

# CATIA V5 Analysis



See what you mean



# CATIA V5 Analysis

**Student Handbook**  
Version 5 Release 19



**16 Hours**

## Copyright DASSAULT SYSTEMES

### ALL RIGHTS RESERVED

No part of this publication may be reproduced, translated, stored in retrieval system or transmitted, in any form or by any means, including electronic, mechanical, photocopying, recording or otherwise, without the express prior written permission of DASSAULT SYSTEMES. This courseware may only be used with explicit DASSAULT SYSTEMES agreement.



## Table of Contents

• Introduction to Finite Element Analysis	7
• Pre-processing	37
• Computation	63
• Post-processing	81
• Mesh Refinement	113
• Assembly Structural Analysis	125
• Master Project	138
• Shortcuts	164
• Glossary	165



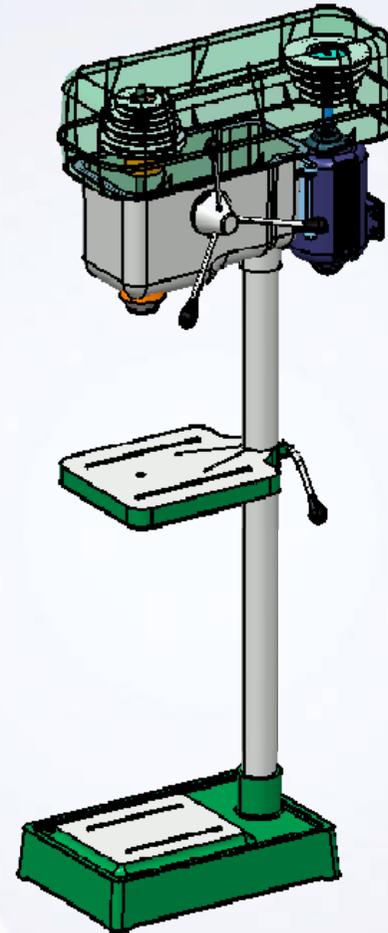
# Introduction to Finite Element Analysis

1

**Learning Objectives:**

*Upon completion of this lesson you will be able to:*

- ✓ Understand the general steps to be followed to perform Finite Element Analysis for structures.
- ✓ Understand the general FEA process in GPS workbench.
- ✓ Understand the preparation of part and surface models for analysis.



2 hours




---



---



---



---



---



---

## Case Study

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 for case study come from the drill press assembly, which is also your master project.

## Design Intent

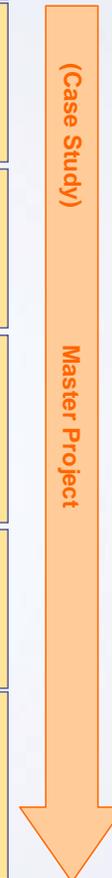
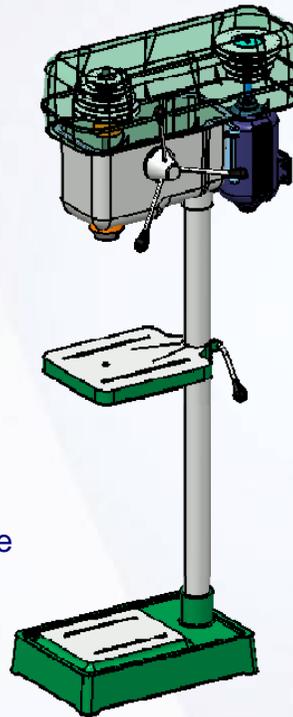
Each case study contains a set of model requirements, known as the design intent. The first case study does not contain a design intent because you are not going to design anything. However, by the end of this lesson you should be able to:

- ✓ 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

## Stages in the Process

Each lesson consists of steps. For lesson 1, you will go through the following steps to start with FEA in Generative Structural Analysis Workbench:

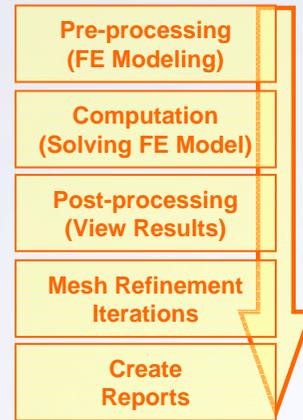
1. What is Finite Element Analysis Process.
2. Introduction to Generative Structural Analysis (GPS) Workbench.
3. Preparing Parts and Surfaces for Analysis.



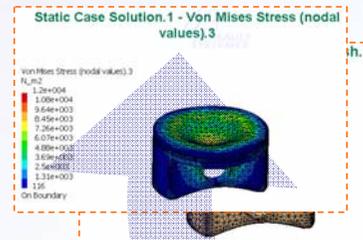
# Finite Element Analysis Process

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

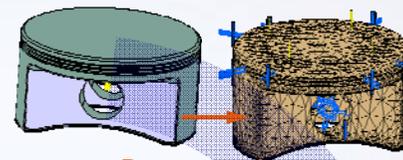
1. Pre-processing: Conversion of actual problem into Finite element problem
2. Computation: Solution of the of the FE problem provided by pre-processing to find out unknown displacement values
3. Post-processing: Calculation of strains and stresses using displacement values. Study of displacements strains 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



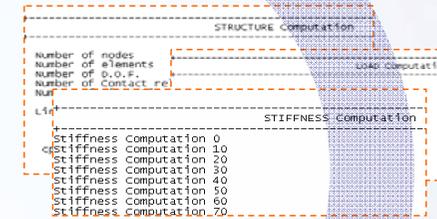
Finite Element Analysis Process



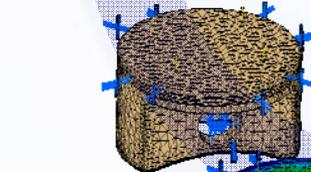
Report Generation



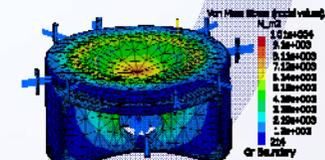
Pre-processing



Computation



Mesh-Refinement



Post-processing

## 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

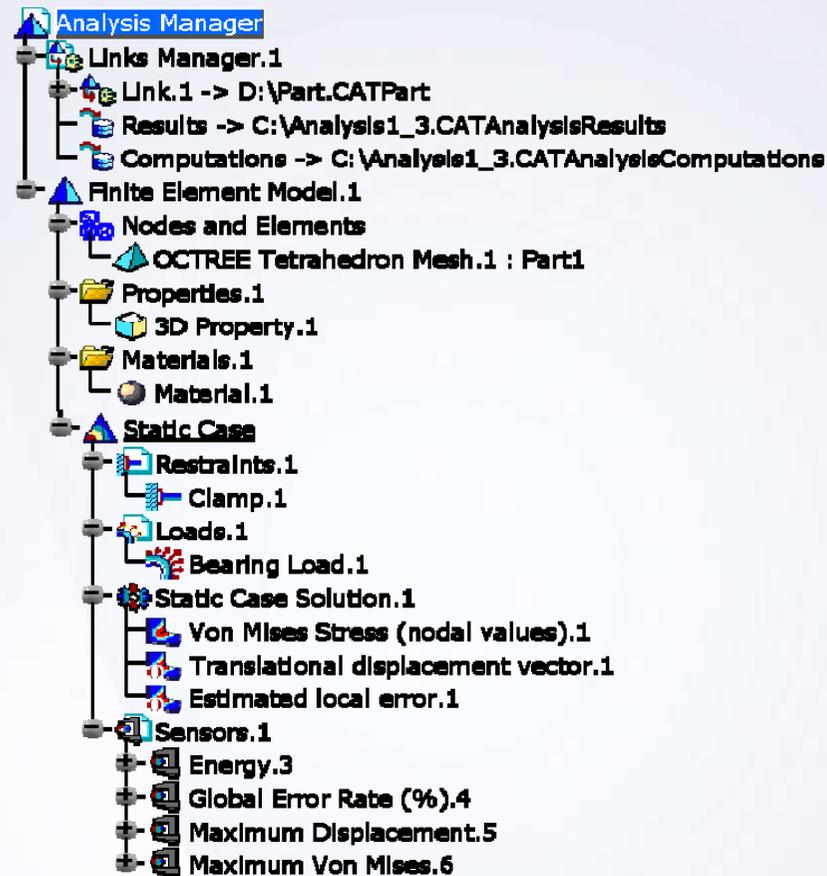
According to variation in load with respect to time, the FEA for structures can be classified as:

- Static Analysis
- Transient Analysis
- Harmonic Analysis

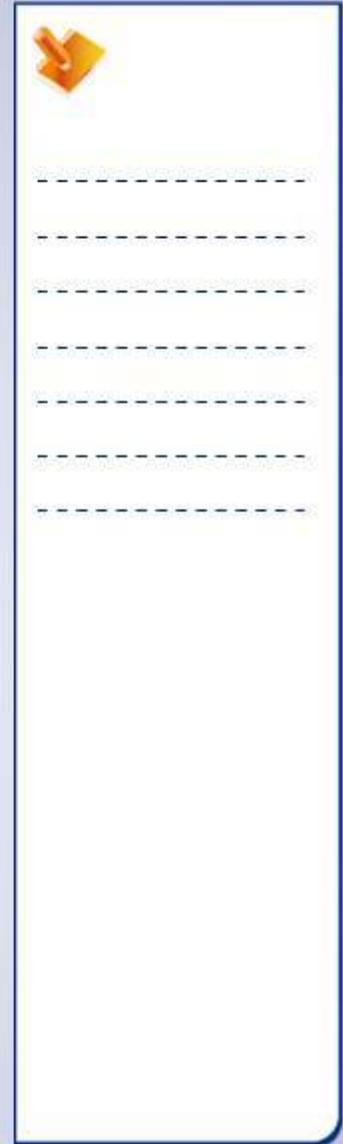
According to the way the structure reacts to the load, the FEA for structures can be classified as:

- 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

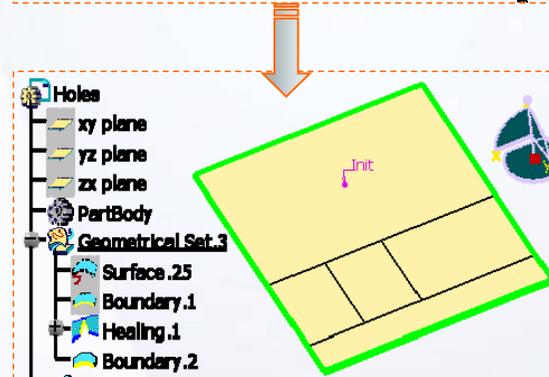
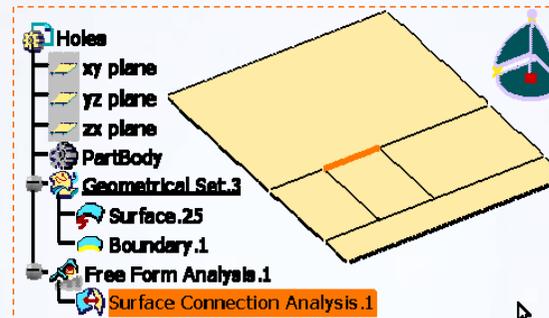
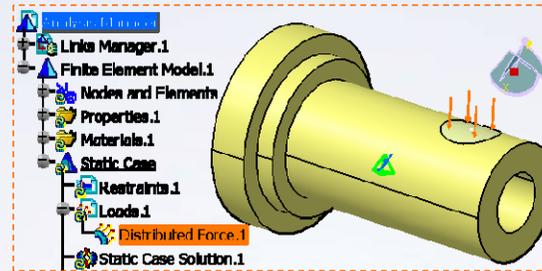


## Preparing Parts and Surfaces for Analysis

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.






---



---



---



---



---



---

## 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.






---



---



---



---



---



---

## 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.






---



---



---



---



---



---

# Exercise 1C

## Recap Exercise

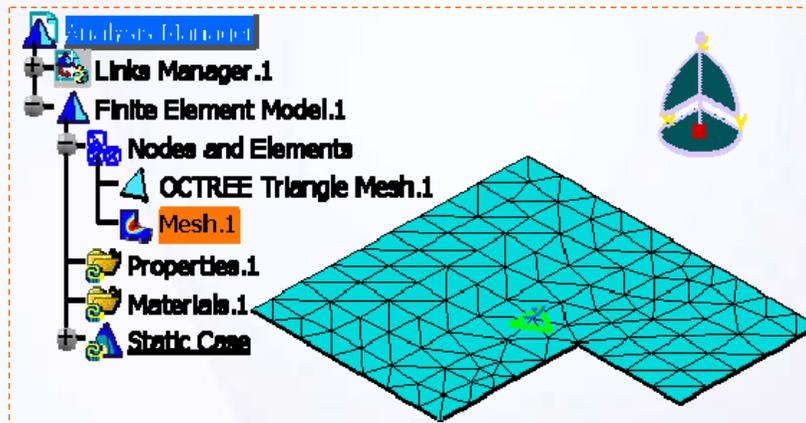


60 min

*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






---



---



---



---



---



---

## Exercise 1C (1/8)

### 1. Open a part.

- Open 1C\_Overlapping\_Start.CATPart

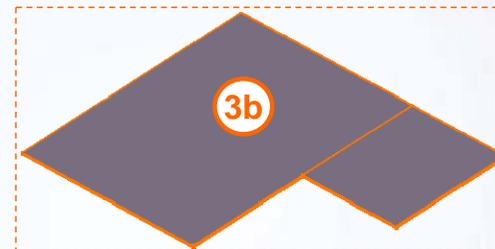
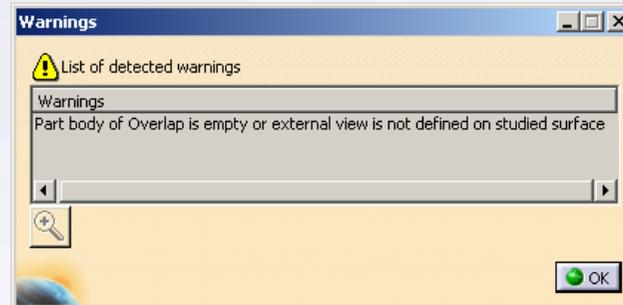
### 2. Create a Static Analysis Case.

- 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.
  - In **Mesh Parts** toolbar, click **OCTREE Triangle Mesher** icon.
  - Click on the surface.
  - Keep default values as shown and click **OK**.

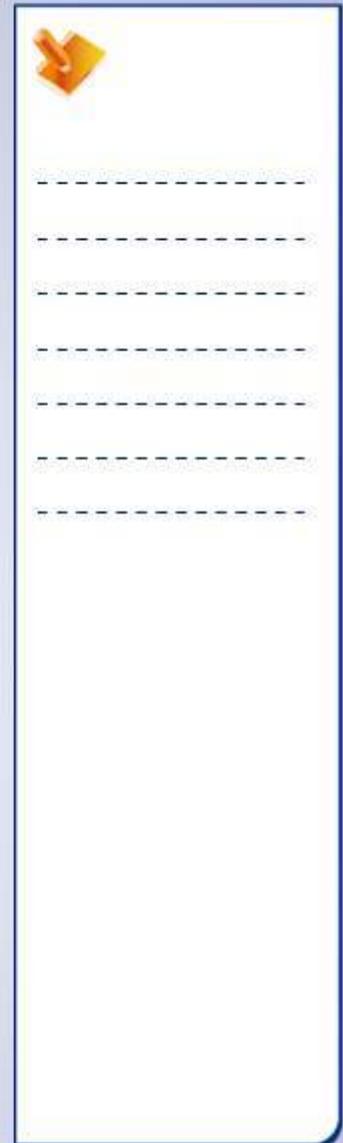
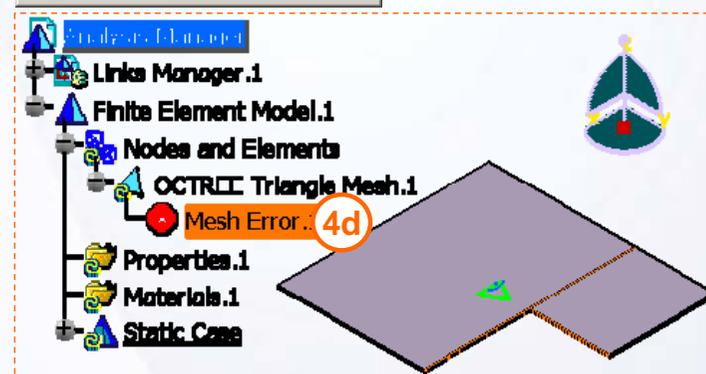
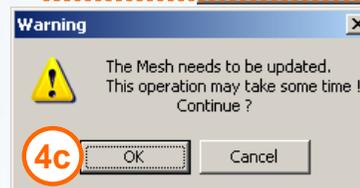
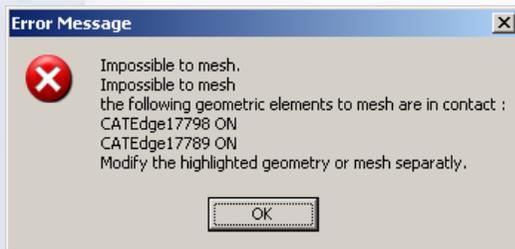
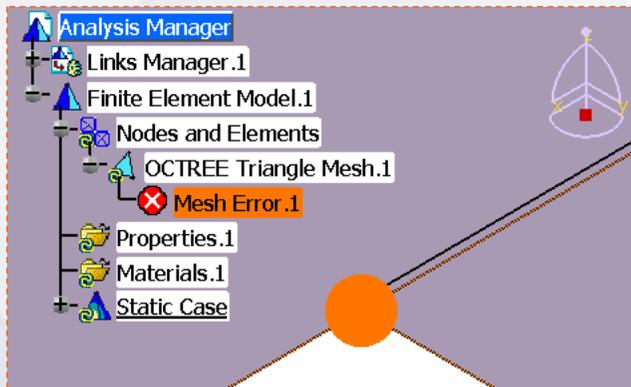
Please ensure that you are set to mm.



## Exercise 1C (2/8)

### 4. View the mesh.

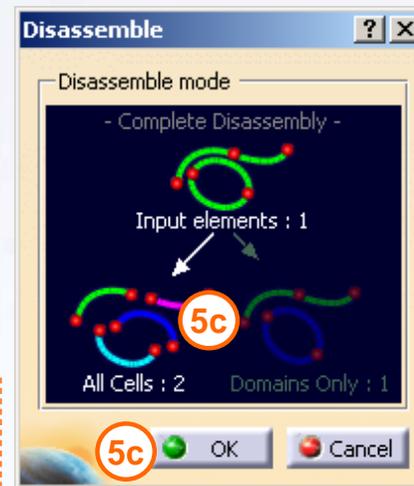
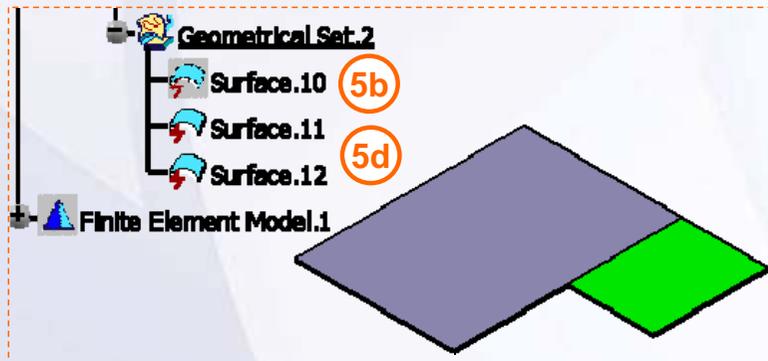
- Create Mesh visualization to create Mesh image.
  - a. Right-click on *Nodes and Elements*.
  - b. Select Mesh visualization.
  - c. Click **OK** for the warning message to update the mesh.
  - d. Double-click on the *Mesh Error.1*.



## Exercise 1C (3/8)

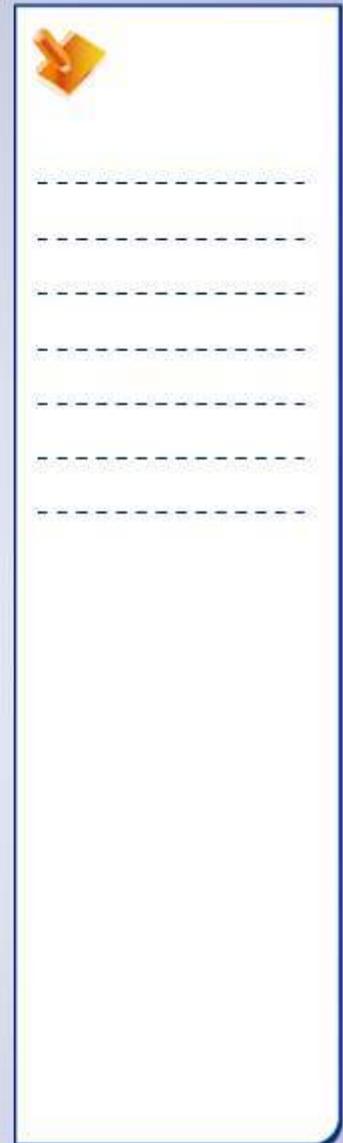
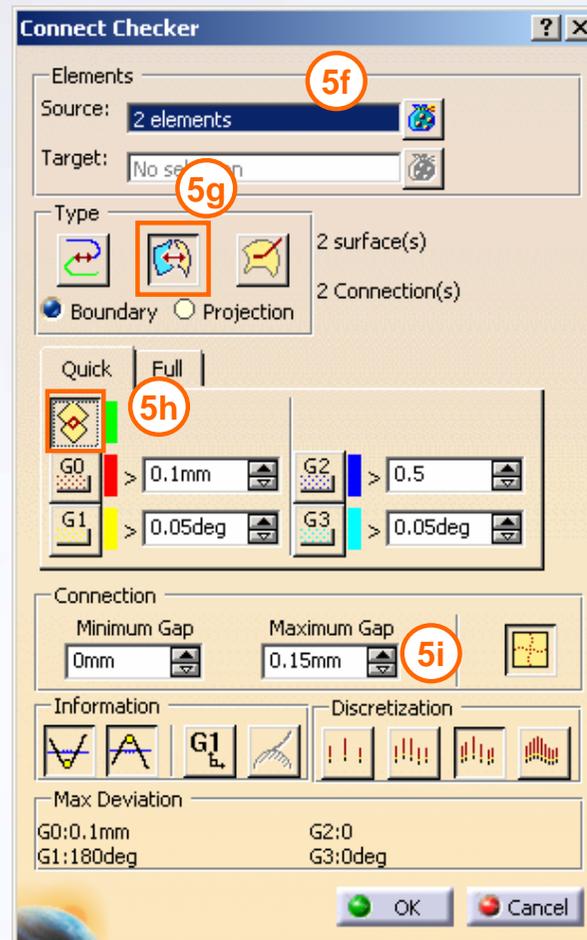
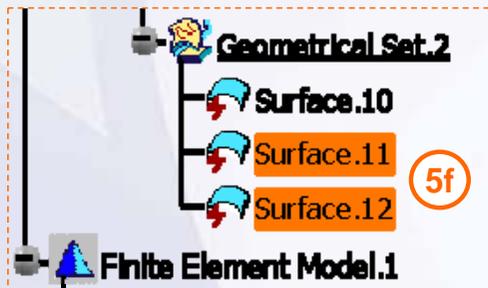
### 5. Correct overlapping surfaces.

- 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.



## 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.

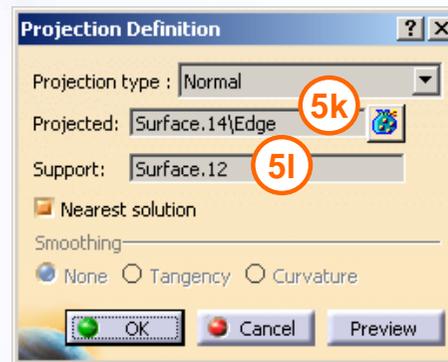
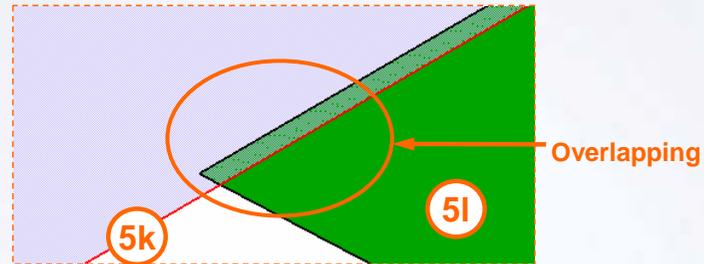


## Exercise 1C (5/8)

- 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**.



5j






---



---



---



---



---



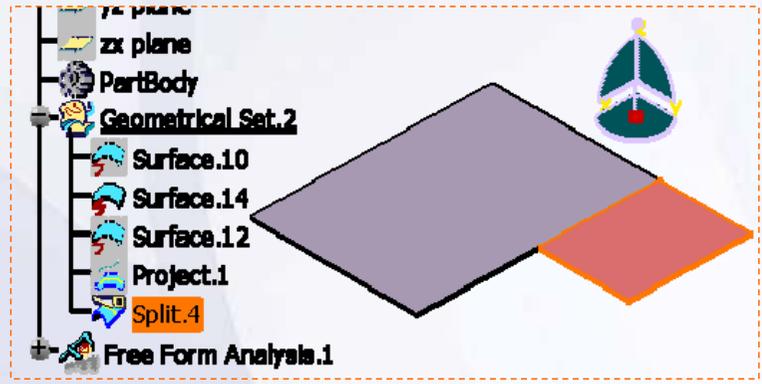
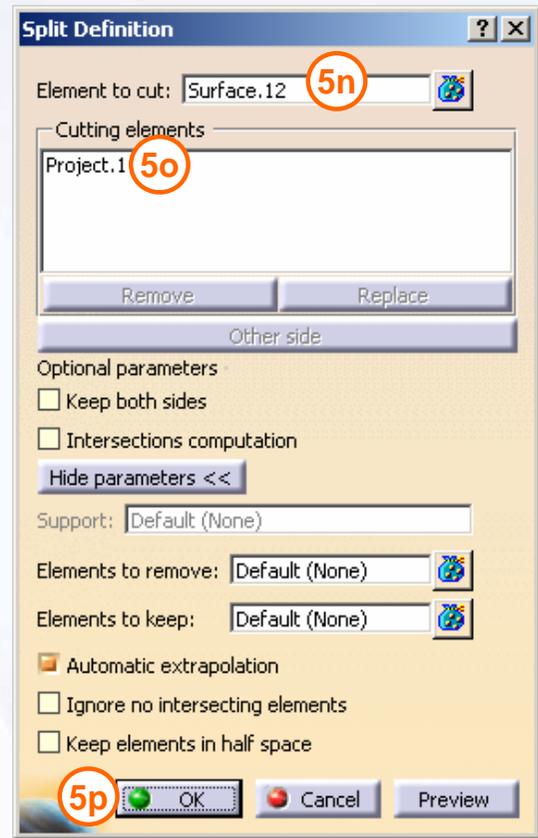
---



---

## 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.




---



---



---



---



---



---



---

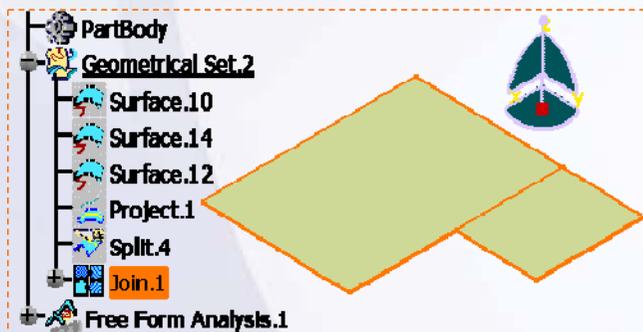
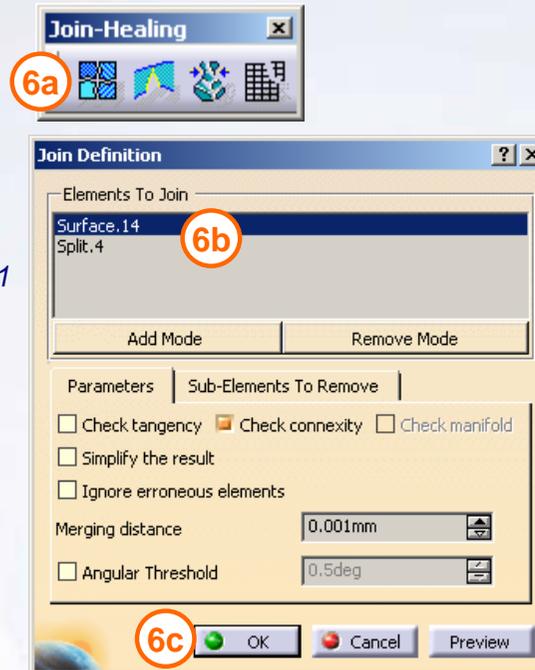


---

## 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 *Surface.14*.
  - c. Click **OK**. This will create new surface *Join.1* in specification tree.






---



---



---



---



---



---

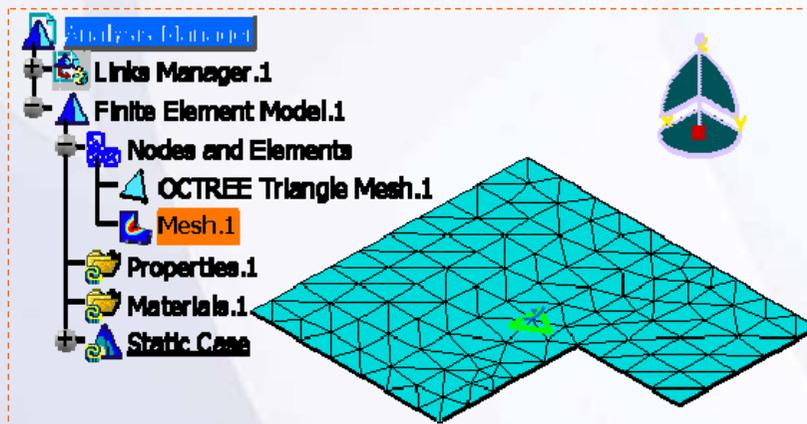


---

## 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.



---



---



---



---



---



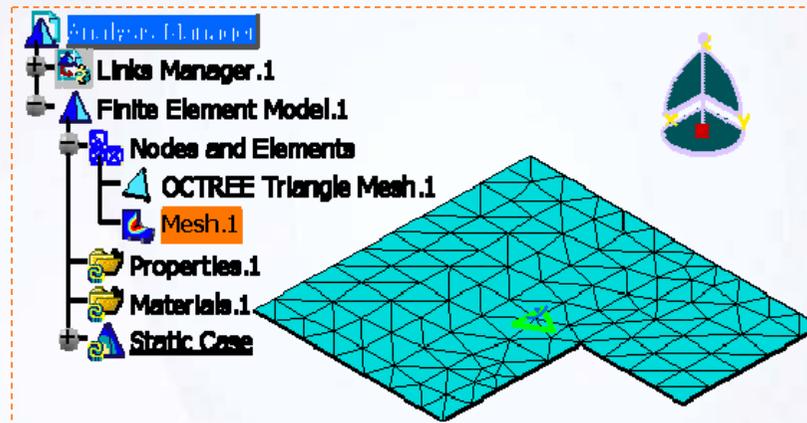
---



---

## Exercise 1C: Recap

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



A vertical rectangular box on the right side of the page. It contains a yellow pushpin icon at the top left corner. Below the icon are several horizontal dashed lines, intended for a student to write their notes or answers during the exercise.

# Exercise 1D

## Recap Exercise

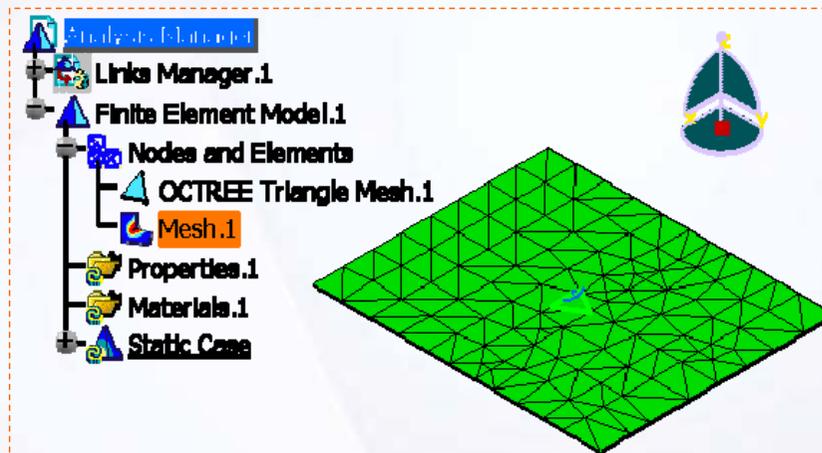


15 min

*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






---



---



---



---



---



---

## Exercise 1D (1/4)

### 1. Open a part.

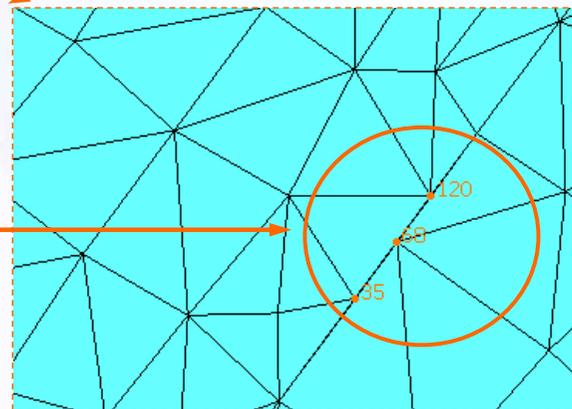
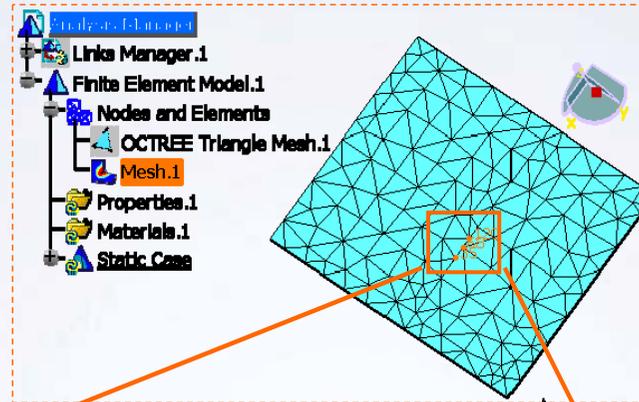
- Open 1D\_Gaps\_Start.CATPart

### 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.

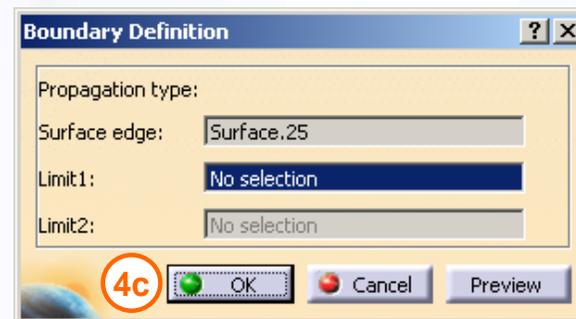
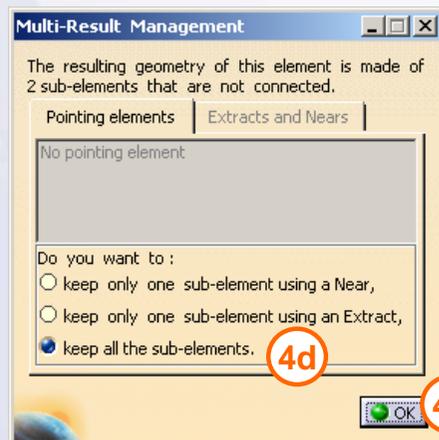
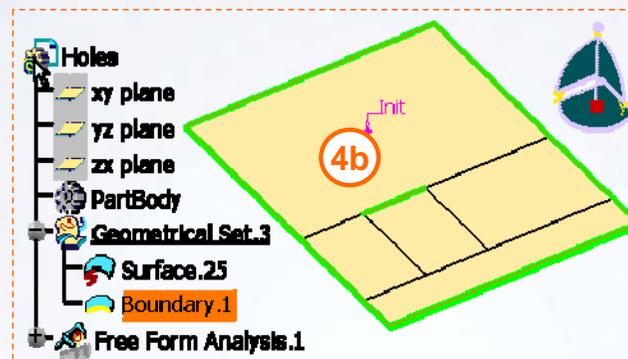


Nodes are not interconnected in meshing if surfaces have gaps

## 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**



---



---



---



---



---



---

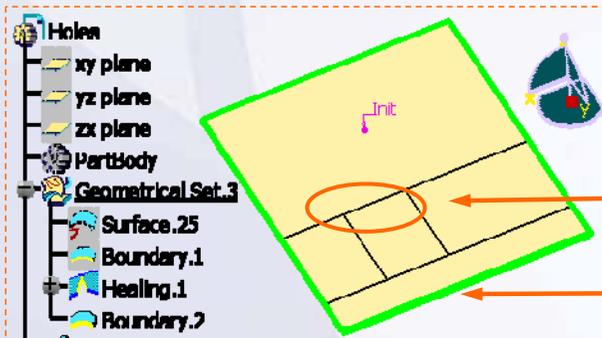


---

## Exercise 1D (3/4)

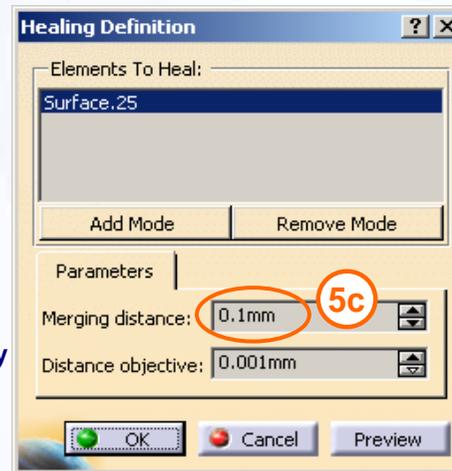
### 5. Merging the gap.

