STUDENT GUIDE

Student Notes:

CATIA V5 Training
Foils

CATIA V5 Analysis

Version 5 Release 19
January 2009

EDU_CAT_EN_V5A_FF_V5R19

Student Notes:

Lesson 1: Introduction to Finite Element Analysis

About this Course

Introduction

CATIA is a robust application that enables you to create rich and complex designs. The goal of the course 'CATIA V5 for Analysis' is to teach you how to assist designer to perform preliminary Static Analysis of parts and assemblies designed in CATIA. This course focuses on the Finite Element Analysis process for static analysis, concepts in static analysis and performing Static analysis for parts and assemblies using Generative Part Structural (GPS) Analysis workbench.

Course Design Philosophy

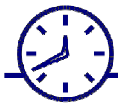
This course is designed using a process-based approach to training. Rather than focusing on individual features and functions, this course emphasizes on the processes and procedures required to complete a particular task. By using the case studies to illustrate these processes, you will learn the necessary commands, options, and menus within the context of completing a design task.

Target audience

- Mechanical Designers, Structural Analysts

Prerequisites

CATIA V5 Fundamentals



16 Hours

Student Notes:

About the Student Guide

Using the Student Guide

This student guide is intended to be used in a classroom environment under the guidance of a certified CATIA Instructor. The exercises and case studies are designed to be demonstrated by the instructor.

Exercises/Case Studies

This course illustrates the process-based approach in two ways: exercises and case studies. Exercises give you the opportunity to apply and practice the material covered during the lecture/demonstration portion of the course. They are designed to represent typical design and modeling situations. Extra exercises have been included in this guide to accommodate those students who may wish to practice more modeling. The case studies provide a context in which you would use particular tools and methods, and illustrate the process flow you would typically follow for a project.

Student Notes:

Conventions Used in the Student Guide

The following typographic conventions are used in the student guide:

- **Bold text** within a sentence denotes options selected from the CATIA menu bar.
- **Red text** denotes the name of a tool, icon, button, or window option.
- *Italic text* within a sentence is used to apply emphasis on key words.
- Numerical lists are used in sequential lists, such as the steps in a procedure.
- Lower-case alphabetical sub-lists are used in sequential sub-lists, as for steps in an exercise procedure.
- 2b identifies areas in a picture that are associated with steps in a sequential list, such as in an exercise.
- Upper-case alphabetical lists are used in non-sequential lists, as for a list of options or definitions.
- Text enclosed in < > brackets represents the names of keyboard keys that must be pressed.
- Text enclosed in [ ] brackets identifies text that must be entered into a text field of a CATIA dialog box or prompt.

Example page:

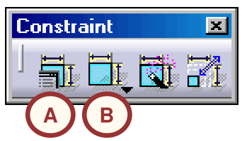
Use the following steps to create a new document in CATIA:

1. Click **Start > Mechanical Design > Part Design**.
2. Create new part.
   a. Click **File > New**.
   b. Select **Part** from the New window.
   c. Select **OK**.

   d. Press <CTRL> + <S> to save the document.
   e. Enter [my first document] as the document name.

You can create the following profile types:

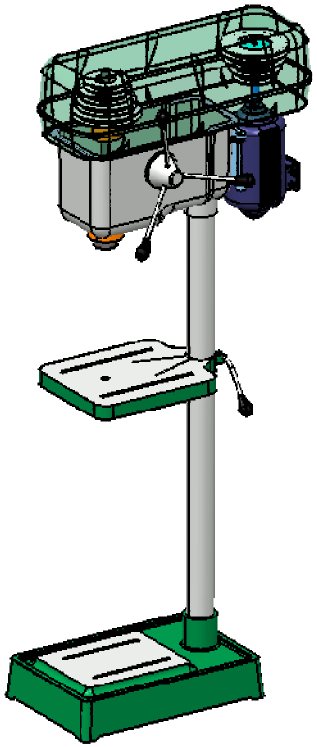
A. User Defined Profiles
B. Pre-Defined Profiles
C. Circles

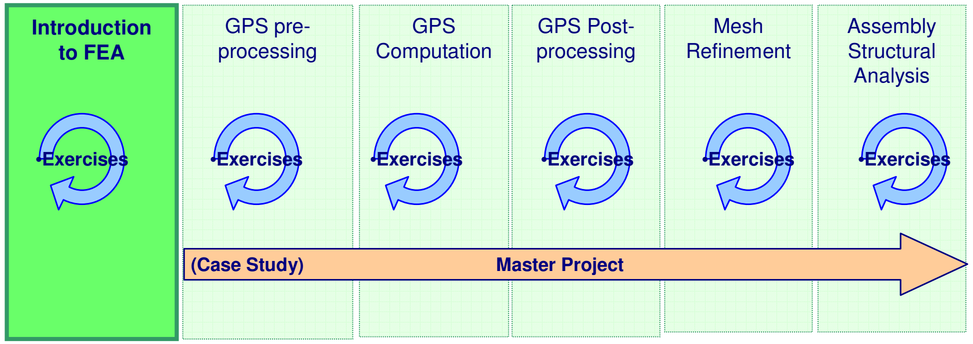


Student Notes:

Case Study: Introduction to Finite Element Analysis (FEA)

Lesson 1 will familiarize you with the Finite Element Analysis process. From lesson 2 onwards, each lesson in this course will contain a case study, which will help to explain the skills and concepts covered in the lesson. Models used in case studies come from the drill press assembly, which is also your master project. In this lesson, you will learn about the types of part or surface preparation that may be required before analysis, and how to open the GPS workbench.



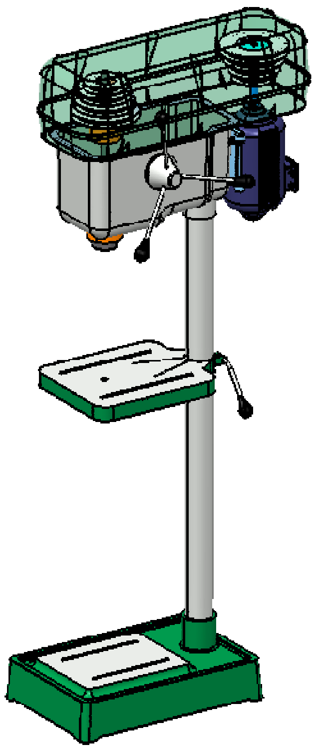


Student Notes:

Design Intent

Each case study contains a set of model requirements, known as the design intent. One interpretation of design intent is how the part model has been constructed in order to properly convey its functional requirements. In subsequent lessons the case study will be analyzed in order to verify these functional requirements. By the end of this lesson you should be able to :

- ✓ Understand FEA process
- ✓ Open GPS Workbench
- ✓ Changing Default Units
- ✓ Choosing Local co-ordinate System
- ✓ Applying Constraints on Part of Face
- ✓ Prepare Surfaces for GPS Analysis



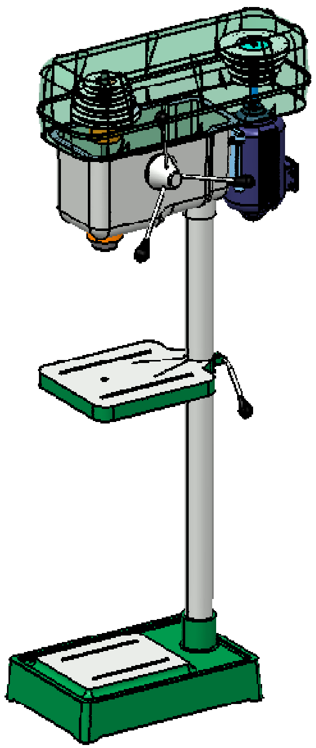
Student Notes:

Stages in the Process

Each lesson explains the topics in steps. These steps outline how to pre-process, compute, and post-process the part or assembly in the case study. Each step contains the information needed to complete the exercise.

For Lesson 1, you will go through the following steps to start yourself with FEA in Generative Structural Analysis Workbench:

1. What is Finite Element Analysis Process
2. Introduction to Generative Structural Analysis (GPS) Workbench
3. Preparing Parts and Surfaces for Analysis



Student Notes:

Step 1: Finite Element Analysis Process

In this section, you will understand the general steps to be followed to perform Finite Element Analysis for structures.

To introduce yourself to Finite Element Analysis use the following steps:

1. **Finite Element Analysis (FEA) Process**
2. Introduction to GPS workbench
3. Preparing Parts and Surfaces for analysis

Student Notes:

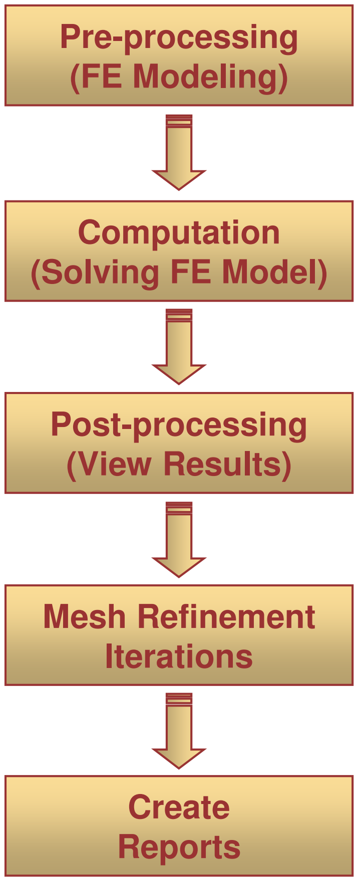
What is Finite Element Analysis (1/4)

Finite Element Analysis (FEA) is a numerical tool used to simulate a physical system. With this method the modeled system is broken down into simple geometric shapes, called finite elements, whose behavior can be described mathematically. The elements and their interrelationships are converted into a system of equations which are solved numerically.

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

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

The overall process can be subdivided into smaller steps as shown in the illustration.



Finite Element Analysis Process

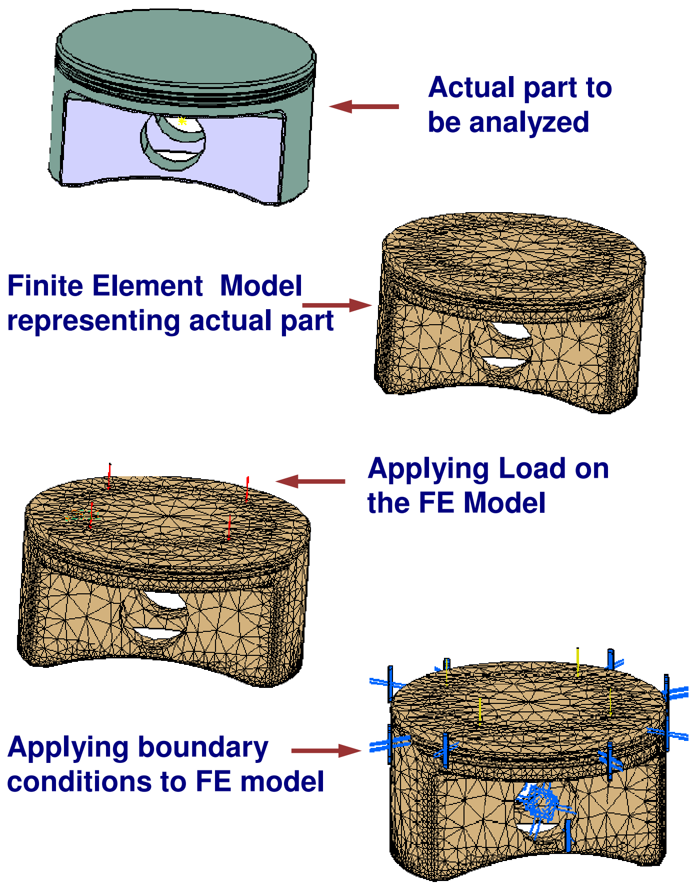
Student Notes:

What is Finite Element Analysis (2/4)

A. Pre-processing

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

- The physical structure is converted into an equivalent Finite Element (FE) model.
- The actual material properties are defined for FE model.
- Actual physical Forces are converted into equivalent FE Loads.
- The actual physical Boundary Conditions are converted into equivalent FE Boundary Conditions.

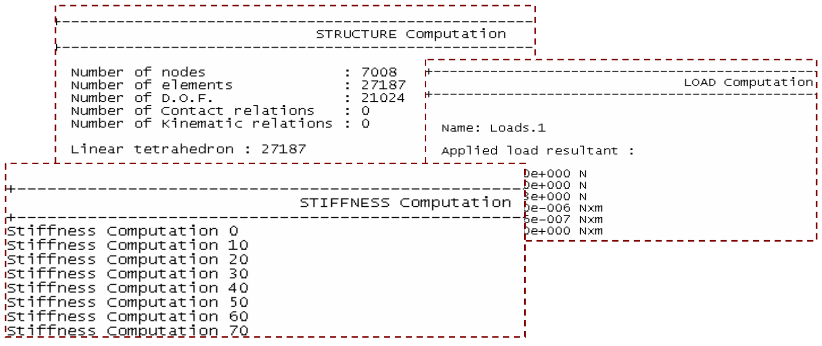


Student Notes:

What is Finite Element Analysis (3/4)

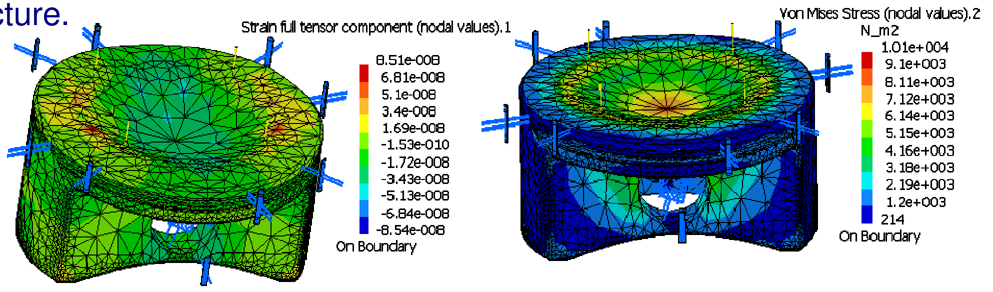
B. Computation

- The FE problem provided by the pre-processing step is solved to find out the unknown displacement values.



C. Post-processing

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



Student Notes:

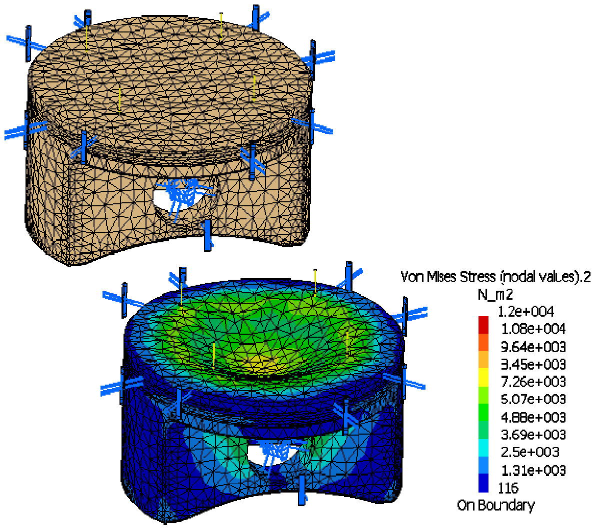
What is Finite Element Analysis (4/4)

D. Mesh Refinement

- The first solution provides initial estimation of stress / strain values. If this is considered to be sufficiently accurate, no further computation will be required.
- To get a more accurate solution, the mesh needs to be refined and the computation is to be done again.
- A number of mesh refinement and computation iterations are performed till the required solution accuracy is achieved.

E. Report Generation

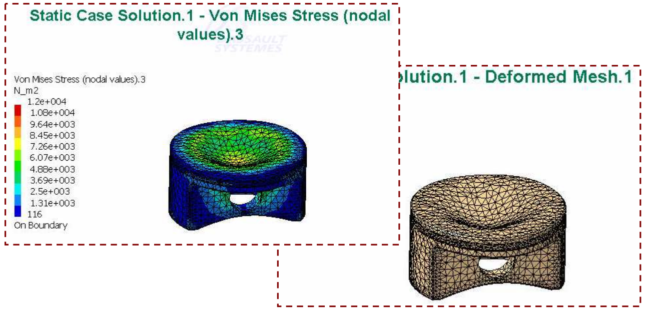
- Once the required accuracy level is achieved, various plots such as Displacement, Principle Stress, Von-Mises Stress can be obtained.



ELEMENT QUALITY:

Criterion	Good	Poor	Bad	Worst	Average
Distortion	13267 (64.35%)	5728 (27.78%)	1623 (7.87%)	57.220	31.031
Stretch	20589 (99.86%)	29 (0.14%)	0 (0.00%)	0.276	0.595
Length Ratio	20615 (99.99%)	3 (0.01%)			

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

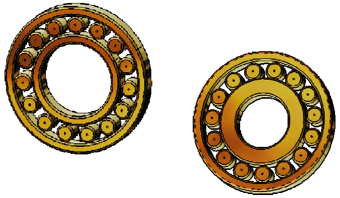


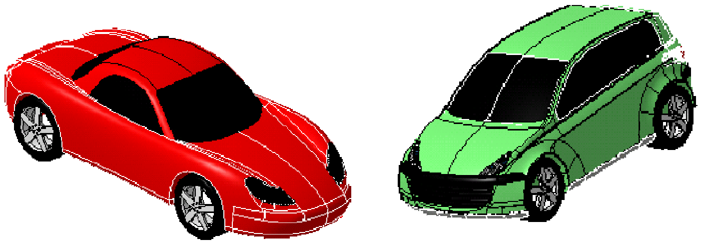
Student Notes:

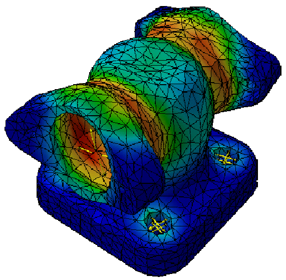
Why to Use Finite Element Analysis

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

- You can validate product modifications to meet new conditions.
- You can verify a proposed product or structure which is intended to meet the customer specification, prior to manufacturing or construction.
- You can evaluate advantages and effectiveness of various product design alternatives without having any kind of experimental test setup.
- It helps to implement the product concept ‘first time right’ with corresponding cost savings thus minimizes the product life cycle time significantly.
- With FEA software tools, you can optimize your product for minimum weight and volume with negligible cost, thus increasing product life and improving product reliability.



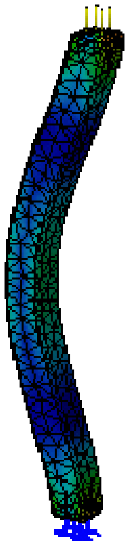




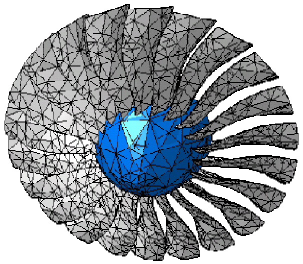
Student Notes:

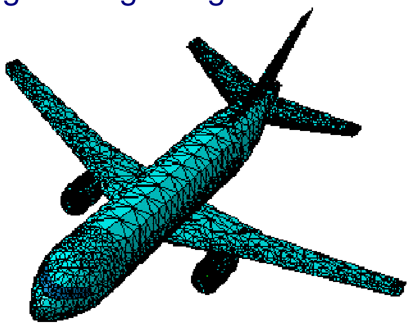
Application of Finite Element Analysis

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

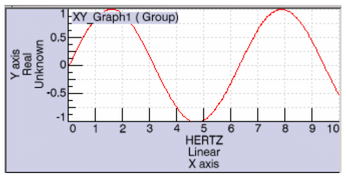


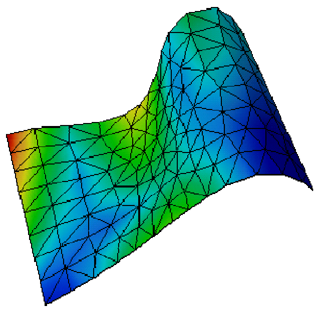
- Structural Analysis
- Dynamic analysis
- Buckling Analysis



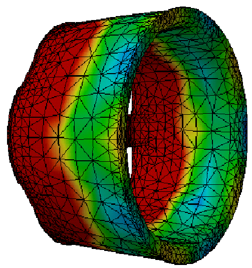


- Vibrations Analysis
- Acoustic Analysis
- Shock Analysis
- Crash Analysis





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



Copyright DASSAULT SYSTEMES

Student Notes:

Step 2: Introduction to GPS workbench

In this section, you will learn:

1. ***How to perform FEA process steps in Generative Part Structural Analysis workbench.***
2. ***How to access the GPS workbench***
3. ***What is Static Analysis***

To introduce yourself to Finite Element Analysis use the following steps:

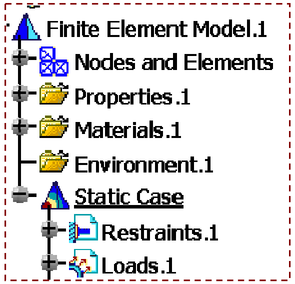
1. Finite Element Analysis (FEA) Process
2. **Introduction to GPS workbench**
3. Preparing Parts and Surfaces for analysis

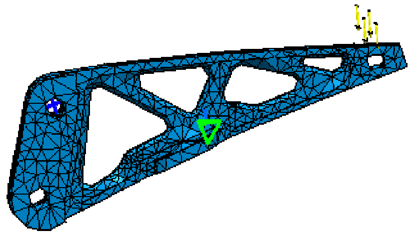
<u>*Student Notes:*</u>

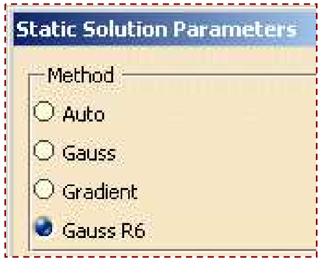
General FEA Process in GPS Workbench (1/2)

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

1. Open the Generative Structural Analysis workbench. Apply material, mesh the part, apply the restraints and loads. (Pre-processing)
2. Compute the Analysis. (Computation)

<table>
<tr><th></th><th>General FEA Process</th><th>FEA Process Steps in GPS Workbench</th></tr>
<tr><td>1</td><td>Pre-processing
(FE Modeling)</td><td>


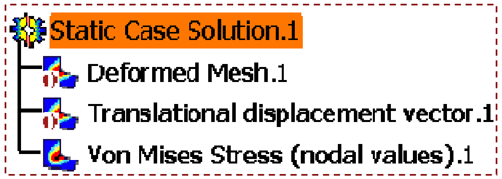
</td></tr>
<tr><td>2</td><td>Computation
(Solving FE Model)</td><td>

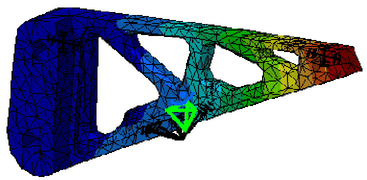
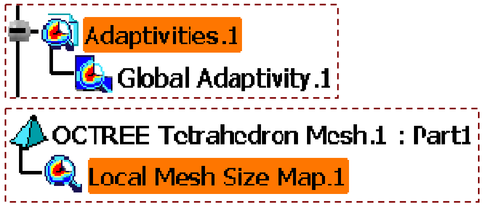

</td></tr>
</table>

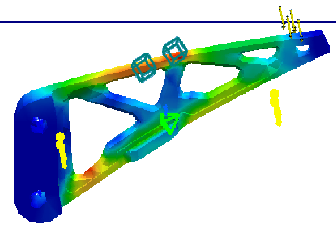
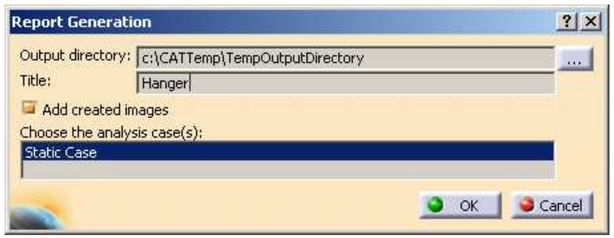
Student Notes:

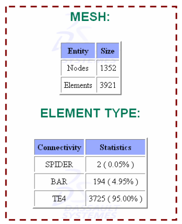
General FEA Process in GPS Workbench (2/2)

3. Visualize the results. (Post-processing)
4. Interpret the results and Mesh Refinement. (Mesh Refinement Iterations)
5. Manage the results. (Report Generation)

<table>
<tr><th></th><th>General FEA Process</th><th>FEA Process Steps in GPS Workbench</th></tr>
<tr><td>3</td><td>Post-processing (View Results)</td><td>

</td></tr>
<tr><td>4</td><td>Mesh Refinement Iterations</td><td>

</td></tr>
<tr><td>5</td><td>Create Reports</td><td>


</td></tr>
</table>

Student Notes:

Finite Element Analysis Types (1/3)

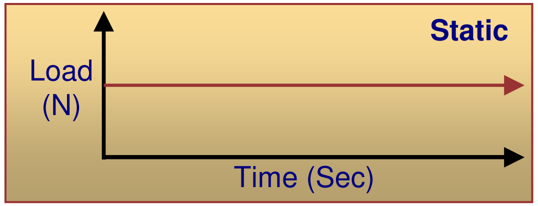
FEA for structures can be broadly classified in the following two ways:

1. According to variation of load with respect to time.

   A. Static analysis

   Static analysis is performed when load vectors and boundary conditions remain constant with respect to time.

