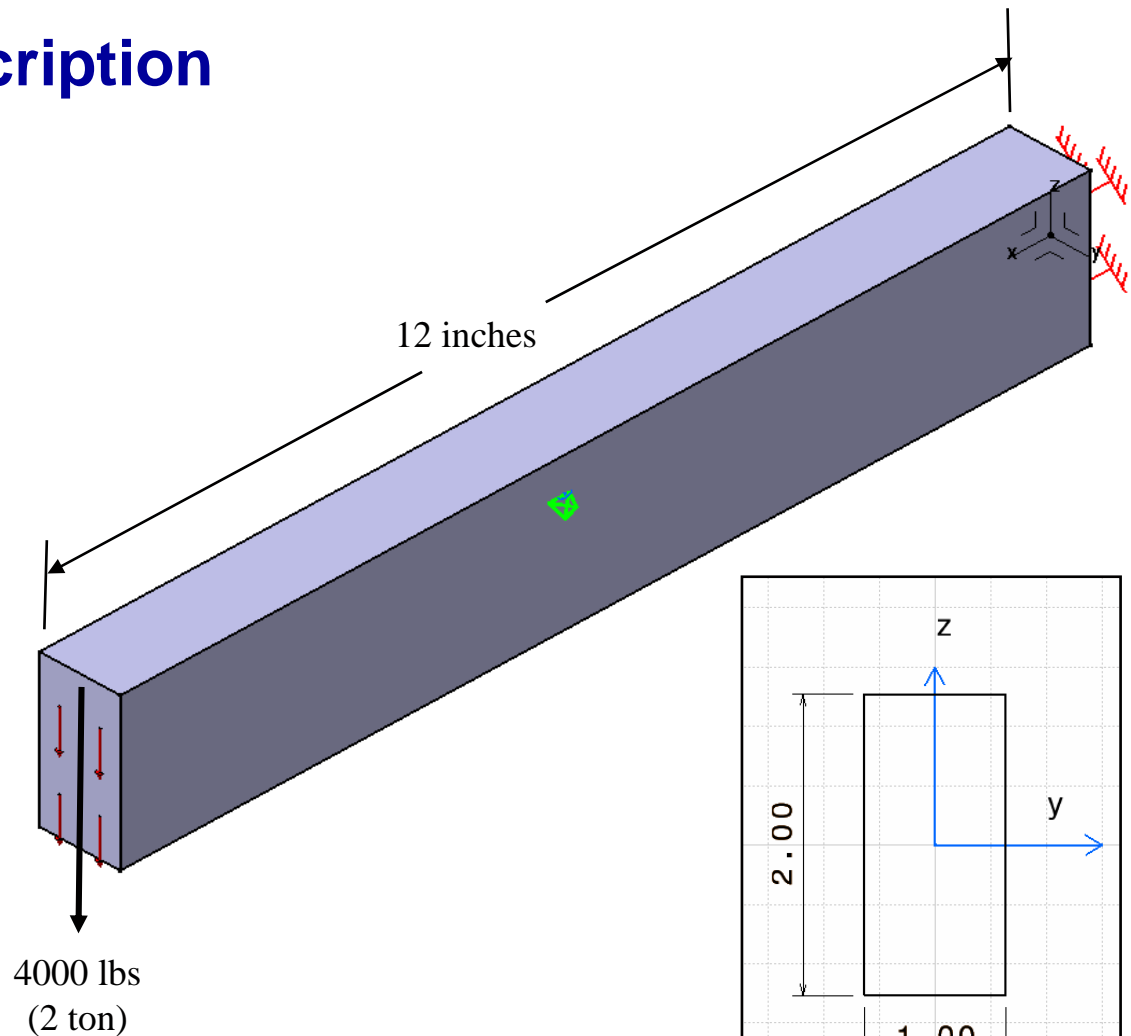
RECTANGULAR SECTION CANTILEVER BEAM



WORKSHOP 8 – RECTANGULAR 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

The screenshot displays the CATIA V5 software interface. The main window shows the 'Analysis Manager' tree on the left, which is organized as follows:

- Analysis Manager
 - Links Manager
 - Results -> C:\ELFINI\cantilever\rectangular.CATAnalysisResults
 - Computations -> C:\ELFINI\cantilever\rectangular.CATAnalysisComputations
 - Link.1 -> C:\CATIA\Training\data\ws8rectangularBeam.CATPart
 - Finite Element Model
 - Nodes and Elements
 - Properties.1
 - Material Property3D.1
 - Static Case

On the right side of the interface, a 3D model of a rectangular beam is visible. The bottom of the screen shows the software's toolbar and a coordinate system (X, Y, Z).

Four red arrows and boxes, numbered 1 through 4, indicate the steps for creating the analysis document:

- 1: Points to the 'Link.1' folder in the Analysis Manager tree.
- 2: Points to the 'Material Property3D.1' folder under 'Properties.1'.
- 3: Points to the 'Generate' button in the software toolbar.
- 4: Points to the 'Computations' folder in the Analysis Manager tree.

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.

CATIA V5 - [ws8rectangularBeam.CATAnalysis]

Start TeamPDM File Edit View Insert Tools Window Help

Analysis Manager

Links Manager

Finite Element Model

Nodes and Elements

OCTREE Tetrahedron Mesh.1 : rectangularBeam

Properties.1

Material Property3D.1

Static Case

OCTREE Tetrah...

Global Local

Size 0.25in

Sag 0.025in

Element type

Linear

Parabolic

OK Cancel

Linear TE4

Parabolic TE10

Step 2. Mesh globally with linear elements

CATIA V5 - [wsRectangularBeam.CATAnalysis]

Start TeamPDM File Edit View Insert Tools Window Help

Analysis Manager

Links Manager

Finite Element Model

Nodes and Elements

Mesh Visualization

OCTREE Tetrahedron Mesh.1 : rectangularBeam

Mesh

Properties.1

Static Case

Compute

Mesh Only

OK Cancel

1

2

3

4

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

CATIA V5 - [wsBrectangularBeam.CATAnalysis]

Start TeamPDM File Edit View Insert Tools Window Help

Analysis Manag Render Style

Links Manager

Finite Element Model 1

Nodes and Elements

OCTREE Tetrahedron Mesh.1 : rectangularBeam

Mesh

Properties.1

Static Case

Custom View Modes

- Edges_points
- Shading
- Outlines
- Hidden edges_points
- Dynamic hidden line removal
- Materials
- Facet
- Isoparametrics

OK Cancel

2

3

4

ject or a command

CATIA P2 SOLUTIONS

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

CATIA V5 - [ws8RectangularBeam.CATAnalysis]

Start TeamPDM File Edit View Insert Tools Window Help

Analysis Manager

Links Manager

Finite Element Model

Nodes and Elements

OCTREE Tetrahedron Mesh.1 : rectan

Mesh ← 1

Properties.1

Static Case

Image Fem Editor

Deformed

Display on Deformed Mesh

Mesh | Selections

Shrink

Shrink Coefficient

0.90

OK Cancel Help

2

ject or a command

CATIA V5 P2

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.

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.

The screenshot displays the CATIA V5 software interface for a finite element analysis. The main window shows a 3D model of a rectangular beam with a mesh applied. The features tree on the left is organized as follows:

- Analysis Manager
 - Links Manager
 - Finite Element Model
 - Nodes and Elements (1)
 - OCTREE Tetrahedron Mesh.1 : rectangularBeam
 - Mesh (1)
 - Prop Image Activate/DeActivate
 - Static Case
 - Restraints.1
 - Clamp.1 (5)
 - Loads.1
 - Static Case Solution.1
 - Sensors.1

Numbered callouts (1-5) indicate the steps: 1 points to the 'Mesh' feature, 2 points to the 'Shading with Edges' display mode icon, 3 points to the 'Clamp Restraint' icon in the 'Restrain' dialog, 4 points to the 'OK' button in the 'Clamp' dialog, and 5 points to the 'Clamp.1' feature in the features tree. The 'Clamp' dialog box shows 'Name: Clamp.1' and 'Supports: 1 Face'. The 'Restrain' dialog box shows the 'Clamp Restraint' icon selected. The 3D model shows the beam with a green arrow indicating the direction of the clamp restraint applied to the face at the origin.

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.

The screenshot shows the CATIA V5 interface for a finite element analysis. The Analysis Manager tree on the left is expanded to show the 'Mesh' feature under 'Nodes and Elements'. A red box labeled '3' highlights the 'Mesh' feature. Below it, the 'Static Case' is expanded to show 'Restraints.1', which contains a 'Clamp.1' feature. A red box labeled '4' highlights the 'Clamp.1' feature. A tooltip for 'Clamp.1' shows 'Restraint visualization on mesh'. To the right, a 3D model of a rectangular beam is shown with a mesh and blue arrows indicating the clamp's effect. A 'Compute' dialog box is open, with a red box labeled '1' highlighting the 'Mesh Only' option in the dropdown menu. A red box labeled '2' highlights the 'Dynamic Hidden Line Removal' icon in the bottom toolbar. The bottom toolbar also shows the 'Image Activate/DeActivate' icon, which is highlighted by a red box labeled '3'.

Step 3. Apply a clamp restraint

Analysis Manager

- Links Manager
- Finite Element Model
 - Nodes and Elements
 - OCTREE Tetrahedron M
 - Mesh ← 1
 - Properties.1
 - Static Case
 - Restraints.1
 - Clamp.1
 - Restraint symbolo
 - Loads.1
 - Static Case Solution.1
 - Sensors.1

Image Fem Editor

Deformed

Display on Deformed Mesh

Mesh Selections

All OCTREE Tetrahedron Mesh.1 : rectangularBeam Clamp.1

OK Cancel Help

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.

The screenshot shows the CATIA V5 interface for a finite element analysis. The Analysis Manager tree on the left contains the following items: Links Manager, Finite Element Model, Nodes and Elements, OCTREE Tetrahedron Mesh.1 : rectangularBeam, Mesh, Properties (Image Activate/DeActivate), Static Case, Restraints.1 (containing Clamp.1 and Restraint symbol), Loads.1 (Image Activate/DeActivate), Static Case Solution.1, and Sensors.1. The 3D model of a rectangular beam is shown with a mesh and blue restraints at one end. The 'Image Fem Editor' dialog is open, showing the 'Selections' tab with 'All' selected. Red boxes and arrows indicate the steps: 1. Double-clicking the Mesh object in the features tree; 2. Selecting the Selections tab and 'All' in the Fem Editor; 3. Deactivating the 'Restraint symbol' and 'Mesh' images; 4. Clicking the Dynamic Hidden Line Removal icon.

Step 4. Apply a distributed force

The screenshot shows the CATIA V5 interface for a static analysis. The Analysis Manager tree on the left shows the following structure:

- Analysis Manager
 - Links Manager
 - Finite Element Model
 - Nodes and Elements
 - OCTREE Tetrahedron Mesh.1 : rectangularBeam
 - Mesh
 - Properties.1
 - Static Case
 - Restraints.1
 - Clamp.1
 - Restraint symbol
 - Loads.1
 - Distributed Force.1
 - Static Case Solution.1
 - Sensors.1

The 3D view shows a rectangular beam with a distributed force applied to its end face. The force is represented by red arrows pointing downwards. The 'Distributed Force' dialog box is open, showing the following settings:

- Name: Distributed Force.1
- Supports: 1 Face
- Axis System: Global
- Type: Global
- Display locally:
- Force Vector
 - Norm: 4000lbf
 - X: 0lbf
 - Y: 0lbf
 - Z: -4000lbf

Red arrows and numbers 1, 2, and 3 indicate the steps:

1. Select the Force icon.
2. Select end face as shown.
3. Enter -4000 lbs in the Z-direction, select OK.

Steps:

1. Select the Force icon.
2. Select end face as shown.
3. Enter -4000 lbs in the Z-direction, select OK.

object or a command

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.

The screenshot shows the CATIA V5 software interface for a finite element analysis. The tree view on the left is expanded to show the 'Distributed Force.1' feature. A context menu is open over this feature, with 'Restraint visualization on mesh' selected. The 3D model on the right shows a rectangular beam mesh with a distributed force applied, indicated by yellow arrows. A 'Compute' dialog box is open, with 'Mesh Only' selected. The 'Compute' dialog box has a '1' in a red box next to it. The 'Restraint visualization on mesh' option in the context menu has a '4' in a red box next to it. The 'Image Activate/DeActivate' option in the context menu has a '3' in a red box next to it. The 'Wireframe' icon in the toolbar has a '2' in a red box next to it.

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.

Point force vector

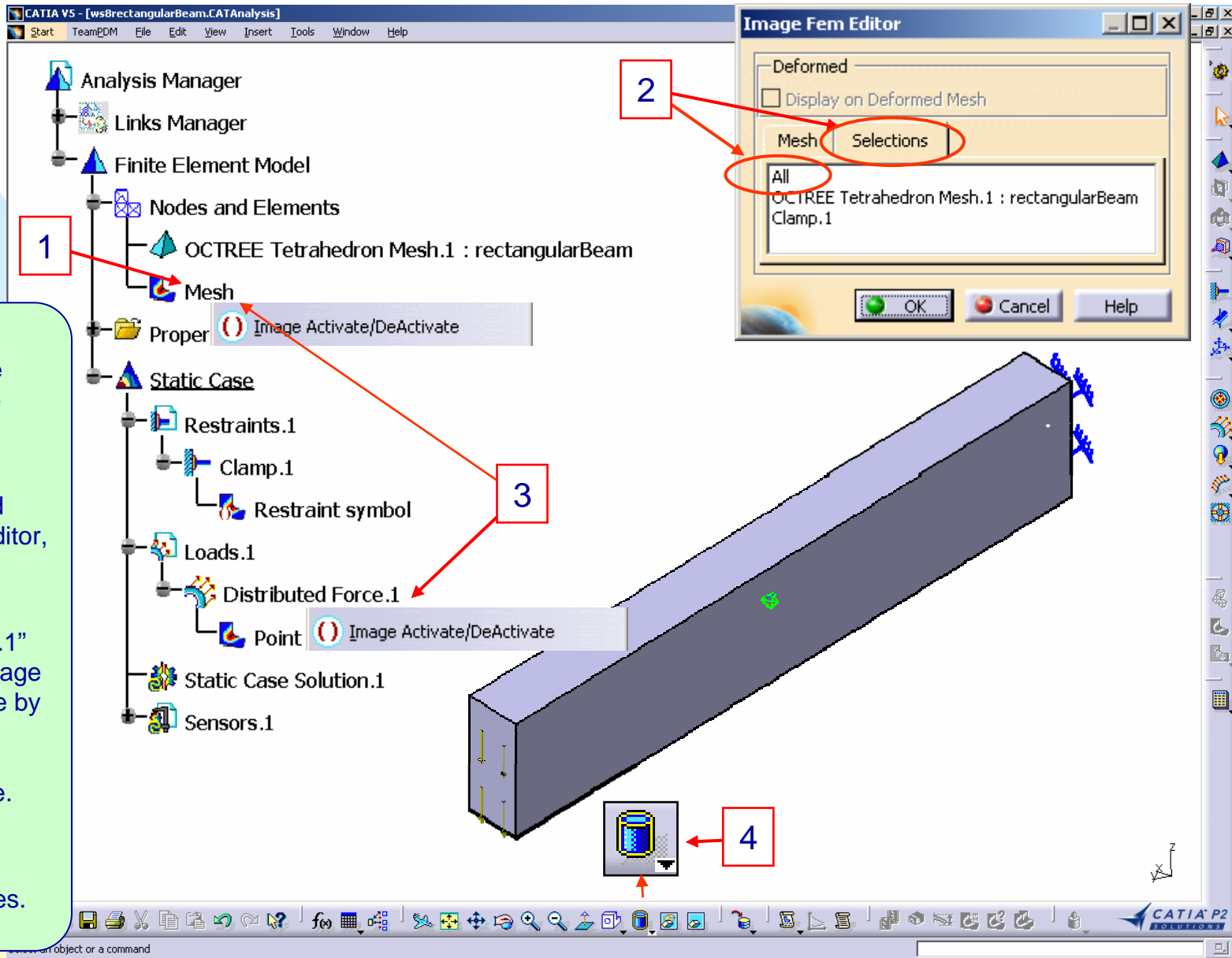
lbf

89
89
88.9
88.9
88.9
88.9
88.8
88.8
88.8
88.8

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

Analysis Manager

Links Manager

Finite Element Model

Nodes and Elements

Properties.1

Static Case

Restraints.1

Loads.1

Static Case Solution.1

Estimated local error

Sensors.1

Estimated local error

Btu

3.61e-006

3.25e-006

2.89e-006

2.53e-006

2.17e-006

1.81e-006

1.44e-006

1.08e-006

7.22e-007

3.61e-007

3.48e-010

Precision Location : Global
Estimated Precision : 1.8288 Btu
Strain Energy : 26.3011 Btu
Global Estimated Error Rate : 18.3299 %

1

2

3

4b

CATIA V5 - [ws8rectangularBeam.CATAnalysis]

Start TeamPDM File Edit View Insert Tools Window Help

MSC SOFTWARE SOLUTIONS

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

Analysis Manager

- Links Manager
- Finite Element Model
 - Nodes and Elements
 - Properties.1
 - Static Case
 - Restraints.1
 - Loads.1
 - Static Case Solution.1
 - Estimated local error
 - Extrema
 - Global Maximum.1
 - Sensors.1

Estimated local error Global Maximum.1: 3.6101e-006 Btu

Extrema Creation

- Global
 - Minimum extrema at most: 0
 - Maximum extrema at most: 2
- Local
 - Minimum extrema at most: 0
 - Maximum extrema at most: 2

OK Cancel

1 2 3

Select an object or a command

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

Analysis Manager

- Links Manager
- Finite Element Model **4**
- Nodes and Elements
- Properties.1
- Static Case
 - Restraints.1
 - Loads.1
 - Static Case Solution.1
 - Estimated local error
 - Extrema **2b**
 - Global Maximum.1
 - Sensors.1
- Adaptivity Process
 - Adaptivity Convergence.1
 - Adaptivities.1
 - Adaptivity Box.1

Adaptivity Box

Name: Adaptivity Box.1

Objective Error (%): 5

Solution: Static Case Solution.1

Local Error (%): 20.174

Select Extremum

OK Cancel

1

2a

2b

3

Estimated local error Global Maximum.1: 3.6101e-006 Btu

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

Analysis Manager

Links Manager

Finite Element Model

- Nodes and Elements
 - OCTREE Tetrahedron Mesh.1 : rectangularBeam
 - Mesh
- Properties.1
- Static Case
 - Restraints.1
 - Loads.1
 - Static Case Solution.1
 - Estimated local error
 - Extrema
 - Sensors.1
- Adaptivity Process
 - Adaptivity Convergence.1
 - Adaptivities.1
 - Adaptivity Box.1

1

2

OCTREE Tetrah...

Global | Local

Size 0.25in

Sag 0.025in

Element type

Linear

Parabolic

OK Cancel

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

Analysis Manager

- Links Manager
- Finite Element Model
 - Nodes and Elements
 - Properties.1
 - Static Case
 - Restraints.1
 - Loads.1
 - Static Case Solution.1
 - Sensors.1

Compute

All

Preview

OK Cancel

Computation Resources Estimation

2e+002 s of CPU
1.38e+004 kilo-bytes of memory
1.03e+005 kilo-bytes of disk
Intel MKL(c) Library found: Intel(R) MKL V5.1.0

Do you want to continue the computation?

Yes No

1

2

3

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

CATIA V5 - [wsBrectangularBeam.CATAnalysis]

Start TeamPDM File Edit View Insert Tools Window Help

Analysis Manager

- Links Manager
- Finite Element Model
 - Nodes and Elements
 - Properties.1
 - Static Case
 - Restraints.1
 - Loads.1
 - Static Case Solution.1
 - Estimated local error
 - Extrema
 - Sensors.1
 - Adaptivity Process

Estimated local error Btu

- 2.5e-007
- 2.25e-007
- 2e-007
- 1.75e-007
- 1.5e-007
- 1.25e-007
- 1e-007
- 7.51e-008
- 5.01e-008
- 2.5e-008
- 2.26e-013

Precision Location : Global
Estimated Precision : 0.00859889 Btu
Strain Energy : 27.3213 Btu
Global Estimated Error Rate : 1.25436 %

1

2

3

Image Activate/DeActivate

object or a command

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

Analysis Manager

- Links Manager
- Finite Element Model
 - Nodes and Elements
 - Properties.1
 - Static Case
 - Restrains.1
 - Loads.1
 - Static Case Solution.1
 - Estimated local error **1**
 - Extrema
 - Sensors.1
 - Adaptivity Process
 - Adaptivity Convergence.1
 - Adaptivities.1 **2**
 - Adaptivity Box.1

Adaptivity Box

Name: Adaptivity Box.1

Objective Error (%): 5

Solution: Static Case Solution.1

Local Error (%): 2.9285 **3**

Select Extremum

OK Cancel

Estimated local error Global Maximum, 1: 2.5043e-007 Btu

CATIA P2 SOLUTIONS

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

CATIA V5 - [wsBrectangularBeam.CATAnalysis]

Start TeamPDM File Edit View Insert Tools Window Help

Analysis Manager

- Links Manager
- Finite Element Model
 - Nodes and Elements
 - Properties.1
 - Static Case
 - Restraints.1
 - Loads.1
 - Static Case Solution.1
 - Estimated local error
 - Translational displacement vector
 - Sensors.1
 - Adaptivity Process

Hide/Show

Translational displacement vector

in

0.122

0.11

0.0974

0.0852

0.0731

0.0609

0.0487

0.0365

0.0244

0.0122

0

On Boundary

1

2

CATIA V5 SOLUTIONS

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

The screenshot displays the CATIA V5 software interface for a finite element analysis. The Analysis Manager tree on the left shows the following structure:

- Analysis Manager
 - Links Manager
 - Finite Element Model
 - Nodes and Elements
 - Properties.1
 - Static Case
 - Restraints.1
 - Loads.1
 - Static Case Solution.1
 - Estimated local error
 - Translational displacement vector
 - Extrema
 - Global Maximum.1
 - Sensors.1
 - Adaptivity Process

The main 3D view shows a beam with a downward load (orange circle) and reaction forces (red arrows). A callout box indicates: "Translational displacement vector Global Maximum.1: 0,121914 in".

The Extrema Creation dialog box is open, showing the following settings:

Category	Minimum extrema at most	Maximum extrema at most
Global	0	2
Local	0	2

Annotations in the image include:

- 1a: Points to the Extrema icon in the tree.
- 1b: Points to the Global checkbox in the Extrema Creation dialog.
- 2: Points to the Global Maximum.1 feature in the tree.

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.

0.0149 = V1 = X
-2.24e-005 = V2 = Y
-0.121 = V3 = Z

Translational displacement vector

Image Edition

Deformed

Display on Deformed Mesh

Visu | Criteria | **Filters** | Selections

Position : Node

Component : ALL

Lamina : ALL

Sup-Mid-Inf : V1
V2
V3

Repeat : V1V2
V1V3
V2V3

Edition : Iso/Fringe | Symbol | Axis System

OK Cancel Help

Step 9. Visualize final results

Change the displacement image from symbols to Average-ISO

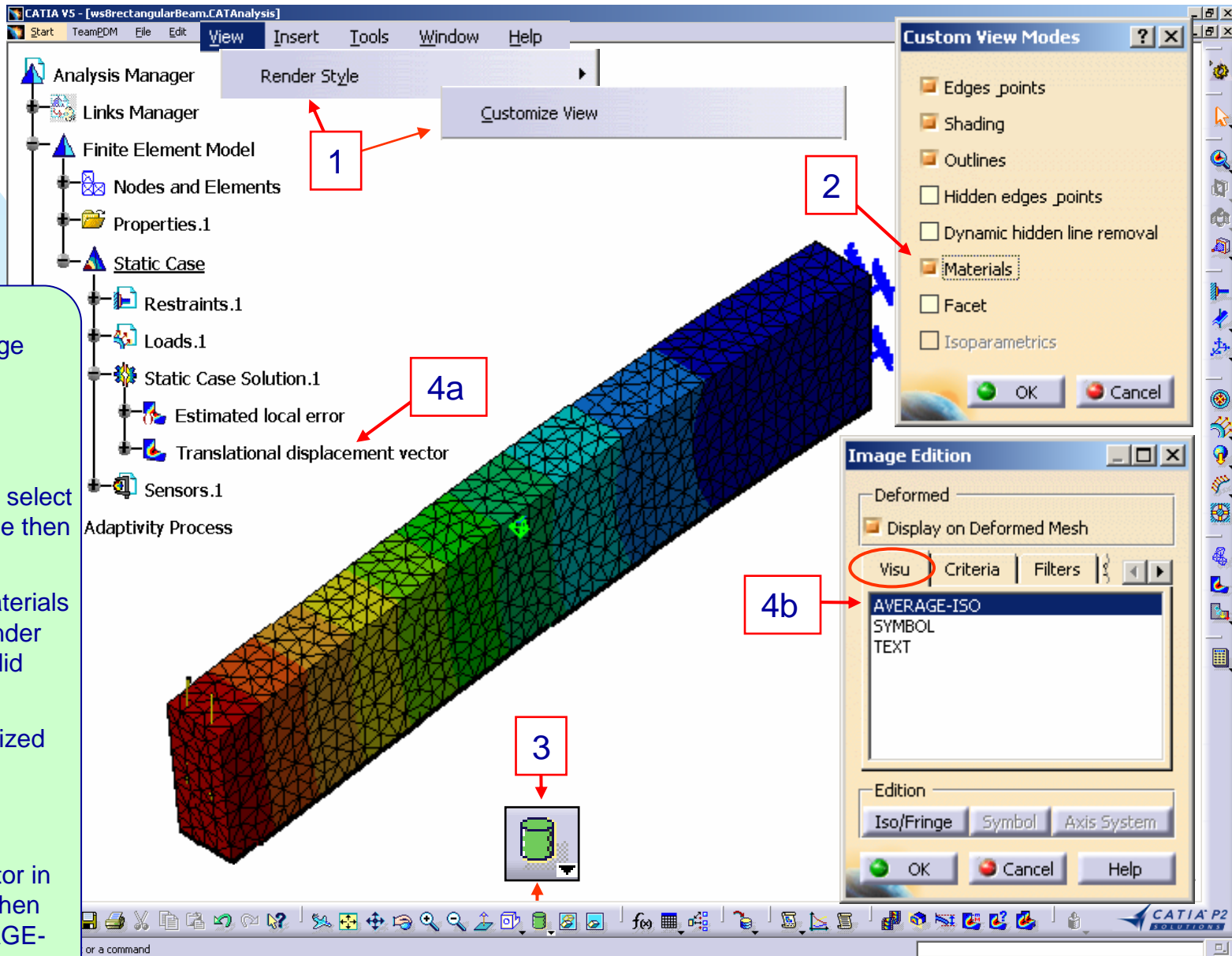
Steps:

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

Analysis Manager

- Links Manager
- Finite Element Model
 - Nodes and Elements
 - Properties.1
 - Static Case
 - Restraints.1
 - Loads.1
 - Static Case Solution.1
 - Estimated local error
 - Translational displacement magnitude
 - Von Mises Stress (nodal value)
 - Sensors.1

Adaptivity Process

Von Mises Stress (nodal value)

psi

8.62e+004
7.77e+004
6.92e+004
6.07e+004
5.22e+004
4.38e+004
3.53e+004
2.68e+004
1.83e+004
9.84e+003
1.36e+003

On Boundary

1

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

Analysis Manager

- Links Manager
- Finite Element Model
 - Nodes and Elements
 - Properties.1
 - Static Case
 - Restraints.1
 - Loads.1
 - Static Case Solution.1
 - Estimated local error
 - Translational displacement magnitude
 - Von Mises Stress (nodal value)
 - Extrema
 - Global Maximum.1
 - Sensors.1

- Adaptivity Process

Von Mises Stress (nodal value) Global Maximum.1: 85427,6 psi

Von Mises Stress (nodal value) Global Maximum.1: 85427,6 psi

Extrema Creation

- Global
 - Minimum extrema at most: 0
 - Maximum extrema at most: 2
- Local
 - Minimum extrema at most: 0
 - Maximum extrema at most: 2

OK Cancel

1a

1b

2

Focus On

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.

The screenshot displays the CATIA V5 interface for a finite element analysis. The Analysis Manager tree on the left shows the following structure:

- Analysis Manager
 - Links Manager
 - Finite Element Model
 - Nodes and Elements
 - Properties.1
 - Static Case
 - Restraints.1
 - Loads.1
 - Static Case Solution.1
 - Estimated local error
 - Translational displacement magnitude
 - Von Mises Stress (nodal value) ← 2a
 - Extrema ← 1
 - Sensors.1
 - Adaptivity Process

Two 'Image Edition' dialog boxes are open. The first, labeled '2b', has the 'Visu' tab selected, showing a list of visualization options: AVERAGE-ISO, DISCONTINUOUS-ISO, and TEXT. The second, labeled '3', has the 'Filters' tab selected, with 'Position' set to 'Node'. A third dialog box, 'Image Iso Fringe Editor', is also open, with 'ISO smooth' and 'Display Element Without Value' options unchecked, labeled '4'. A 'Hide/Show' button is visible next to the 'Extrema' object in the tree.

Step 9. Visualize final results

Find exact recommend design stress.

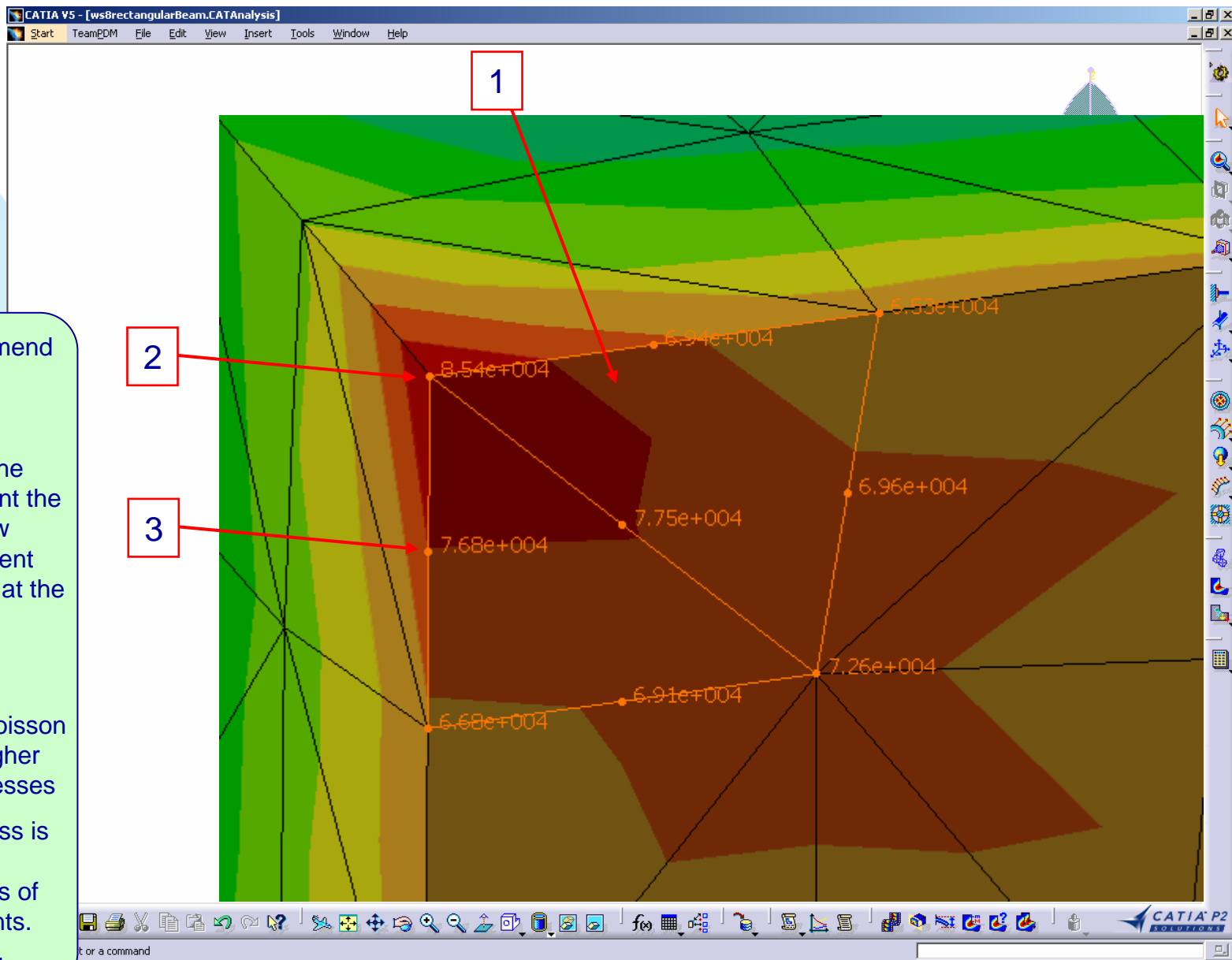
Steps:

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



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.

The screenshot displays the CATIA V5 interface for visualizing simulation results. The Analysis Manager on the left shows a tree structure with the following items: Analysis Manager, Links Manager, Finite Element Model, Nodes and Elements, Properties.1, Static Case, Restraints.1, Loads.1, Static Case Solution.1, Estimated local error, Translational displacement magnitude, Von Mises Stress (nodal value), Stress full tensor symbol (column), and Sensors.1. The 'Stress full tensor symbol (column)' is highlighted with a red box and labeled '2'. The 'Image Edition' dialog box is open, showing the 'Criteria' tab selected with a red box and labeled '3'. The list of criteria includes 'PRINCIPAL-VALUE', 'ABS-PRINCIPAL-VALUE', and 'MATRIX-COLUMN', with 'MATRIX-COLUMN' selected. The 'Image Edition' dialog box is also open, showing the 'Filters' tab selected with a red box and labeled '4'. The 'Component' is set to 'Col3'. The 3D model on the right shows a deformed mesh with blue stress vectors and yellow arrows indicating the direction of the stress. A red box labeled '1' points to a small icon in the bottom right corner of the 3D view.

Step 9. Visualize final results

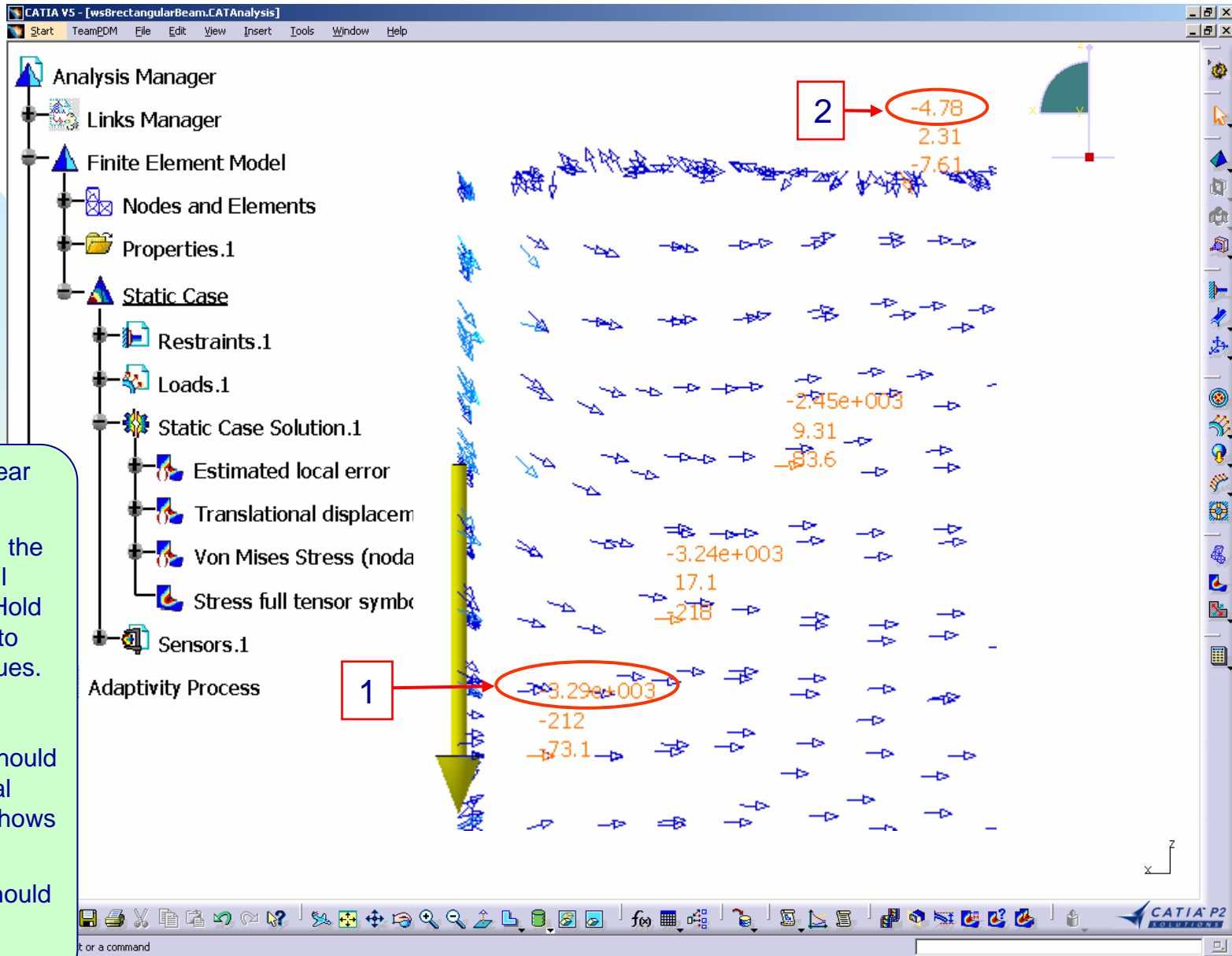
Find horizontal shear stress.

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.



Step 9. Visualize final results

■ Conclusions

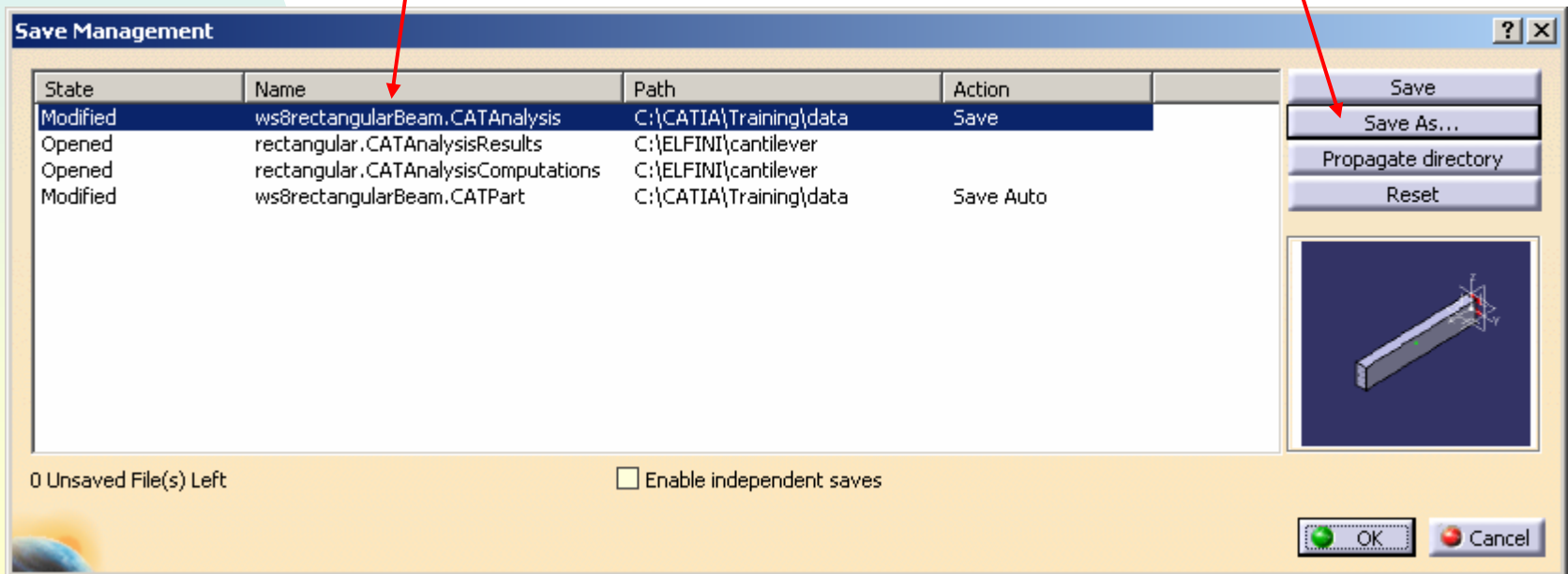
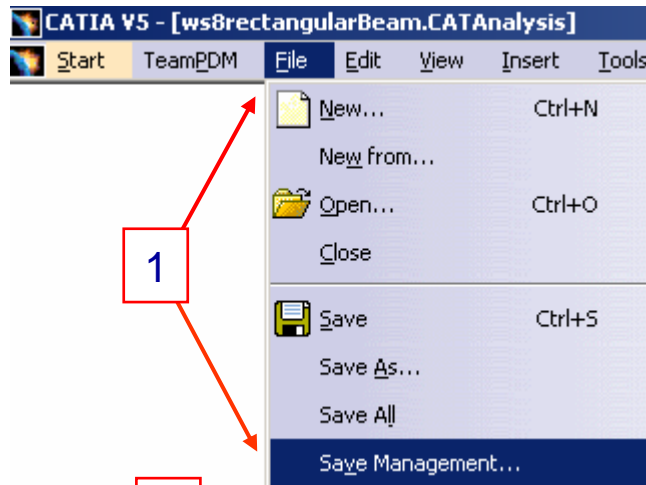
- ◆ CATIA V5 GSA workbench is validated for a rectangular cantilever beam scenario. To be conservative, increase material strength to a minimum yield of 77000 psi for the described load case.

	Hand Calculations	.25" Parabolic Global Mesh, .025" sag
Global % Precision error	NA	1.25 %
Local % Precision error	NA	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.