- 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.



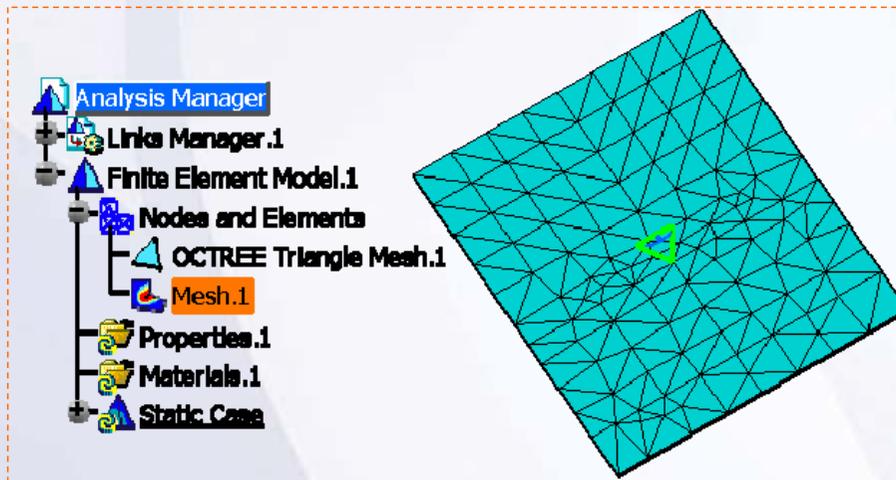
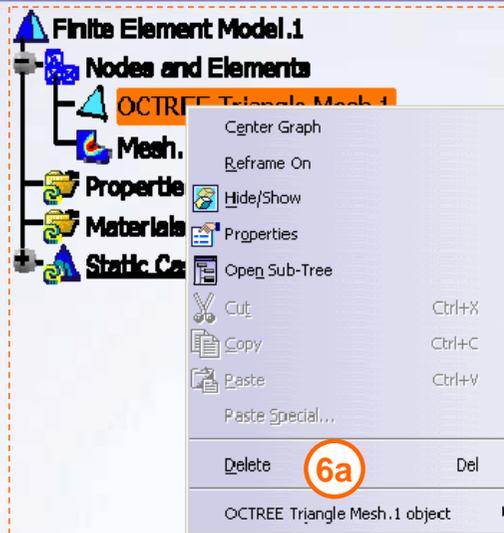
No gap present

Only Boundary is highlighted



## 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.






---



---



---



---



---



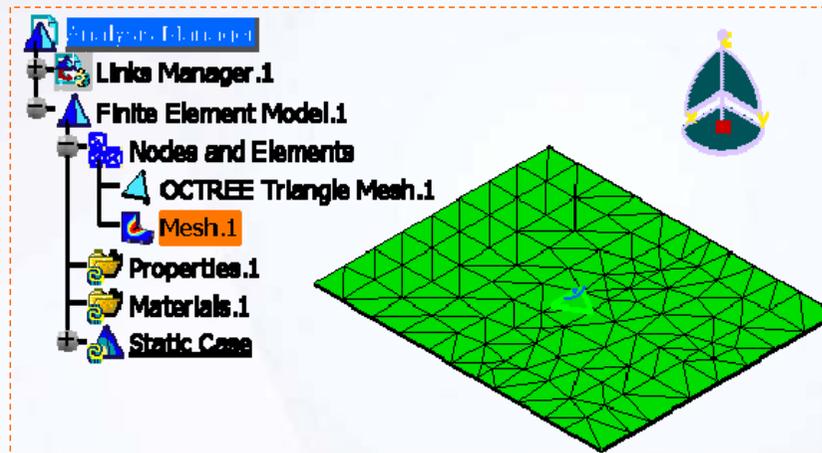
---



---

## Exercise 1D: Recap

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



A vertical rectangular box containing a yellow arrow icon at the top and several horizontal dashed lines for writing.

# Exercise 1E

## Recap Exercise

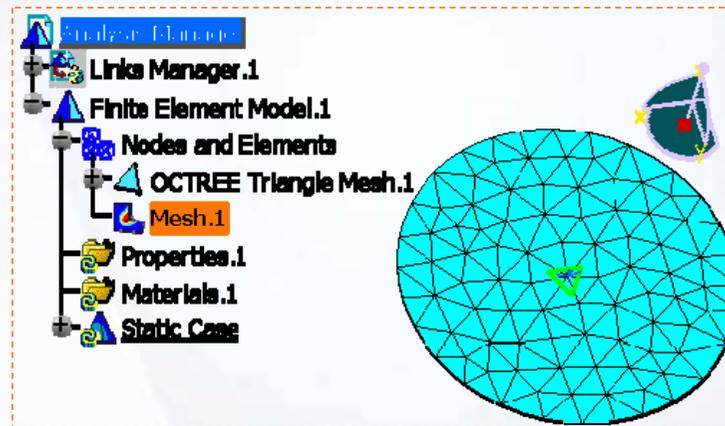


5 min

*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






---



---



---



---



---



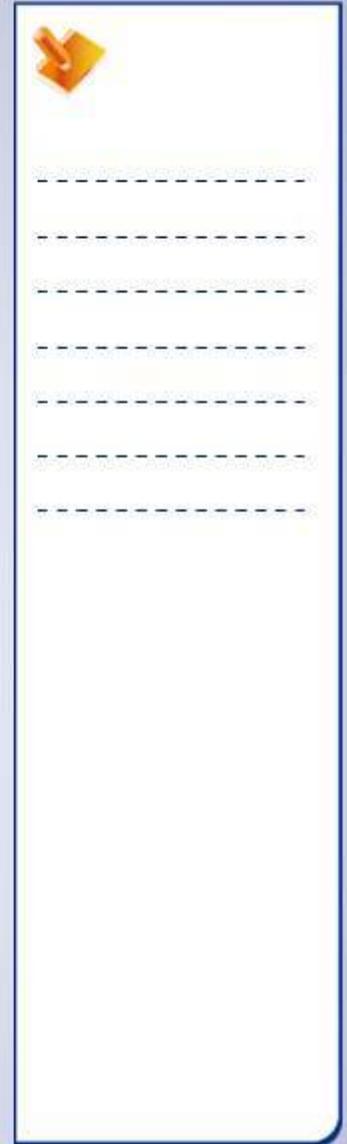
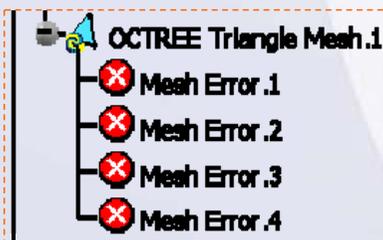
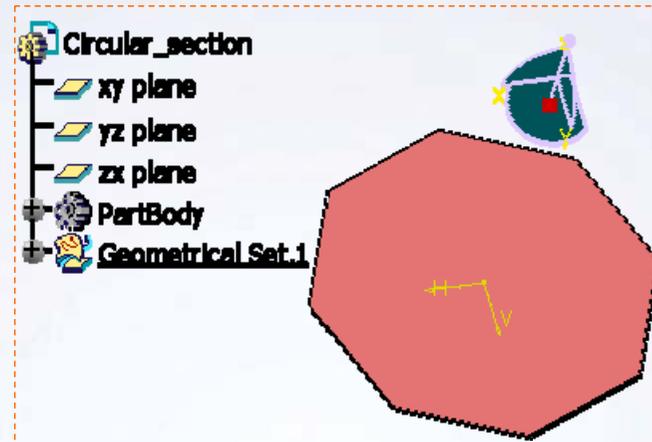
---



---

## Exercise 1E (1/2)

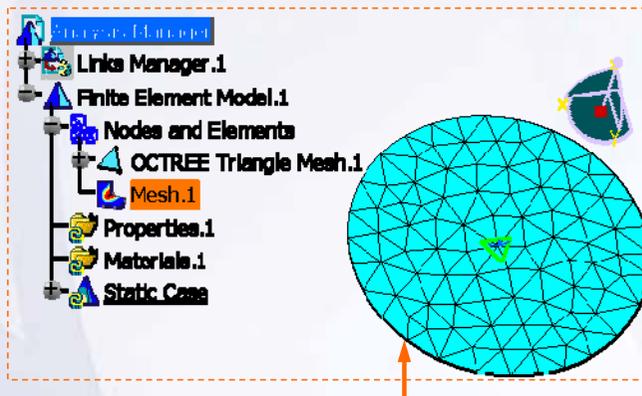
1. **Open a part.**
  - Open 1E\_Mesh\_Size\_Start.CATPart
2. **Create a Static Analysis Case.**
  - Access Generative Structural Analysis workbench.
  - Create a Static Analysis Case.
3. **Mesh the surface.**
  - Mesh the surface with OCTREE Triangle Mesher.
    - a. Enter 0.003 in **Size** field.
    - 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.



## Exercise 1E (2/2)

### 5. Change the mesh size.

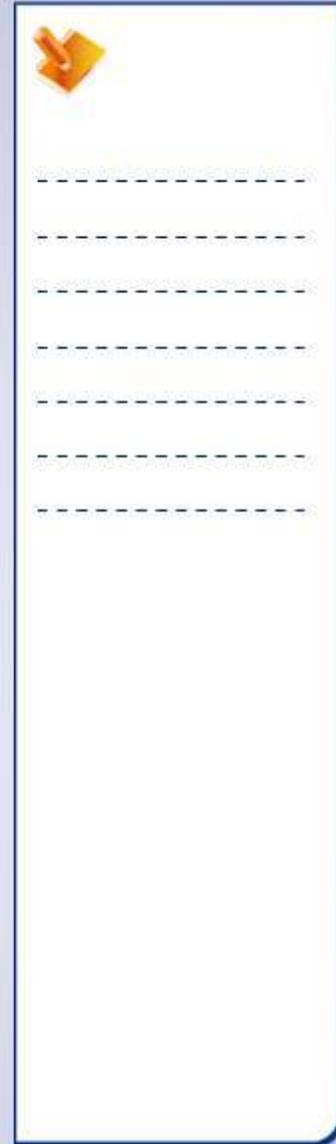
- Change the mesh size to 0.1 mm
  - a. Double-click on *OCTREE Triangle Mesh.1* in specification tree. Enter *0.1* in **Size** field. Enter **Absolute sag** as *0.01* and click **OK**.
  - b. Generate the Mesh Image. Once again you will get the same warning message to update the mesh. When you select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



Mesh Image with 0.1 mm mesh size

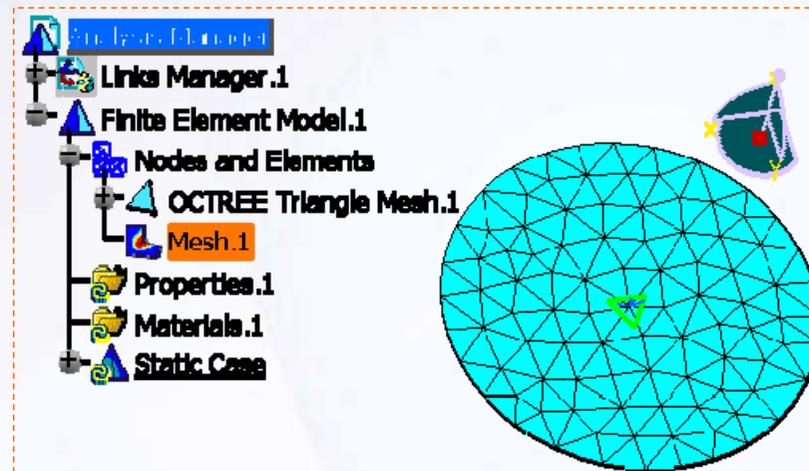


The minimum mesh size value 0.2 mm is a purely theoretical limit above which you are sure to be able to create a mesh. If you have less than this value, may or may not be able to create a mesh. This will be indeterminate because will depend upon the geometry.



## Exercise 1E: Recap

- ✓ Mesh the surface with value above recommended mesh size.



Handwritten notes area with a yellow arrow icon at the top and several horizontal dashed lines for writing.

# Exercise 1F

Recap exercise

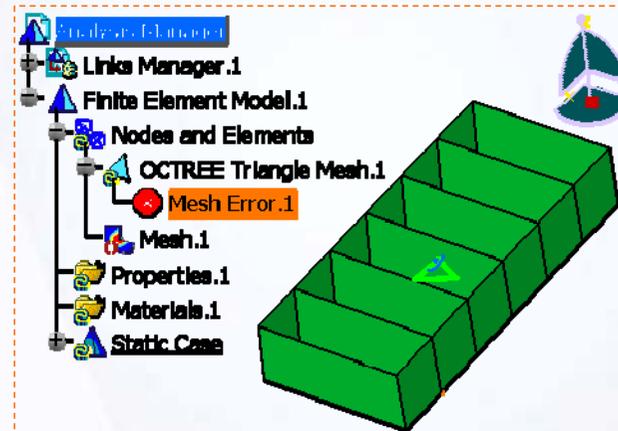


5 min

*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






---



---



---



---



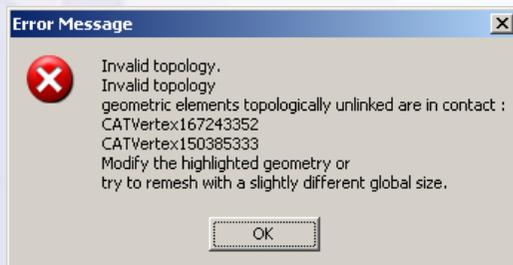
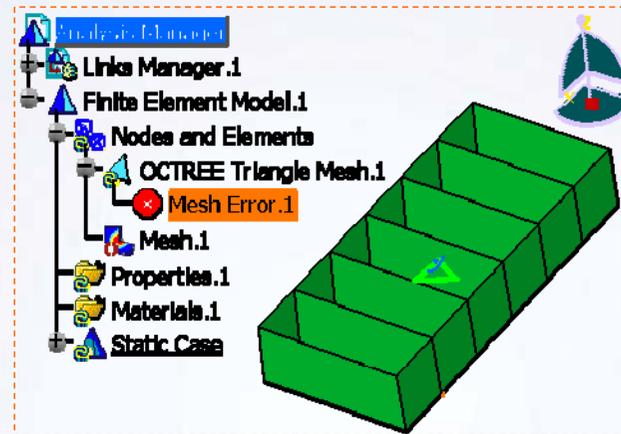
---



---

## 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.




---



---



---



---



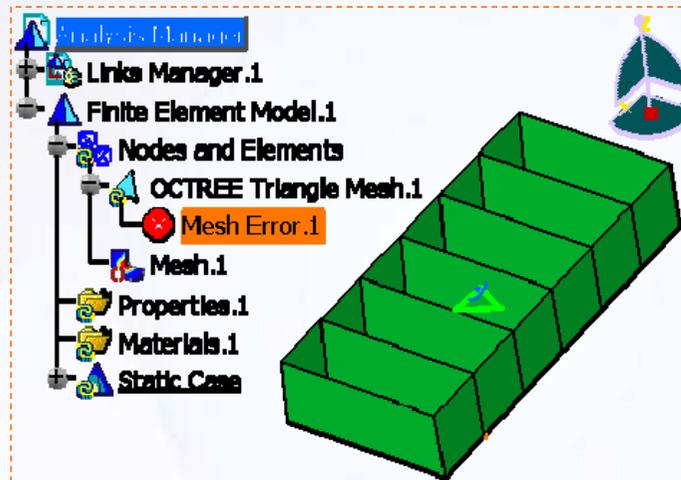
---



---

## Exercise 1F: Recap

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



Handwritten notes area with a yellow arrow icon at the top and several horizontal dashed lines for writing.

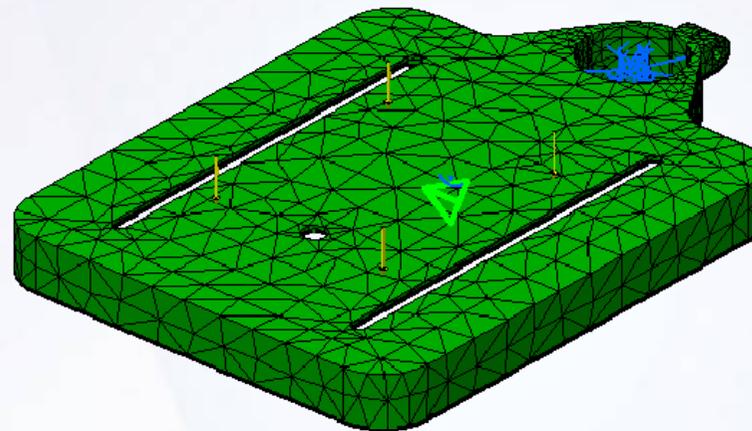
# Pre-processing

2

**Learning Objectives:**

*Upon completion of this lesson you will be able to:*

- ✓ Understand what is Pre-processing
- ✓ Apply material with specified properties
- ✓ Generate mesh
- ✓ Assign physical properties
- ✓ Apply restraints and loads to the model
- ✓ Check the Finite Element model



2 hours




---



---



---



---



---



---



---

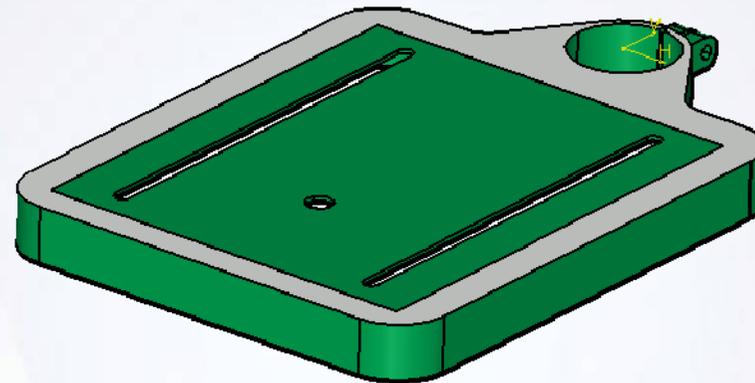
## Case Study

The case study for this lesson is the Drill Press Table. The focus of this case study is to create a Finite Element model for Drill Press Table.

## Design Intent

You will perform the static analysis of Drill Press Table. The actual physical problem will be described which will provide input data for FE modeling of the drill press Table. You have to use this information to build an appropriate FE model.

- ✓ The table of drill press is composed of steel having specified properties. You will create the user material with these properties.
- ✓ The mesh type must be 3D OCTREE Tetrahedron.
- ✓ The table is fixed to the stand with a Clamp restraint.
- ✓ A downward force of 10 N will be applied and transmitted to the top surface of the table through the drill-bit and work-piece.
- ✓ The validity of the created FE model must be checked.



Drill Press Table Component




---



---



---



---



---



---

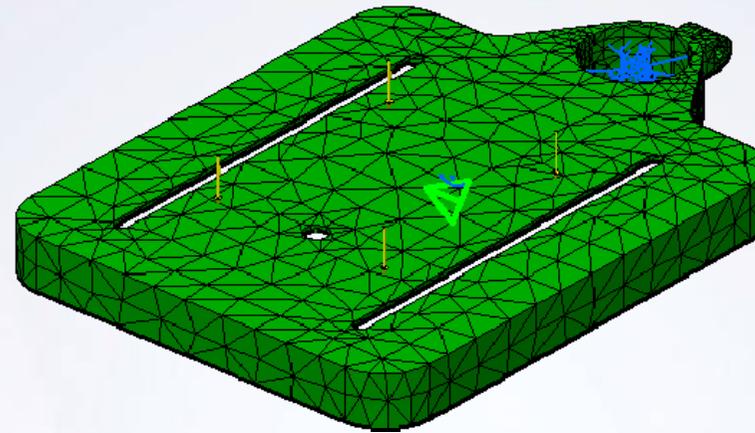


---

## Stages in the Process

You will see how to perform each Pre-processing step in the GPS workbench:

1. What is Pre-processing?
2. Applying material
3. Generating mesh
4. Assigning physical property
5. Applying Restraints and Loads
6. Checking the Finite Element Model



Drill press Table FE Model



---

---

---

---

---

---

---

---

## What is Pre-processing

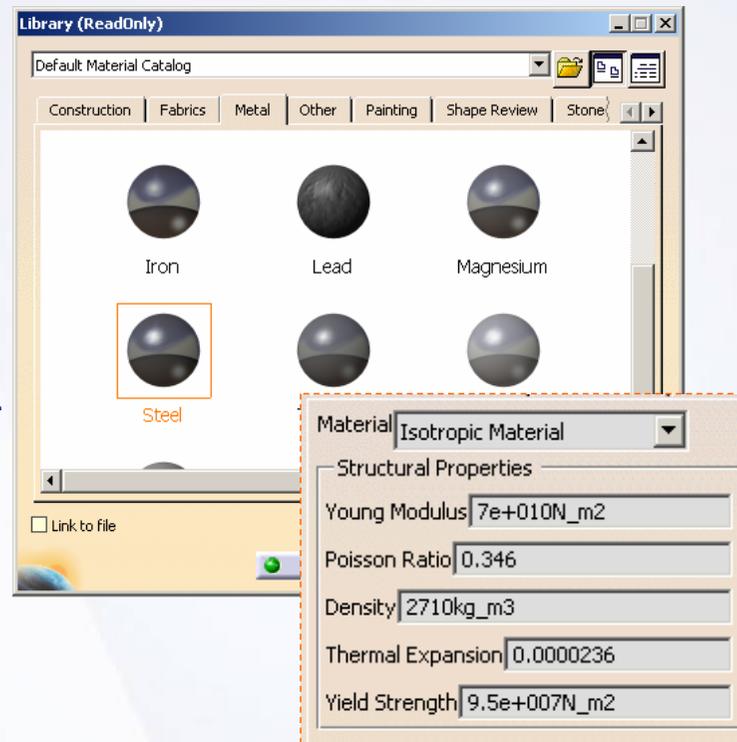
Pre-processing involves all the steps which are required to convert a given physical problem into an equivalent Finite Element problem. This involves following steps:

1. Applying material structural property to a part
2. Meshing the part using elements and nodes
3. Applying physical property to the mesh
4. Applying restraints to the mesh
5. Applying loads to the mesh
6. Model checking to validate that all the pre-processing steps are done.

## Applying Material

Structural properties are required to calculate the deformation, strains and stresses. You can apply material by selecting it from the default material catalog.

You may need to create the material with specified properties, which is not available in the standard material catalog. In this case, you can use User Material option to create the material with your own specification.






---



---



---



---



---



---

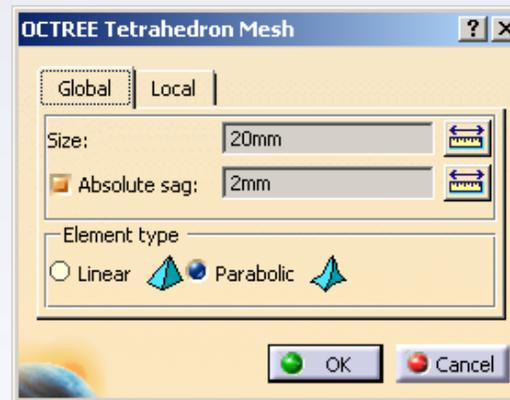


---

## Generating Mesh

Meshing involves approximating actual physical structure into Finite Element Model using several simple geometric shapes called elements. Mesh is representation of the mathematical idealization of the structure.

On the basis of dimensionality, elements can be classified as 1D, 2D, and 3D elements. According to the order of the displacement equation, elements can be classified as Linear and Higher order elements. You can use Beam Mesher, OCTREE Triangle Mesher, and OCTREE Tetrahedron Mesher to mesh wireframes, surfaces, and solids respectively.



## Assigning Physical Property

In CATIA, physical property means cross-section, thickness, and material properties. It uses geometry of the mesh or mesh part as support to apply Physical Property. You can also apply physical property directly to orphan meshes. The physical property to be attached depends on the mesh used to create FE model as shown in the table.

Mesh Type	Physical Property Name	Physical Property Assigned to Mesh
Beam	1D Property	1. Cross-section 2. Material
OCTREE Triangle	2D Property	1. Thickness 2. Material
OCTREE Tetrahedron	3D Property	1. Material




---



---



---



---



---



---

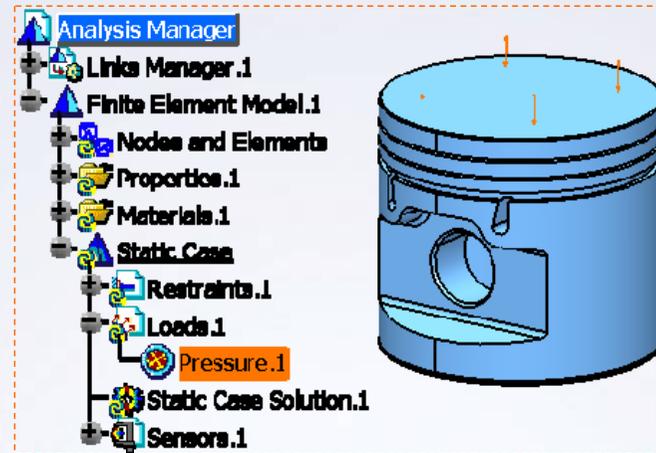


---

## Applying Restraints and Loads

The restraints are used to specify the support or boundary conditions for FE model. Restraints restrict the displacement of supports of the structure in required direction, by providing zero displacement values for specific Degrees of Freedoms at nodes present at support in FE model.

Loads are inputs to the system. They can be form of forces, moments, pressures, temperatures, accelerations, etc. The loads are directly applied onto the geometry (surfaces, lines, points). The type of load that can be applied will depend upon the geometry selected.



## Checking the Finite Element Model

Model Check allows you to verify that all pre-processing steps are completed. If any information is missing, it shows status KO against that row and provides related error message.

Model Checker contains following 3 tabs.

- Bodies
- Connections
- Others

**Status:**  
 OK  Green  
 KO  Red

Model Checker

One or several irrelevancies found

Part	Feature	Mesh Part	Property	Material	Status
Part.1.1	PartBody	OCTREE Tetrahed...	3D Property.1	Steel	OK
Part.2.1	PartBody	OCTREE Tetrahed...	3D Property.2	Steel	OK
Part.3.1	PartBody	OCTREE Tetrahed...	3D Property.3	Steel	OK
Part.4.1	PartBody	OCTREE Tetrahed...	3D Property.4	Steel	OK

## Main Tools

### Material

- 1 **Apply Material:** Lets you apply the material directly from the default Material catalog.



1

### Mesh Parts

- 2 **Beam Mesher:** Lets you create 1-D element mesh.
- 3 **OCTREE Triangle Mesher:** Lets you create 2D triangular element mesh.
- 4 **OCTREE Tetrahedron Mesher:** Lets you create 3D tetrahedron element mesh.



4

3

2

### Model Manager

- 5 **1D Property:** offers you a set of standard cross-sections and material to be associated to the beam mesh.
- 6 **2D Property:** Lets you to associate thickness and material to the OCTREE Triangle mesh.
- 7 **3D Property:** Lets you to assign material to be associated to the OCTREE Tetrahedron mesh.
- 8 **User Material:** Lets you create your own user-defined material with required properties.



7

6

5

8




---



---



---



---



---



---

# Exercise 2E

## Recap Exercise

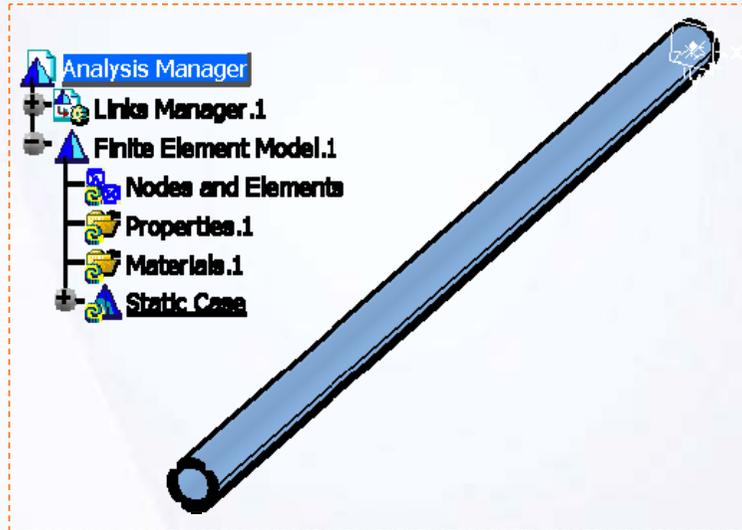


15 min

*In this exercise, you will apply boundary conditions, load and check the FE model for 2D mesh. High-level instructions are provided for this exercise.*

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

- Apply Clamp Restraint
- Apply Distributed Force
- Use Model Checker tool






---



---



---



---



---

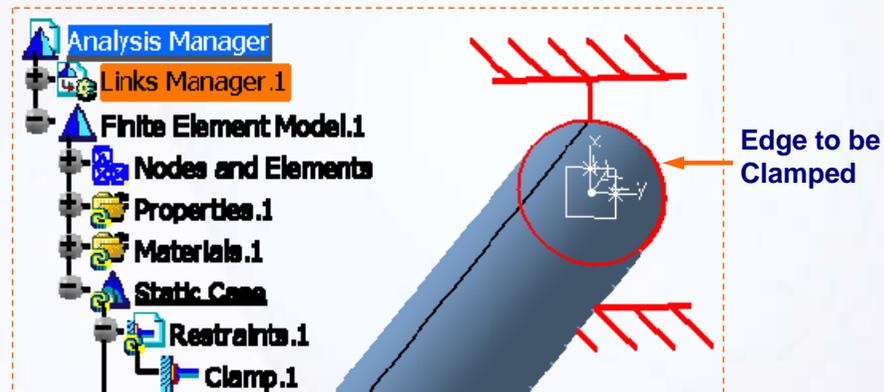


---

## Exercise 2E (1/3)

### 1. Apply Boundary Conditions.

- Open  
2E\_Hollow\_Shaft\_2D\_Start.CATAnalysis
- Apply Clamp Restraint.
- Select Edge shown as support for clamp.






---



---



---



---



---



---

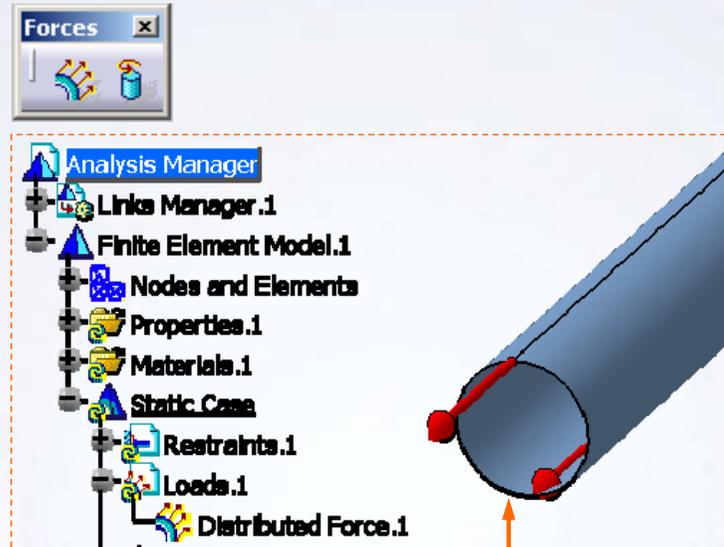
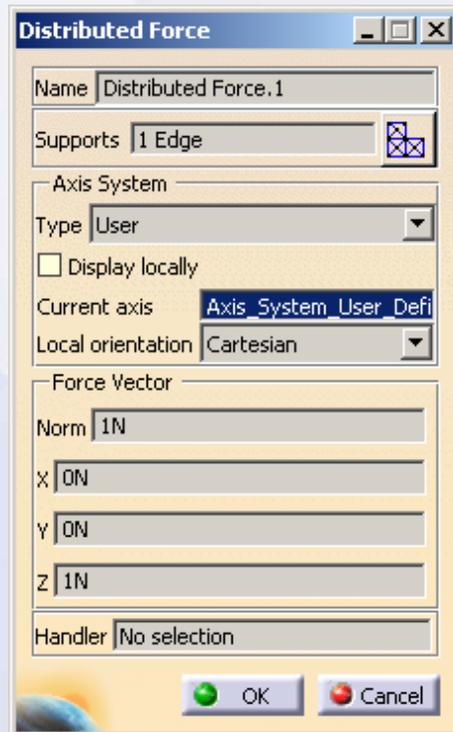


---

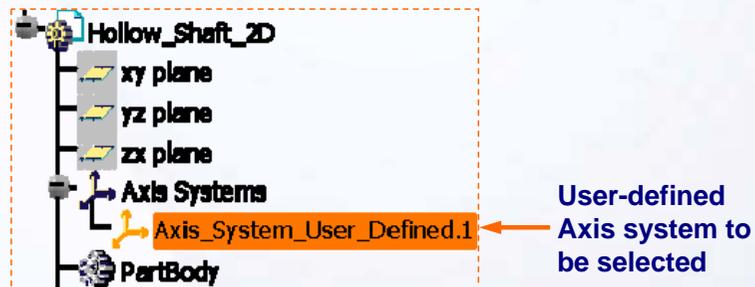
## Exercise 2E (2/3)

### 2. Applying Load.

- Apply Distributed Force.
- Select the edge shown as support.
- Select User-defined Axis System.
- Enter 1N force in Z field as shown.