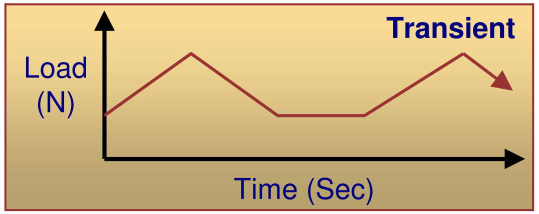
   Example: Static loading on machine frames due to weight, static loading on trusses, pressurized containers subjected to internal static pressure.

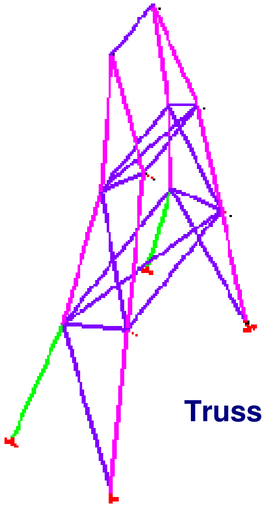


   B. Transient Analysis

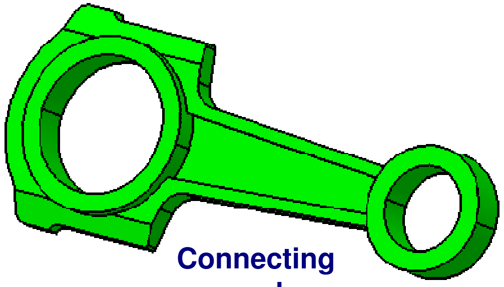
   Transient analysis is performed when load vectors and boundary conditions change with respect to time.

   Example: Load on connecting rod in piston-cylinder assembly.





Truss



Connecting rod

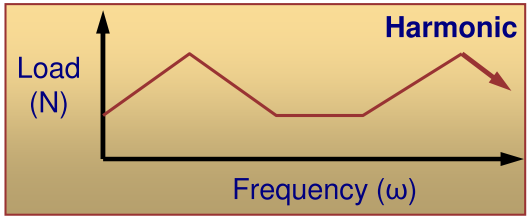
Student Notes:

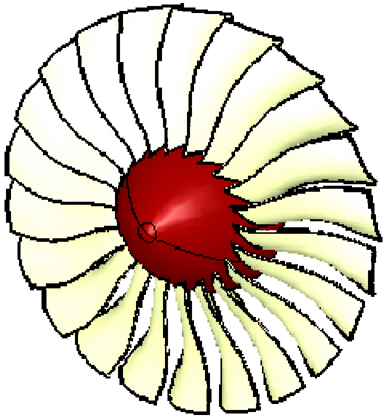
Finite Element Analysis Types (2/3)

C. Harmonic Analysis

Harmonic Analysis is performed when Load vectors change with the frequency.

Example: Analysis of crankshaft of an automobile engine, analysis of turbine and propellers.





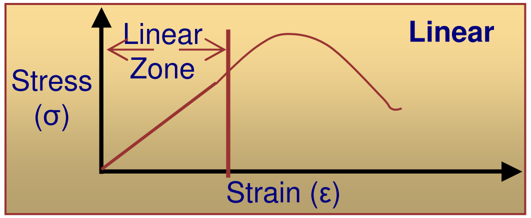
Propeller

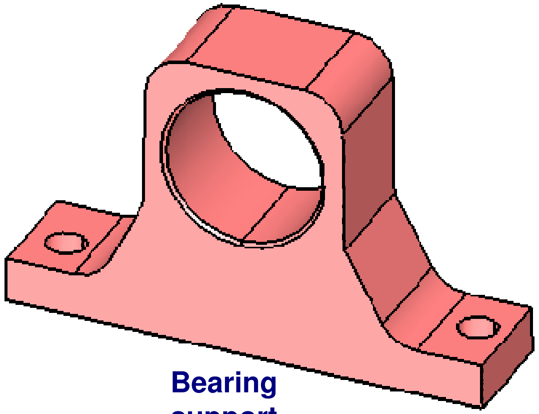
2. According to the way the structure reacts to the load.

A. Linear Analysis

Linear analysis is performed when the structural deformations due to loads are within the elastic limit of stress-strain curve for structure material.

Example. Static analysis of components where deformations produced are very small. (Static Load on bearing support)





Bearing support

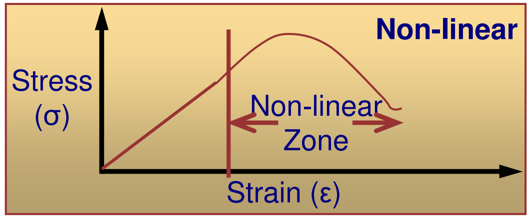
Student Notes:

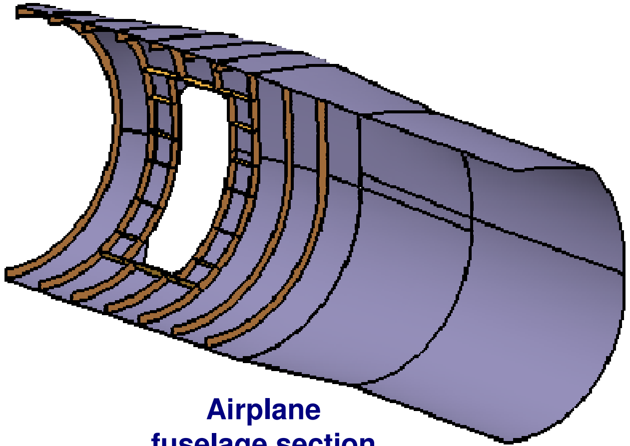
Finite Element Analysis Types (3/3)

B. Non-Linear Analysis

Non-linear analysis is performed when structural deformations due to load do not follow linear stress-strain relationship.

Example: Analysis of components using non-linear materials like composites, analysis involving non-linear phenomena like plastic deformation (metal forming), friction, crash analysis of automobiles, airplane structures





Airplane fuselage section

In CATIA, you can perform Static linear analysis, free vibration analysis to find out mode shapes of a structure, linear transient analysis and harmonic analysis. SIMULIA helps you to perform non-linear analysis.

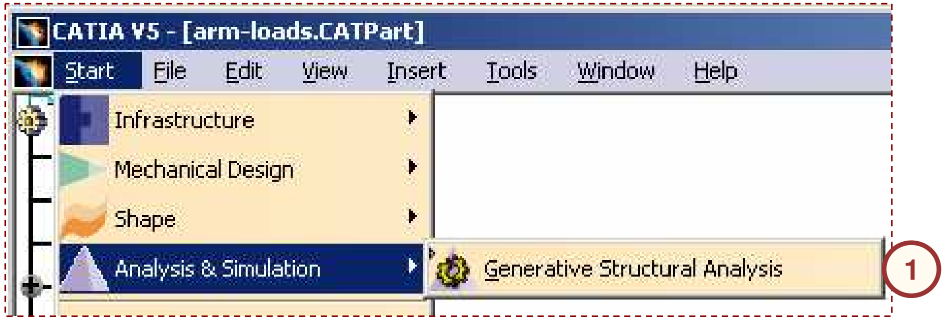
Using the GPS workbench with a GPS license, you can perform Static Linear analysis and free vibration analysis.

Student Notes:

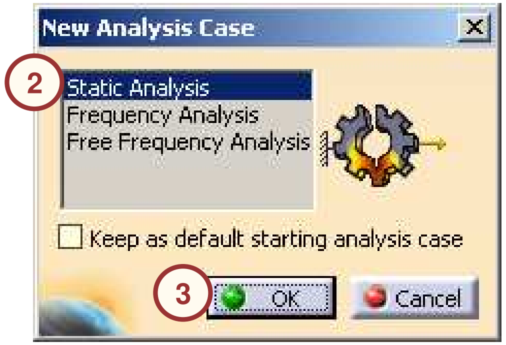
Accessing the Generative Structural Analysis Workbench

You can access Generative Structural Analysis (GPS) workbench using the following steps.

1. From the main menu select
   Start > Analysis & Simulation > Generative Structural Analysis



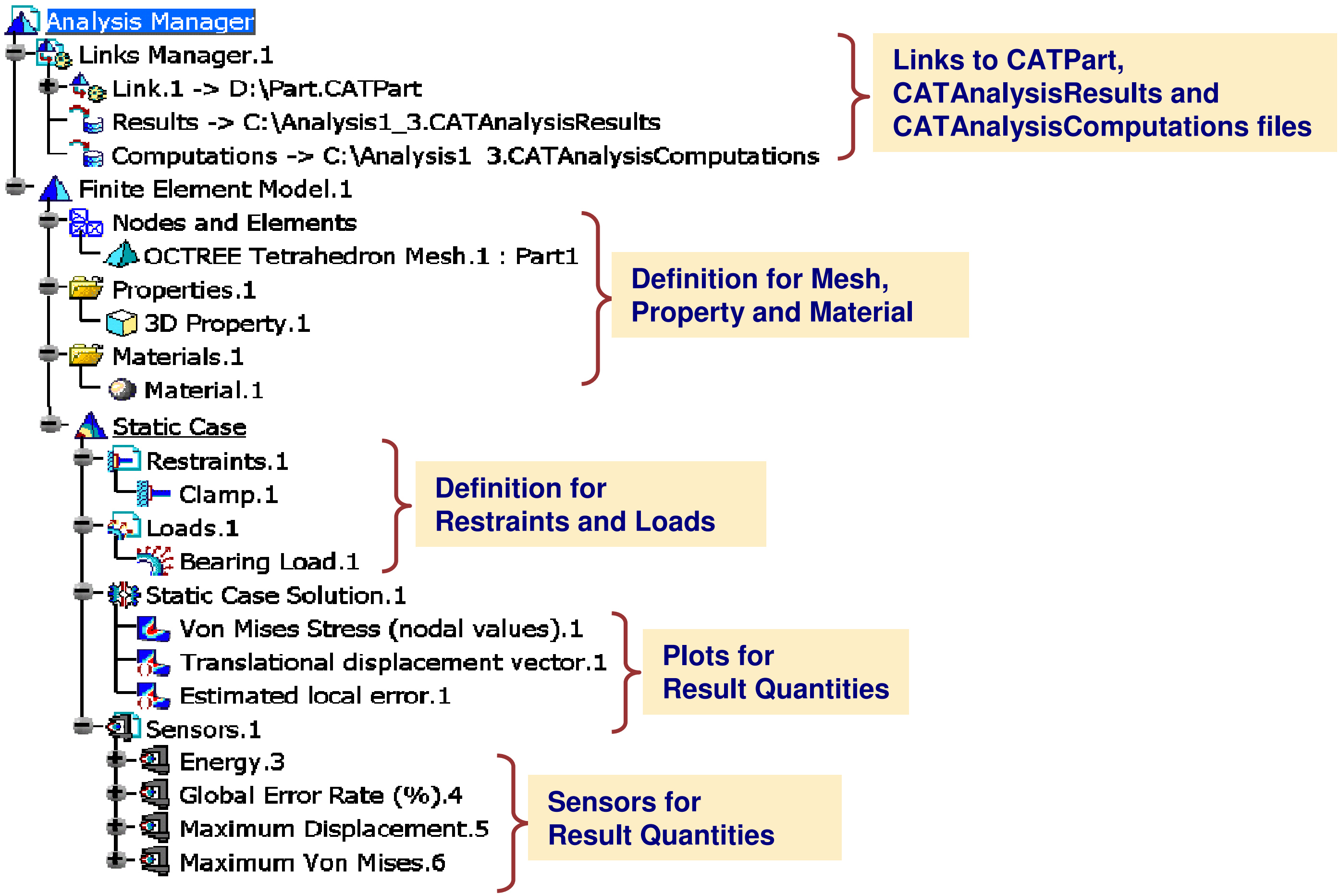
2. Select the **Static Analysis.**
3. Click **OK.**



Student Notes:

GPS Static Analysis Tree Structure

Entities created during process of GPS Static Analysis get mapped in tree structure as shown.



Student Notes:

Exercise 1A

Recap Exercise

15 min

In this exercise, you will open a Generative Structural Analysis (GPS) workbench, study the specification tree for analysis and see licensing information. Detailed instructions for this exercise are provided.

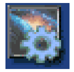
By the end of this exercise you will be able to:

- **View licensing information for CATIA V5**
- **Open the GPS workbench**
- **Change default units**

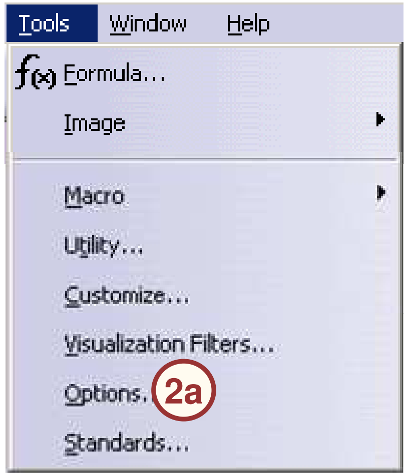
Student Notes:

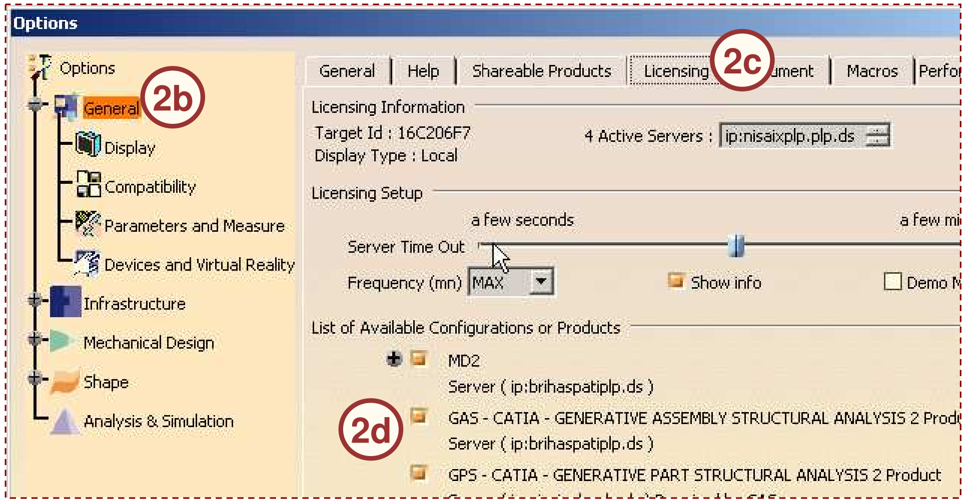
Exercise 1A (1/4)

1. **Open a part.**
   - Open 1A_Change_Units_Start.CATPart
     a. Select **File > Open.**
     b. Browse the folder and click on the file.
     c. Click **OK**.
     d. While using the companion, click the link on the companion, *Load a document*, to open the part.



2. **View Licensing Information.**
   a. Select on **Tools > Options.**
   b. Click on **General** in **Options** specification tree.
   c. Select **Licensing** Tab.
   d. Observe the checked products in **List of Available configurations or Products.**





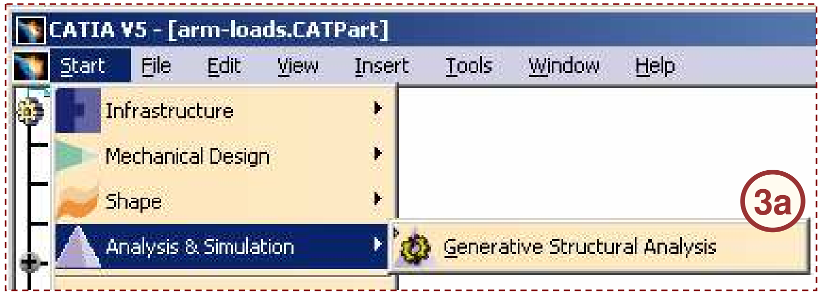
Please ensure that GPS and GAS licenses are checked.

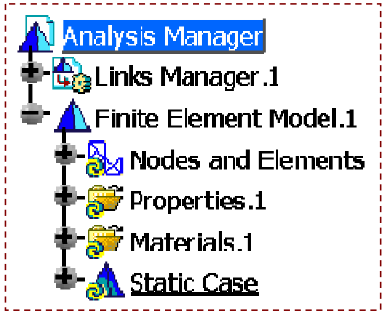
Student Notes:

Exercise 1A (2/4)

3. Access Generative Structural Analysis workbench.

- To access workbench from start menu.
  - a. Select **Start** > **Analysis & Simulation** > **Generative Structural Analysis**

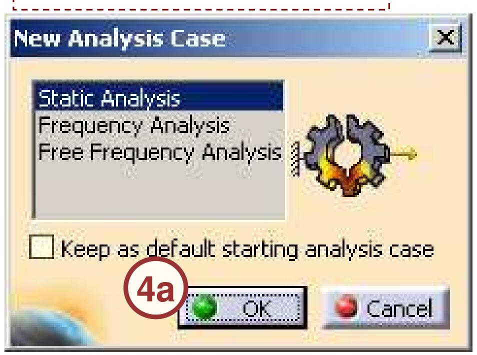




Analysis Manager Specification tree.

4. Create a Static Analysis Case.

- 'Static Analysis' case will be default selection.
  - a. Click **OK**.
  - b. A *Static Case* is added to the specification tree.

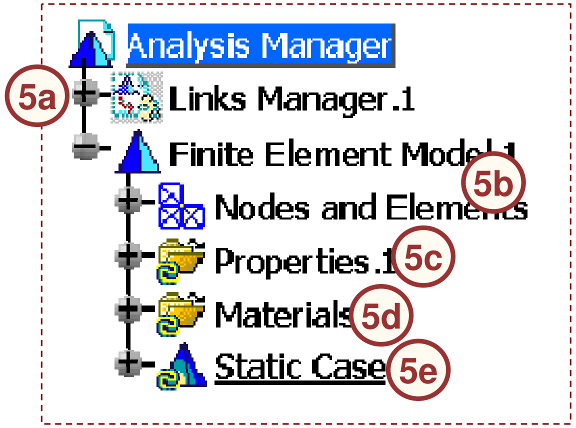


Student Notes:

Exercise 1A (3/4)

5. Study specification tree.

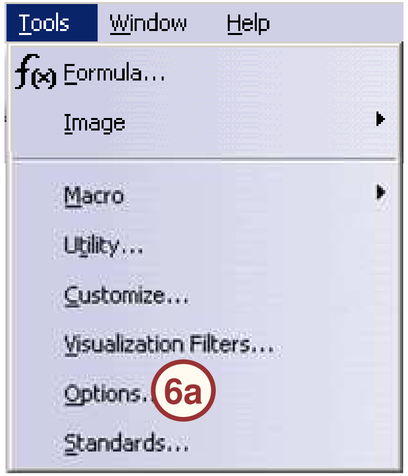
- Expand various nodes in the specification tree to observe the details.
  - a. Click *Links Manager.1* node on "+" sign to expand. Observe how the part is linked to Analysis Document.
  - b. Double-click *Nodes and Elements* in specification tree. Click on *OCTREE Tetrahedron Mesh.1 : Cylinder*. See which type of mesh is applied on Part.
  - c. Double-click *Properties.1* node in specification tree. Observe the Property Type applied to Part.
  - d. Double-click *Materials.1* node in specification tree. Then double-click on *Material.1*. Observe the Material applied to Part.
  - e. Double-click *Static Case* node in specification tree. Observe the new nodes added like Restraints, Loads, Static Case Solution and Sensors.