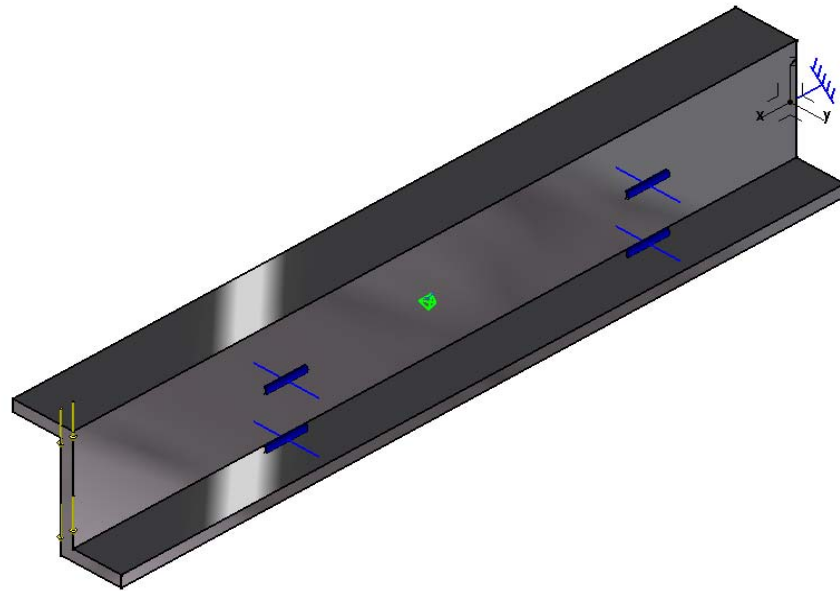
a = area of section where stress is desired or applied (in²)
b = width of section (in)
c = distance from neutral axis to extreme fiber (in)
d = depth of section (in)
e = eccentricity of applied load (in)
f = force per linear inch (in)
g = acceleration of gravity (386.4 inch/sec²)
h = height (in)
k = 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

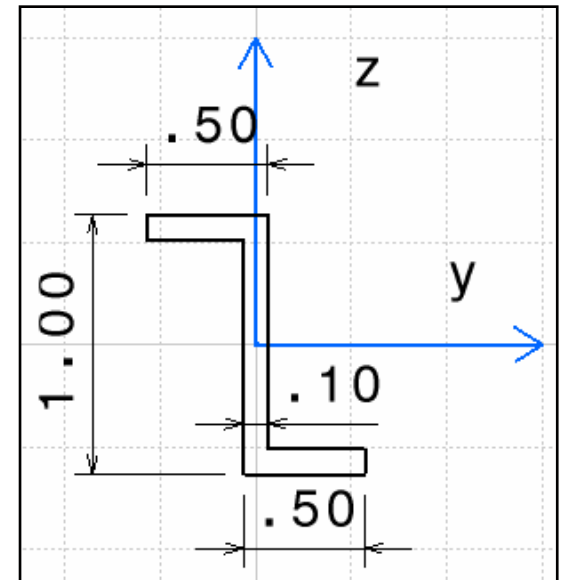
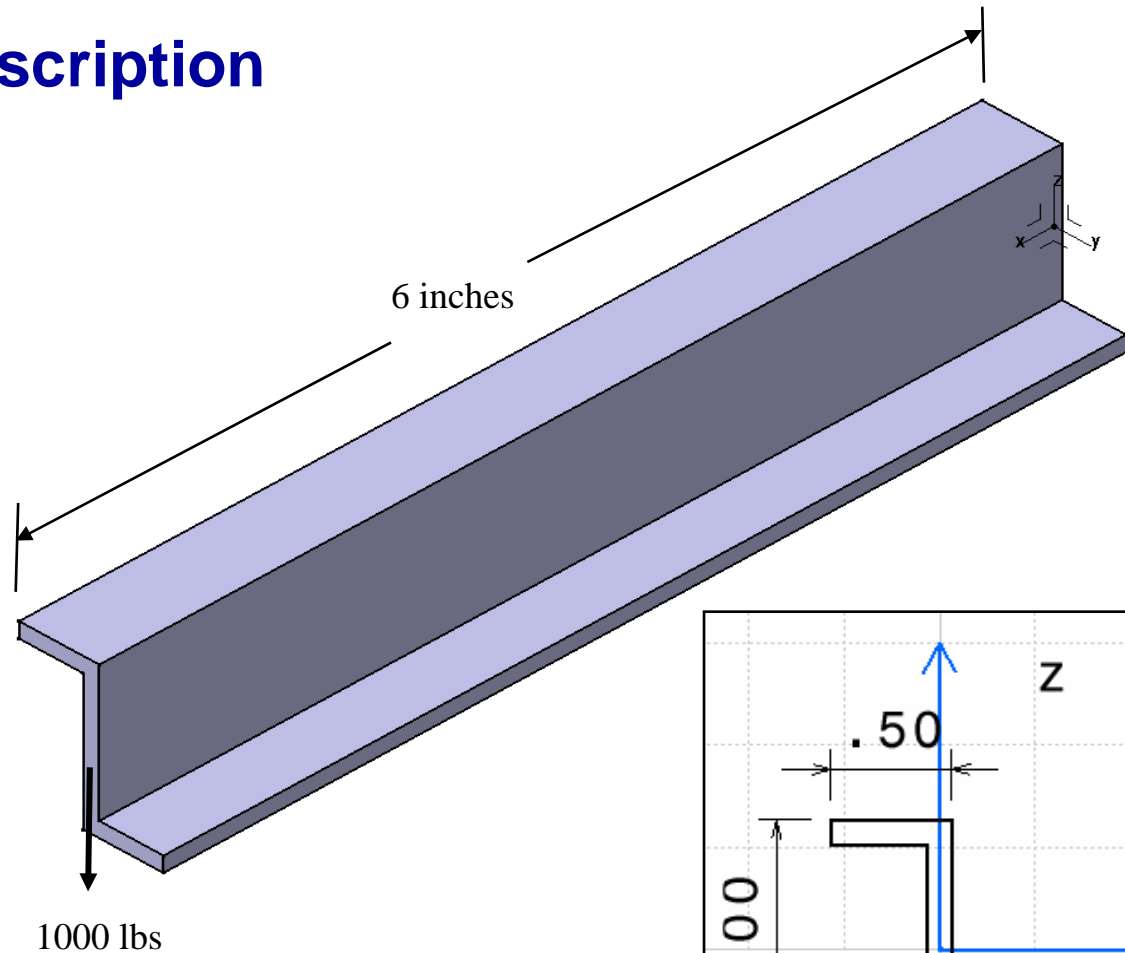
Z-SECTION CANTILEVER BEAM



WORKSHOP 8b – Z-SECTION CANTILEVER BEAM

■ Problem Description

- ◆ Load case.



Material: Aluminum
Young Modulus = 10.15×10^6 psi
Shear Modulus = 3.77×10^6 psi
Poisson Ratio = .346
Density = .098 lb_in³
Yield Strength = 13778 psi

■ Problem Description

◆ Bending and shear displacement

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

$$\text{Bending Displacement} = \delta = \frac{P \cdot L^3}{(3 \cdot E \cdot I)} = \frac{1000 \text{ lbs} \cdot 6^3 \text{ inch}}{(3 \cdot 10.15 \times 10^6 \text{ psi} \cdot 0.0246 \text{ in}^4)} = 0.288 \text{ inch}$$

$$\text{Shear Displacement} = \delta = \frac{P \cdot L}{(E_s \cdot A)} = \frac{1000 \text{ lbs} \cdot 6 \text{ inch}}{(3.77 \times 10^6 \text{ psi} \cdot 0.1 \text{ in}^2)} = 0.0159 \text{ inch} \quad \delta_{\text{Combined}} = 0.288 \text{ in} + 0.0159 \text{ in} = 0.304 \text{ in}$$

◆ Bending stress

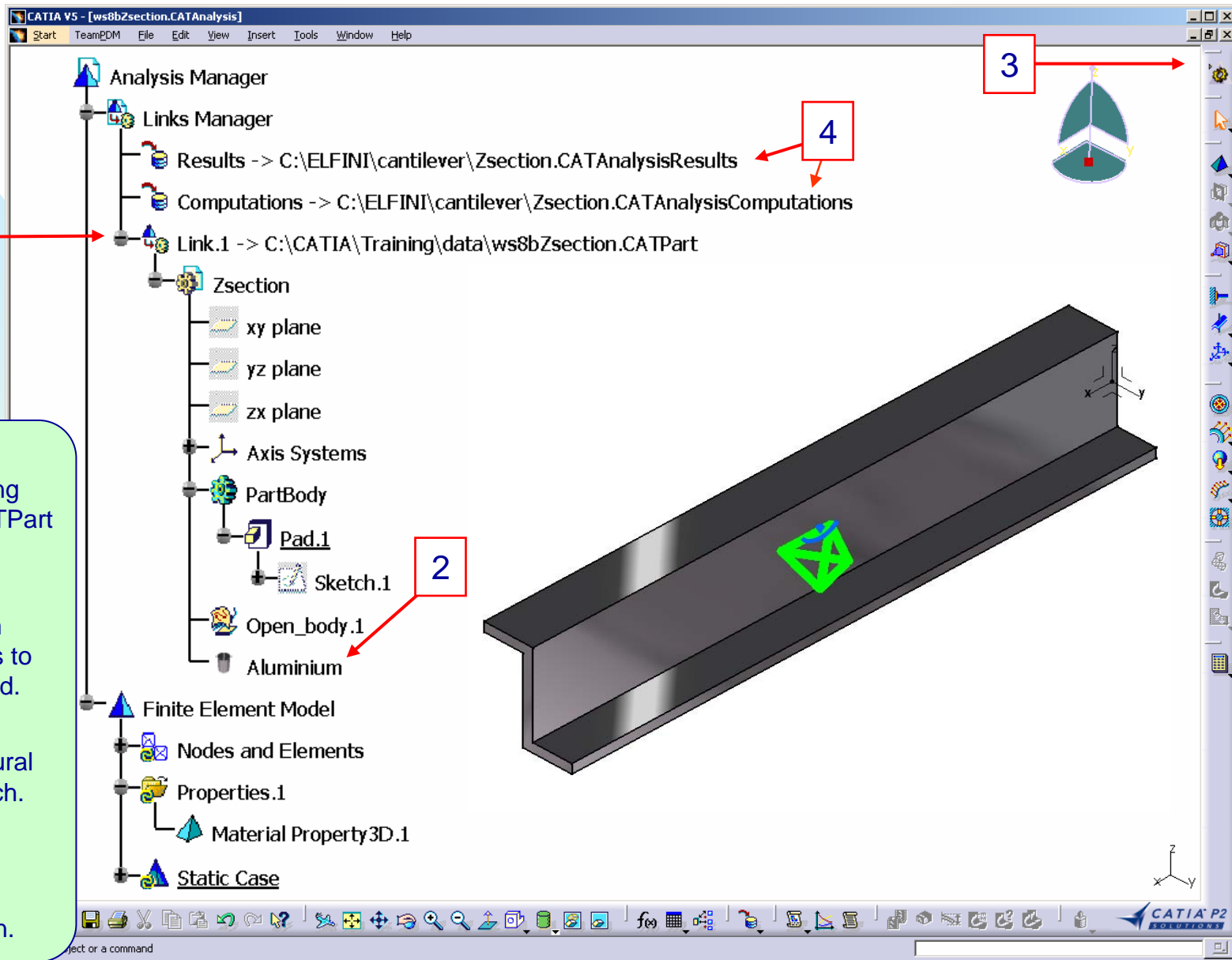
$$\text{Bending Moment} = M_y = P \cdot L = 1000 \text{ lbs} \cdot 6 \text{ inch} = 6000 \text{ in lbs}$$

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

■ 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

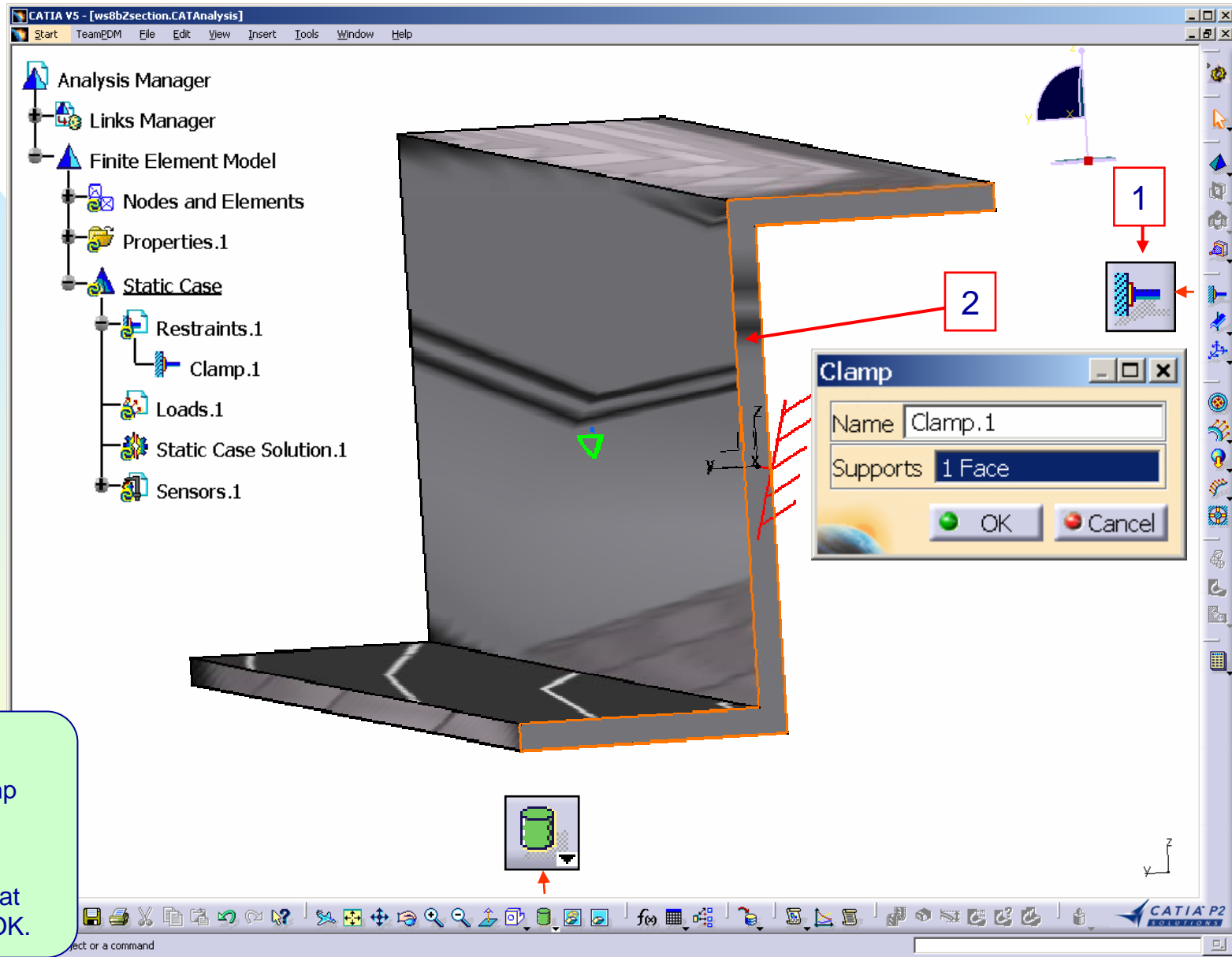
Steps:

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).

The screenshot shows the CATIA V5 interface with the following elements:

- Tree View:** Analysis Manager, Links Manager, Finite Element Model, Nodes and Elements, OCTREE Tetrahedron Mesh.1 : Zsection, Properties.1, Static Case.
- 3D Model:** A Z-section part with a green mesh overlay. Red arrows and numbers 1-5 indicate the steps.
- OCTREE Tetrahedron Mesh Dialog (Top):** Global tab selected. Size: 0.75in, Sag: 0.07in. Element type: Linear (selected). A context menu is open over the Size field with options: Measure... and Add Range...
- OCTREE Tetrahedron Mesh Dialog (Bottom):** Global tab selected. Size: 0.1in, Sag: 0.07in. Element type: Linear (selected). A callout box says "Change to .01in" pointing to the Sag field.
- Measure command ended Dialog:** A question mark icon and the text "Do you want to copy the result of this measure in this parameter?". A "Yes" button is highlighted with a red arrow and the number 4.

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

CATIA V5 - [ws8bZsection.CATAnalysis]

Analysis Manager

- Links Manager
- Finite Element Model
 - Nodes and Elements
 - Properties.1
 - Static Case
 - Restraints.1
 - Clamp.1
 - Loads.1
 - Distributed Force.1
 - Static Case Solution.1
 - Sensors.1

1

2

3

Distributed Force

Name: Distributed Force.1

Supports: 2 Edges

Axis System

Type: Global

Display locally

Force Vector

Norm: 1000lbf

X: 0lbf

Y: 0lbf

Z: -1000lbf

OK Cancel

select or a command

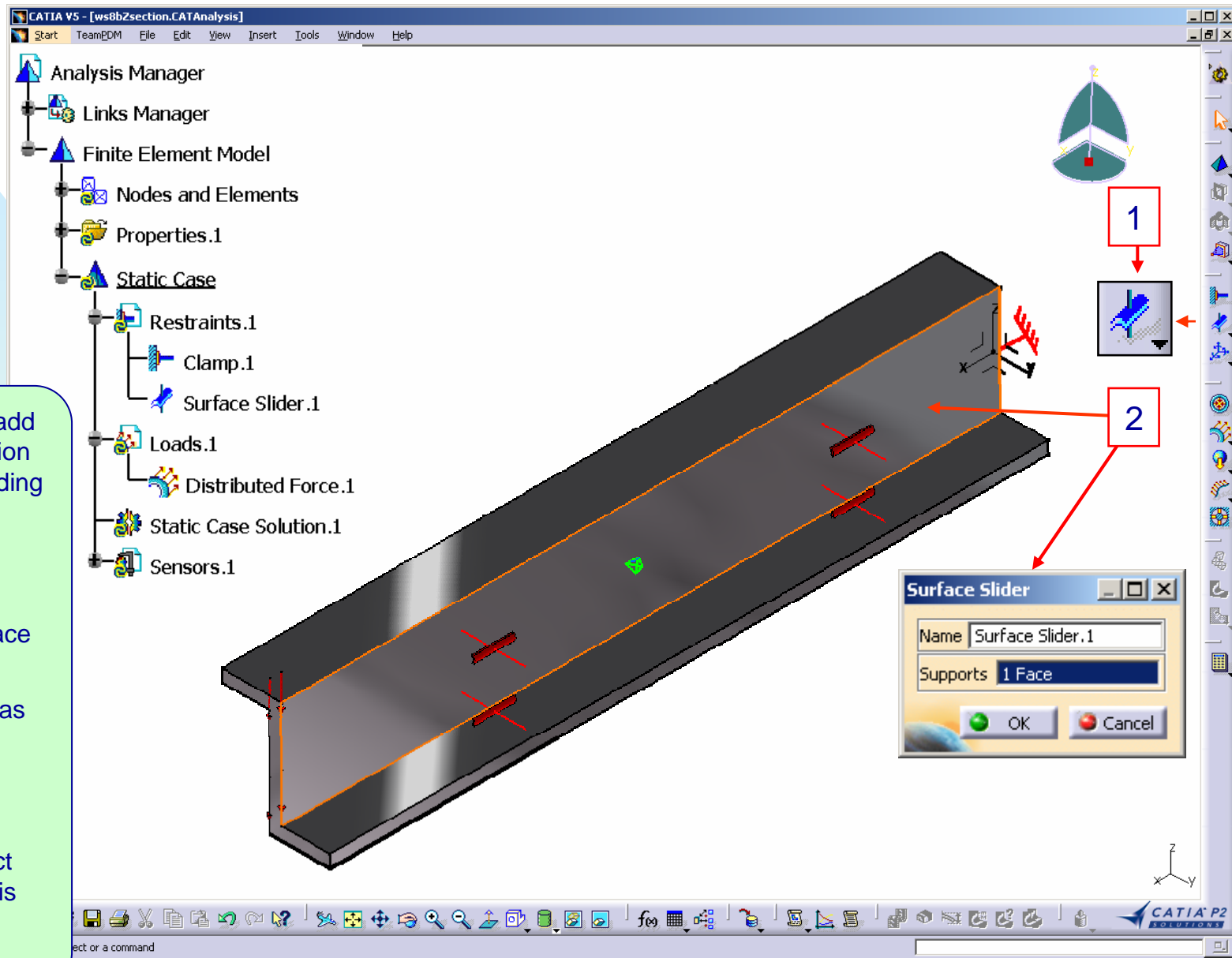
Load only on the web

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.

4 s of CPU
2.06e+003 kilo-bytes of memory
6.22e+003 kilo-bytes of disk
Intel MKL(c) Library found: Intel(R) MKL V5.1.0

Do you want to continue the computation?

Yes No

Step 6. Check global and local precision

CATIA V5 - [ws8bZsection.CATAnalysis]

Analysis Manager

- Links Manager
- Finite Element Model
 - Nodes and Elements
 - Properties.1
 - Static Case
 - Restraints.1
 - Loads.1
 - Static Case Solution.1
 - Deformed Mesh
 - Sensors.1

ISO View

Front View

1

2

Animate Window

Steps Number 10

Speed

Close

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

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

Analysis Manager

- Links Manager
- Finite Element Model
 - Nodes and Elements
 - Properties.1
 - Static Case
 - Restrains.1
 - Loads.1
 - Static Case Solution.1
 - Deformed Mesh
 - Estimated local error
 - Sensors.1

Precision Location : Global
Estimated Precision : 0.687878 Btu
Strain Energy : 16.8486 Btu
Global Estimated Error Rate : 14.1439 %

Estimated local error

Btu

3.26e-006
2.93e-006
2.6e-006
2.28e-006
1.95e-006
1.63e-006
1.3e-006
9.77e-007
6.51e-007
3.26e-007
6.96e-011

1

2

3

CATIA V5 - [ws8bZsection.CATAnalysis]

Start TeamPDM File Edit View Insert Tools Window Help

object or a command

CATIA P2 SOLUTIONS

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

Analysis Manager
Links Manager
Finite Element Model
Nodes and Elements
Properties.1
Static Case
Restrains.1
Loads.1
Static Case Solution.1
Deformed Mesh
Estimated local error
Extrema
Global Maximum.1
Sensors.1

Estimated local error Global Maximum.1: 3.25528e-006 Btu

Extrema Creation

Global
Minimum extrema at most 0
Maximum extrema at most 2

Local
Minimum extrema at most 0
Maximum extrema at most 2

OK Cancel

Focus On

1 2 3

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

Adaptivity Box

Name: Adaptivity Box.1
Objective Error (%): 5
Solution: Static Case Solution.1
Local Error (%): 21.498
Select Extremum
OK Cancel

Analysis Manager

- Links Manager
- Finite Element Model **4**
- Nodes and Elements
- Properties.1
- Static Case
 - Restraints.1
 - Loads.1
 - Static Case Solution.1
 - Deformed Mesh
 - Estimated local error
 - Extrema
 - Global Maximum.1 **2b**
 - Sensors.1
- Adaptivity Process
 - Adaptivity Convergence.1
 - Adaptivities.1
 - Adaptivity Box.1

Steps:

1. Select the adaptivity box icon. **1**
2. Select the "Select Extremum" button then Global Maximum.1 in the features tree to locate box. **2a**
3. Use the compass and green dots to locate and size box around meshed areas. **3**
4. Since local error is above 10% try changing the mesh element to Parabolic. **4**

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

The screenshot shows the CATIA V5 software interface. On the left, the Analysis Manager tree is visible, with 'OCTREE Tetrahedron Mesh.1 : Zsection' highlighted. A red box labeled '1' points to this entry. In the center, a 3D model of a mechanical part is shown with a mesh. On the right, the 'OCTREE Tetrahedron Mesh' dialog box is open, showing the 'Global' tab. The 'Element type' section has the 'Parabolic' radio button selected, indicated by a red box labeled '2'. The dialog also shows 'Size' set to 0.1in and 'Sag' set to 0.01in. The 'OK' button is highlighted.

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

The screenshot shows the CATIA V5 interface with the following components:

- Analysis Manager:** A tree view on the left containing:
 - Links Manager
 - Finite Element Model
 - Nodes and Elements
 - Properties.1
 - Static Case
 - Restraints.1
 - Loads.1
 - Static Case Solution.1
 - Sensors.1
 - Adaptivity Process

- Compute Dialog:** A dialog box titled 'Compute' with a dropdown menu set to 'All'. A red box with the number '2' points to this dropdown.
- Computation Resources Estimation Dialog:** A dialog box showing resource requirements:
- 2e+002 s of CPU
- 1.66e+004 kilo-bytes of memory
- 1.11e+005 kilo-bytes of disk
- Intel MKL(c) Library found: Intel(R) MKL V5.1.0A red box with the number '3' points to this dialog. Below the list, it asks 'Do you want to continue the computation?' with 'Yes' and 'No' buttons. The 'Yes' button is highlighted with a red box.
- Toolbar:** A vertical toolbar on the right side of the interface. A red box with the number '1' points to the 'Compute' icon (a calculator icon).
- 3D Model:** A 3D model of a mechanical part, possibly a bracket, with blue arrows indicating loads and red arrows indicating restraints.

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

Analysis Manager

- Links Manager
- Finite Element Model
 - Nodes and Elements
 - OCTREE Tetrahedron M
 - Properties.1
 - Static Case
 - Restrains.1
 - Loads.1
 - Static Case Solution.1
 - Deformed Mesh
 - Estimated local error
 - Extrema
 - Sensors.1
 - Adaptivity Process

Estimated local error
Btu

3.11e-007
2.79e-007
2.48e-007
2.17e-007
1.86e-007
1.55e-007
1.24e-007
9.32e-008
6.21e-008
3.11e-008
3.99e-014

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.

Precision Location : Global
Estimated Precision : 0.019525 Btu
Strain Energy : 17.2517 Btu
Global Estimated Error Rate : 2.37816 %

Step 8. Compute the precise analysis

The screenshot displays the CATIA V5 interface for a finite element analysis. The Analysis Manager tree on the left shows the following structure:

- Analysis Manager
 - Links Manager
 - Finite Element Model
 - Nodes and Elements
 - OCTREE Tetra
 - Properties.1
 - Static Case
 - Restrains.1
 - Loads.1
 - Static Case Solution.1
 - Deformed Mesh
 - Estimated local error (1)
 - Extrema
 - Local Update
 - Global Maximum.1
 - Sensors.1 (2)
 - Adaptivity Process (3)

The Adaptivity Box dialog is open, showing the following settings:

- Name: Adaptivity Box.1
- Objective Error (%): 5
- Solution: Static Case Solution.1
- Local Error (%): 5.7693
- Buttons: Select Extremum, OK, Cancel

The main 3D view shows a finite element mesh of a mechanical part. A red wireframe box highlights a specific region of the mesh. An orange text box overlaid on the mesh reads: "Estimated local error Global Maximum, 1: 3.10539e-007 Btu".

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

CATIA V5 - [ws8b2section.CATAnalysis]

Start TeamPDM File Edit View Insert Tools Window Help

Analysis Manager

Links Manager

Finite Element Model

Nodes and Elements

Properties.1

Static Case

Restrains.1

Loads.1

Static Case Solution.1

Deformed Mesh

Estimated local error

Translational displacement vector

Sensors.1

Adaptivity Process

Hide/Show

Translational displacement vector

in

0.308

0.277

0.246

0.216

0.185

0.154

0.123

0.0924

0.0616

0

On Boundary

1

2

CATIA V5 SOLUTIONS

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

Analysis Manager

- Links Manager
- Finite Element Model
 - Nodes and Elements
 - Properties.1
 - Static Case
 - Restraints.1
 - Loads.1
 - Static Case Solution.1
 - Deformed Mesh
 - Estimated local error
 - Translational displacement vector
 - Von Mises Stress (nodal value)**
 - Sensors.1

Adaptivity Process

Von Mises Stress (nodal value)
psi

1.51e+005
1.36e+005
1.21e+005
1.05e+005
9.04e+004
7.54e+004
6.03e+004
4.53e+004
3.02e+004
1.52e+004
151

On Boundary

1

CATIA V5 - [ws8b2section.CATAnalysis]

Start TeamPDM File Edit View Insert Tools Window Help

CATIA P2 SOLUTIONS

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

Von Mises Stress (nodal value) Global Maximum.1: 150612 psi

Von Mises Stress (nodal value) Global Maximum.1: 150612 psi

Extrema Creation

Global

Minimum extrema at most 0

Maximum extrema at most 2

Local

Minimum extrema at most 0

Maximum extrema at most 2

OK Cancel

1a

1b

2

Focus On

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.

The screenshot displays the CATIA V5 software interface for a finite element analysis. The Analysis Manager tree on the left shows the following structure:

- Analysis Manager
 - Links Manager
 - Finite Element Model
 - Nodes and Elements
 - Properties.1
 - Static Case
 - Restraints.1
 - Loads.1
 - Static Case Solution
 - Deformed Mesh
 - Estimated local error
 - Translational displacement vector
 - Von Mises Stress (nodal value)
 - Extrema
 - Sensors.1
 - Adaptivity Process

Two **Image Edition** dialog boxes are open. The left one (labeled 2b) has the **Visu** tab selected, showing a list of visualization options: AVERAGE-ISO, DISCONTINUOUS-ISO, and TEXT. The right one (labeled 3) has the **Filters** tab selected, with the **Position** dropdown set to **Node**. The **Image Iso Fringe Editor** dialog box (labeled 4) is also open, showing the **ISO value** dropdown set to **IsoContour**, and the **ISO smooth** checkbox is unchecked.

A 3D model of a mechanical part is shown in the bottom right, with a stress distribution visualization. The model is colored with a gradient from blue (low stress) to red (high stress). A coordinate system (X, Y, Z) is visible in the bottom right corner.

Red arrows and boxes labeled 1, 2a, 2b, 3, and 4 point to specific actions in the software:

- 1: Points to the **Extrema** object in the Analysis Manager tree.
- 2a: Points to the **Hide/Show** button below the **Extrema** object.
- 2b: Points to the **Visu** tab in the left **Image Edition** dialog box.
- 3: Points to the **Filters** tab in the right **Image Edition** dialog box.
- 4: Points to the **ISO smooth** checkbox in the **Image Iso Fringe Editor** dialog box.

Step 9. Visualize final results

Find exact recommend design stress.

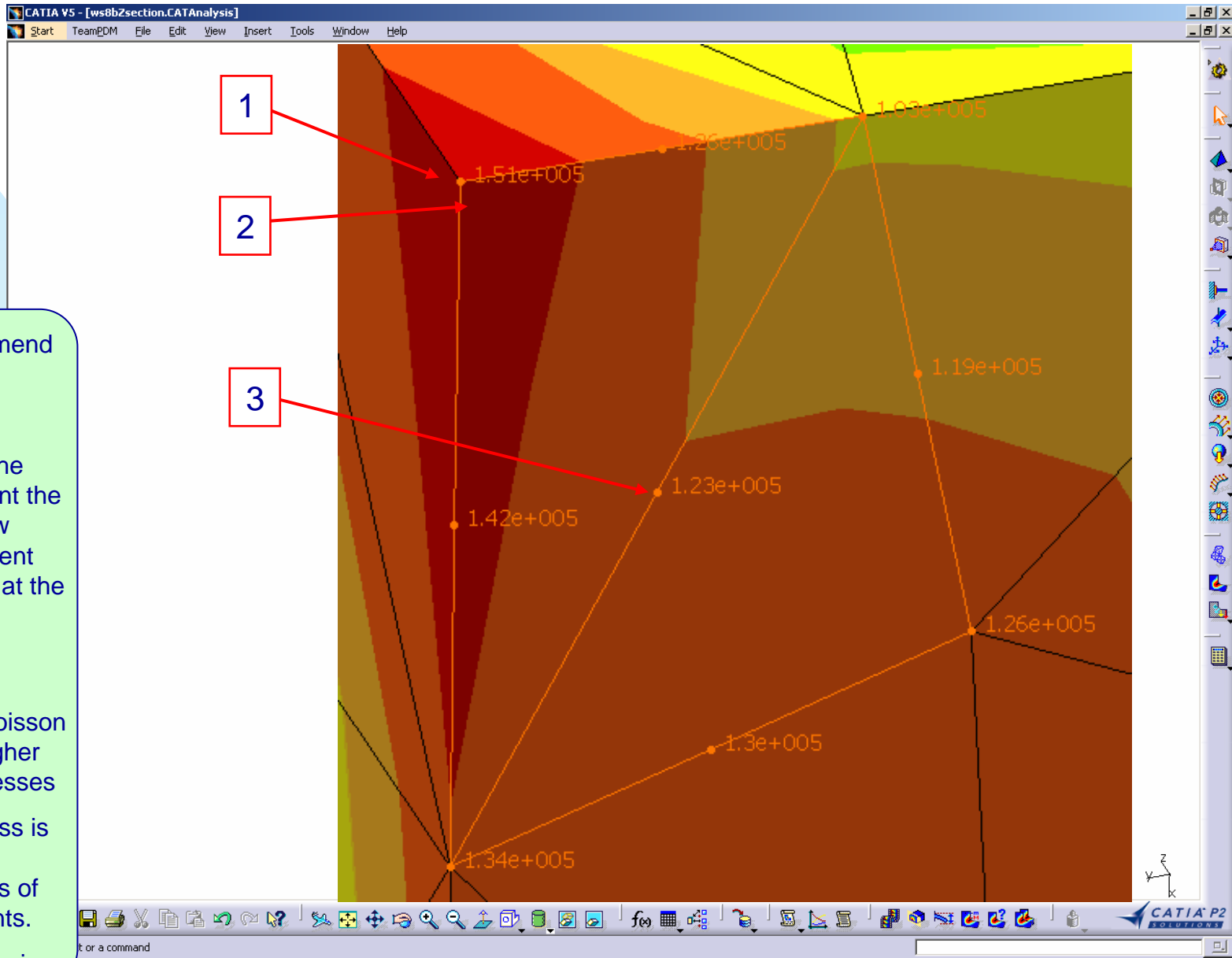
Steps:

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

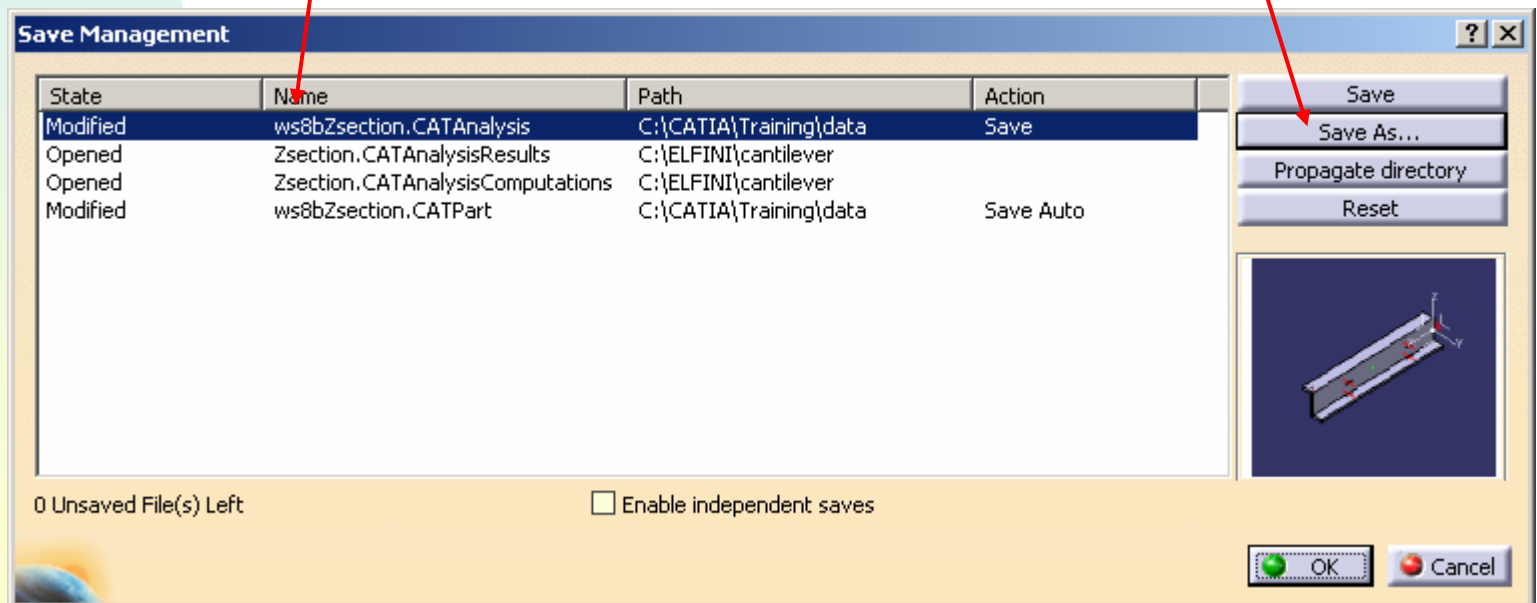
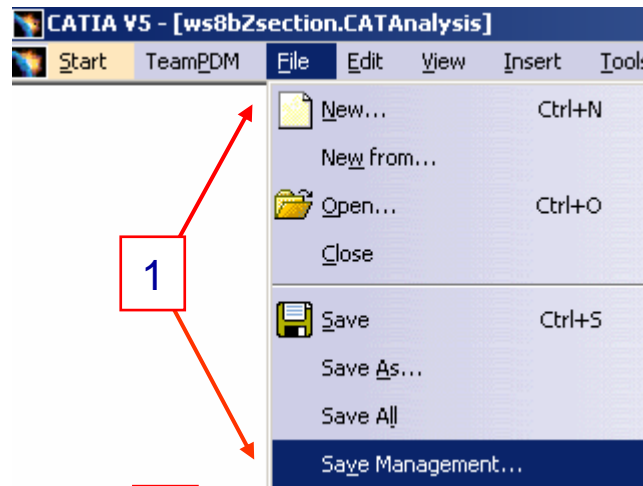
- ◆ CATIA V5 GSA workbench is validated for a Z-section cantilever beam scenario. To be conservative, increase material strength to a minimum yield of 134000 psi for the described load case.

	Hand Calculations	.1 inch Parabolic Global Mesh, .01 inch sag
Global % Precision error	NA	2.38 %
Local % Precision error	NA	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

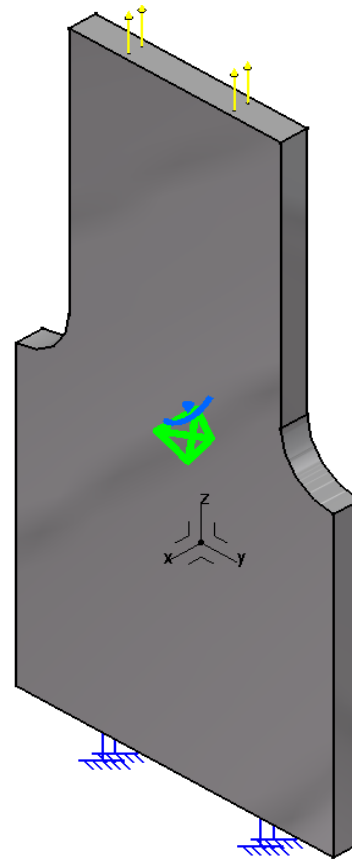
Steps:

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



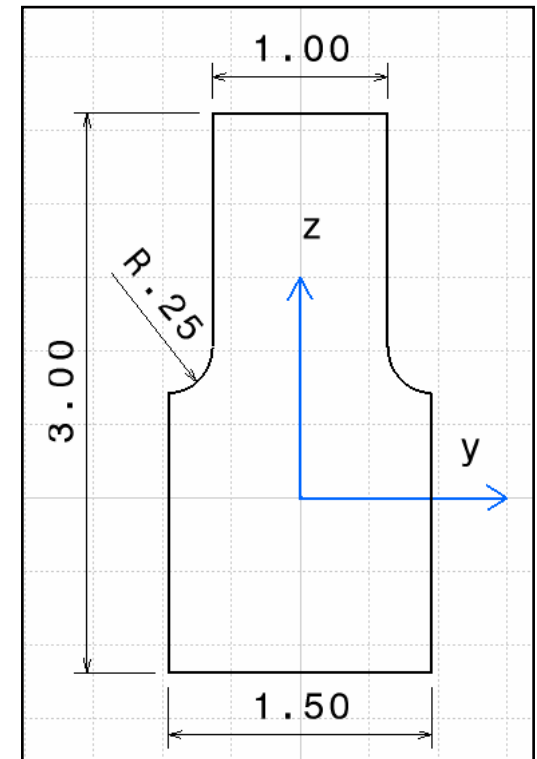
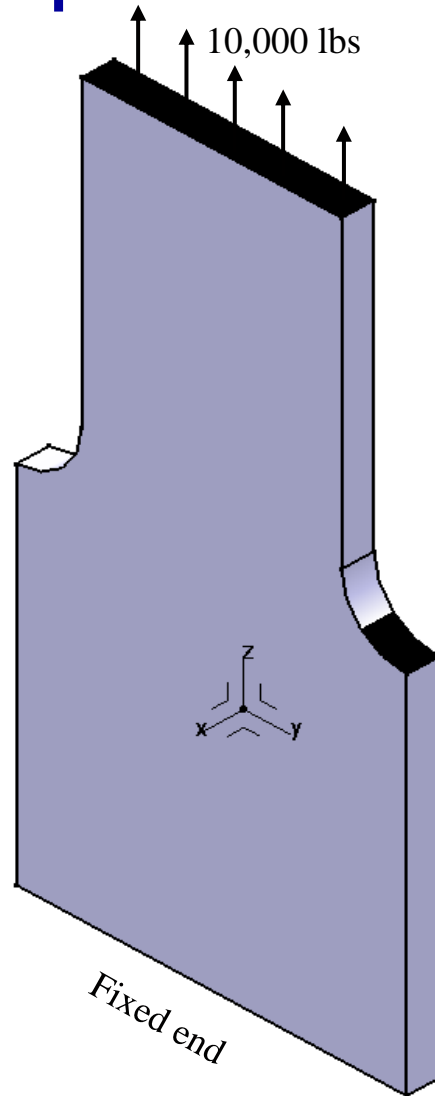
WORKSHOP 9

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

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

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

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

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

■ 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

The screenshot shows the CATIA V5 interface with the Analysis Manager tree on the left and a 3D model of a part on the right. The tree structure is as follows:

- Analysis Manager
 - Links Manager
 - Results -> C:\ELFINI\concentration\stepped.CATAnalysisResults
 - Computations -> C:\ELFINI\concentration\stepped.CATAnalysisComputations
 - Link.1 -> C:\CATIA\Training\data\ws9stepped.CATPart
 - ws9stepped
 - xy plane
 - yz plane
 - zx plane
 - Axis Systems
 - PartBody
 - Pad.1
 - Steel
 - Finite Element Model
 - Nodes and Elements
 - Properties.1
 - Material Property3D.1
 - Static Case

Red boxes with numbers 1, 2, 3, and 4 point to the following elements:

- 1: Link.1 in the Links Manager tree.
- 2: Material Property3D.1 in the Properties.1 folder.
- 3: The 3D model of the part.
- 4: The Results folder in the Links Manager tree.

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

The screenshot shows the CATIA V5 software interface. On the left, the Analysis Manager tree is expanded to show a 'Static Case' with a 'Clamp.1' restraint applied. In the center, a 3D model of a mechanical part is shown with a red hatched area indicating the selected face for the clamp. On the right, a 'Clamp' dialog box is open, showing the name 'Clamp.1' and 'Supports: 1 Face'. A red box labeled '1' points to the Clamp Restraint icon in the toolbar, and another red box labeled '2' points to the selected face on the 3D model.