Edge to be selected to apply Distributed force



User-defined Axis system to be selected




---



---



---



---



---



---



---

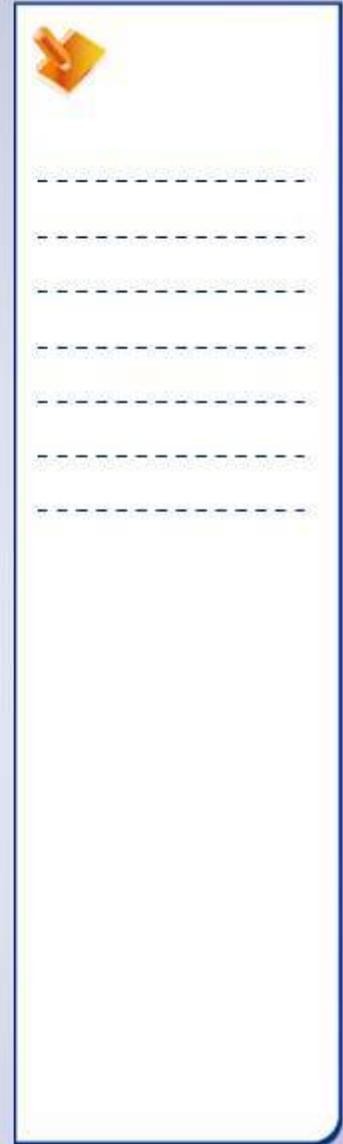
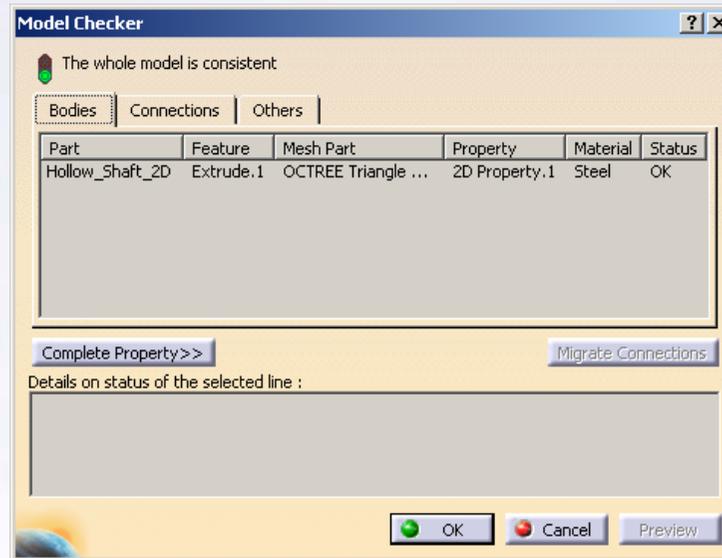
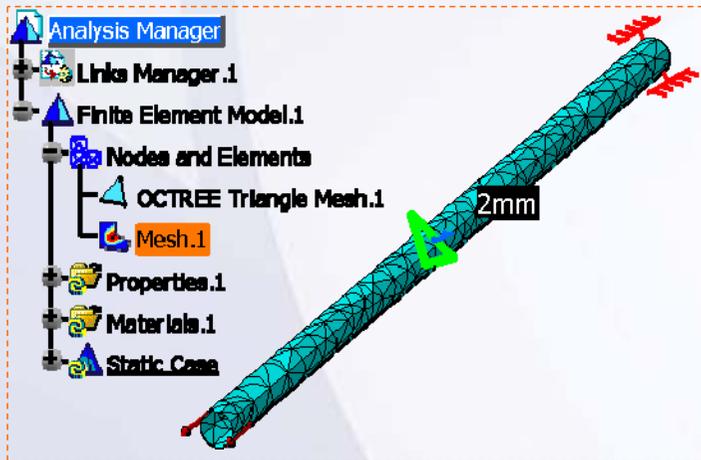
## Exercise 2E (3/3)

### 3. Checking the FE Model.

- Use Model checker tool to check model inconsistency.

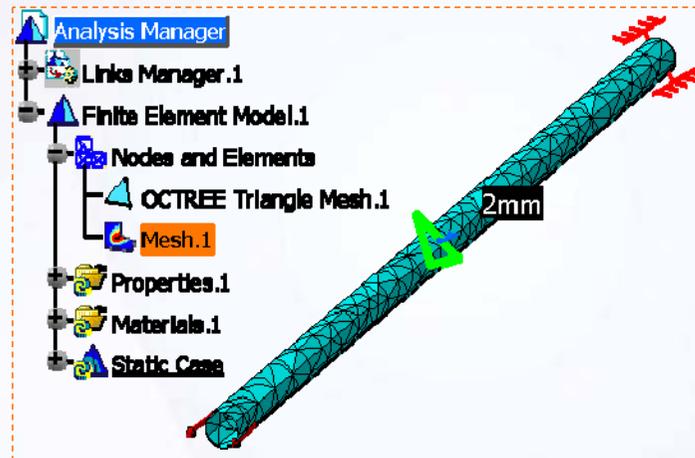
### 4. View Mesh Image.

- Activate the Mesh Image by double-click on Mesh image in specification tree.



## Exercise 2E: Recap

- ✓ Apply Clamp Restraint.
- ✓ Apply Distributed Force.
- ✓ Use Model Checker tool.



✎

---



---



---



---



---



---



---

## Exercise 2F

### Recap Exercise

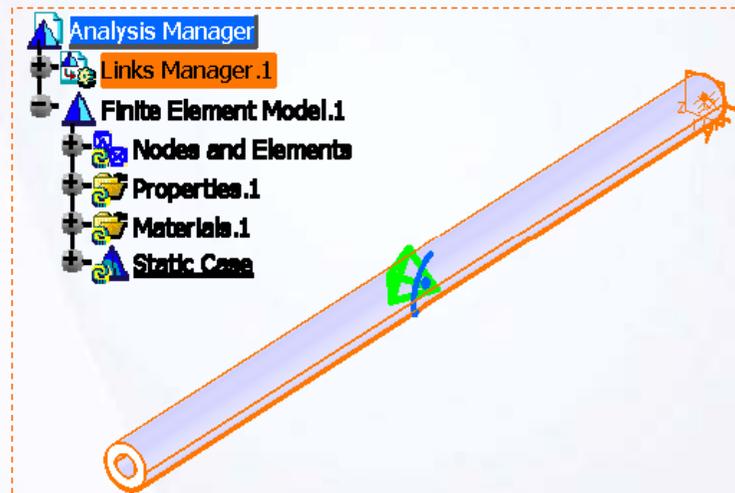


15 min

*In this exercise, you will apply boundary conditions, load and check the FE model for 3D mesh. High-level instructions are provided for this exercise.*

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

- Apply Clamp Restraint
- Apply Pressure
- Use Model Checker tool






---



---



---



---



---

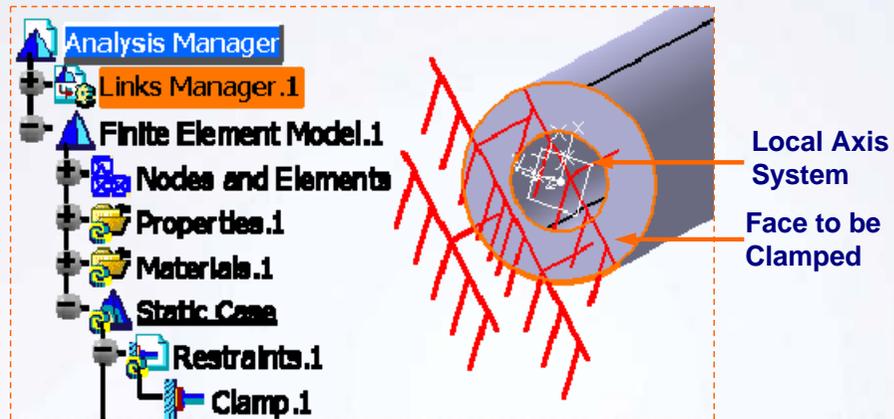


---

## Exercise 2F (1/3)

### 1. Apply Boundary Conditions.

- Open  
2F\_Hollow\_Shaft\_3D\_Start.CA  
TAnalysis
- Apply Clamp Restraint.
- Select the face of hollow shaft  
shown as support for clamp.




---



---



---



---



---

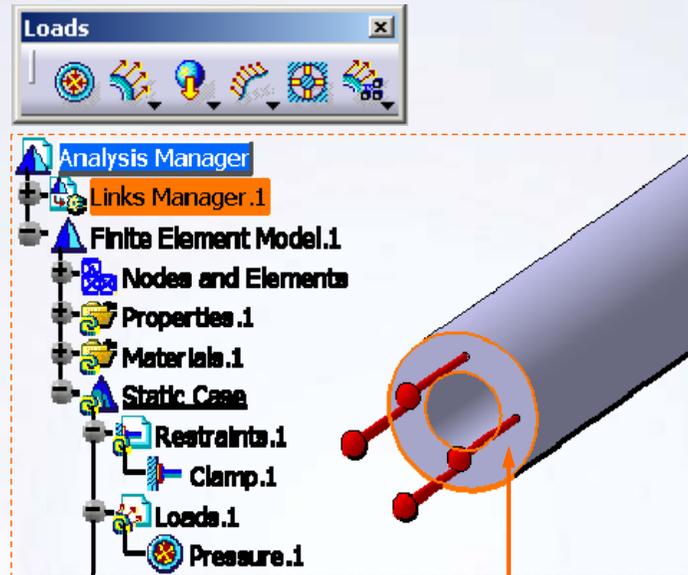


---

## Exercise 2F (2/3)

### 2. Applying Load.

- Apply Pressure.
- Select the surface shown as support.
- Enter -1N\_m2 in Pressure field as shown.



Surface to be selected to apply Pressure






---



---



---



---



---



---



---

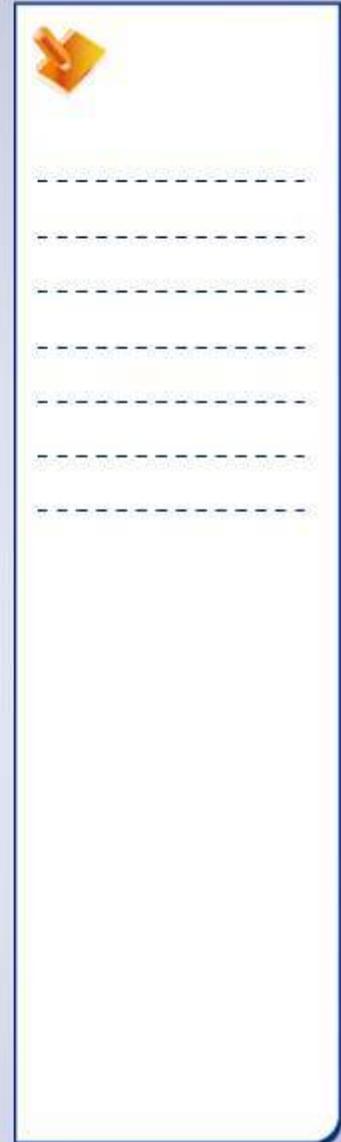
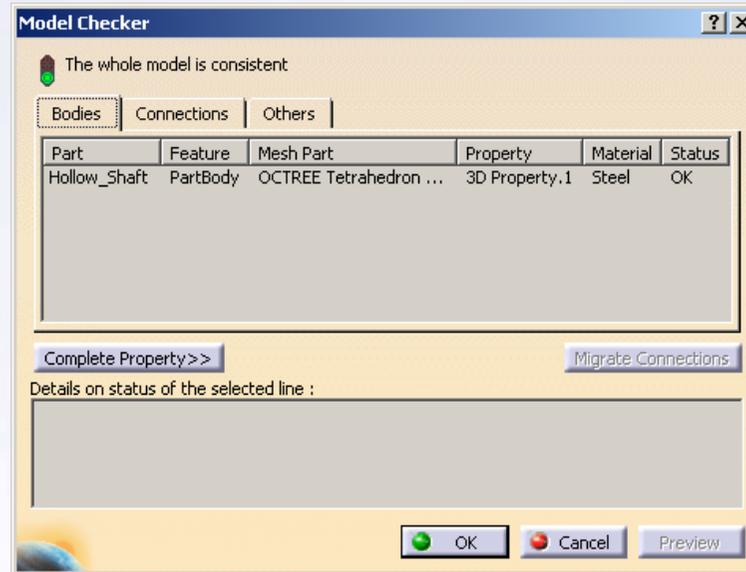
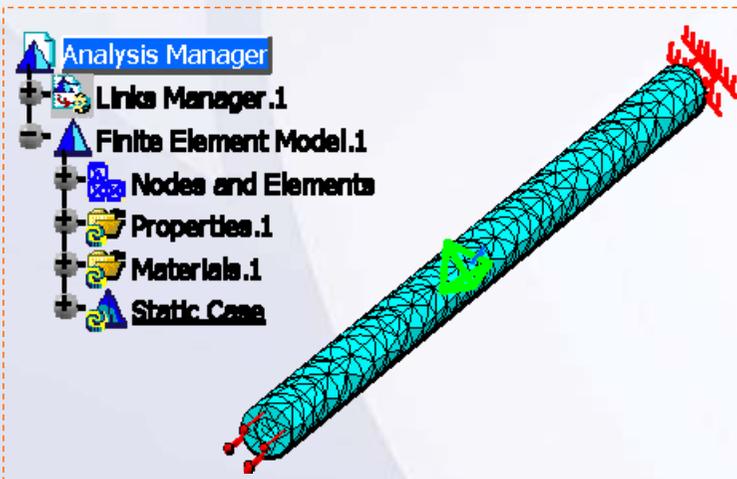
## Exercise 2F (3/3)

### 3. Checking the FE Model.

- Use Model checker tool to check model inconsistency.

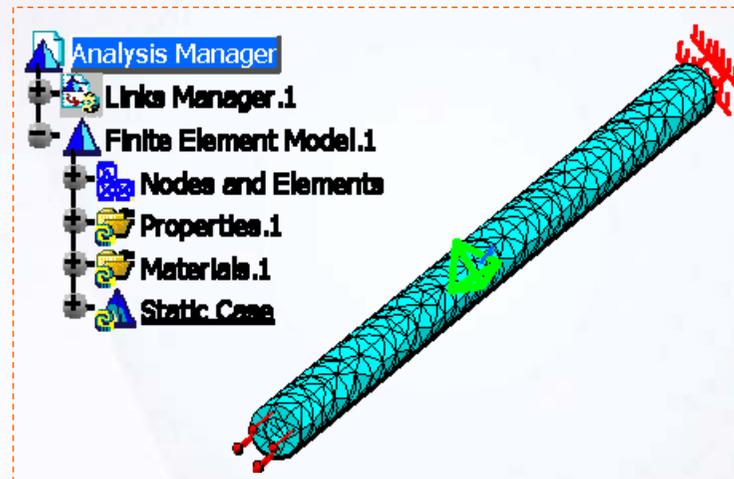
### 4. View Mesh Image.

- Activate the Mesh Image by double-click on Mesh image in the specification tree.



## Exercise 2F: Recap

- ✓ Apply Clamp Restraint.
- ✓ Apply Distributed Force.
- ✓ Use Model Checker tool.



Handwritten notes area with a yellow arrow icon at the top and several horizontal dashed lines for writing.

## Case Study: FE Modeling of Drill Press Table

### Recap Exercise

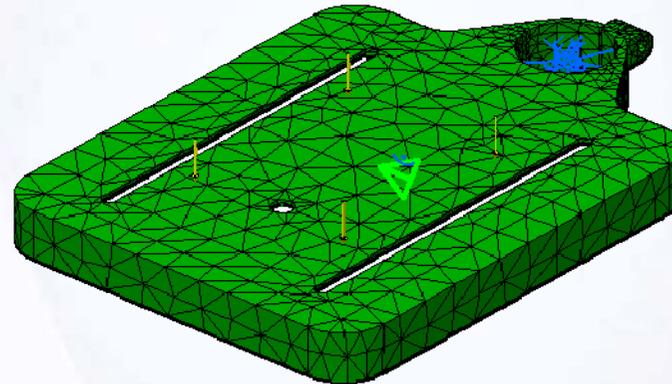


30 min

**You will practice what you learned, by completing the case study model. In this exercise, you will create FE model for Drill Press Table.**

**Recall the design intent of this model:**

- ✓ Understand the structural properties of Drill Press Table. Check for standard material with these properties in material catalogue or create new User material.
- ✓ Decide the mesh (1D, 2D or 3D) to be used to map actual problem into FE model. As per the geometry dimensions use appropriate Mesher.
- ✓ Understand and assign the physical property to the created mesh part.
- ✓ Understand and apply the restraints (boundary conditions) on the mesh part.
- ✓ Understand and apply the forces acting on the Drill Press Table.
- ✓ Check the created FE model.

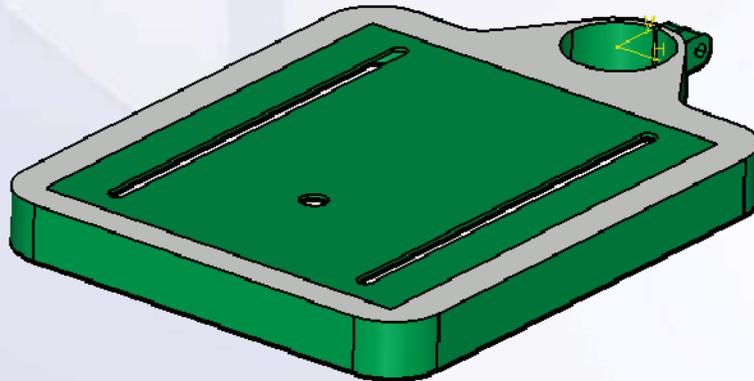


## Case Study: Introduction

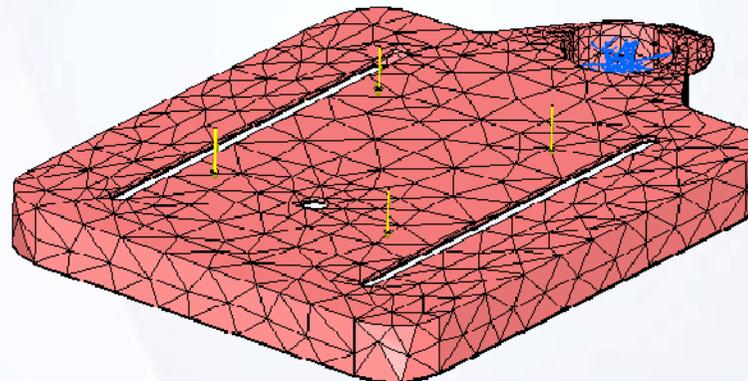
The case study for this lesson is the Drill Press Table. The focus of this case study is to create a Finite Element model for Drill Press Table. You will perform following steps independently in order to achieve this.

1. Create and apply specified Material.
2. Decide and create relevant mesh.
3. Decide and assign corresponding physical property.
4. Decide and apply boundary conditions.
5. Decide the loading conditions and apply load.
6. Check the FE model.

You will understand the actual physical problem and use this information to perform above steps to create FE model.



Drill press Table Component



Drill press Table FE Model




---



---



---



---



---



---



---

## Design Intent (1/4)

You will now understand the physical problem. The figure shows the drill press assembly. You can see, while drilling operation, the Drill Press Table is fixed to the stand as shown in fig. The workpiece is located firmly on the table with clamps in order to get drilled.

- ✓ Understand the structural properties of Drill Press Table. The table of drill press is composed of steel having following properties.

The material is an isotropic material.

Young's Modulus =  $2.1 \times 10^{11}$  N\_m2.

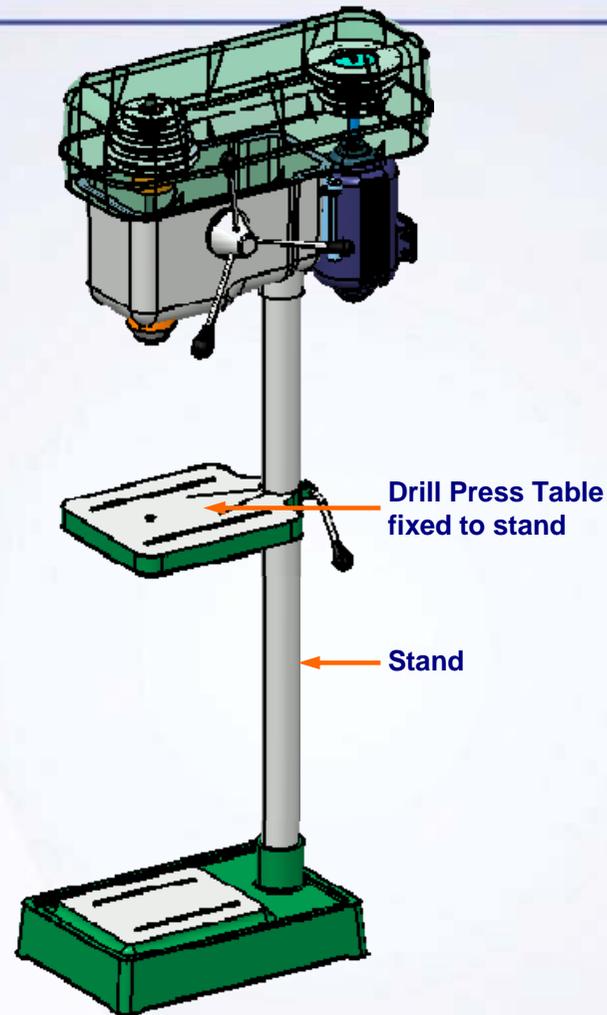
Poisson's Ratio = 0.268.

Density = 7862 kg/m3.

Thermal Expansion =  $1.19 \times 10^{-5}$  /Kdeg.

Yield Strength =  $2.6 \times 10^8$  N\_m2.

- Check for the standard material with these properties in material catalogue or create new material with User Material and edit properties.



-----

-----

-----

-----

-----

-----

## Design Intent (2/4)

- ✓ Decide the mesh (1D, 2D or 3D) to be used to map actual problem into FE model.

In this case, it is not possible to use beam FE model. The surface FE model will take more efforts to convert part model to surface model and may oversimplify the geometry by removing stiffeners. You can directly model this part as 3D mesh to take advantage of existing part model.

- Use OCTREE Tetrahedron mesh

- ✓ Understand the physical property to be applied on the mesh parts:

The type of physical property to be applied depends on the type of mesh used.

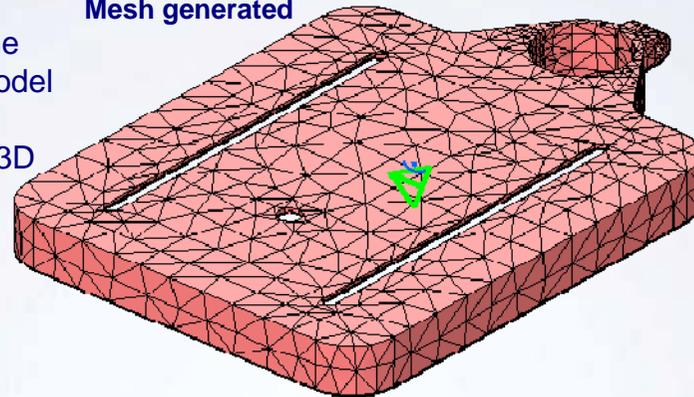
- If Part model is 3D Geometry, the OCTREE Tetrahedron mesh and 3D Property is automatically applied when you switch to GPS workbench.

- ✓ Understand the restraints (boundary conditions) on the parts:

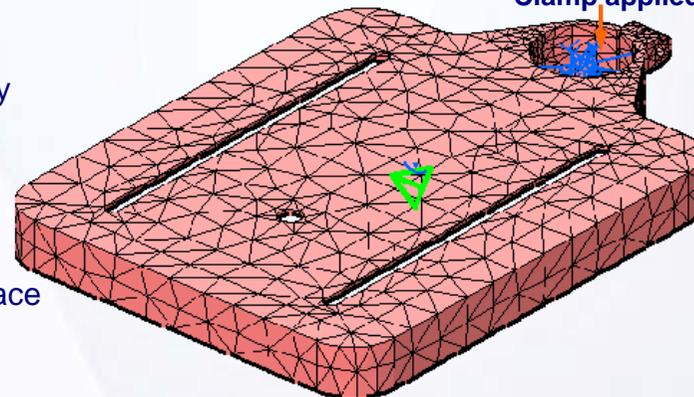
As the Table is fixed to the stand, it means that the surface of table which is in contact with stand surface will not deform. It means all DOFs are fixed for this surface of table.

- Thus, you can apply Clamp restraint for this surface.

Mesh generated



Clamp applied






---



---



---



---



---



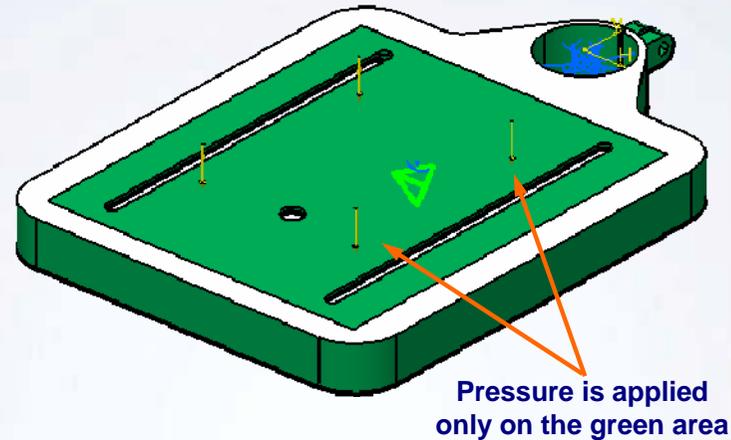
---

## Design Intent (3/4)

- ✓ Understand the forces acting on the Drill Press Table.

Now you will study the forces acting on the table. The torque applied by the electric motor is transmitted to table through drill-bit and workpiece. The drill press operator applies downward force of 10 N to move drill in vertically downward direction. This force will be transmitted to table through drill-bit and workpiece. The torque and downward force will be transmitted to table through bottom surface of workpiece. You can assume both forces remain constant with respect to time. As the applied loads are stationary with time, you will perform Static Analysis. For the purpose of simplicity you will neglect Torque and you will perform the analysis assuming only downward force acting on the table.

- Use Static Analysis Case
- The downward force can be approximated as pressure on the area equal to bottom of workpiece.



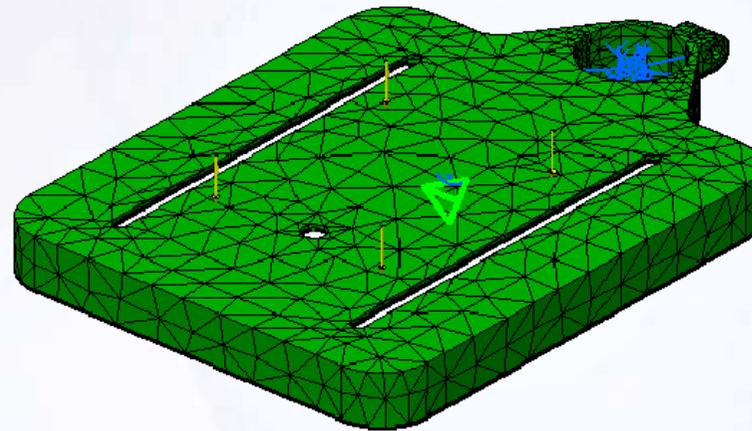
## Design Intent (4/4)

- ✓ Validate the FE model created.

It is necessary to check the FE model to ensure that all steps required to perform computation are done.

- Use Model Check tool.

You can summarize the pre-processing information received from the above study: Now you can map this pre-processing information into FE Model in CATIA.



Drill press Table FE Model




---



---



---



---



---



---



---

## Do It Yourself : FE Modeling of Drill Press Table

The following steps offer hints to guide you through the creation of FE model for Drill Press Table.

**1. Open the given part Drill\_Press\_Table\_Start model. Switch to GPS workbench.**

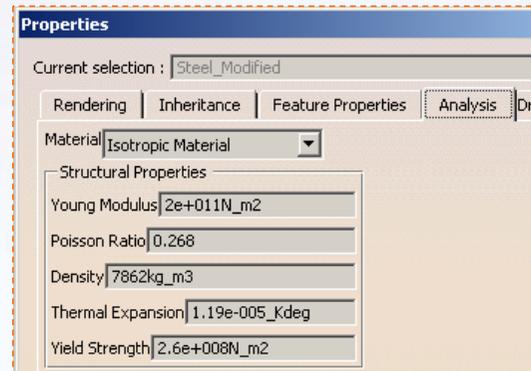
Browse through the files and open the model 'Drill\_Press\_Table\_Start .CATPart

**2. Create User Material.**

Create a User Material steel and edit its structural properties for material specifications provided. Name the Material as Modified\_Steel.

**3. Edit OCTREE Tetrahedron Mesh.**

Keep the default mesh size and sag values as 10% of mesh size. Generate Mesh Visualization.






---



---



---



---



---



---

## Do It Yourself : FE Modeling of Drill Press Table

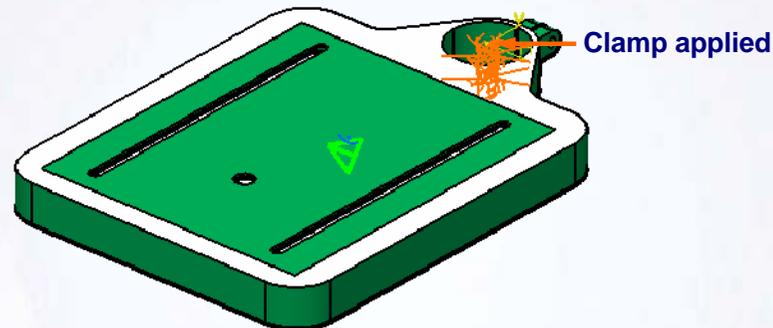
**4. Edit 3D Property.**

Select the Modified\_Steel in property.



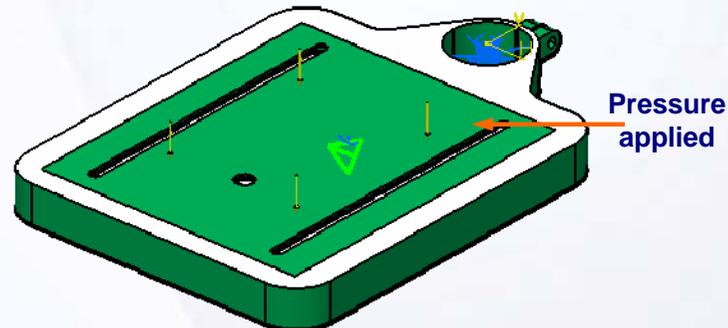
**5. Apply Boundary Conditions.**

Apply Clamp Restraint to the surfaces shown.



**6. Apply Loads.**

You have to apply 10 N Load as pressure on selected area shown. Measure the surface area and calculate the pressure value.

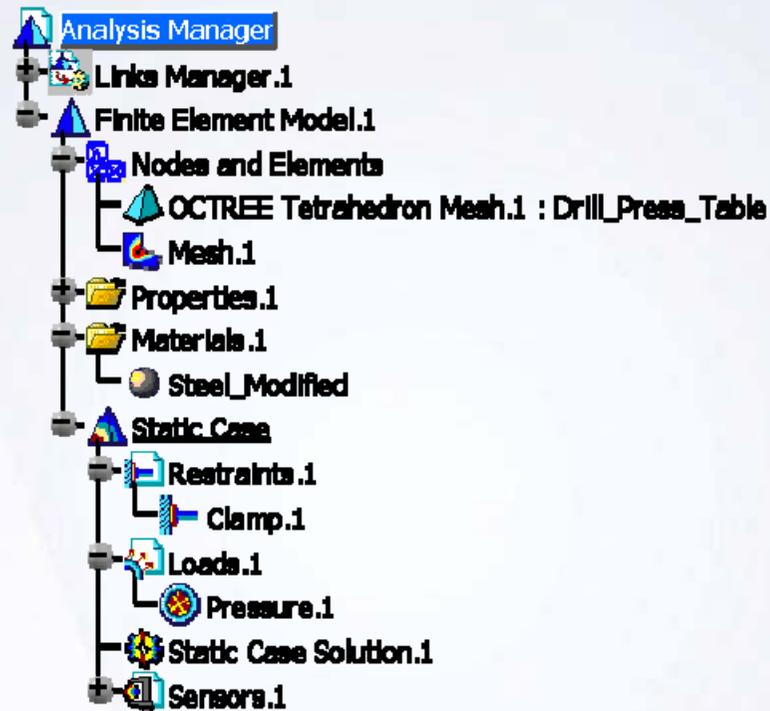


**7. Check the Model.**

Check the model for consistency.

## Case Study: FE Modeling of Drill Press Table Recap

- ✓ Understand the structural properties of Drill Press Table. Check for standard material with these properties in material catalogue or create new material.
- ✓ Decide the mesh (1D, 2D or 3D) to be used to map actual problem into FE model. As per the geometry dimensions use appropriate Mesher.
- ✓ Understand and assign the physical property to be applied on the created mesh part.
- ✓ Understand and apply the restraints (boundary conditions) to be applied on the mesh part.
- ✓ Understand and apply the forces acting on the Drill Press Table.
- ✓ Check the created FE model.






---



---



---



---



---



---

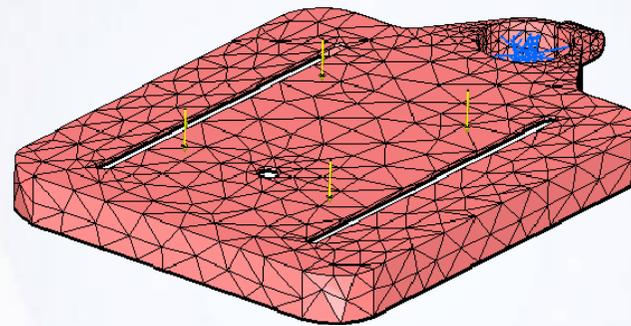
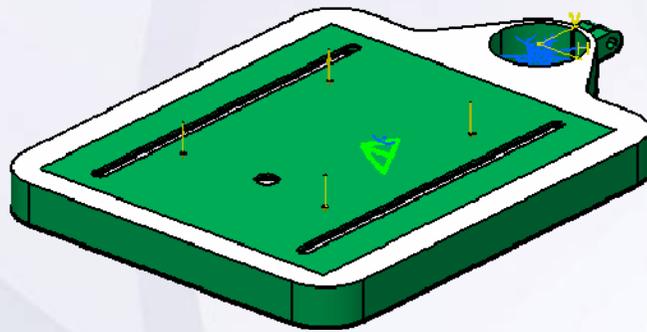
# Computation

3

**Learning Objectives:**

*Upon completion of this lesson you will be able to:*

- ✓ Introduce yourself with Computation process
- ✓ Understand different Solving Methods
- ✓ Compute a Static Case



2 hours

Handwriting practice area with a yellow arrow icon at the top left and several horizontal dashed lines for writing.

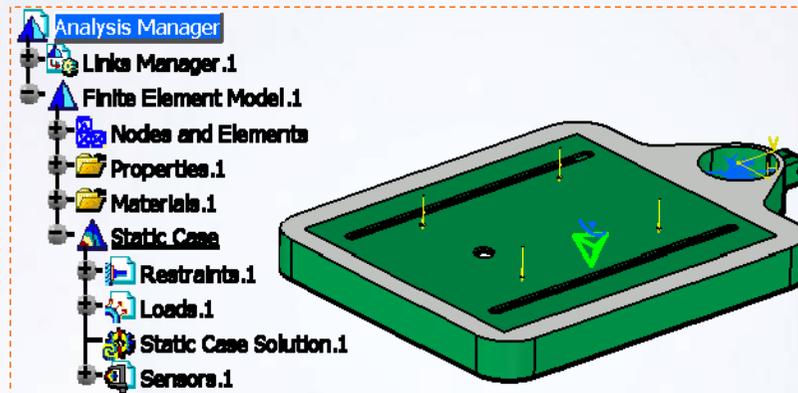
## Case Study

The case study for this lesson is the Drill Press Table FE model computation. The focus of this case study is to select proper computation method to solve the finite element model.

## Design Intent

You have to solve the Finite Element model of the Drill press Table.

- ✓ You will provide path for results and computation files.
  - Specify External Storage file path
- ✓ You will find the number of Degrees of Freedom (DOF) for the model and based on that select the appropriate solving method.
- ✓ Perform the computation of the FE model.
- ✓ Check the ELFINI log file.
- ✓ Save all the files.



## Stages in the Process

This will involve following steps to perform in the case study. First you will understand what is Computation. Later you will see how to perform computation step in the GPS workbench.

1. What is Computation?
2. Solving Methods
3. Computing a Static Case




---



---



---



---



---



---

## Introduction to Computation

In Finite Element Analysis, computation is performed to calculate unknown displacement values at nodal points of FE model. From these displacement values, other solution quantities such as Strains, Principal stresses, and Von Mises stresses are calculated.