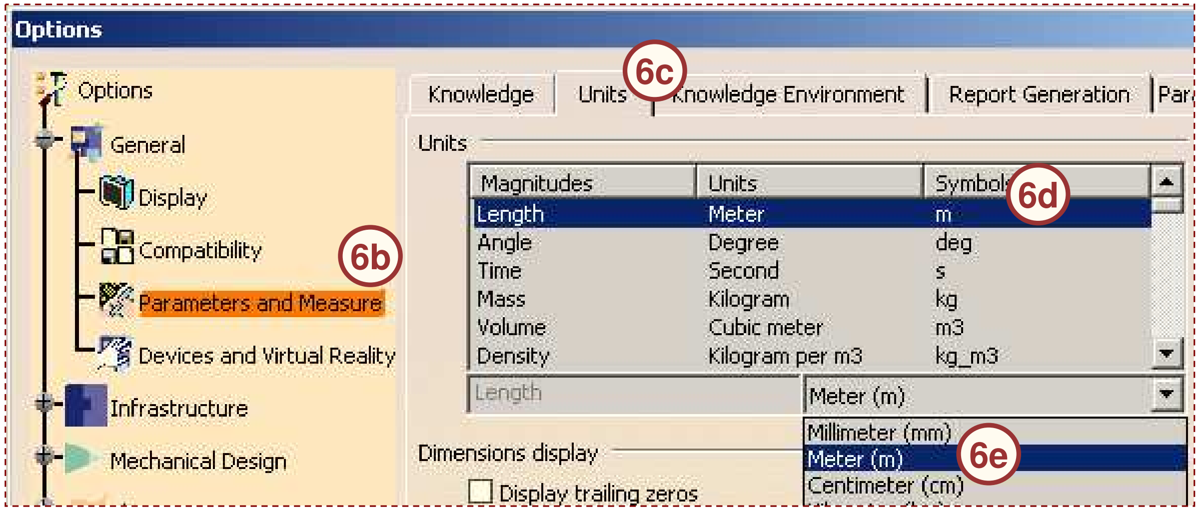


Student Notes:

Exercise 1A (4/4)

6. **Change Units.**
   - Change the unit of length to meters.
     - a. Select **Tools > Options.**
     - b. Click on **Parameters and Measures** in **Options** specification tree.
     - c. Select **Units** Tab.
     - d. Click on Length in **Units** field.
     - e. Select **Meter** in the dropdown box.
     - f. Click **OK.**

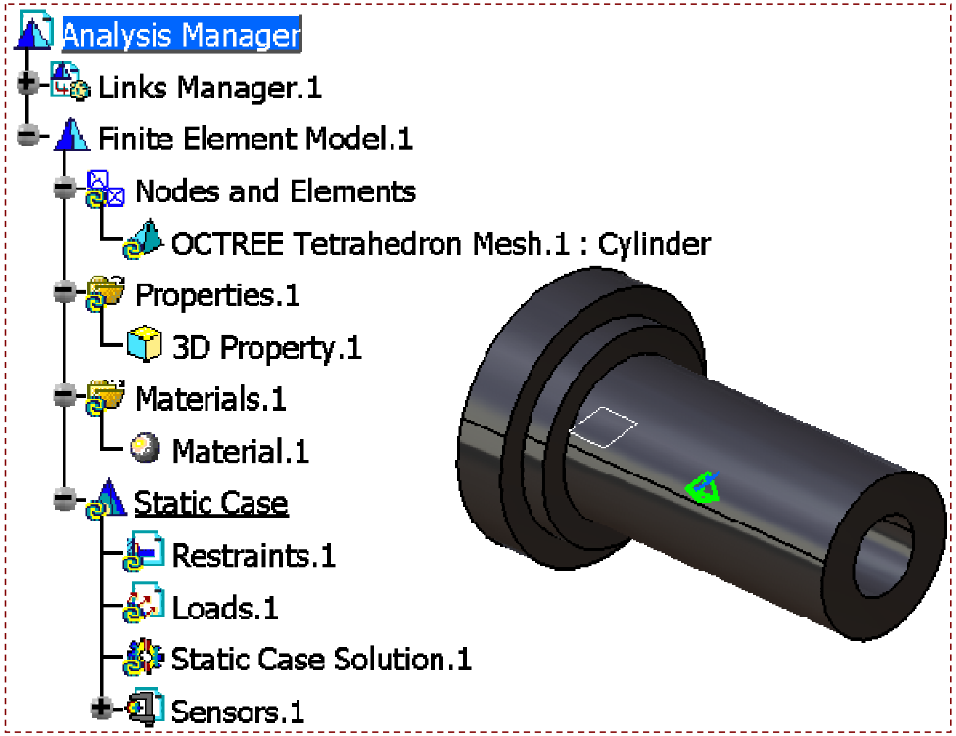




Student Notes:

Exercise 1A: Recap

- ✓ View licensing information for CATIA V5.
- ✓ Open the GPS workbench.
- ✓ Change default units.



Student Notes:

Step 3: Preparing Parts and Surfaces for Analysis

- ***In this section, you will learn which tools are used to prepare part and surface models for Analysis.***

To introduce yourself to Finite Element Analysis use the following steps:

1. Finite Element Analysis (FEA) Process
2. Introduction to GPS workbench
3. **Preparing Parts and Surfaces for analysis**

Student Notes:

Preparing Parts and Surfaces for Analysis

Before you switch to Generative Structural Analysis workbench, you may need to make some modifications in the existing CATIA geometry or modify some of the CATIA settings. This may include :

1. Changing default units
2. Creating support on part of a Face
3. Creating Local Axis System
4. Handling Non-Manifold Surfaces
5. Handling Overlapping Surfaces
6. Preparing Surfaces with Gaps for Analysis

In addition, having switched to the Generative Structural Analysis workbench, you may need to take into account the following when meshing very small models :

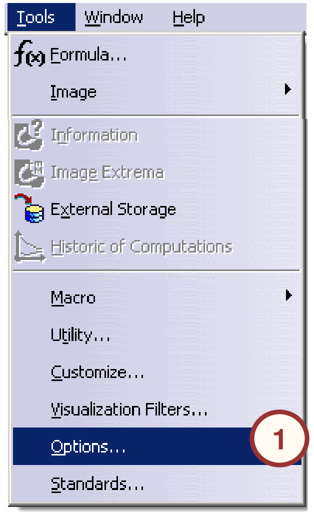
7. Lowest Mesh size value for Analysis

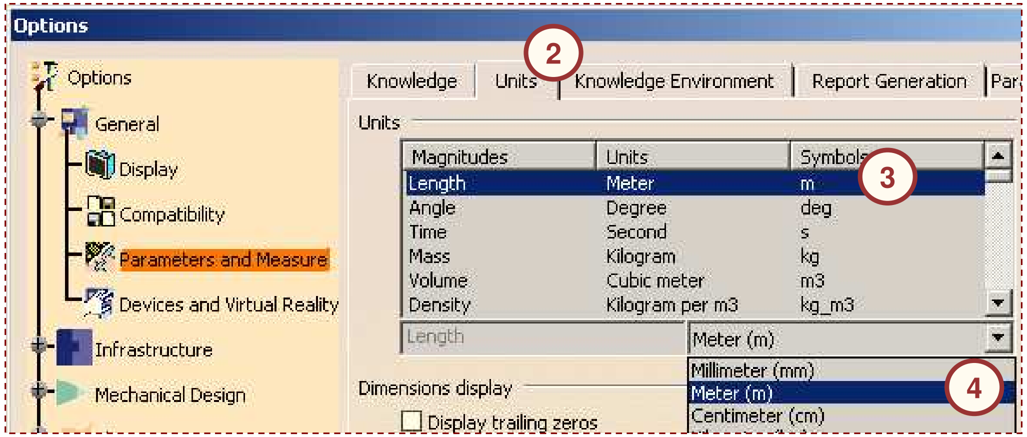
Student Notes:

Changing Default Units

You may need to change the default CATIA units as per your requirements. This can be done in following way.

1. In main menu select **Tools > Options.**
2. Click on **Parameters and Measure** in options tree and click on **Units** Tab.
3. Click on the unit in **Units** field which you want to change.
4. From the drop-down list select the required unit and click **OK.**

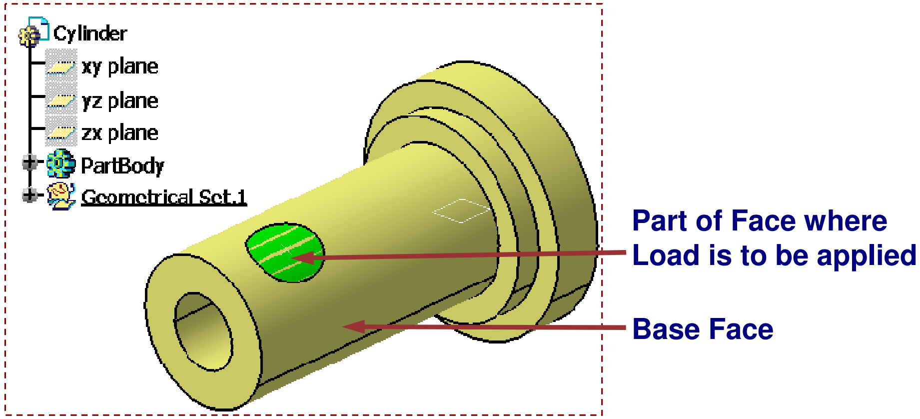




Student Notes:

Creating support on part of a Face (1/3)

Consider a case where you have to apply a load or constraint on part of a face as shown in the picture. In this case it is not possible to select only part of the face as shown, as a support for applying a load or constraints.

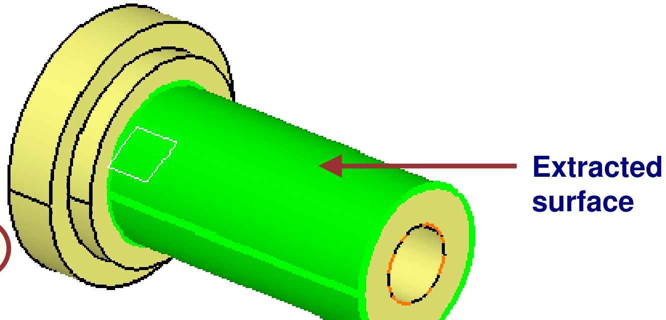


You will see how to create such a support using Part Design and Generative Shape Design tools.

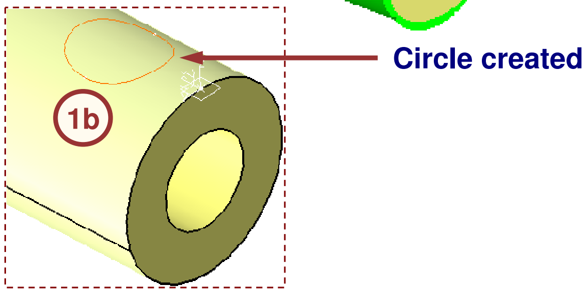
The general steps to create such a support are:

1. Create the required surface you want to select as support on the base face as geometry support. You can create this by following methods in Generative Shape Design Workbench:

   a. Extract the Base surface



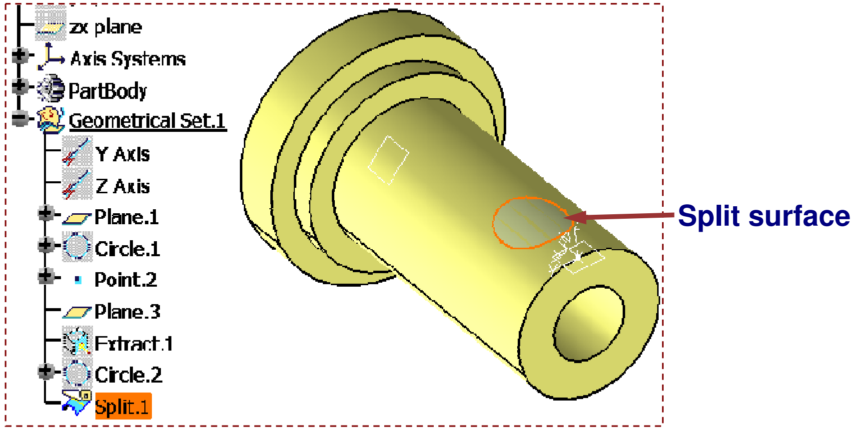
   c. Create the sketch/circle directly on the base surface with base surface as geometry support.



Student Notes:

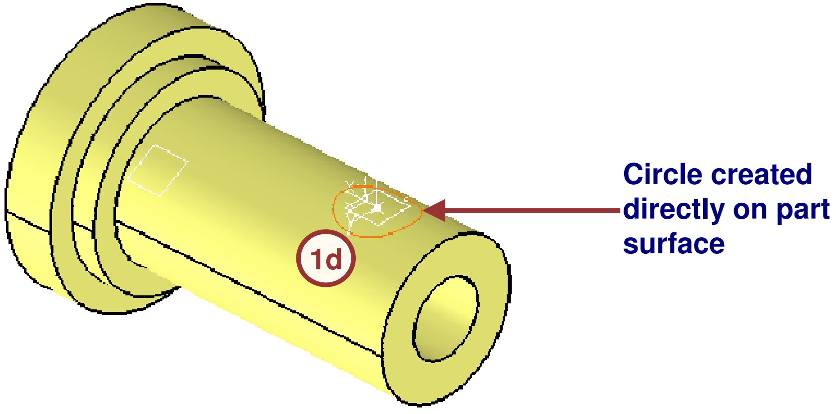
Creating support on part of a Face (2/3)

c. Split the extracted surface with sketch.



You can also create such support surface using following steps:

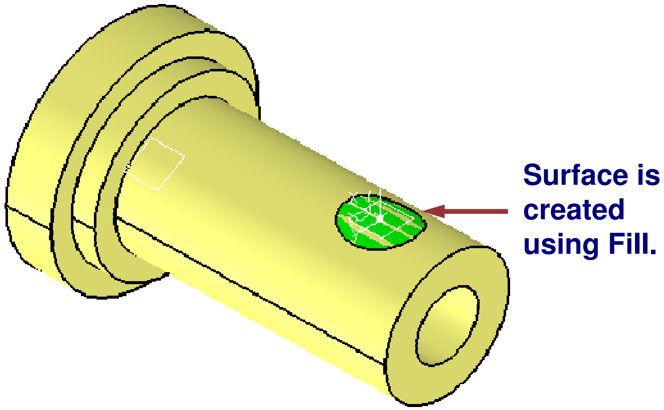
d. Create the sketch/circle directly on the base surface with base surface as geometry support. You can also create User coordinate system to create this sketch/circle.



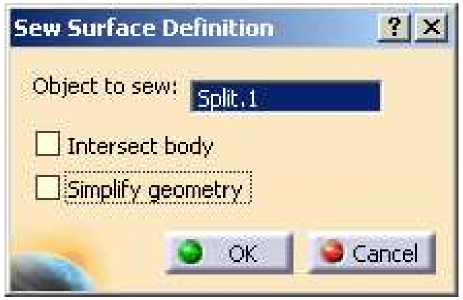
Student Notes:

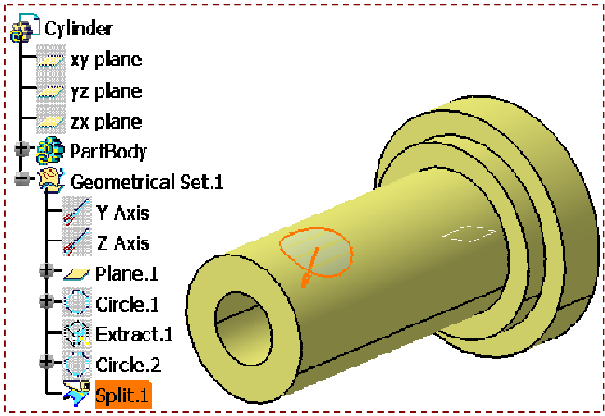
Creating support on part of a Face (3/3)

e. Fill the sketch/circle using to create required surface.



2. Sewing the created surface to the Part.
   In Part Design Workbench, click **Sew** icon from **Surface-Based Features** toolbar. Select the created surface and click **OK** with options shown unchecked.





Student Notes:

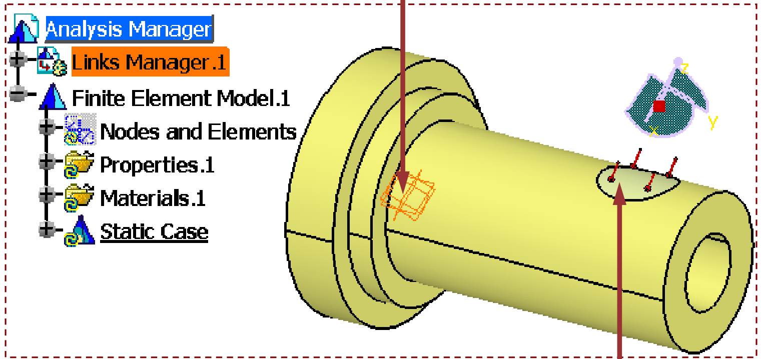
Creating Local Axis System (1/2)

The loads applied on the FE model are by default applied with respect to Global Axis System. Consider a case where part is not aligned with global Axis System as shown in the figure. In this case, you have to apply a distributed force vertically downward on the highlighted surface.

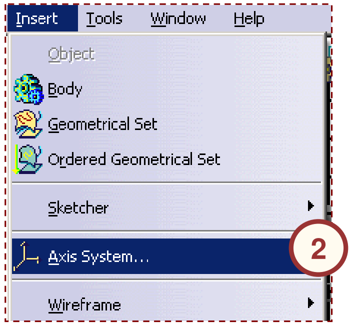
You can create your own user coordinate system with one axis vertically downward and apply a load with respect to this user coordinate system. The general steps to create a user coordinate system are:

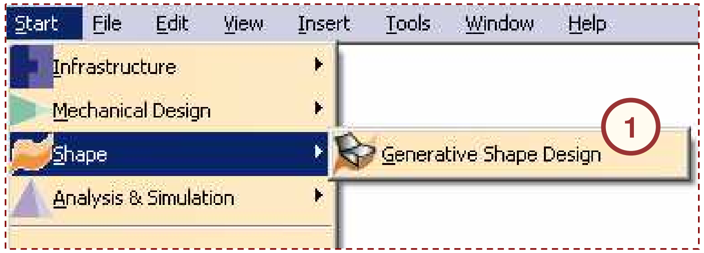
1. Switch to GSD Workbench.
2. In main toolbar select **Insert > Axis-System**

Global coordinate system not aligned with part



Highlighted surface where vertically downward Load to be applied. Load with respect to global axis is not vertically downward.

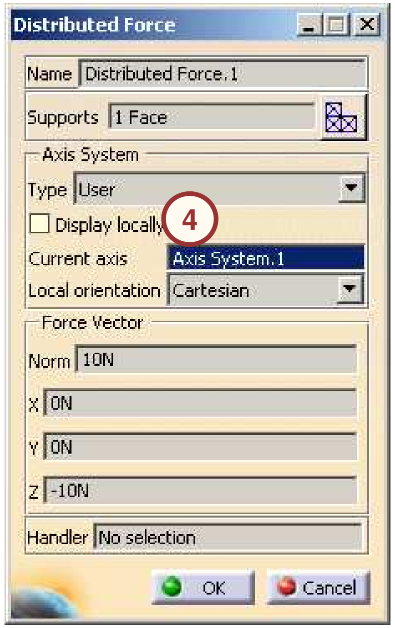




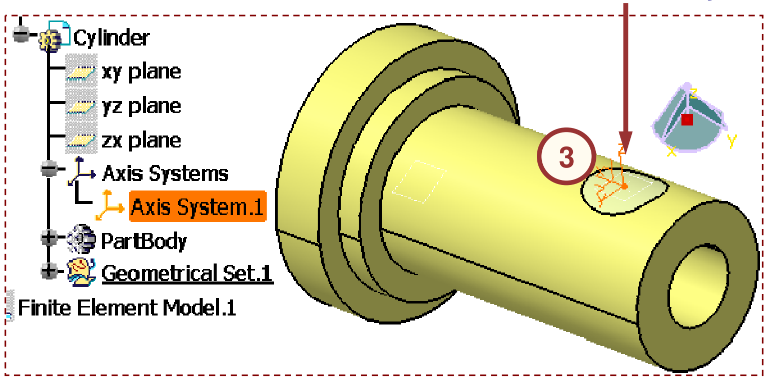
Student Notes:

Creating Local Axis System (2/2)

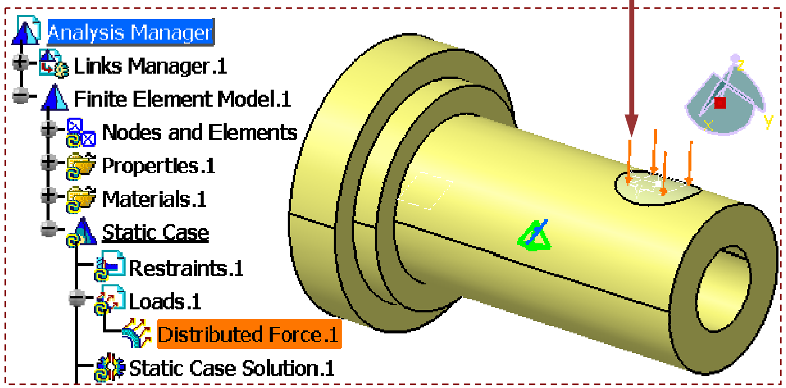
3. Create user-defined Axis System using various options available.
4. While applying the load, select the **Type** as **User** and select *Axis System.1* from specification tree.



User-defined Axis System



Load applied using user defined Axis System



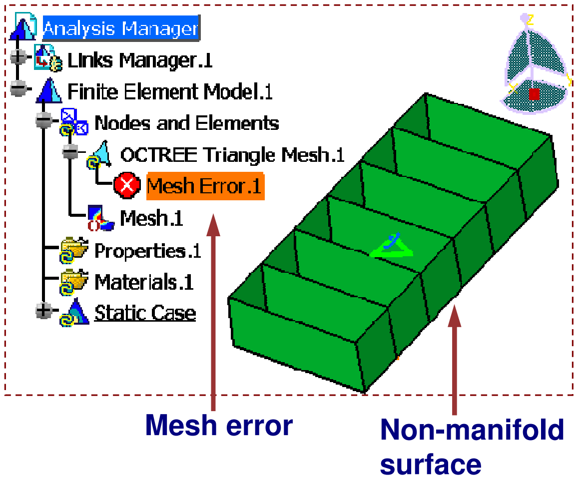
Student Notes:

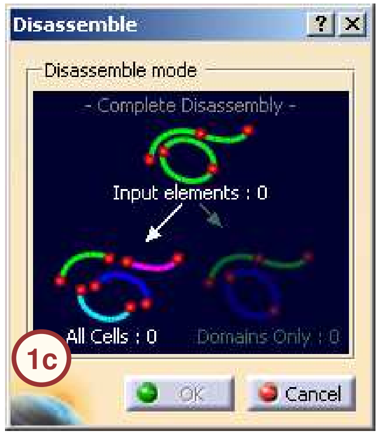
Handling Non-Manifold Surfaces for Analysis (1/2)

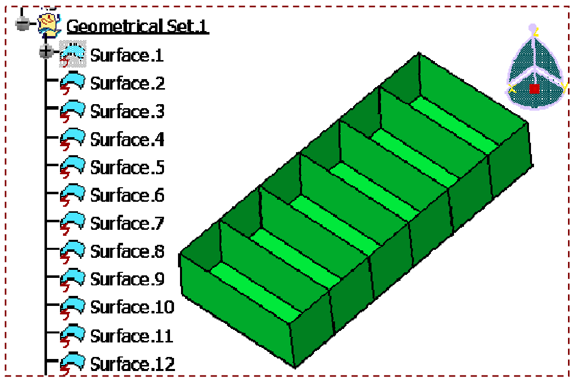
Before you start FE modeling for a surface, ensure that it is a manifold surface as non-manifold surfaces cannot be meshed.

However in certain situations where you have to analyze part with a non manifold surfaces, you can use the following process:

1. Switch to GSD workbench.
2. Disassemble the surfaces.
   a. In **Join-Healing** toolbar, click **Disassemble** icon.
   b. Click on the surface to be disassembled.
   c. Select **All Cells** option and click **OK**.





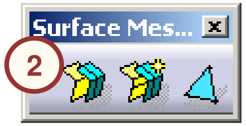


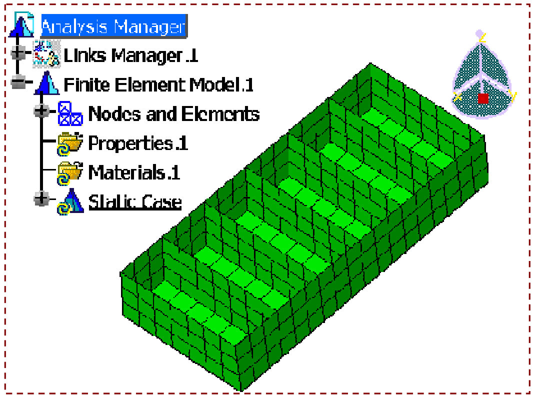
Student Notes:

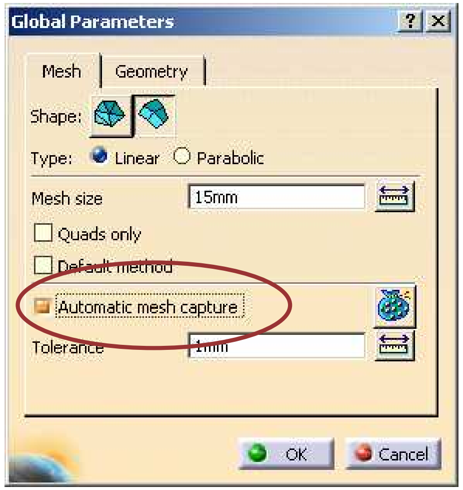
Handling Non-Manifold Surfaces for Analysis (2/2)

2. Mesh each surface independently using Surface Mesher with **Automatic mesh capture** option checked. (This option is available with the Advanced Meshing Tools Workbench).