Steps:

1. Select the Clamp Restraint icon.
2. Select the face as shown, select OK.

Step 4. Apply a pressure force

The screenshot shows the CATIA V5 interface with the following elements:

- Tree View (Left):** Analysis Manager, Links Manager, Finite Element Model, Nodes and Elements, Properties.1, Static Case, Restraints.1 (Clamp.1), Loads.1 (Distributed Force.1), Static Case Solution.1, Sensors.1.
- 3D Model (Center):** A grey mechanical part with a coordinate system (x, y, z). Red arrows on the top face indicate the direction of the pressure force. A green arrow points to the bottom face, which is supported by red hatching.
- Pressure Dialog Box (Right):** A dialog box titled 'Pressure' with the following fields:
 - Name: Pressure.1
 - Supports: 1 Face
 - Pressure: -80000psi
 - Optional Elements: (empty)
 - Data Mapping:
 - Buttons: OK, Cancel
- Annotations:** Red boxes labeled 1, 2, and 3 with arrows pointing to the toolbar icon, the top face, and the dialog box respectively.

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.

Click or a command

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.

1e+001 s of CPU
3.98e+003 kilo-bytes of memory
1.59e+004 kilo-bytes of disk
Intel MKL(c) Library found: Intel(R) MKL V5.1.0

Do you want to continue the computation?

Yes No

Step 6. Check global and local precision

Analysis Manager

- Links Manager
- Finite Element Model
 - Nodes and Elements
 - Properties.1
 - Static Case
 - Restraints.1
 - Loads.1
 - Static Case Solution.1
 - Deformed Mesh
 - Sensors.1

Animate Window

7

Steps Number 10

Speed

Close

Right Side View

ISO View

1

2

CATIA V5 - [ws9stepped.CATAnalysis]

Start TeamPDM File Edit View Insert Tools Window Help

Project or a command

CATIA P2 SOLUTIONS

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

Analysis Manager

- Links Manager
- Finite Element Model
 - Nodes and Elements
 - Properties.1
 - Static Case
 - Restraints.1
 - Loads.1
 - Static Case Solution.1
 - Deformed Mesh
 - Estimated local error
 - Sensors.1

Precision Location : Global
Estimated Precision : 0.00126126 Btu
Strain Energy : 3.95554 Btu
Global Estimated Error Rate : 1.26256 %

Estimated local error

Btu

- 2.23e-008
- 2.01e-008
- 1.78e-008
- 1.56e-008
- 1.34e-008
- 1.12e-008
- 8.92e-009
- 6.69e-009
- 4.46e-009
- 2.23e-009
- 2.46e-015

1

2

3

CATIA V5 - [ws9stepped.CATAnalysis]

Start TeamPDM File Edit View Insert Tools Window Help

Project or a command

CATIA V5 SOLUTIONS

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%).

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.

The screenshot displays the CATIA V5 interface for a finite element analysis. The left-hand tree view shows the following structure:

- Analysis Manager
 - Links Manager
 - Finite Element Model
 - Nodes and Elements
 - Properties.1
 - Static Case
 - Restraints.1
 - Loads.1
 - Static Case Solution.1
 - Deformed Mesh
 - Estimated local error
 - Extrema
 - Global Maximum.1
 - Sensors.1

The main view shows a 3D model of a mechanical part with a finite element mesh. A red triangle highlights the element with the highest estimated error. A text box indicates the estimated local error for the Global Maximum.1 is $2.2304e-008$ Btu. A 'Focus On' button is visible below the tree view. A 'Extrema Creation' dialog box is open on the right, showing settings for Global and Local extrema. A 'Search Image Extrema' icon is highlighted in the bottom toolbar.

Extrema Creation

<input checked="" type="checkbox"/> Global	
Minimum extrema at most	0
Maximum extrema at most	2
<input type="checkbox"/> Local	
Minimum extrema at most	0
Maximum extrema at most	2

Buttons: OK, Cancel

Step 6. Check global and local precision

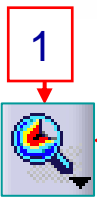
The screenshot shows the CATIA V5 interface with a finite element model of a mechanical part. The model is colored by stress, with a red box highlighting a specific region. The left-hand side shows the Analysis Manager tree with the following structure:

- Analysis Manager
 - Links Manager
 - Finite Element Model
 - Nodes and Elements
 - Properties.1
 - Static Case
 - Restrains.1
 - Loads.1
 - Static Case Solution.1
 - Deformed Mes...
 - Estimated local error
 - Extrema
 - Global Maximum.1
 - Sensors.1
 - Adaptivity Process
 - Adaptivity Convergence.1
 - Adaptivities.1
 - Adaptivity Box.1

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%.

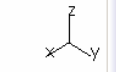


Adaptivity Box

Name	Adaptivity Box.1
Objective Error (%)	5
Solution	Static Case Solution.1
Local Error (%)	3,565

Select Extremum

OK Cancel



Step 6. Check global and local precision

Adaptivity Box

Name: Adaptivity Box.2

Objective Error (%): 5

Solution: Static Case Solution.1

Local Error (%): 2.1772

Select Extremum

OK Cancel

1

2

3

Analysis Manager

Links Manager

Finite Element Model

Nodes and Elements

Properties.1

Static Case

Restrains.1

Loads.1

Static Case Solution.1

Deformed Mesh

Estimated local error

Extrema

Global Maximum.1

Sensors.1

Adaptivity Process

Adaptivity Convergence.1

Adaptivities.1

Adaptivity Box.1

CATIA P2 SOLUTIONS

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%.

Step 7. Change mesh size and add local meshing

Analysis Manager

- Links Manager
- Finite Element Model
 - Nodes and Elements
 - OCTREE Tetrahedron Mesh.1 : 7
 - Properties.1
 - Static Case
 - Restraints.1
 - Loads.1
 - Static Case Solution.1
 - Sensors.1
- Adaptivity Process

OCTREE Tetrahedron Mesh

Global | Local

Size: 0.063in
Sag: 0.006in

Element type

Linear

Parabolic

OK Cancel

Local Mesh Size

Name: Local Mesh Size
Supports: 2 Faces
Value: 0.04in

OK Cancel

Local Mesh Sag

Name: Local Mesh Sag
Supports: 2 Faces
Value: 0.004in

OK Cancel

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.

Computation Resources Estimation	
4e+002 s of CPU	
2.3e+004 kilo-bytes of memory	
1.63e+005 kilo-bytes of disk	
Intel MKL(c) Library found: Intel(R) MKL V5.1.0	
Do you want to continue the computation?	
Yes	No

Step 8. Compute the precise analysis

Analysis Manager

- Links Manager
- Finite Element Model
 - Nodes and Elements
 - OCTREE Tetrahedron Mesh.1 : Zsection
 - Properties.1
 - Static Case
 - Restrains.1
 - Loads.1
 - Static Case Solution.1
 - Deformed Mesh
 - Estimated local error
 - Extrema
 - Sensors.1
 - Adaptivity Process

Estimated local error
Btu

7.04e-009
6.34e-009
5.63e-009
4.93e-009
4.22e-009
3.52e-009
2.82e-009
2.11e-009
1.41e-009
7.04e-010
3.41e-018

Precision Location : Global
Estimated Precision : 0.000287749 Btu
Strain Energy : 3.95692 Btu
Global Estimated Error Rate : 0.602983 %

1
2
3

Image Activate/DeActivate

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

The screenshot shows the CATIA V5 interface with the following components:

- Analysis Manager Tree:**
 - Analysis Manager
 - Links Manager
 - Finite Element Model
 - Nodes and Elements
 - OCTREE Tetrahedron Mesh.1 : Zsection
 - Properties.1
 - Static Case
 - Restrains.1
 - Loads.1
 - Static Case Solution.1
 - Deformed Mesh
 - Estimated local error (1) (with Local Update button)
 - Extrema (2) (with Global Maximum.1)
 - Sensors.1
 - Adaptivity Process (3)

- Adaptivity Box Dialog:**
- Name: Adaptivity Box.2
- Objective Error (%): 5
- Solution: Static Case Solution.1
- Local Error (%): 0,4722
- Buttons: Select Extremum (3), OK, Cancel
- Mesh Visualization:** A blue tetrahedral mesh with a red wireframe bounding box around a portion of it.

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

CATIA V5 - [ws9stepped.CATAnalysis]

Start TeamPDM File Edit View Insert Tools Window Help

Analysis Manager

- Links Manager
- Finite Element Model
 - Nodes and Elements
 - Properties.1
 - Static Case
 - Restraints.1
 - Loads.1
 - Static Case Solution.1
 - Deformed Mesh
 - Estimated local error
 - Translational displacement vector
 - Sensors.1
 - Adaptivity Process
 - Hide/Show

Translational displacement vector

in

0.00702

0.00632

0.00561

0.00491

0.00421

0.00351

0.00281

0.00211

0.0014

0.000702

0

On Boundary

1

2

CATIA V5 P2 SOLUTIONS

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

Analysis Manager

- Links Manager
- Finite Element Model
 - Nodes and Elements
 - Properties.1
 - Static Case
 - Restraints.1
 - Loads.1
 - Static Case Solution.1
 - Deformed Mesh
 - Estimated local error
 - Translational displacement vector
 - Von Mises Stress (nodal value)
 - Sensors.1

Adaptivity Process

Von Mises Stress (nodal value)
psi

1.39e+005
1.25e+005
1.12e+005
9.77e+004
8.37e+004
6.98e+004
5.59e+004
4.2e+004
2.81e+004
1.41e+004
231

On Boundary

1

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.

Von Mises Stress (nodal value) Global Maximum.1: 139407 psi

Analysis Manager

- Links Manager
- Finite Element Model
 - Nodes and Elements
 - Properties.1
 - Static Case
 - Restraints.1
 - Loads.1
 - Static Case Solution.1
 - Deformed Mesh
 - Estimated local error
 - Translational displacement vector
 - Von Mises Stress (nodal value)
 - Extrema
 - Global Maximum.1
 - Sensors.1
 - Adaptivity Process

Extrema Creation

- Global
 - Minimum extrema at most 0
 - Maximum extrema at most 2
- Local
 - Minimum extrema at most 0
 - Maximum extrema at most 2

OK Cancel

1a

1b

2

t or a command

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.

The screenshot displays the CATIA V5 software interface for visualizing simulation results. The Analysis Manager tree on the left shows the following structure:

- Analysis Manager
 - Links Manager
 - Finite Element Model
 - Nodes and Elements
 - Properties.1
 - Static Case
 - Restraints.1
 - Loads.1
 - Static Case Solution
 - Deformed Mesh
 - Estimated local error
 - Translational displacement vector
 - Von Mises Stress (nodal value)
 - Extrema
 - Sensors.1
 - Adaptivity Process

Key dialog boxes and their settings are shown:

- Image Edition (2b):** The 'Visu' tab is selected. The 'AVERAGE-ISO' filter is chosen from the list. The 'Edition' buttons are 'Iso/Fringe', 'Symbol', and 'Axis System'.
- Image Edition (3):** The 'Filters' tab is selected. The 'Position' is set to 'Node'. The 'Edition' buttons are 'Iso/Fringe', 'Symbol', and 'Axis System'.
- Image Iso Fringe Editor (4):** The 'ISO smooth' checkbox is unchecked.

The 3D model on the right shows a stress distribution on a part, with a color scale from blue (low stress) to red (high stress). The 'Extrema' feature in the tree is highlighted with a red box labeled '1' and a 'Hide/Show' button. Red arrows labeled '2a' and '2b' point to the 'Von Mises Stress (nodal value)' and 'Image Edition (2b)' respectively. A red arrow labeled '3' points to the 'Image Edition (3)' dialog box. A red arrow labeled '4' points to the 'ISO smooth' checkbox in the 'Image Iso Fringe Editor' dialog box.

Step 9. Visualize final results

■ Conclusions

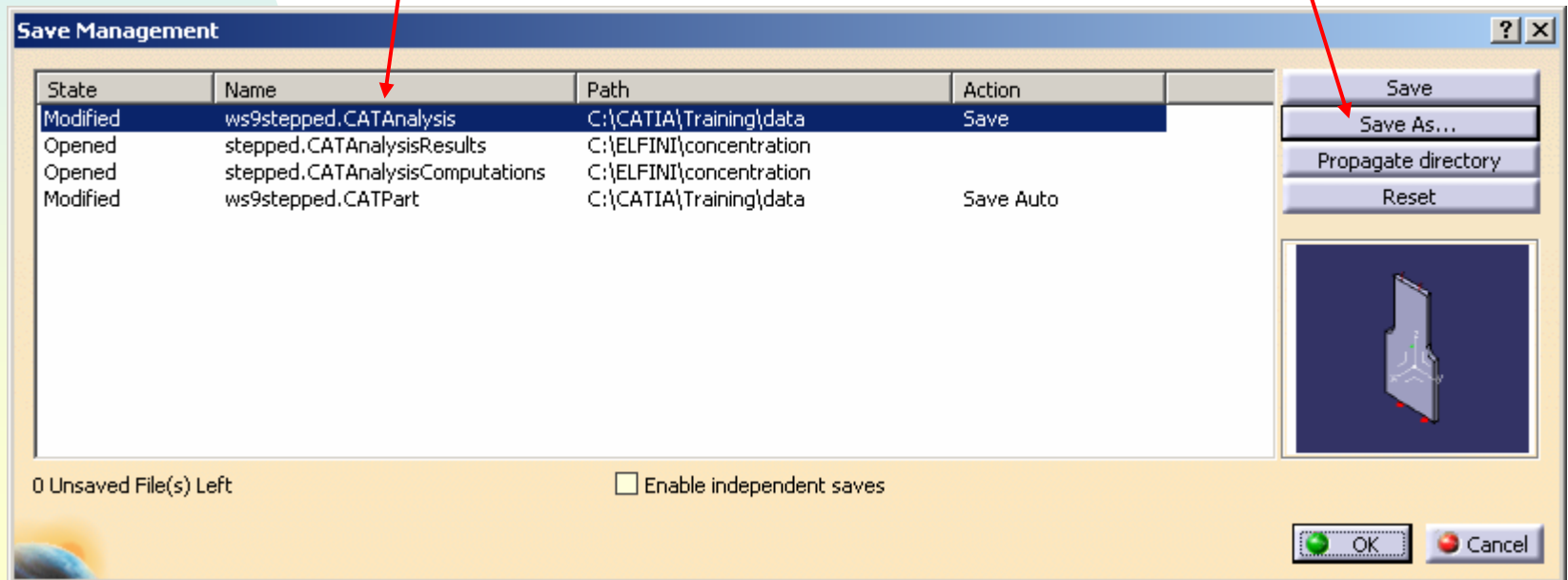
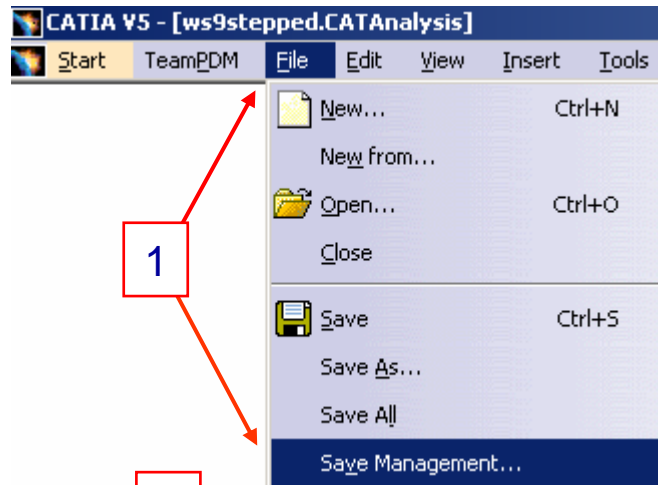
- ◆ CATIA V5 GSA workbench is validated for a stepped flat tension bar with shoulder fillets scenario.

	Hand Calculations	.1 inch Parabolic Global Mesh, .01 inch sag
Global % Precision error	NA	0.6 %
Local % Precision error	NA	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

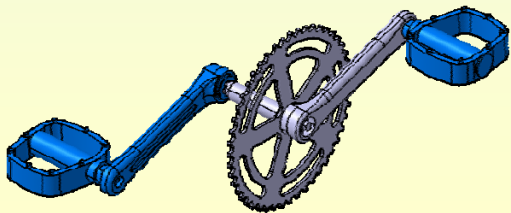
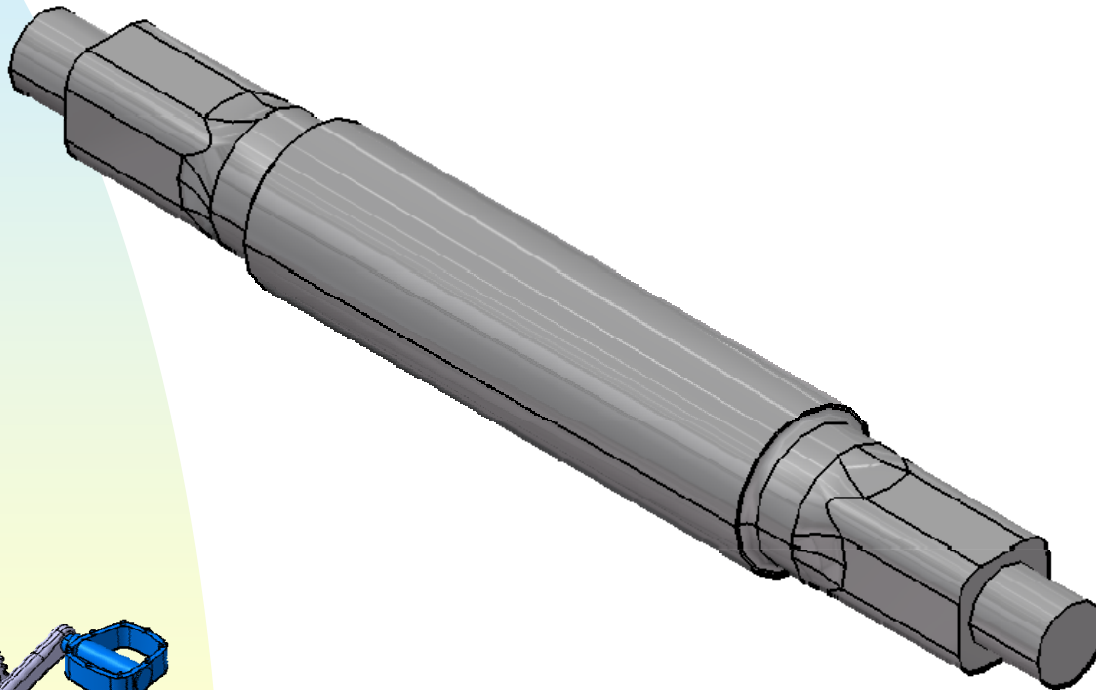
Steps:

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



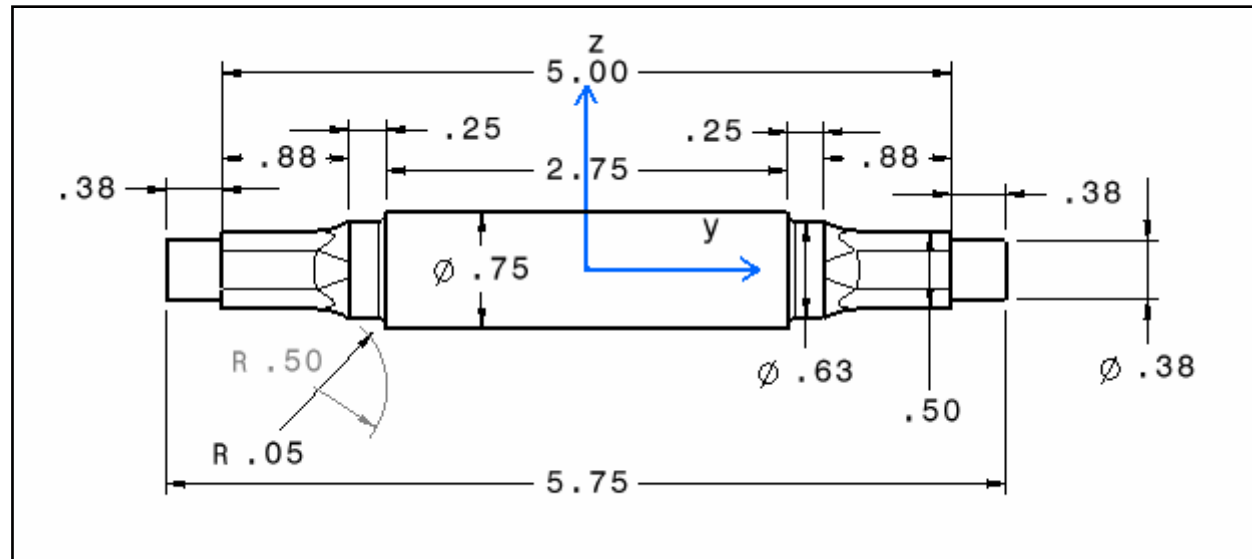
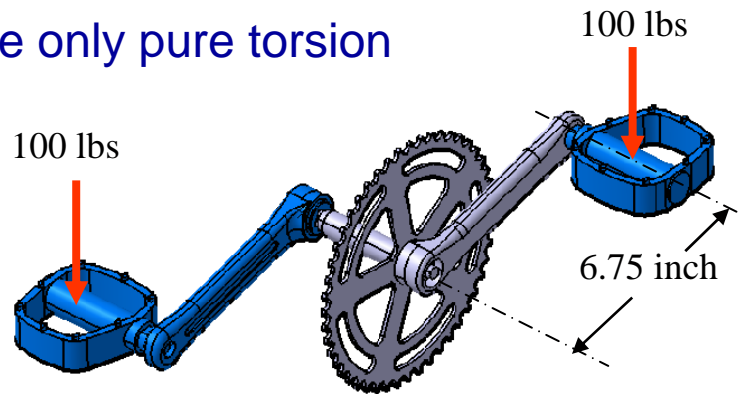
WORKSHOP 9b

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

$$\begin{aligned}\text{Maximum Shear Stress} = \tau_{max} &= \tau_{nom} \cdot K_{ts} & \tau_{nom} &= \frac{16 \cdot T}{\pi \cdot d^3} = \frac{16 \cdot 1350 \text{ lb-inch}}{3.14 \cdot 0.63^3} = 27502 \text{ psi} \\ K_{ts} &= 1.42 & \tau_{max} &= 27502 \text{ psi} \cdot 1.42 = 39053 \text{ psi}\end{aligned}$$

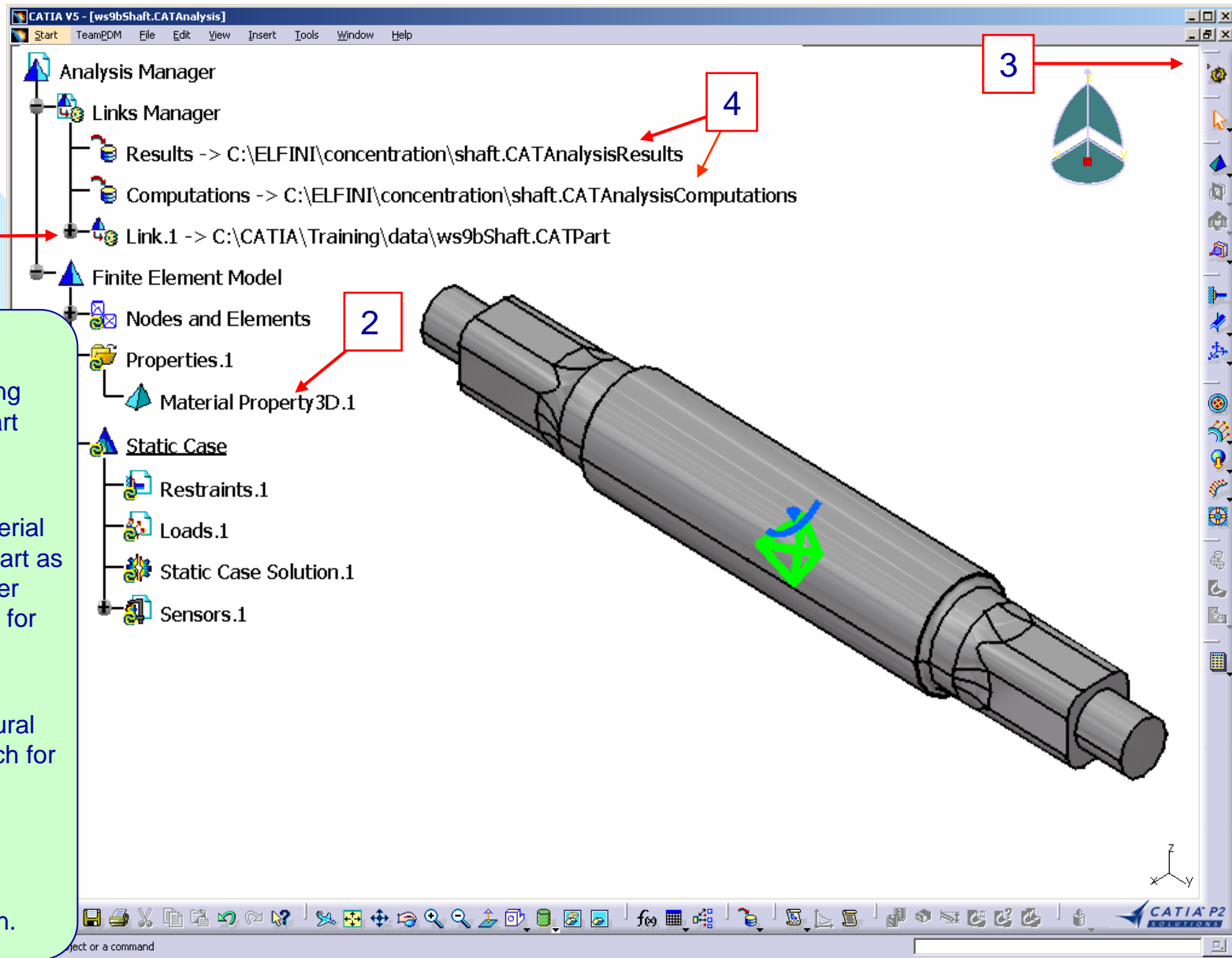
◆ Maximum angle of twist

$$\begin{aligned}\text{Angle of twist (radians)} = \theta &= \frac{T \cdot l}{J \cdot G} & J &= 2 \cdot I = 2 \cdot \frac{\pi}{4} \cdot R^4 = 2 \cdot \frac{3.14}{4} \cdot 0.375^4 \text{ inch} = 0.031 \text{ inch}^4 \\ \theta &= \frac{1350 \text{ lb-inch} \cdot 5.01 \text{ inch}}{0.031 \text{ inch}^4 \cdot 12 \times 10^6 \text{ psi}} = 0.0182 \text{ radians (1.04 degrees)}\end{aligned}$$

■ 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

CATIA V5 - [ws9bShaft.CATAnalysis]

Start TeamPDM File Edit View Insert Tools Window Help

Analysis Manager

- Links Manager
- Finite Element Model
 - Nodes and Elements
 - OCTREE Tetrahedron Mesh.1 : shaft
 - Properties.1
 - Material Property3D.1
 - Static Case
 - Restraints.1
 - Loads.1
 - Static Case Solution.1
 - Sensors.1

OCTREE Tetrah...

Global Local

Size 0.125in

Sag 0.013in

Element type

Linear

Parabolic

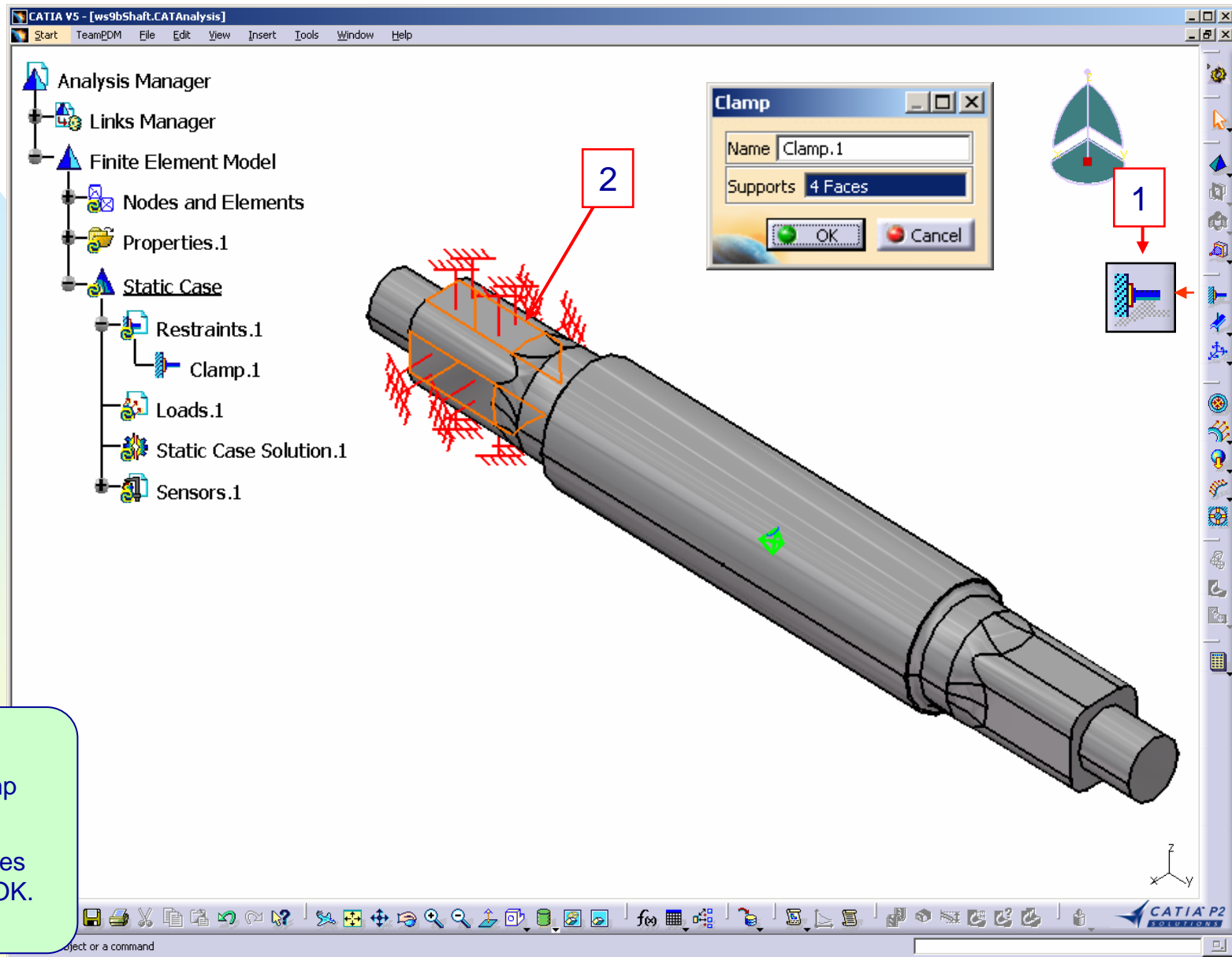
OK Cancel

Object or a command

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

CATIA V5 - [ws9bShaft.CATAnalysis]

Start TeamPDM File Edit View Insert Tools Window Help

Analysis Manager

- Links Manager
- Finite Element Model
 - Nodes and Elements
 - OCTREE Tetrahedron Mesh.1 : shaft
 - Properties.1
 - Material Property3D.1
 - Static Case
 - Restraints.1
 - Loads.1
 - Moment.1
 - Static Case Solution.1
 - Sensors.1

Moment

Name: Moment.1

Supports: 4 Faces

Axis System

Type: Global

Display locally

Moment Vector

Norm: 1350lbf·in

X: 0lbf·in

Y: 1350lbf·in

Z: 0lbf·in

OK Cancel

1

2

3

Load only one end.

Steps:

1. Select the Moment icon.
2. Select four faces as shown.
3. Enter 1350lbf·in in the positive y-direction.

Step 5. Compute the initial analysis

The screenshot shows the CATIA V5 interface for a finite element analysis of a shaft. The left-hand 'Analysis Manager' tree is expanded to show the 'Static Case' folder, which contains sub-items for 'Restraints.1', 'Loads.1', 'Static Case Solution.1', and 'Sensors.1'. The main 3D view displays a grey shaft model with blue arrows representing restraints and yellow arrows representing loads. A 'Computation Resources Estimation' dialog box is open in the foreground, showing the following details:

Resource	Value
CPU	4 s
Memory	1.81e+003 kilo-bytes
Disk	5.6e+003 kilo-bytes
Library	Intel MKL(c) Library found: Intel(R) MKL V5.1.0

The dialog box asks, 'Do you want to continue the computation?' with 'Yes' and 'No' buttons. A red box with the number '1' and an arrow points to the 'Compute All' button in the software's main toolbar.

Save first.

Steps:

1. Compute All

Step 6. Check global and local precision

Analysis Manager

- Links Manager
- Finite Element Model
 - Nodes and Elements
 - Properties.1
 - Static Case
 - Restraints.1
 - Loads.1
 - Static Case Solution.1
 - Deformed Mesh
 - Sensors.1

Animate Window

7

Steps Number 10

Speed

Close

1

2

Visualize the **Deformation** and animate.

Steps:

1. Select the Deformation icon.
2. Select on the Animate icon.

Verify that deformation is what you expect.

Step 6. Check global and local precision

Analysis Manager

- Links Manager
- Finite Element Model
 - Nodes and Elements
 - Properties.1
 - Static Case
 - Restraints.1
 - Loads.1
 - Static Case Solution.1
 - Deformed Mesh
 - Estimated local error
 - Sensors.1

Estimated local error

Btu

- 1.53e-006
- 1.38e-006
- 1.22e-006
- 1.07e-006
- 9.17e-007
- 7.64e-007
- 6.12e-007
- 4.59e-007
- 3.06e-007
- 1.53e-007
- 2.94e-019

Precision Location : Global
Estimated Precision : 0.473102 Btu
Strain Energy : 1.51133 Btu
Global Estimated Error Rate : 36.788 %

3

1

2

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%).

Step 6. Check global and local precision

Analysis Manager

- Links Manager
- Finite Element Model
 - Nodes and Elements
 - Properties.1
 - Static Case
 - Restraints.1
 - Loads.1
 - Static Case Solution.1
 - Deformed Mesh
 - Estimated local error
 - Extrema
 - Global Maximum.1
 - Sensors.1

Estimated local error Global Maximum.1: 1.52898e-006 Btu

2

Focus On

1

Select an object or a command

Find the global element with the highest estimated error.

Steps:

1. Select the Search Image Extrema icon.
2. Focus On the worst element.

Step 7. Change global and local mesh size

The screenshot displays the CATIA V5 interface for a finite element analysis of a shaft. The Analysis Manager tree on the left shows the following structure:

- Analysis Manager
 - Links Manager
 - Finite Element Model
 - Nodes and Elements
 - OCTREE Tetrahedron Mesh.1 : shaft
 - Local Mesh Size
 - Local Mesh Sag
 - Properties.1
 - Static Case
 - Restraints.1
 - Loads.1
 - Static Case Solution.1
 - Sensors.1
 - Adaptivity Process

The 3D model shows a shaft with a tetrahedral mesh. A red circle highlights a shoulder area, and a yellow circle highlights another shoulder area. Red arrows point from numbered boxes to the 'Global' tab of the 'OCTREE Tetrah...' dialog (labeled '1') and the 'Local Mesh Size' dialog (labeled '2').

The 'OCTREE Tetrah...' dialog has the following settings:

- Global | Local
- Size: 0.06in
- Sag: 0.006in
- Element type: Linear, Parabolic
- Buttons: OK, Cancel

The 'Local Mesh Size' dialog has the following settings:

- Name: Local Mesh Size
- Supports: 22 Faces
- Value: 0.03in
- Buttons: OK, Cancel

The 'Local Mesh Sag' dialog has the following settings:

- Name: Local Mesh Sag
- Supports: 22 Faces
- Value: 0.003in
- Buttons: OK, Cancel

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

CATIA V5 - [ws9bShaft.CATAnalysis]

Start TeamPDM File Edit View Insert Tools Window Help

Analysis Manager

- Links Manager
- Finite Element Model
 - Nodes and Elements
 - Properties.1
 - Static Case
 - Restraints.1
 - Loads.1
 - Static Case Solution.1
 - Sensors.1
- Adaptivity Process

Computation Resources Estimation

3e+002 s of CPU
2.16e+004 kilo-bytes of memory
1.21e+005 kilo-bytes of disk
Intel MKL(c) Library found: Intel(R) MKL V5.1.0

Do you want to continue the computation?

Yes No

1

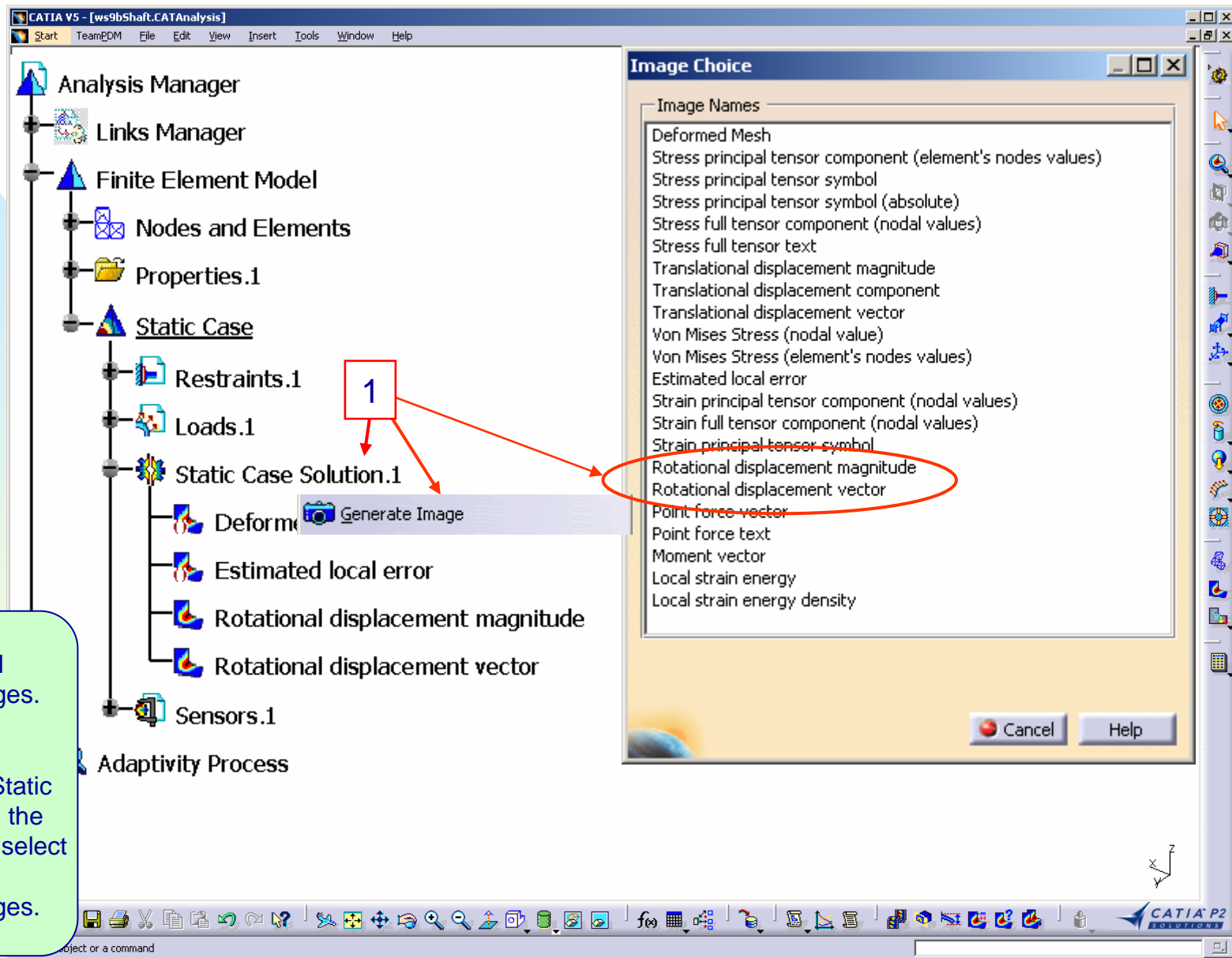
CATIA P2 SOLUTIONS

Save first.

Steps:

1. Compute All

Step 9. Visualize final results

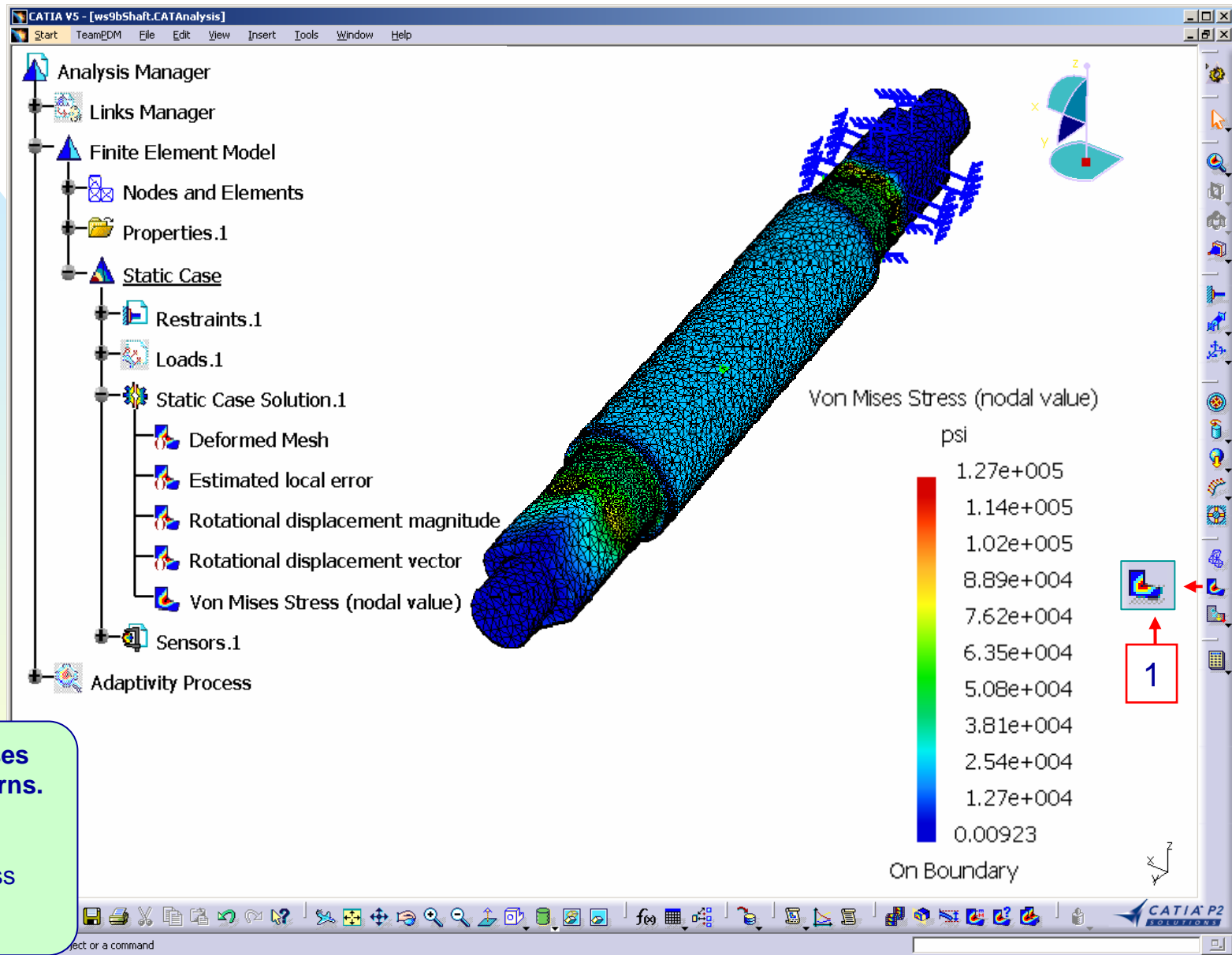


Add the Rotational displacement images.