After discretization, nodal displacements are calculated by solving the global equation -- (1). From displacements, using Strain-Displacement relations -- (2), strains are calculated. From strains using Stress-Strain relations -- (3), stresses are calculated.

$$\{F\} = [K] \{U\} \quad -- (1)$$

$$\{\epsilon\} = [B] \{U\} \quad -- (2)$$

$$\{\sigma\} = [C] \{\epsilon\} \quad -- (3)$$

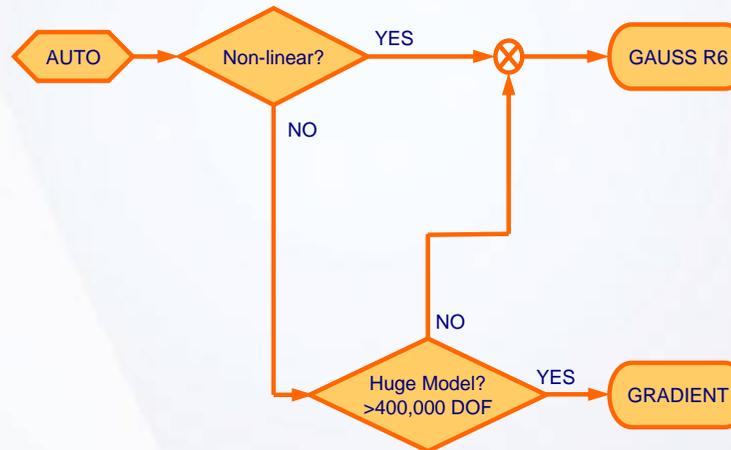
Where  $\{\epsilon\}$ : Strain Matrix  
 [B]: Strain-Displacement Matrix  
 $\{\sigma\}$ : Stress Matrix  
 [C]: Stress-Strain Matrix  
 $\{F\}$ : Nodal Force Matrix  
 [K]: Global Stiffness Matrix  
 $\{U\}$ : Nodal Displacement Matrix

## Solving Methods

For a static analysis, the displacement vector can be computed using various methods, depending on the model configuration. There are the following four options for the methods:

- A. GAUSS
- B. GAUSS R6
- C. GRADIENT
- D. AUTO

For the AUTO option, the criteria used for selection of the method internally, is as shown.



Criterion used for selection of method.




---



---



---



---



---



---



---

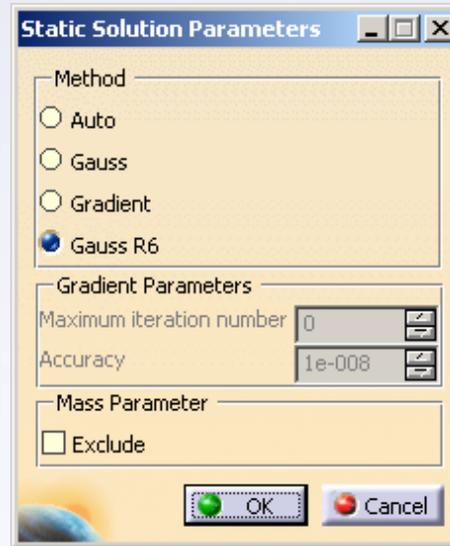
## Computing a Static Case

Before computing a static case, you need to set proper Data Save Options.

Also, for the first time you need to specify some paths. You need to specify the temporary external storage path for temporary computation and ELFINI Solver Log path for storing the log of ELFINI solver. Also you need to specify ELFINI Solver External Storage path, for storing these files.

Before starting the computation select the method of computation. Select the analysis case to compute and complete the computation process.

You can also submit the job in batch mode, to continue the interaction with CATIA while the analysis runs.






---



---



---



---



---



---



---

## Main Tools

### Solver Tools

- 1 **Temporary External Storage:** lets you specify the temporary external storage path.
- 2 **External Storage:** lets you specify the external storage folder path for CATAnalysisResultss file and CATAnalysisComputations file.

### Analysis Results

- 3 **Elfini Solver Log:** lets you specify the folder for Elfini solver log file.

### Compute

- 4 **Compute:** lets you compute the analysis.






---



---



---



---



---



---

# Exercise 3B

## Recap Exercise

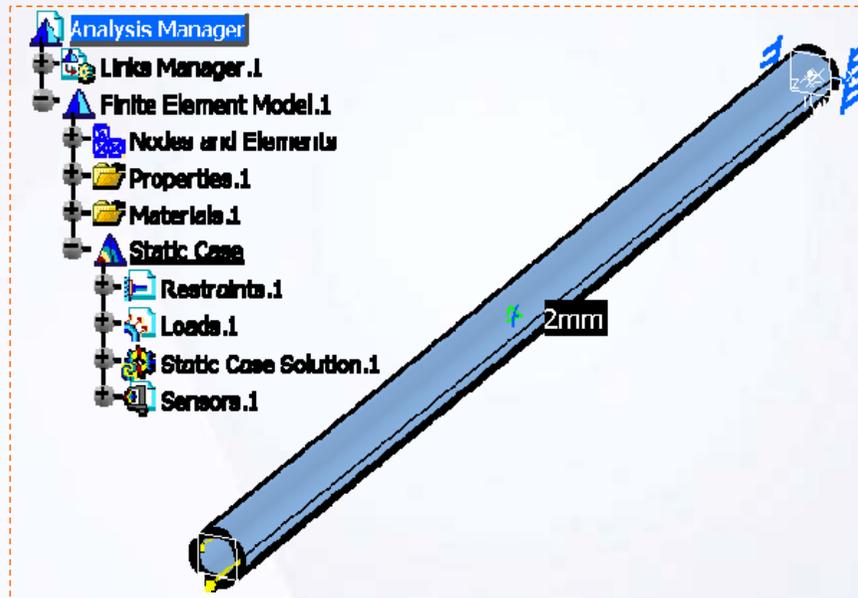


15 min

*In this exercise, you will perform computation for 2D component. High-level instructions are provided for this exercise.*

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

- Specify External Storage path
- Compute Static Analysis Case






---



---



---



---



---



---

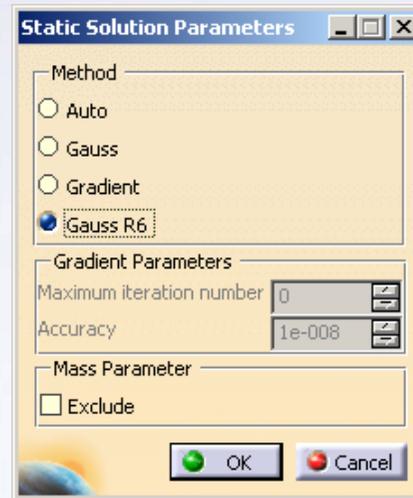


---



## Exercise 3B (2/2)

3. **Specify the Solving Method.**
  - Use Gauss R6 as solving method, as it is fast and uses improved algorithm .
  
4. **Compute the Analysis Case.**
  - Compute the Analysis Case with Preview option unchecked.
  
5. **Check ELFINI Log file.**
  
6. **Save all the files.**






---



---



---



---



---



---



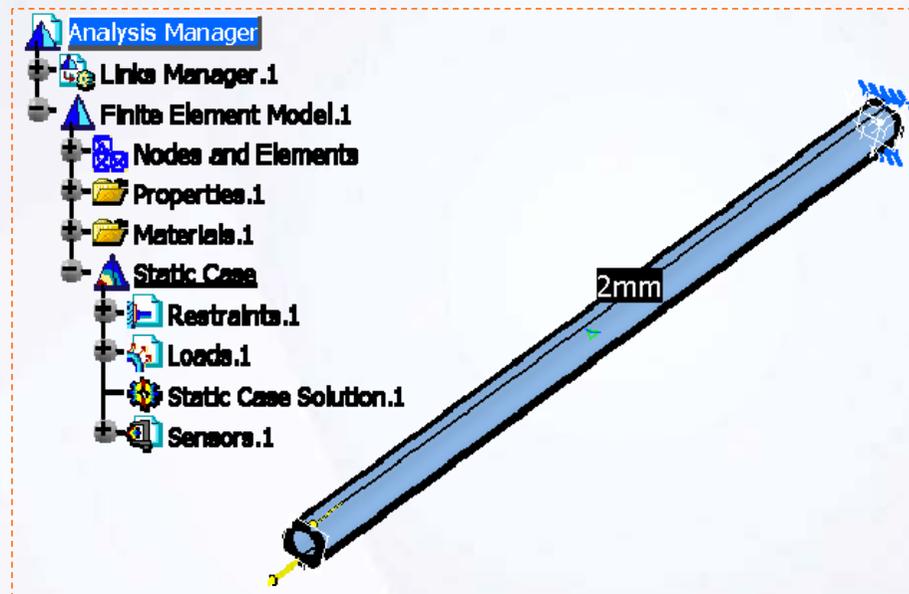
---



---

## Exercise 3B: Recap

- ✓ Specify External Storage path.
- ✓ Compute Static Analysis Case.



Handwritten notes area with a yellow arrow icon at the top and several horizontal dashed lines for writing.

# Exercise 3C

## Recap Exercise

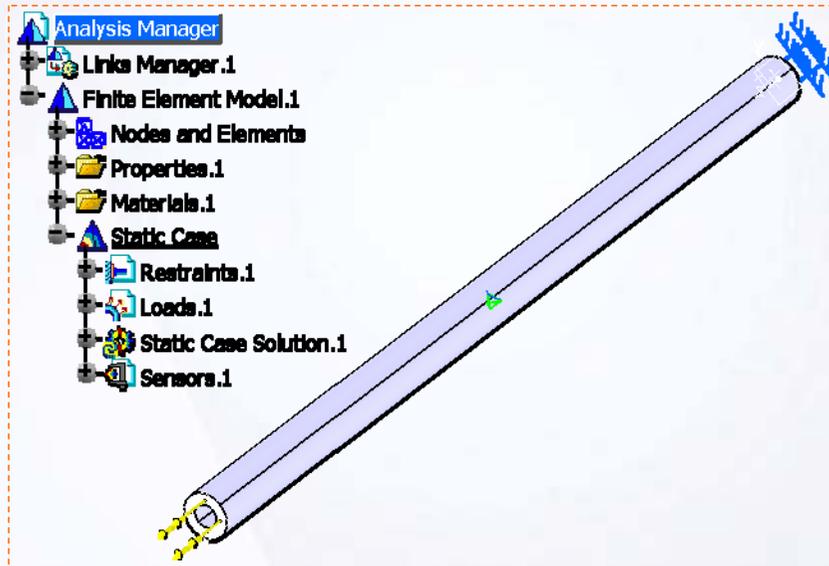


15 min

*In this exercise, you will perform computation for 3D component. High-level instructions are provided for this exercise.*

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

- Specify External Storage path
- Compute Static Analysis Case






---



---



---



---



---



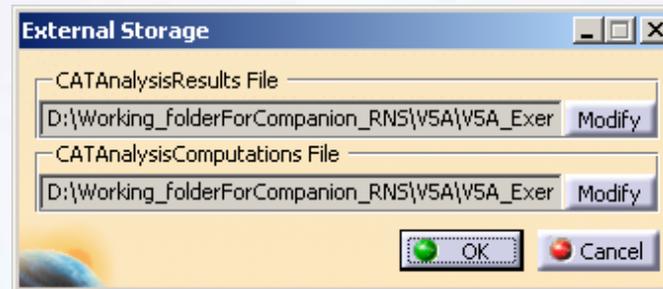
---



---

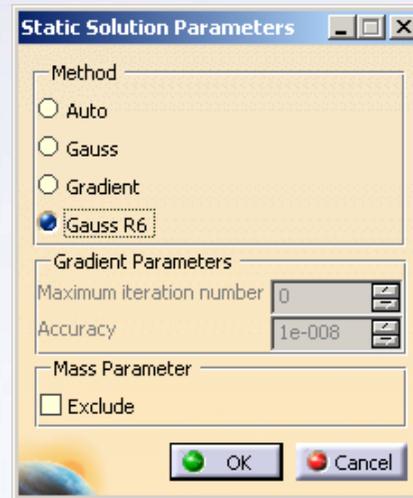
## Exercise 3C (1/2)

1. Open a Analysis Document  
3C\_Hollow\_Shaft\_3D\_Start.CATAnalysis.
2. Specify External Storage file path.

A vertical rectangular box on the right side of the page. It contains a yellow arrow icon pointing downwards at the top. Below the arrow are several horizontal dashed lines, intended for a student to write their answers to the exercise steps.

## Exercise 3C (2/2)

3. **Specify the Solving Method.**
  - Use Gauss R6 as solving method, as it is fast and uses improved algorithm.



4. **Compute the Analysis Case.**
5. **Check ELFINI Log file.**
6. **Save all the files.**






---



---



---



---



---



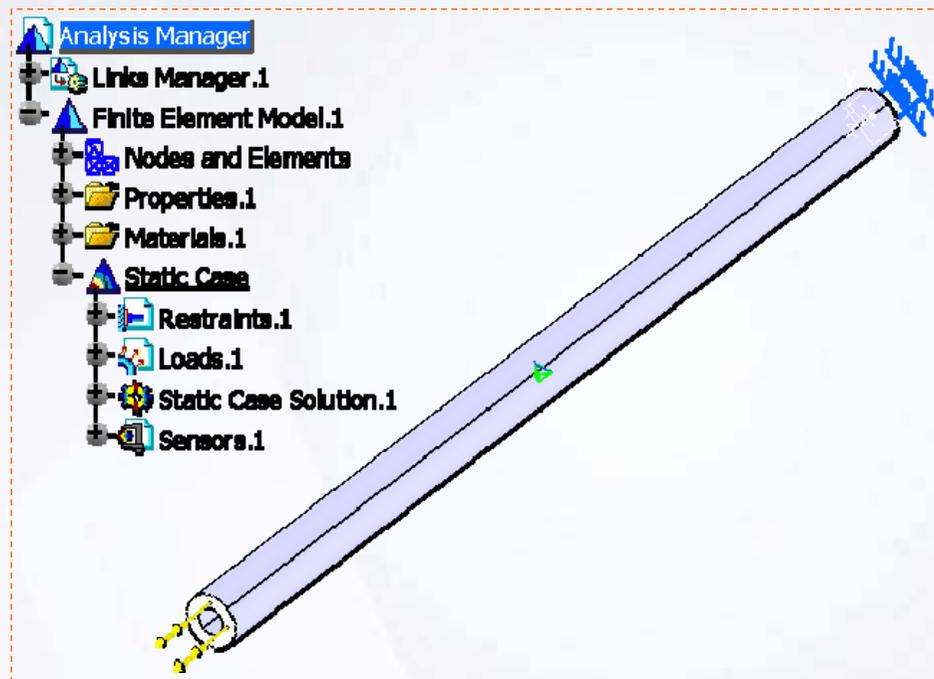
---



---

## Exercise 3C: Recap

- ✓ Specify External Storage path.
- ✓ Compute Static Analysis Case.



Handwritten notes area with a yellow arrow icon and several horizontal dashed lines for writing.

# Case Study: Computing Drill Press Table

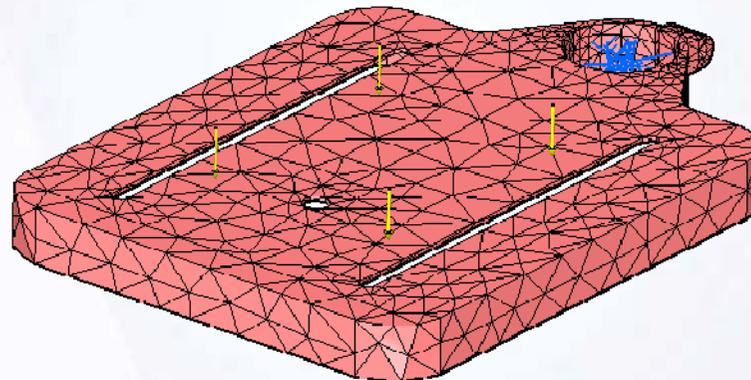
## Recap Exercise



30 min

**You will practice what you have learned, by completing the case study model. In this exercise, you will compute FE model for Drill Press Table. Recall the design intent of this model:**

- ✓ Specify the External Storage path.
- ✓ Select the solving method.
- ✓ Compute the Analysis Case.
- ✓ View the ELFINI Log file.
- ✓ Save all the documents






---



---



---



---



---



---

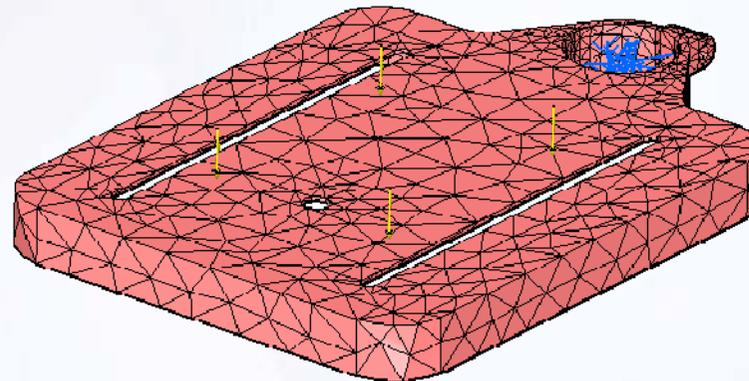


---

## Case Study: Introduction

The case study for this lesson is the Drill Press Table FE model computation. The focus of this case study is to carry out computation task independently. You will perform the following steps in order to achieve this.

1. Specify the External Storage path.
2. Select the appropriate solving method based on solving method guidelines.
3. Compute the Analysis Case.
4. View the ELFINI Log file.
5. Save all the documents



Drill press Table FE Model



-----

-----

-----

-----

-----

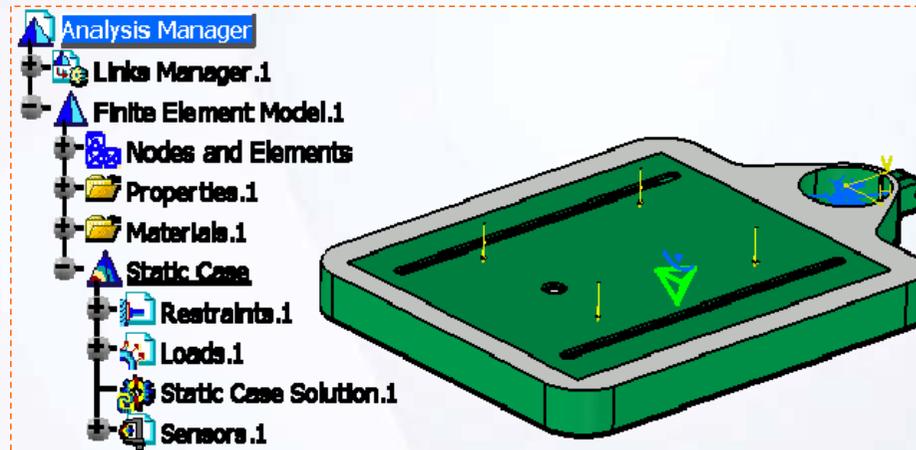
-----

## Design Intent

- ✓ You will provide paths for results and computation files.
  - Specify External Storage file path
- ✓ You will then find the number of DOF for the FE model and based on that you will select the solving method.

You can get the number of DOF in following way. In Analysis Tools toolbar, use information icon on Mesh Image. You will get number of nodes for the FE model. For Tetrahedron element in CATIA you have 3 DOF per node. From this you will get total number of DOF for the FE model. Use this information along with solving method guidelines.

- Use solving method based on above information.
- ✓ Compute the Analysis Case.
- ✓ Check ELFINI Log file.
- ✓ Save all the files.






---



---



---



---



---



---

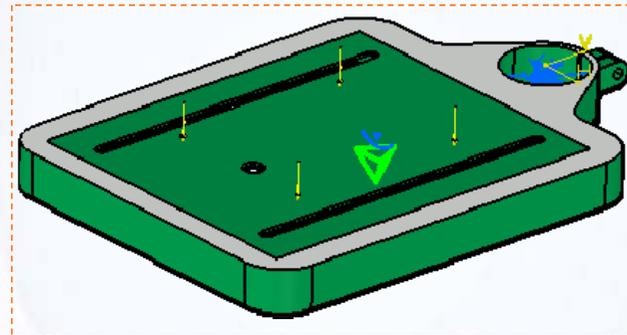


---

## Do It Yourself

The following steps offer hints to guide you through the computation of FE model for Drill Press Table.

1. **Open an Analysis Document**  
Drill\_Press\_Table\_Computation\_Start.CATAnalysis.
2. **Specify the External Storage file path.**
3. **Specify the Solving Method.**
  - Use Gauss R6 as solving method, as it is fast and uses improved algorithm.
4. **Compute the Analysis Case.**
5. **Check the ELFINI Log file.**
6. **Save all the files.**






---



---



---



---



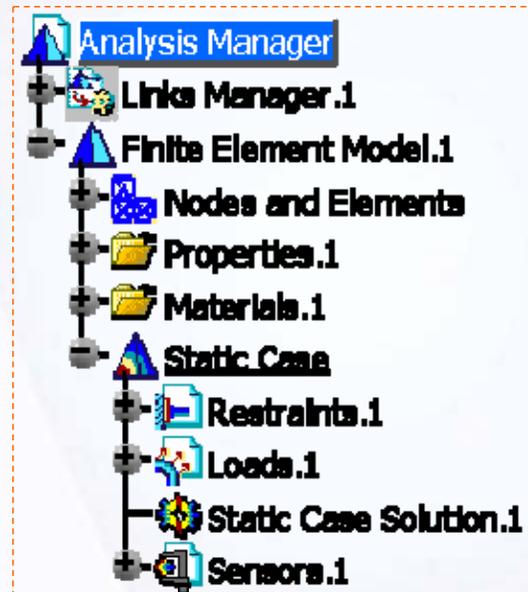
---



---

## Case Study: Computing Drill Press Table Model Recap

- ✓ Specify External Storage path.
- ✓ Select the solving method.
- ✓ Compute the Analysis Case.
- ✓ View ELFINI Log file.
- ✓ Save all the documents.






---



---



---



---



---



---



---

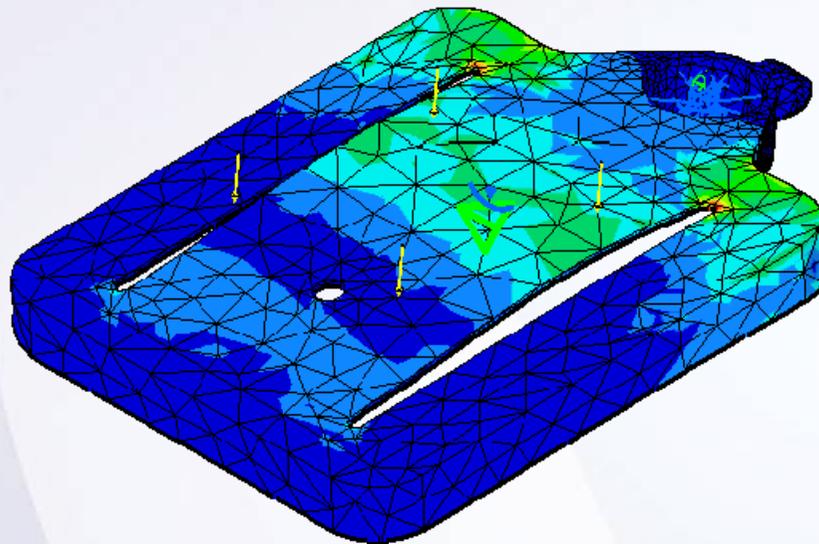
# Post-processing

4

**Learning Objectives:**

*Upon completion of this lesson you will be able to:*

- ✓ Understand post-processing.
- ✓ Visualize the required result images.
- ✓ Understand the use of sensors.
- ✓ Generate reports.



2 hours

✎

---

---

---

---

---

---

---

---

---

---

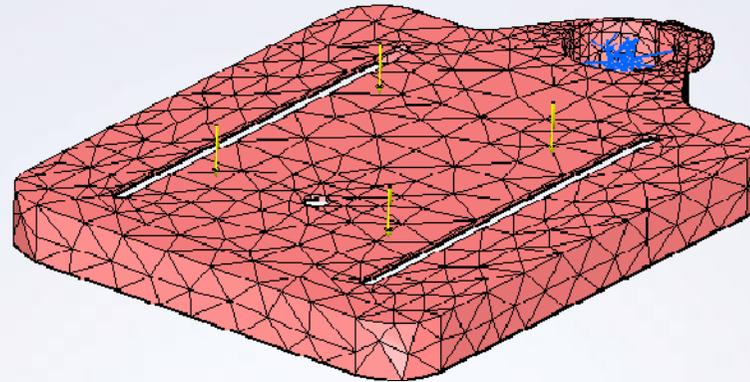
## Case Study

The case study for this lesson is the Drill Press Table FE model post-processing. The focus of this case study is to post-process the computed data as per requirement.

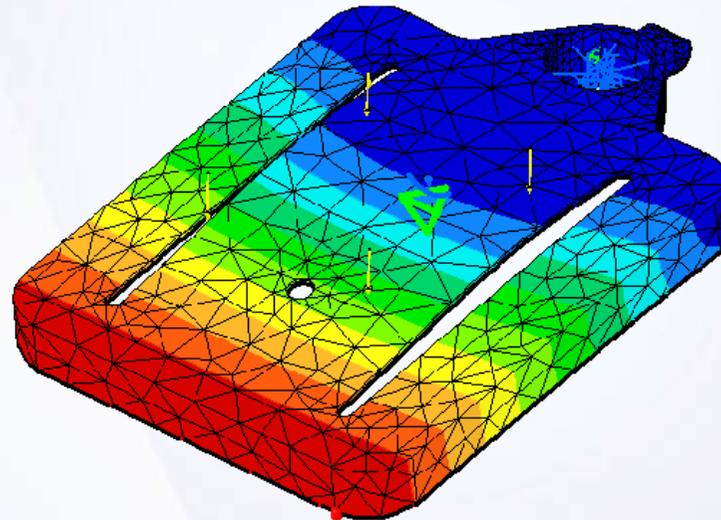
## Design Intent

You must decide which results are required to be viewed. Based on this decision you will create the visualization images and an analysis report which will clearly summarize the results of the computation.

- ✓ You will create Translational Displacement Magnitude Plot, also find the value of maximum displacement and its location.
- ✓ You will create plots for Maximum and Minimum Principle stress distribution with discontinuous iso. Find out the location of the maximum values.
- ✓ You will create Von Mises stress distribution plot and find the location of maximum value in FE model.
- ✓ You will create Local error distribution and find the location of maximum value.
- ✓ You will create output sensors for Error in Energy, Global Error Rate percentage.
- ✓ You will create Resultant sensor to know reactions at restraints.
- ✓ You will create Analysis Report with Images.



Drill press Table FE Model



Drill press Table Translational Displacement Image




---



---



---



---



---



---

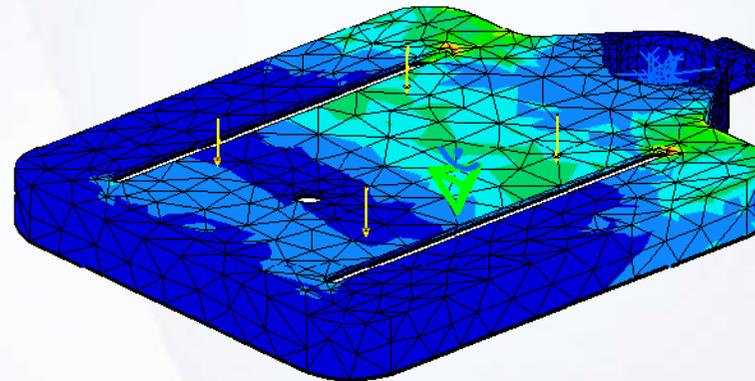
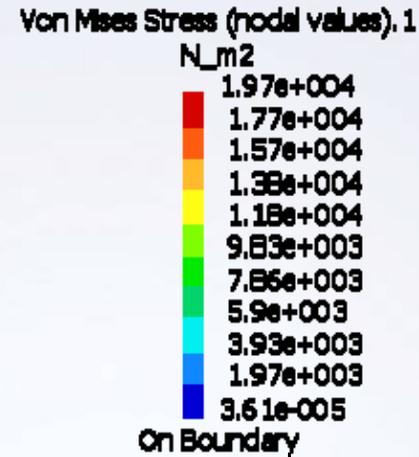


---

## Stages in the Process

This will involve the following steps to perform the case study. First, you will learn what is meant by Post-processing. Later, you will see how to use the various result visualization functionalities in the GPS workbench.

1. Understand Post-processing.
2. Create Visualization Images.
3. Create Extrema on created images.
4. Use sensors.
5. Generate analysis report.






---



---



---



---



---



---



---

## What is Post-processing

Post-processing involves all those steps which are carried out after computation of results. In this step you will:

- ✓ Create different images like Deformation, Displacement magnitude, Stress, Reaction force, and other available images from computed solution data.
- ✓ Find location of result values in FE model.
- ✓ Validate the results using different images and study these images to understand and interpret the solution.
- ✓ Make decisions for further improving the solution with mesh refinement iterations or other solution types.
- ✓ Validate the current design or provide the changes based on the results.

## Visualization Images

CATIA provides the different visualization images to study the results of the analysis as shown in the table.

When you edit an image, you can choose the image type. Location of the result values, provided by the solver, changes according to the selected type. You can choose the location for text and symbol types only.

Image	Purpose of the image
Deformation	To visualize finite element mesh in the deformed configuration
Displacement	To visualize displacement field patterns which represent variation of position vectors of material particles
Principal Stresses	To visualize principal stress field patterns which represent a tensor field quantity used to measure the state of stress
Von Mises Stresses	To visualize the Von Mises Stress field patterns, which represent a scalar field quantity obtained from the volume distortion energy density
Precision	To visualize computation error maps




---



---



---



---



---



---



---

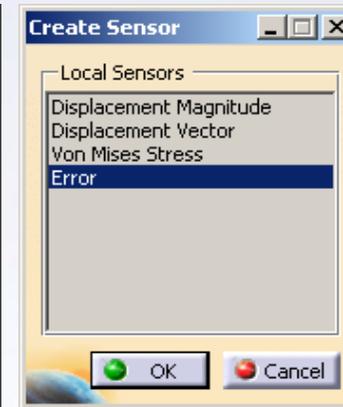
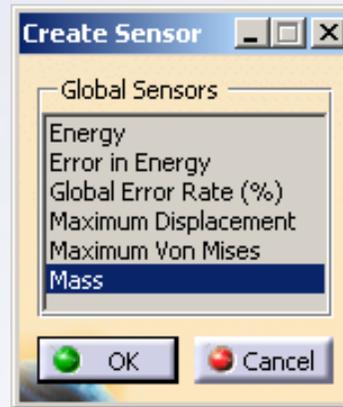
## Sensors

Sensor is a physical output of a computation. Sensors allow you to produce specialized output provided as a single value rather than a range of values displayed on the model.

You can use sensor output to:

- ✓ Validate the Analysis Results
- ✓ Synthesize the analysis results and use it as a parameter to improve and optimize design specifications.

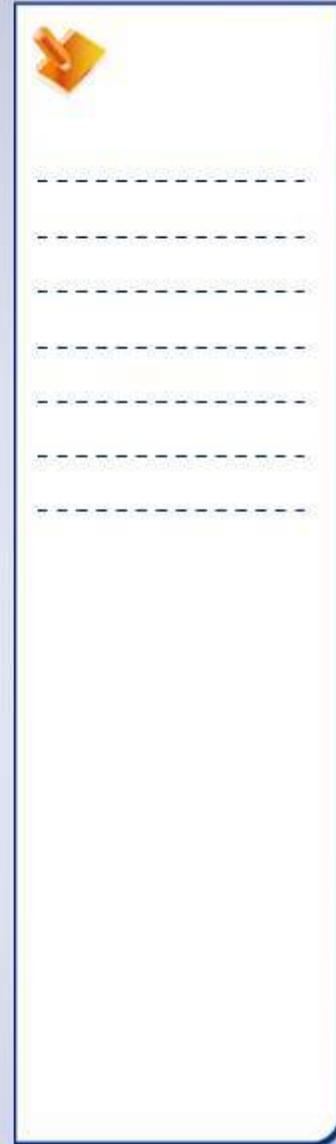
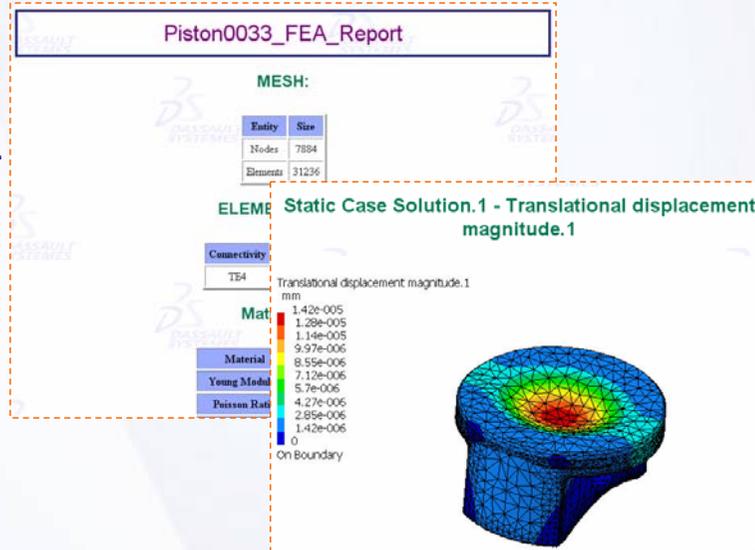
Global sensor provides an output value for the entire FE model, while local sensor provides an output value for a local region in the FE model.



## Report Generation

After the required level of solution accuracy is reached, after and several mesh refinement iterations, you can generate reports.

In this step of report generation, you will present the information generated during all stages of the FE Analysis process: Pre-processing, Computation, and Post-processing.



## Main Tools (1/2)

### View Mode

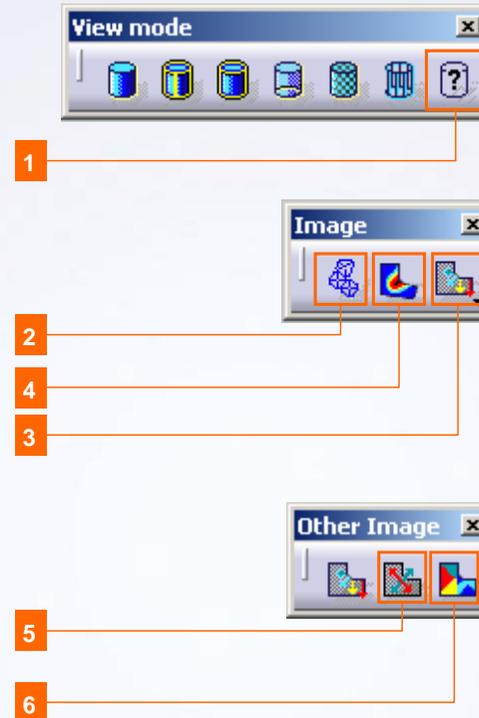
- 1 Customize View Parameters:** lets you customize the view mode.

### Image

- 2 Deformation:** lets you specify the finite element mesh in the deformed configuration under applied loading condition.
- 3 Displacement:** lets you visualize the displacement field patterns, which represent variation of position vectors of material particles as a result of applied load.
- 4 Von Mises stress:** lets you visualize the Von Mises Stress field pattern.

### Other Image

- 5 Principle stress:** lets you visualize the principle stress field patterns which represent a tensor field quantity used to measure the state of stress and to determine the load path on loaded part.
- 6 Precision:** lets you visualize the computation error maps.






---



---



---



---



---



---

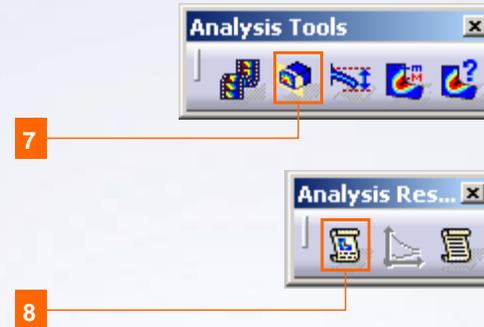
## Main Tools (2/2)

### Analysis tools

- 7 Cut Plane Analysis:** lets you to visualize sections of the structure to allow you to visualize the results within the material.

### Analysis Results

- 8 Generate Reports:** lets you to generates report for computed solutions.