   This will help to capture the nodes of adjacent surface meshes.





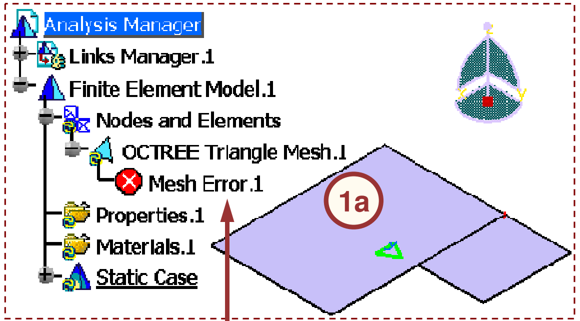


Student Notes:

Handling Overlapping Surfaces for Analysis (1/5)

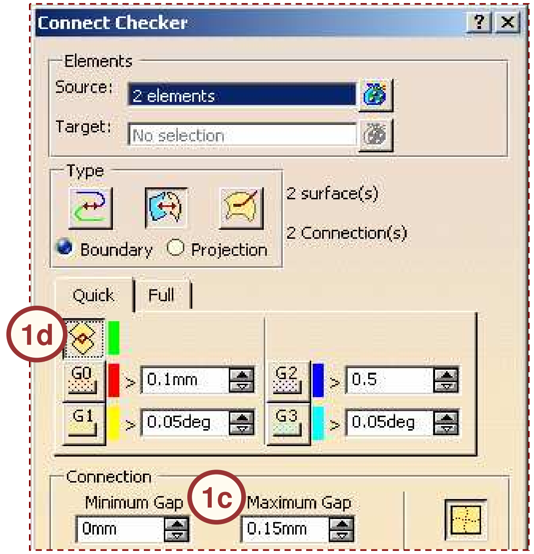
If there are overlapping surfaces, an error message will be displayed while attempting to generate the mesh.

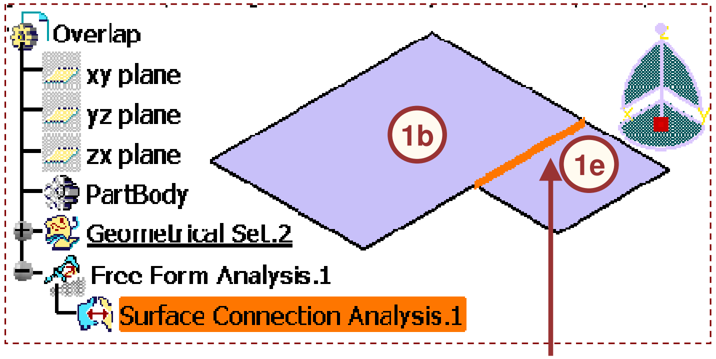
1. Check for overlapping surfaces.
   a. Double-click the surface to enter in GSD workbench.
   b. In main toolbar select **Insert > Analysis > Connect Checker** and select the surface to be checked.
   c. Enter **Minimum Gap** and **Maximum gap** value. This should be greater than **Max:** Value.
   d. Check the **overlapping** option.
   e. Click **OK**. The overlapped area will get highlighted.



Mesh error

Two surfaces are overlapped at common boundary



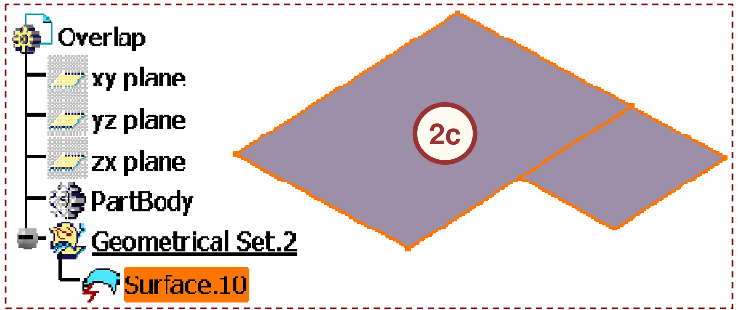


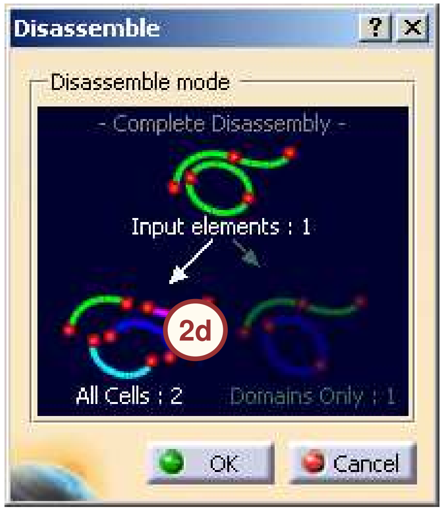
Overlapped area is highlighted using Connect Checker

Student Notes:

Handling Overlapping Surfaces for Analysis (2/5)

2. Correction of the overlapping surface
   a. Double-click on the surface to enter GSD workbench.
   b. In **Join-Healing** toolbar, click **Disassemble** icon.
   c. Click on the surface to be disassembled.
   d. Select **All cells** option and click **OK**.





Student Notes:

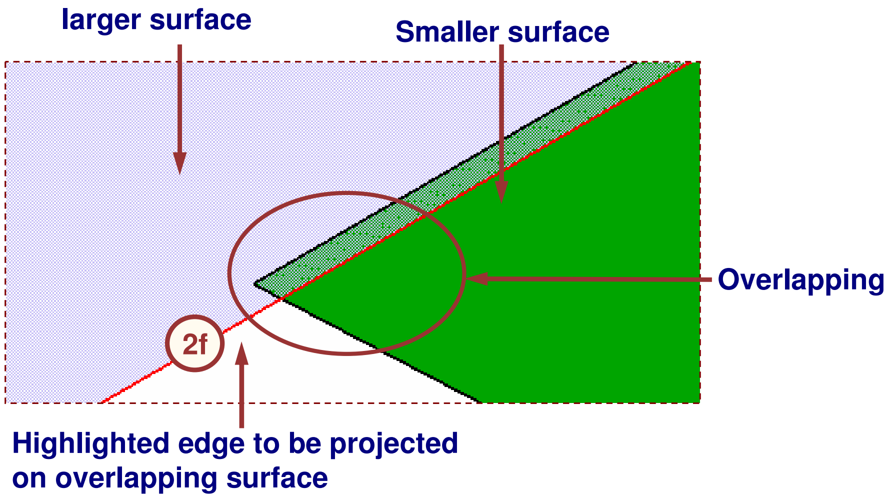
Handling Overlapping Surfaces for Analysis (3/5)

2. Correction of the overlapping surface (continued). Project the edge of one surface on overlapping surface.

   e. In **Project-combine** toolbar, click **Projection** icon.

   f. In **Projected** textbox, select edge of larger surface to be projected on smaller surface.

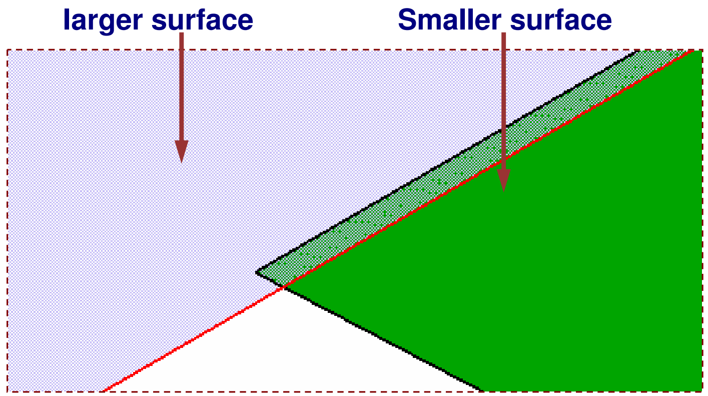
   g. In **support** select smaller surface and click **OK**.

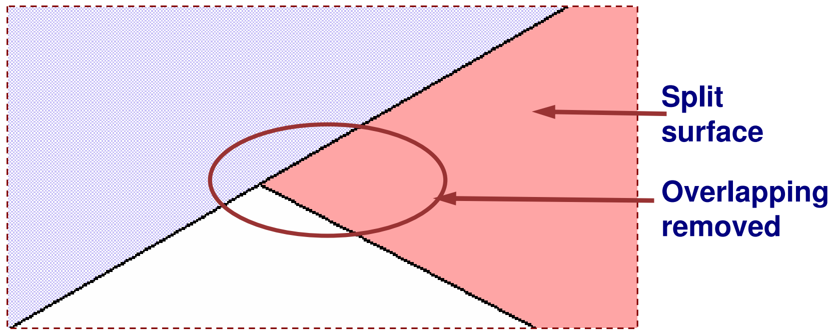


Student Notes:

Handling Overlapping Surfaces for Analysis (4/5)

2. Correction of the overlapping surface (continued). Cut the overlapped portion of the surface.
   h. In **Trim-Split** toolbar, click **Split** icon.
   i. Select smaller surface from specification tree (newly created using disassemble) in **Elements to cut**.
   j. Select the *Project.1* just created in **Cutting Elements**.
   k. Click on **Preview** to ensure that overlapped portion of surface is removed.





Surfaces after removing overlapped part

Student Notes:

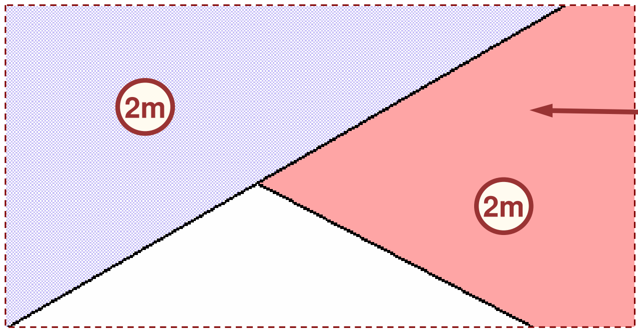
Handling Overlapping Surfaces for Analysis (5/5)

2. Correction of the overlapping surface (continued). Joining the newly created Split Surface and Non-overlapping surface.

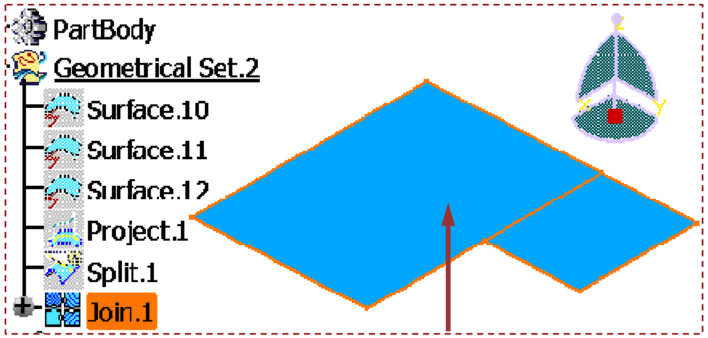
   l. In **Join-Healing** toolbar, click **Join** icon.

   m. Select newly created *split.1* surface and disassembled, larger surface *Surface.11* from specification tree. This will create new surface *Join.1* in specification tree.

Now the geometry can be meshed.



Split surface

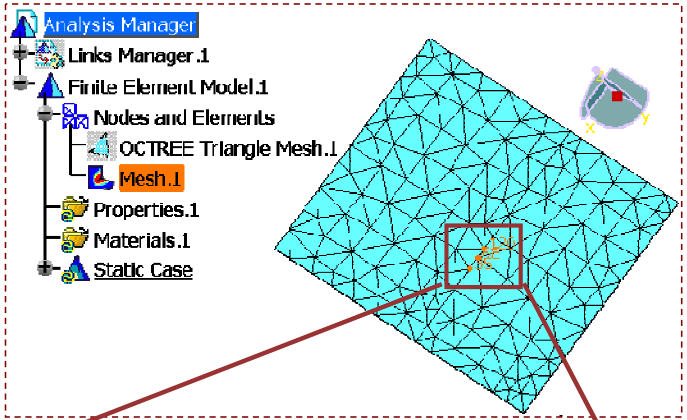


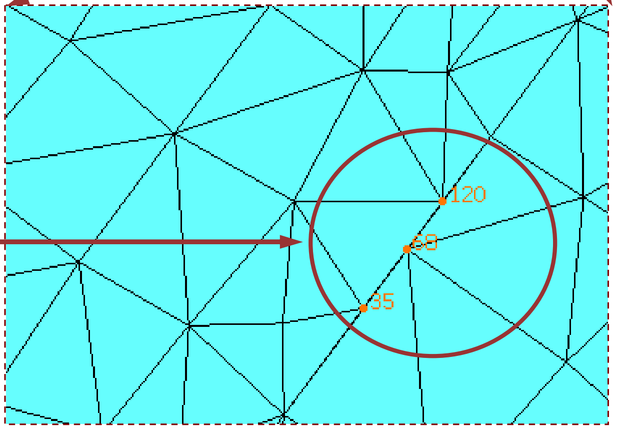
Joined surface

Student Notes:

Preparing Surfaces with Gaps for Analysis (1/4)

If the surfaces have gaps, the surfaces will be meshed, but you will find that the nodes on the gap edges are not interconnected.





Nodes are not interconnected in meshing if surfaces have gaps

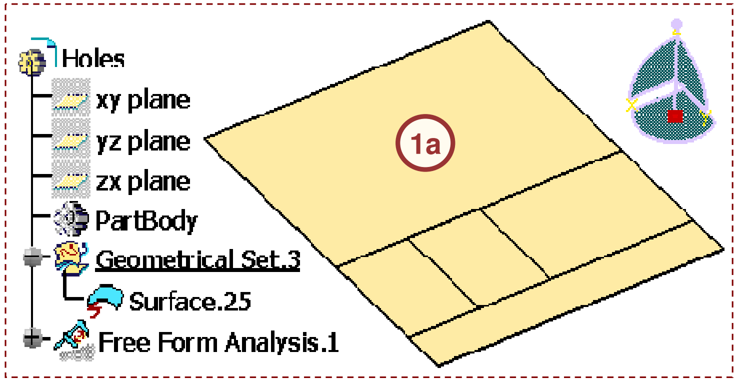
Preparing Surfaces with Gaps for Analysis (2/4)

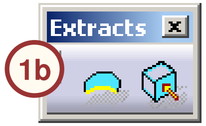
You will see how to correct the surfaces which have gaps.

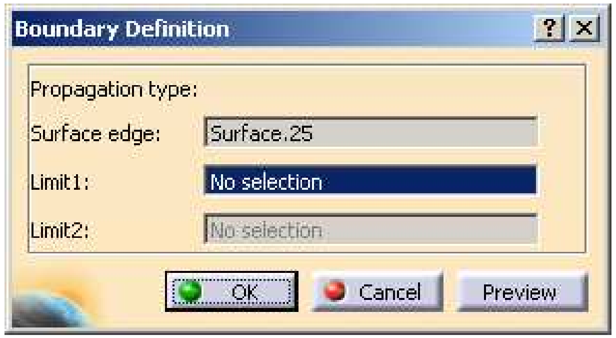
1. Find location of gaps.

   a. Double-click the surface to enter in GSD workbench.

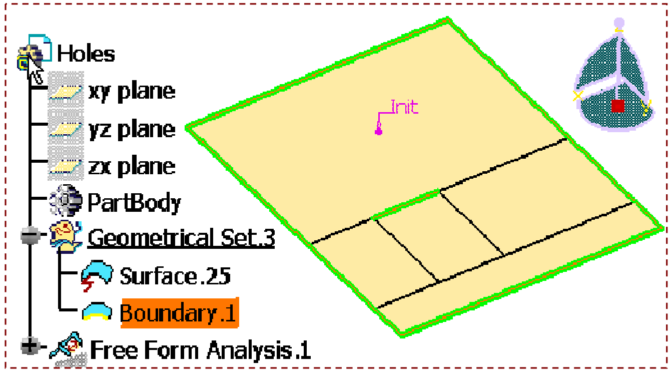
   b. In **Extracts** toolbar, click the **Boundary** icon. Select the *Surface.25* by directly clicking on it. You can also select it through the specification tree. If the surface has gaps the internal boundaries will be highlighted. Click **OK**.







Gap within surface is highlighted

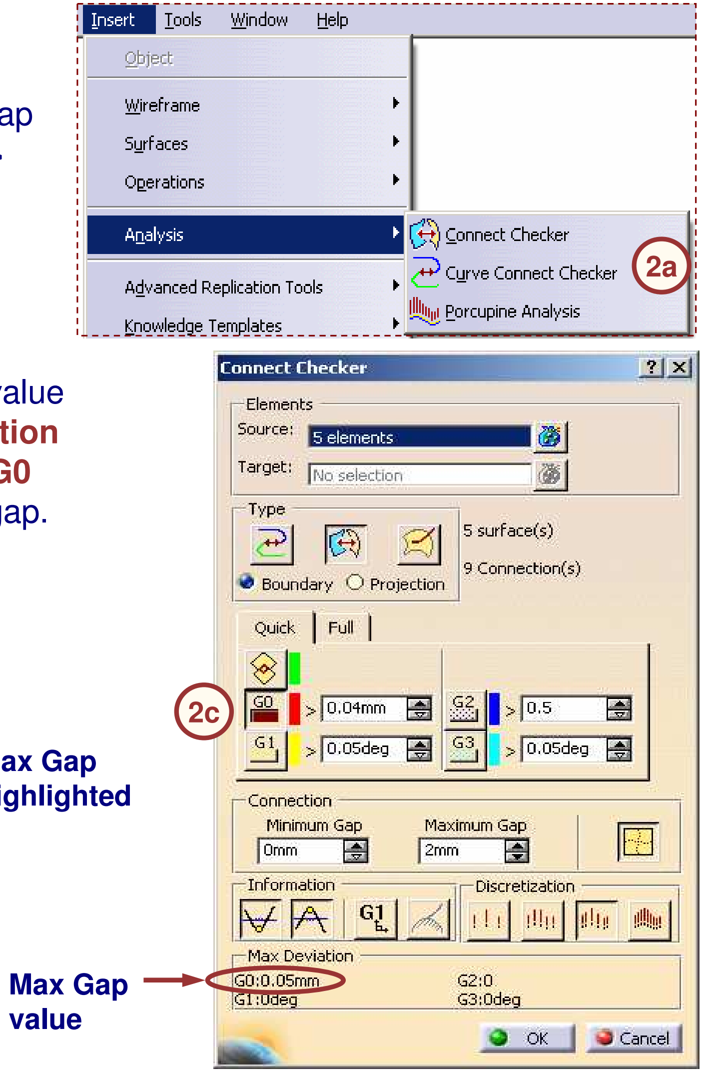


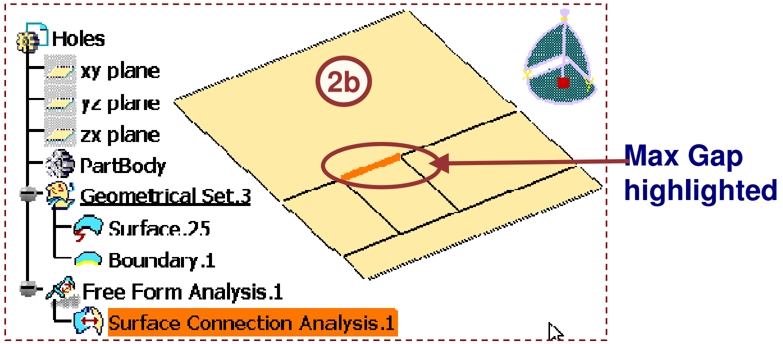
Student Notes:

Student Notes:

Preparing Surfaces with Gaps for Analysis (3/4)

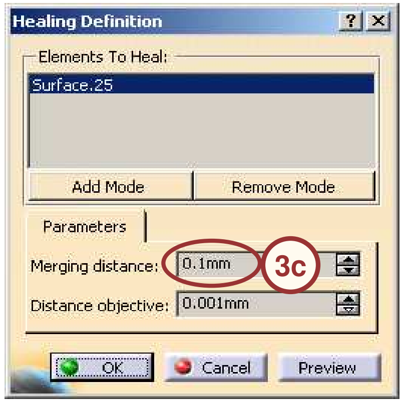
2. Check gap value
   a. Use Connect Checker tool to find gap size. In main toolbar select **Insert > Analysis > Connect Checker**.
   b. Select the surface to be checked.
   c. Select **G0** option. Enter its value as 0.04, keep the value less than **G0** value of **Max Deviation**. Say **Max Deviation** is 0.05 mm then enter 0.04 mm in **G0** field. This will help to highlight the gap.

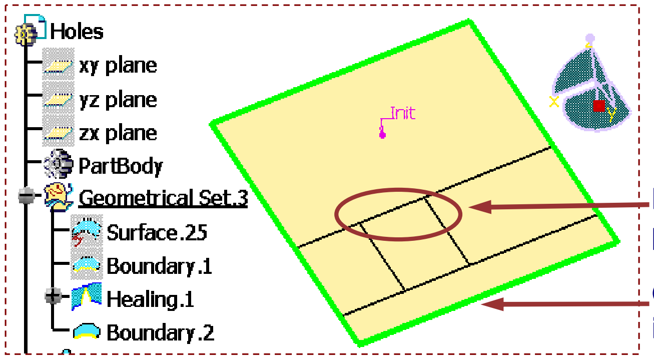




Preparing Surfaces with Gaps for Analysis (4/4)

3. Merging the Gaps
   a. In **Join-Healing** toolbar, select **Healing** icon.
   b. Select the surface to heal.
   c. Enter *0.1* in **Merging Distance** field and click **OK**. (Merging distance should be greater than Max gap)
4. To ensure that gaps are merged, in the **Extracts** toolbar use the **Boundary** icon again. Now it will show you only the outer boundary highlighted.





No gap present

Only Boundary is highlighted

Student Notes:

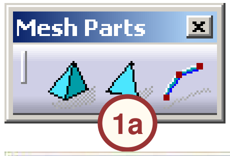
Student Notes:

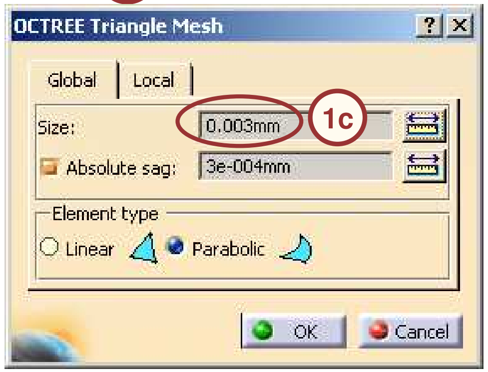
Lowest Mesh size value for Analysis (1/2)

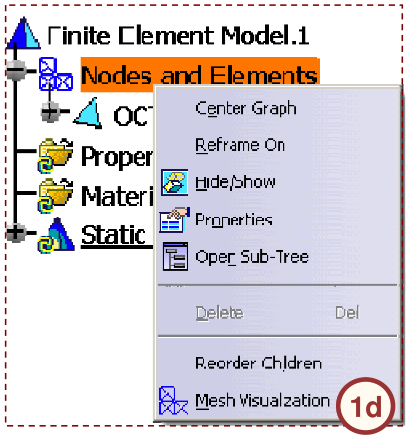
The lowest recommended value for mesh size is 0.2 mm. Mesh size values below this may lead to error while meshing due to the automatic meshing algorithm. You will try meshing the surface with the following mesh sizes.