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.

Step 9. Visualize final results

CATIA V5 - [ws9bShaft.CATAnalysis]

Static Case Solution.1

- Deformed Mesh
- Estimated local error
- Rotational displacement magnitude
- Rotational displacement vector
- Von Mises Stress (nodal value)
- Extrema
 - Global Maximum.1

Adaptivity Box

Name: Adaptivity Box.1

Objective Error (%): 10

Solution: Static Case Solution.1

Local Error (%): 32.096

Select Extremum

OK Cancel

Focus On

Von Mises Stress (nodal value) Global Maximum.1: 126962 psi

1

2

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

Static Case Solution.1

- Deformed Mesh
- Estimated local error
- Rotational displacement magnitude
- Rotational displacement vector
- Von Mises Stress (nodal value)
- Stress principal tensor component (nodal values)

Image Edition

Deformed

Display on Deformed Mesh

Visu Criteria Filters

DISCONTINUOUS-ISO
SYMBOL
AVERAGE-ISO
TEXT

Image Edition

Deformed

Display on Deformed Mesh

Visu Criteria Filters

Position : Node

Component : C1

Lamina : 1 [Min-Max]

Sup-Mid-Inf :

Repeat :

Edition

Iso/Fringe Symbol Axis System

OK Cancel Help

1

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

Static Case Solution.1

- Deformed Mesh
- Estimated local error
- Rotational displacement magnitude
- Rotational displacement vector
- Von Mises Stress (nodal value)
- Stress principal tensor component (nodal values)

Stress principal tensor component (nodal values)

MAX: 88270.2 psi
min: -2678.25 psi

3.9e+004

3.48e+004

3.07e+004

2.65e+004

2.23e+004

1.82e+004

1.4e+004

9.83e+003

5.66e+003

1.49e+003

-2.68e+003

On Boundary

Color Map Editor

On Boundary

Number of Colors 10

Imposed Max 39000

Imposed Min -2678.25

More >>

OK Apply Cancel

1

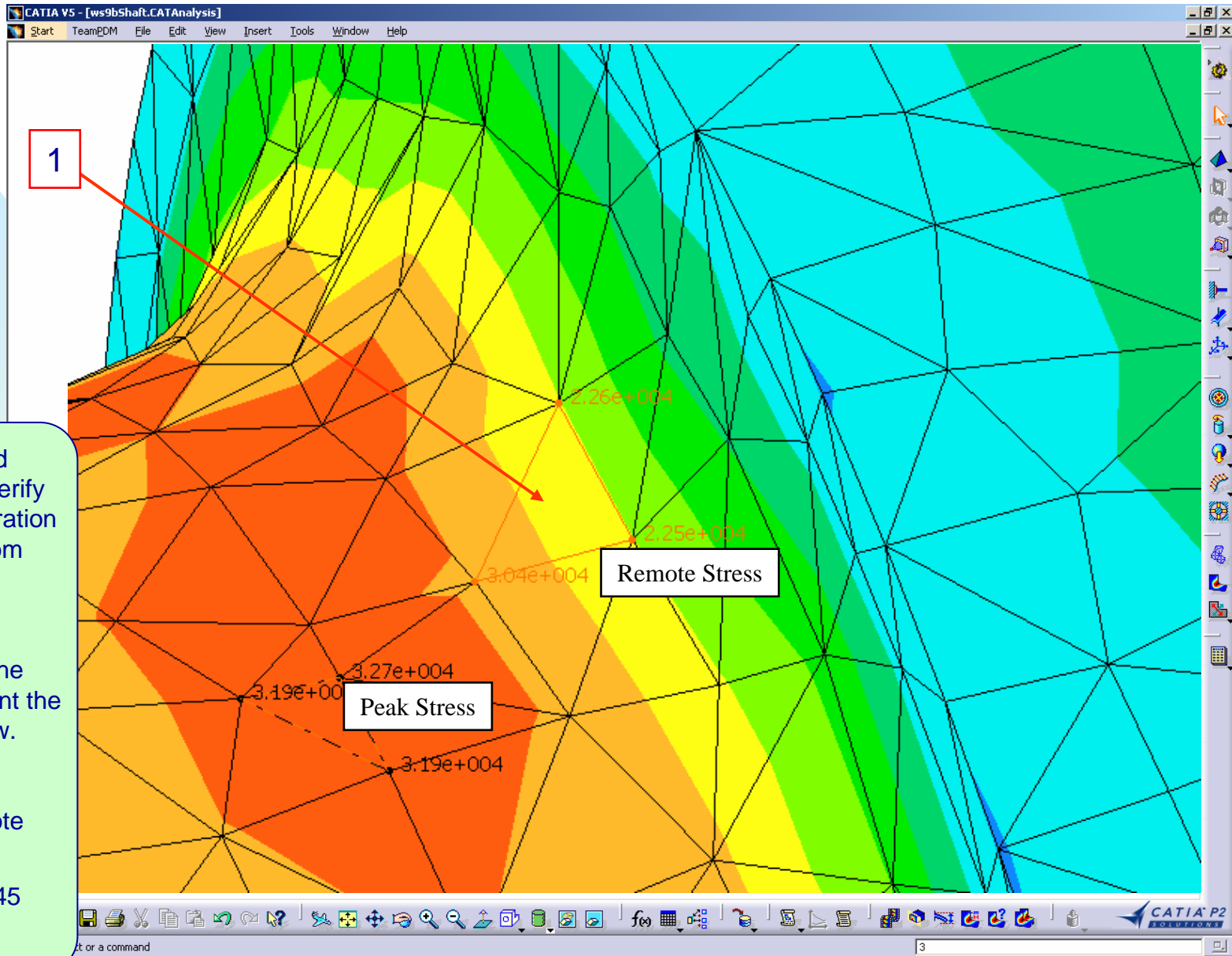
Hand calculations indicate max shear of 39,000 psi at the shoulder, so...

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.

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

CATIA V5 - [ws9bShaft.CATAnalysis]

Start TeamPDM File Edit View Insert Tools Window Help

2

Stress principal tensor component (nodal values) Global Maximum, 1: 88270.2 psi

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.

Static Case Solution.1

- Deformed Mesh
- Estimated local error
- Rotational displacement magnitude
- Rotational displacement vector
- Von Mises Stress (nodal value)
- Stress principal tensor component (nodal values)
- Extrema
 - Global Maximum.1

1

Select an object or a command

CATIA P2 SOLUTIONS

Step 9. Visualize final results

■ Conclusions

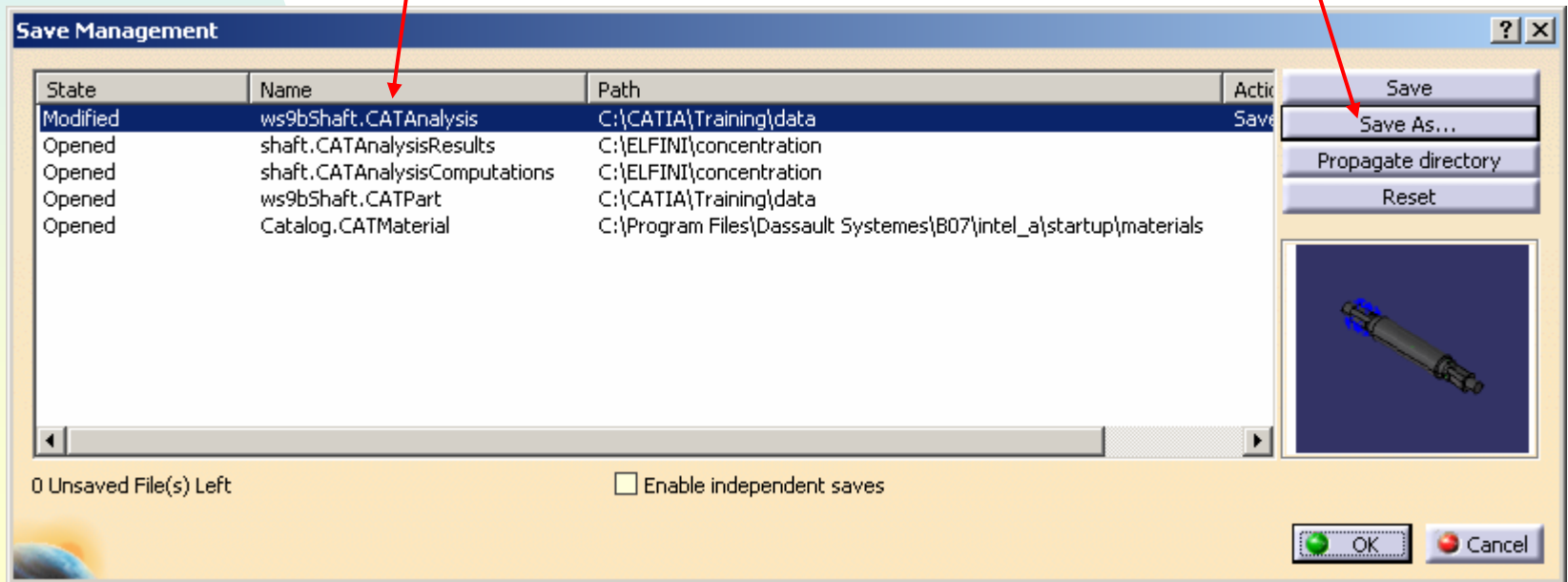
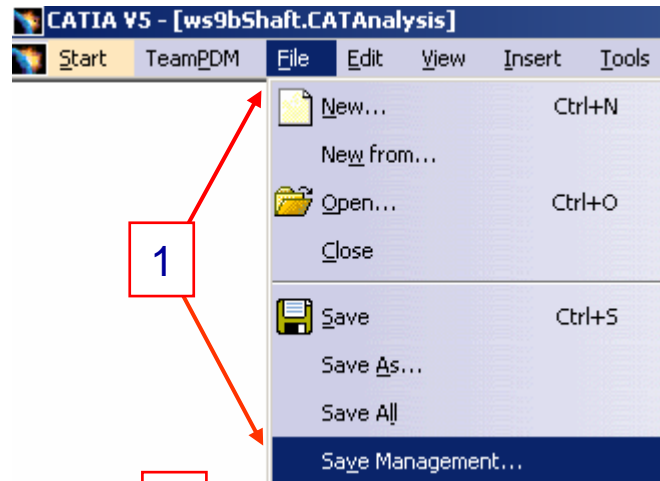
- ◆ CATIA V5 GSA workbench is validated for a shaft with a shouldered fillet scenario.

	Hand Calculations	.06 inch Linear Global Mesh, .006 inch sag .03 inch Linear Local Mesh, .003 inch sag
Global % Precision error	NA	19 %
Local % Precision error	NA	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

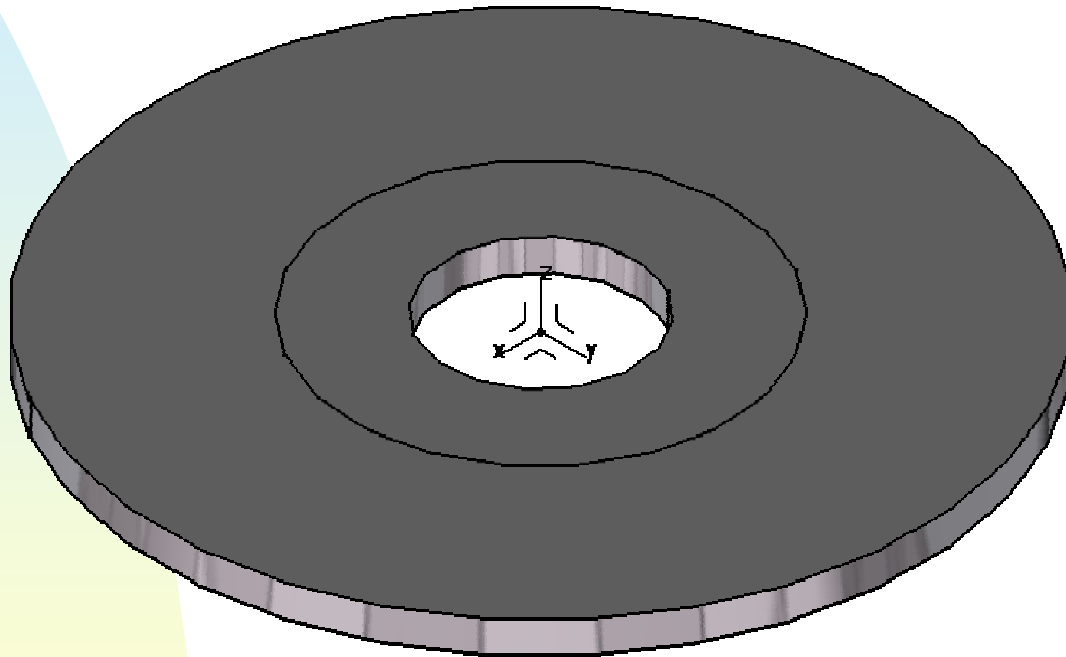
Steps:

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.



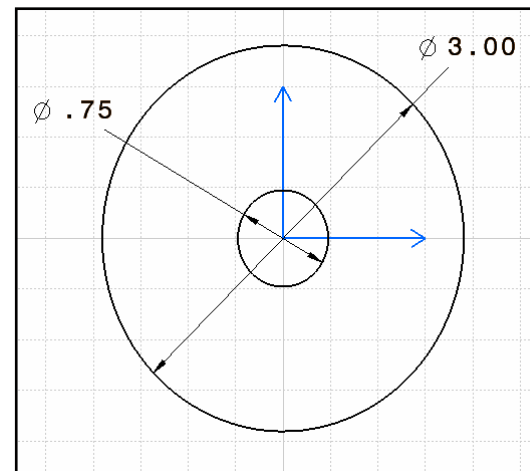
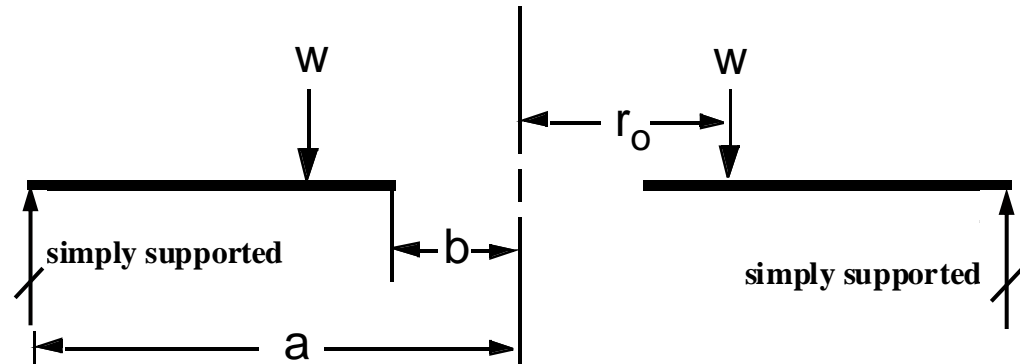
WORKSHOP 10

ANNULAR PLATE



■ 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

■ Hand calculations

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

◆ Plate constant:
$$D = \frac{Et^3}{12(1-\nu^2)}$$

◆ Plate constants dependent on the ratio a/b:

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

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

◆ Loading constants dependent upon the ratio a/r_o:

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

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

■ Hand calculations (cont.)

- ◆ Plate constant:

$$D = 1788.576$$

- ◆ Plate constants dependent on the ratio a/b:

$$C_1 = 0.8815 \quad C_7 = 1.7062$$

- ◆ Loading constants dependent upon the ratio a/r_o:

$$L_3 = 0.0582 \quad L_9 = 0.3346$$

- ◆ Maximum vertical displacement:

$$y = -0.018$$

- ◆ Maximum bending stress:

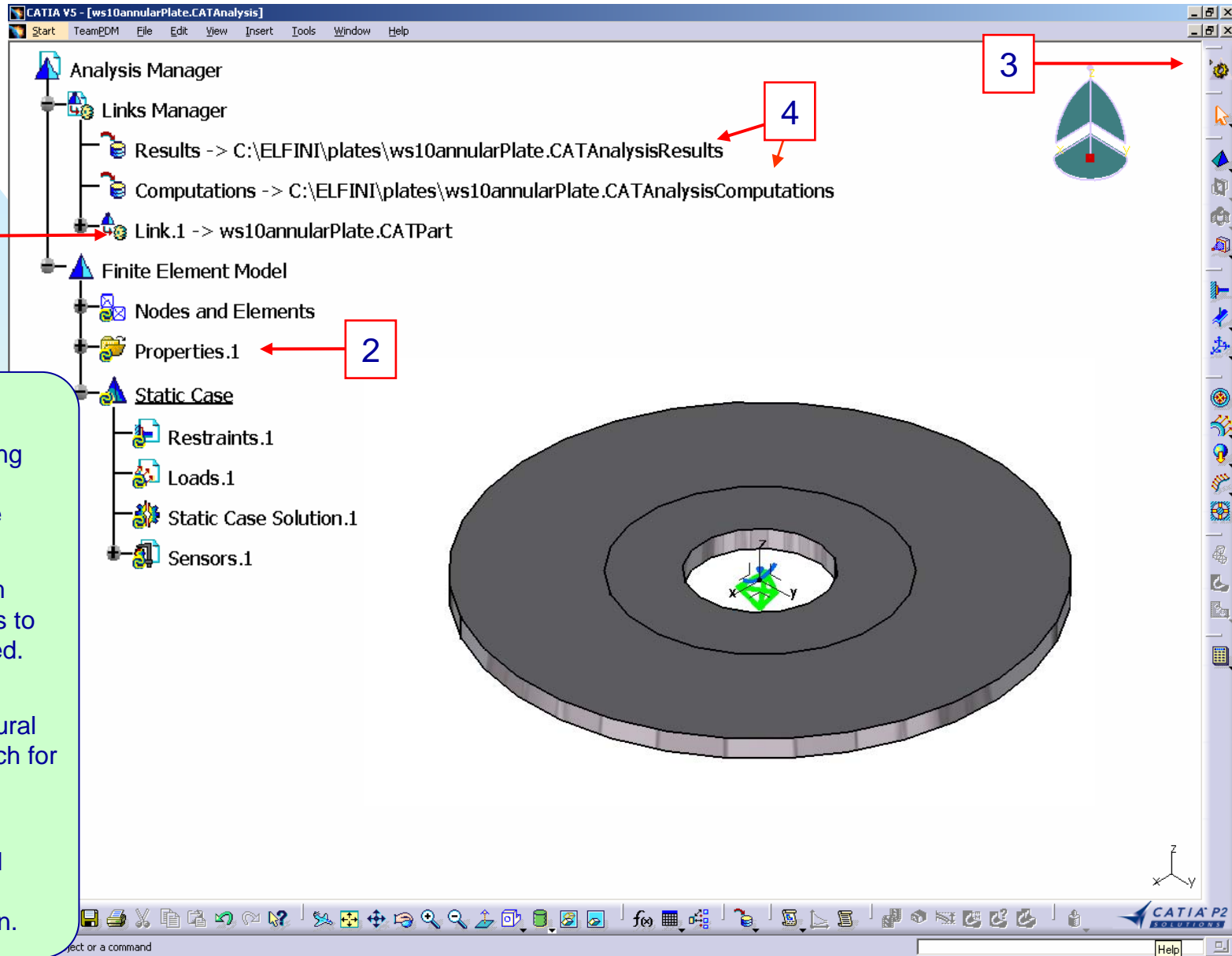
$$\text{Maximum bending stress} = \sigma = \frac{6 \cdot M}{t^2} \quad M = K_m \cdot w \cdot a = 0.5501 \cdot 85 \text{ lb/in} \cdot 1.5 \text{ inch} = 70.14 \text{ lbs}$$

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

■ 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.

The screenshot displays the CATIA V5 software interface. The Analysis Manager tree on the left shows the following structure:

- Analysis Manager
 - Links Manager
 - Finite Element Model
 - Nodes and Elements
 - OCTREE Tetrahedron Mesh.1 : ws10annularPlate
 - Properties.1
 - Static Case
 - Restraints.1
 - Loads.1
 - Static Case Solution.1
 - Sensors.1

The 'OCTREE Tetrahedron Mesh.1 : ws10annularPlate' item is highlighted with a red box containing the number '1'. A red arrow points from this box to the meshing icon in the tree. Another red box containing the number '2' points to the 'OCTREE Tetrah...' dialog box, which is open and shows the following settings:

- Global | Local
- Size: 0.125in
- Sag: 0.013in
- Element type:
 - Linear
 - Parabolic
- OK | Cancel

The 3D model of the annular plate is shown in the center, with a red arrow pointing to the meshing icon in the tree. The software interface includes a menu bar, toolbars, and a status bar.

Step 3. Apply an advanced restraint

Advanced Restraint

Name: Restraint.1

Supports: 1 Edge

Axis System

Type: Global

Display locally

Restrain Translation 1

Restrain Translation 2

Restrain Translation 3

Restrain Rotation 1

Restrain Rotation 2

Restrain Rotation 3

OK Cancel

Analysis Manager

- Links Manager
- Finite Element Model
 - Nodes and Elements
 - Properties.1
 - Static Case
 - Restraints.1
 - Restraint.1
 - Loads.1
 - Static Case Solution.1
 - Sensors.1

1

2

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

The screenshot shows the CATIA V5 interface for a finite element analysis. The tree view on the left shows the following structure:

- Analysis Manager
 - Links Manager
 - Finite Element Model
 - Nodes and Elements
 - Properties.1
 - Static Case
 - Restraints.1
 - Restraint.1
 - Isostatic.1
 - Loads.1
 - Line Force Density.1
 - Static Case Solution.1
 - Sensors.1
 - Energy

The 3D model in the center shows a ring with several blue isostatic restraint icons applied to its outer surface. A red box with the number '1' and an arrow points to the Isostatic Restraint icon in the toolbar on the right.

The 'Isostatic Restraint' dialog box is open, showing the name 'Isostatic.1' and 'OK' and 'Cancel' buttons.

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.

The screenshot shows the CATIA V5 interface with the following elements:

- Analysis Manager:** A tree view on the left showing the model structure: Analysis Manager > Links Manager > Finite Element Model > Nodes and Elements > Properties.1 > Static Case > Restraints.1 > Loads.1 > Line Force Density.1 > Static Case Solution.1 > Sensors.1.
- 3D Model:** A gray ring with a red line and arrows indicating the force density load. Red 'X' marks are visible on the ring's surface.
- Line Force Density Dialog:** A dialog box on the right with the following settings:
 - Name: Line Force Density.1
 - Supports: 1 Edge
 - Axis System: (empty)
 - Type: Global
 - Display locally:
 - Force Vector:
 - Norm: 85lbf_in
 - X: 0lbf_in
 - Y: 0lbf_in
 - Z: -85lbf_in
 - Optional Elements:
 - Data Mapping:
 - Buttons: OK, Cancel
- Callouts:** Red boxes with numbers 1, 2, and 3. Box 1 points to the Line Force Density icon in the toolbar. Box 2 points to the red line on the ring. Box 3 points to the Z field in the dialog.
- Other UI:** A coordinate system icon (x, y, z) is visible in the top right. The bottom status bar shows 'object or a command' and 'Help'.

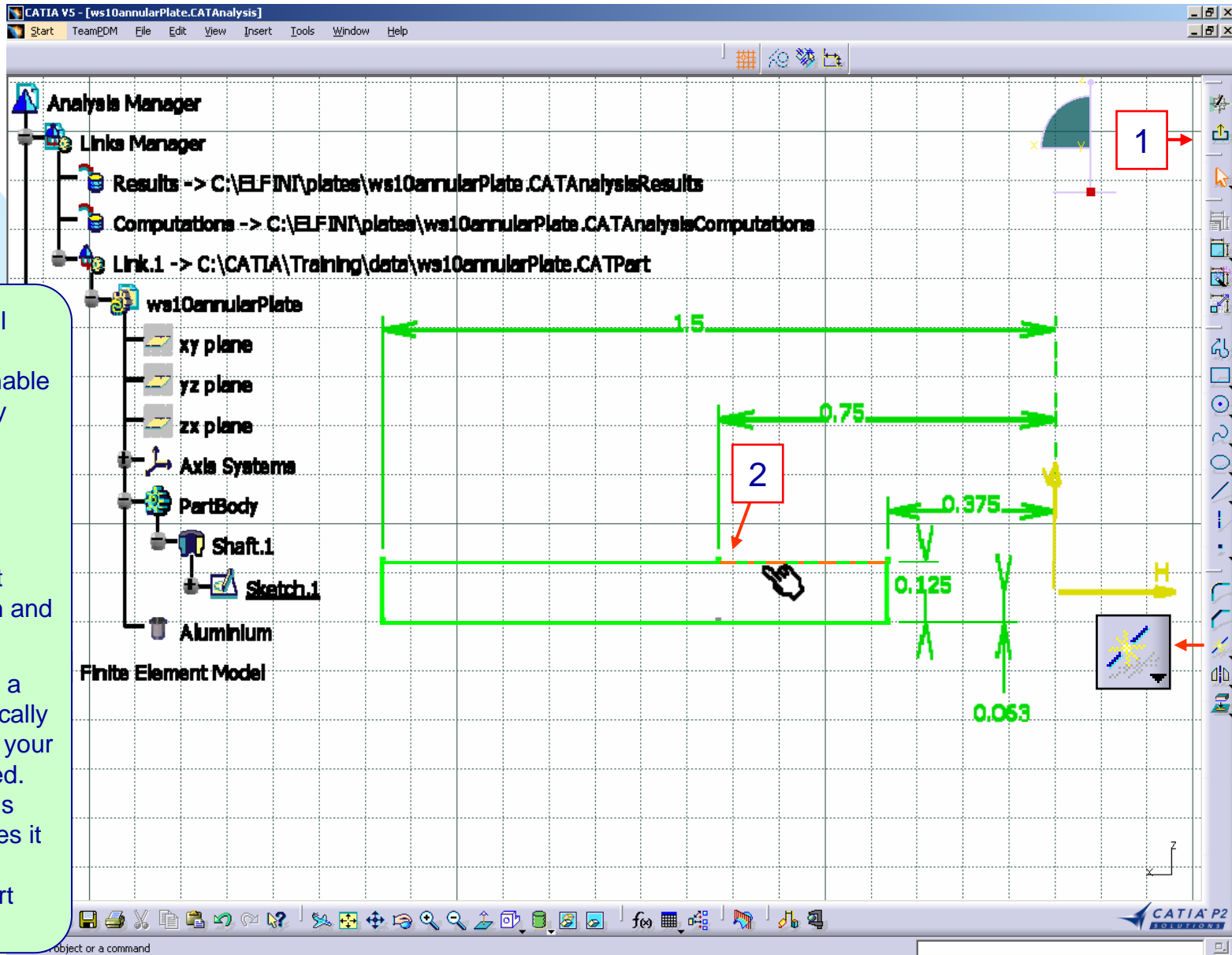
Step 4. Apply a line force density load

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

Analysis Manager

- Links Manager
- Finite Element Model
 - Nodes and Elements
 - OCTREE Tetrahedron Mesh.1
 - Properties.1
 - Static Case
 - Restraints.1
 - Restraint.1
 - Isostatic.1
 - Loads.1
 - Line Force Density.1
 - Static Case Solution.1
 - Sensors.1

Computation Resources Estimation

1e+002 s of CPU
1.01e+004 kilo-bytes of memory
6.13e+004 kilo-bytes of disk
Intel MKL(c) Library found: Intel(R) MKL V5.1.0

Do you want to continue the computation?

Yes No

1

Steps:

1. Compute All

Save often.

Step 6. Check global and local precision

CATIA V5 - [ws10annularPlate.CATAnalysis]

Analysis Manager

- Links Manager
- Finite Element Model
 - Nodes and Elements
 - Properties.1
 - Static Case
 - Restraints.1
 - Loads.1
 - Static Case Solution.1
 - Deformed Mesh
 - Sensors.1

Animate Window

Steps Number: 10

Speed

Close

1

2

Visualize the **Deformation** and animate.

Steps:

1. Select the Deformation icon.
2. Select on the Animate icon.

Step 6. Check global and local precision

Analysis Manager

- Links Manager
- Finite Element Model
 - Nodes and Elements
 - Properties.1
 - Static Case
 - Restrains.1
 - Loads.1
 - Static Case Solution.1
 - Deformed Mesh
 - Estimated local error
 - Sensors.1

Estimated local error

Btu

- 6.12e-009
- 5.51e-009
- 4.9e-009
- 4.28e-009
- 3.67e-009
- 3.06e-009
- 2.45e-009
- 1.84e-009
- 1.22e-009
- 6.13e-010
- 5.96e-013

Precision Location : Global
Estimated Precision : 0.00106225 Btu
Strain Energy : 0.343216 Btu
Global Estimated Error Rate : 3.93079 %

1

2

3

CATIA V5 - [ws10annularPlate.CATAnalysis]

Start TeamPDM File Edit View Insert Tools Window Help

Project or a command

Help

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.

Estimated local error Global Maximum, 1: 6.12046e-009 Btu

Analysis Manager
Links Manager
Finite Element Model
Nodes and Elements
Properties.1
Static Case
Restraints.1
Loads.1
Static Case Solution.1
Deformed Mesh
Estimated local error
Extrema
Global Maximum.1
Sensors.1

Focus On

Extrema Creation

Global
Minimum extrema at most 0
Maximum extrema at most 2
Local
Minimum extrema at most 0
Maximum extrema at most 2
OK Cancel

1

2

3

Select an object or a command

CATIA V5 - [ws10annularPlate.CATAnalysis]
Start TeamPDM File Edit View Insert Tools Window Help

CATIA V5 SOLUTIONS

Step 6. Check global and local precision

Analysis Manager

- Links Manager
- Finite Element Model
 - Nodes and Elements
 - Properties.1
 - Static Case
 - Restraints.1
 - Loads.1
 - Static Case Solution.1
 - Deformed Mesh
 - Estimated local error
 - Extrema
 - Global Maximum.1
 - Sensors.1
 - Adaptivity Process
 - Adaptivity Convergence.1
 - Adaptivities.1
 - Adaptivity Box.1

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%.

Adaptivity Box	
Name	Adaptivity Box.1
Objective Error (%)	5
Solution	Static Case Solution.1
Local Error (%)	5.5258
Select Extremum	
OK	Cancel



1

3

4

2b

2a



Step 7. Refine the mesh locally

CATIA V5 - [ws10annularPlate.CATAnalysis]

Start TeamPDM File Edit View Insert Tools Window Help

Analysis Manager

- Links Manager
- Finite Element Model
 - Nodes and Elements
 - OCTREE Tetrahedron Mesh.1
 - Properties.1
 - Static Case
 - Restraints.1
 - Loads.1
 - Static Case Solution.1
 - Sensors.1
- Adaptivity Process

Refine the mesh by $\frac{1}{2}$ 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.

Local Mesh Size

Name: Local Mesh Size

Supports: 1 Face

Value: 0.06in

OK Cancel

Local Mesh Sag

Name: Local Mesh Sag

Supports: 1 Face

Value: 0.006in

OK Cancel

Select or a command

CATIA V5 SOLUTIONS

Step 8. Compute the precise analysis

Steps:

1. Compute All.

Step 9. Visualize final results

Analysis Manager

- Links Manager
- Finite Element Model
 - Nodes and Elements
 - OCTREE Tetrahedron
 - Properties.1
 - Static Case
 - Restraints.1
 - Loads.1
 - Static Case Solution.1
 - Deformed Mesh
 - Estimated local error (2)
 - Extrema
 - Sensors.1
- Adaptivity Process

Estimated local error
Btu

3.02e-009
2.71e-009
2.41e-009
2.11e-009
1.81e-009
1.51e-009
1.21e-009
9.06e-010
6.04e-010
3.03e-010
1.35e-012

Precision Location : Global
Estimated Precision : 0.000633323 Btu
Strain Energy : 0.343535 Btu
Global Estimated Error Rate : 3.03467 % (1)

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

The screenshot shows the CATIA V5 interface with the following components:

- Analysis Manager Tree:**
 - Analysis Manager
 - Links Manager
 - Finite Element Model
 - Nodes and Elements
 - OCTREE Tetrahedron Mesh.1
 - Properties.1
 - Static Case
 - Restraints.1
 - Loads.1
 - Static Case Solution.1
 - Deformed Mesh
 - Estimated local error (1)
 - Extrema (2)
 - Global Maximum.1
 - Sensors.1
 - Adaptivity Process (3)

- Adaptivity Box Dialog:**
- Name: Adaptivity Box.1
- Objective Error (%): 5
- Solution: Static Case Solution.1
- Local Error (%): 3.757
- Buttons: Select Extremum, OK, Cancel
- 3D Visualization:** A blue mesh of a ring with a central hole. A red box highlights a specific region of the mesh.

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

Analysis Manager

- Links Manager
- Finite Element Model
 - Nodes and Elements
 - Properties.1
 - Static Case
 - Restraints.1
 - Loads.1
 - Static Case Solution.1
 - Def Static Case Solution.1 object
 - Est Report
 - Tra Generate Image
 - Sensors Clear Solution Storage
 - Adaptivity Process

Image Choice

Image Names

- Deformed Mesh
- Stress principal tensor component (element's nodes values)
- Stress principal tensor symbol
- Stress principal tensor symbol (absolute)
- Stress full tensor component (nodal values)
- Stress full tensor text
- Translational displacement magnitude
- Translational displacement component
- Translational displacement vector
- Von Mises Stress (nodal value)
- Von Mises Stress (element's nodes values)
- Estimated local error
- Strain principal tensor component (nodal values)
- Strain full tensor component (nodal values)
- Strain principal tensor symbol
- Rotational displacement magnitude
- Rotational displacement vector
- Point force vector
- Point force text
- Moment vector
- Local strain energy
- Local strain energy density

Cancel Help

1

2

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.

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.

The screenshot shows the CATIA V5 interface for a finite element analysis. The Analysis Manager tree on the left shows the hierarchy: Analysis Manager > Links Manager > Finite Element Model > Nodes and Elements > Properties.1 > Static Case > Restraints.1 > Restraint.1 > Isostatic.1 > Inside edge Dummy restraint. The 3D model of the annular plate shows various restraints (blue anchors) and loads (yellow arrows). The 'Advanced Restraint' dialog box is open, showing 'Name: Restraint.2', 'Supports: 1 Edge', 'Axis System: Global', and 'Type: Global'. The 'Restraining Translation' options (1, 2, 3) are unselected, while 'Restraining Rotation 3' is selected. The 'Image Edition' dialog box is also open, showing the 'Selections' tab with a list of elements including 'Inside edge Dummy restraint'. The 'Compute All' button is highlighted with a red box and arrow. The 'Image Edition' dialog box is also highlighted with a red box and arrow.

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.

CATIA V5 - [ws10annularPlate.CATAnalysis]

Start TeamPDM File Edit View Insert Tools Window Help

Analysis Manager

Links Manager

Finite Element Model

Nodes and Elements

Properties.1

Static Case

Restrains.1

Loads.1

Static Case Solution.1

Deformed Mesh

Estimated local error

Translational displacement vector

Extrema

Global Maximum.1

Translational displacement text

Sensors.1

Adaptivity Process

Translational displacement vector Global Maximum.1: 0.0219658 in

Focus On

0.0219658

0.02186

0.02186

0.02186

0.02185

0.02185

0.02129

0.02129

0.02183

0.02119

0.02064

0.02071

0.02183

0.02127

44

7

1055

28

1

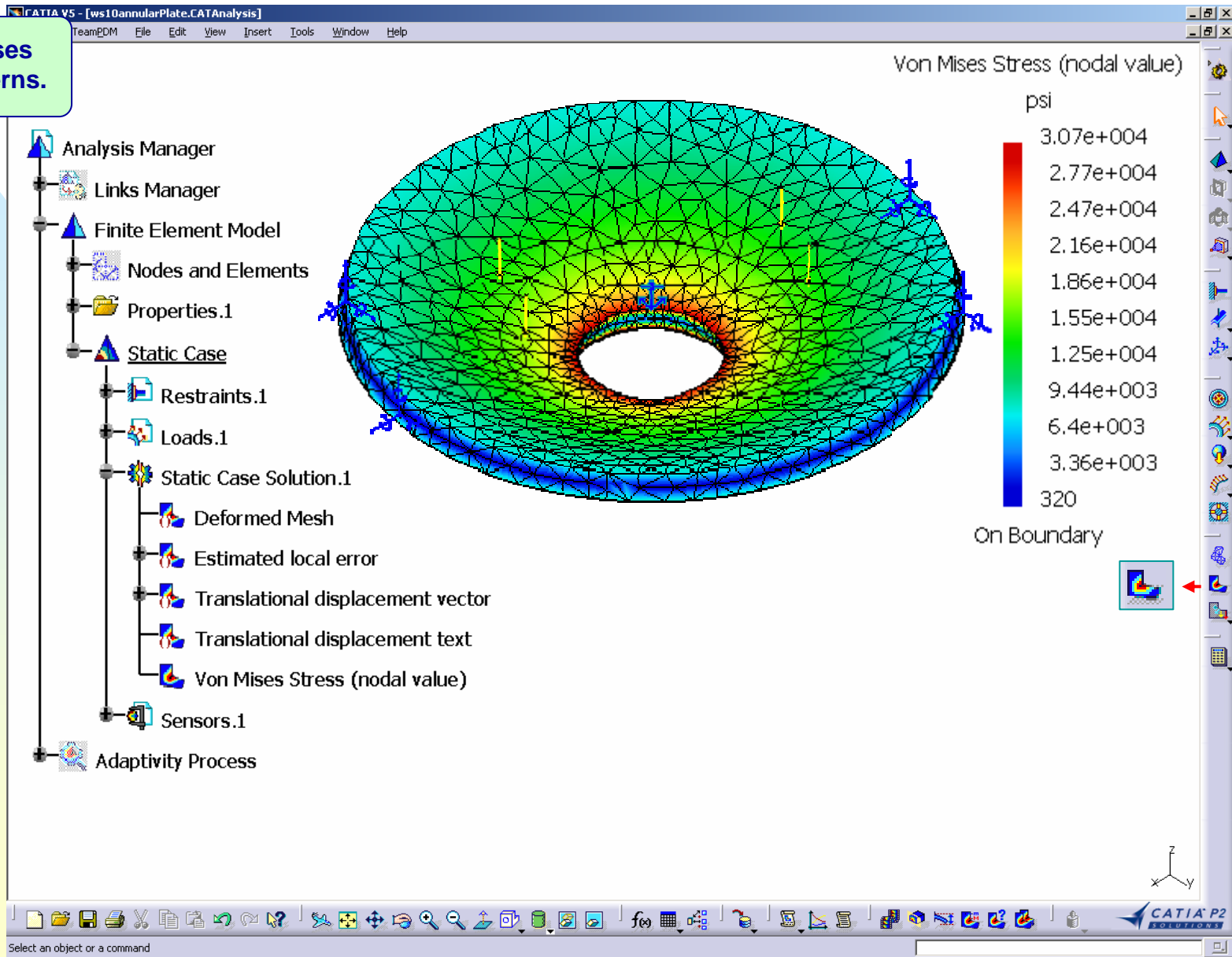
2

Focus On

CATIA V5 P2 SOLUTIONS

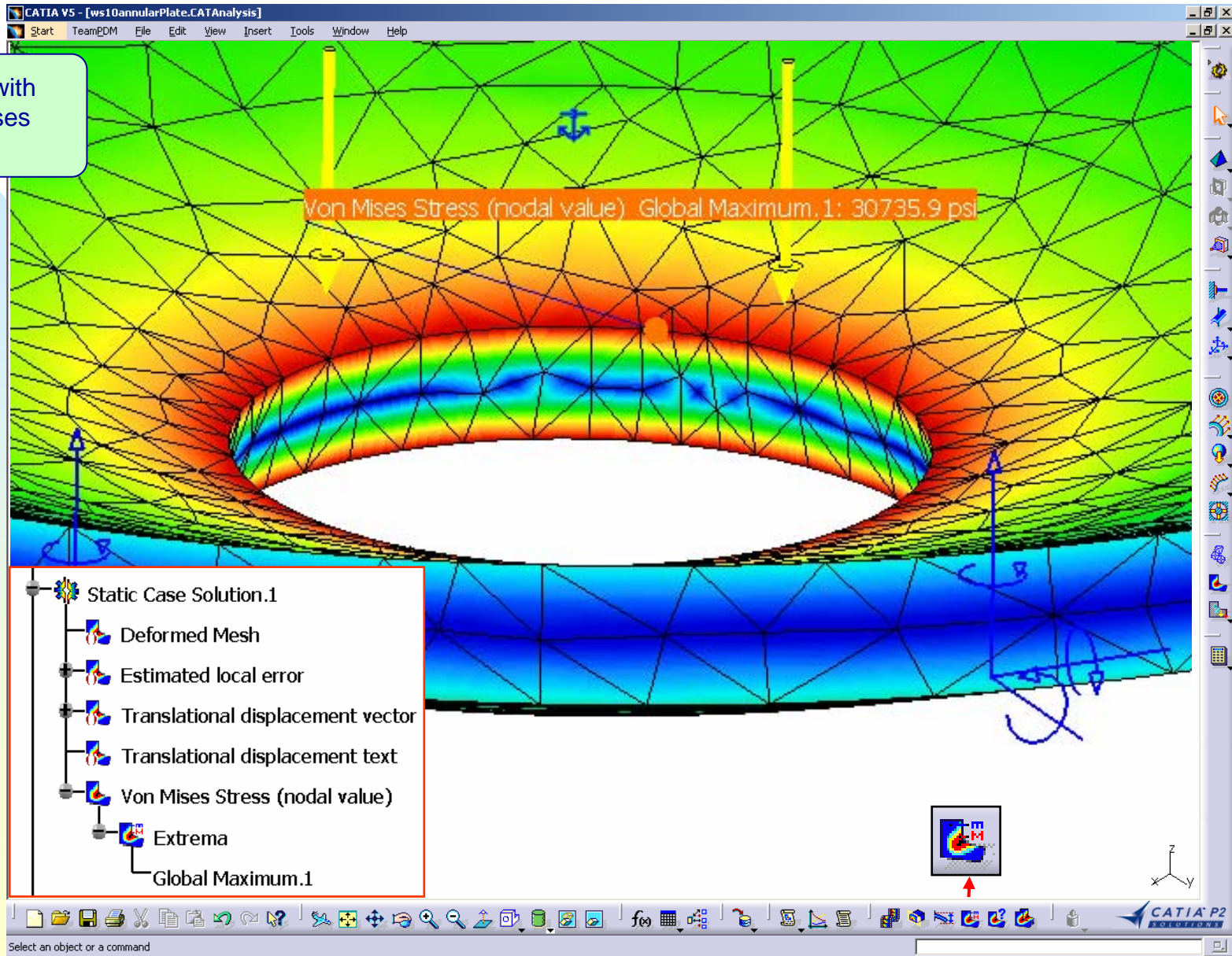
Step 9. Visualize final results

Visualize Von Mises stress field patterns.