---



---



---



---



---



---



---

## Exercise 4B

### Recap Exercise



15 min

*In this exercise, you will create result visualization images for 2D component.  
Detailed instructions for new topics are provided for this exercise.*

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

- Create Translational Displacement image
- Create Principal Stress Image
- Create Von Mises Stress Image
- Create Precision Image
- Create Energy and Global error Rate sensor
- Animate the displacement Image



---

---

---

---

---

---

---

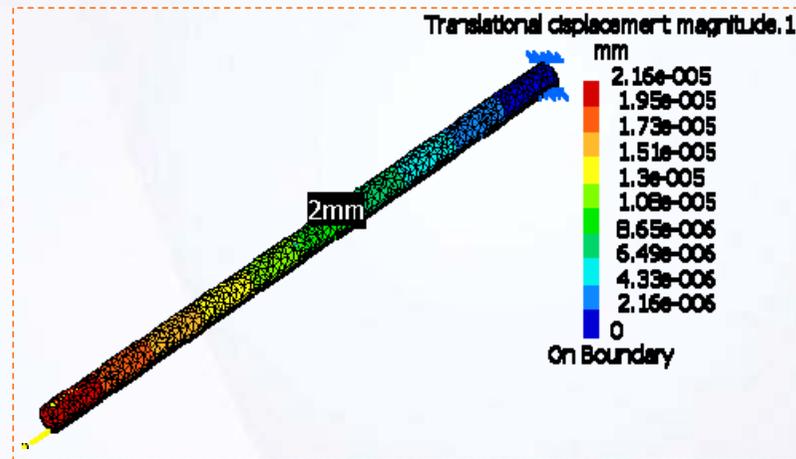
---

---

---

## Exercise 4B (1/7)

1. **Open a Analysis Document.**
  - Open 4B\_Hollow\_Shaft\_2D\_Start.CATAnalysis.
  
2. **Visualize Translational Displacement results.**
  - Create Translational Displacement Vector Image.
    - a. Select **Type** as **average iso**.
    - b. Uncheck the **Deform according to** option and observe the difference in image.

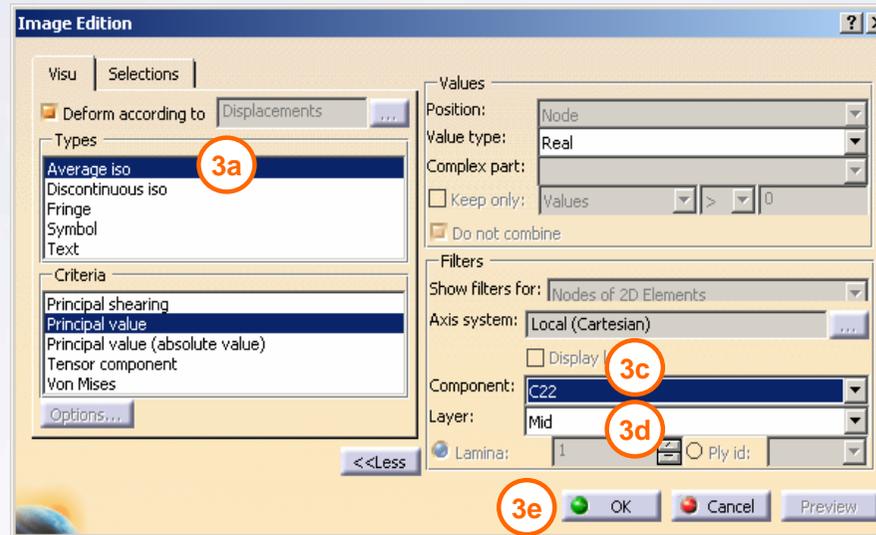


Handwritten notes area with a yellow arrow icon at the top and several horizontal dashed lines for writing.

## Exercise 4B (2/7)

### 3. Display Average Minimum Principal Stress Distribution.

- Create Principal Stress Tensor Image.
  - a. Select **Type** as **average iso** for average values.
  - b. Click **More >>** button.
  - c. In **Component** field, select **C22** for minimum principal stress.
  - d. In **Layer** field, select **Mid**.
  - e. Click **OK**.



For Stress Principal Tensors Image:

In the case of 3D elements:

C11: is the maximum principal stress

C22: is the middle principal stress

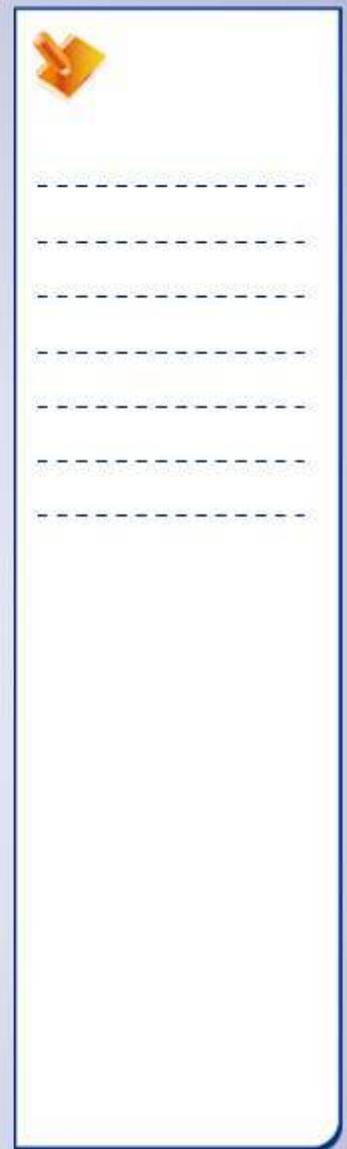
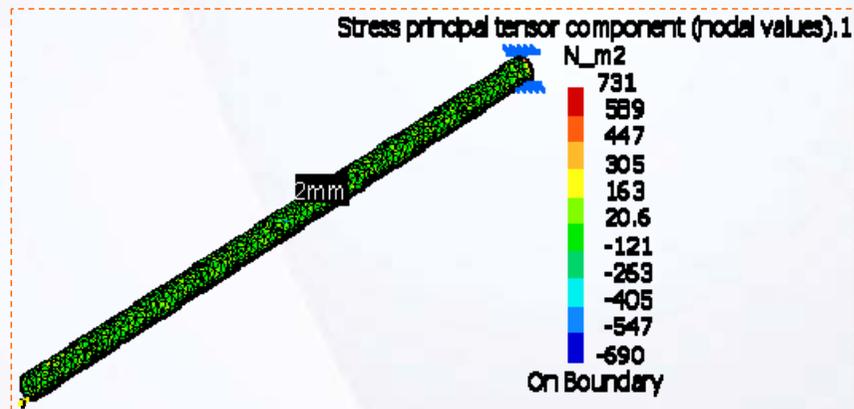
C33: is the minimum principal stress

You can also have a combination of these components (for example, C11 C22).

In the case of 2D elements:

C11: is the maximum principal stress

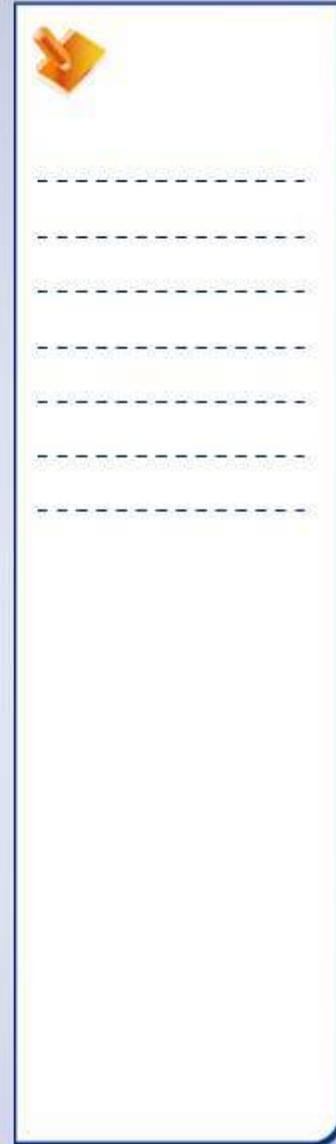
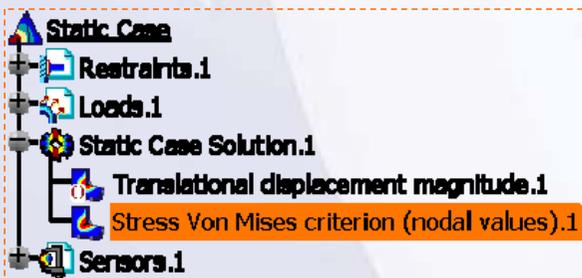
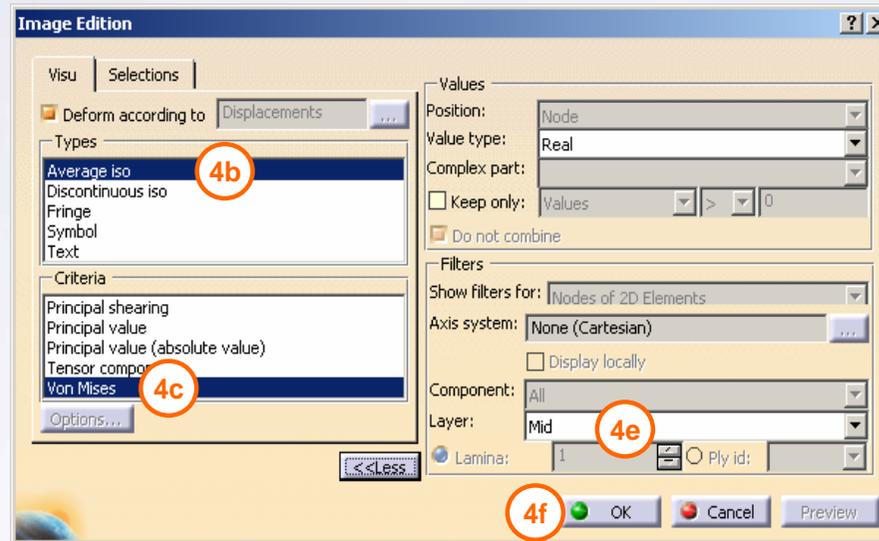
C22: is the minimum principal stress



## Exercise 4B (3/7)

### 4. Display Von Mises using Principal Stress Distribution Image

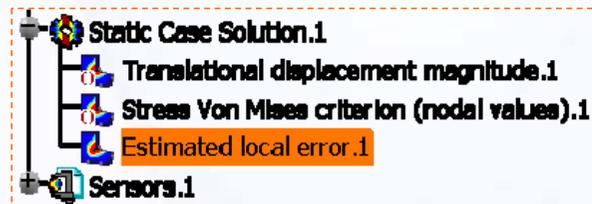
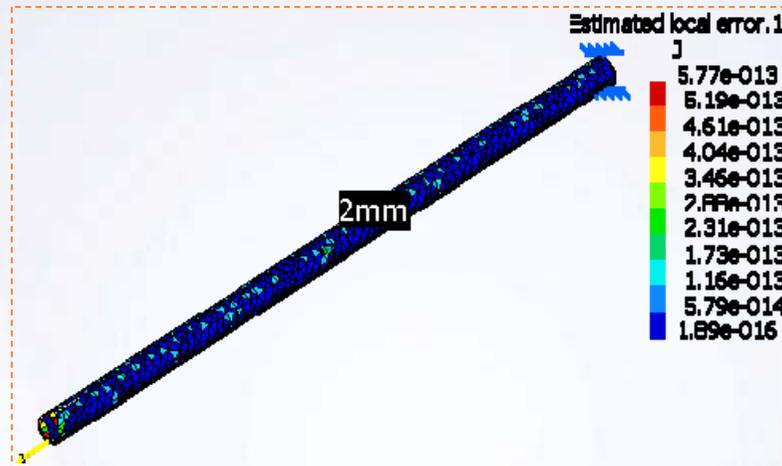
- Modify Principal Stress Tensor Image.
  - a. Double-click the *Stress principal tensor component (nodal values).1* image in specification tree.
  - b. Change the type to **Average iso** in the **Types** field.
  - c. Click on **Von Mises** in **criteria** field.
  - d. Click **More >>** button.
  - e. In **Layer** field, select **Mid**.
  - f. Click **OK**.



## Exercise 4B (4/7)

### 5. Display Local Error Distribution

- Create Precision Image.
  - a. In **Other Image** toolbar, click **Precision** icon.






---



---



---



---



---



---

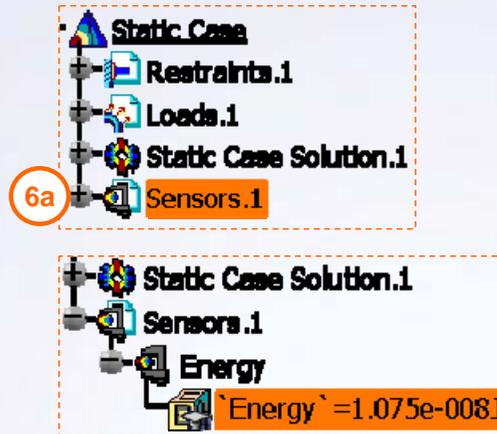


---

## Exercise 4B (5/7)

### 6. Create Sensors.

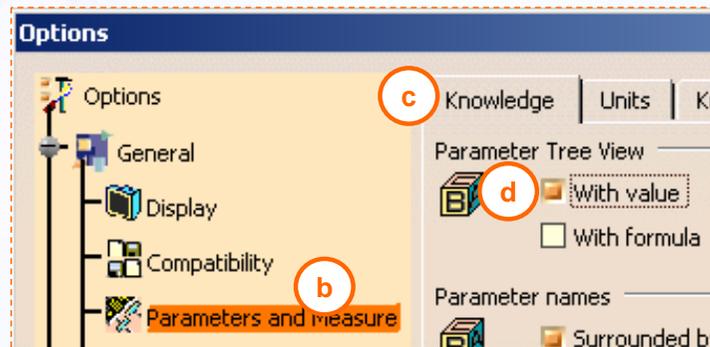
- Create Energy sensor.
  - a. Expand *Sensors.1 > Energy* node in specification tree. You will get the energy value.



The energy value calculated is the total strain energy of the model.

If this node does not display the value then:

- In Main menu, select **Tools > Options**
- Click on **Parameters and Measure** in Options specification tree.
- Click on **Knowledge** Tab.
- In Parameter Tree View, check option **With value**.
- Click **OK**.






---



---



---



---



---



---

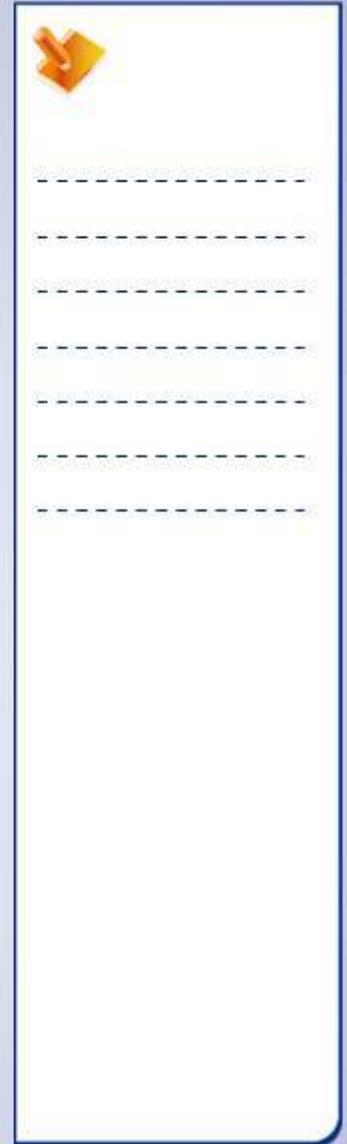
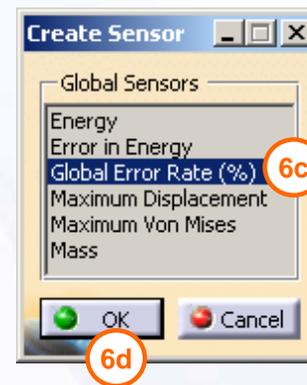
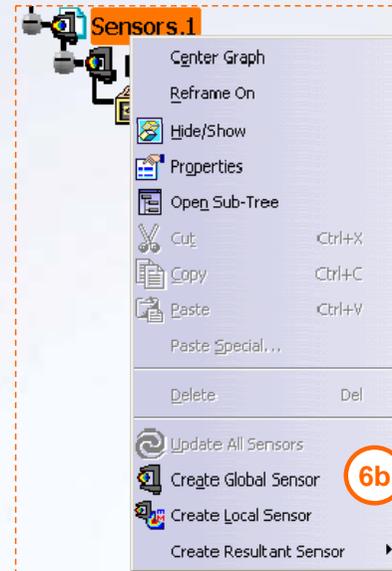
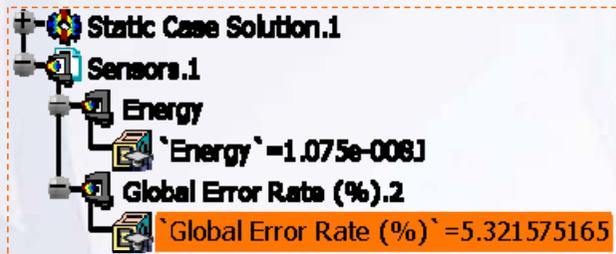


---

## Exercise 4B (6/7)

### 6. Create Sensors (continued)

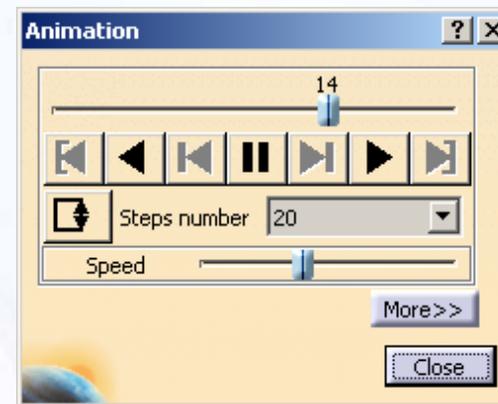
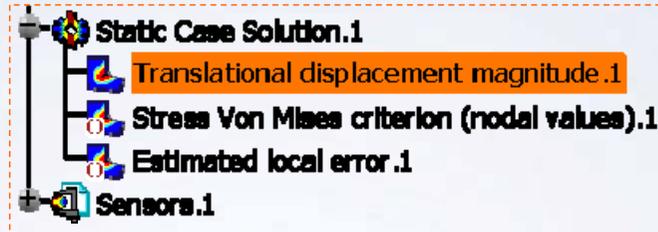
- Create global Error Rate sensor.
  - b. In *Sensors.1* contextual menu, click **Create Global Sensor**.
  - c. Select **Global Error Rate (%)** in the **Create Sensor** dialogue box.
  - d. Click **OK**.
  - e. Expand the new node *Global Error Rate (%)*.2 which get added in *Sensors.1* node in specification tree. You will get **Global Error Rate** value.



## Exercise 4B (7/7)

### 7. Visualize Animation.

- Animate the displacement image.
  - a. Deactivate *Estimated local error.1* image.
  - b. Activate the *Translational displacement magnitude.1* image.
  - c. In **Analysis Tools** toolbar, click **Animate** icon.
  - d. In **Steps number**, select **20** from dropdown.
  - e. Adjust the **Speed** button as shown.






---



---



---



---



---



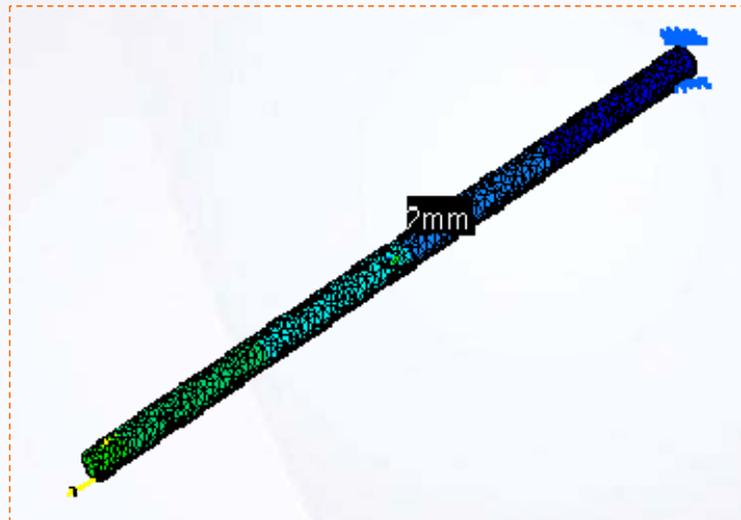
---



---

## Exercise 4B: Recap

- ✓ Create Translational Displacement image.
- ✓ Create Principal Stress Image.
- ✓ Create Von Mises Stress Image.
- ✓ Create Precision Image.
- ✓ Create Energy and Global error Rate sensor
- ✓ Animate the displacement Image



-----

-----

-----

-----

-----

-----

## Exercise 4C

### Recap Exercise



15 min

*In this exercise, you will create result visualization images for 3D component.  
Detailed instructions for new topics are provided for this exercise.*

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

- Create Translational Displacement image
- Create Von Mises Stress Image
- Create Precision Image
- Create Global error Rate sensor
- Use Cut Plane Analysis
- Find Image Extrema
- Display Image Information
- Generate Report



---

---

---

---

---

---

---

---

---

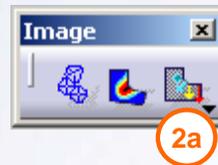
---

## Exercise 4C (1/7)

1. **Open a Analysis Document.**
  - Open 4C\_Hollow\_Shaft\_3D\_Start.CATAnalysis.

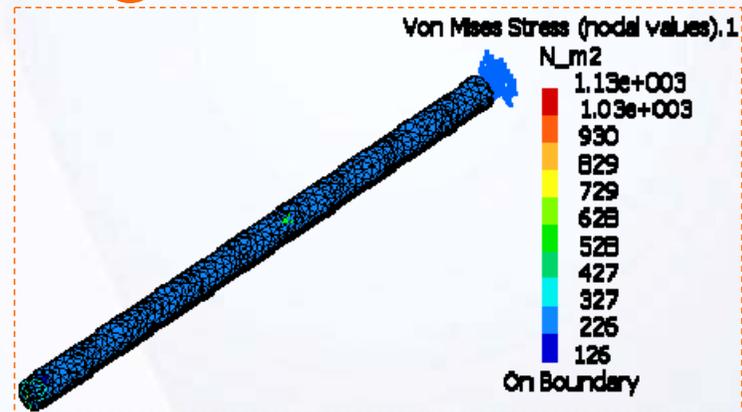
2. **Visualize Translational Displacement results.**

- Create Translational Displacement Vector Image.
  - a. Select **Type** as **average iso**.



3. **Display Von Mises Stress Distribution.**

- Create Von Mises stress Image.
  - a. In **Image** toolbar, click **Von Mises Stress** icon.






---



---



---



---



---



---

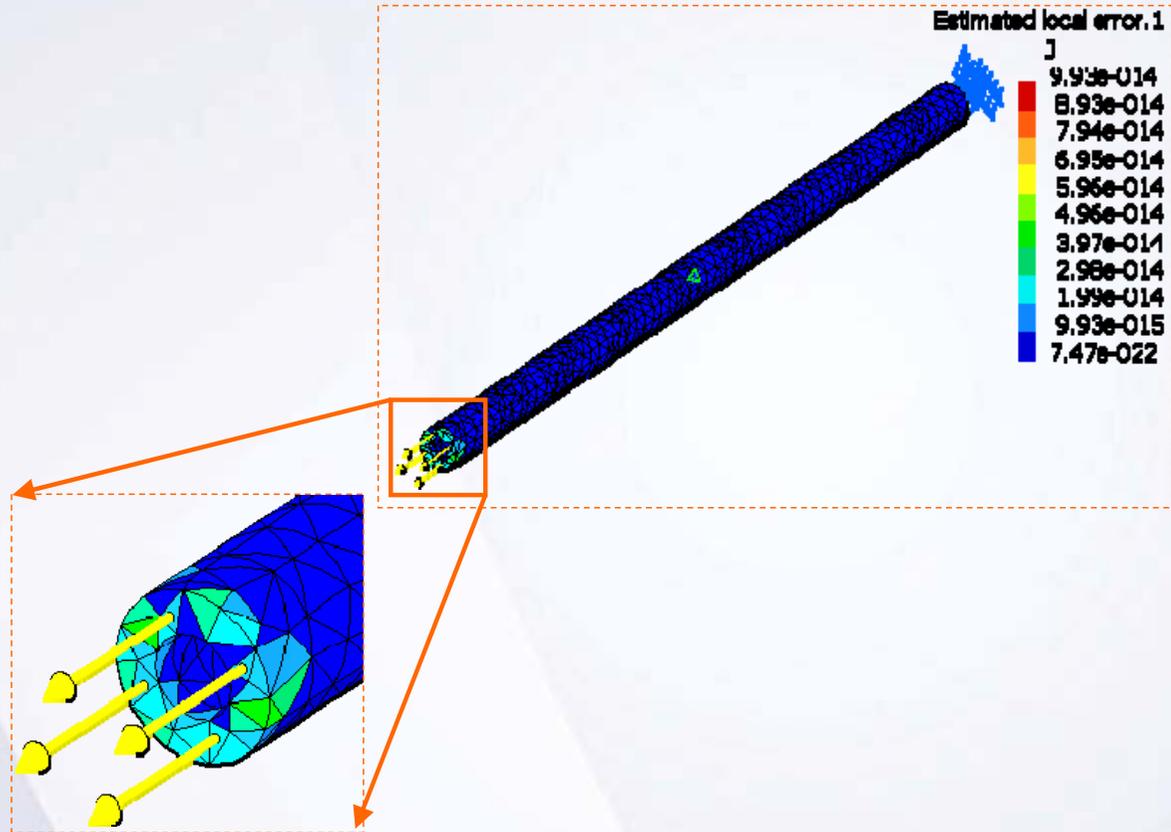
## Exercise 4C (2/7)

### 4. Display Local Error Distribution.

- Create Precision Image.
  - a. In **Other Image** toolbar, click **Precision** icon.



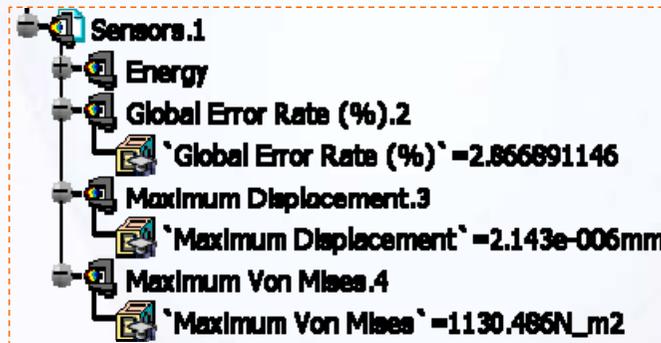
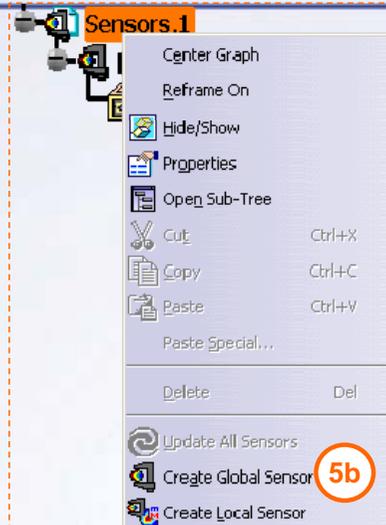
4a



## Exercise 4C (3/7)

### 5. Create Sensors.

- Create Global Error Rate sensor.
- Create Maximum Displacement sensor.
- Create Maximum Von Mises sensor.

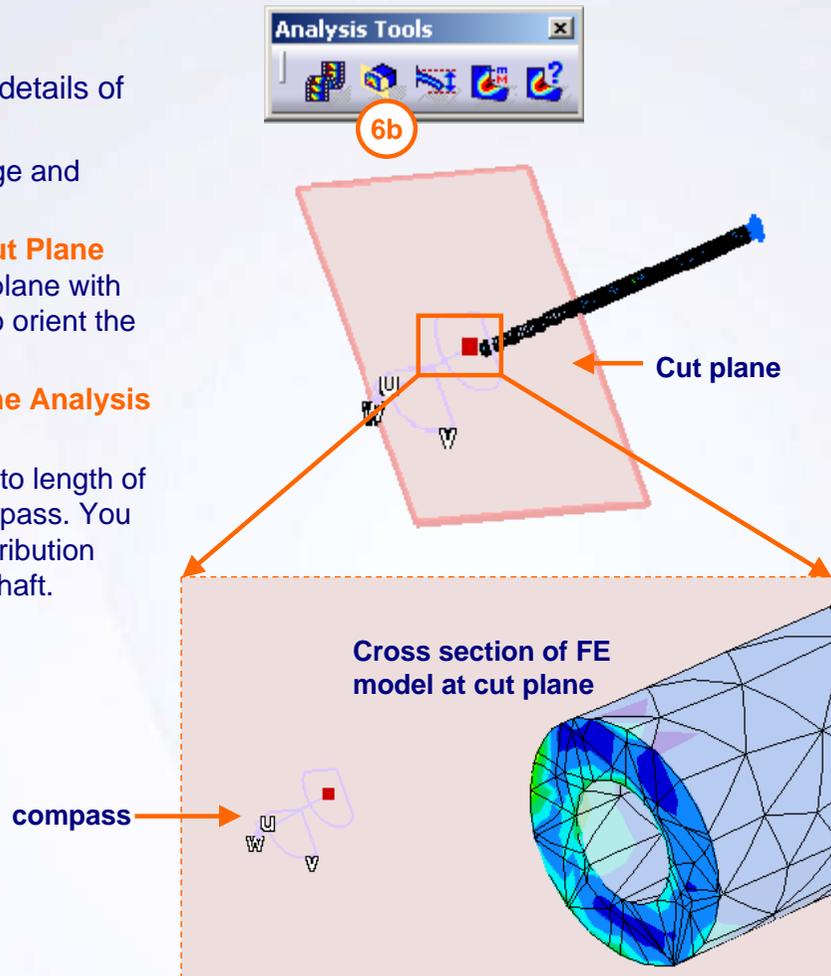
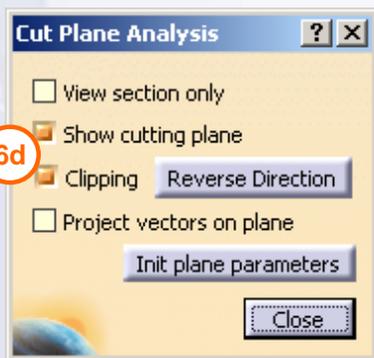


A vertical rectangular box containing a yellow arrow icon at the top and several horizontal dashed lines for writing.

## Exercise 4C (4/7)

### 6. Cut Plane Analysis.

- Use cut plane analysis to view inside details of FE model.
  - a. Activate the Von Mises stress image and deactivate other images.
  - b. In **Analysis Tools** toolbar, click **Cut Plane analysis** icon. It will display a cut plane with compass. You can use compass to orient the cut plane.
  - c. Use the options shown in **Cut Plane Analysis** dialogue box.
  - d. Orient the cut plane perpendicular to length of hollow shaft using mouse and compass. You can view the Von Mises stress distribution along the cross section of hollow shaft.




---



---



---



---



---



---



---

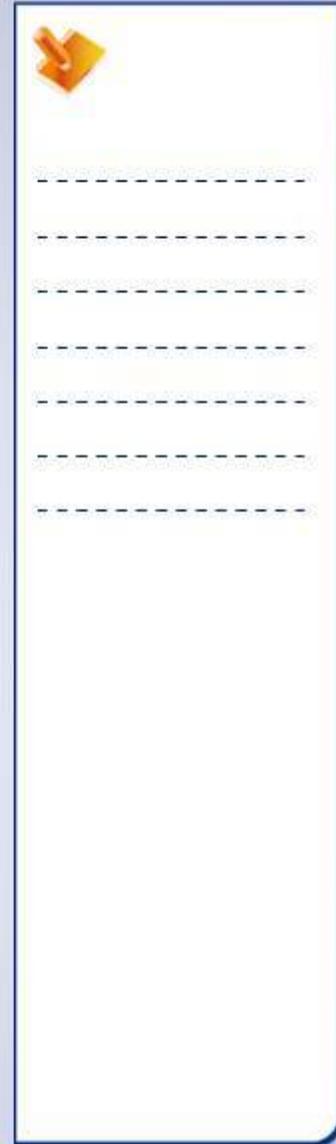
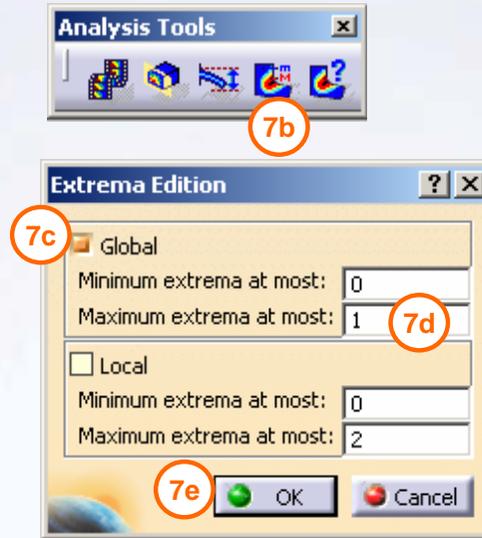
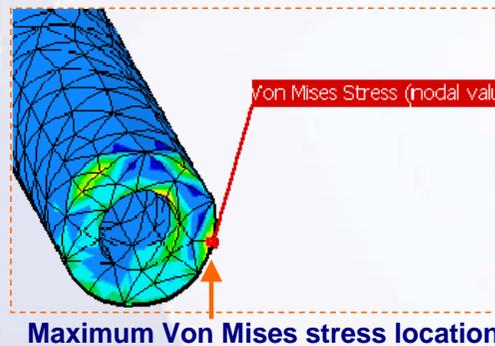
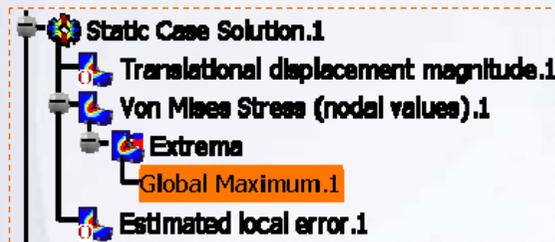


---

## Exercise 4C (5/7)

### 7. Find Maximum Von Mises stress value and location.

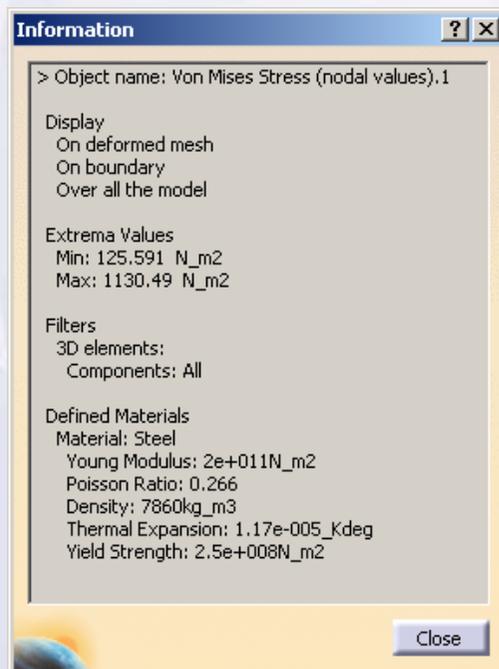
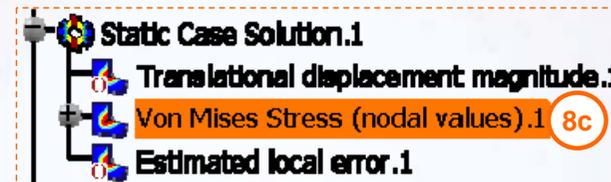
- Find Image extrema for Von Mises stress.
  - a. Activate the Von Mises stress image and deactivate other images.
  - b. In **Analysis Tools** toolbar, click **Image Extrema** icon.
  - c. Check the **Global** option.
  - d. In **Maximum extrema at most** field, enter 1.
  - e. Click **OK**.