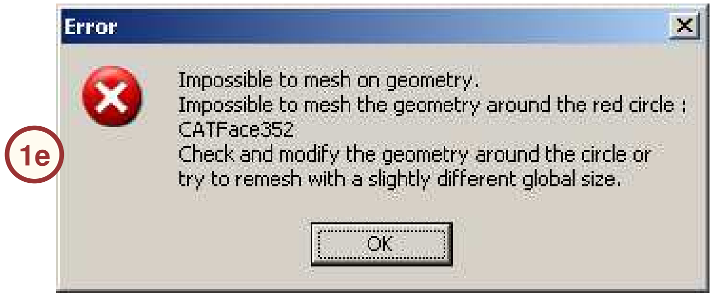
1. Create a mesh whose size is too small.

   a. In **Mesh Parts** toolbar, select **OCTREE Triangle Mesher** icon.
   b. Select the surface to mesh.
   c. Enter *0.003* in **Size** field. Check **Absolute sag** option and enter *0.0003*. Select **Element Type** as **Parabolic** and click **OK**.
   d. Right-click on *Nodes and elements* in specification tree and click on **Mesh Visualization**.
   e. An error message will be displayed.









Error message with 0.003 mm mesh size

Student Notes:

Lowest Mesh size value for Analysis (2/2)

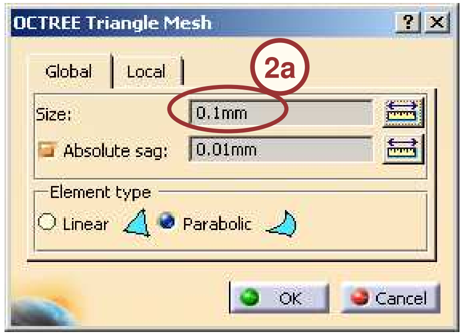
2. Modify the mesh to an acceptable size.

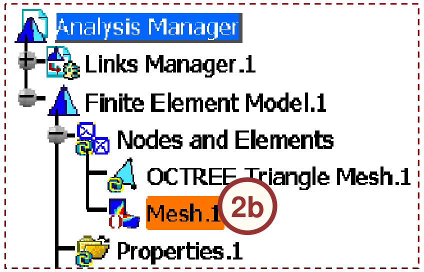
    a. Double-click on **OCTREE Triangle Mesh.1** in the specification tree. Enter *0.1* in **Size** field. Enter **Absolute sag** as *0.01* and click **OK**.

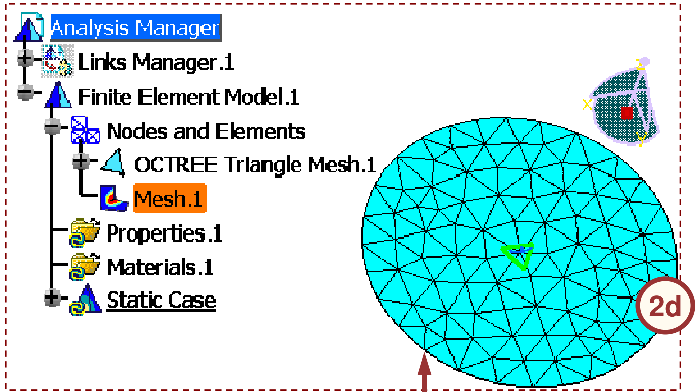
    b. Double-click on *Mesh.1*.

    c. You will get a warning message to update the mesh. Click **OK** to update.

    d. The mesh will be created and displayed.



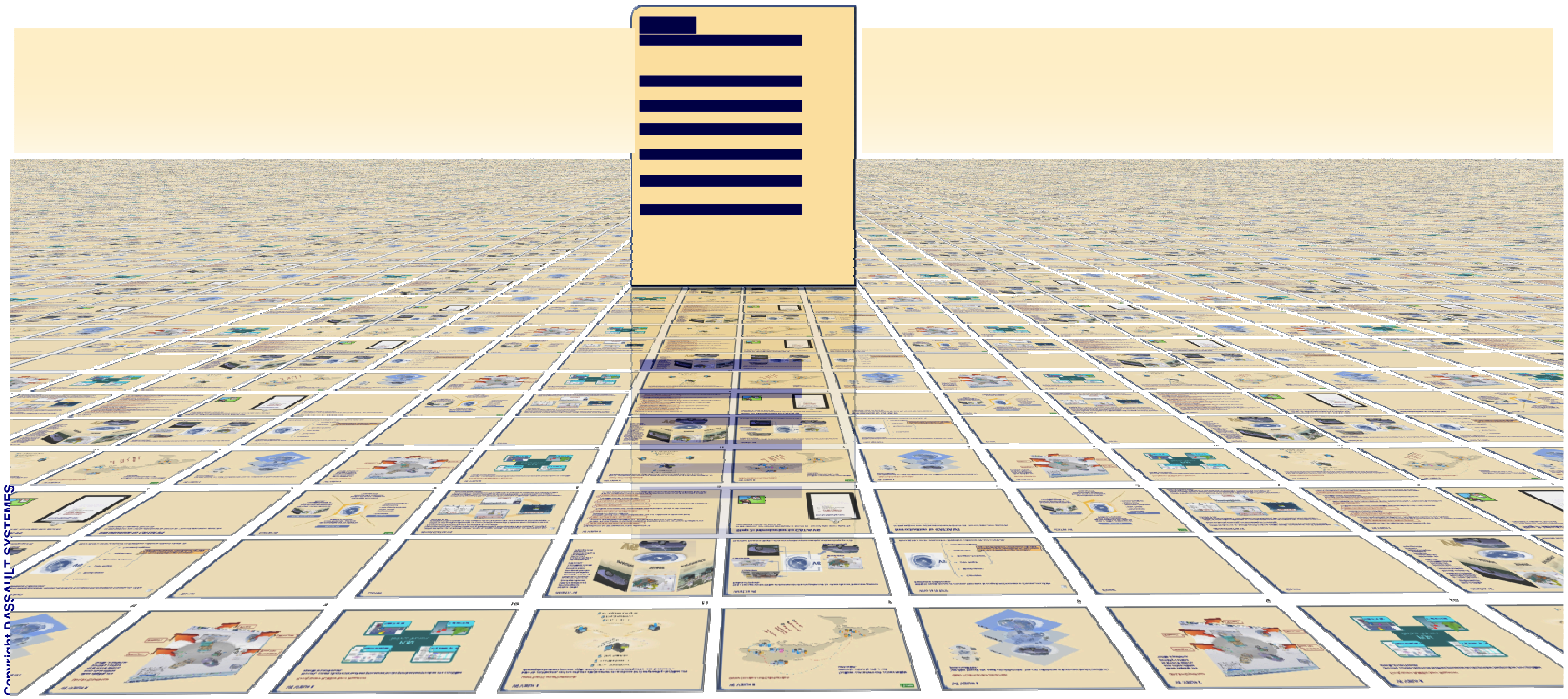




Mesh Image with 0.1 mm mesh size

To Sum Up

In the following slides you will find a summary of the topics covered in this lesson.



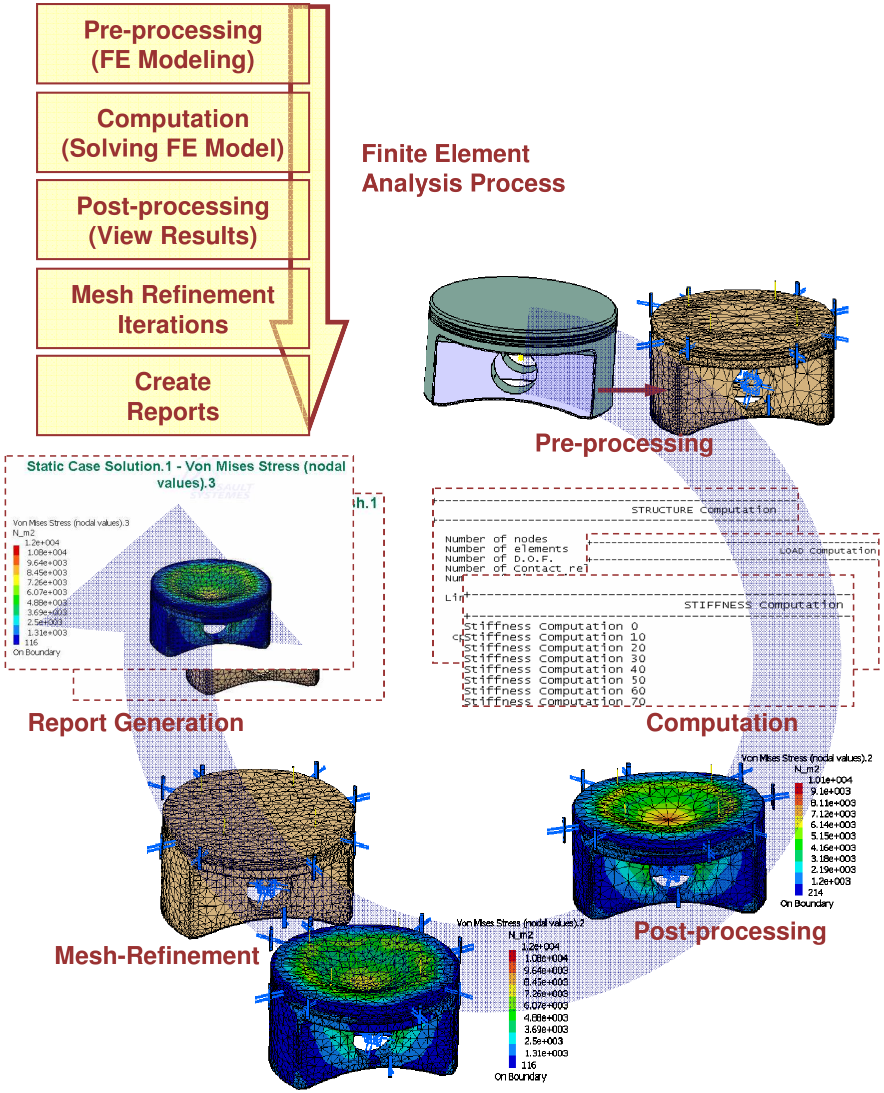
Student Notes:

Student Notes:

Finite Element Analysis Process

Finite Element analysis (FEA) is a numerical tool used to simulate the physical system. In this method the modeled system is broken into smaller geometric shapes, called finite elements, whose behavior can be described mathematically. The elements and their interrelationships are converted into a system of equations which are solved numerically. The overall process is divided into smaller steps as follow

1. Pre-processing: Conversion of actual problem into Finite element problem
2. Computation: Solution of the of the FE problem provided by pre-processing to find out unknown displacement values
3. Post-processing: Calculation of strains and stresses using displacement values. Study of displacements strains and stresses
4. Mesh Refinement: Refinement of the mesh and computation to achieve the required level of accuracy
5. Report Generation: Generation of various plots such as displacements, strains and stresses once the required level of accuracy is generated



Student Notes:

Introduction to GPS workbench

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

1. Open the Generative Structural Analysis workbench. Apply material, mesh the part, apply the restraints and loads
2. Compute the Analysis
3. Visualize the results
4. Interpret the results and Mesh Refinement
5. Manage the results

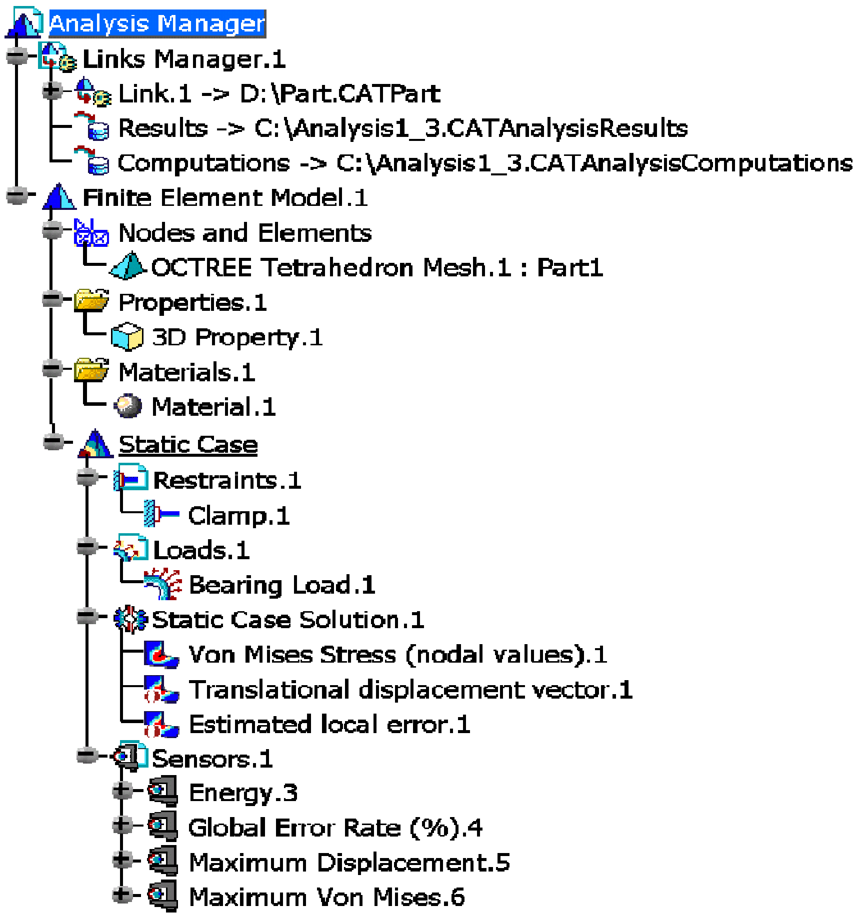
According to variation in load with respect to time, the FEA for structures can be classified as:

- Static Analysis
- Transient Analysis
- Harmonic Analysis

According to the way the structure reacts to the load, the FEA for structures can be classified as:

- Linear analysis
- Non-linear Analysis

Entities created during process of GPS Static Analysis gets mapped in the tree-structure as shown.



GPS Static Analysis Tree Structure

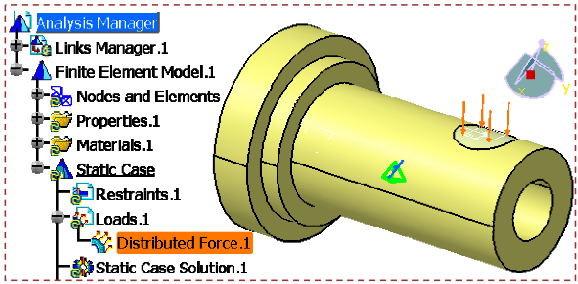
Student Notes:

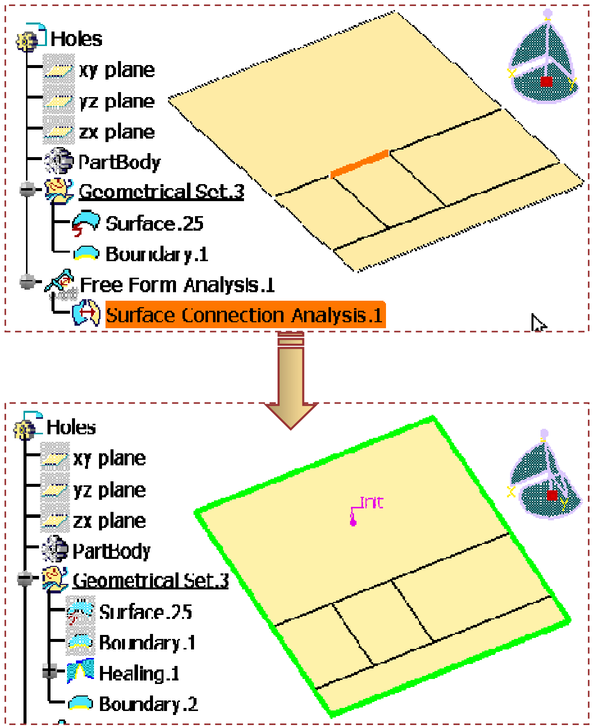
Preparing Parts and Surfaces for Analysis

Before you switch to Generative Structural Analysis workbench, you may need to make some modifications in the existing CATIA geometry or modify some of the CATIA settings. This may include:

- Changing default units
- Creating support on part of a Face
- Creating Local Axis System
- Handling Non-Manifold Surfaces
- Handling Overlapping Surfaces
- Preparing Surfaces with Gaps for Analysis

In addition, when meshing very small models you may need to take into account Lowest Mesh Size value for Analysis.





Student Notes:

Tools Used for Preparing Parts and Surfaces for Analysis (1/2)

1 **Extract:** lets you perform an extract from elements (curves, points, surfaces, solids, volumes etc).

2 **Split:** lets you split the surfaces.

3 **Fill:** lets you create fill surfaces between number of boundary segments.

4 **Sew Surface:** lets you add or remove material by modifying the surface of the volume.

5 **Disassemble:** lets you disassemble the multi-cell bodies into mono-cell or mono-domain bodies, whether curves or surfaces.

6 **Surface Mesher:** lets you mesh the surface part by entering into the Surface Mesher workshop.

7 **Connect Checker:** lets you analyze the connection between the surfaces' borders and their projection on a surface.

Student Notes:

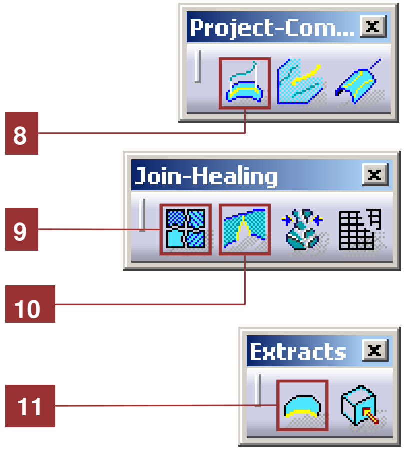
Tools Used for Preparing Parts and Surfaces for Analysis (2/2)

8 **Projection:** lets you create geometry by projecting one more elements onto a support.

9 **Join:** lets you join the multi-sections and swept surfaces.

10 **Healing:** lets you heal the surfaces (i.e. fill any gap that may be appearing between two surfaces).

11 **Boundary:** lets you create the boundary curve of a surface or the boundary point of a curve.



Student Notes:

Exercise 1B

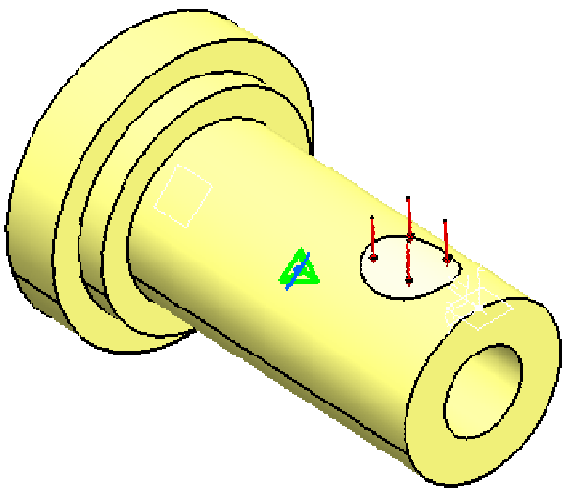
Recap Exercise

30 min

In this exercise, you will apply a load on a part of the face. You will use a User-defined coordinate system to apply a Distributed load. Detailed instructions for this exercise are provided.

By the end of this exercise you will be able to:

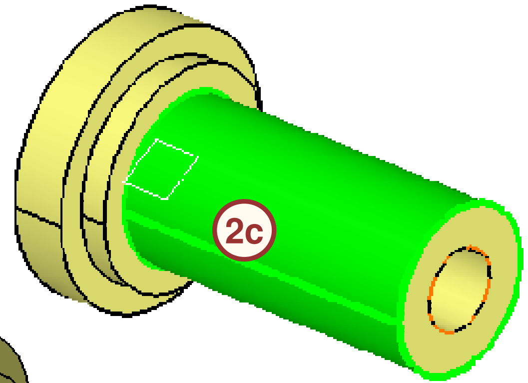
- **Create a support to apply constraints or loads on a part of the face**
- **Create a user defined Axis System**
- **Apply a load with a user-defined coordinate system**

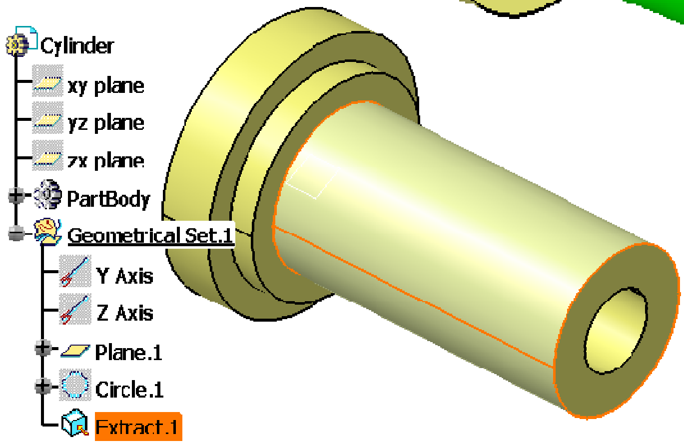


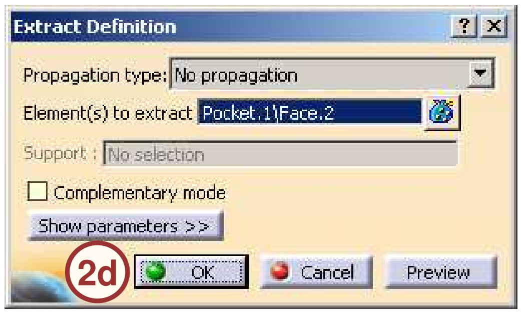
Exercise 1B (1/8)

1. **Open a part.**
   - Open 1B_Part_of_Face_Start.CATPart
     a. Select **File > Open.**
     b. Browse the file and select the file.
     c. Click **OK**.

2. **Extract the outer surface of shaft.**
   - Extract the outer surface of the shaft to provide the base surface.
     a. Access the GSD workbench.
     b. In **Extracts** toolbar, click on the **Extract** icon.
     c. Select the outer surface of shaft as shown.
     d. Click **OK**.