Step 9. Visualize final results

Find the element with maximum Von Mises Stress.



Step 9. Visualize final results

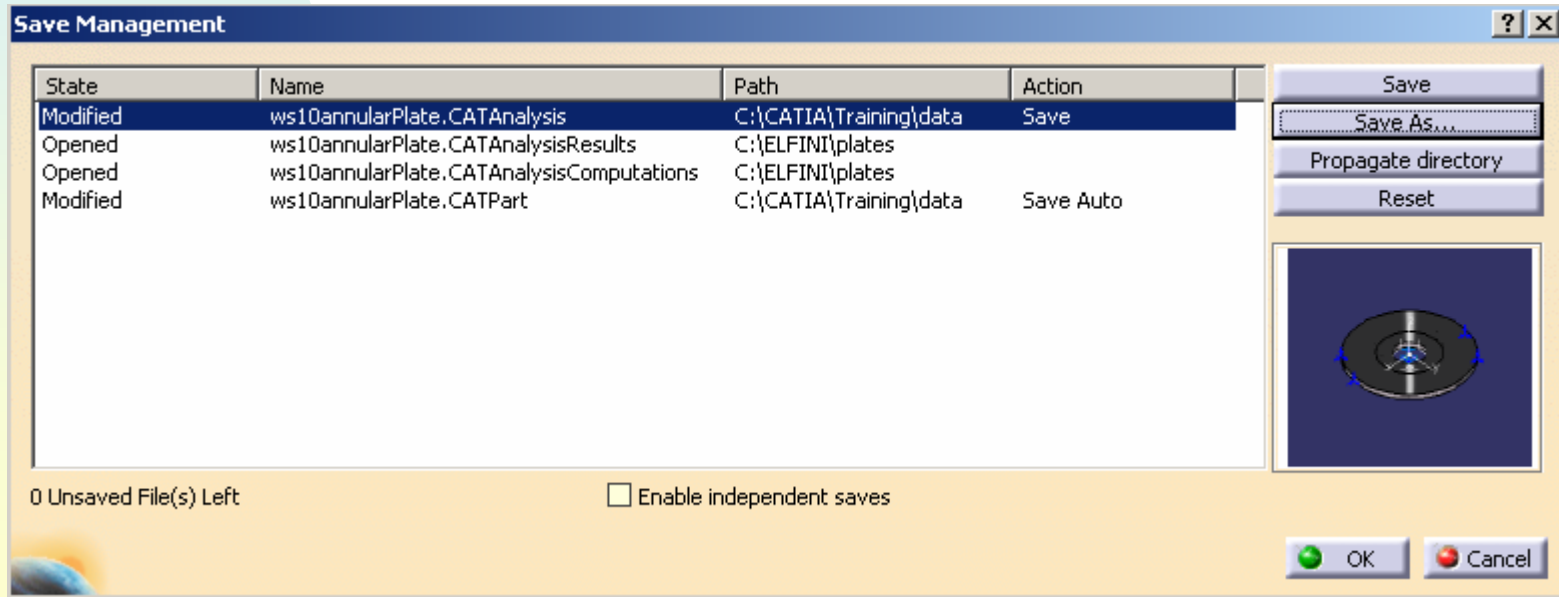
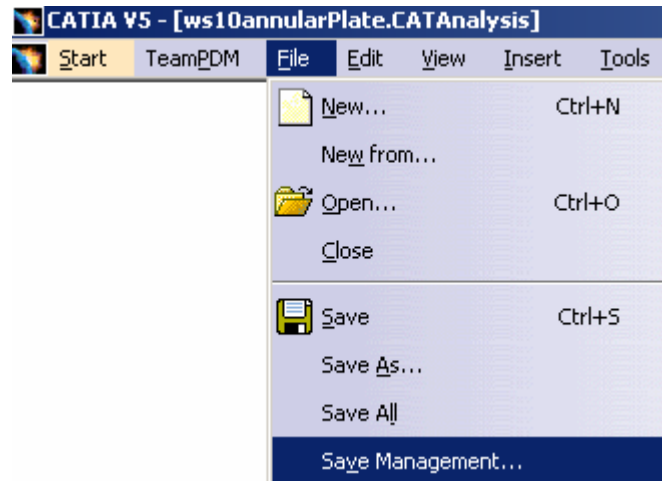
■ Conclusions

- ◆ 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	NA	3.03 %
Local % Precision error	NA	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

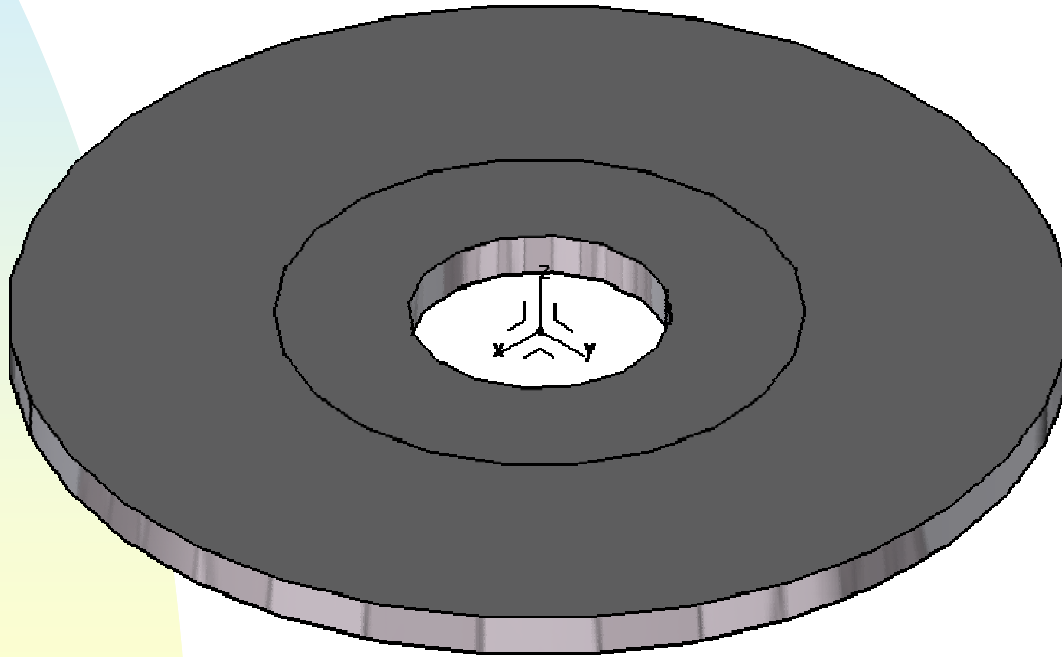
Step 10. Save the analysis document

Save your documents



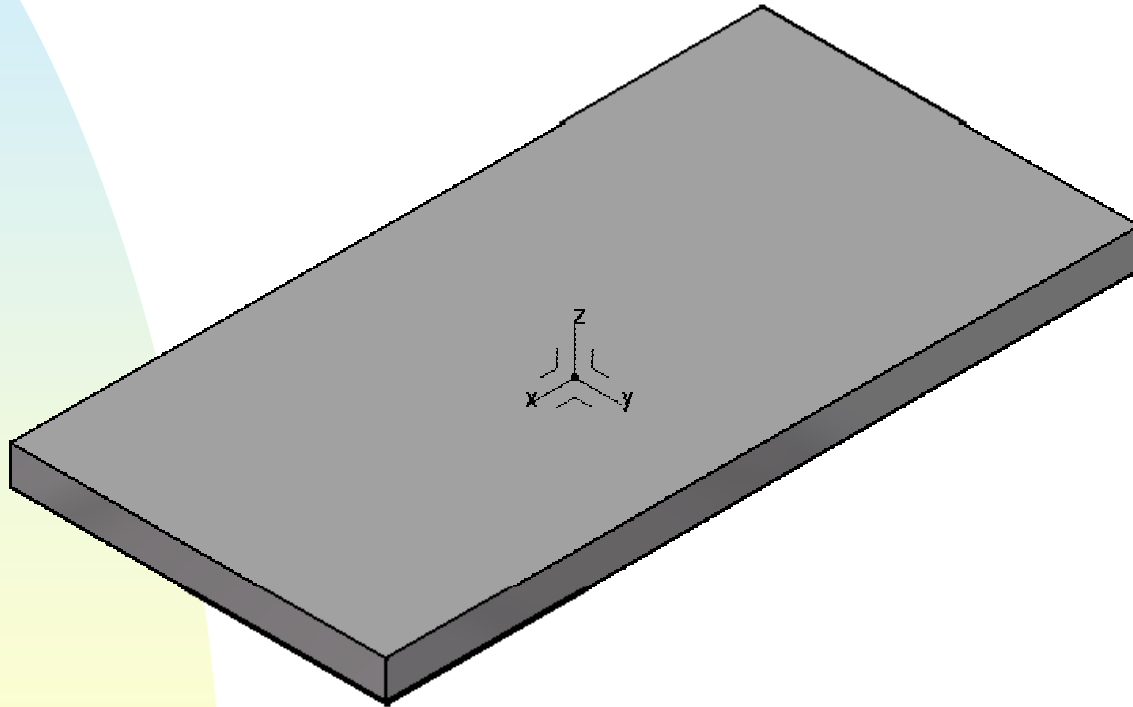
WORKSHOP 10...

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



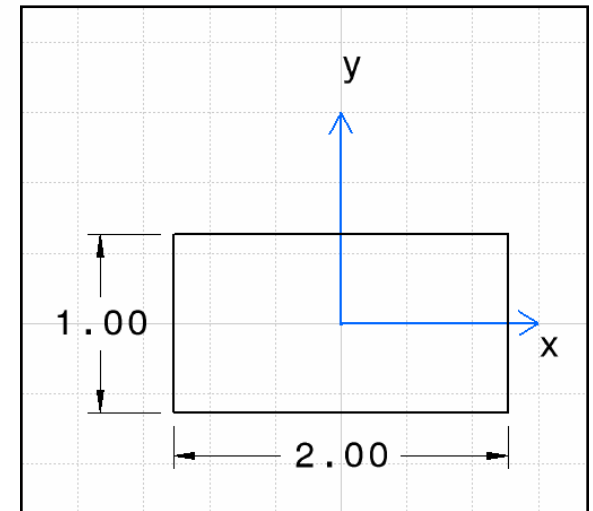
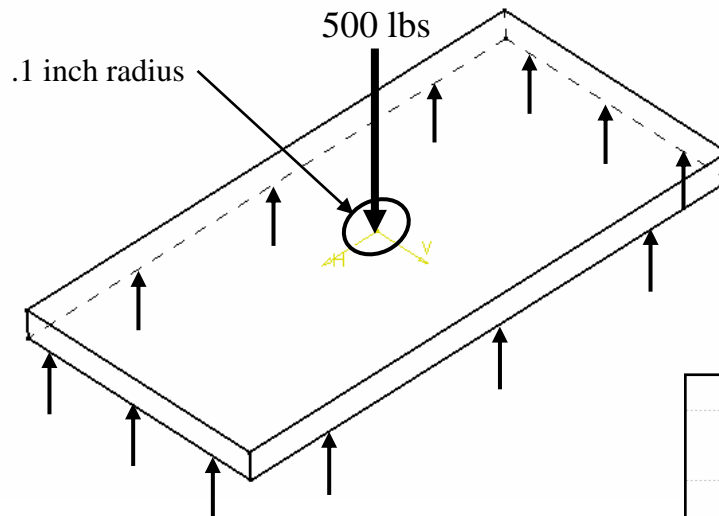
WORKSHOP 10b

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

■ 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

The screenshot shows the CATIA V5 software interface with the Analysis Manager tree on the left and a 3D model of a rectangular plate on the right. The tree structure is as follows:

- Analysis Manager
 - Links Manager
 - Results -> C:\ELFINI\plates\ws10aRectPlate.CATAnalysisResults
 - Computations -> C:\ELFINI\plates\ws10aRectPlate.CATAnalysisComputations
 - Link.1 -> C:\CATIA\Training\data\ws10aRectPlate.CATPart
 - Finite Element Model
 - Nodes and Elements
 - OCTREE Tetrahedron Mesh.1 : rectangularPlate
 - Properties.1
 - Static Case
 - Restraints.1
 - Loads.1
 - Static Case Solution.1
 - Sensors.1
 - Energy

Red arrows and numbered boxes indicate the following steps:

1. Points to the Link.1 node in the Links Manager.
2. Points to the Properties.1 node in the Finite Element Model.
3. Points to the Analysis Manager icon in the top right corner.
4. Points to the Results and Computations nodes in the Links Manager.

The 3D model shows a gray rectangular plate with a green coordinate system (x, y, z) centered on its top surface.

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

Analysis Manager

- Links Manager
- Finite Element Model
 - Nodes and Elements
 - OCTREE Tetrahedron Mesh.1 : rectangularPlate
 - Properties.1
 - Static Case
 - Restraints.1
 - Loads.1
 - Static Case Solution.1
 - Sensors.1

OCTREE Tetrah...

Global | Local

Size 0.1in

Sag 0.01in

Element type

Linear

Parabolic

OK Cancel

Select or a command

Steps:

1. Globally mesh as shown.

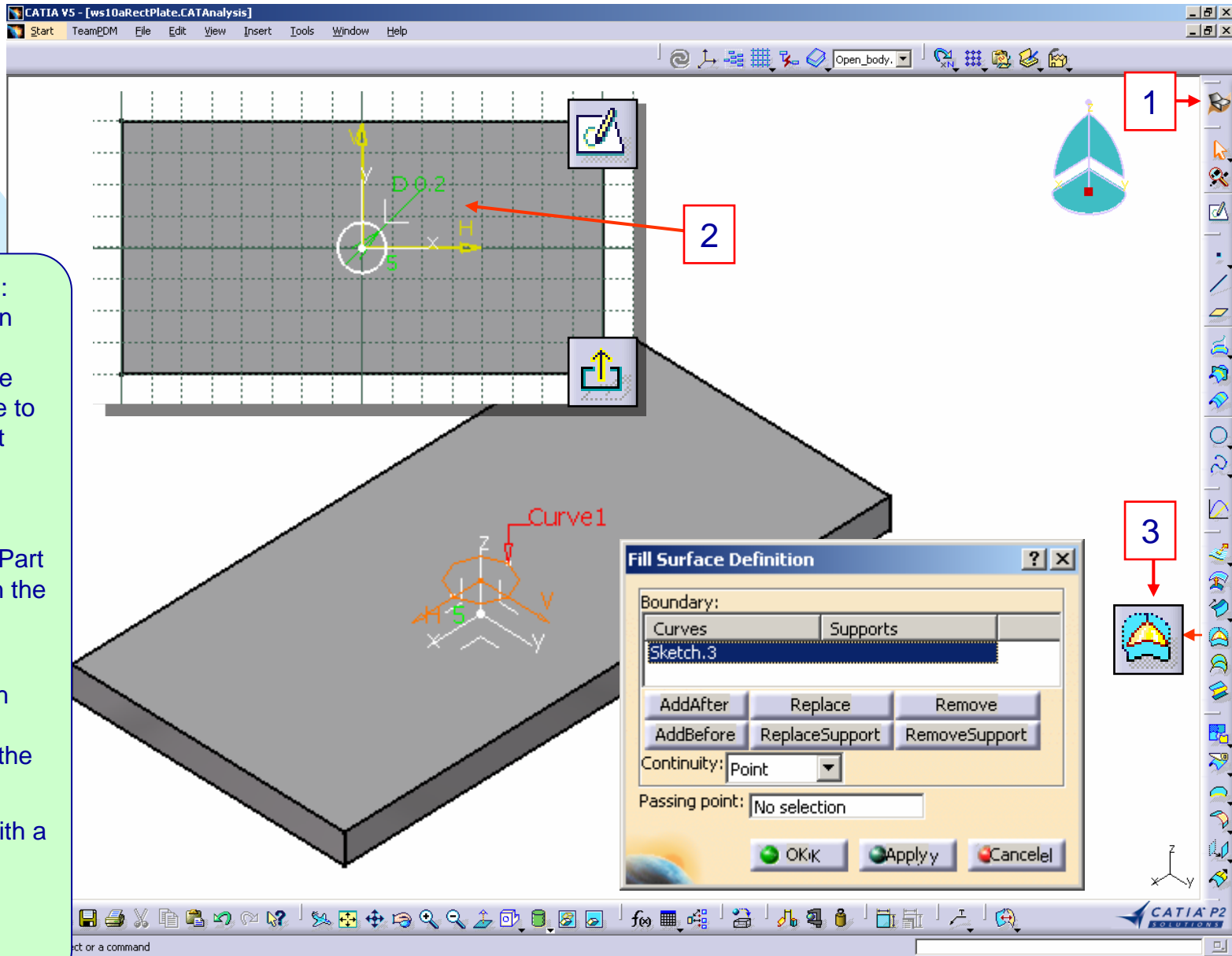
As plates typically are large using one mesh element through the thickness is a good way to start. Then use localized adaptive meshing for precise results.

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:

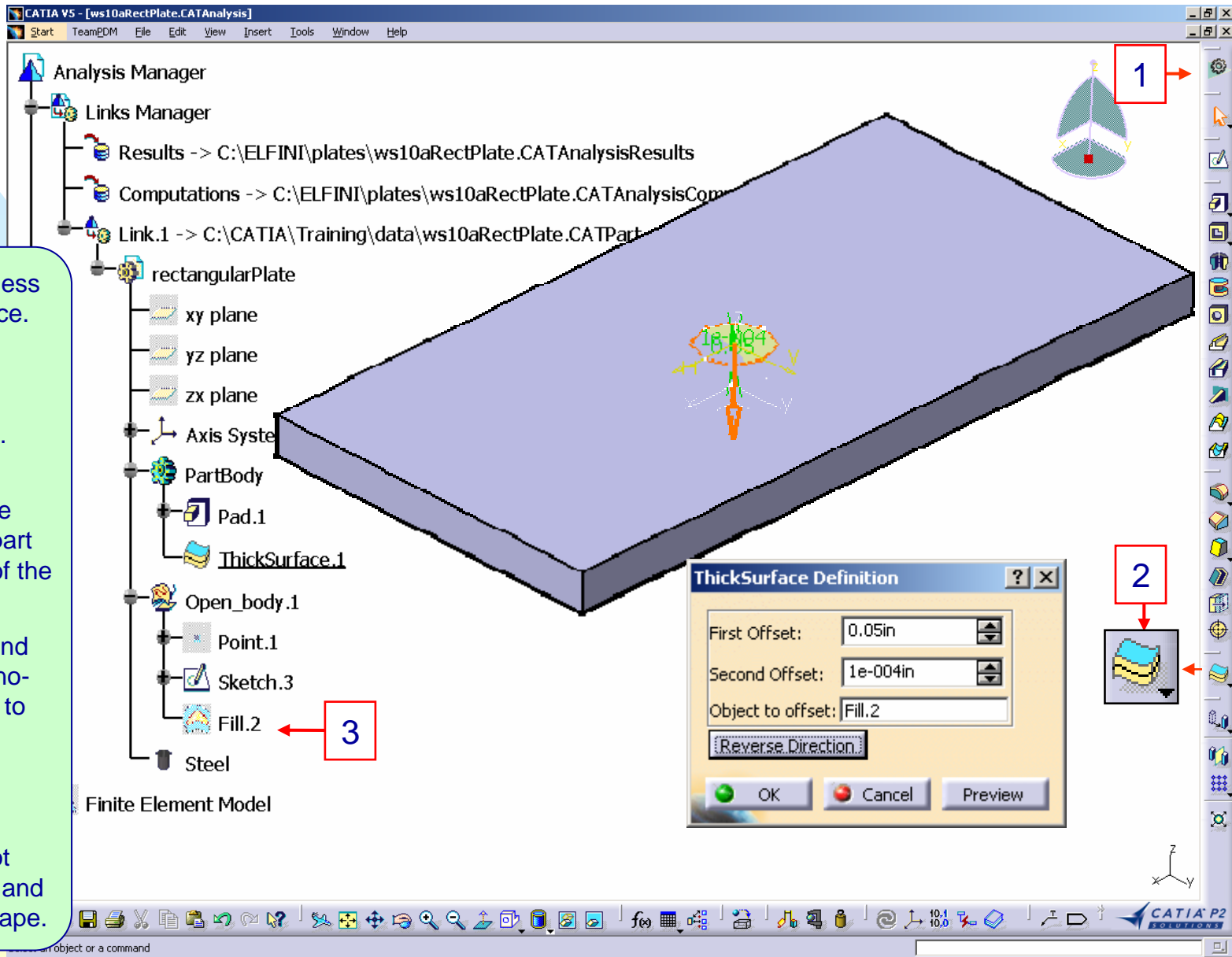
1. Make your .CATPart current and Launch the Generative Shape Design workbench.

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.

Step 4. Apply a force

CATIA V5 - [ws10aRectPlate.CATAnalysis]

Analysis Manager

- Links Manager
- Finite Element Model
 - Nodes and Elements
 - Properties.1
 - Static Case
 - Restraints.1
 - Loads.1
 - Distributed Force.1
 - Static Case Solution.1
 - Sensors.1
 - Energy

Distributed Force

Name: Distributed Force.1

Supports: 1 Face

Axis System

Type: Global

Display locally

Force Vector

Norm: 500lbf

X: 0lbf

Y: 0lbf

Z: -500lbf

OK Cancel

1

Select an object or a command

CATIA V5 SOLUTIONS

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

Analysis Manager

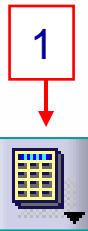
- Links Manager
- Finite Element Model
 - Nodes and Elements
 - OCTREE Tetrahedron Mesh
 - Properties.1
 - Static Case
 - Restraints.1
 - Restraint.1
 - Isostatic.1
 - Loads.1
 - Distributed Force.1
 - Static Case Solution.1
 - Sensors.1
 - Energy

Computation Resources Estimation

2e+001 s of CPU
4.26e+003 kilo-bytes of memory
2.07e+004 kilo-bytes of disk
Intel MKL(c) Library found: Intel(R) MKL V5.1.0

Do you want to continue the computation?

Yes No



Select an object or a command

Step 6. Check global and local precision

Analysis Manager

- Links Manager
- Finite Element Model
 - Nodes and Elements
 - Properties.1
 - Static Case
 - Restraints.1
 - Loads.1
 - Static Case Solution.1
 - Deformed Mesh
 - Estimated local error
 - Sensors.1
 - Energy

Estimated local error
Btu

- 1.43e-008
- 1.29e-008
- 1.15e-008
- 1e-008
- 8.59e-009
- 7.16e-009
- 5.73e-009
- 4.3e-009
- 2.87e-009
- 1.44e-009
- 3.94e-012

1a

2a

2b

1b

Precision Location : Global
Estimated Precision : 0.00101514 Btu
Strain Energy : 0.0881795 Btu
Global Estimated Error Rate : 7.56515 %

CATIA V5 - [ws10aRectPlate.CATAnalysis]
Start TeamPDM File Edit View Insert Tools Window Help

Object or a command

CATIA P2 SOLUTIONS

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

The screenshot displays the CATIA V5 software interface. On the left, the Analysis Manager tree shows the following structure:

- Analysis Manager
 - Links Manager
 - Finite Element Model
 - Nodes and Elements
 - Properties.1
 - Static Case
 - Restraints.1
 - Loads.1
 - Static Case Solution.1
 - Deformed Mesh
 - Estimated local error
 - Extrema
 - Global Maximum.1
 - Sensors.1
 - Adaptivity Process
 - Adaptivity Convergence.1
 - Adaptivities.1
 - Adaptivity Box.1

In the center, the 'Adaptivity Box' dialog box is open, showing the following fields:

- Name: Adaptivity Box.1
- Objective Error (%): 5
- Solution: Static Case Solution.1
- Local Error (%): 8.045 (circled in red)

Buttons include 'Select Extremum', 'OK', and 'Cancel'.

On the right, a 3D model of a rectangular plate is shown with a red search box around a specific element. A magnifying glass icon is also visible on the right toolbar.

At the bottom, a toolbar contains various icons, with a red box labeled '1' highlighting the 'Adaptivity Box' icon.

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%.

Step 8. Visualize final results

Analysis Manager

- Links Manager
- Finite Element Model
 - Nodes and Elements
 - Properties.1
 - Static Case
 - Restraints.1
 - Loads.1
 - Static Case Solution.1
 - Deformed Mesh ← 2
 - Estimated local error
 - Extrema
 - Sensors.1
- Adaptivity Process
 - Adaptivity Convergence
 - Adaptivities.1
 - Adaptivity Box.1 ← 1

Adaptivity Box

Name: Adaptivity Box.1

Objective Error (%): 3

Solution: Static Case Solution.1

Local Error (%): 3.8071

Select Extremum

OK Cancel

Precision Location : Global
Estimated Precision : 0.000594382 Btu
Strain Energy : 0.0895442 Btu
Global Estimated Error Rate : 5.75148 %

Steps:

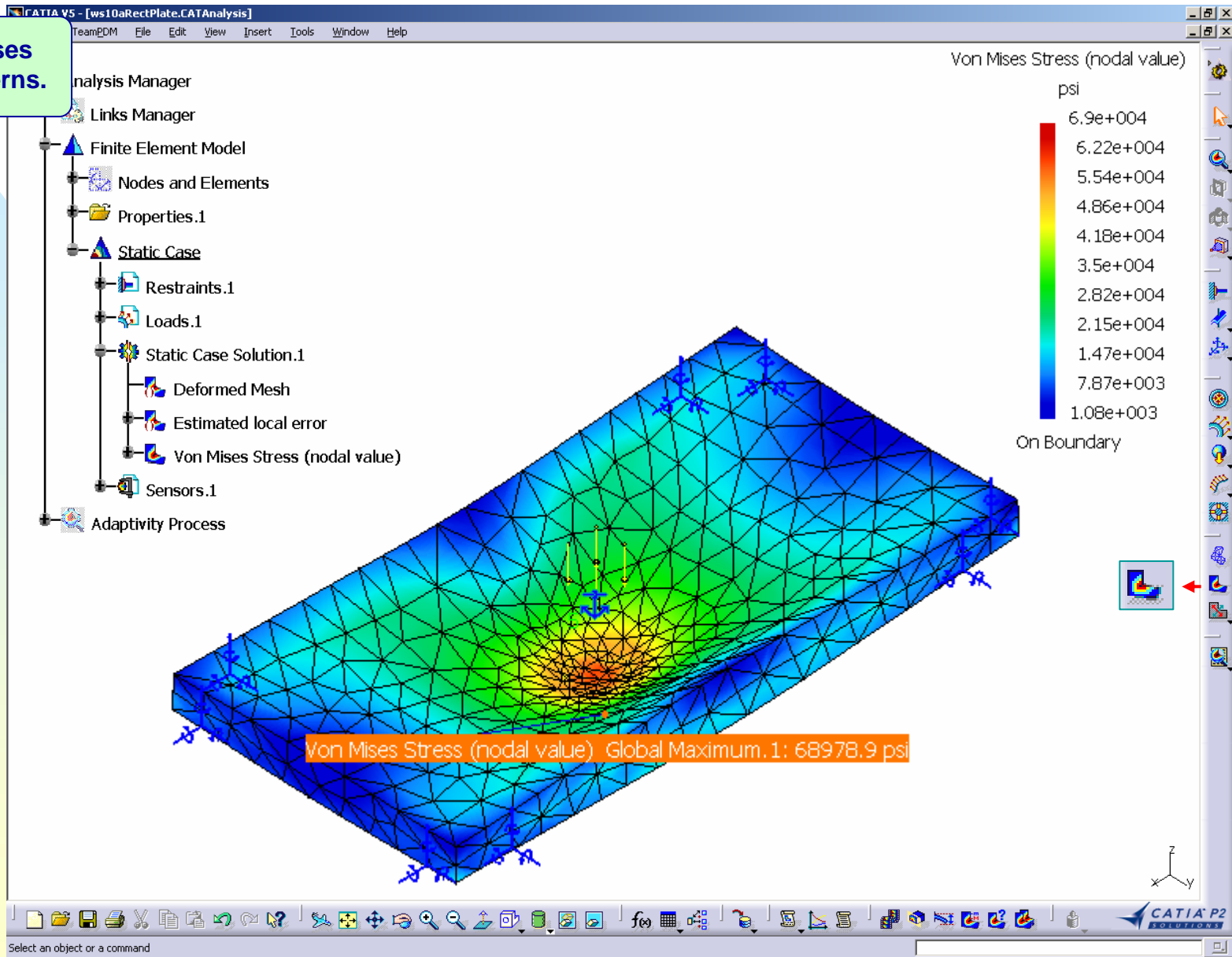
1. Check local precision again.

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

We now have a precise model.

Step 8. Visualize final results

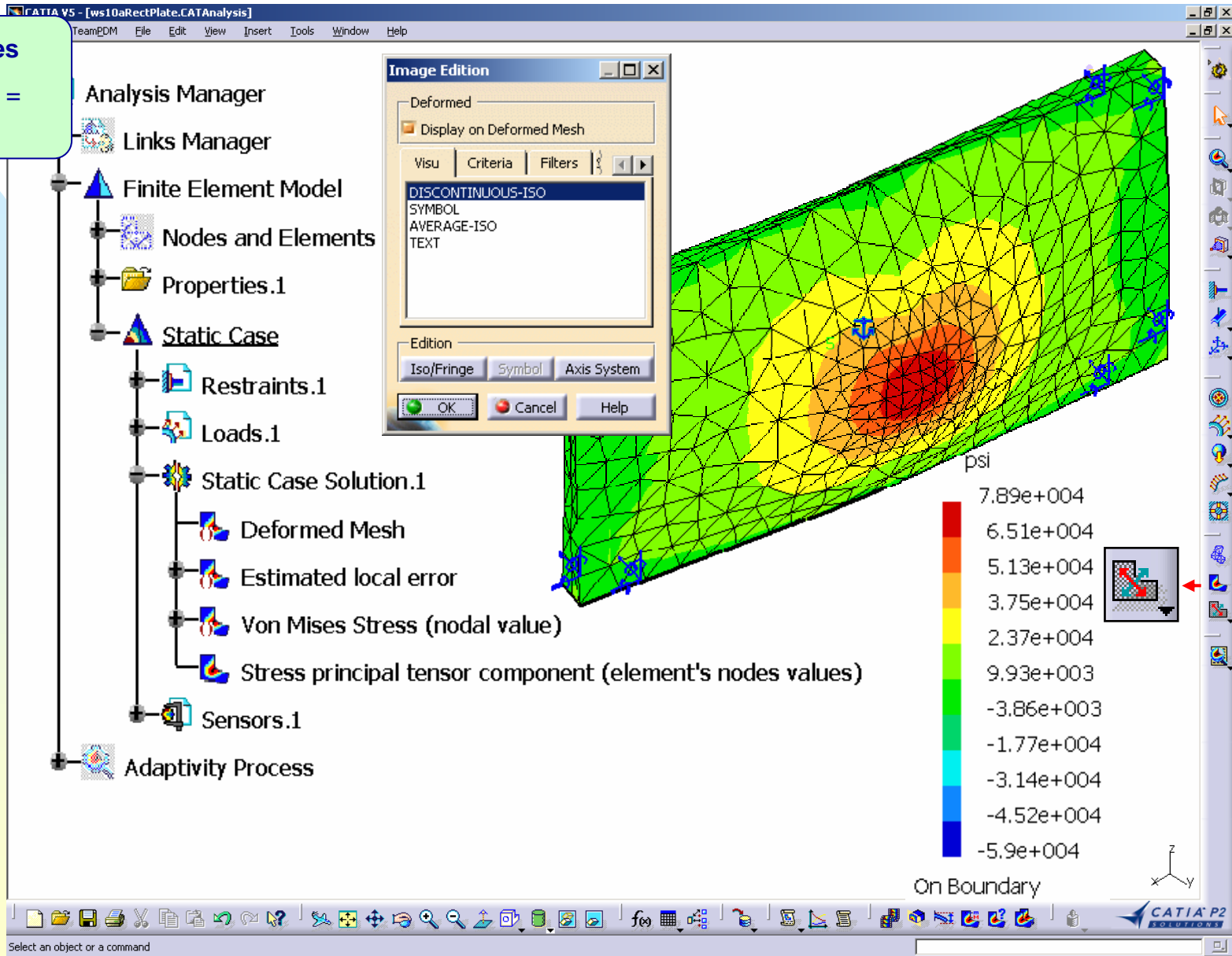
Visualize Von Mises stress field patterns.



Step 8. Visualize final results

Principal Stresses

Hand calculations =
80,317 psi



Step 8. Visualize final results

Vertical displacement

Hand calculations =
0.003 inches

Analysis Manager

- Links Manager
- Finite Element Model
 - Nodes and Elements
 - Properties.1
 - Static Case
 - Restraints.1
 - Loads.1
 - Static Case Solution.1
 - Deformed Mesh
 - Estimated local error
 - Von Mises Stress (nodal value)
 - Stress principal tensor component (element)
 - Translational displacement magnitude
 - Sensors.1
 - Adaptivity Process

Image Edition

Deformed

Display on Deformed Mesh

Visu Criteria Filters

DISCONTINUOUS-ISO

SYMBOL

AVERAGE-ISO

TEXT

Edition

Iso/Fringe Symbol Axis System

OK Cancel Help

in

0.00328

0.00295

0.00263

0.0023

0.00197

0.00164

0.00131

0.000985

0.000657

0.000328

0

On Boundary

Step 8. Visualize final results

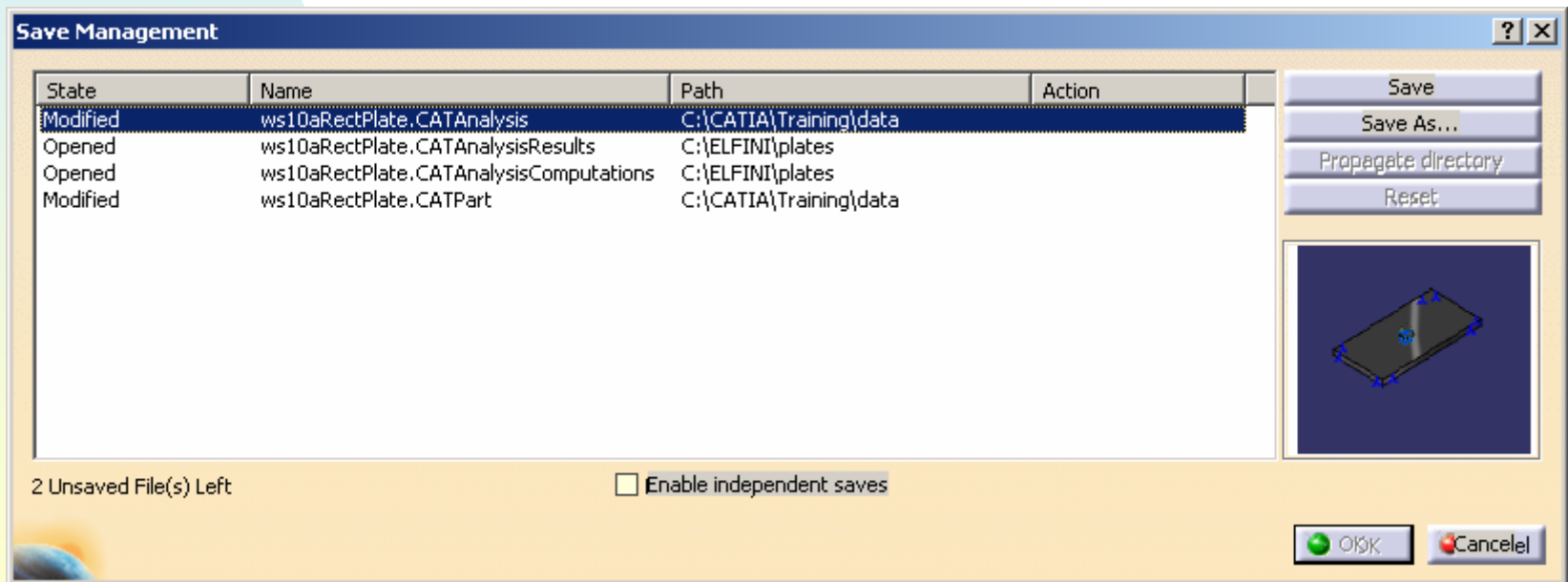
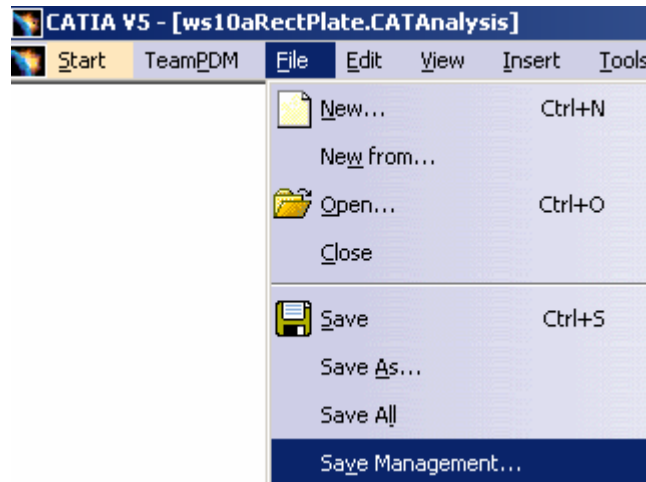
■ Conclusions

- ◆ 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	NA	5.7 %
Local % Precision error	NA	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

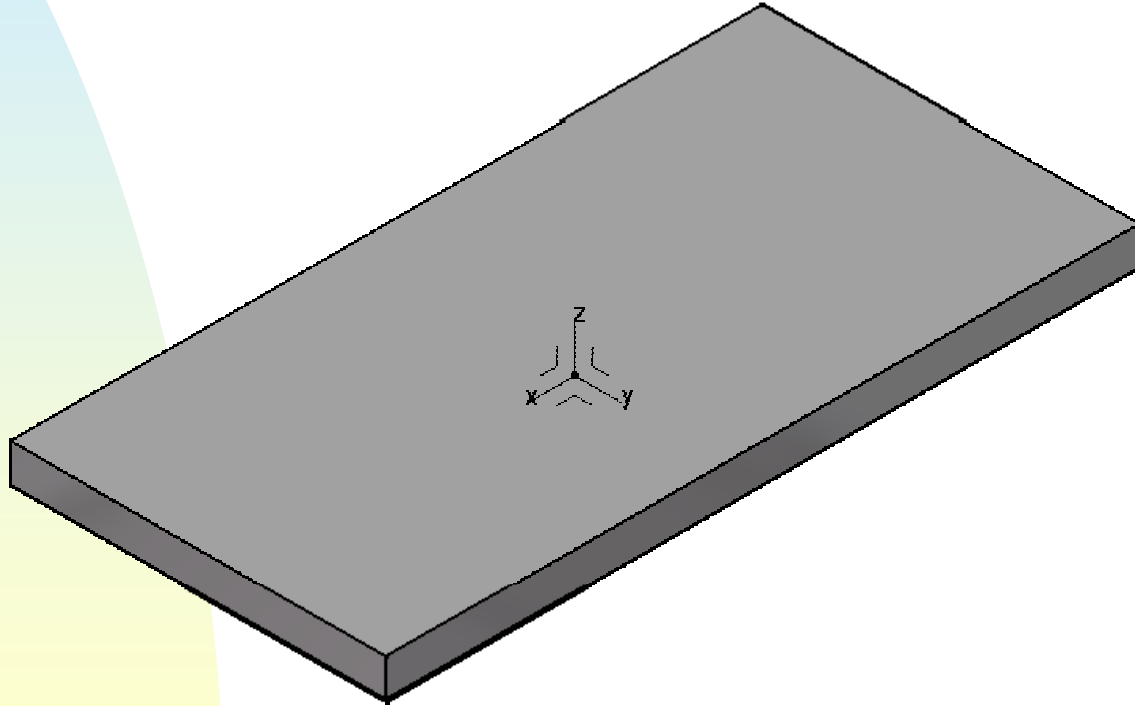
Step 9. Save the analysis document

Save your documents



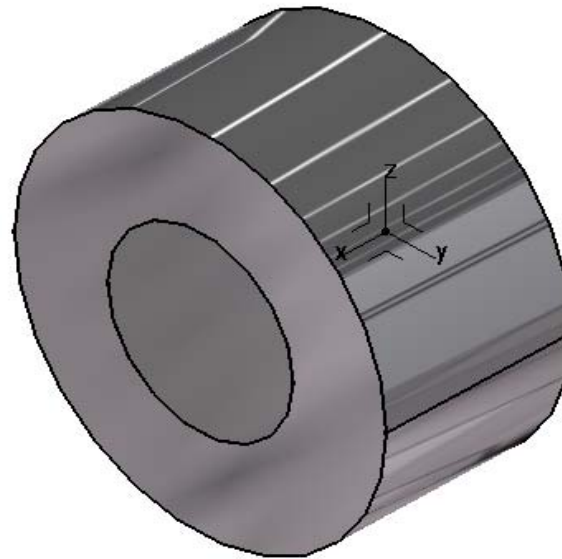
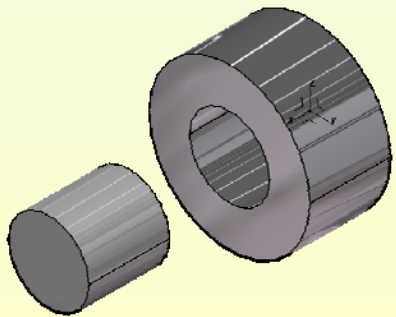
WORKSHOP 10...