## Exercise 4C (6/7)

### 8. Display Image information.

- Find Image Information for Von Mises stress.
  - a. Activate the Von Mises stress image and deactivate other images.
  - b. In **Analysis Tools** toolbar, click **Information** icon.
  - c. Click the *Von Mises Stress (nodal values).1* option. You will get **Information** dialogue box.






---



---



---



---



---



---

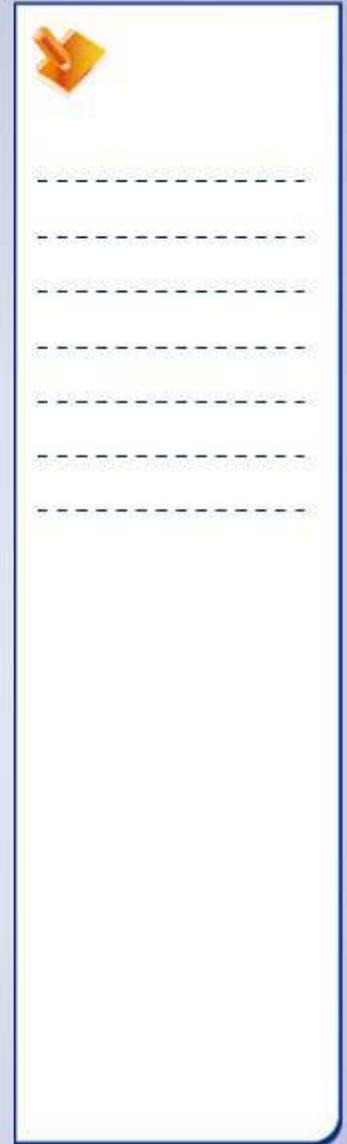
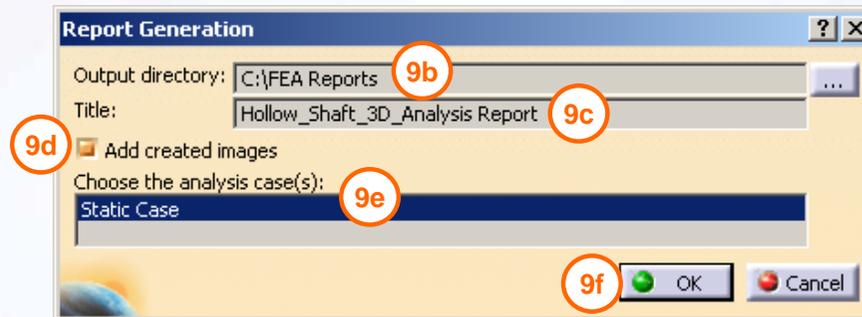


---

## Exercise 4C (7/7)

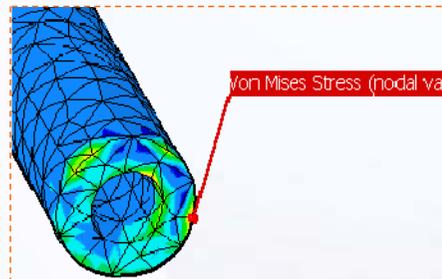
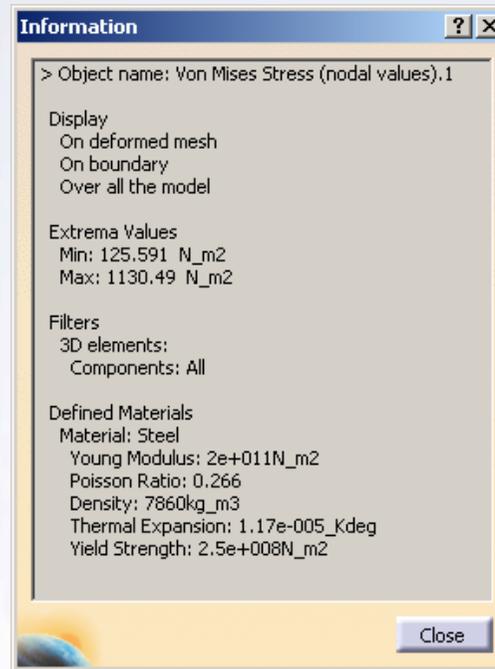
### 9. Create Analysis Report.

- Generate Report for Analysis Case.
  - a. In **Analysis Results** toolbar, click **Generate Report** icon.
  - b. Select the desired folder in the **Output directory** field.
  - c. In **Title** field write the title for the report.
  - d. Check **Add created images** option. This will add all created result visualization images in report.
  - e. In **Choose the analysis cases(s)**, you can choose the analysis cases to be included in report. This is applicable in case you have multiple analysis cases.
  - f. Click **OK**.



## Exercise 4C: Recap

- ✓ Create Translational Displacement image.
- ✓ Create Von Mises Stress Image.
- ✓ Create Precision Image.
- ✓ Create Global error Rate sensor.
- ✓ Use Cut Plane Analysis.
- ✓ Find Image Extrema.
- ✓ Display Image Information.
- ✓ Generate Report.






---



---



---



---



---



---



---

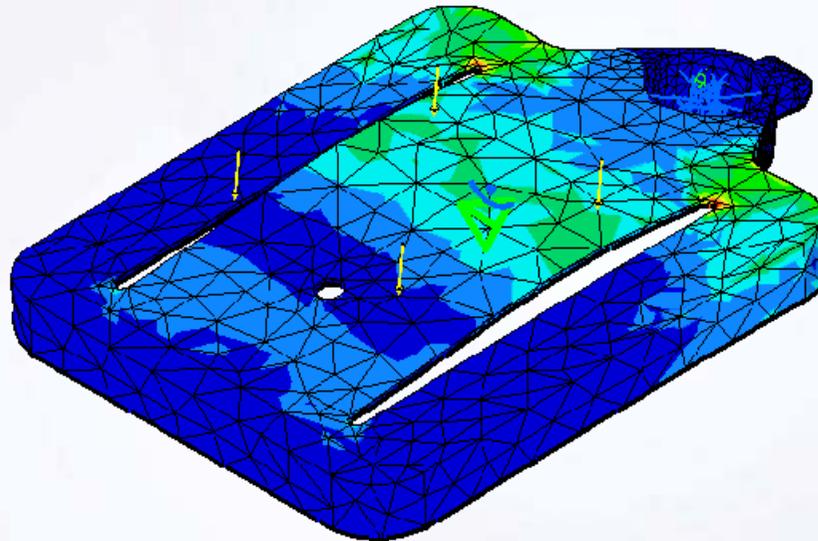
# Case Study: Drill Press Table Post-processing

## Recap Exercise



**You will practice what you learned, by completing the case study model.  
In this exercise, you will post process FE model for Drill Press Table.  
Recall the design intent of this model:**

- ✓ Create required Visualization Images.
- ✓ Create Extrema on created images.
- ✓ Create sensors.
- ✓ Generate analysis report.






---



---



---



---



---



---

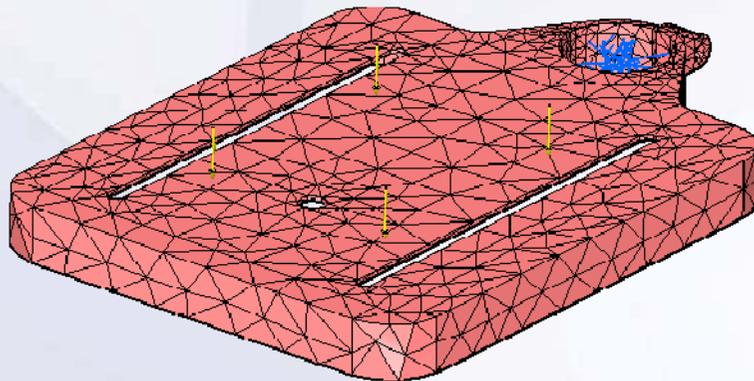


---

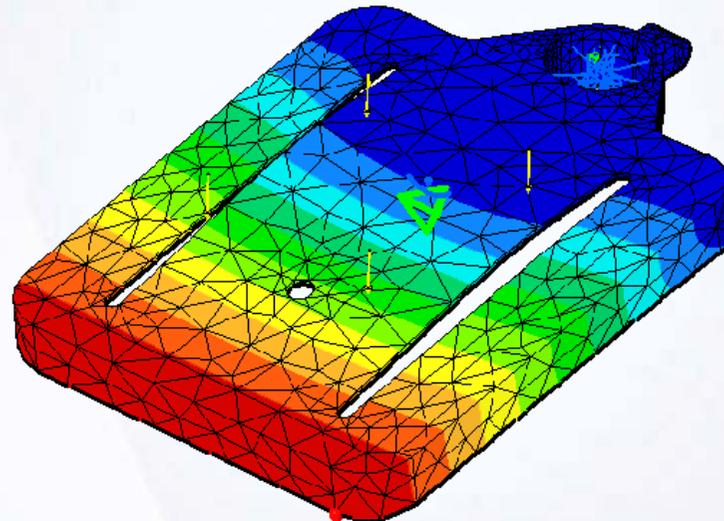
## Case Study: Introduction

The case study for this lesson is the Drill Press Table FE model post-processing. The focus of this case study is to post-process the computed data as per requirement. You will perform the following steps in order to achieve this.

1. Create required Visualization Images.
2. Create Extrema on created images.
3. Create sensors.
4. Generate analysis report.



Drill press Table FE Model



Drill press Table Translational Displacement Image




---



---



---



---



---



---

## Design Intent (1/2)

- ✓ You will create Translational Displacement Magnitude plot. You will find the magnitude of maximum average displacement. Also find the location of maximum value of translational displacement. Also observe the Z-direction component image for translational displacement.
  - Create Translational Displacement Magnitude image and create Global Extrema for this image.
  - Create Translational Displacement Component with Z-direction component image and create Global Extrema for this image.
  
- ✓ You will create plots for Maximum Principal stress and Minimum Principal stress distribution with discontinuous iso and find out the location of maximum values.
  - Create Principle Stress image with Maximum Principal Stress component and create Global Extrema for this image.
  - Create Principal Stress image with Minimum Principal Stress component and create Global Extrema for this image.



-----

-----

-----

-----

-----

-----

## Design Intent (2/2)

- ✓ You will create Von Mises stress distribution plot and its location of maximum value in FE model.
  - Create Von Mises stress image and create Global Extrema for this image.
- ✓ You will create Local error distribution and its location of maximum value.
  - Create Precision image and create Global Extrema for this image.
- ✓ You will create output sensors for Error in Energy, Global Error Rate percentage.
- ✓ You will create Resultant sensor to know reactions at restraints.
  - Create Resultant sensor in reaction sensor.
- ✓ You will create Analysis Report with Images.




---



---



---



---



---



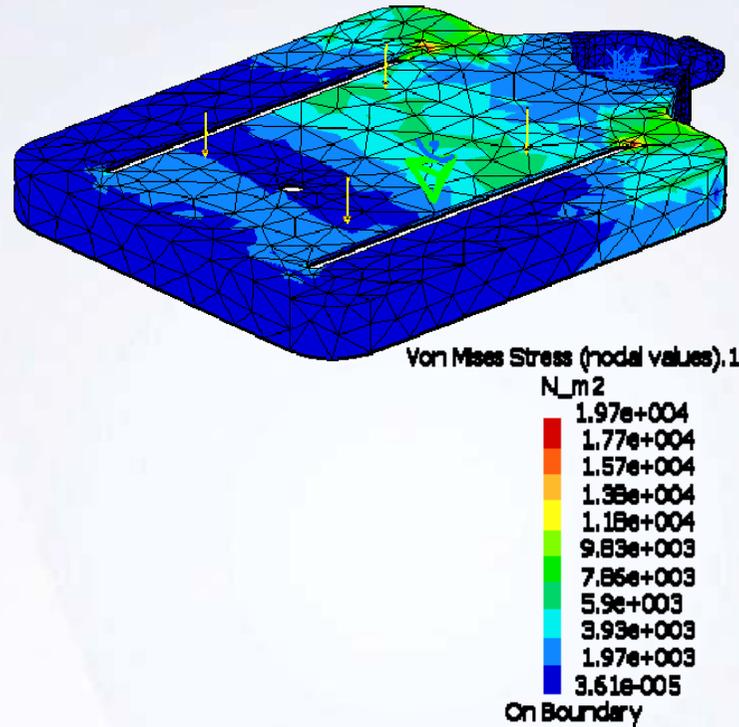
---



---

## Do It Yourself

1. Open **Drill\_Press\_Table\_Post\_Processing\_Start.CATAnalysis.**
2. Create required Visualization Images.
3. Create Extrema on created images.
4. Create sensors.
5. Generate analysis report.



Drill press Table Von Mises Stress Image




---



---



---



---



---



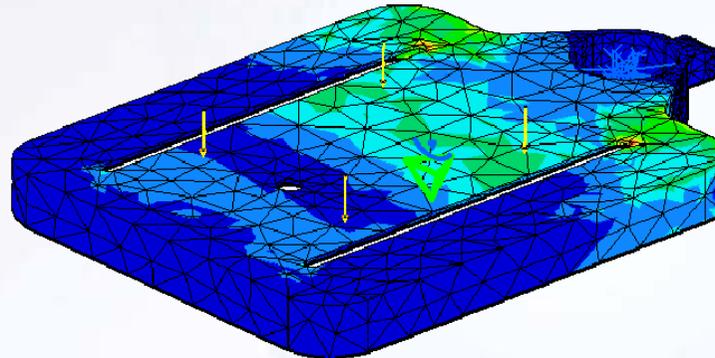
---



---

## Case Study: Drill Press Table Post-processing Recap

- ✓ Create required Visualization Images.
- ✓ Create Extrema on created images.
- ✓ Create sensors.
- ✓ Generate analysis report.



-----

-----

-----

-----

-----

-----



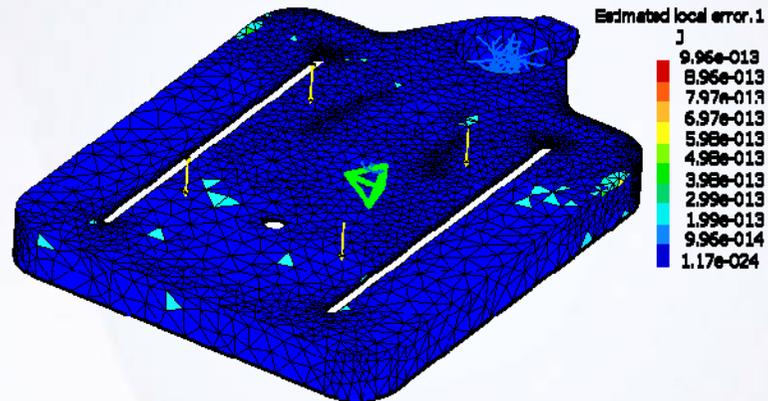
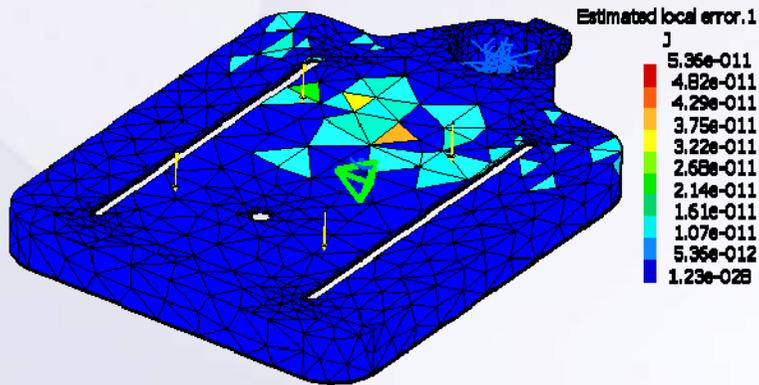
# Mesh Refinement

5

**Learning Objectives:**

*Upon completion of this lesson you will be able to:*

- ✓ Understand Mesh Refinement.
- ✓ Perform adaptive mesh refinement.






---



---



---



---



---



---



---

## Case Study

The case study for this lesson is the Drill Press Table FE model Post-processing. The case study focuses on mesh refinement in order to obtain required visualization images.

## Design Intent

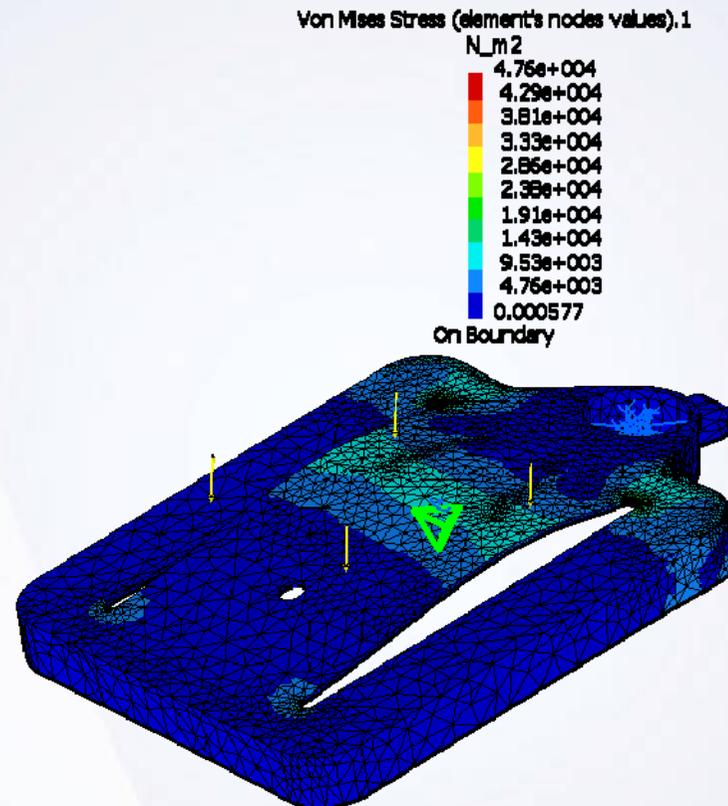
The analysis must be performed with the provided mesh and adaptivity parameters:

- ✓ You will solve the FE Model with initial mesh
- ✓ You will study the following results
  - Local error distribution
  - % global error rate
  - Compare the difference between values of Von Mises stress with Discontinuous iso and average iso.
- ✓ You will specify the Global Adaptivity parameters.
- ✓ You will compute with Adaptivity
- ✓ You will again study the results.

## Stages in the Process

The following steps are used to perform the case study:

1. Solve the model with initial mesh.
2. Specify Global Adaptivity parameters.
3. Compute with Adaptivity.
4. Check the results.






---



---



---



---



---



---



---



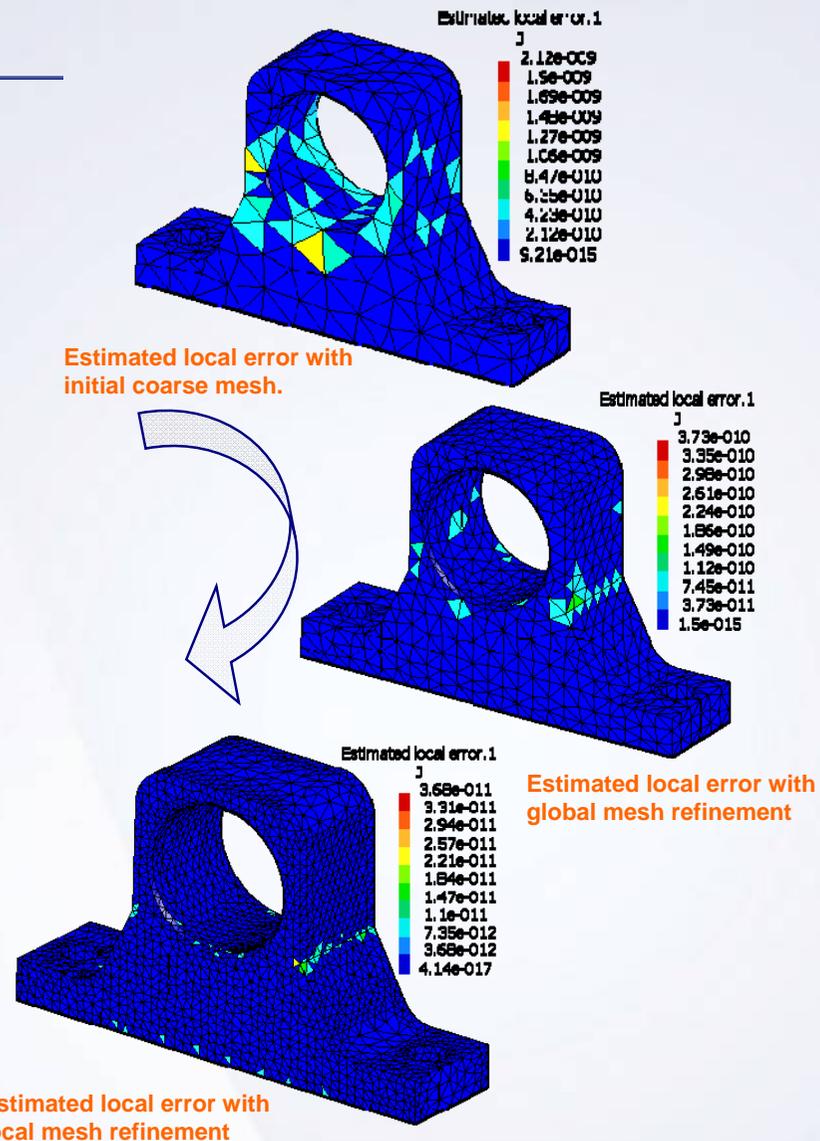
---

## What is Mesh Refinement

Initial FE solution provides you the results that are generally obtained using simple element types and coarse meshing. From this solution, you can get the rough idea of the results using simple FE modeling which requires minimum computation time. These results may not be sufficiently accurate. In the process of mesh refinement you will increase the accuracy of the solution, by reducing the discretization error.

Global mesh refinement involves decreasing the mesh size and sag for the whole FE model. Successive global mesh refinement may not lead to significant change in the stress values in the major areas of the component; however it will increase the CPU computation time. So using precision plots you can spot the regions of high local error values, and refine the mesh in those regions locally.

The 'Global Error Rate' sensor indicates overall accuracy of the solution. In each mesh refinement step the 'Global Error Rate' decreases.



Handwritten notes area with a yellow sticky note icon at the top and several horizontal dashed lines for writing.

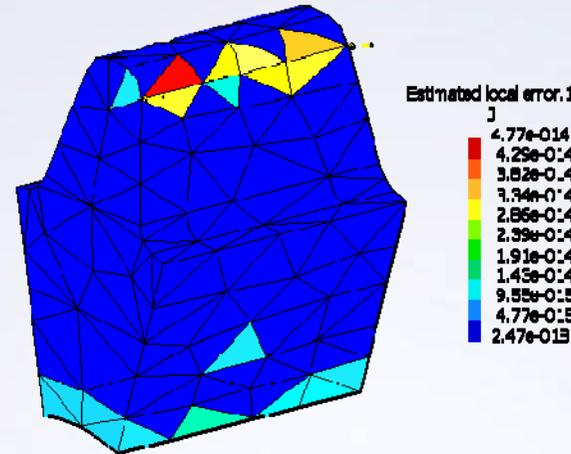
## Adaptive Mesh Refinement

In Adaptive Mesh Refinement, the mesh is refined by increasing or decreasing the element sizes. The order of the element remains the same. The mesh is refined in the areas where local error estimates are high.

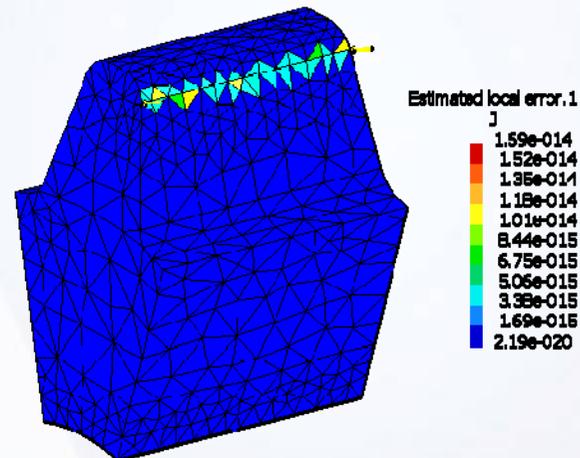
In general following steps are involved in Computing the mesh using Adaptive Mesh Refinement technique.

1. Solve the FE model with initial mesh.
2. Specify the Adaptivity parameters.
3. Compute with Adaptivity.
4. Check the results.

After refinement, the difference between stress values with Average iso and Discontinuous iso decreases, as the solution converges.



Mesh to be refined using Adaptive mesh refinement



You will observe that the areas of high Local error in previous mesh are refined with a fine mesh.




---



---



---



---



---



---



---

## Main Tools

### Adaptive Mesh Refinement

- 1 New Adaptivity Entity:** Lets you create the adaptivity on a mesh part for a static analysis case solution.
- 2 Compute with Adaptivity:** Lets you compute the solution with adaptivity.



A vertical sidebar on the right side of the page, containing a yellow arrow icon at the top and several horizontal dashed lines below it, serving as a workspace for notes.

## Case Study: Drill Press Table Mesh Refinement

### Recap Exercise



30 min

***You will practice what you learned, by completing the case study model. In this exercise, you will how to use adaptive mesh refinement to improve analysis results for Drill Press Table FE model. Recall the design intent of this model:***

- ✓ Solve the model with initial mesh
- ✓ Specify Global Adaptivity parameters
- ✓ Compute with Adaptivity
- ✓ Check the results



---

---

---

---

---

---

---

---

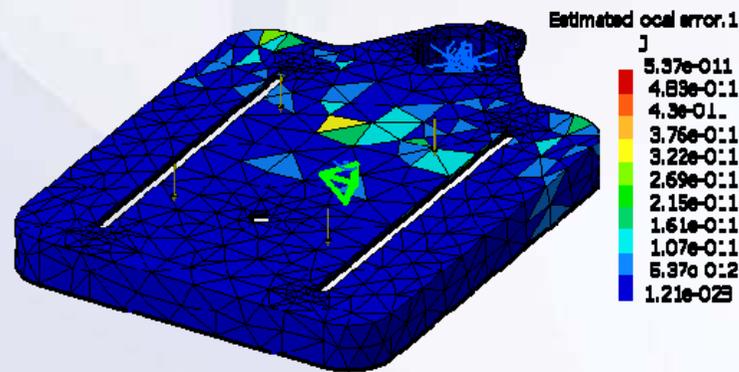
---

---

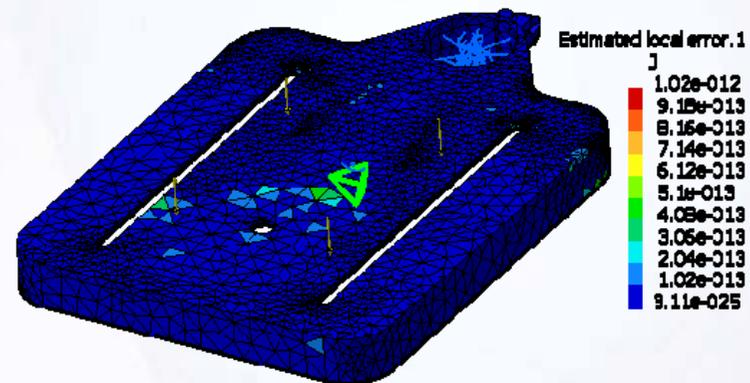
## Case Study: Introduction

The case study for this lesson is the Drill Press Table FE Model Mesh Refinement. The focus of this case study is to perform mesh refinement using adaptive mesh refinement. You will perform following steps in order to achieve this.

1. Solve the model with initial mesh.
2. Specify Global Adaptivity parameters.
3. Compute with Adaptivity.
4. Check the results.



Drill press Table FE Model Local error distribution for initial mesh



Drill press Table FE Model Local error distribution for final mesh

## Design Intent (1/2)

- ✓ You will solve the initial FE model with mesh size 33 mm, Absolute sag 3 mm and parabolic element type.
- ✓ You will study the following results.
  - View the local error distribution. Note the maximum local error value.
  - View the % global error rate. Note the % global error rate by creating the global error sensor.
  - Compare the difference in values of maximum Von Mises stress with Discontinuous iso and maximum Von Mises stress with average iso.
- ✓ Specify the Global Adaptivity parameters
  - Select the mesh part to be refined.
  - Define the required % global error rate as % objective error. Specify % objective error as 12%.




---

---

---

---

---

---

---

---

## Design Intent (2/2)

- ✓ Compute with Adaptivity.
  - Specify the iterations to be performed. Specify number iterations as 30.
  - Keep Global mesh size unmodified. Uncheck allow mesh Unrefinement.
  - Allow Global sags to modify. Check Deacitvate global sags.
  
- ✓ You will again study the following results.
  - View the local error distribution. Note the maximum local error value.
  - View the % global error rate. Note the % global error rate by creating the global error sensor.
  - Compare the difference in values of maximum Von Mises stress with Discontinuous iso and maximum Von Mises stress with average iso.

The above two values should be approximately same in order to achieve convergence.

**In case the Objective error is not reached, increase the number of iterations and again compute with Adaptivity  
More the number of iterations more memory is required for computation. Please make sure that you have enough disk space compute the solution.**



-----

-----

-----

-----

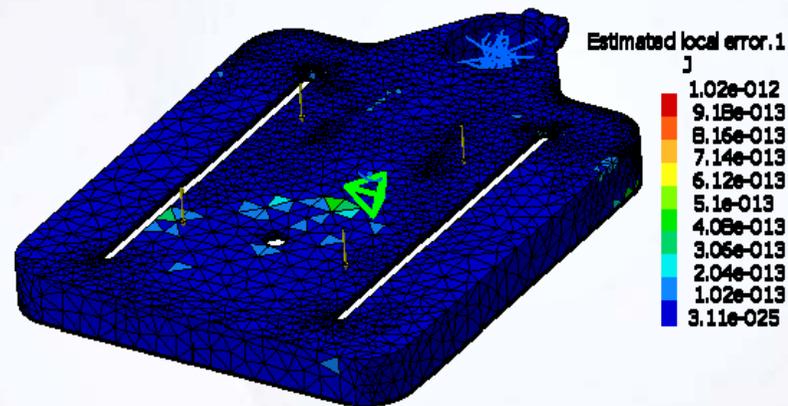
-----

-----

## Do It Yourself

The following steps offer hints to guide you through the mesh refinement of FE model for Drill Press Table.

1. **Open an Analysis Document**  
Drill\_Press\_Table\_Mesh\_Refinement\_Start.CATAnalysis.
  
2. **Solve the model with initial mesh.**
  - Use mesh size 33 mm, Absolute sag 3 mm and parabolic element type.
  
3. **Specify Global Adaptivity parameters.**
  
4. **Compute with Adaptivity.**
  
5. **Check the results.**






---



---



---



---



---



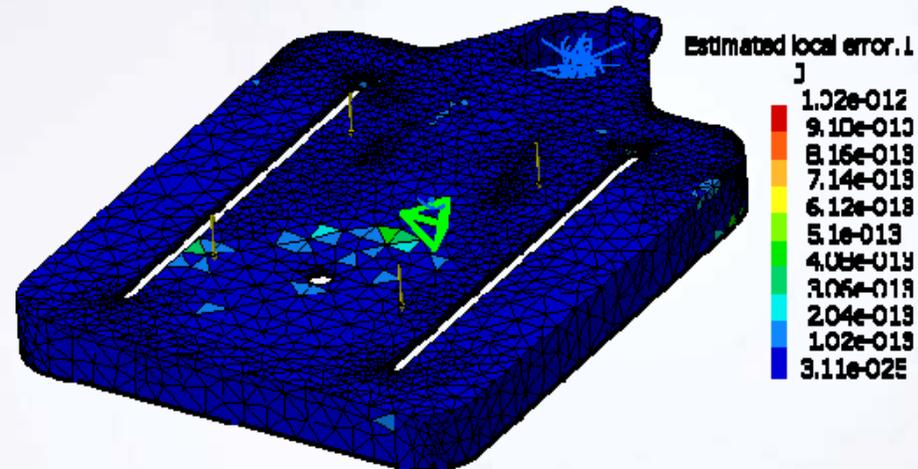
---



---

## Case Study: Drill Press Table Mesh Refinement Recap

- ✓ Solve the model with initial mesh.
- ✓ Specify Global Adaptivity parameters.
- ✓ Compute with Adaptivity.
- ✓ Check the results.



Handwritten notes area with a yellow arrow icon at the top and several horizontal dashed lines for writing.



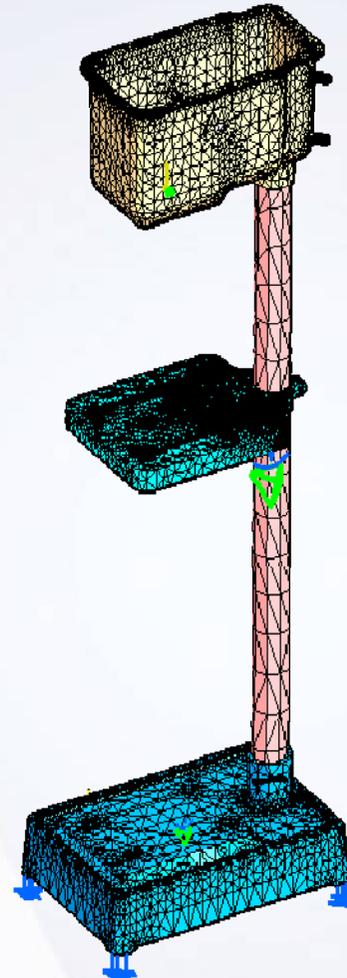
# Assembly Structural Analysis

6

**Learning Objectives:**

*Upon completion of this lesson you will be able to:*

- ✓ Understand Assembly Structural Analysis.
- ✓ Understand about Analysis Assembly.
- ✓ Learn about different analysis connections and connections properties.
- ✓ Understand the use of virtual parts and distributed mass.



2 hours




---



---



---



---



---



---

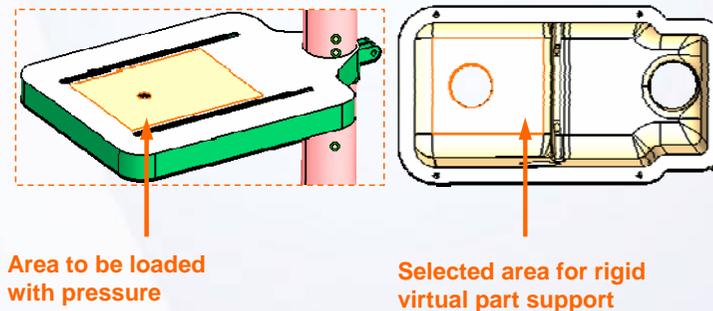
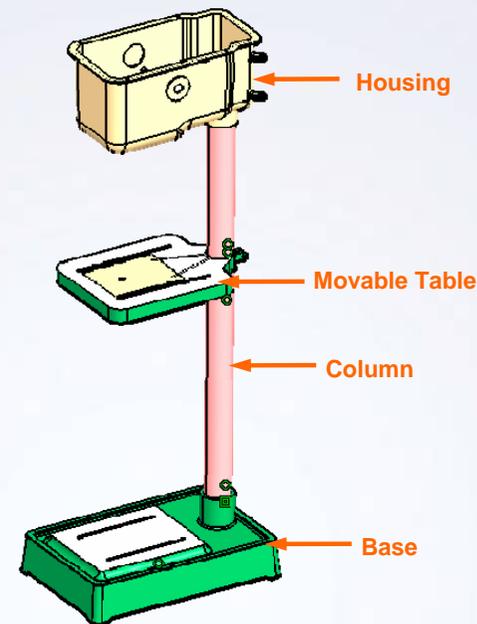
## Case Study

The case study for this lesson is the Analysis Assembly of Drill Press Sub-Assembly. The focus of this case study is to create FE model of the Drill Press Sub-Assembly using Analysis Assembly approach.