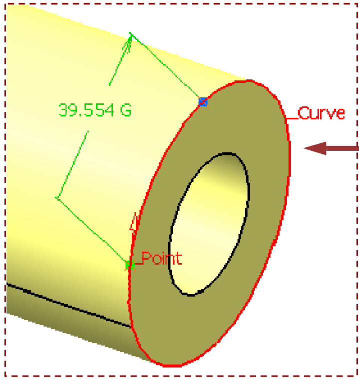


Student Notes:

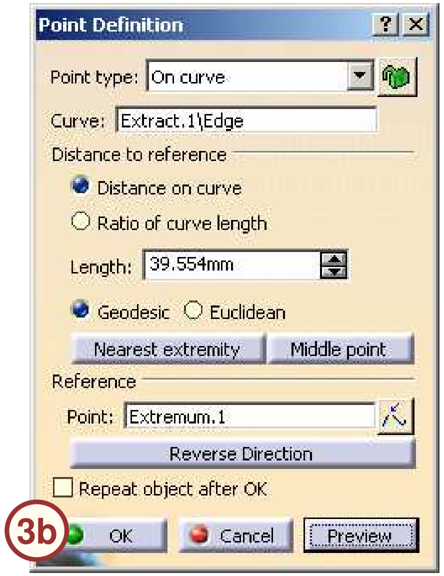
Student Notes:

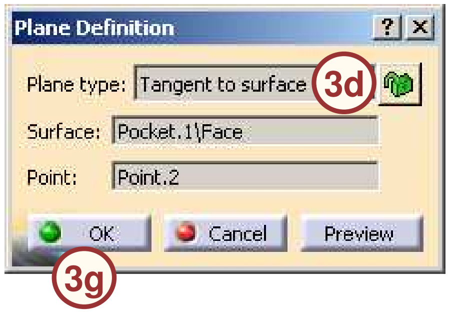
Exercise 1B (2/8)

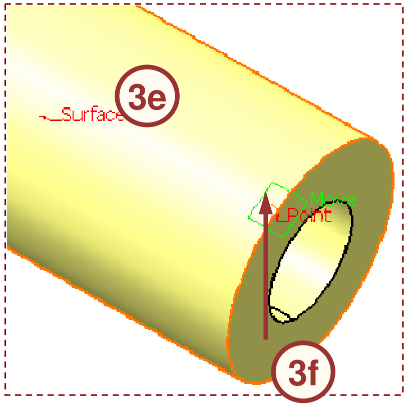
3. **Create an Axis System for geometry on support surface.**
   - Define an Axis system by creating a plane tangent to the surface.
     a. In **Reference Elements** toolbar, click on the **Point** icon.
     b. Use options as shown and enter *39.554mm* in the **Length:** field. Click **OK.**
     c. In **Reference Elements** toolbar, click the **Plane** icon.
     d. Select **Plane Type** as **Tangent to surface.**
     e. Select surface in **Surface** field as shown.
     f. Select Point.2 in **Point** field as shown.
     g. Click **OK.**



Curve to be selected to as support to create a point







Point to be selected

Student Notes:

Exercise 1B (3/8)

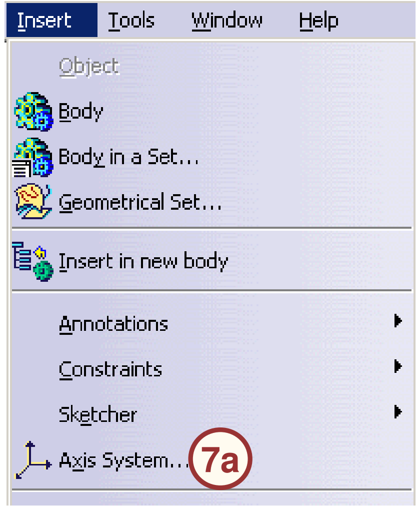
h. Select **Insert > Axis System…**

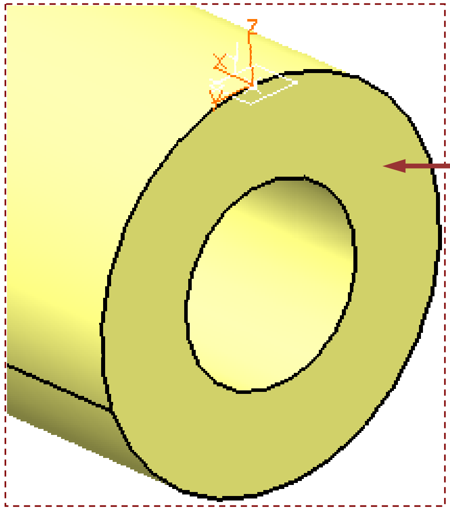
i. In **Origin** field, select the *Point.2* from specification tree.

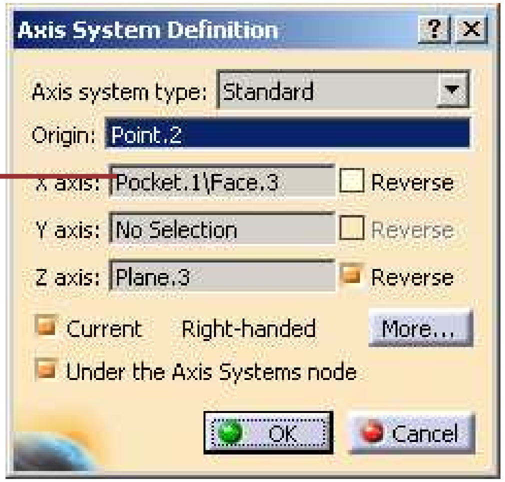
j. In **X axis** field, select the vertical surface of shaft as shown. Keep **Y axis** field as it is.

k. In **Z axis** field, select the *Plane.3* just created, from specification tree. Check **Reverse** option to make upward direction positive.

l. Click **OK**.





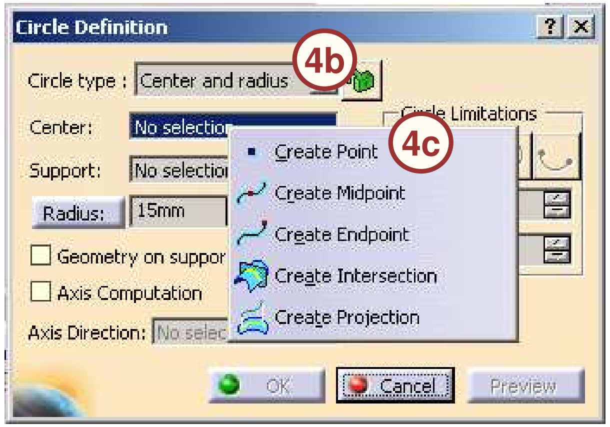


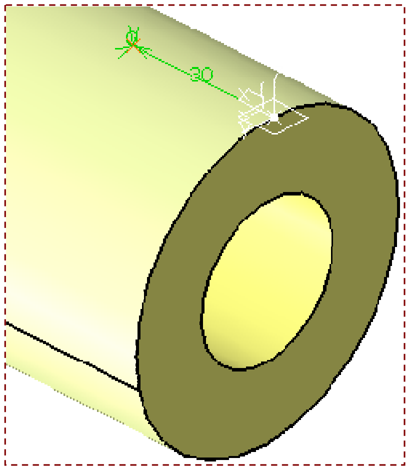
Student Notes:

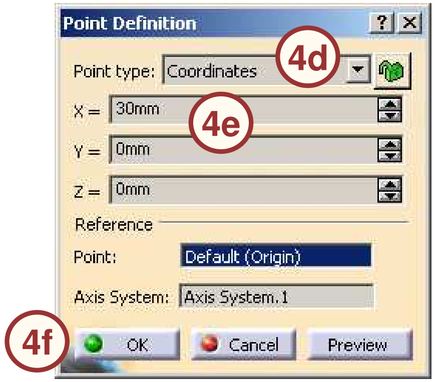
Exercise 1B (4/8)

4. Create a geometry on support surface.

- Create a circle using User-defined Axis System.
  - a. In **Circle-Conic** toolbar, click **Circle** icon.
  - b. Select **Center and radius** in **Circle type.**
  - c. In **Center** field, right-click to open the contextual menu and select **Create Point** option.
  - d. In **Point Definition** dialogue box, select **Point type** as **Coordinates**.
  - e. Enter distance *30 mm* in **X =** field.
  - f. Click **OK. Point.3** will be created in **Center** field.







Student Notes:

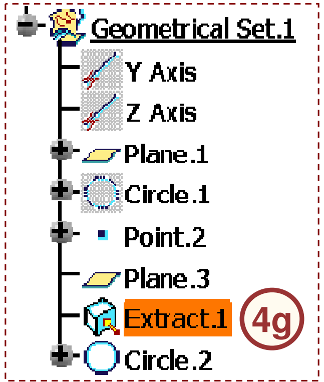
Exercise 1B (5/8)

g. In **Circle Definition** dialogue box, click in the **Support** field and select *Extract.1* from specification tree.

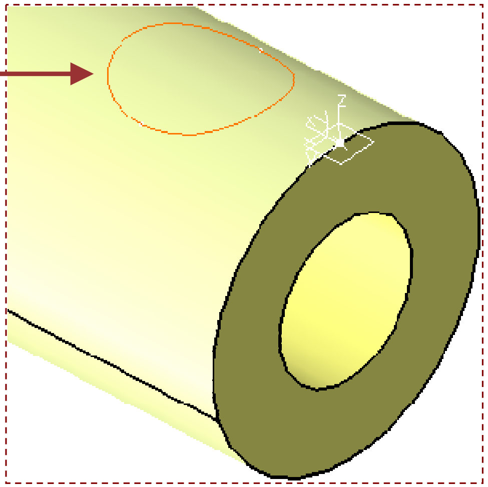
h. Enter Radius as *15 mm.*

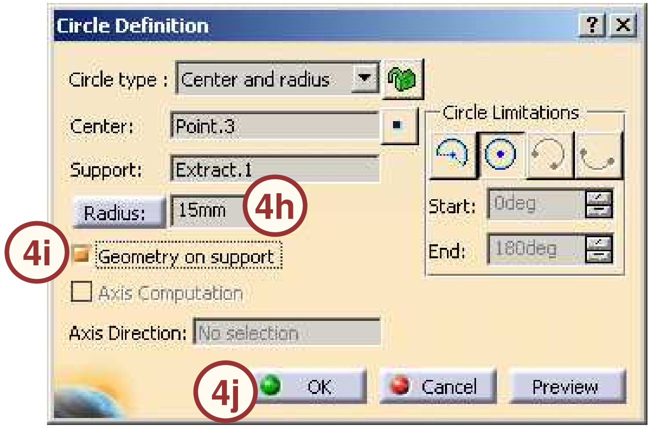
i. Check the option **Geometry on support.**

j. Click **OK.**



Circle created



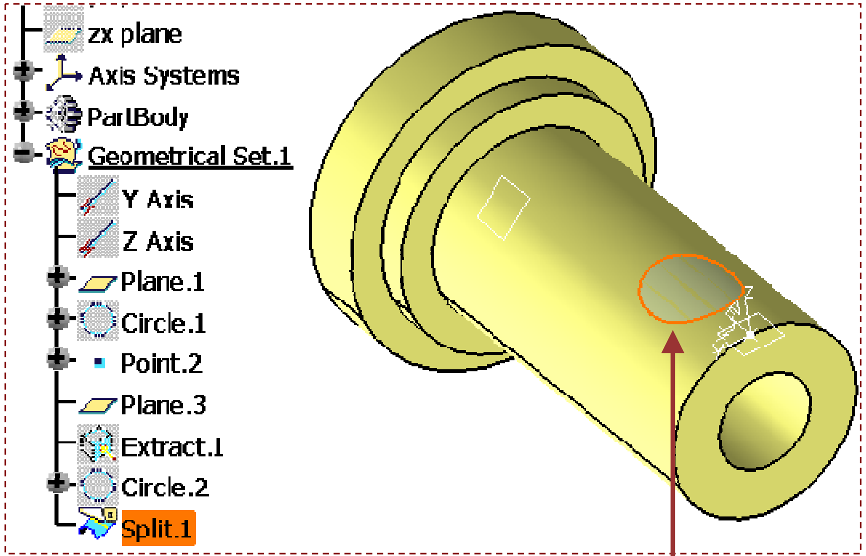


Student Notes:

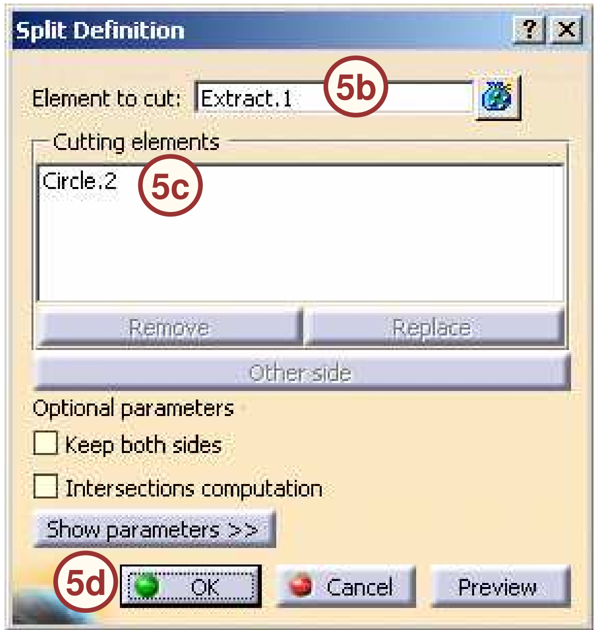
Exercise 1B (6/8)

5. Cut the extracted surface with geometry.

- Cut the surface with geometry to create support surface.
  - a. In **Trim-Split** toolbar, click **Split** Icon.
  - b. Select *Extract.1* in **Element to cut** field.
  - c. Select *Circle.2* in **Cutting elements** field.
  - d. Click **OK**.



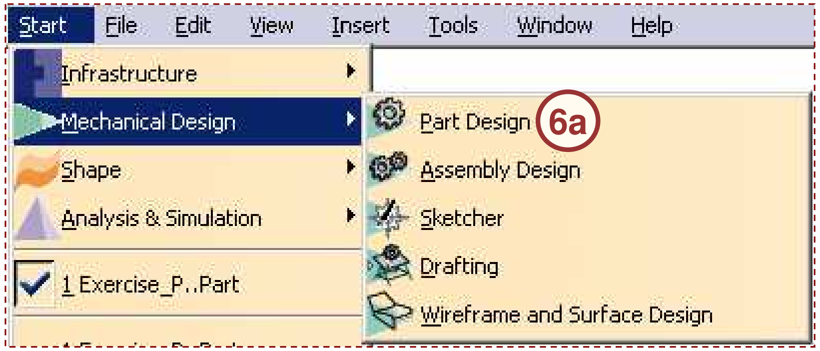
Split surface

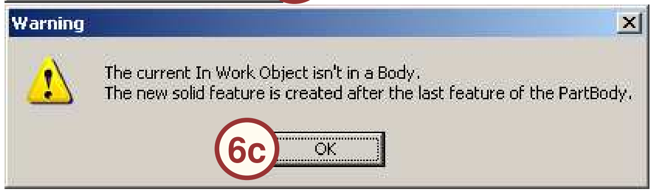


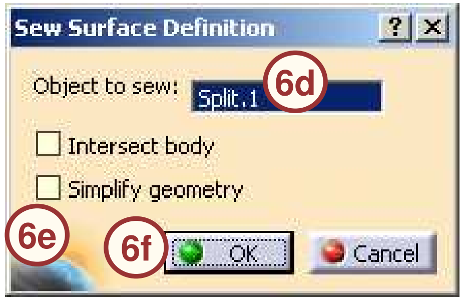
Student Notes:

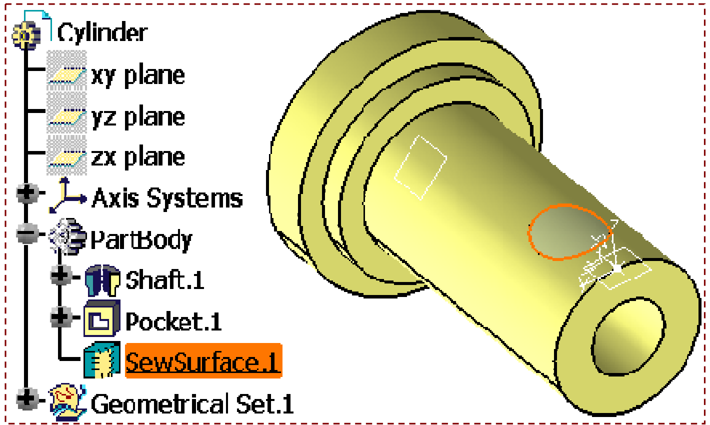
Exercise 1B (7/8)

6. **Sew the support surface to Part.**
   - Sew the surface created to part.
     a. Switch to **Part Design** workbench.
     b. In **Surface-Based Features** toolbar, click **Sew Surface** icon.
     c. Click **OK** on the warning message.
     d. Select *Split.1 from* specification tree in **Object to sew** field.
     e. Uncheck **Simplify geometry** option.
     f. Click **OK**.







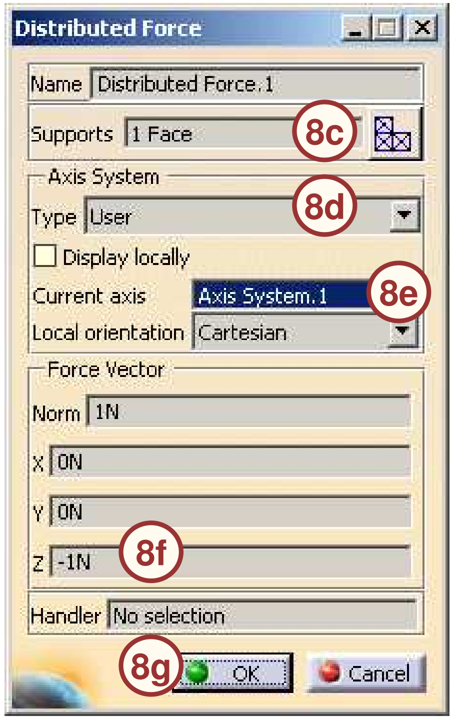


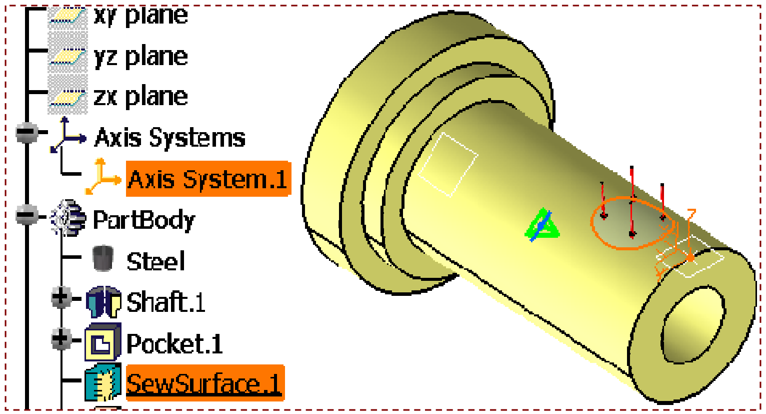
Student Notes:

Exercise 1B (8/8)

7. Apply Force using user-defined Axis System

- Apply Distributed Force.
  a. Select **Start > Analysis & Simulation > Generative Structural Analysis.** Click **OK** to the warning detected.
  b. In **Forces** toolbar, click **Distributed Force** icon.
  c. Select *SewSurface.1* from the specification tree in **Supports** field.
  d. In **Axis System**, select **User** in **Type** dropdown.
  e. Select *Axis System.1* in **Current axis** field.
  f. In **Force Vector**, enter *-1N* in **Z** field.
  g. Click **OK**.

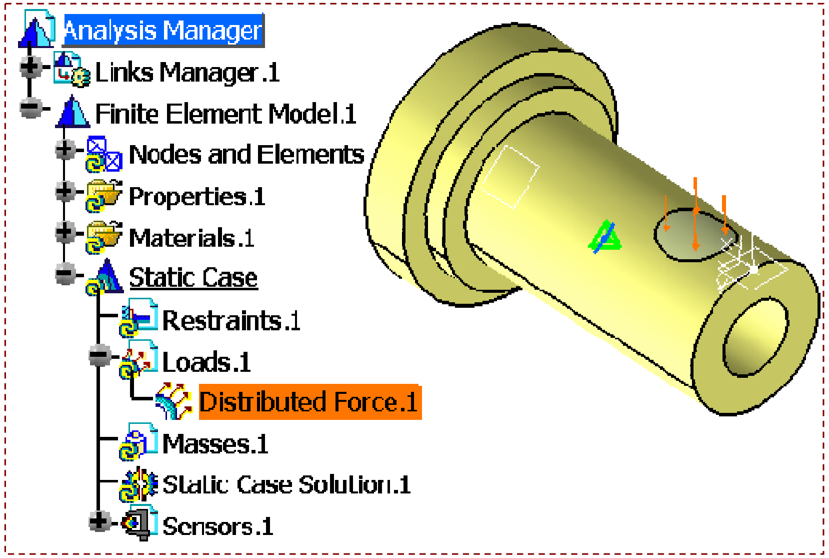




Student Notes:

Exercise 1B: Recap

- ✓ Create a support to apply constraints or loads on a part of the face.
- ✓ Create a user defined Axis System.
- ✓ Apply a load with a user-defined coordinate system.



Student Notes:

Exercise 1C

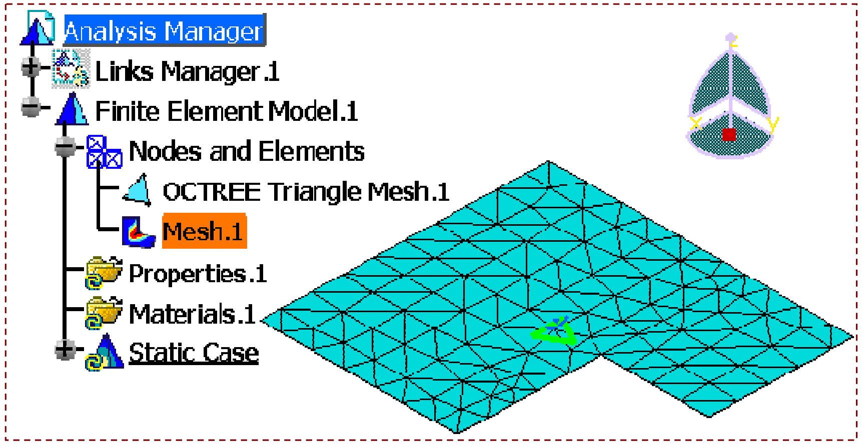
Recap Exercise

60 min

In this exercise, you will mesh an overlapping surface, correct the overlapping surfaces and re-mesh the corrected surface. Detailed instructions for new topics are provided for this exercise.

By the end of this exercise you will be able to:

- **Detect overlapping surfaces**
- **Correct the overlapping surfaces**
- **Mesh the corrected surface**

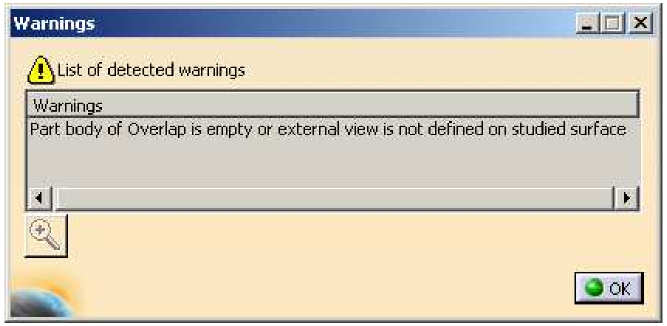


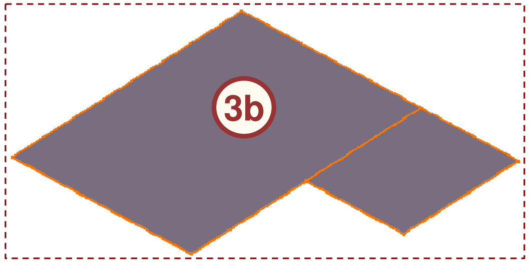
Student Notes:

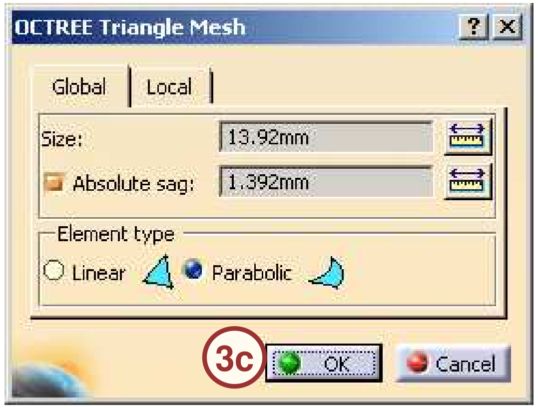
Exercise 1C (1/8)

1. **Open a part.**
   - Open 1C_Overlapping_Start.CATPart
2. **Create a Static Analysis Case.**
   - Access the Generative Structural Analysis workbench.
   - Create a Static Analysis Case. Click **OK** for the warning message.
3. **Mesh the surface.**
   - Mesh the surface with OCTREE Triangle Mesher.
     a. In **Mesh Parts** toolbar, click **OCTREE Triangle Mesher** icon.
     b. Click on the surface.
     c. Keep default values as shown and click **OK**.