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



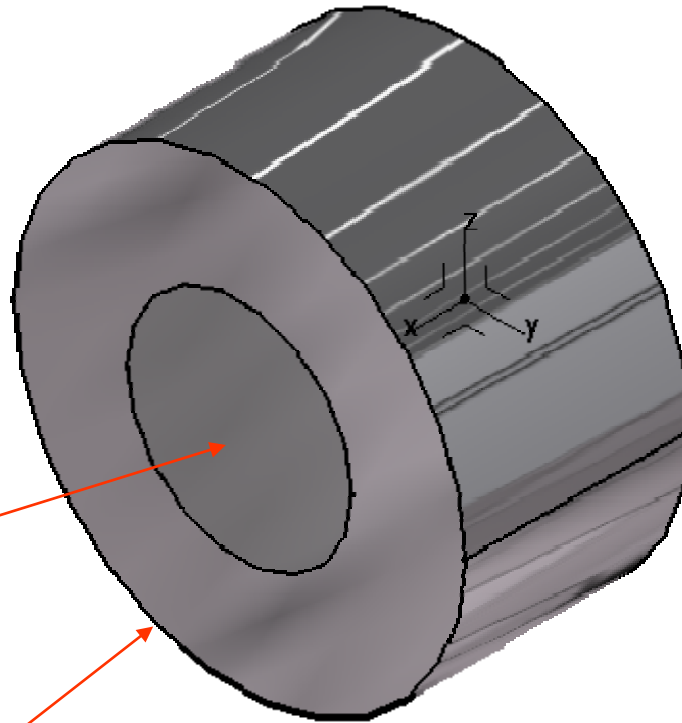
WORKSHOP 11

PRESS FIT



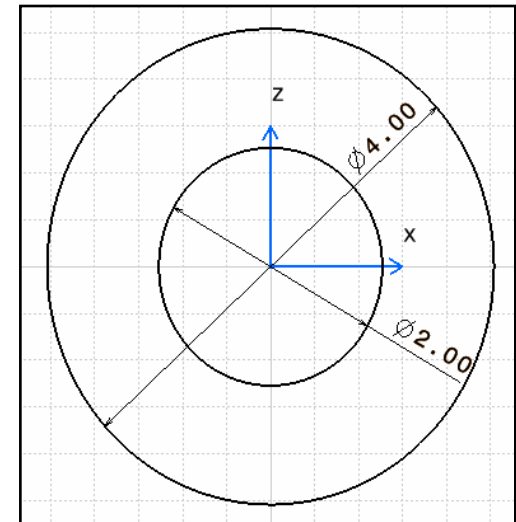
■ Problem Description

- ◆ 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

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

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

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

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

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

- ◆ Hoop stress at outside and inside diameters.

$$\sigma_{\text{hoop}} = P \cdot \frac{(r_{\text{outer cylinder}}^2 + r_{\text{inner cylinder}}^2)}{(r_{\text{outer cylinder}}^2 - r_{\text{inner cylinder}}^2)}$$

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

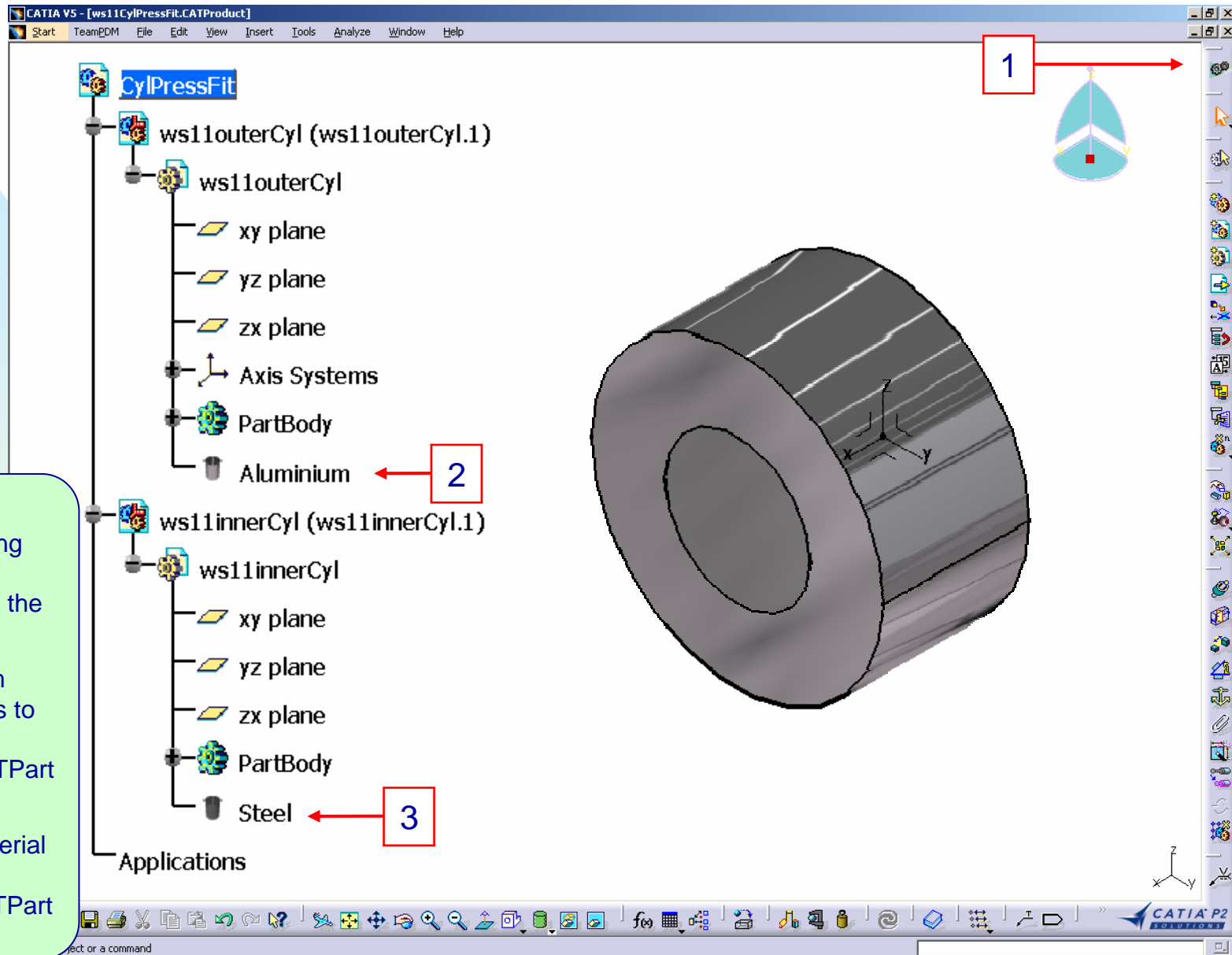
■ Suggested Exercise Steps

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

1

2

3

4

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.

Geometry not published.
Select another geometry or publish it before use it.

Close

CATIA V5 - [ws11CylPressFit.CATProduct]

Start TeamPDM File Edit View Insert Tools Analyze Window Help

f(x) Formula...
Image
Macro
Customize...
Visualization Filter
Options...

Options

General Constraints

Paste components

- Without the assembly constraints
- With the assembly constraints only after a Copy
- With the assembly constraints only after a Cut
- Always with the assembly constraints

Constraints creation

- Use any geometry
- Use published geometry of child components only
- Use published geometry of any level

Assistant

Geometry not published.
Select another geometry or publish it before use it.

Close

CATIA V5 P2 SOLUTIONS

Step 2. Create assembly constraints

Using the Compass

Steps:

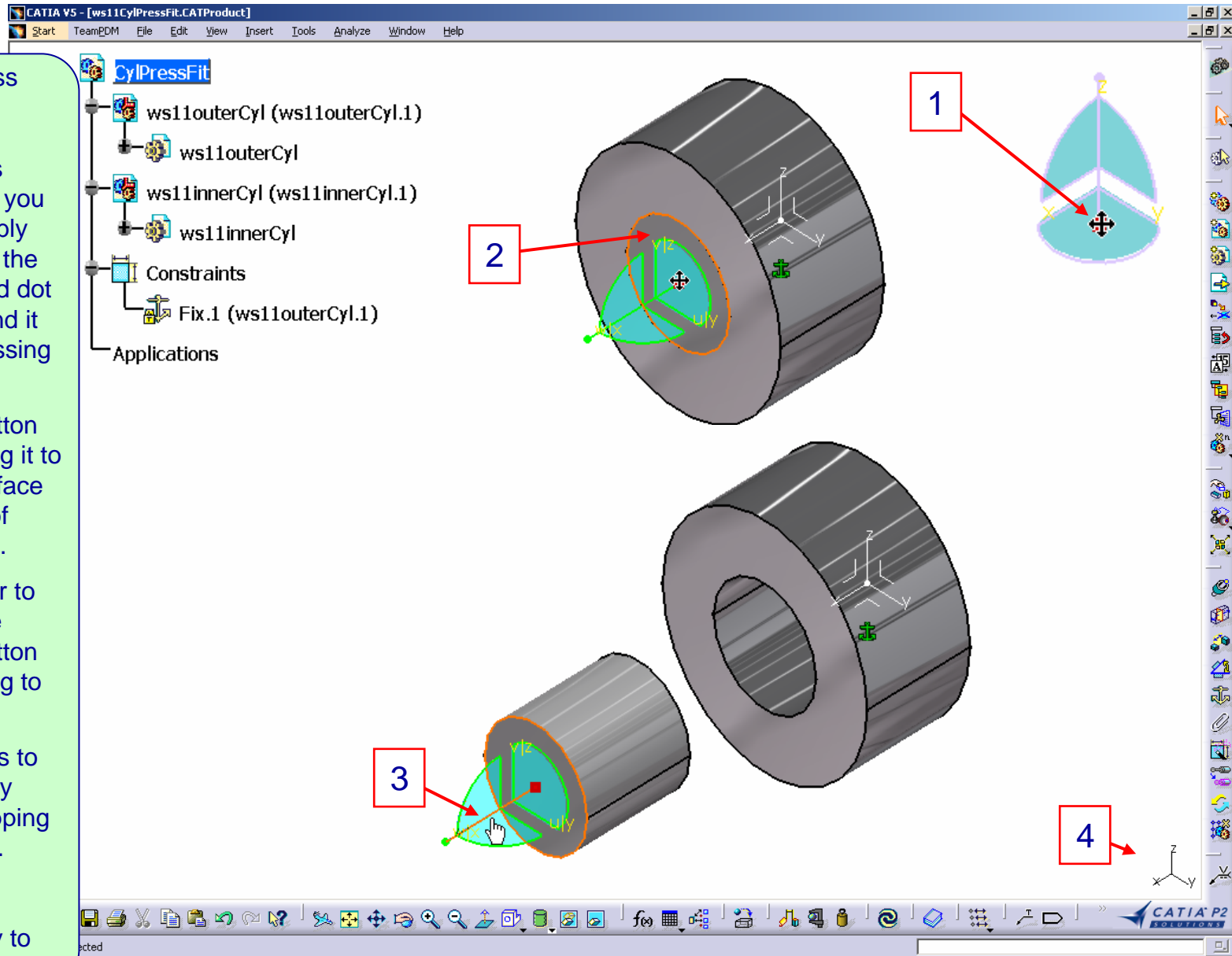
1. The compass is available to assist you in creating assembly constraints. Move the cursor onto the red dot of the compass and it will change to crossing arrows.

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 cursor 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.

The screenshot displays the CATIA V5 interface for a product named 'ws11CylPressFit.CATProduct'. The 'Options' dialog box is open, with the 'Constraints' tab selected. A red box labeled '1' points to the 'Manual' radio button under the 'Update' section. The 'Assembly Design' option is highlighted in the left-hand tree. Below the tree, a 3D model of two cylinders is shown. A red box labeled '2' points to the 'Contact Constraint' icon in the software's toolbar. A red box labeled '3' points to the two cylinders in the 3D model, which have orange selection boxes around their inner and outer surfaces. In the bottom-left corner, the 'Constraint Properties' dialog box is open, showing 'Surface contact' selected. The 'Name' field contains 'Surface contact.2'. The 'Supporting Elements' table is as follows:

Type	Component	Status
Cylinder	ws11outerCyl (ws11outerCyl.1)	Connected
Cylinder	ws11innerCyl (ws11innerCyl.1)	Connected

The 'OK' button is highlighted in the dialog box.

Step 2. Create assembly constraints

CATIA V5 - [ws11CylPressFit.CATProduct]

Start TeamPDM File Edit View Insert Tools Analyze Window Help

CylPressFit

- ws11outerCyl (ws11outerCyl.1)
- ws11innerCyl (ws11innerCyl.1)
- Constraints
 - Fix.1 (ws11outerCyl.1)
 - Surface contact.2 (ws11outerCyl.1,ws11innerCyl.1)
 - Offset.4 (ws11outerCyl.1,ws11innerCyl.1)
- Applications

Constraint Properties

Name: Offset.4

Supporting Elements

Type	Component	Status
Plane	ws11outerCyl (ws11outerCyl.1)	Connected
Plane	ws11innerCyl (ws11innerCyl.1)	Connected

Orientation: Same

Offset: 0in

OK Cancel

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

The screenshot shows the CATIA V5 interface with the Analysis Manager tree on the left and a 3D model of a cylinder with a hole on the right. The tree structure is as follows:

- Analysis Manager
 - Links Manager
 - Results -> C:\ELFINI\pressFit\pressFit.CATAnalysisResults
 - Computations -> C:\ELFINI\pressFit\pressFit.CATAnalysisComputations
 - Link.1 -> C:\CATIA\Training\data\ws11CylPressFit.CATProduct
 - Finite Element Model
 - Nodes and Elements
 - OCTREE Tetrahedron Mesh.1 : ws11outerCyl.1
 - OCTREE Tetrahedron Mesh.2 : ws11innerCyl.1
 - Properties.1
 - Material Property3D.1
 - Material Property3D.2
 - Static Case
 - Restraints.1
 - Loads.1
 - Static Case Solution.1
 - Sensors.1
 - Energy

Red arrows and boxes labeled '1' and '2' indicate the following actions:

- Box '1' points to the 'Results' folder in the Links Manager.
- Box '2' points to the 'Computations' folder in the Links Manager.
- Box '1' also points to the 'Results' folder in the Links Manager.

The 3D model shows a cylinder with a hole, with a coordinate system (X, Y, Z) and a green arrow indicating a direction of force or displacement.

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.

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.

The screenshot displays the CATIA V5 interface for a contact analysis. The **Features Tree** on the left shows the model structure: Analysis Manager, Links Manager, Results, Computations, Link.1, CylPressFit (containing ws11outerCyl.1 and ws11innerCyl.1), Constraints (containing Fix.1, Surface contact.2, and Offset.4), Applications, Finite Element Model (containing Nodes and Elements with two OCTREE meshes), Properties.1 (containing two Material Property3D objects and Contact Connection.1), and Static Case. The **Contact Connection** dialog box is open, showing 'Name: Contact Connection.1', 'Supports: 1 Constraint', and 'Clearance: -0.01in'. Two 3D views of the cylindrical assembly are shown at the bottom, with red boxes highlighting the contact area. The bottom toolbar and status bar are also visible.

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.

The screenshot displays the CATIA V5 interface with the following elements:

- Analysis Manager:** Shows the project structure including Results, Computations, Link.1, and Applications.
- Links Manager:** Lists the results and computation paths.
- Link.1:** Points to the product file: C:\CATIA\Training\data\ws11CylPressFit.CATProduct.
- CylPressFit:** The main assembly structure containing:
 - ws11outerCyl (ws11outerCyl.1)
 - ws11innerCyl (ws11innerCyl.1)
 - Constraints:
 - Fix.1 (ws11outerCyl.1)
 - Surface contact.2 (ws11outerCyl.1,ws11innerCyl.1)
 - Offset.4 (ws11outerCyl.1,ws11innerCyl.1) - highlighted with a red box and arrow labeled '2'.
- Finite Element Model:** Shows the meshed model with:
 - Nodes and Elements
 - Properties.1:
 - Material Property3D.1
 - Material Property3D.2
 - Contact Connection.1
 - Smooth Connection.1 - highlighted with a red box and arrow labeled '3'.
 - Static Case

Two 3D views of the assembly are shown at the bottom. The left view is a top-down cross-section showing the inner and outer cylinders with a red dashed line indicating the offset constraint. The right view is a perspective view showing the smooth connection between the two cylinders, with a red box and arrow labeled '1' pointing to the Smooth Connection icon in the toolbar.

A 'Smooth Connection' dialog box is open, showing:

- Name: Smooth Connection.1
- Supports: 1 Constraint
- Buttons: OK, Cancel

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.

The screenshot shows the CATIA V5 software interface. The top menu bar includes Start, TeamPDM, File, Edit, View, Insert, Tools, Analyze, Window, and Help. The Analysis Manager tree on the left contains the following items:

- Analysis Manager
 - Links Manager
 - Results -> C:\ELFINI\pressFit\pressFit.CATAnalysisResults
 - Computations -> C:\ELFINI\pressFit\pressFit.CATAnalysisComputations
 - Link.1 -> C:\CATIA\Training\data\ws11CylPressFit.CATProduct
 - CylPressFit (highlighted with a red box and arrow labeled '1')
 - Finite Element Model (highlighted with a red box and arrow labeled '4')
 - Nodes and Elements
 - OCTREE Tetrahedron Mesh.1 : ws11outerCyl.1
 - OCTREE Tetrahedron Mesh.2 : ws11innerCyl.1
 - Contact Connection Mesh.1
 - Smooth Connection Mesh.1
 - Properties.1
 - Static Case

The 3D model on the right shows a grey cylindrical part with a mesh. A red box labeled '2' points to a center plug being moved. A red box labeled '3' points to the compass tool in the bottom right corner. A red box labeled '4' points to the Finite Element Model object in the tree. The bottom status bar shows 'PartBody selected'.

Step 5. Mesh globally and locally

Analysis Manager

- Links Manager
- Finite Element Model
 - Nodes and Elements
 - OCTREE Tetrahedron Mesh.1 : ws11outerCyl.1
 - Local Mesh Size
 - Local Mesh Sag
 - OCTREE Tetrahedron Mesh.2 : ws11innerCyl.1

Local Mesh Size

Name: Local Mesh Size
Supports: 1 Face
Value: 0.2in

Local Mesh Sag

Name: Local Mesh Sag
Supports: 1 Face
Value: 0.02in

OCTREE Tetrahedron Mesh

Global | Local

Size: 0.3in
Sag: 0.03in

Element type

Linear

Parabolic

OK Cancel

1

2

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

Analysis Manager

- Links Manager
- Finite Element Model
 - Nodes and Elements
 - OCTREE Tetrahedron Mesh.1 : ws11outerCyl.1
 - Local Mesh Size
 - Local Mesh Sag
 - OCTREE Tetrahedron Mesh.2 : ws11innerCyl.1
 - Contact Connection Mesh.1
 - Smooth Connection Mesh.1
 - Properties.1
 - Static Case

OCTREE Tetrahedron Mesh

Global Local

Size 0.2in

Sag 0.02in

Element type

Linear

Parabolic

OK Cancel

1

Mesh the inner cylinder.

Steps:

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

Step 5. Mesh globally and locally

The screenshot shows the CATIA V5 interface with the Analysis Manager tree on the left and a 3D model of a cylindrical part on the right. The tree structure is as follows:

- Analysis Manager
 - Links Manager
 - Results -> C:\ELFINI\pressFit\pressFit.CATAnalysisResults
 - Computations -> C:\ELFINI\pressFit\pressFit.CATAnalysisComputations
 - Link.1 -> C:\CATIA\Training\data\ws11CylPressFit.CATProduct
 - CylPressFit (highlighted with a red box labeled '1')
 - Finite Element Model (highlighted with a red box labeled '3')
 - Nodes and Elements
 - OCTREE Tetrahedron Mesh.1 : ws11outerCyl.1
 - OCTREE Tetrahedron Mesh.2 : ws11innerCyl.1
 - Contact Connection Mesh.1
 - Smooth Connection Mesh.1
 - Properties.1
 - Static Case

The 3D model shows a cylindrical part with a mesh applied to its surface. A red arrow labeled '2' points to the 'Mesh' icon in the bottom toolbar.

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

The screenshot displays the CATIA V5 environment. On the left, the 'Analysis Manager' tree shows a 'Static Case' containing 'Restrains.1', 'Loads.1', and 'Static Case Solution.1'. The 'Restrains.1' folder is expanded to show 'Isostatic.1'. The main workspace shows a 3D model of a circular part with a central hole. A blue anchor icon is placed on the inner surface of the hole, indicating the application of an isostatic restraint. A red box with the number '1' and an arrow points to the Isostatic Restraint icon in the software's toolbar. A dialog box titled 'Isostatic Restraint' is open, showing 'Name Isostatic.1' and 'OK'/'Cancel' buttons.

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

Analysis Manager

- Links Manager
- Finite Element Model
 - Nodes and Elements
 - Properties.1
 - Static Case
 - Restraints.1
 - Isostatic.1
 - Loads.1
 - Static Case Solution.1
 - Sensors.1

Computation Resources Estimation

4e+002 s of CPU
4.9e+004 kilo-bytes of memory
7.9e+004 kilo-bytes of disk
Intel MKL(c) Library found: Intel(R) MKL V5.1.0

Do you want to continue the computation?

Yes No

Save all before computing.

Steps:

1. Compute All.

Should take about 7 minutes.

Step 8. Check global and local precision

Analysis Manager

- Links Manager
- Finite Element Model
 - Nodes and Elements
 - Properties.1
 - Static Case
 - Restraints.1
 - Loads.1
 - Static Case Solution.1
 - Deformed Mesh
 - Sensors.1

Image Fem Editor

Deformed

Display on Deformed Mesh

Mesh | Selections

- All
- OCTREE Tetrahedron Mesh.1 ; ws11outerCyl.1
- OCTREE Tetrahedron Mesh.2 ; ws11innerCyl.1
- Contact Connection Mesh.1
- Smooth Connection Mesh.1
- Isostatic.1

OK Cancel Help

1

2

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

Analysis Manager

- Links Manager
- Finite Element Model
 - Nodes and Elements
 - Properties.1
 - Static Case
 - Restraints.1
 - Loads.1
 - Static Case Solution.1
 - Deformed Mesh
 - Estimated local error
 - Extrema
 - Global Maximum.1
 - Sensors.1

Precision Location : Global
Estimated Precision : 10.7918 Btu
Strain Energy : 322.13 Btu
Global Estimated Error Rate : 12.8354 %

Estimated local error
Btu

1.03e-005
9.26e-006
8.24e-006
7.21e-006
6.18e-006
5.15e-006
4.12e-006
3.09e-006
2.06e-006
1.03e-006
5.18e-011

Estimated local error Global Maximum.1: 1.02943e-005 Btu

1 2 3 4

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

Analysis Manager

- Links Manager
- Finite Element Model
 - Nodes and Elements
 - Properties.1
 - Static Case
 - Restraints.1
 - Loads.1
 - Static Case Solution.1
 - Deformed Mesh
 - Estimated local error
 - Extrema
 - Global Maximum.1
 - Sensors.1
 - Adaptivity Process
 - Adaptivity Convergence.1
 - Adaptivities.1
 - Adaptivity Box.1

Determine maximum local error %.

Steps:

1. Select the adaptivity box icon.
2. Since the extrema is not anywhere 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.

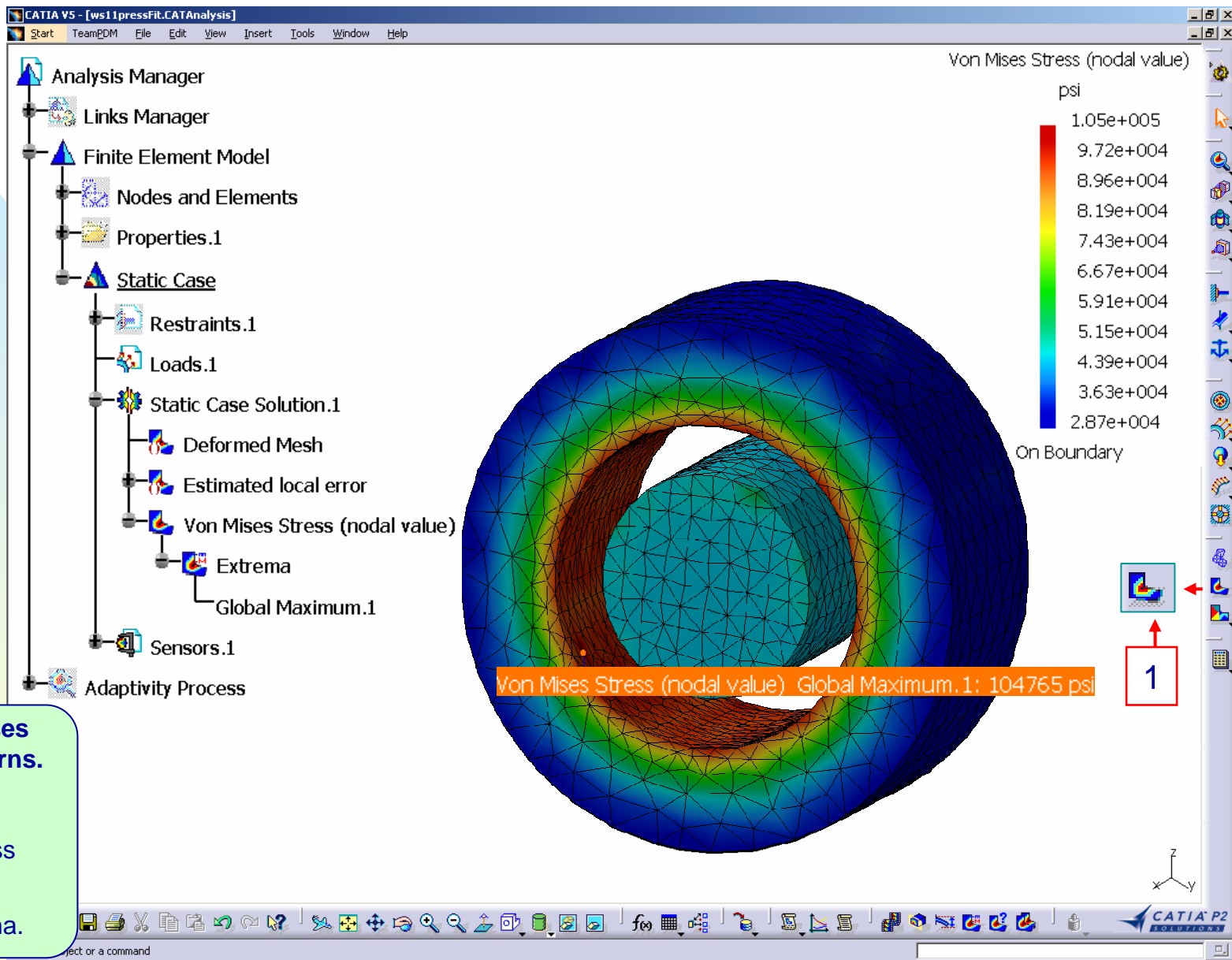
Adaptivity Box

Name	Adaptivity Box.1
Objective Error (%)	5
Solution	Static Case Solution.1
Local Error (%)	12.653

Select Extremum

OK Cancel

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 in the features tree and modify the filter and selections tabs as shown.

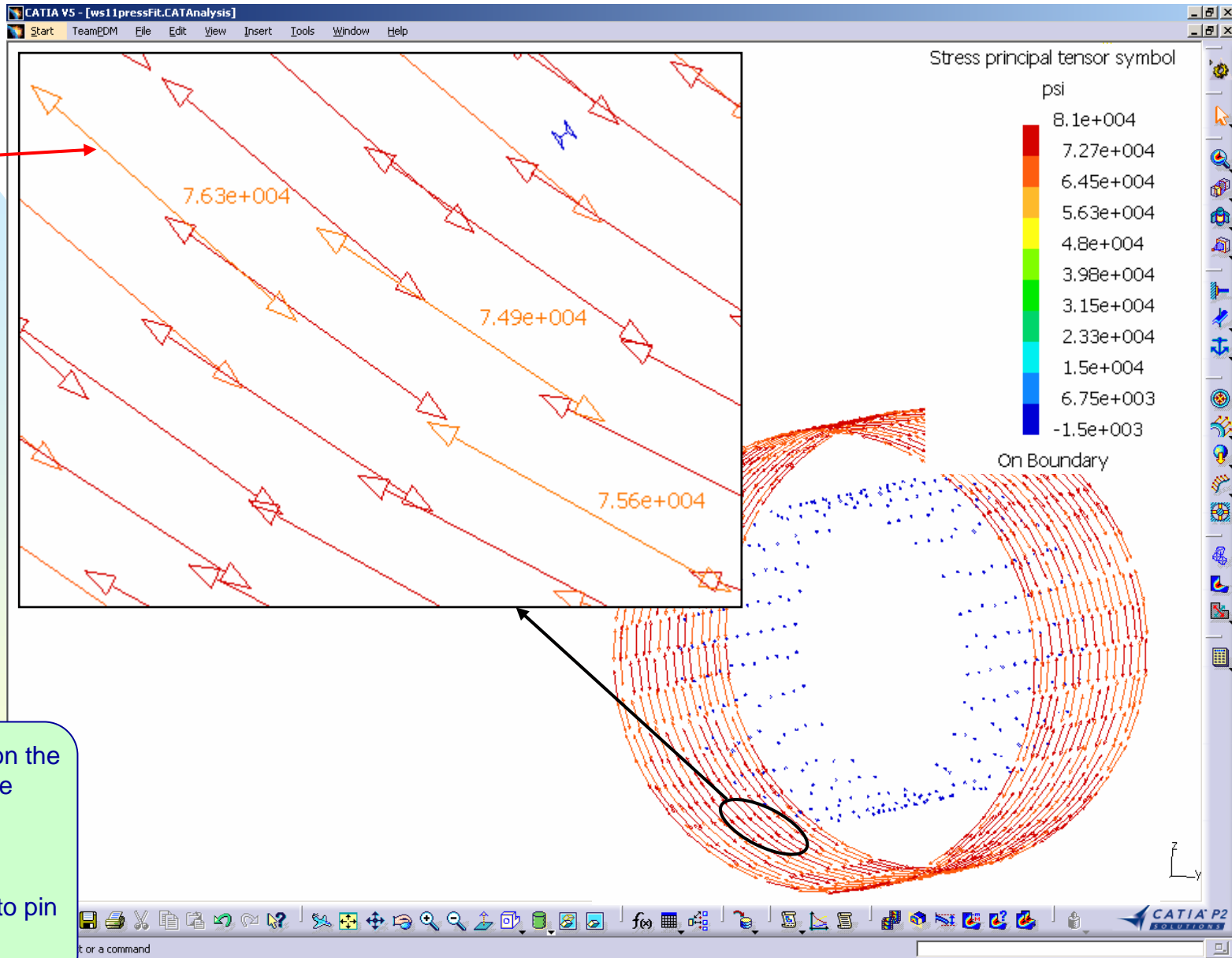
Image shown on next page.

The screenshot displays the CATIA V5 interface for a finite element analysis. The Analysis Manager tree on the left shows the following structure:

- Analysis Manager
 - Links Manager
 - Finite Element Model
 - Nodes and Elements
 - Properties.1
 - Static Case
 - Restraints.1
 - Loads.1
 - Static Case Solution.1
 - Deformed Mesh
 - Estimated local error
 - Von Mises Stress (nodal value)
 - Stress principal tensor symbol
 - Sensors.1

Two 'Image Edition' dialog boxes are open. The top dialog box has the 'Position' set to 'Node', 'Component' set to 'C1', and 'Lamina' set to '1'. The bottom dialog box has the 'Selections' tab active, with 'Contact Connection Mesh.1' selected in the list. A red box labeled '1' is around the selection list, and a red box labeled '2' is around the 'Stress principal tensor symbol' in the tree. Red arrows point from box '2' to the 'Image Edition' dialog boxes.

Step 9. Visualize final results



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

Steps:

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



Step 9. Visualize final results

Verify the contact pressure due to 0.010 inch interference fit.

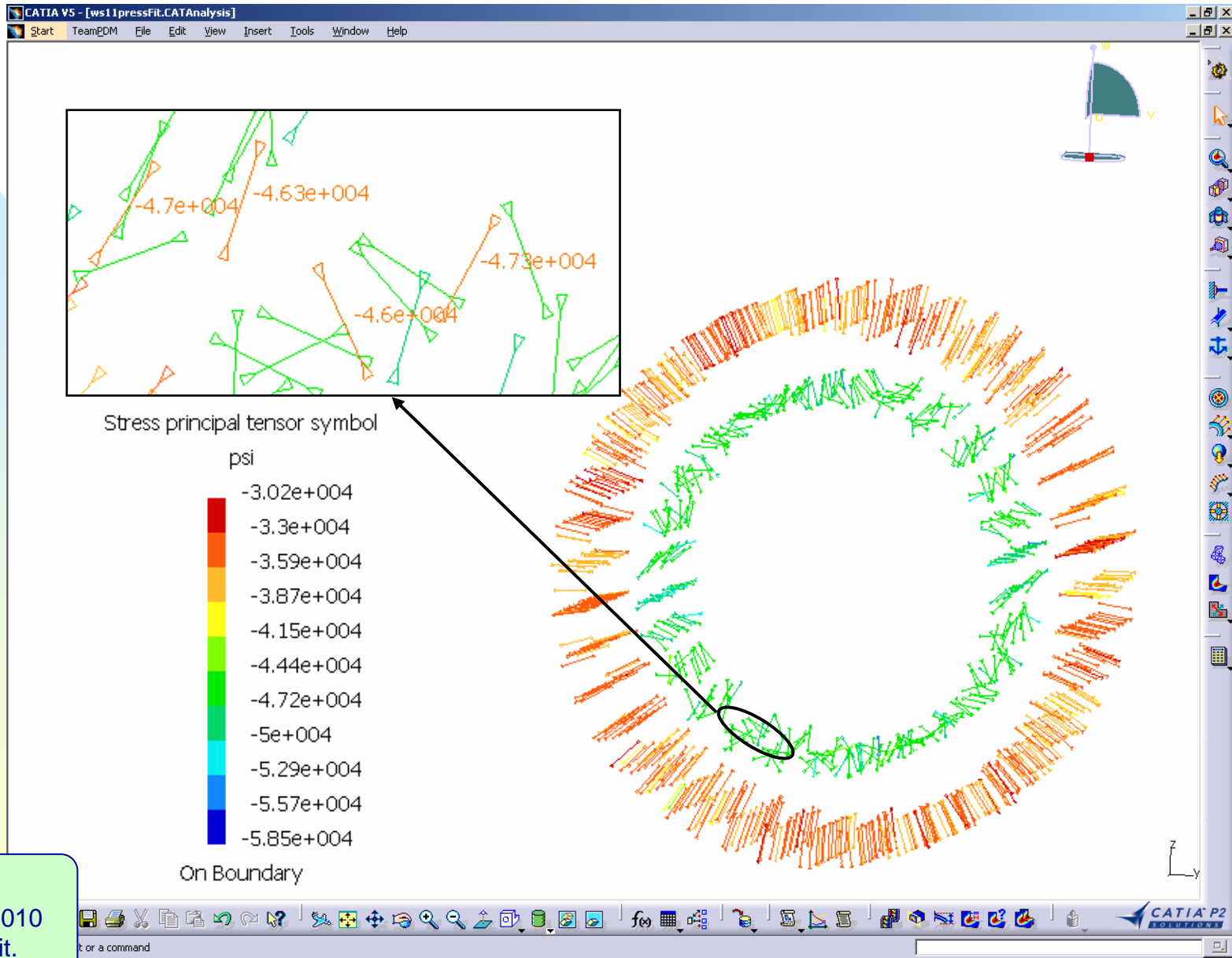
Steps:

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.

The screenshot displays the CATIA V5 software interface. On the left, the Analysis Manager tree is visible, showing a hierarchy of objects: Analysis Manager, Links Manager, Finite Element Model, Nodes and Elements, Properties.1, Static Case, Restraints.1, Loads.1, Static Case Solution.1, Deformed Mesh, Estimated local error, Von Mises Stress (nodal value), Stress principal tensor symbol, and Sensors.1. A red arrow points to the 'Stress principal tensor symbol' object, which is enclosed in a red box with the number '1'. Two 'Image Edition' dialog boxes are open. The top dialog box has the 'Filters' tab selected, showing 'Position: Node', 'Component: C3', 'Lamina: 1', and 'Repeat:'. The bottom dialog box has the 'Selections' tab selected, showing a list of mesh objects: 'All', 'OCTREE Tetrahedron Mesh.1 : ws11outerCyl.1', 'OCTREE Tetrahedron Mesh.2 : ws11innerCyl.1', 'Contact Connection Mesh.1' (highlighted), 'Smooth Connection Mesh.1', and 'Isostatic.1'. The bottom dialog box also shows 'Edition' options: 'Iso/Fringe', 'Symbol', and 'Axis System'. The bottom of the interface shows the standard Windows taskbar and the CATIA V5 toolbar.

Step 9. Visualize final results



Step 9. Visualize final results

■ Conclusions

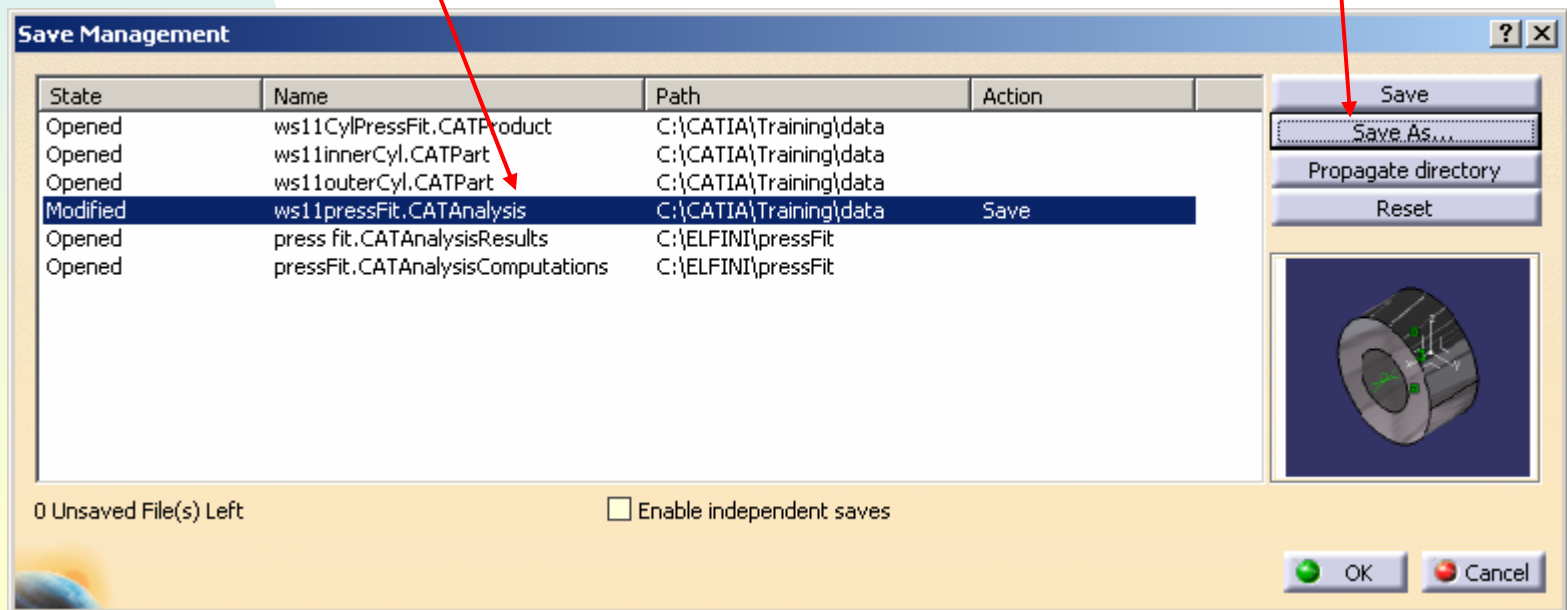
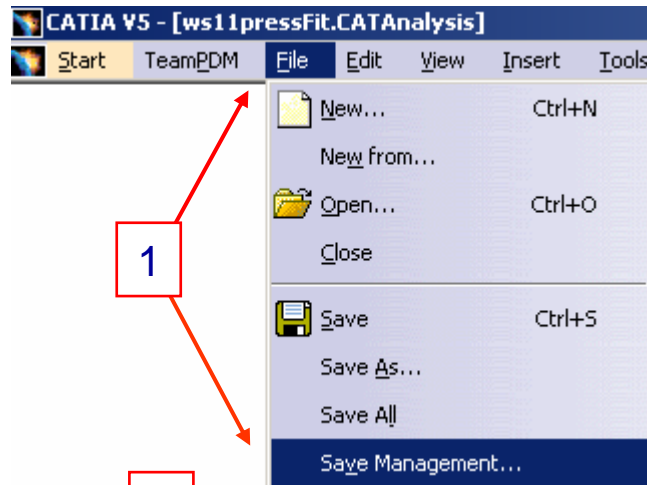
- ◆ CATIA V5 GSA workbench is validated for a press fit scenario.