## Design Intent

The analysis must be performed with the following parameters and component connections:

- ✓ The Component level analysis documents for Table and Housing will be reused in this assembly.
- ✓ For other components the automatically applied mesh parameters and properties will be applied.
- ✓ The components will be connected as follows:
  - A rigid connection between the Base and Column.
  - A fastened connection between the Column and Table.
  - A rigid connection between the Column and Housing.
- ✓ The Table is to be loaded with a uniform pressure of 1N per square meter.
- ✓ A Rigid Virtual part will be used to represent a sub-assembly mass load of 10 Kg on a limited area of the housing lower surface.
- ✓ The bottom of the base must be fixed.
- ✓ The FE model will be checked and computed.
- ✓ The results must be viewed.






---



---



---



---



---

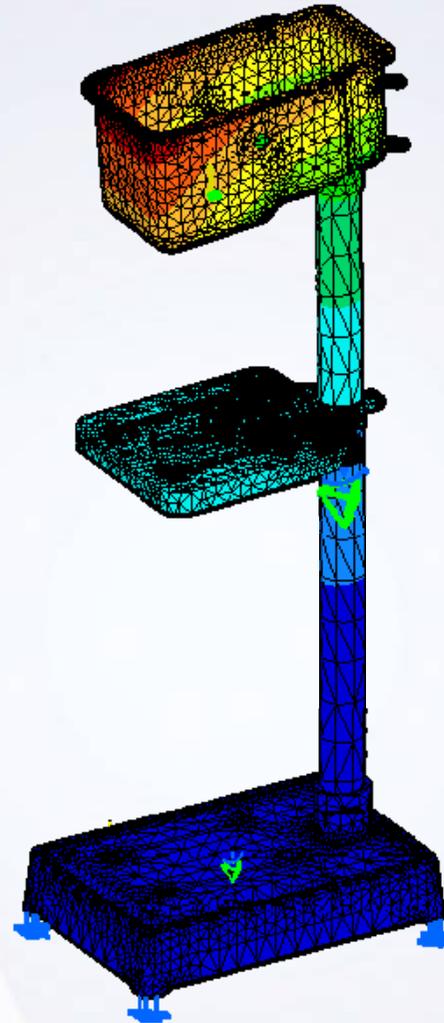


---

## Stages in the Process

The following steps are used to perform the case study:

1. Learn different Assembly Structural Analysis approaches.
2. Create Assembly connections.
3. Apply Connection properties.
4. Create a Virtual Part.
5. Create a Distributed Mass.



Handwritten notes area with a yellow arrow icon at the top left and several horizontal dashed lines for writing.

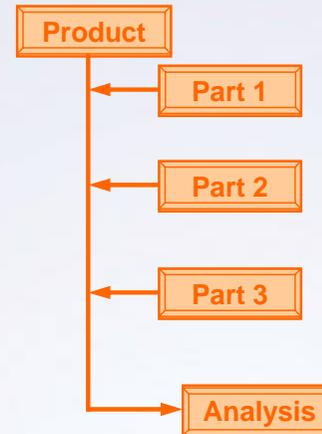
## Assembly Structural Analysis

Assembly Structural Analysis is generally used to perform the structural analysis of assemblies, by modeling the physical assemblies using Finite Element Assemblies.

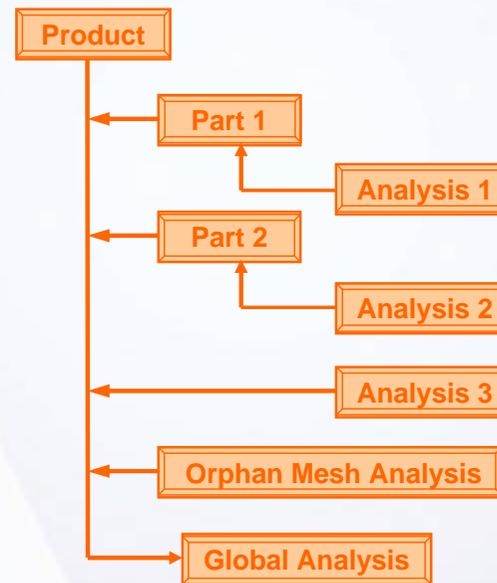
To simulate the real connection behavior you can define connections between different assembly components and assign different types of connection properties to these connections, using GAS license through Generative Structural Analysis workbench. You can also use 'assembly constraints' as a support to assign the connection properties.

In the Assembly Analysis approach, the product document is available. You need to mesh and apply the properties. Assign the connection properties between the components. Finally the analysis of the complete assembly is performed.

In the Analysis Assembly approach, the product document may or may not be available. However you have individual component analysis documents. Attach the component analysis document to the corresponding component in the product. The component mesh parameters and Properties information is reused from the component analysis documents for the global analysis. Finally analysis of the complete assembly is performed.



Assembly Analysis Approach



Analysis Assembly Approach




---



---



---



---



---



---

## Analysis Assembly

In this approach, analysis of individual parts in a product which are already done, are assembled to form an 'Analysis Assembly' and then a final 'Global Analysis' is performed for the product.

In case a product is not available, you can create a product document and directly attach individual part analysis files in the product document. You can also attach orphan mesh analyses to the product.

The analysis document which is alternate shape representation of the part document, is attached to the part using 'Manage Shape Representation' option.

With this approach of Analysis Assembly it is possible to mesh individual parts in an assembly simultaneously by different users at different locations.

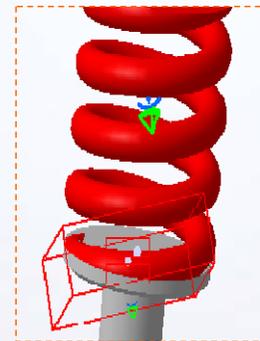


## Analysis Connections and Connection Properties

In the finite element model, GAS connections are an easy way to define interactions between the parts without modeling all the physical details of the joint.

General Analysis connection is used for connecting parts of an assembly with or without handler points. You can create different sets of analysis connections as per requirement.

You need to choose the proper connection properties to simulate the real behavior of the joint.






---



---



---



---



---



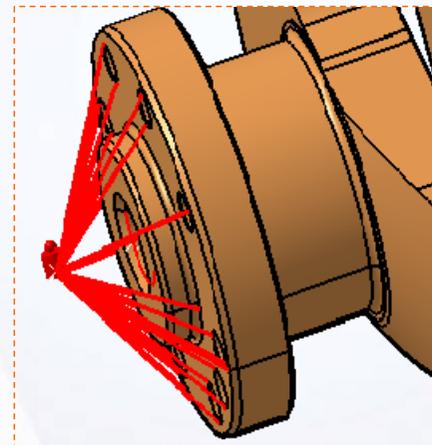
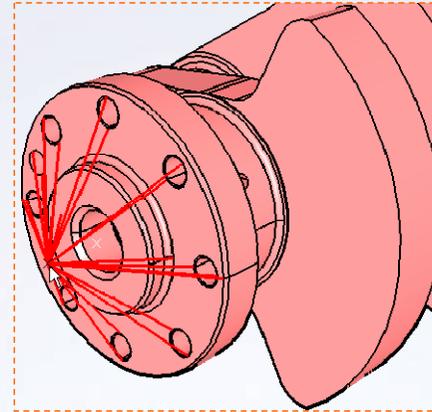
---

## Virtual Parts

Virtual parts represent bodies for which no geometry model is available, but which play an important role in the analysis of the single part or assembly systems. Virtual parts are connected to components and are used to transmit the displacement at a distance.

Following are the different types of virtual parts:

- ✓ Rigid Virtual Part
- ✓ Smooth Virtual Part.
- ✓ Contact Virtual Part
- ✓ Rigid Spring Virtual Part
- ✓ Smooth Spring Virtual Part



## Distributed Mass

The Distributed Mass is used to represent purely inertial systems. The distributed mass is the point mass equivalent to the total mass concentrated at a given point.

Generally it is distributed on a virtual part or on a geometric entity. For virtual parts, the total mass is concentrated at the handler point of the virtual part.




---



---



---



---



---



---

## Main Tools (1/2)

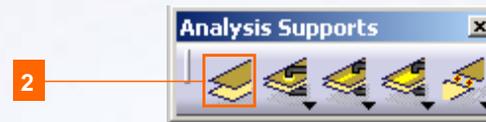
### Analysis Assembly

- 1 Analysis Assembly 2D Viewer:** enables you to add or remove a shape, activate or deactivate an existing shape, and add or remove a product component in Analysis Assembly.



### Analysis Supports

- 2 General Analysis Connection:** connects two or more parts of an assembly, with or without a handler point.



### Face Face Connection Properties

- 3 Fastened Connection Property:** creates a link between two bodies which are fastened together at their common boundary.
- 4 Contact Connection Property:** creates link between two part bodies which are prevented from inter-penetrating at their common boundary. The two bodies are free to move arbitrarily relative to each other as long as they do not come in contact.






---



---



---



---



---



---



---

## Main Tools (2/2)

### Face Face Connection Properties

- 5 Rigid Connection Property:** creates link between two bodies which are stiffened and fastened together at their common boundary, and will behave as if their interface is infinitely rigid.

### Distant Connection Properties

- 6 Bolt-tightening Connection Property:** creates connection that takes into account pre-tension in a bolted assembly.






---



---



---



---



---



---



---

# Exercise 6C

## Recap Exercise

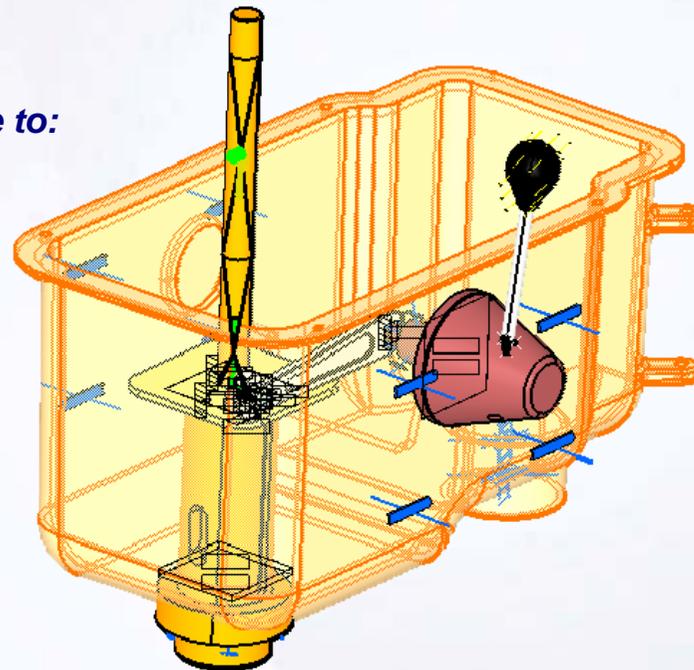


15 min

*In this exercise, you will use Virtual Part and Additional Mass features. Detailed instructions for new topics are provided for this exercise.*

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

- Create Virtual Parts
- Create Additional Mass






---



---



---



---



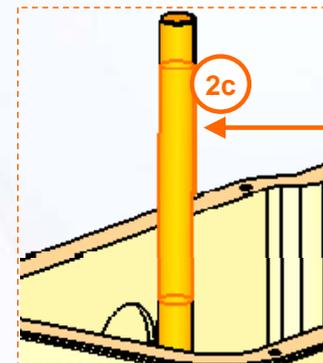
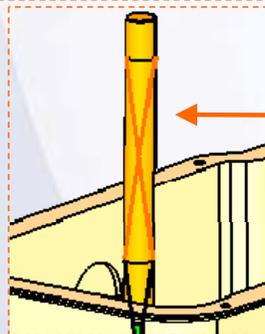
---



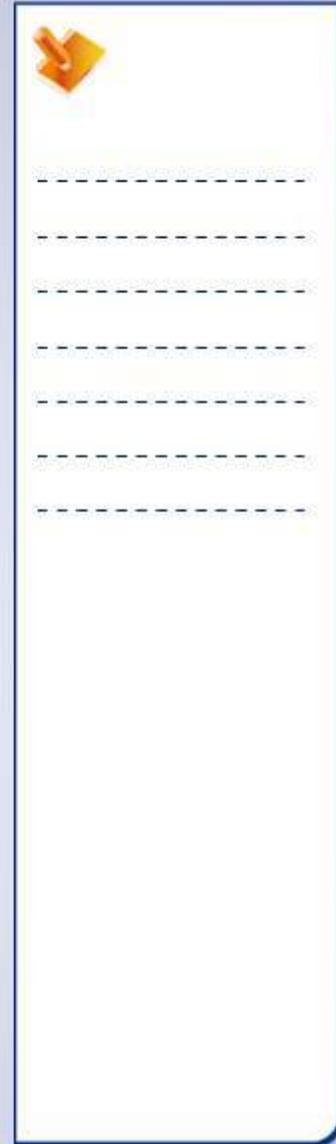
---

## Exercise 6C (1/3)

1. **Open a Analysis Document.**
  - V5A\_Ex\_6C\_Virtual\_Parts\_Distributed\_Mass\_Start.CATAnalysis.
2. **Create Virtual Part.**
  - Create a Rigid Virtual Part on Candela Axis which represents the pulley.
    - a. In **Virtual Parts** toolbar, click **Rigid Virtual Part** icon.
    - b. Name virtual part as *Rigid Virtual Part for Pulley*.
    - c. Select the highlighted face in **Supports** field as shown in image.
    - d. Click **OK**.



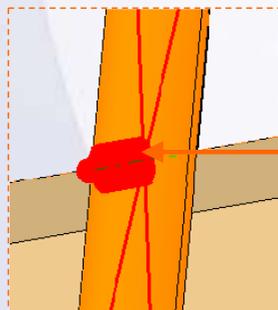
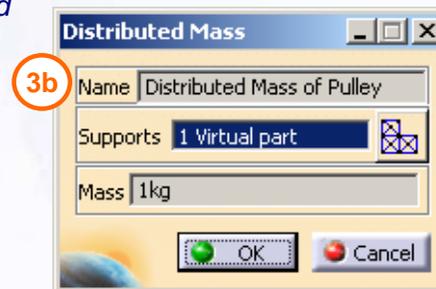
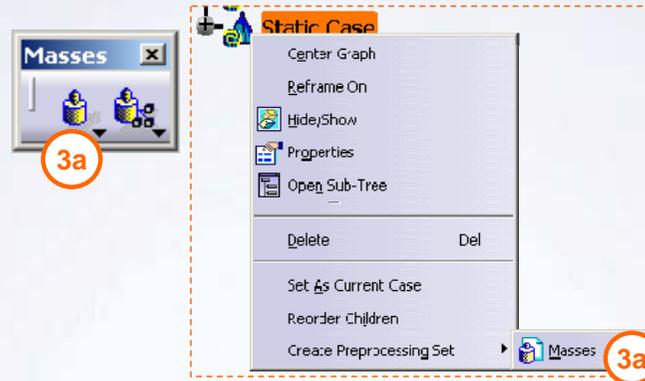
Select face as Support



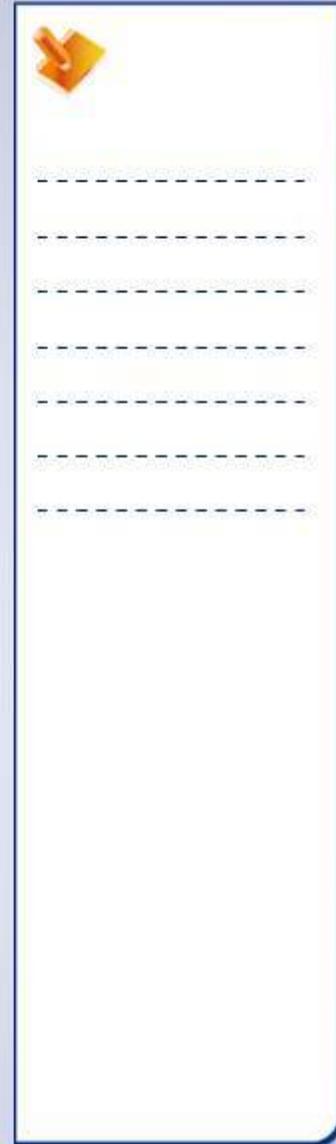
## Exercise 6C (2/3)

### 3. Add Pulley mass to virtual part

- Create a Distributed mass on the virtual part
  - a. Click **Distributed Mass** icon. If the **Masses** toolbar is not available, right-click on *Analysis Manager > Finite Element Model.1 > Static Case* to open contextual menu. Select **Create Preprocessing Set > Masses**. Click on **Masses**. **Masses** toolbar will get activated. Click on **Distributed Mass** icon.
  - b. Enter **Name** as *Distributed Mass for Pulley*
  - c. Select the virtual part *Properties.1 > Rigid Virtual Part for Pulley* from specification tree in **Supports** field.
  - d. Enter **1 Kg** in **Mass** field.
  - e. Click **OK**.



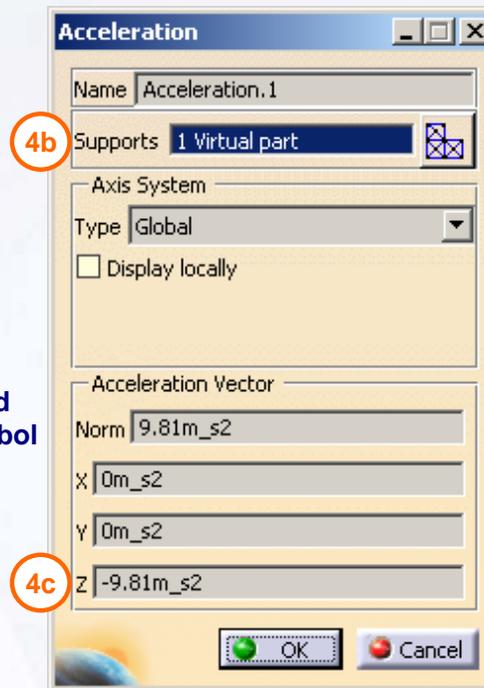
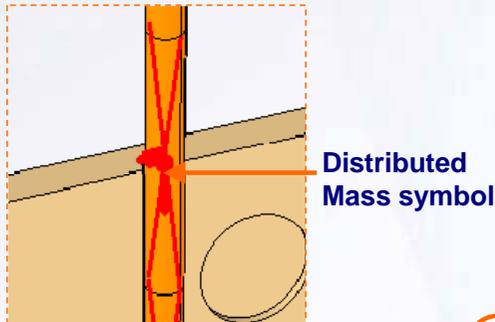
Distributed Mass symbol



## Exercise 6C (3/3)

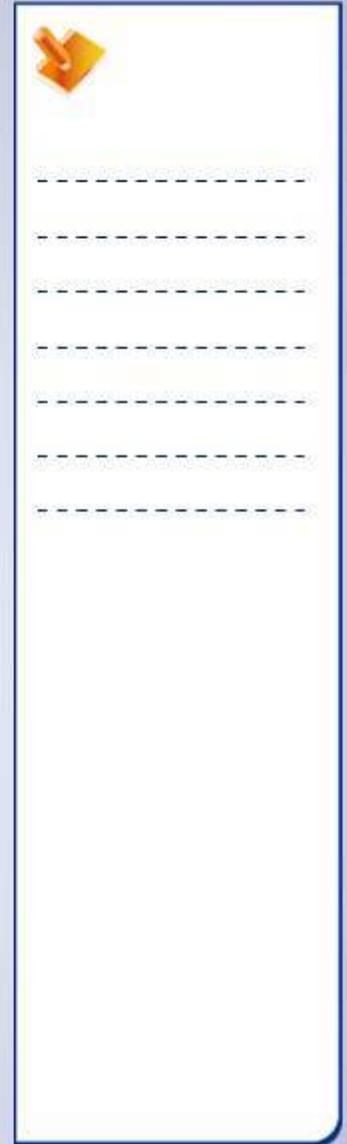
### 4. Convert pulley mass into Load

- Apply Gravitation acceleration to Pulley mass
  - a. In **Loads** toolbar, click on the **Acceleration** icon.
  - b. Select the *Rigid Virtual Part for Pulley* from specification tree in **Supports** field .
  - c. Enter  $-9.81m\_s^2$  in **Z** field.
  - d. Click **OK**.



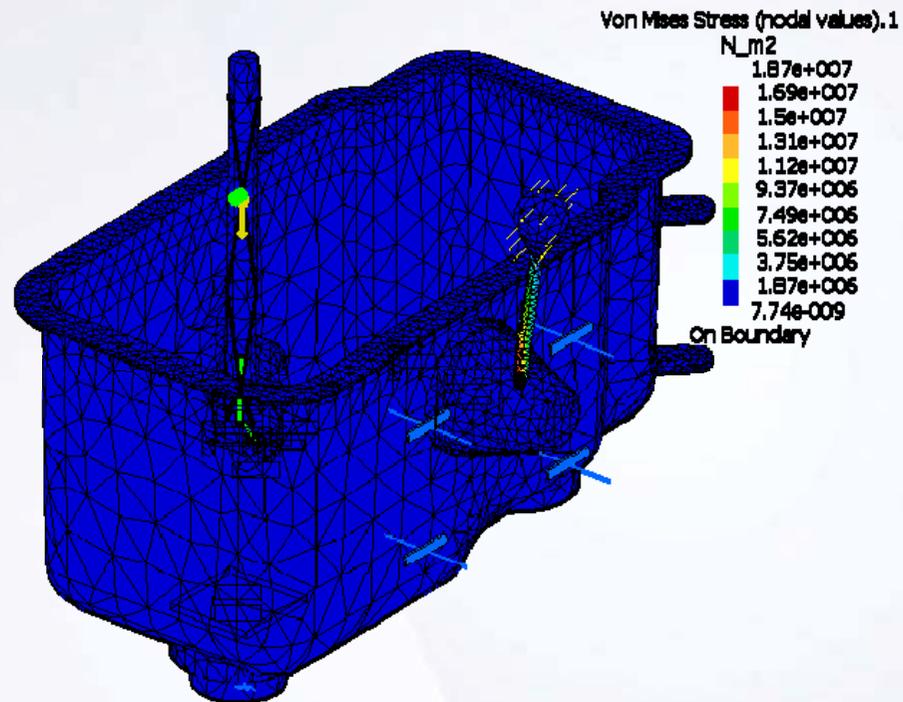
### 5. Compute the Model and visualize results.

- Use Model Checker to validate the model.
- Compute the Model by setting Result file paths.
- Visualize Displacement and Von Mises contours.



## Exercise 6C: Recap

- ✓ Create Virtual Parts.
- ✓ Create Additional Mass.






---



---



---



---



---



---

## Master Project

### Analysis of Drill Press Assembly



2 hours

*The objective of this project is to analyze the complete assembly of the Drill Press System. You have to study every component of the system, create connections and apply the correct material, forces and boundary conditions to analyze the assembly.*

*By the end of this Project you will be able to:*

- Understand Actual Physical system
- Apply Material Properties
- Create Connections
- Apply Forces
- Apply Boundary Conditions
- Check the FE Model
- Compute the FE Model
- Post-processing



## Master Project: Overview (1/3)

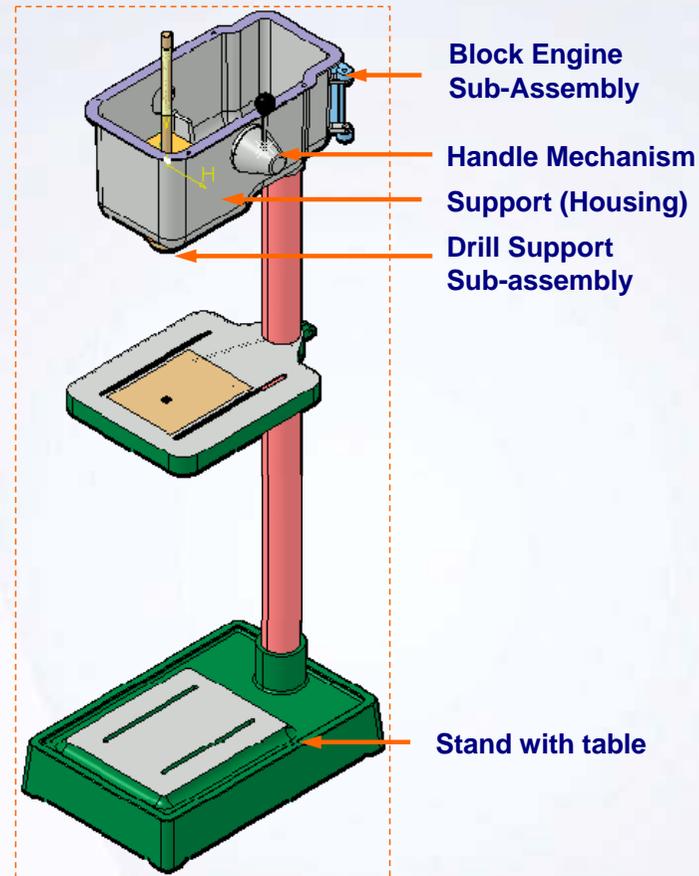
The following is a list of the steps that are required to complete the master project.

Open DrillPress.CATProduct

### 1. Understand Actual Physical system.

- This step is to Understand the actual physical Drill Press System.

### 2. Apply Material Properties.



✚

---



---



---



---



---



---



---

## Master Project: Overview (2/3)

The following is a list of the steps that are required to complete the master project (continued).

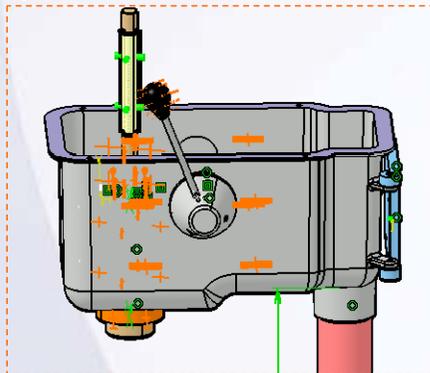
### 3. Create Connections.

- This step is to create connections and assigning connection properties between the components. Create virtual components and distributed masses.

### 4. Apply forces.

- This step is to apply different loads coming on the assembly.

### 5. Apply boundary conditions.



- Rigid Connection Property .1 Handle and handle Block
- Rigid Connection Property .2 Handle Block and Connecting Rod
- Fastened Connection Property .1 Bolt and Bearing
- Contact Connection Property .1 Bearing and Support Plaque
- Fastened Connection Property .2 Barrel and Support Plaque
- Fastened Connection Property .3 Barrel and Support Plaque
- Fastened Connection Property .4 Barrel and Support Plaque
- Contact Connection Property .2 Barrel and Housing
- Rigid Connection Property .3 Barrel and Upper Bearing
- Rigid Connection Property .4 Upper Bearing and Canella Axis
- Rigid Connection Property .5 Barrel and Lower Bearing
- Rigid Connection Property .6 Lower Bearing and Canella Axis
- Fastened Connection Property .5 Engine Support and Housing Upper
- Fastened Connection Property .6 Engine Support and Housing Lower
- Rigid Connection Property .7 Column and Housing
- Rigid Connection Property .8 Column and Base
- Rigid Connection Property .9 Connecting Rod and Bolt
- Contact Connection Property .3 Support and handle Block
- Rigid Connection Property .10 Column and Table
- Rigid Virtual Part .1 Engine Block and Pulley Support




---



---



---



---



---

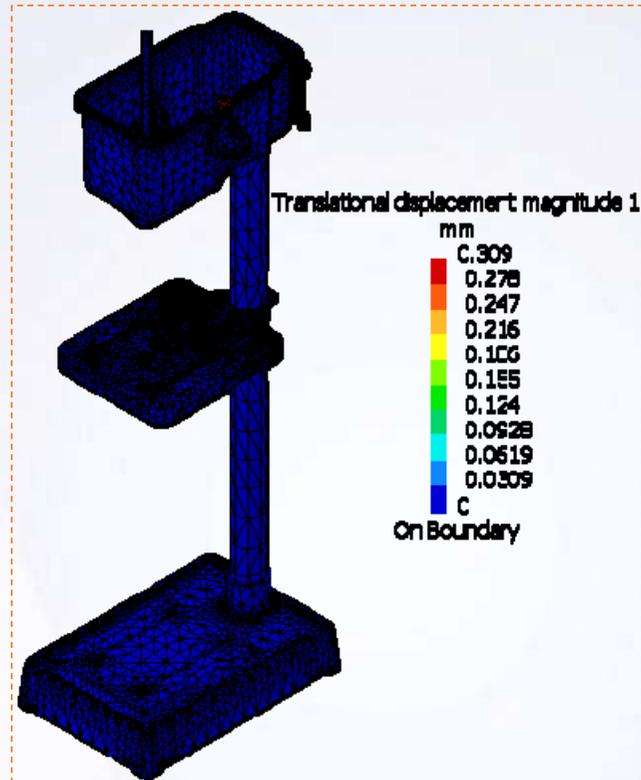


---

## Master Project: Overview (3/3)

The following is a list of the steps that are required to complete the master project (continued).

6. Check the FE model.
7. Compute the FE model.
  - This step is to compute the FE Model using proper solution method
8. Post processing
  - This step is to create the different result images and generate report.






---



---



---



---



---



---

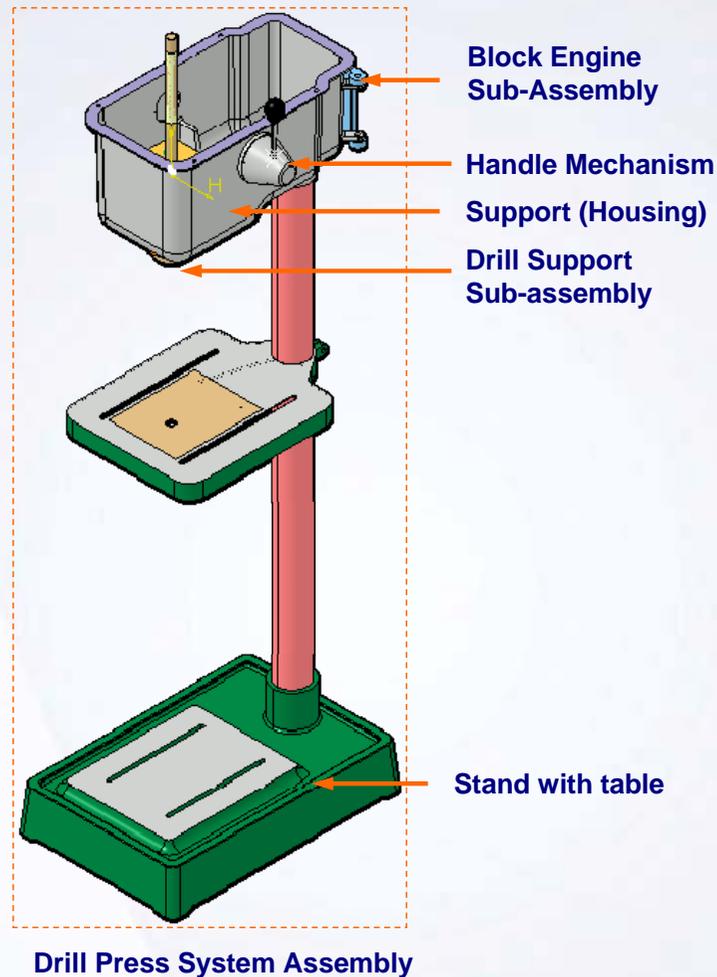
## Step 1 – Understanding Actual Physical System (1/3)

This step is to Understand the actual physical Drill Press System. Open the product DrillPress.CATProduct

The assembly consists of the following components

1. Stand with Table
2. Support (Housing)
3. Block Engine Sub-Assembly with Engine support, Engine and Pulley
4. Handle Mechanism
5. Drill Support Sub-Assembly

These components are the subassemblies and consist of the CATParts as their components.



✚

-----

-----

-----

-----

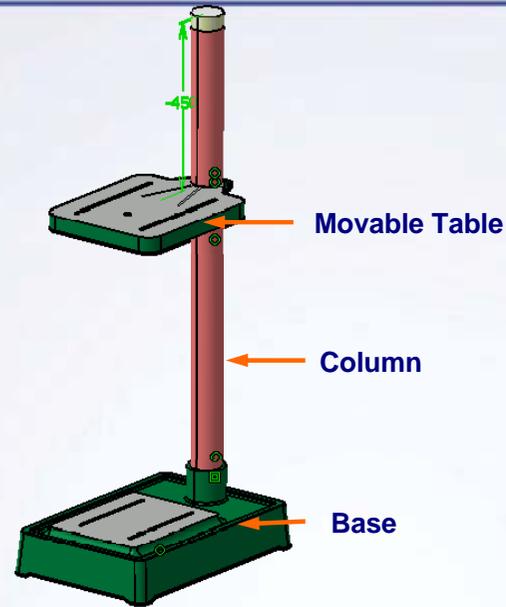
-----

-----

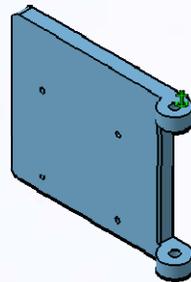
## Step 1 – Understanding Actual Physical System (2/3)

The components are explained in detail here.

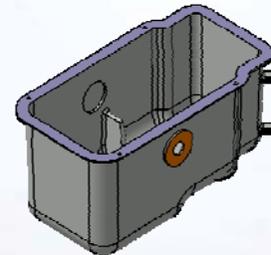
1. Stand with Table
  - a. Base
  - b. Column
  - c. Movable table
  
2. Support (Housing)
  - a. Support (Housing)
  
3. Block Engine Sub-Assembly with Engine support, Engine and Pulley
  - a. Engine Support



Stand with Table Sub-Assembly



Engine Support



Support (Housing)




---



---



---



---



---

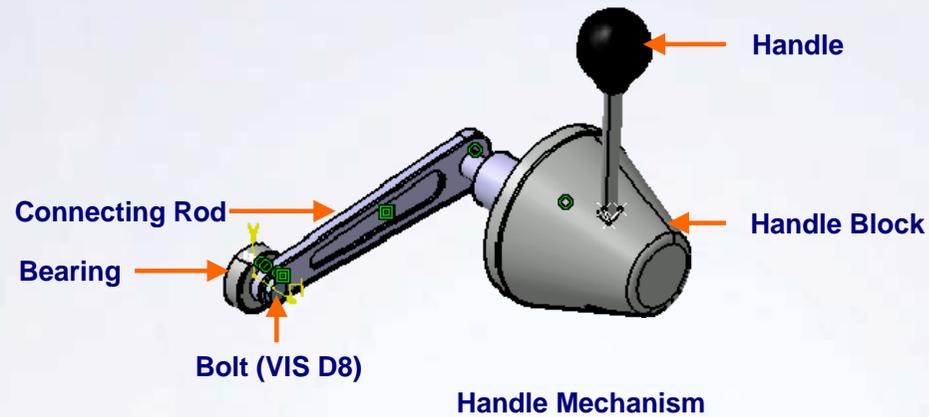


---

## Step 1 – Understanding Actual Physical System (3/3)

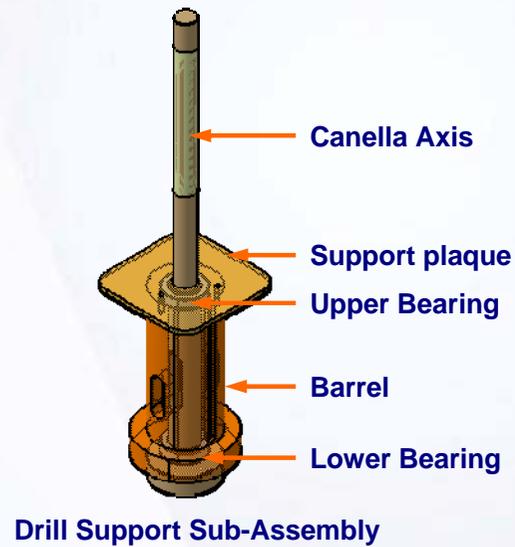
### 4. Handle Mechanism

- a. Connecting Rod
- b. Handle Block
- c. Handle
- d. Bearing
- e. Bolt (VIS D8)



### 5. Drill Support Sub-Assembly

- a. Barrel
- b. Canella Axis
- c. Support plaque
- d. Upper bearing (Bearing D30)
- e. Lower bearing (Bearing D30)



✂

---



---



---



---



---



---



---

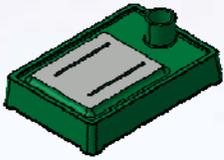
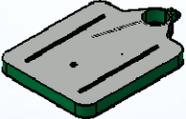
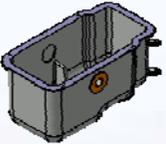
## Step 2 – Applying Material Properties (1/2)

This step is to apply correct material properties to each component.