Please ensure that you are set to mm.





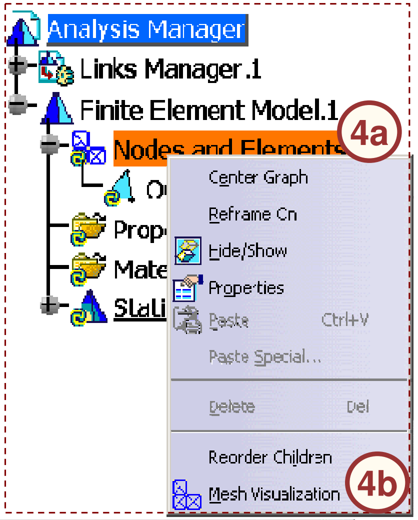


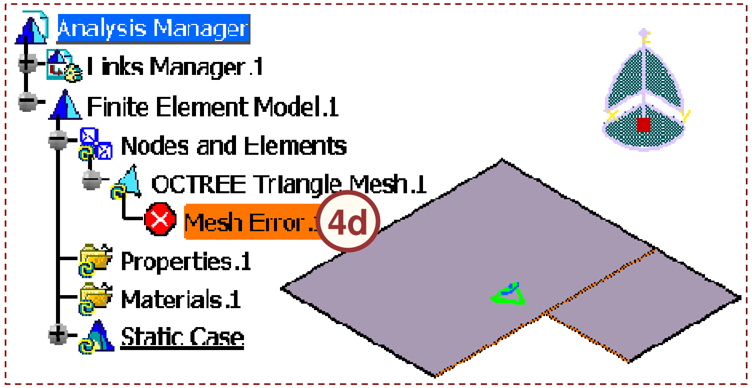
Student Notes:

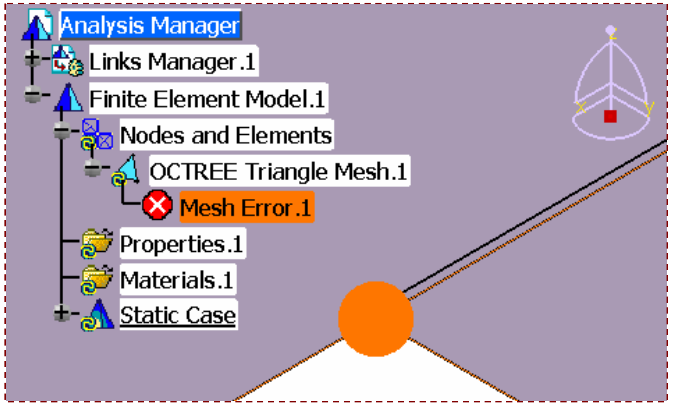
Exercise 1C (2/8)

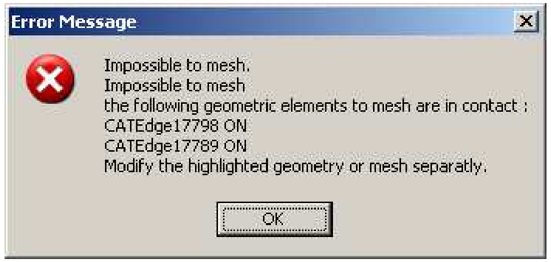
4. View the mesh.

- Create Mesh visualization to create Mesh image.
  - a. Right-click on *Nodes and Elements.*
  - b. Select Mesh visualization.
  - c. Click **OK** for the warning message to update the mesh.
  - d. Double-click on the *Mesh Error.1.*





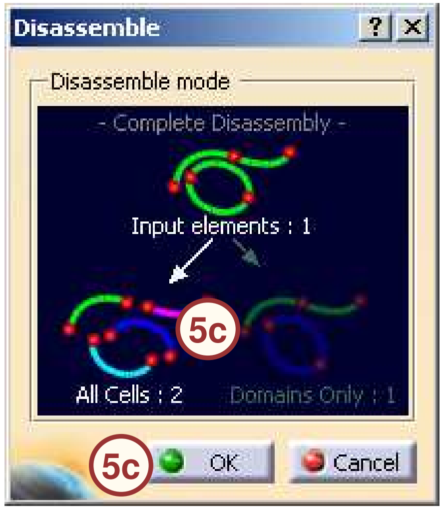


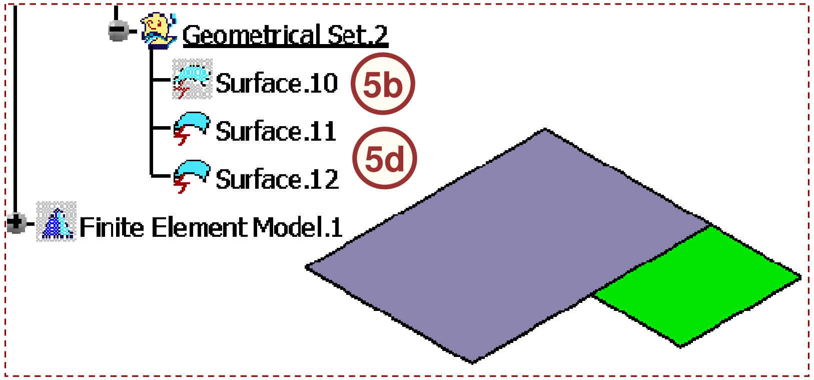


Student Notes:

Exercise 1C (3/8)

5. **Correct overlapping surfaces.**
   - Double-click on the surface to switch to GSD workbench.
   - Disassemble the surfaces.
     a. In **Join-Healing** toolbar , click **Disassemble** icon.
     b. Select the surface to be disassembled.
     c. Select **All cells** option and click **OK** to create the two new surfaces.
     d. Hide the original surface *Surface.10* in order to see only the new surfaces. The different colors shown below are only to simplify identification.

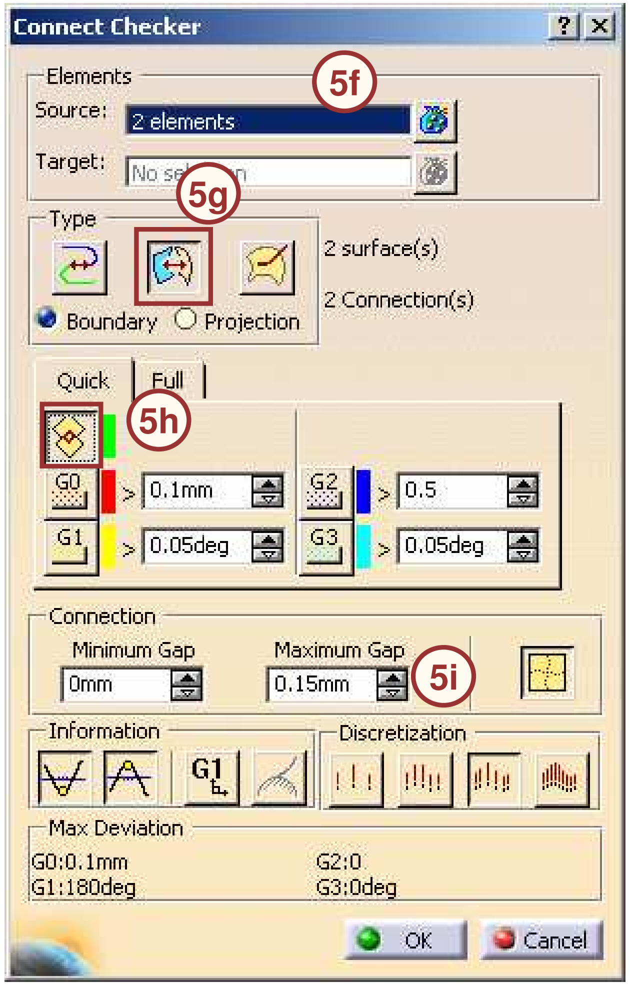


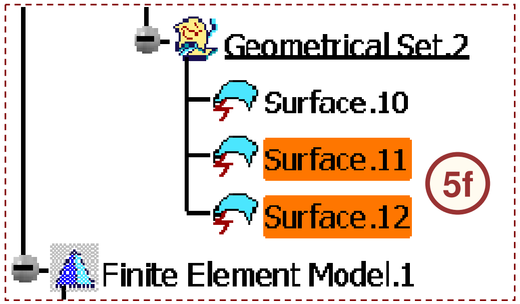


Student Notes:

Exercise 1C (4/8)

- Use Connect Checker tool to highlight gap.
  - e. Select **Insert > Analysis > Connect Checker.**
  - f. Select the *Surface.11* and *surface.12*.
  - g. Select the **Surface-Surface connection** as type.
  - h. In the Quick tab select **overlap Defect** option.
  - i. Enter *0.15 mm* in **Maximum gap** and click **OK**. The overlapped area will get highlighted.

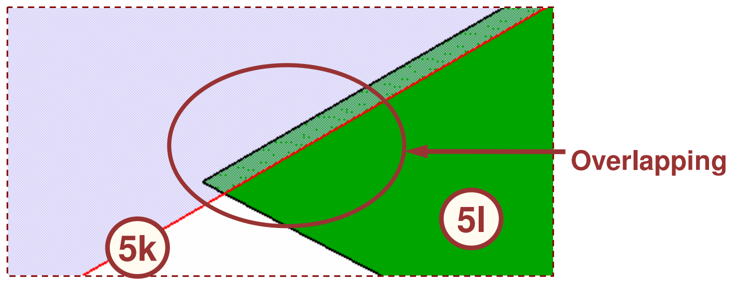


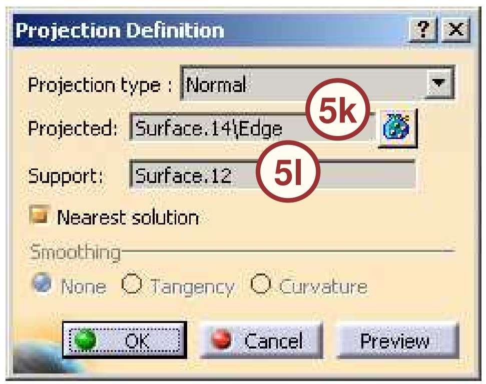


Student Notes:

Exercise 1C (5/8)

- Split an overlapping surface.
  - j. In **Project -Combine** toolbar, click **Projection** icon.
  - k. In **Projected** field, select edge of large surface to be projected on overlapping surface.
  - l. In **Support** field, select small surface and click **OK**.





Student Notes:

Exercise 1C (6/8)

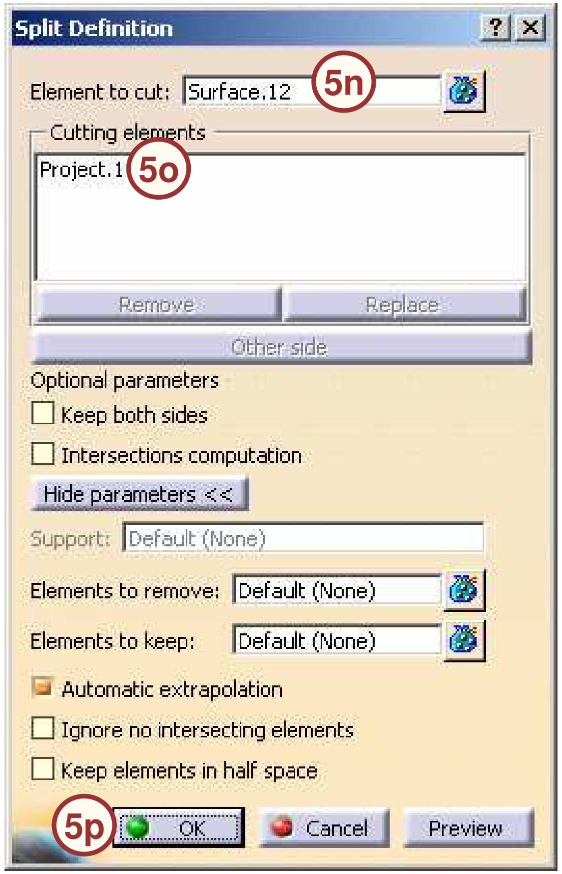
m. In **Trim-Split** toolbar, click **Split** icon.

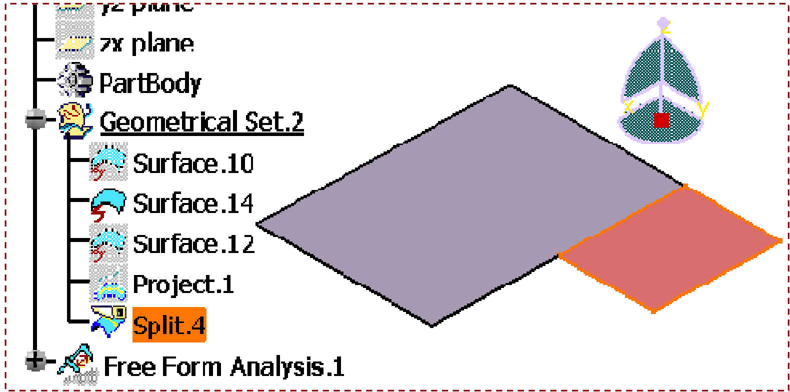
n. Select the small surface *surface.12* in **Elements to cut field**.

o. Select the projected edge *Project.1* in **Cutting elements** field.

p. Click **OK**.

The different colors shown below are only to simplify identification of the new split surface.



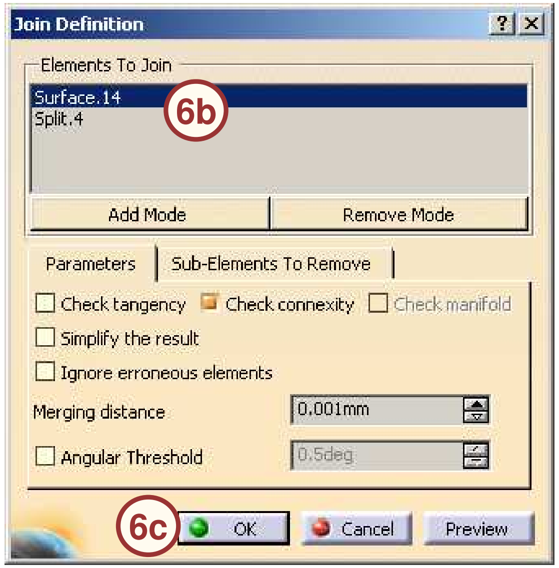


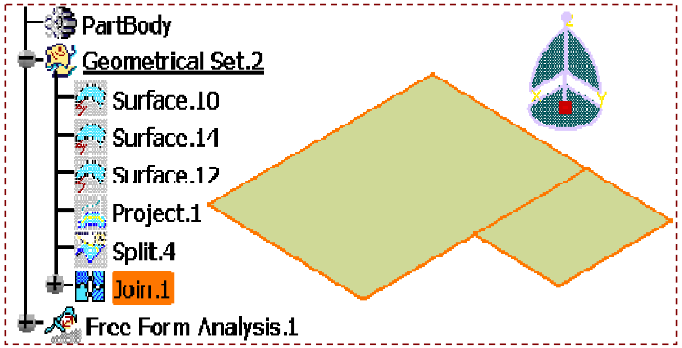
Student Notes:

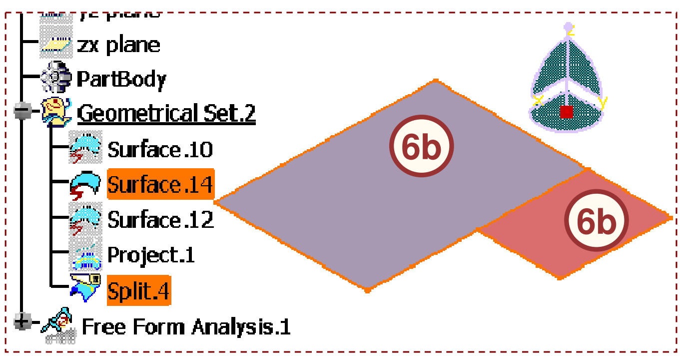
Exercise 1C (7/8)

6. Create a single surface element.

- Join the two surfaces.
  a. In **Join-Healing** toolbar, click **Join** icon.
  b. Select newly created *Split.4* surface and disassembled, larger surface *Surface14*.
  c. Click **OK**. This will create new surface *Join.1* in specification tree.





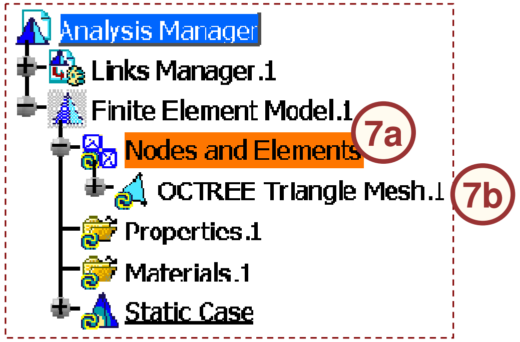


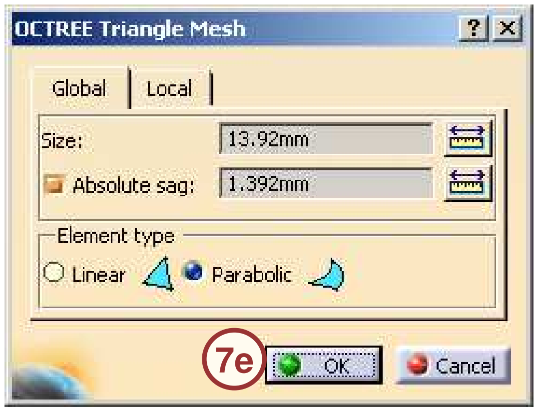
Student Notes:

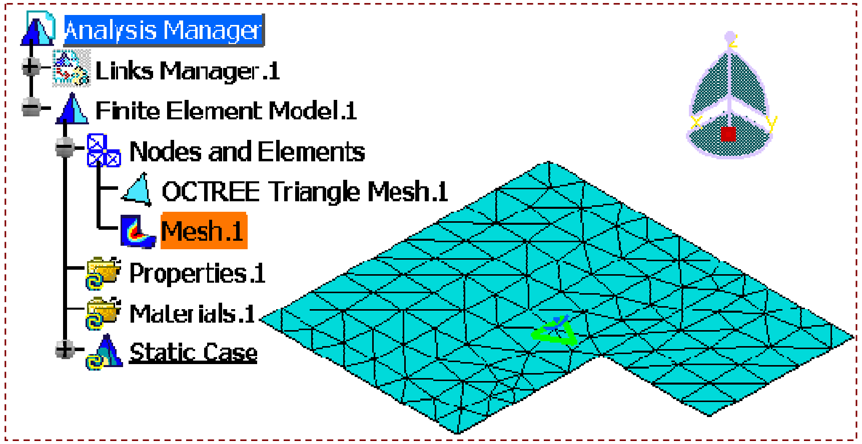
Exercise 1C (8/8)

7. Create and view the Mesh Image.

- Switch to GPS workbench.
  - a. Double-click on Nodes and Elements to switch to GPS workbench.
- Create new Mesh.
  - b. Right-click on OCTREE Triangle Mesh.1 and select **Delete.**
  - c. In **Mesh Parts** toolbar, select **OCTREE Triangle Mesher** icon.
  - d. Click the surface.
  - e. Keep default values as shown and click **OK**.
- Create Mesh visualization to create Mesh image.



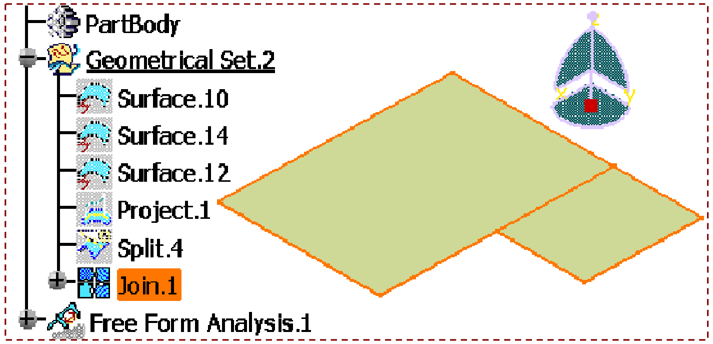


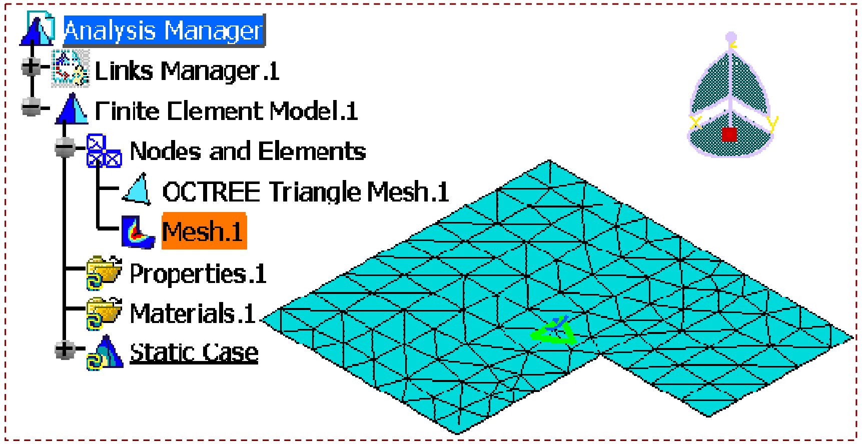


Student Notes:

Exercise 1C: Recap

- ✓ Detect overlapping surfaces.
- ✓ Correct the overlapping surfaces.
- ✓ Mesh the corrected surface.





Student Notes:

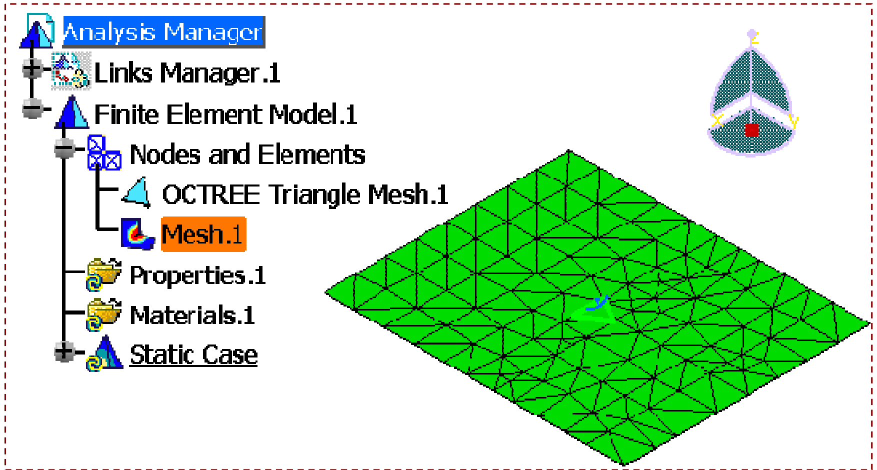
Exercise 1D

Recap Exercise

In this exercise, you will mesh the surface having gaps, check for the gaps, merge the gaps and mesh the corrected surface. Detailed instructions for new topics are provided for this exercise.