	Hand Calculations	various Linear Global Mesh
Global % Precision error	NA	12.8 %
Local % Precision error	NA	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

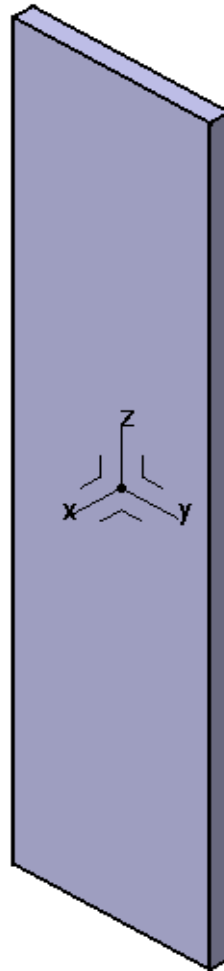
Steps:

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



WS12-1

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 = 10.15×10^6 psi

Poisson Ratio = .346

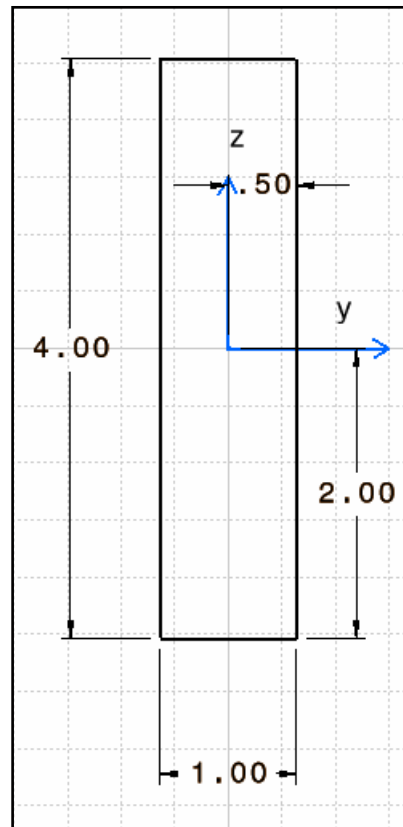
Density = .098 lb/in³

Yield Strength = 13778 psi

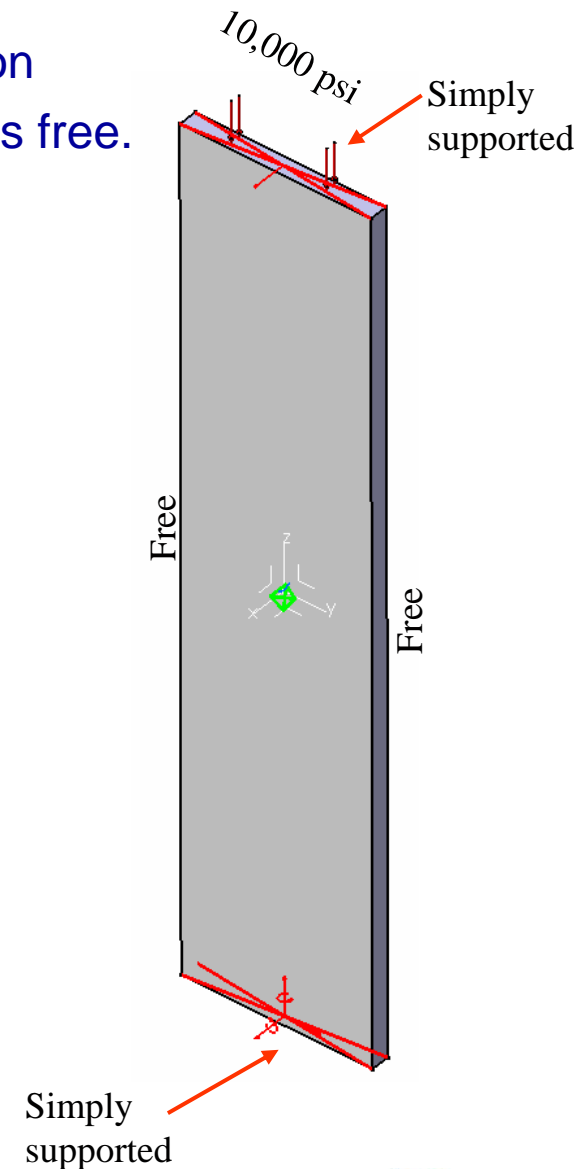
Design requirements:

Thickness, $t =$ 0.1 inch

Vertical Load, $w =$ 100 lbs/in



WS12-3

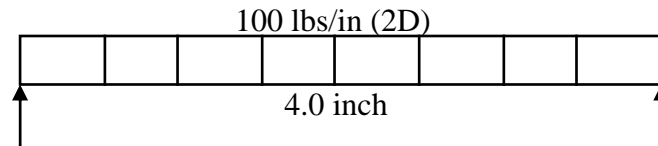


Hand calculations

- ◆ Critical load of a long slender column:

$$\text{Critical Load} = P_{cr} = \frac{\pi^2 \cdot E \cdot I_{min}}{L^2} \quad I_{min} = \frac{b \cdot t^3}{12} = \frac{1.0 \cdot 0.1^3}{12} = 0.00008333 \text{ inch}^4$$
$$P_{cr} = \frac{\pi^2 \cdot 10.15e6 \text{ psi} \cdot 0.00008333 \text{ inch}^4}{4.0^2 \text{ inch}} = 522 \text{ lbs}$$

- ◆ 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 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.

The screenshot displays the CATIA V5 interface for a finite element analysis. The Analysis Manager tree on the left shows the following structure:

- Analysis Manager
 - Links Manager
 - Finite Element Model
 - Nodes and Elements
 - OCTREE Tetrahedron Mesh.1 : ws12columnPlateBuck
 - Properties.1
 - Static Case
 - Restraints.1
 - Loads.1
 - Static Case Solution.1
 - Sensors.1

A red box with the number '1' and an arrow points to the 'OCTREE Tetrahedron Mesh.1 : ws12columnPlateBuck' entry in the tree. Another red arrow points from this box to a 3D model of a thin plate with a mesh. A dialog box titled 'OCTREE Tetrah...' is open, showing the following settings:

- Global | Local
- Size: 0.1in
- Sag: 0.01in
- Element type:
 - Linear
 - Parabolic
- OK | Cancel

Step 3. Create virtual parts

The screenshot displays the CATIA V5 interface for a finite element analysis. The tree view on the left shows the following structure:

- Analysis Manager
 - Links Manager
 - Finite Element Model
 - Nodes and Elements
 - Properties.1
 - Material Property3D.1
 - Rigid V. Part.1
 - Rigid V. Part.2
 - Static Case
 - Restraints.1
 - Loads.1
 - Static Case Solution.1
 - Sensors.1

The central 3D model shows a vertical plate with a coordinate system. Two red arrows point to the top and bottom faces of the plate, labeled with red boxes containing the numbers 1 and 2. Two 'Rigid V. Part' dialog boxes are shown. The first dialog box, labeled '1', has 'Name' set to 'Rigid V. Part.1' and 'Supports' set to '1 Face'. The second dialog box, labeled '2', has 'Name' set to 'Rigid V. Part.2' and 'Supports' set to '1 Face'. Both dialog boxes have 'Part Handler' set to 'No selection' and 'OK' and 'Cancel' buttons.

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.

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.

The screenshot displays the CATIA V5 interface for a finite element analysis. The left-hand tree structure shows the following hierarchy: Analysis Manager > Links Manager > Finite Element Model > Nodes and Elements > Properties.1 > Material Property3D.1 > Rigid V. Part.1 > Rigid V. Part.2 > Static Case > Restraints.1 > Restraint.1. The main 3D view shows a vertical plate with a red line indicating the top edge. A coordinate system is visible on the plate's surface. An inset image shows a close-up of the top edge of the plate with a red line and a coordinate system. The 'Advanced Restraint' dialog box is open, showing the following settings: Name: Restraint.1; Supports: 1 Virtual part; Axis System: Global; Type: Global; Display locally: unchecked. The 'Restrain Translation 1' checkbox is checked, while 'Restrain Translation 2', 'Restrain Translation 3', 'Restrain Rotation 1', 'Restrain Rotation 2', and 'Restrain Rotation 3' are unchecked. The 'OK' button is highlighted. A red arrow points from the top edge of the plate to the '1 Virtual part' support selection. Another red arrow points from the 'Restrain Translation 1' checkbox to the 'OK' button. A small icon in the top right corner of the software interface is labeled with a red box containing the number '1', and the 'OK' button is labeled with a red box containing the number '2'.

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.

The screenshot displays the CATIA V5 software interface for a finite element analysis. The left-hand tree view shows the model structure, including 'Analysis Manager', 'Links Manager', 'Finite Element Model', 'Nodes and Elements', 'Properties.1', 'Material Property3D.1', 'Rigid V. Part.1', 'Rigid V. Part.2', 'Static Case', 'Restraints.1', and 'Loads.1'. The 3D model shows a plate with a coordinate system and red arrows indicating the application of restraints. The 'Advanced Restraint' dialog box is open, showing the following details:

- Name: Restraint.2
- Supports: 1 Virtual part
- Axis System: (empty)
- Type: Global
- Display locally:
- Restrained options:
 - Restrain Translation 1
 - Restrain Translation 2
 - Restrain Translation 3
 - Restrain Rotation 1
 - Restrain Rotation 2
 - Restrain Rotation 3

Red arrows and boxes highlight the 'Advanced Restraint' icon in the toolbar (labeled '1') and the 'Restrain Translation 1' option in the dialog box (labeled '2').

Step 4. Apply a force

Analysis Manager

- Links Manager
- Finite Element Model
 - Nodes and Elements
 - Properties.1
 - Material Property3D.1
 - Rigid V. Part.1
 - Rigid V. Part.2
 - Static Case
 - Restraints.1
 - Loads.1
 - Surface Force Density.1

Surface Force Density

Name: Surface Force Density.1

Supports: 1 Face

Axis System

Type: Global

Display locally

Force Vector

Norm: 10000psi

X: 0psi

Y: 0psi

Z: -10000psi

Optional Elements

Data Mapping

OK Cancel

1

2

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

The screenshot displays the CATIA V5 interface for a finite element analysis. The Analysis Manager tree on the left shows a hierarchy: Analysis Manager > Links Manager > Finite Element Model > Nodes and Elements > Properties.1 > Static Case > Restraints.1 > Loads.1 > ColumnPlateSolution > Sensors.1 > Buckling Case > ColumnPlateSolution > Buckling Case Solution.2 > Sensors.3. Two red arrows originate from a box labeled '1' and point to the 'ColumnPlateSolution' node under the 'Static Case' and the 'ColumnPlateSolution' node under the 'Buckling Case'. A second red arrow, labeled '2', points to the 'Buckling Case Solution.2' node. Two dialog boxes are open: 'Static solution Parameters' and 'Buckling solution Parameters'. The 'Static solution Parameters' dialog shows the 'Method' set to 'gauss' and 'Gradient parameters' with 'maximum iteration number' at 0 and 'accuracy' at 1e-008. The 'Buckling solution Parameters' dialog shows 'Number of modes' set to 10, 'Method' set to 'gauss', and 'Dynamic parameters' with 'maximum iteration number' at 50 and 'accuracy' at 0.001. On the right, a 3D model of a vertical rectangular plate is shown with a coordinate system (x, y, z) at its base.

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

Analysis Manager

- Links Manager
- Finite Element Model
 - Nodes and Elements
 - Properties.1
 - Static Case
 - Buckling Case
 - ColumnPlateSolution
 - Buckling Case Solution.3
 - Sensors.4

Computation Resources Estimation

3e+001 s of CPU
5.85e+003 kilo-bytes of memory
2.92e+004 kilo-bytes of disk
Intel MKL(c) Library found: Intel(R) MKL V5.1.0

Do you want to continue the computation?

Yes No

1

Select an object or a command

CATIA V5 - [ws12columnPlateBuck.CATAnalysis]

Start TeamPDM File Edit View Insert Tools Window Help

CATIA V5 SOLUTIONS

Save first.

Steps:

1. Compute all objects.

Step 8. Check global and local precision

Analysis Manager

- Links Manager
- Finite Element Model
 - Nodes and Elements
 - Properties.1
 - Static Case
 - Buckling Case**
 - ColumnPlateSolution
 - Estimated local error
 - Buckling Case Solution.3
 - Deformed Mesh
 - Sensors.4

Precision Location : Global
Estimated Precision : 0.000110767 Btu
Strain Energy : 0.220174 Btu
Global Estimated Error Rate : 1.58582 %

1a
2a
2b
1b

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).

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.

Analysis Manager

- Links Manager
- Finite Element Model
 - Nodes and Elements
 - Properties.1
 - Static Case
 - Buckling Case
 - ColumnPlateSolution
 - Estimated local error
 - Extrema
 - Global Maximum.1
 - Buckling Case Solution.3
 - Deformed Mesh
 - Sensors.4
 - Adaptivity Process
 - Adaptivity Convergence.1
 - Adaptivities.1
 - Adaptivity Box.1

Estimated local error Global Maximum.1: 1.44148e-009 Btu

Adaptivity Box

Name: Adaptivity Box.1

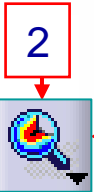
Objective Error (%): 3

Solution:

Local Error (%): 0

Select Extremum

OK Cancel



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.

The screenshot shows the CATIA V5 interface with the Analysis Manager tree on the left. The tree is expanded to show the Buckling Case, which is highlighted with a red box and the number 1. A context menu is open over the Buckling Case, with 'Set As Current Case' selected. The Sensors.4 folder is highlighted with a red box and the number 2. A context menu is open over the Sensors.4 folder, with 'Create Sensor' selected. The Sensor Creation dialog is open, with 'bucklingfactors' selected in the Functions list, highlighted with a red box and the number 3. The 'OK' button is highlighted with a red box and the number 4. The Output Values dialog is open, showing a table of buckling factors. The first row, 'buckfactor1' with a value of 0.519465625, is circled in red.

Parameter	Value
buckfactor1	0.519465625
buckfactor2	2.1823283
buckfactor3	3.996623039
buckfactor4	4.766805649
buckfactor5	8.472147942
buckfactor6	12.188117981
buckfactor7	13.137681007
buckfactor8	16.271015167
buckfactor9	18.459434509
buckfactor10	18.659662247

$$\text{Critical Load} = (.519)(10,000 \text{ psi}) = 5190 \text{ psi}$$

Compare with hand calculations:

$$\text{Critical Load} = (5190 \text{ psi})(0.1 \text{ inch}^2) = 519 \text{ lbs.}$$

$$\text{Hand calculations} = 522 \text{ lbs.}$$

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.

The screenshot shows the CATIA V5 software interface. The Analysis Manager tree on the left contains the following objects: Analysis Manager, Links Manager, Finite Element Model, Nodes and Elements, Properties.1, Static Case, Restraints.1 (containing Restraint.1 and Restraint.2), Loads.1, ColumnPlateSolution, Sensors.1, Buckling Case, SimplySupportedBeam (containing Restraints.1 with Restraint.1 and Isostatic.1), Loads.2, Masses.1, Static Case Solution.2, Sensors.5, and Energy. A context menu is open over the 'Static Case' object, showing options: Object, Static Case (highlighted with a red arrow and box '1'), Frequency Case, and Buckling Case. A 'Static Case' dialog box is open in the center, with 'Restraints.1' selected under the 'Restraints' section (indicated by a red arrow and box '2'). The 'SimplySupportedBeam' object is highlighted in the tree with a red arrow and box '3'. The bottom status bar reads 'Select an object or a command'.

Step 9. Visualize final results

CATIA V5 - [ws12columnPlateBuck.CATAnalysis]

Start TeamPDM File Edit View Insert Tools Window Help

Analysis Manager

- Links Manager
- Finite Element Model
 - Nodes and Elements
 - Properties.1
 - Static Case
 - Buckling Case
 - SimplySupportedBeam
 - Restraints.1
 - Loads.2
 - Masses.1
 - Static Case Solution.2
 - Translational displaceme
 - Sensors.5
 - Energy
 - Adaptivity Process

Translational displacement vector

in

0.398

0.358

0.318

0.278

0.239

0.199

0.159

0.119

0.0796

0.0398

0

On Boundary

1

Select an object or a command

CATIA V5 SOLUTIONS

Compare hand calculation displacements.

Steps

1. Select the Displacement icon.

Hand calc's = .394 in.

FEA = .398 in.

Looks good.

Step 9. Visualize final results

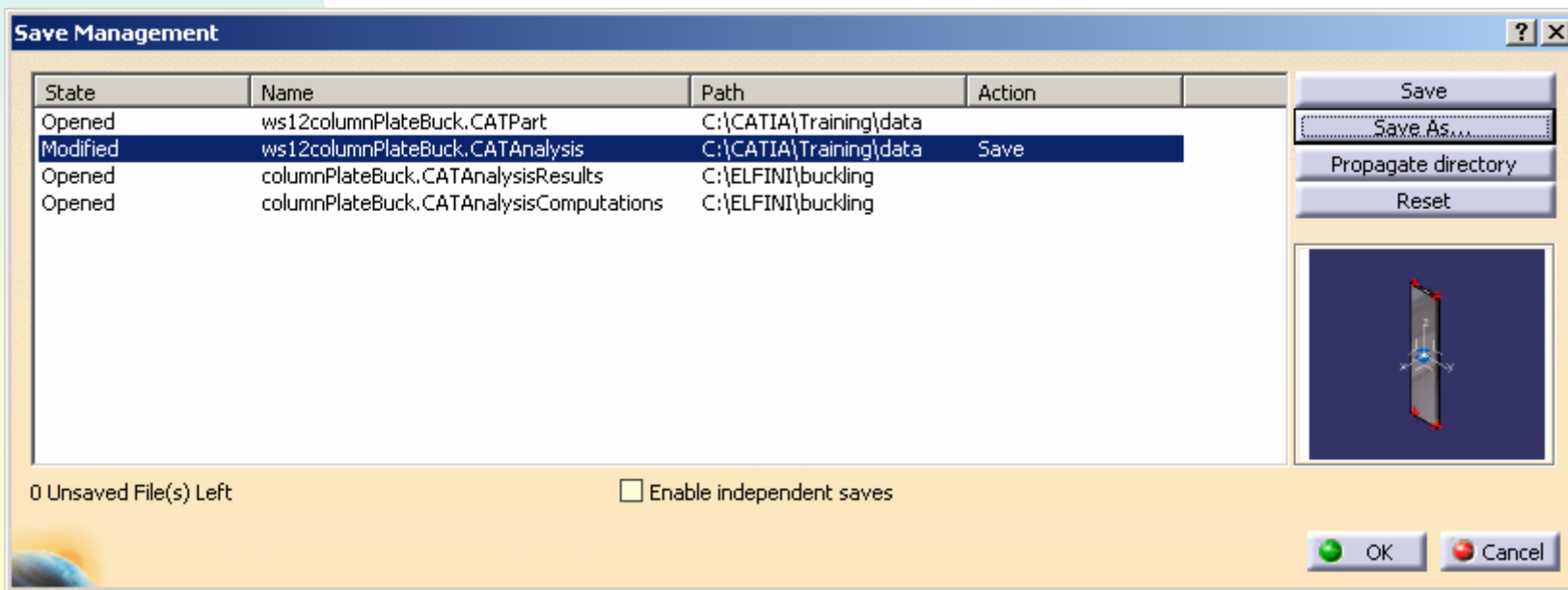
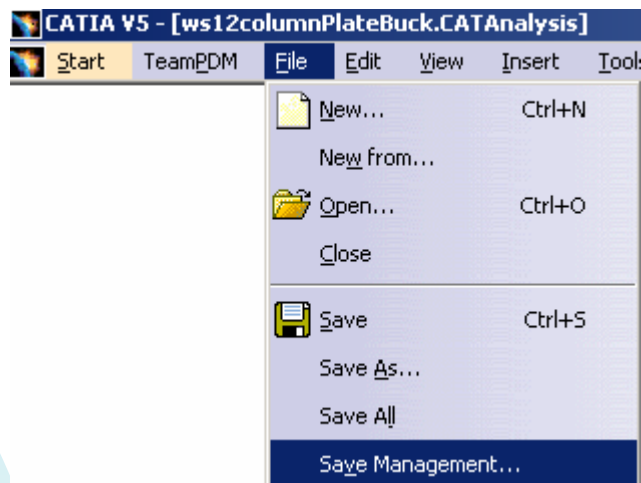
■ Conclusions

- ◆ 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	NA	1.58 %
Local % Precision error	NA	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

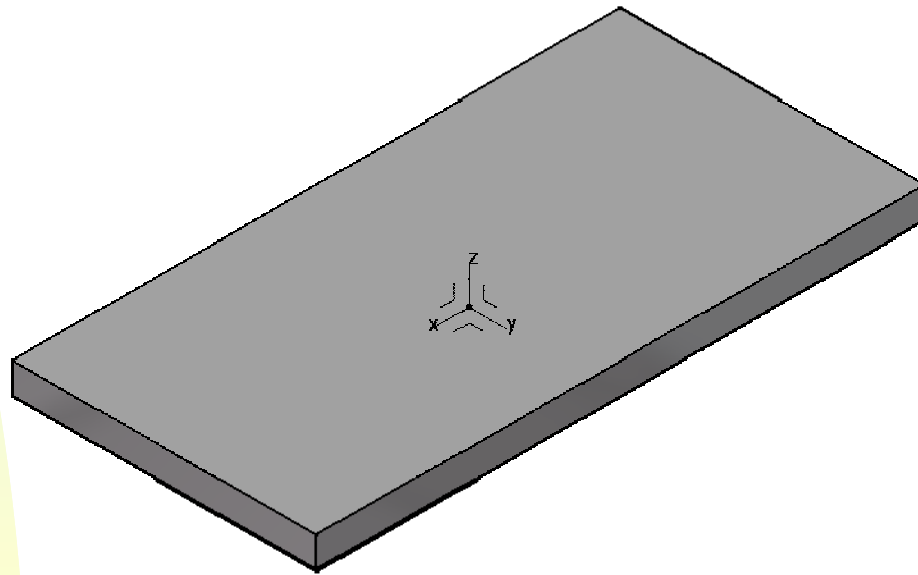
Step 10. Save the analysis document

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**



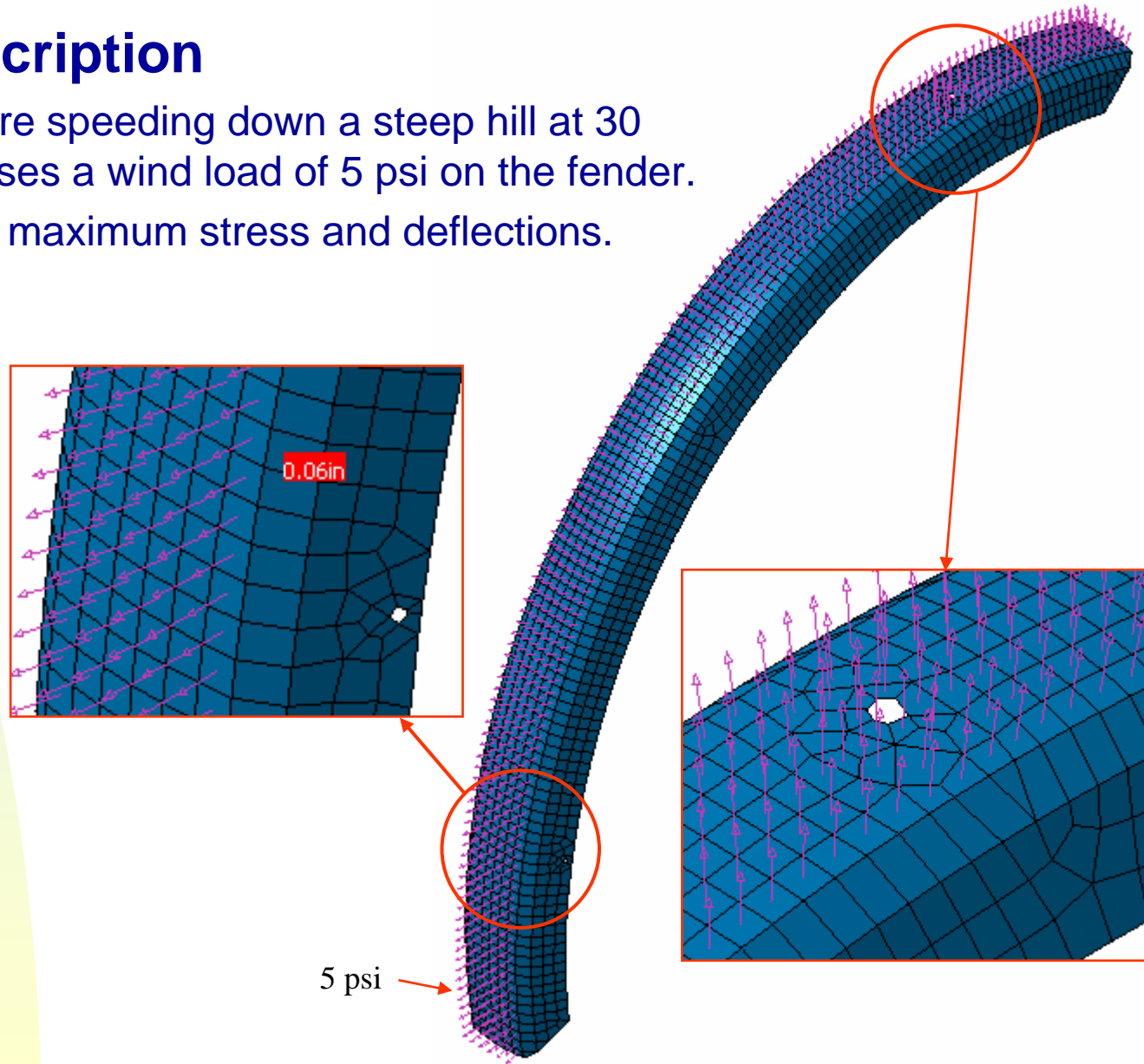
WORKSHOP 13

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.

The screenshot displays the CATIA V5 software interface. The main menu bar includes 'Start', 'TeamPDM', 'File', 'Edit', 'View', 'Insert', 'Tools', 'Window', and 'Help'. The 'Analysis & Simulation' menu is expanded, showing 'Advanced Meshing Tools' (highlighted with a red box and arrow labeled '2'), 'Generative Structural Analysis', and 'Analysis Connections'. The 'Analysis Manager' tree on the left shows a 'Finite Element Model' containing 'Nodes and Elements', 'Properties.1', 'Static Case', 'Restrains.1', 'Loads.1', 'Static Case Solution.1', 'Sensors.1', and 'Energy'. A red box and arrow labeled '1' points to the 'Static Case' folder. The 'New Analysis Case' dialog box is open, showing 'Static Analysis' selected in a list, with 'Frequency Analysis' and 'Free Frequency Analysis' also visible. The dialog includes an 'OK' button, a 'Cancel' button, and a 'Help' button. A green 3D model of a curved fender part is visible in the background.

Step 2. Specify global surface mesh parameters

Analysis Manager

- Links Manager
- Finite Element Model
 - Nodes and Elements
 - Smart Surface Mesh.1
 - Properties.1
 - Static Case

Free edges are displayed with a green color.

Unspecified edges are displayed with a white color.

Gaps would be displayed with a pink color.

Mesh Parameters

Element shape	
Mesh size	0.25in
Maximum sag	0.025in
Offset	0in
Min holes size	0.1in
<input checked="" type="checkbox"/> Merge during simplification	
Minimum size	0.079in

More >>>

OK Cancel

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

Analysis Manager

- Links Manager
- Finite Element Model
 - Nodes and Elements
 - Smart Surface Mesh.1
 - Constrained Geometries
 - Properties.1
 - Static Case

Add/Remove Constraints

N°	Geometry	Status
1	Split. 4/Sweep. 1/Edge	Constrained
2	Split. 4/Sweep. 1/Edge	Constrained
3	Split. 4/Sweep. 1/Edge	Constrained
4	Split. 4/Sweep. 1/Edge	Constrained

Remove Remove All Zoom

OK Cancel

Select or a command

CATIA V5 [ws13fender.CATAnalysis]

Start TeamPDM File Edit View Insert Tools Window Help

CATIA V5 SOLUTIONS

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 1/2 circle.

You can specify node distributions on constrained or free edges.

Analysis Manager

- Links Manager
- Finite Element Model
 - Nodes and Elements
 - Smart Surface Mesh.1
 - Constrained Geometries
 - Imposed Nodes
 - Properties.1
 - Static Case

Edit Nodes Distribution

Uniform

Number of nodes: 5

Size: 0.591in

OK Apply Cancel

Imposed Nodes

N°	Geometry	Description
1	Split.4/Split.3/Edge	Uniform, 5 nodes
2	Split.4/Split.3/Edge	Uniform, 5 nodes
3	Split.4/Edge	Uniform, 5 nodes
4	Split.4/Edge	Uniform, 5 nodes
5	Split.4/Edge	Uniform, 5 nodes
6	Split.4/Edge	Uniform, 5 nodes

Edit Remove Remove All Zoom

OK Cancel

Step 5. Mesh the part

Analysis Manager

- Links Manager
- Finite Element Model
 - Nodes and Elements
 - Smart Surface Mesh.1
 - Constrained Geometries
 - Imposed Nodes
 - Properties.1
 - Static Case

Mesh The Part

Frontal quadrangle, size = 6.35 mm
1825 nodes created
1694 elements created
Mesh OK

OK

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.

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.

The screenshot shows the CATIA V5 interface with a finite element model. The Analysis Manager tree on the left shows the following structure:

- Analysis Manager
 - Links Manager
 - Finite Element Model
 - Nodes and Elements
 - Smart Surface Mesh.1
 - Constrained Geometries
 - Imposed Nodes
 - Properties.1
 - Static Case

The main window displays a 3D model of a mechanical part with a mesh. The mesh is color-coded: red for the worst elements, yellow for moderate quality, and green for good quality. A circular hole is visible in the center of the part.

The Quality Analysis dialog box is open, showing the following options:

- Select: Taper, Skewness, Distortion, Jacobian, Warp Factor, Warp Angle, Skew Angle, Stretch, Min. Length, Max. Length, Shape Factor, Length Ratio
- Options: All, None
- Buttons: OK, Apply

The Worst Elements Browser dialog box is open, showing the following options:

- Number of worst elements to look at: 10
- Show single analysis window:
- AutoFocus on element:
- Buttons: Prev, Next, Current 1, OK

Red arrows indicate the following 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

The screenshot shows the CATIA V5 interface with the 'Edit Nodes Distribution' dialog box open. The dialog box has a dropdown menu set to 'Uniform', a radio button for 'Number of nodes' set to '2', and a text field for 'Size' set to '0.2in'. There are 'OK', 'Apply', and 'Cancel' buttons at the bottom. The 'Analysis Manager' tree on the left shows the 'Imposed Nodes' icon selected. Two views of a meshed part are shown: the left view shows a good mesh, and the right view shows a mesh with a red triangle indicating a problem. Red arrows labeled '1' and '2' point to the 'Imposed Nodes' icon and the boundary edges, respectively.

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

CATIA V5 - [ws13fender.CATAnalysis]

Analysis Manager
Links Manager
Finite Element Model
Nodes and Elements
Smart Surface Mesh.1
Constrained Geometries
Imposed Nodes
Properties.1
Static Case

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.
3. Notice the Mesh methods that are available to you here.

Re-Mesh A Domain

Mesh method: Front trias
Mesh size: 0.25in
 Impact neighbour domains

OK Apply Close

Front quads
Front trias
Mapped quads
Mapped Free quads
Bead quads
Half Bead quads
Projection

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.

CATIA V5 - [ws13fender.CATAnalysis]

Start TeamPDM File Edit View Insert Tools Window Help

Analysis Manager

Links Manager

Finite Element Model

Nodes and Elements

Smart Surface Mesh.1

Constrained Geometries

Imposed Nodes

Properties.1

Static Case

2

Before

1

3

Remove edge

Condense nodes

Insert node

Mesh Editing Options

Smooth around modifications

Combine around modifications

Swap around modifications

Propagate to neighbour domains

Global Optimization

OK

Select an object or a command

CATIA P2 SOLUTIONS

Step 6. Check mesh quality and repair

Analysis Manager
Links Manager
Finite Element Model
Nodes and Elements
Smart Surface Mesh.1
Constrained Geometries
Imposed Nodes
Properties.1
Static Case

Before

Step One

Step Two

After

1

2

Select or a command

CATIA V5 - [ws13fender.CATAnalysis]
Start TeamPDM File Edit View Insert Tools Window Help

MSC SOFTWARE
SIMULATING REALITY

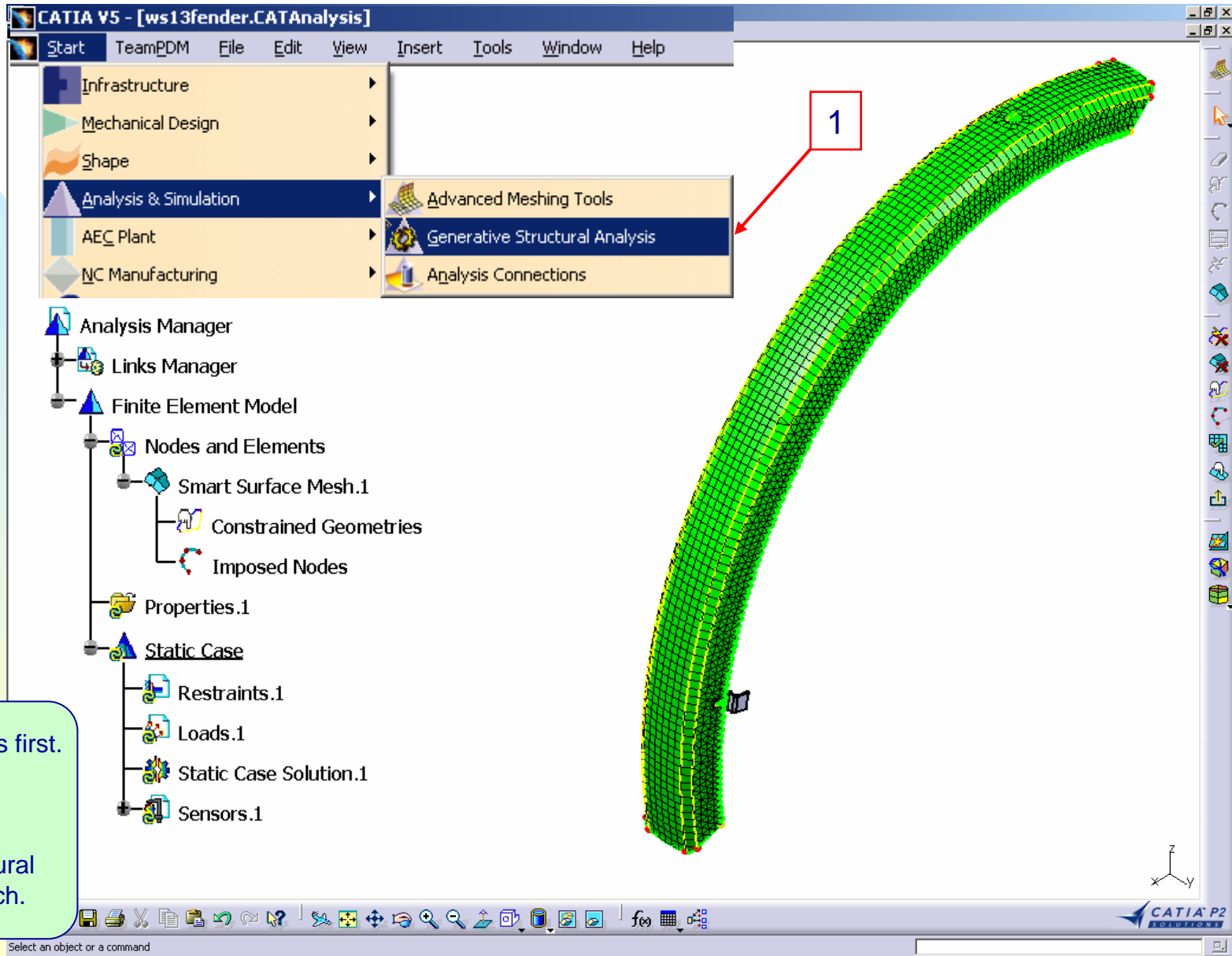
Manually Edit mesh
(cont.).

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

The screenshot displays the CATIA V5 software interface. On the left, the Analysis Manager tree is visible, showing the following structure:

- Analysis Manager
 - Links Manager
 - Results -> C:\ELFINI\surf\fender.CATAnalysisResults
 - Computations -> C:\ELFINI\surf\fender.CATAnalysisComputations
 - Link.1 -> C:\CATIA\Training\data\ws13fender.CATPart
 - Finite Element Model
 - Nodes and Elements
 - Smart Surface Mesh.1
 - Constrained Geometries
 - Imposed Nodes
 - Properties.1
 - Material Property2D.1
 - Static Case
 - Restraints.1
 - Loads.1
 - Static Case Solution.1
 - Sensors.1

On the right, a 3D view shows a green curved part. A coordinate system (X, Y, Z) is visible in the bottom right corner.

The Property2D dialog box is open, showing the following fields:

- Name: Material Property2D.1
- Supports: 1 Body
- Thickness: 0.06in

Buttons for OK and Cancel are visible at the bottom of the dialog box.

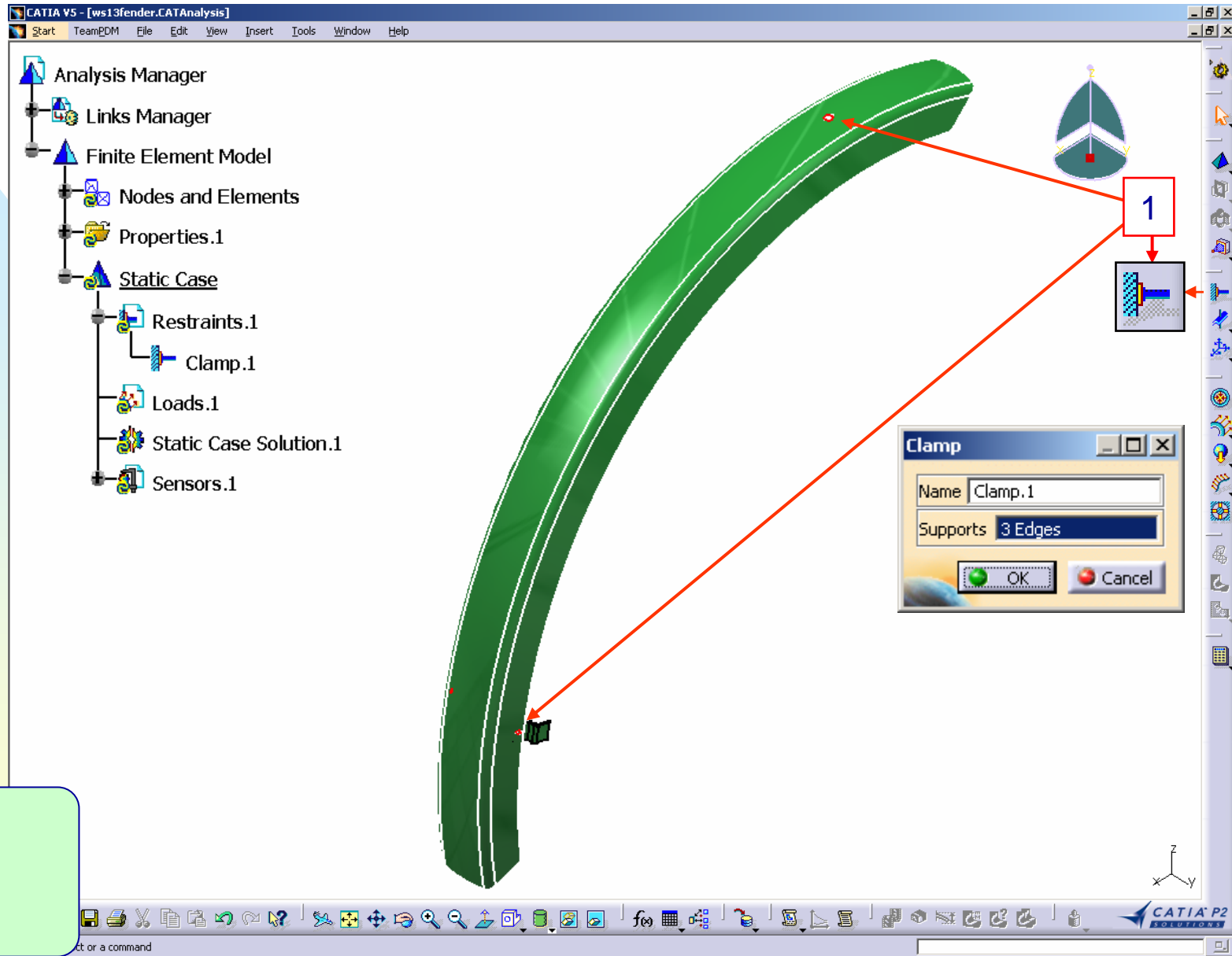
Red arrows and boxes labeled '1' and '2' point to the Results folder and the Material Property2D.1 feature in the tree, respectively.

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

The screenshot shows the CATIA V5 interface with the following elements:

- Tree View (Left):**
 - Analysis Manager
 - Links Manager
 - Finite Element Model
 - Nodes and Elements
 - Properties.1
 - Static Case
 - Restraints.1
 - Clamp.1
 - Loads.1
 - Pressure.1
 - Static Case Solution.1
 - Sensors.1

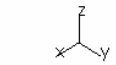
- 3D Model (Center):** A curved green part with a red arrow pointing to the outer surface.
- Pressure Dialog Box (Right):**
- Name: Pressure.1
- Supports: 1 Face
- Pressure: -5psi
- Optional Elements:
 - Data Mapping
- Buttons: OK, Cancel
- Toolbar (Right):** A red box with the number '1' and an arrow pointing to the Pressure icon.

Assume the pressure is from the inside out.

Steps:

1. Select the Pressure icon and the outside face as shown, key in -5psi.

Select or a command



CATIA V5 P2
SOLUTIONS

Step 11. Compute all

Analysis Manager

- Links Manager
- Finite Element Model
 - Nodes and Elements
 - Properties.1
 - Static Case
 - Restrains.1
 - Clamp.1
 - Loads.1
 - Pressure.1
 - Static Case Solution.1
 - Sensors.1

Computation Resources Estimation

4 s of CPU
2.4e+003 kilo-bytes of memory
7.37e+003 kilo-bytes of disk
Intel MKL(c) Library found: Intel(R) MKL V5.1.0

Do you want to continue the computation?

Yes No

1

Select an object or a command

Save first.

Steps:

1. Compute All.

Step 12. Check global precision

Analysis Manager

- Links Manager
- Finite Element Model
 - Nodes and Elements
 - Properties.1
 - Static Case
 - Restrains.1
 - Loads.1
 - Static Case Solution.1
 - Deformed Mesh
 - Estimated local error
 - Sensors.1

Estimated local error
Btu

1.08e-005
9.75e-006
8.67e-006
7.59e-006
6.5e-006
5.42e-006
4.33e-006
3.25e-006
2.17e-006
1.08e-006
2.45e-010

Precision Location : Global
Estimated Precision : 0.602471 Btu
Strain Energy : 5.85222 Btu
Global Estimated Error Rate : 22.1255 %

1a
2a
2b
1b

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.

Step 12. Check global precision

CATIA V5 - [ws13fender.CATAnalysis]

Analysis Manager

- Links Manager
- Finite Element Model
 - Nodes and Elements
 - Properties.1
 - Static Case
 - Restraints.1
 - Loads.1
 - Static Case Solution.1
 - Deformed Mesh
 - Estimated local error
 - Extrema
 - Global Maximum.1
 - Sensors.1

Estimated local error Global Maximum, 1: 1.08358e-005 Btu

1

Select an object or a command

CATIA P2 SOLUTIONS

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

The screenshot shows the CATIA V5 interface with the following elements:

- Menu Bar:** Start, TeamPDM, File, Edit, View, Insert, Tools, Window, Help.
- Tree View:** Infrastructure, Mechanical Design, Shape, Analysis & Simulation (highlighted), AEC Plant, NC Manufacturing. Under Analysis & Simulation, Advanced Meshing Tools is selected.
- 3D Model:** A curved part with a green mesh. A red arrow labeled '3' points to the 'Global Meshing Properties' icon in the top right toolbar.
- Mesh Parameters Dialog:** Shows settings for Element shape, Mesh size (0.1in), Maximum sag (0.01in), Offset (0in), Min holes size (0.1in), Merge during simplification (checked), and Minimum size (0.079in).
- Warning Dialog:** A warning message: "Editing the mesh part will remove the existing mesh. Continue anyway?" with Yes and No buttons.
- Annotations:** Red boxes with numbers 1, 2, and 3. Box 1 points to the 'Advanced Meshing Tools' menu item. Box 2 points to 'Smart Surface Mesh.1' in the tree view. Box 3 points to the 'Global Meshing Properties' icon.

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

CATIA V5 - [ws13fender.CATAnalysis]

Start TeamPDM File Edit View Insert Tools Window Help

Analysis Manager

- Links Manager
- Finite Element Model
 - Nodes and Elements
 - Smart Surface Mesh.1
 - Constrained Geometries
 - Imposed Nodes
 - Properties.1
 - Static Case
 - Restraints.1
 - Loads.1
 - Static Case Solution.1
 - Sensors.1

Mesh The Part

Frontal quadrangle, size = 2.54 mm
11752 nodes created
11430 elements created
Mesh OK

OK

1

Smart Surface Mesh.1/Nodes and Elements/Finite Element Model selected

Refine the global surface mesh (cont.)