By the end of this exercise you will be able to:

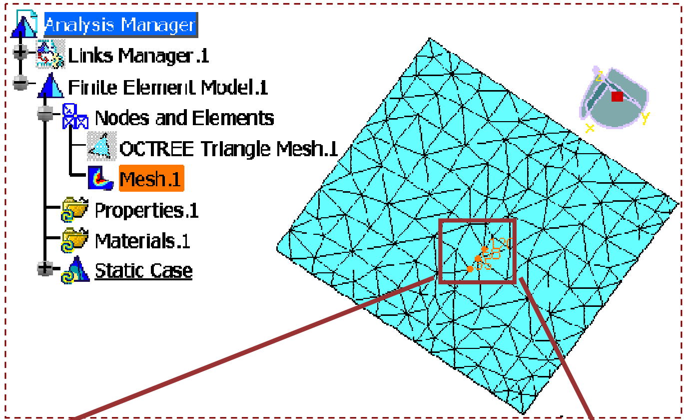
- Check the gaps in surface
- Merge the gaps in surface
- Mesh the corrected surface



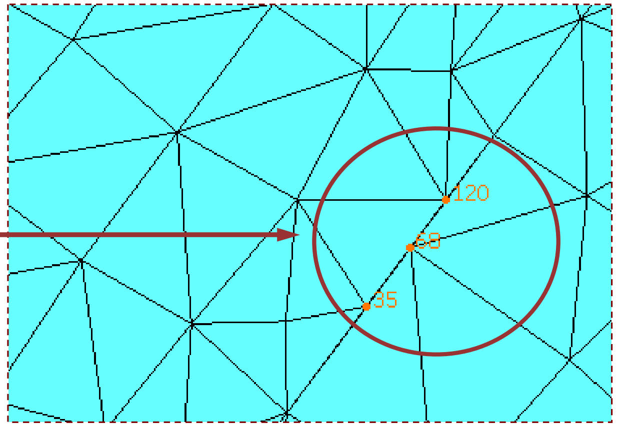
Student Notes:

Exercise 1D (1/4)

1. **Open a part.**
   - Open 1D_Gaps_Start.CATPart
2. **Create a Static Analysis Case.**
   - Access Generative Structural Analysis workbench.
   - Create a Static Analysis Case.
3. **Mesh the surface.**
   - Mesh the surface with OCTREE Triangle Mesher with mesh **Size** *20 mm* and **Absolute sag** *2 mm*.
   - Visualize the mesh.



Nodes are not interconnected in meshing if surfaces have gaps

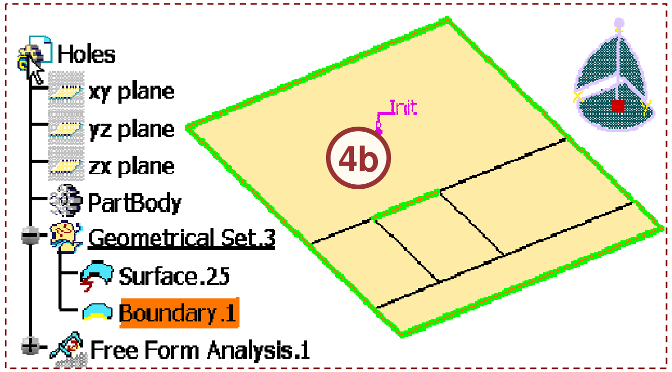


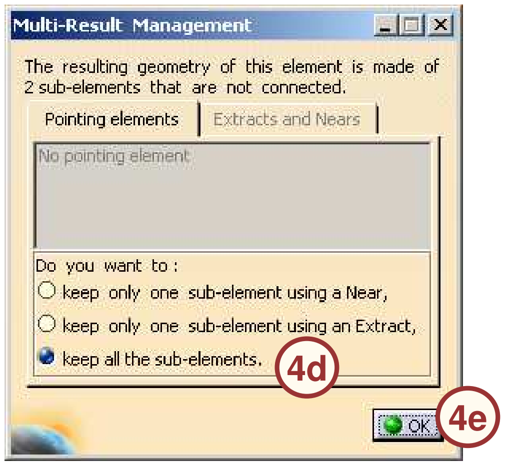
Student Notes:

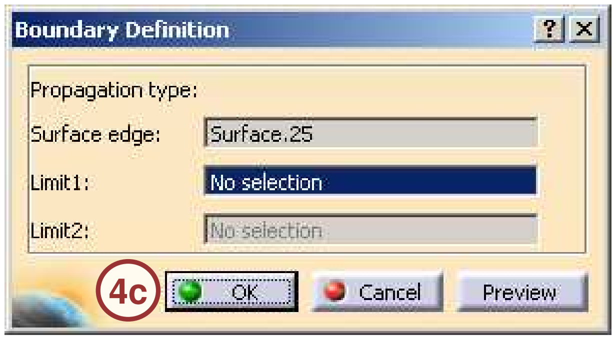
Exercise 1D (2/4)

4. Check if there are gaps.

- Switch to GSD workbench.
- Check the Boundary.
    a. In **Extracts** toolbar click **Boundary** icon.
    b. Select the *Surface.25* by directly clicking on it. You can also select it through specification tree. The surface is highlighted with gap.
    c. Click **OK.**
    d. In Multi-Result Management window select the option **keep all the sub-elements.**
    e. Click **OK**





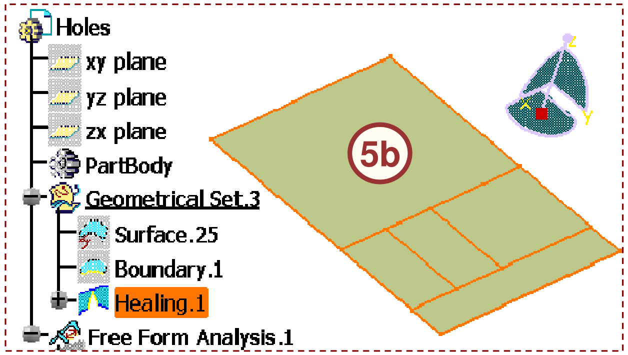


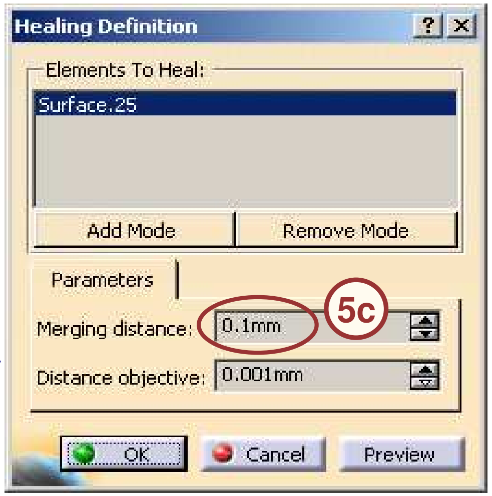
Student Notes:

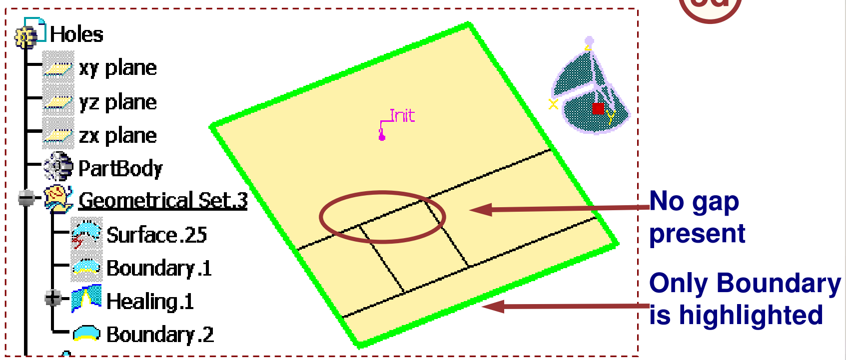
Exercise 1D (3/4)

5. Merging the gap.

- Use Healing to merge the gaps.
  - a. In **Join-Healing** toolbar, click **Healing** icon.
  - b. Select the surface to heal.
  - c. Enter *0.1* in **Merging Distance** field and click **OK**. **Merging distance** should be greater than gap
- Ensure the merging of gaps.
  - d. In **Extracts** toolbar, click **Boundary** icon again. Now it will show you only Outer boundary highlighted.





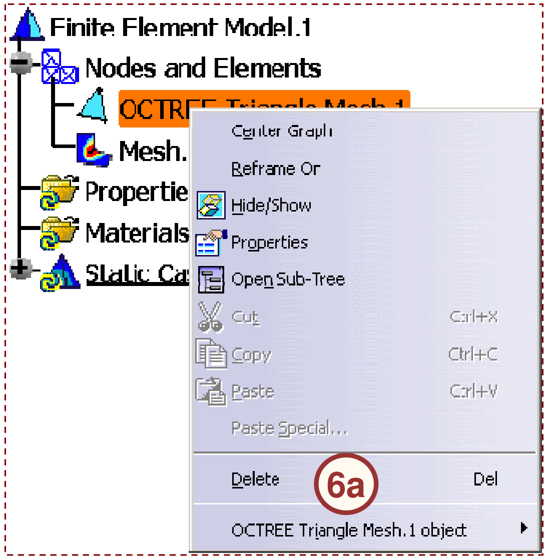


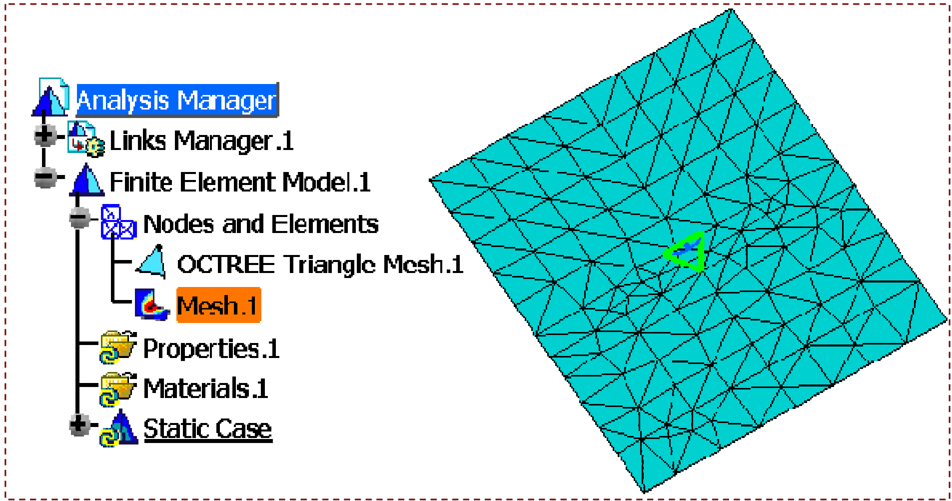
Student Notes:

Exercise 1D (4/4)

6. Mesh the corrected surface.

- Mesh the surface using OCTREE Triangle Mesher.
  a. Delete existing *OCTREE Triangle. Mesh.1* using **Delete** in contextual menu.
  b. Create new mesh **OCTREE Triangle Mesher.**
  c. Activate the *Mesh.1* Image.

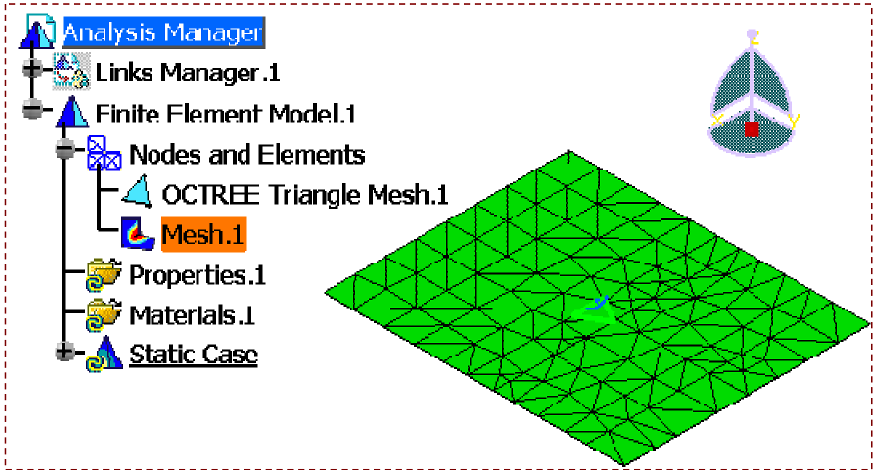




Student Notes:

Exercise 1D: Recap

- ✓ Check the gaps in surface.
- ✓ Merge the gaps in surface.
- ✓ Mesh the corrected surface.



Student Notes:

Exercise 1E

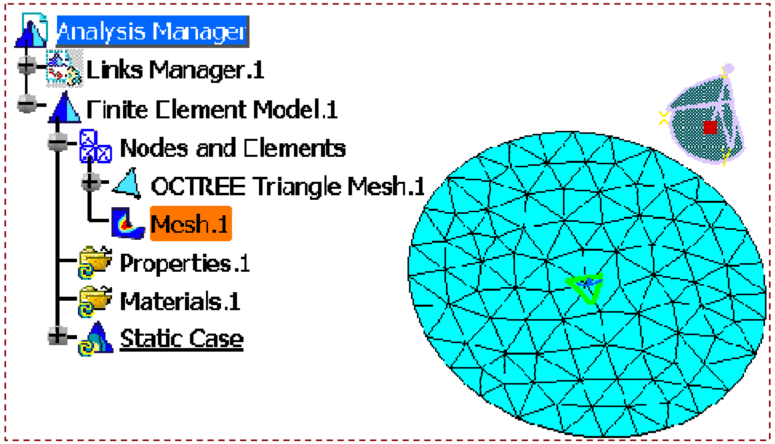
Recap Exercise

5 min

In this exercise, you will mesh the surface with value less than 0.2 mm and then with higher values. Detailed instructions for new topics are provided for this exercise.

By the end of this exercise you will be able to:

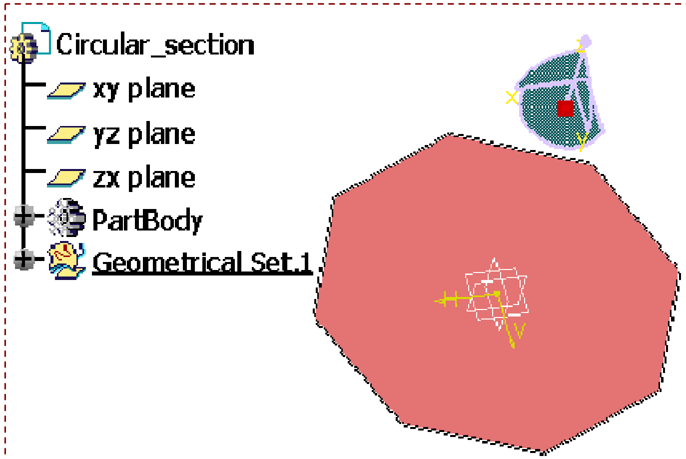
- **Mesh the surface with recommended mesh size**

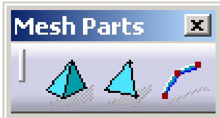


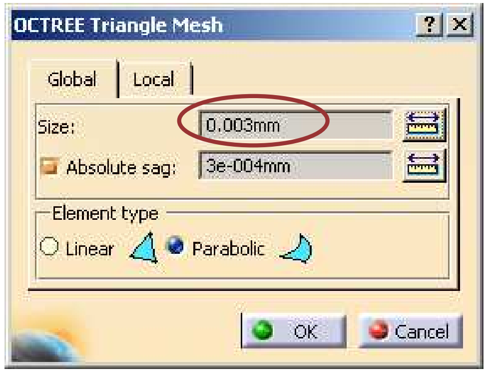
Student Notes:

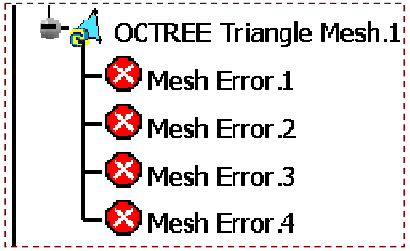
Exercise 1E (1/2)

1. **Open a part.**
   - Open 1E_Mesh_Size_Start.CATPart
2. **Create a Static Analysis Case.**
   - Access Generative Structural Analysis workbench.
   - Create a Static Analysis Case.
3. **Mesh the surface.**
   - Mesh the surface with OCTREE Triangle Mesher.
     a. Enter *0.003* in **Size** field.
     b. Check **Absolute sag** option and enter *0.0003*.
     c. Select **Element Type** as **Parabolic** and click **OK**.
4. **View Mesh.**
   - Generate Mesh Image. The following warning message will be displayed. When you select OK, following four mesh errors are seen in the specification tree.



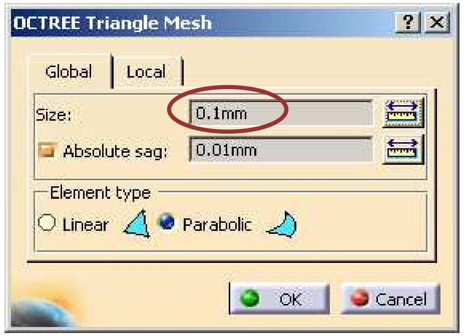






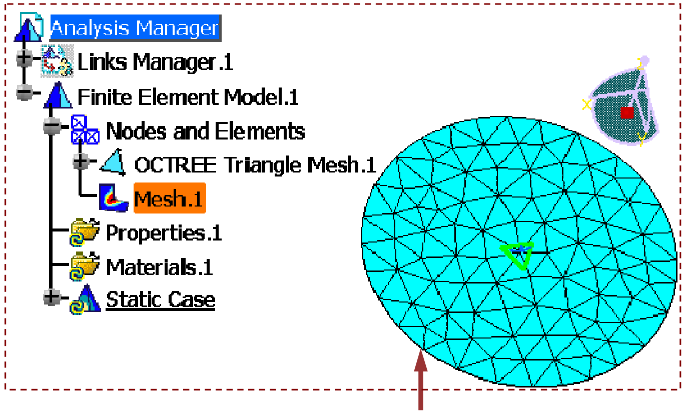
Student Notes:

Exercise 1E (2/2)



5. Change the mesh size.

- Change the mesh size to 0.1 mm
  a. Double-click on *OCTREE Triangle Mesh.1* in specification tree. Enter *0.1* in **Size** field. Enter **Absolute sag** as *0.01* and click **OK**.
  b. Generate the Mesh Image. Once again you will get the same warning message to update the mesh. When you select **OK**, *Mesh.1* will appear in the specification tree.



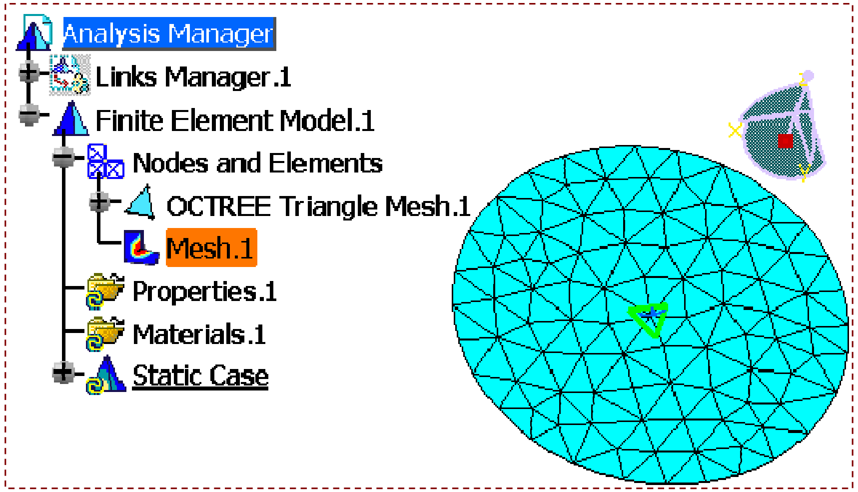
Mesh Image with 0.1 mm mesh size

The minimum mesh size value 0.2 mm is a purely theoretical limit above which you are sure to be able to create a mesh. If you have less than this value, may or may not be able to create a mesh. This will be indeterminate because will depend upon the geometry.

Student Notes:

Exercise 1E: Recap

✓ Mesh the surface with value above recommended mesh size.



Student Notes:

Exercise 1F

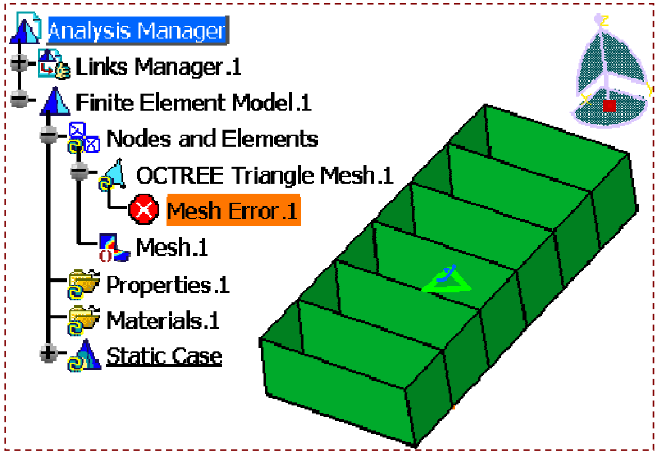
Recap exercise

5 min

In this exercise, you will mesh the non-manifold surface. High-level instructions are provided for this exercise.

By the end of this exercise you will be able to:

- **Verify that non-manifold surfaces can not be meshed with the GPS workbench**



Student Notes:

Exercise 1F

1. Open a part.

- Open 1F_NonManifold_Start.CATPart.

2. Create a Static Analysis Case.

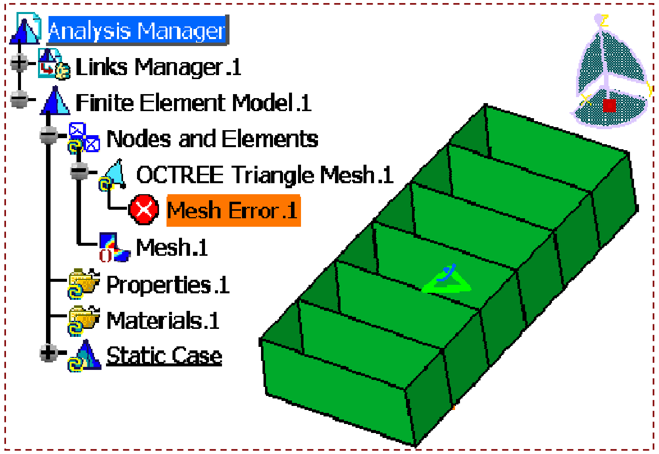
- Access Generative Structural Analysis workbench.
- Create a Static Analysis Case.

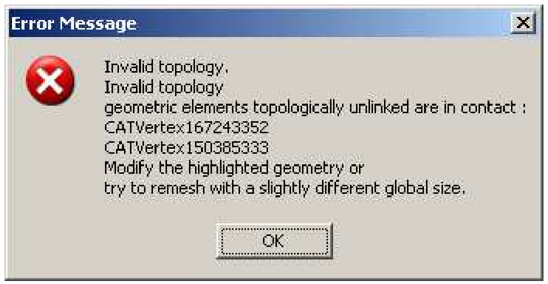
3. Mesh the surface.

- Mesh the surface with OCTREE Triangle Mesher with default values.

4. View Mesh.

- Generate Mesh Image. The following error message will be displayed.





In order to mesh such a part, Advanced Meshing Tools workbench has to be used.

Student Notes:

Exercise 1F: Recap

✓ **Understand that non-manifold surfaces can not be meshed.**