Steps:

1. Select the Mesh The Part icon, select OK.

Then go back to the GSA workbench.

Step 14. Visualize final results

Analysis Manager

- Links Manager
- Finite Element Model
 - Nodes and Elements
 - Properties.1
 - Static Case
 - Restraints.1
 - Loads.1
 - Static Case Solution.1
 - Deformed Mesh
 - Estimated local error **2** → Estimated local error Global Maximum.1: 8.76674e-006 Btu
 - Extrema
 - Global Maximum.1
 - Sensors.1

Computation Resources Estimation

6e+001 s of CPU
1.33e+004 kilo-bytes of memory
7.24e+004 kilo-bytes of disk
Intel MKL(c) Library found: Intel(R) MKL V5.1.0

Do you want to continue the computation?

Yes No

1 →

Precision Location : Global
Estimated Precision : 0.331126 Btu
Strain Energy : 5.96468 Btu
Global Estimated Error Rate : 16.434 %

Save first.

Steps

1. Compute All.
2. Find the Estimated Global error again. Good, below 20%.

Step 14. Visualize final results

CATIA V5 - [ws13fender.CATAnalysis]

Start TeamPDM File Edit View Insert Tools Window Help

Analysis Manager

- Links Manager
- Finite Element Model
 - Nodes and Elements
 - Properties.1
 - Static Case
 - Restraints.1
 - Loads.1
 - Static Case Solution.1
 - Deformed Mesh
 - Estimated local error
 - Translational displacement magnitude
 - Sensors.1

Translational displacement magnitude

in

0.598

0.538

0.478

0.419

0.359

0.299

0.239

0.179

0.12

0.0598

0

On Boundary

1

Select an object or a command

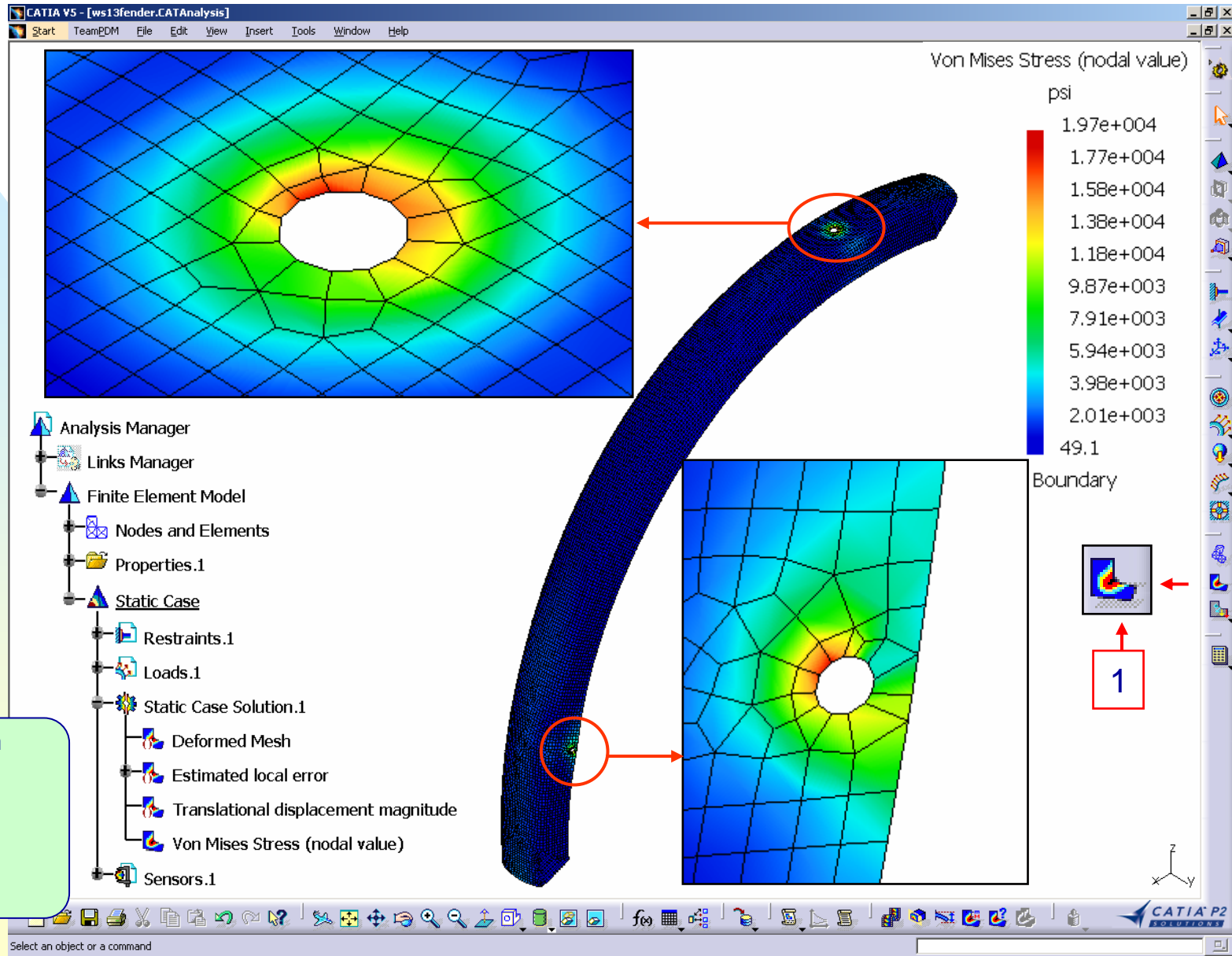
CATIA P2 SOLUTIONS

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.

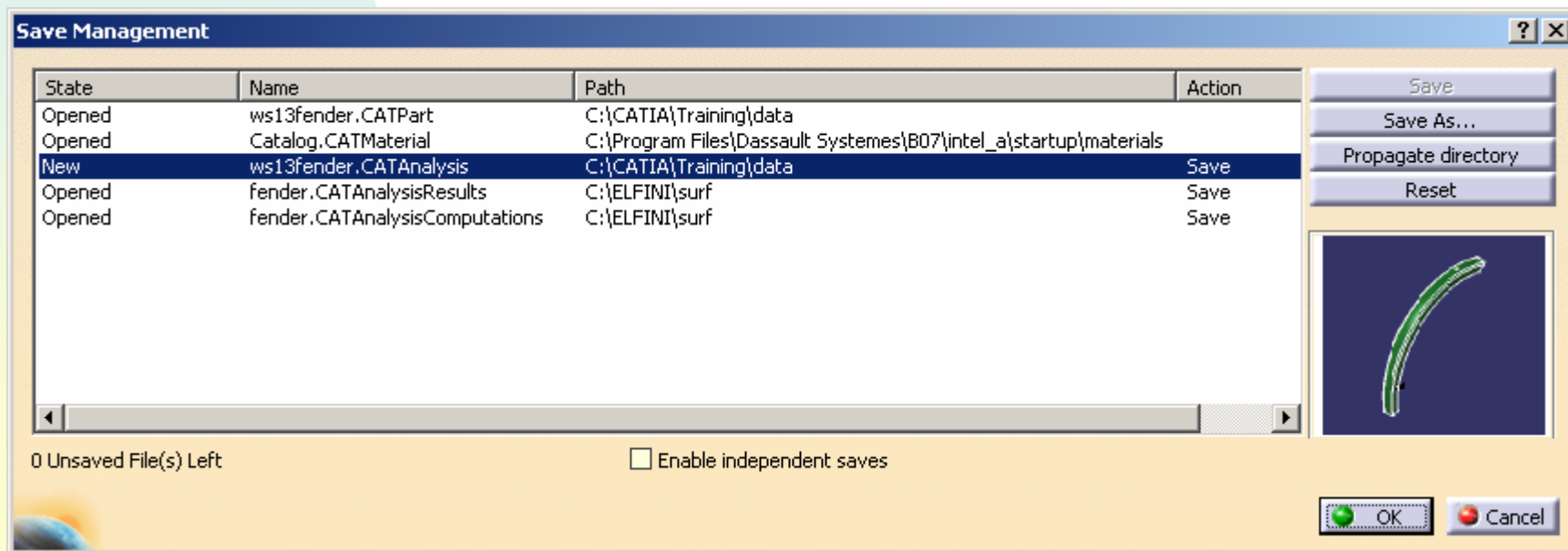
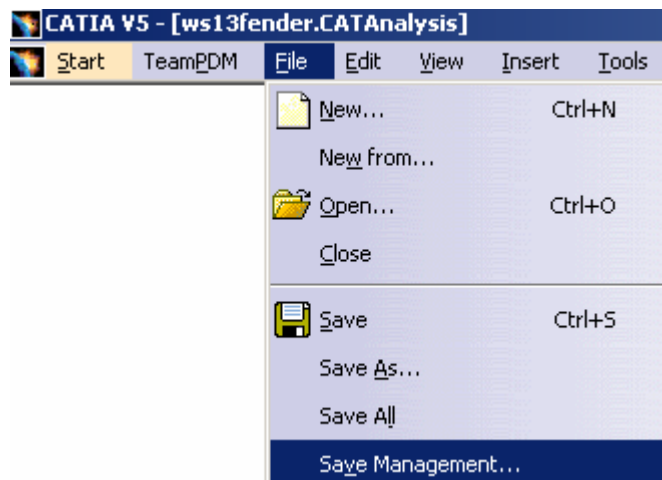
■ Conclusions

- ◆ 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

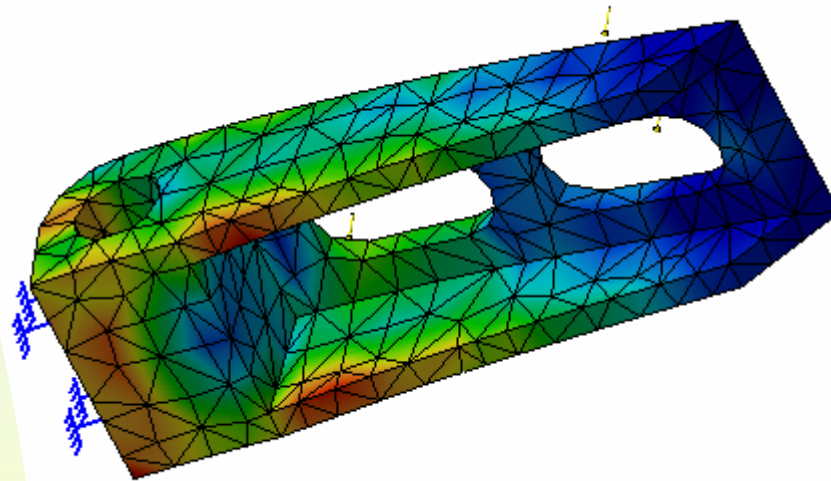
Step 15. Save the analysis document

Save your documents



WORKSHOP 14

KNOWLEDGEWARE



■ Problem Description

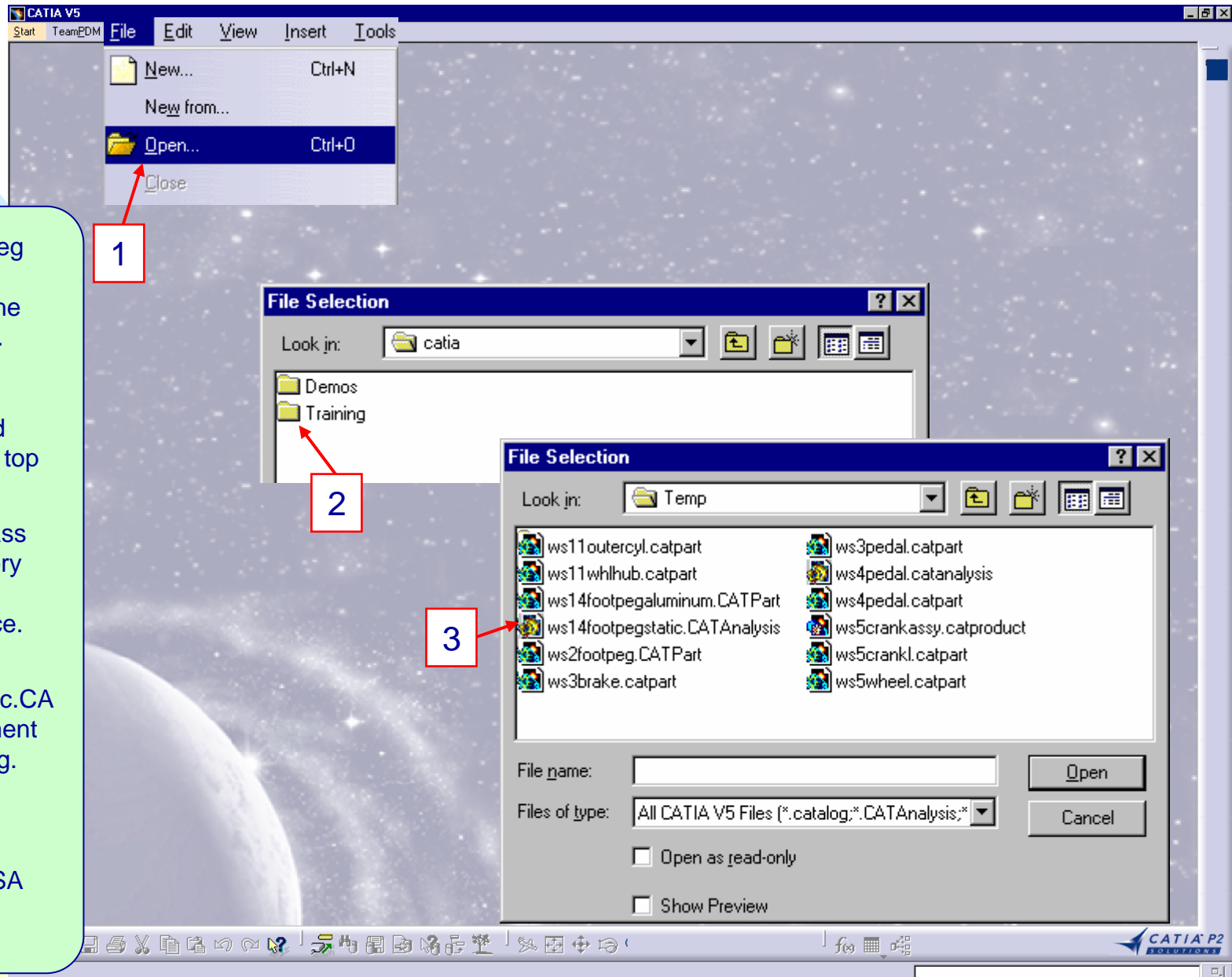
- ◆ The preliminary design of the ATV Foot Peg must be completed as soon as possible and must meet the given structural requirements. The design must not exceed the material yield strength under loading and it must not deform in a manner causing interference with other parts of the vehicle.
- ◆ An initial 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



1

2

3

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).

The screenshot shows the CATIA V5 interface for a static analysis. The specification tree on the left shows the following structure:

- Analysis Manager
 - Links Manager
 - Finite Element Model
 - Nodes and Elements
 - Properties.1
 - Static Case
 - Restraints.1
 - Clamp.1
 - Loads.1
 - Distributed Force.1
 - Static Case Solution.1
 - Von Mises Stress (nodal value)
 - Sensors.1

The 3D model shows a foot peg with a mesh and a color scale for Von Mises Stress (nodal value) in psi, ranging from 220 to 3.4e+003. The 'Sensor Creation' dialog box is open, showing the following functions:

- misesmax
- dispmax
- reaction
- globalerror
- dispmaxongroup

The 'OK' button is highlighted. Red boxes and arrows with numbers 1 through 5 indicate the sequence of actions described in the text.

Step 2. Create analysis sensors

The screenshot displays the CATIA V5 interface for a static analysis of a foot peg. The Analysis Manager tree on the left shows the following structure:

- Analysis Manager
 - Links Manager
 - Finite Element Model
 - Nodes and Elements
 - Properties.1
 - Static Case
 - Restraints.1
 - Clamp.1
 - Loads.1
 - Distributed Force.1
 - Static Case Solution.1
 - Von Mises Stress (nodal value)
 - Sensors.1 (expanded to show:
 - Energy (Default Energy sensor)
 - Maximum Displacement ("dispmax" sensor)
 - Maximum Von Mises ("misesmax" sensor))

The 3D model on the right shows the Von Mises Stress distribution on the foot peg. A color scale legend indicates stress values in psi, ranging from 855 (blue) to 3.4e+003 (red). The legend is labeled "Von Mises Stress (nodal value) psi" and "On Boundary".

Annotations in the image include:

- A red box with the number "1" pointing to the plus sign on the "Sensors.1" branch in the tree.
- A red box with the number "2" pointing to the expanded "Sensors.1" branch.
- Red boxes with labels: "Default Energy sensor" pointing to the Energy sensor, "dispmax" sensor pointing to the Maximum Displacement sensor, and "misesmax" sensor pointing to the Maximum Von Mises sensor.

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:

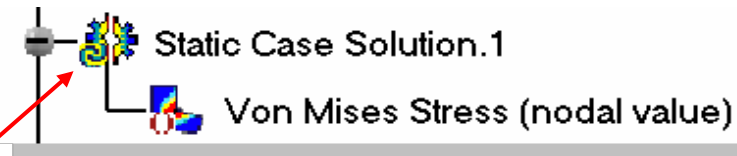
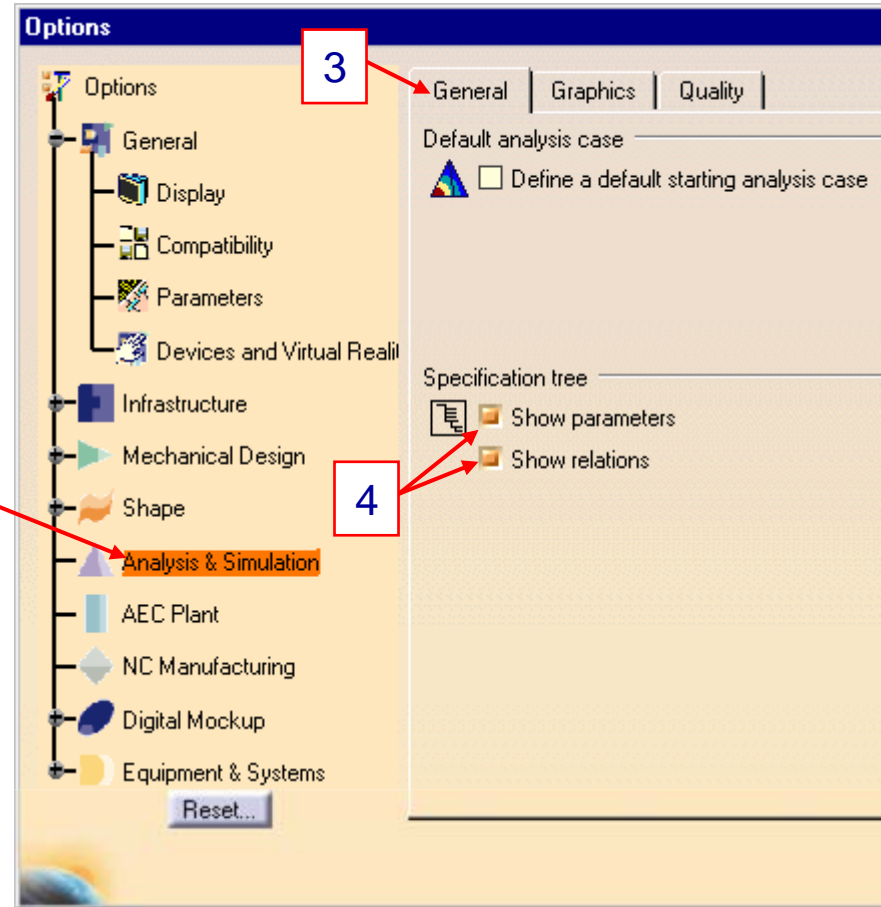
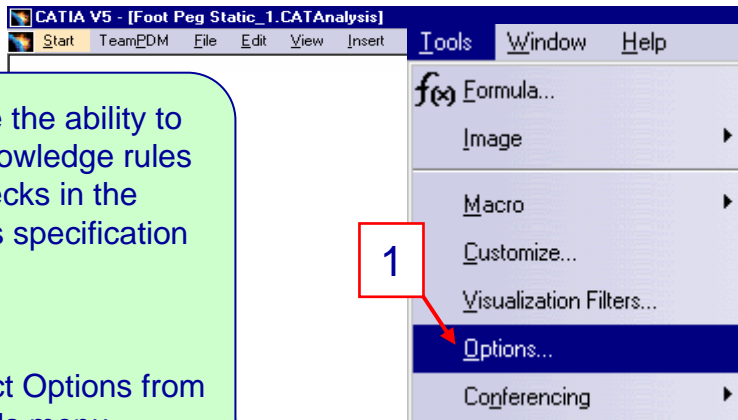
1. Select Options from the Tools menu.
2. Select Analysis & Simulation branch.
3. Select General tab.
4. Activate both boxes to show parameters and relations.

Update the analysis solution if needed.

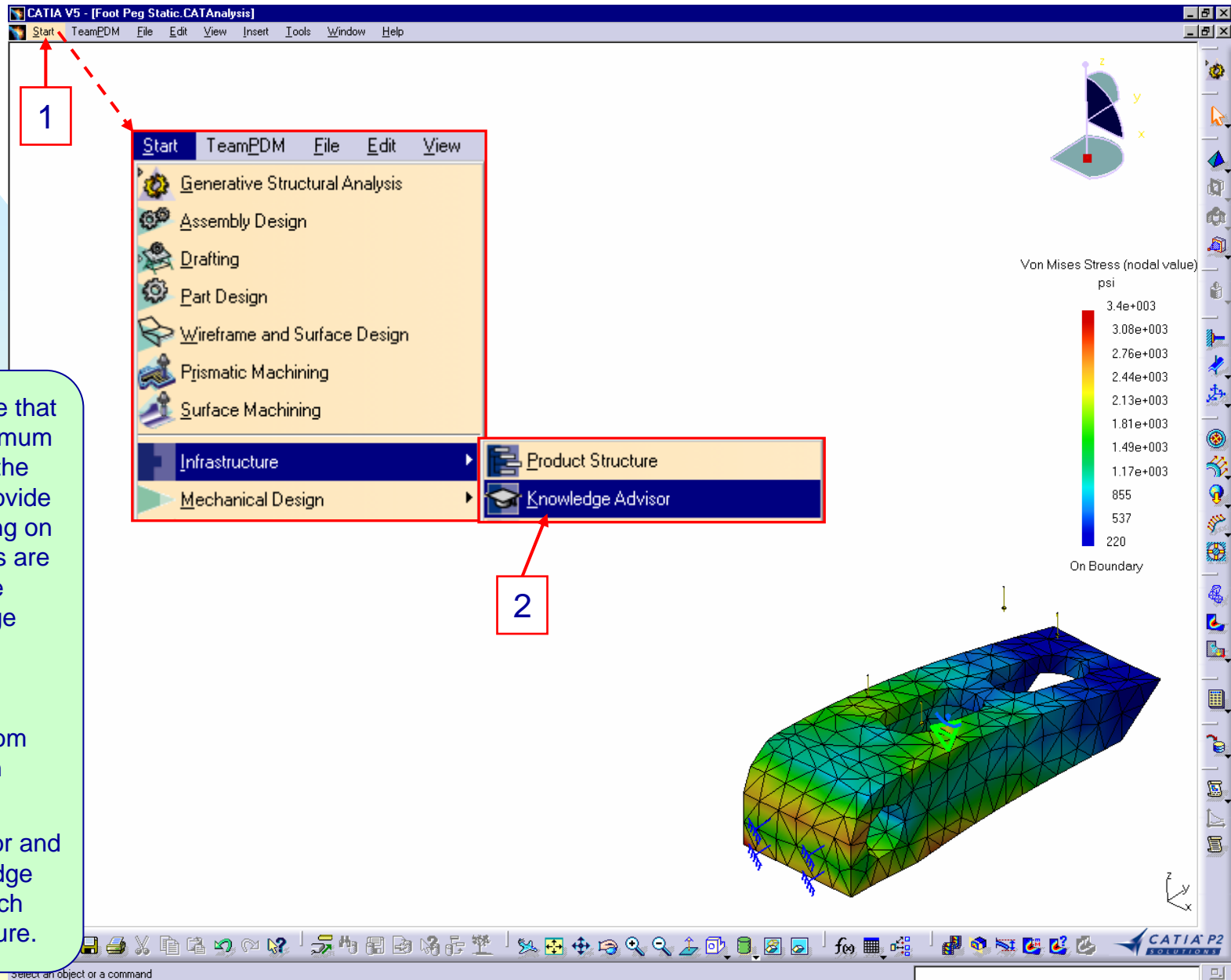
Steps:

1. Check for “update needed” symbol on the Static Case Solution.1
2. Compute to update the analysis results (see Section 3).

Symbols shows update needed



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).

Rule Editor

Name of Rule :
Displacement Max

Description :
Rule created by Student 12/20/01

Destination :
Relations

OK Cancel Help

Rule Editor : Displacement Max Active

Incremental

/*Rule created by Student 12/20/01*/

Dictionary	Members of Parameters	Members of All
Parameters	Renamed parameters	'Foot Peg\PartBody\Material'
Keywords	String	'Foot Peg\PartBody\Sketch.1\Par
Operators	Boolean	'Foot Peg\PartBody\Sketch.1\Par
Point Constructors	Length	'Foot Peg\PartBody\Sketch.1\Tar
Analysis operators	Integer	'Foot Peg\PartBody\Sketch.1\Tar
Surface Constructors	Angle	

OK Apply Cancel

Rule definition entered here

Dictionary categories to assist in defining rules

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).

1st line: Rule description

1

2

3

4

5

6

7

Rule Editor : Displacement Max Active

Incremental

/*Rule created by Student 12/20/01*/

Dictionary: Parameters, Keywords, Operators

Members of Keywords: if, else, else if

Rule Editor : Displacement Max Active

Incremental

/*Rule created by Student 12/20/01*/

if |

Dictionary: Parameters, Keywords, Operators

Members of Parameters: All, Renamed parameters, Length

Members of All: 'Maximum displacement Value', 'Finite Element Model\Maximum D

Rule Editor : Displacement Max Active

Incremental

/*Rule created by Student 12/20/01*/

if 'Maximum displacement Value'

Dictionary: Parameters, Keywords

Members of Parameters: All, Renamed parameters

Members of All: 'Maximum c

'Maximum displacement Value' = 0.011 in

OK Apply Cancel

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.

The screenshot shows the CATIA V5 interface with the Analysis Manager tree on the left and the Rule Editor on the right. The Rule Editor is titled "Rule Editor : Displacement Max Active" and contains the following code:

```
/*Rule created by Student 12/20/01*/  
if "Maximum displacement Value" > 0.009 in  
{  
  Message("The Max. Displacement is > .009in  
  Design modification required")  
}  
else  
  Message("The Max. Displacement is < .009in")
```

Number 8 points to the code. Below the code is a "Dictionary" list with "Keywords" selected, and a "Members of Keywords" list containing "if", "else", and "else if". Number 9 points to the "OK" button. The Analysis Manager tree shows a "Displacement Max" rule created under "Relations". Number 10 points to an "InformationDisplacement Max : Information" dialog box with the message: "The Max. Displacement is > .009in Design modification required". Number 11 points to the "OK" button in the dialog box. A red box labeled "Rule created" points to the "Displacement Max" rule in the tree.

Rule created

10

11

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.

The screenshot shows the CATIA V5 interface with the Check Editor dialog box open. The dialog box has the following fields:

- Name of Check : Von Mises Max
- Description : Check created by Student 12/20/01
- Destination : Relations

Buttons: OK, Cancel, Help

The Check Editor: Von Mises Max Inactive window is also shown, with the following settings:

- Incremental:
- Type of Check : Silent
- Message : Von Mises Max is not valid
- /*Check created by Student 12/20/01*/

The Dictionary section shows the following parameters:

Dictionary	Members of Parameters	Members of All
Parameters	Renamed parameters	'Foot Peg\PartBody\Material'
Keywords	String	'Foot Peg\PartBody\Sketch.1\'
Operators	Boolean	'Foot Peg\PartBody\Sketch.1\'
Point Constructors	Length	'Foot Peg\PartBody\Sketch.1\'
Analysis operators	Integer	'Foot Peg\PartBody\Sketch.1\'
Surface Constructors	Angle	'Foot Peg\PartBody\Sketch.1\'

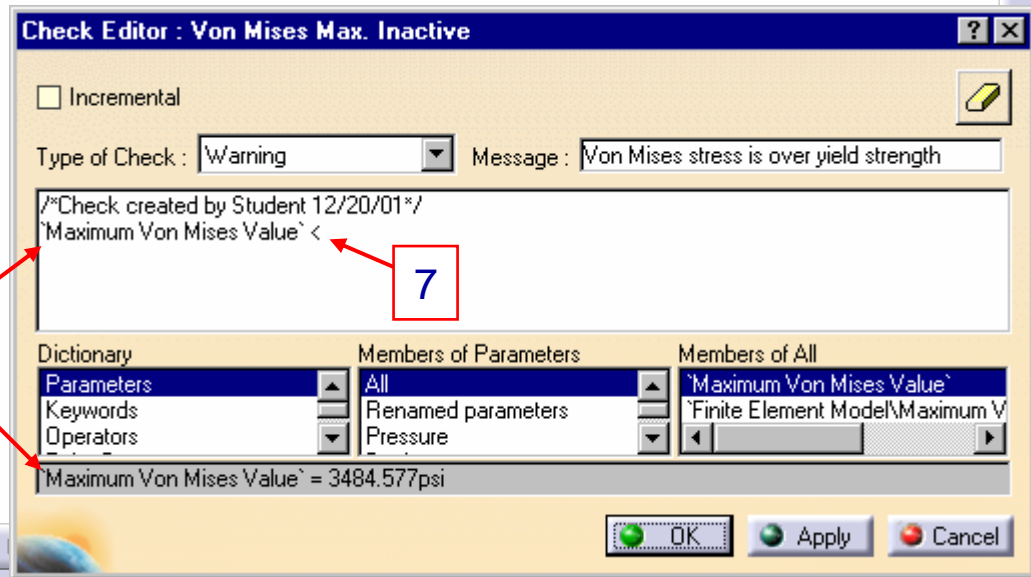
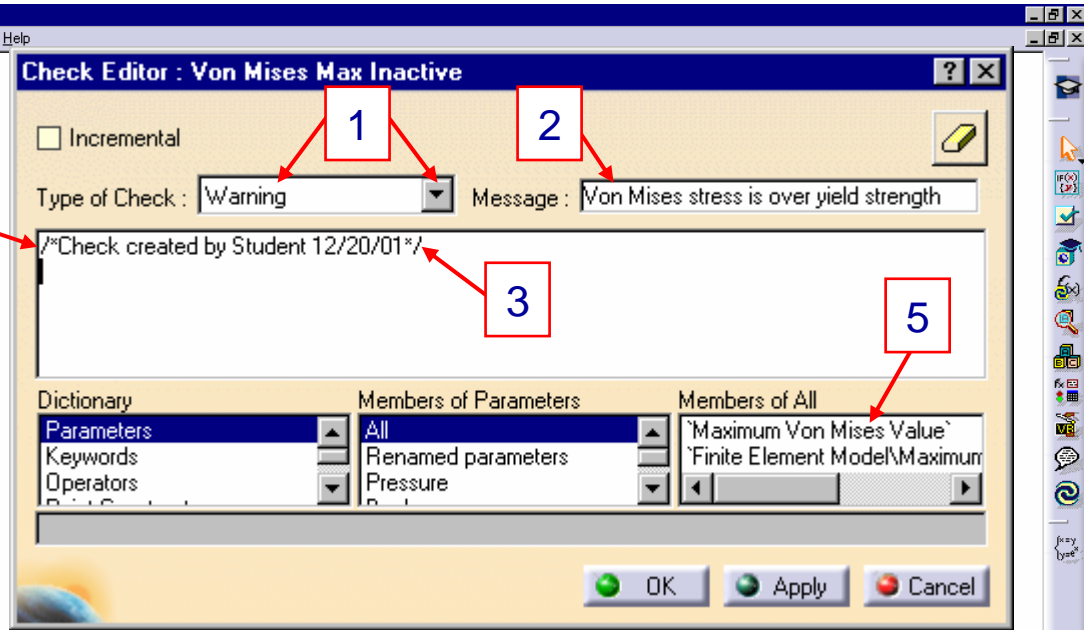
Step 4. Create knowledge check

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.

1st line: Rule description



Select an object or a command

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.

Check created and shows green light (check not violated)

The screenshot displays the CATIA V5 interface for creating a knowledge check. The 'Analysis Manager' tree on the left shows the 'Foot Peg' selected, with its parameters listed. The 'Check Editor : Von Mises Max Inactive' dialog box is open, showing the configuration for a 'Von Mises Max' check. The 'Members of Parameters' list includes 'Pressure', and the 'Members of Pressure' list includes 'Aluminium\Aluminium.1.1\Young Modulus' and 'Aluminium\Aluminium.1.1\Yield Strength'. The 'Check Editor : Von Mises Max Active' dialog box is also open, showing the configuration for a 'Von Mises Max' check. The 'Members of Parameters' list includes 'Pressure', and the 'Members of Pressure' list includes 'Aluminium\Aluminium.1.1\Young Modulus' and 'Aluminium\Aluminium.1.1\Yield Strength'. The 'OK' button is highlighted with a green light.

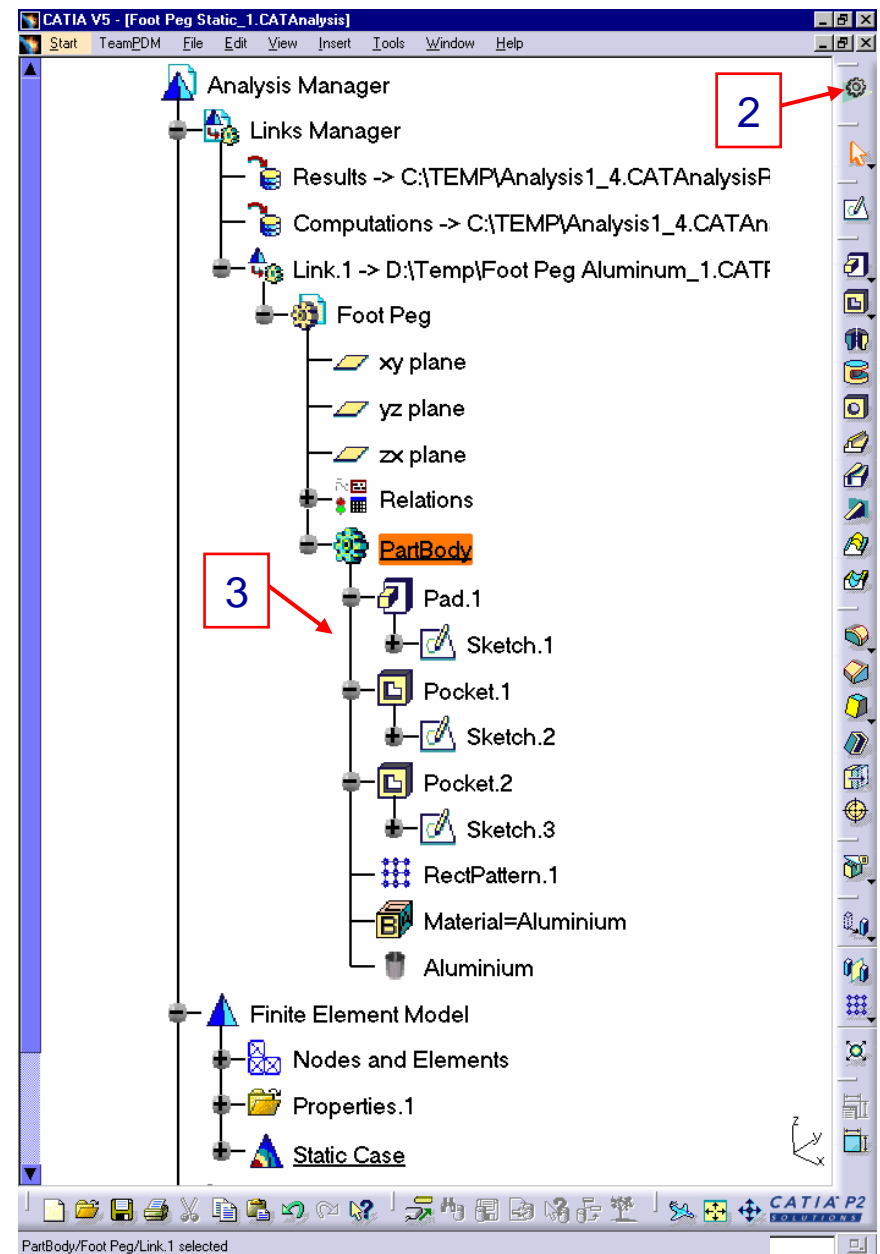
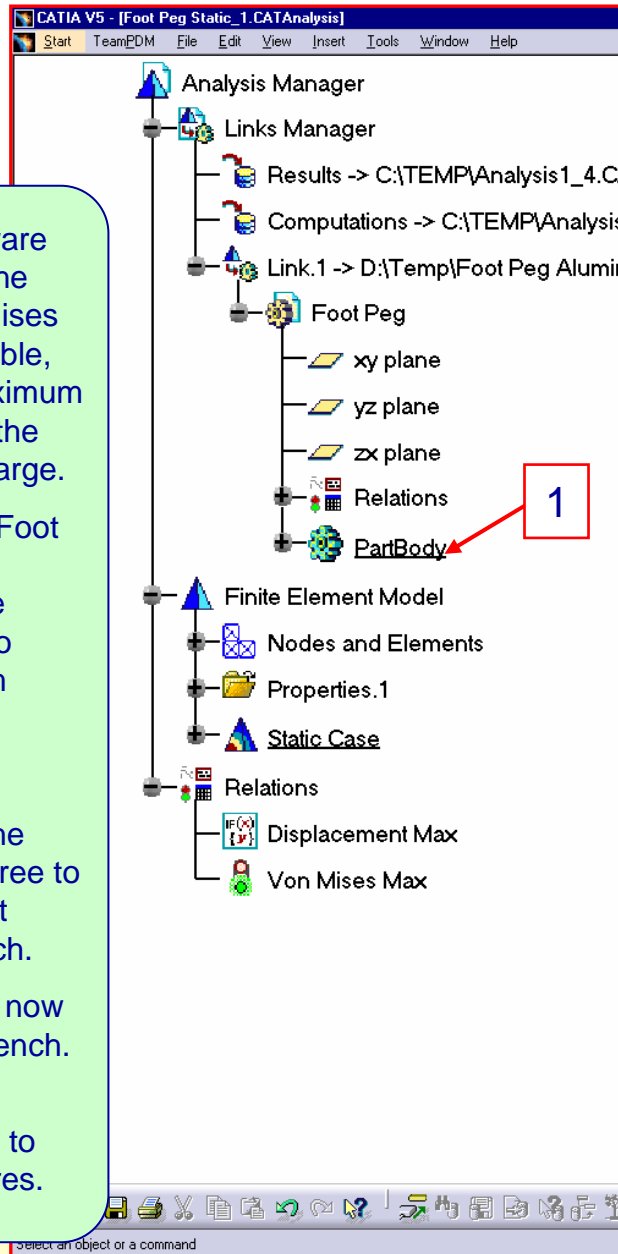
Step 5. Modify Foot Peg design

The knowledgware 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.

CATIA V5 - [Foot Peg Static_1.CATAnalysis]

Start TeamPDM File Edit View Insert Tools Window Help

Analysis Manager

Links Manager

Results -> C:\TEMP\Analysis1_4.CATAnalysisResults

Computations -> C:\TEMP\Analysis1_4.CATAnalysisComputations

Link.1 -> D:\Temp\Foot Peg Aluminium_1.CATPart

Foot Peg

- xy plane
- yz plane
- zx plane
- Relations
- PartBody
 - Pad.1
 - Sketch.1
 - Pocket.1
 - Sketch.2
 - Pocket.2
 - Sketch.3
 - RectPattern.1
 - Material=Aluminium
 - Aluminium
- Finite Element Model
 - Nodes and Elements
 - Properties.1
 - Static Case

Pad Definition

First Limit

Type: Dimension

Length: 2.5in

Limit: No selection

Profile

Selection: Sketch.1

Reverse Side

Mirrored extent

Reverse Direction

More>>

OK Cancel Preview

Constraint Definition

Value: 7in

Reference

OK Cancel

Enter new data to edit the pad

Dim1 | 2.5in

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.

Preview of modified spacing

Arrow shows 1st direction

1

2

3

4

5

Rectangular Pattern Definition

First Direction | Second Direction

Parameters: Instance(s) & Spacing

Instance(s): 2

Spacing: 2.5in

Length: 2.75in

Reference Direction

Reference element: Edge.1

Reverse

Object to Pattern

Object: Pocket.2

Keep specifications

More >>

OK Cancel Preview

Change the initial feature or body or modify parameters.

Step 6. Compute analysis

CATIA V5 - [Foot Peg Static.CATAnalysis]

Start TeamPDM File Edit View Insert Tools Window Help

Analysis Manager

- Links Manager
 - Results -> C:\TEMP\Analysis1_4.CATAnalysisResults
 - Computations -> C:\TEMP\Analysis1_4.CATAnalysisComputations
 - Link.1 -> D:\Temp\Foot Peg Aluminum.CATPart
- Finite Element Model **1**
- Nodes and Elements
- Properties.1
- Static Case **Symbol showing analysis case not updated**
- Relations
 - Displacement
 - Von Mises Ma

3D Model: Foot Peg Aluminum.CATPart with analysis symbols (red arrows, green/blue symbols).

Compute Dialog Box **2**

- Dropdown: **3** All
- Preview:
- Buttons: **4** OK, Cancel

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.

The screenshot displays the CATIA V5 interface for a static analysis. The Analysis Manager tree on the left shows the following structure:

- Analysis Manager
 - Links Manager
 - Results -> C:\TEMP\Analysis1_4.CATAnalysisResults
 - Computations -> C:\TEMP\Analysis1_4.CATAnalysisComputations
 - Link.1 -> D:\Temp\Foot Peg Aluminum.CATPart
 - Finite Element Model
 - Nodes and Elements
 - Properties.1
 - Static Case
 - Restrains.1
 - Loads.1
 - Static Case Solution.1
 - Sensors.1
 - Energy
 - Maximum Displacement
 - Maximum Von Mises
 - Relations
 - Displacement Max
 - Von Mises Max

The 3D model of the foot peg shows displacement vectors. A red box labeled '1' points to the InformationDisplacement Max dialog box, which displays: "The Max. Displacement is < .009in". A red box labeled '2' points to the Von Mises Max sensor in the tree, which is green. A red box labeled '3' points to the Output Values dialog box for Maximum Von Mises, which displays: "Maximum Von Mises Value 2896.478psi". Another Output Values dialog box for Maximum displacement shows: "Maximum displacement Value 0.008in". A text box on the right states: "Max. displacement is now acceptable".