The material for each component of the assembly is shown in the table. There are two different materials used in the system, one is steel and other is Cast Iron. The required properties of the two materials are provided in the table below.

- Change the existing material properties to the provided one.

Material	Steel	Cast Iron
Young Modulus (N/m <sup>2</sup> )	2.06 x 10 <sup>11</sup>	1.28 x 10 <sup>11</sup>
Poisson Ratio	0.30	0.26
Density (Kg/m <sup>3</sup> )	7840	7200

Name of the Component	3D View	Material
Base		Cast Iron
Column		Steel
Movable Table		Steel
Support (Housing)		Cast Iron




---



---



---



---

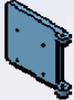
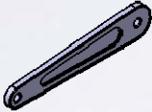
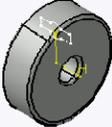


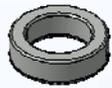
---



---

## Step 2 – Applying Material Properties (2/2)

Name of the Component	3D View	Material
Engine Support		Steel
Connecting Rod		Steel
Handle Block		Steel
Handle		Steel
Bearing		Steel
Bolt		Steel

Name of the Component	3D View	Material
Barrel		Steel
Canella Axis		Steel
Support Plaque		Steel
Upper Bearing		Steel
Lower Bearing		Steel




---



---



---



---



---



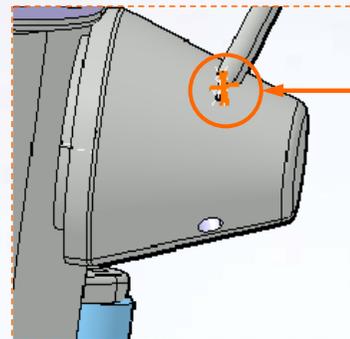
---

## Step 3 – Creating Connections (1/8)

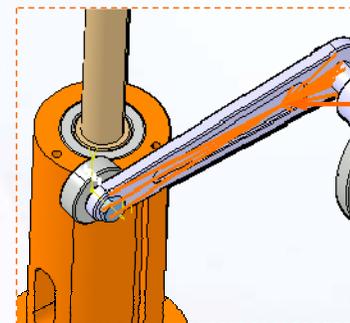
Study the structural connections between the components and assign the connection properties accordingly. Create the virtual part and insert masses wherever required.

Create the analysis connections or use the assembly constraints as per requirements and apply the connection property.

1. Apply Rigid Connection Property between Handle and Handle Block.
  - Create General Analysis Connection between the Handle and Handle Block
  - Apply Rigid Connection Property to the connection.
  
2. Apply Rigid Connection Property between Handle Block and Connecting Rod.
  - Use the appropriate assembly constraint.
  - Apply Rigid Connection Property to the connection.



A symbol representing the Rigid Connection Property



A symbol representing the Rigid Connection Property




---



---



---



---



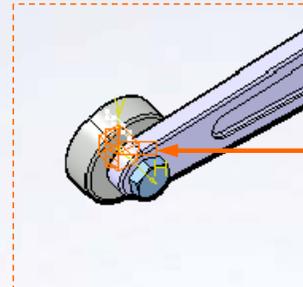
---



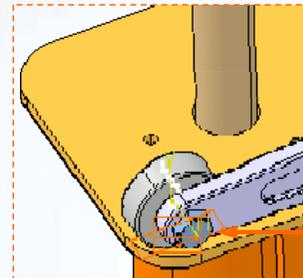
---

## Step 3 – Creating Connections (2/8)

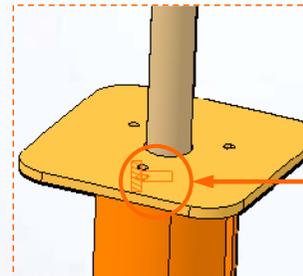
3. Apply Fastened Connection Property between Bolt and Bearing.
  - Use the appropriate assembly constraint.
  - Apply Fastened Connection Property to the connection.
  
4. Apply Contact Connection Property between Bearing and Support Plaque.
  - Create General Analysis Connection between the Bearing and Support plaque
  - Apply Contact Connection Property to the connection.
  
5. Apply Fastened Connection Property between Barrel and Support Plaque, for the first hole pair.
  - Use the appropriate assembly constraint
  - Apply Fastened Connection Property to the connection.



A symbol representing the fastened Connection Property



A symbol representing the contact Connection Property



A symbol representing the fastened Connection Property




---



---



---



---



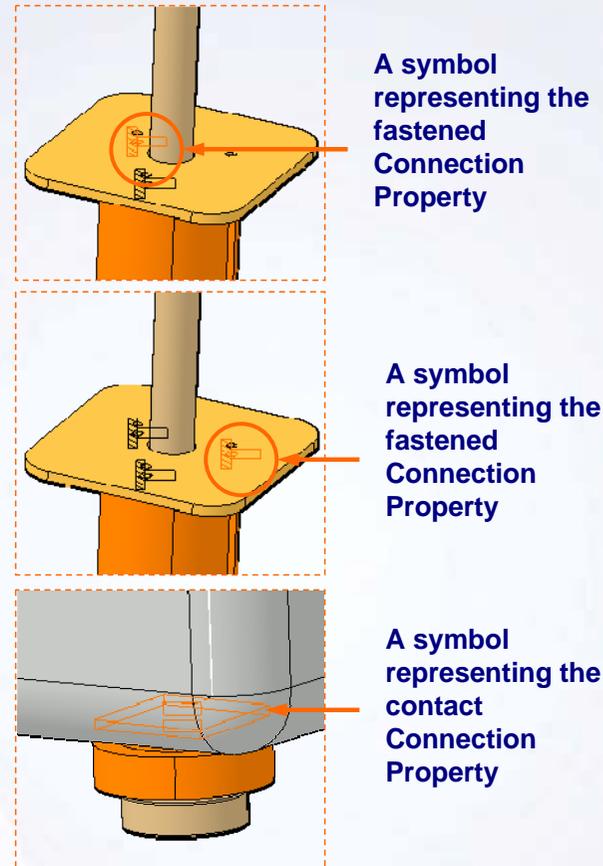
---



---

## Step 3 – Creating Connections (3/8)

6. Apply Fastened Connection Property between Barrel and Support Plaque, for the second hole pair.
  - Use the appropriate assembly constraint.
  - Apply Fastened Connection Property to the connection.
7. Apply Fastened Connection Property between Barrel and Support Plaque, for the third hole pair.
  - Use the appropriate assembly constraint.
  - Apply Fastened Connection Property to the connection.
8. Apply Contact Connection Property between Barrel and Housing.
  - Create General Analysis Connection between the Barrel and Housing.
  - Apply Contact Connection Property to the connection.






---



---



---



---



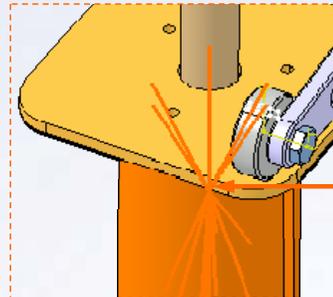
---



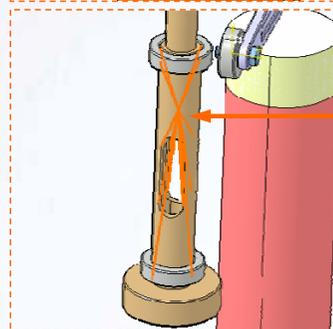
---

## Step 3 – Creating Connections (4/8)

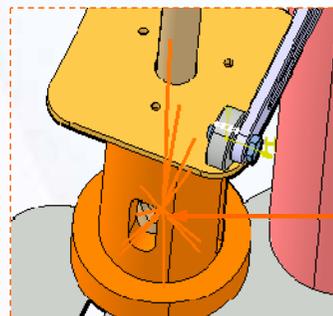
9. Apply Rigid Connection Property between Barrel and Upper Bearing.
  - Create General Analysis Connection between the Barrel and Upper Bearing.
  - Apply Rigid Connection Property to the connection.
  
10. Apply Rigid Connection Property between Upper bearing and Canella axis.
  - Create General Analysis Connection between the Upper bearing and Canella axis.
  - Apply Rigid Connection Property to the connection.
  
11. Apply Rigid Connection Property between Barrel and lower bearing.
  - Create General Analysis Connection between the Barrel and lower bearing.
  - Apply Rigid Connection Property to the connection.



A symbol representing the rigid Connection Property



A symbol representing the rigid Connection Property



A symbol representing the rigid Connection Property




---



---



---



---



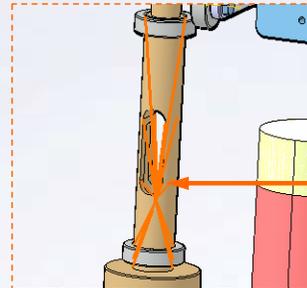
---



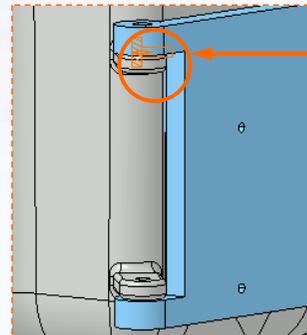
---

## Step 3 – Creating Connections (5/8)

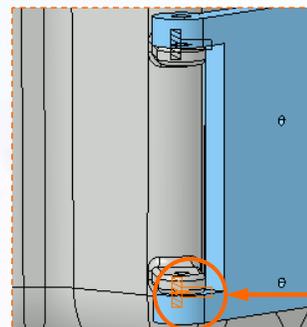
12. Apply Rigid Connection Property between Lower bearing and Canella axis.
  - Create General Analysis Connection between the Lower bearing and Canella axis.
  - Apply Rigid Connection Property to the connection.
  
13. Apply Fastened Connection Property to upper joint of Engine Support and Housing.
  - Use the appropriate assembly constraint.
  - Apply Fastened Connection Property to the connection.
  
14. Apply Fastened Connection Property to lower joint of Engine Support and Housing.
  - Use the appropriate assembly constraint.
  - Apply Fastened Connection Property to the connection.



A symbol representing the rigid Connection Property



A symbol representing the fastened Connection Property



A symbol representing the fastened Connection Property




---



---



---



---



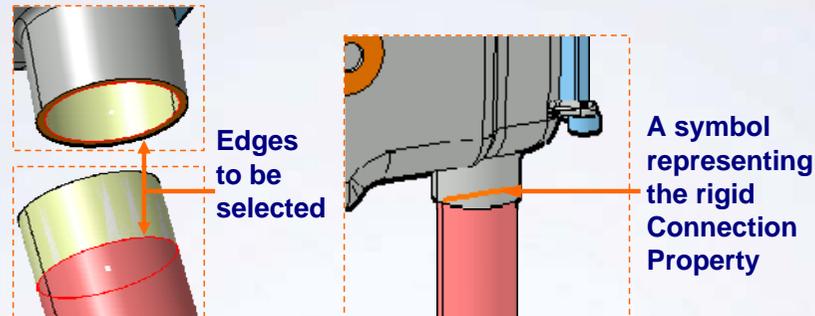
---



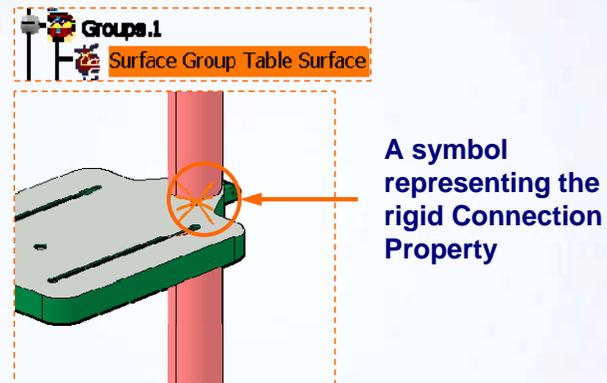
---

## Step 3 – Creating Connections (6/8)

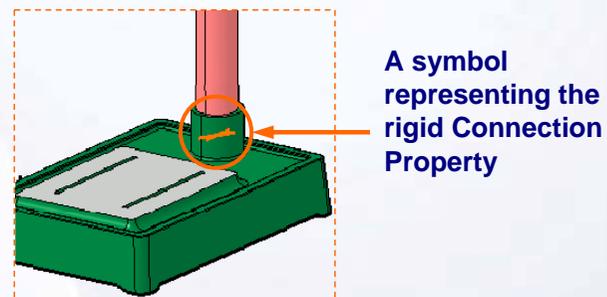
15. Apply Rigid Connection Property between Column and Support (Housing).
- Create General Analysis Connection between the Column and Housing by selecting edges as shown.
  - Apply Rigid Connection Property to the connection.



16. Apply Rigid Connection Property between Column and Table.
- Attach the sub-analysis of the table to the CATPart. Activate the specific CATAnalysis file from Manage Representations. Then synchronize the model from Analysis Assembly 2D Viewer.
  - Use the appropriate group and SewSurface for Table Connection from the Specification tree to create General analysis connection.
  - Apply Rigid Connection Property to the connection.



17. Apply Rigid Connection Property between Column and Base.
- Use the appropriate assembly constraint.
  - Apply Rigid Connection Property to the connection.






---



---



---



---



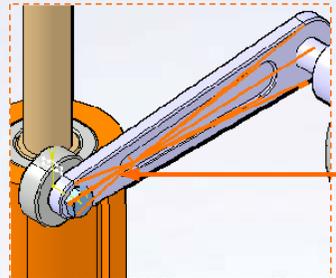
---



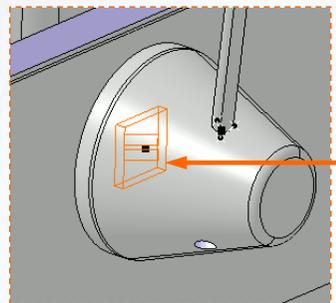
---

## Step 3 – Creating Connections (7/8)

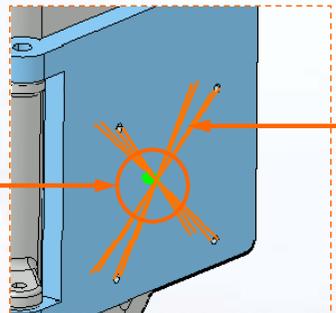
18. Apply Rigid Connection Property between Connecting rod and Bolt.
  - Use the appropriate assembly constraint.
  - Apply Rigid Connection Property to the connection.
  
19. Apply Contact Connection Property between Support and Handle Block.
  - Create General Analysis Connection between the Support and Handle Block.
  - Apply Contact Connection Property to the connection.
  
20. Create Rigid Virtual Part.
  - Create the rigid virtual part representing Engine Block and pulley support using faces of the four holes.
  - Create distributed mass of 11 kg using the virtual part as a support.



A symbol representing the rigid Connection Property



A symbol representing the contact Connection Property



A symbol representing the Distributed mass

A symbol representing the rigid virtual part




---



---



---



---



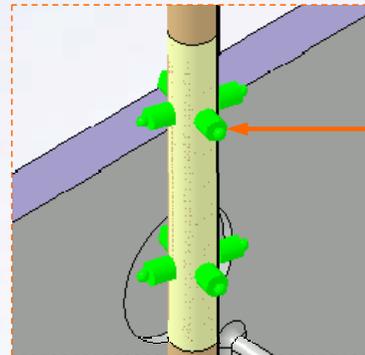
---



---

## Step 3 – Creating Connections (8/8)

21. Create distributed mass representing pulley.
  - Create distributed mass of 2 kg on the outer surface of the canella axis.



A symbol  
representing the  
Distributed mass

✎

---

---

---

---

---

---

---

---

## Step 4 – Applying forces (1/2)

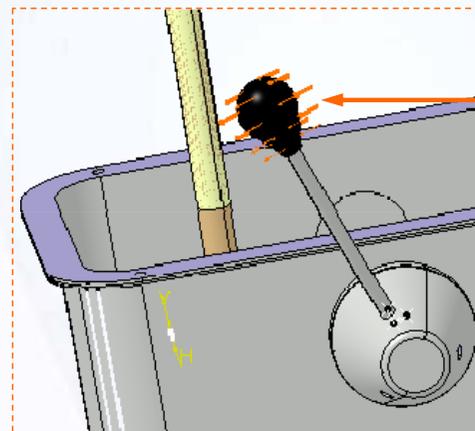
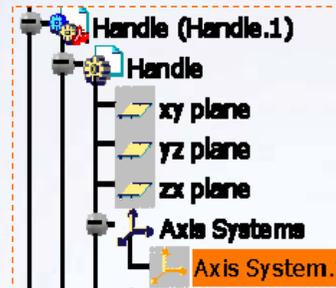
This step is to study the different Loads acting on the Drill Press Assembly.

The following are the forces acting on the assembly components.

1. Force applied on the handle, for drilling operation
2. Gravitational force due to weight of the Engine Block and pulley
3. Force on the Table surface, during drilling. This load will act as pressure, distributed on the area specified over the table to.
4. Gravitational force due to weight of the Pulley on the canella axis

Apply different forces on the assembly as follows

1. Apply force on handle.
  - Apply the Distributed force of 10N on the handle.
  - Direction of the force is perpendicular to the axis of the handle. Use the user axis system.



A symbol representing the Distributed force

**Before opening the exercise data for this step, please ensure that you have closed all the previously opened files in this CATIA session.**




---



---



---



---



---



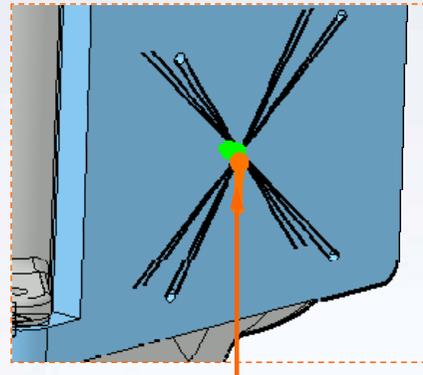
---



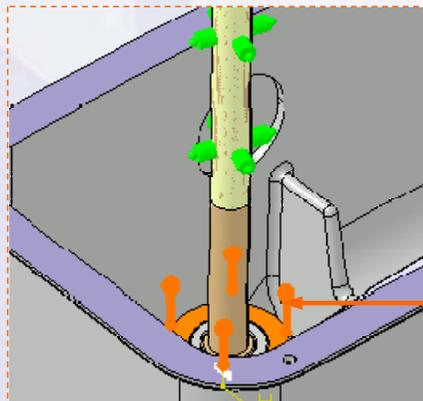
---

## Step 4 – Applying forces (2/2)

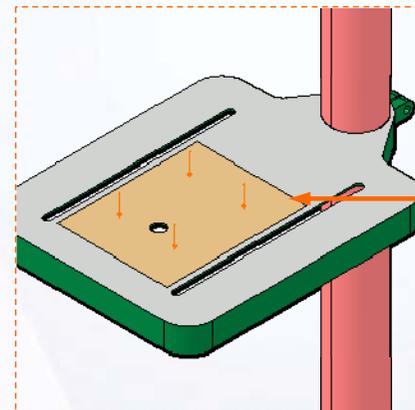
2. Apply gravitational force due to weight of Engine block and pulley.
  - Apply acceleration due to gravity ( $9.81 \text{ m/s}^2$ ) on the virtual part.
3. Apply pressure on the table surface.
  - Apply pressure of  $1 \text{ N/m}^2$ .
  - Use the group created on the top face of the table using group.
4. Apply Gravitational force due to weight of the Pulley on the canella axis.
  - Apply acceleration due to gravity ( $9.81 \text{ m/s}^2$ ) on the surface where distributed mass is applied.



A symbol representing the acceleration



A symbol representing the acceleration



A symbol representing the pressure




---



---



---



---



---



---

## Step 5 – Applying Boundary Conditions (1/2)

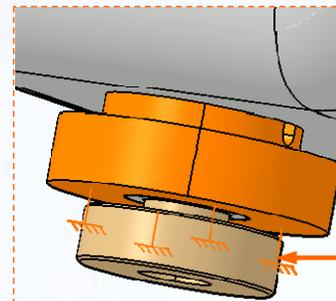
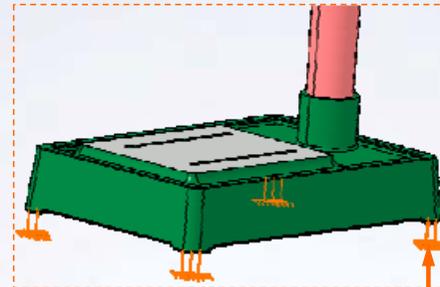
This step is to apply the boundary conditions. The first step is to understand the actual boundary conditions.

The bottom surface of the Base and the Barrel are restrained to move. So they are totally clamped.

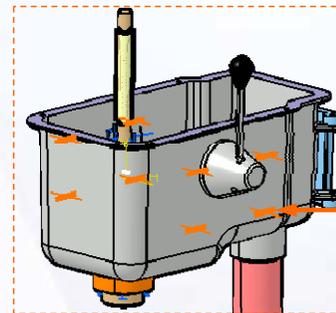
The side surfaces of the Support (housing) and the outer surfaces of the barrel and canella axis are free to move in vertical direction only. So they are restrained to move in the direction normal to surface.

The vertical surface of the Handle block, which is in contact with the housing, is not allowed to move in the direction perpendicular surface plane. So the surface slider restraint is applied to the surface.

1. Apply Clamp to the Base .
  - Bottom of the base is clamped.
2. Apply Clamp to the Barrel.
  - Bottom of the Barrel is clamped.
3. Apply Surface Slider restraints to Housing .
  - Apply Surface Slider restraints to side surfaces of the Housing.



Symbols representing the Clamp



Symbols representing the surface Slider




---



---



---



---



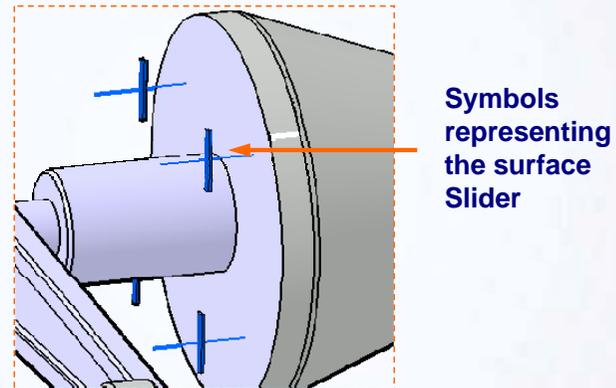
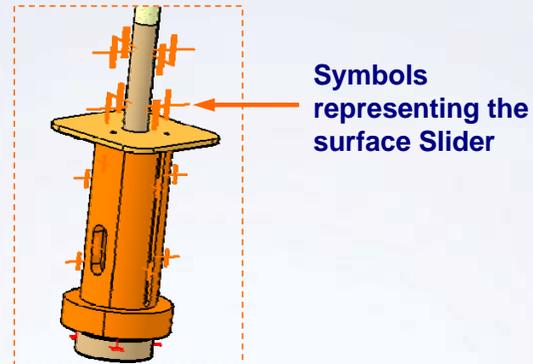
---



---

## Step 5 – Applying Boundary Conditions (2/2)

4. Apply Surface Slider restraints to Barrel and Canella Axis.
  - Apply Slider restraints to outer surface of the Barrel and Canella Axis.
5. Apply Surface Slider restraints to Handle block.
  - Apply Slider restraints to vertical surface of Handle block.






---



---



---



---



---



---



---

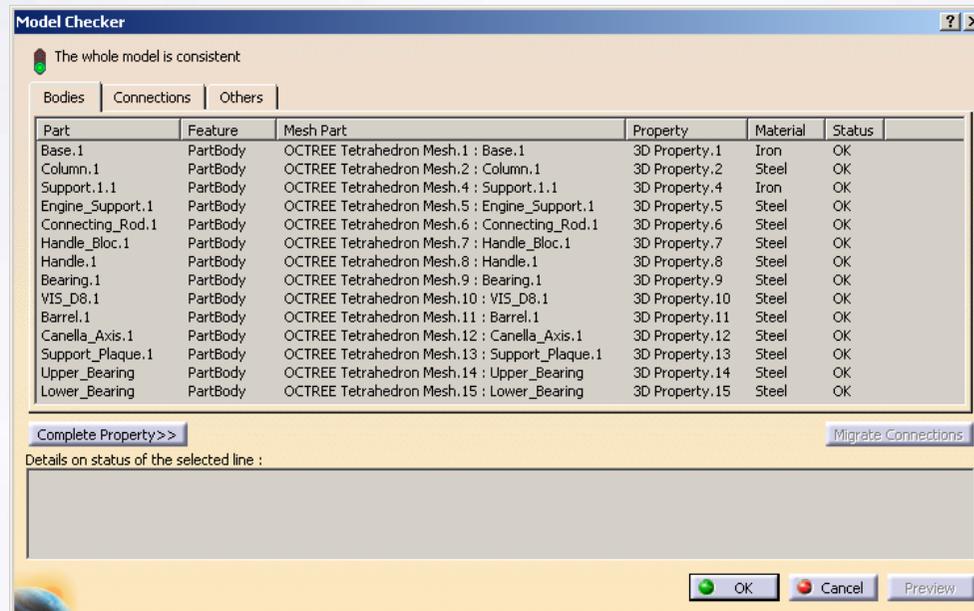
## Step 6 – Checking the FE Model

This step is to Check the created Finite Element model.

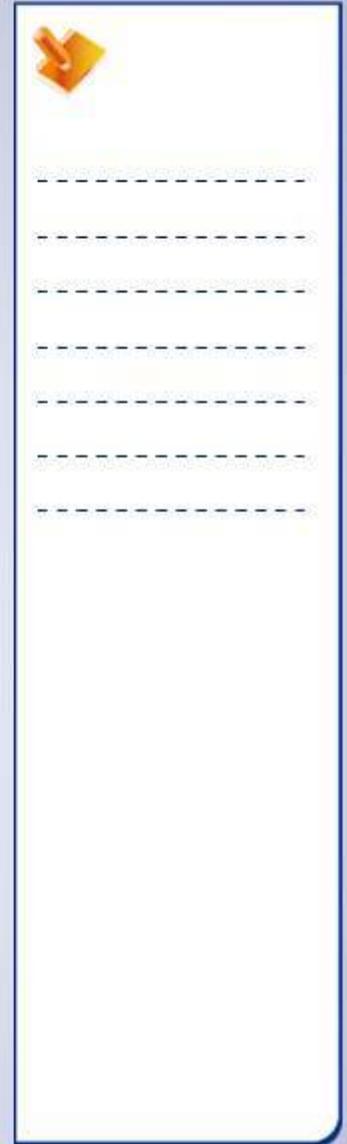
From the first step to the previous step, the FE preprocessing of the assembly model is completed. It is necessary to validate the model, before carrying out the computation step.

Check the Model using Model Checker.

- Confirm the message “The whole model is consistent”, with green signal.



Before opening the exercise data for this step, please ensure that you have closed all the previously opened files in this CATIA session.

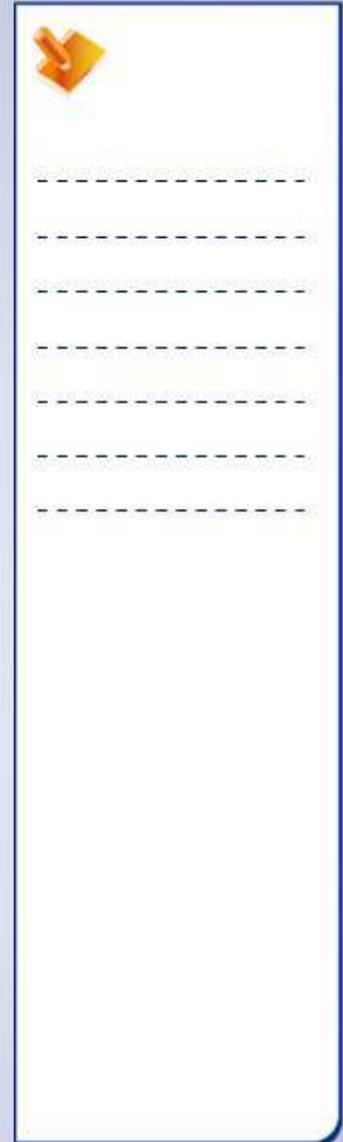
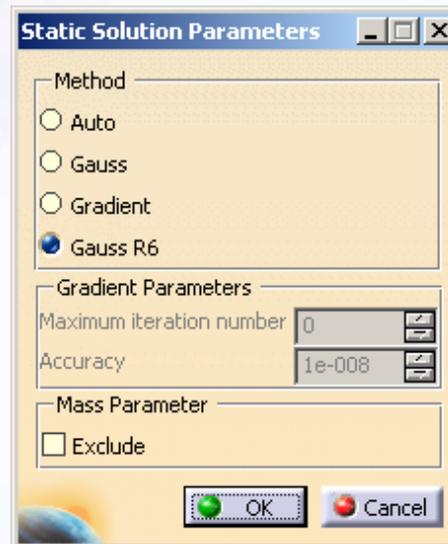
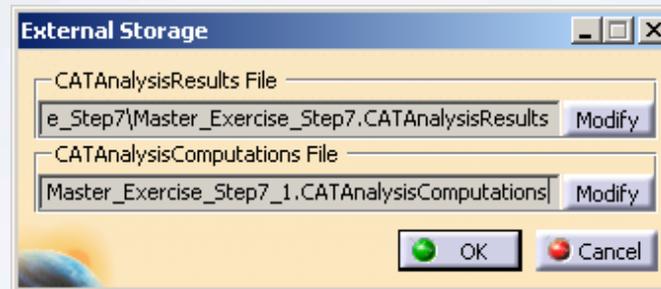


## Step 7 – Computing the FE Model

This step is to solve the model of the assembly.

Select the proper method for solution, based on the number of Degrees Of Freedom (DOF) for FE model.

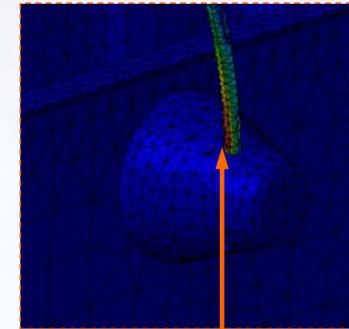
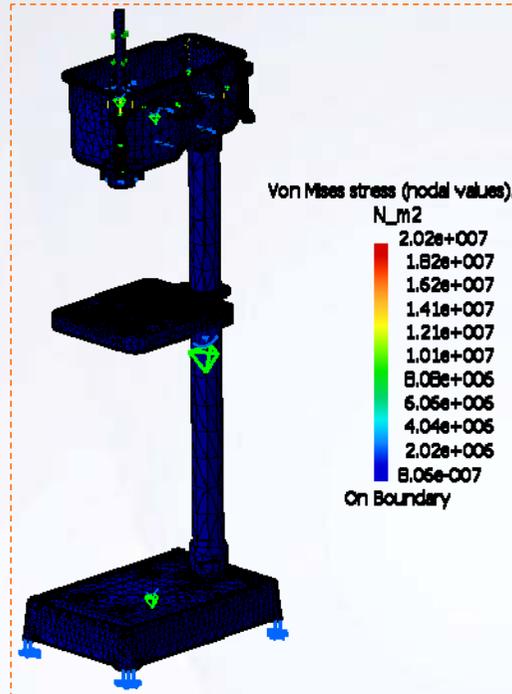
1. Specify the external storage paths.
  - Specify the external storage path for CATAnalysisResult file.
  - Specify the external storage path for CATAnalysisComputations file.
2. Choose the Solution Method.
  - Select the proper method in Static Case Solution parameters.
3. Compute the Model.



## Step 8 – Post-processing (1/3)

This step is to carry out the post processing and study the results.

1. Post-processing: Obtain different result images of the analysis.
  - Create Von Mises Stress (Nodal values) image.
  - Create Translational displacement magnitude image.
  - Create Estimated local error image.
  
2. Generate the Report.
  - Specify the path for the output directory.
  - Specify the title for report.
  - Add created images in the report.
  - Choose the analysis case.
  - Create the report.



Location of Maximum Von Mises stress

Before opening the exercise data for this step, please ensure that you have closed all the previously opened files in this CATIA session.




---



---



---



---



---

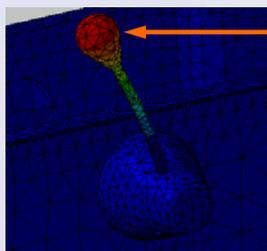
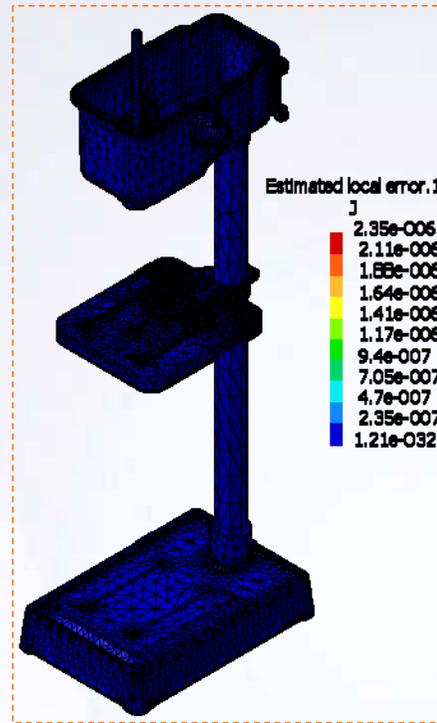
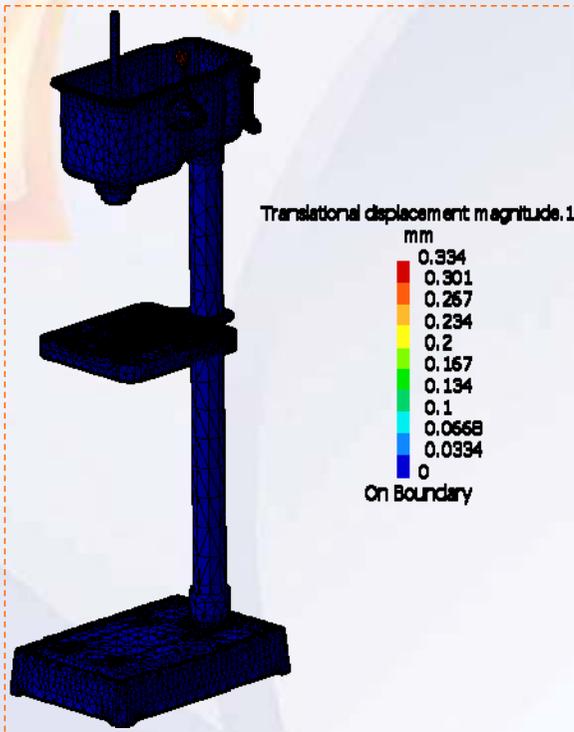


---

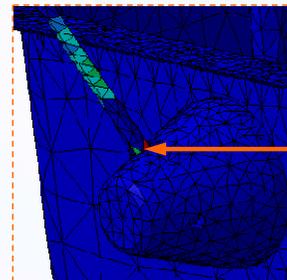


---

## Step 8 – Post-processing (2/3)



Location of Maximum Displacement



Location of Maximum Error

## Step 8 – Post-processing (3/3)

### Master\_Exercise\_Report

#### MESH:

Entity	Size
Nodes	190095
Elements	109478

#### ELEMENT TYPE:

Connectivity	Statistics
SPIDER	1432 ( 1.31% )
NSBAR	10 ( 0.01% )
TE10	108036 ( 98.68% )

#### Materials.1

<b>Material</b>	Iron
<b>Young's modulus</b>	1.28e+011N_m2
<b>Poisson's ratio</b>	0.26

## Shortcuts

<b>F1</b>	Link to on-line documentation	<b>Ctrl</b> + several selections	Multiple selection
<b>Shift F1</b>	Contextual help for an icon	<b>Shift</b> + 2 selections	Selection of all elements between and including the 2 selected elements
<b>Shift F2</b>	Overview of the specification tree		
<b>F3</b>	Hide/Show the specification tree		
<b>Ctrl</b> + Tab	Change CATIA V5 window	<b>Alt</b> F8	Macros
<b>Ctrl</b> N	New file	<b>Alt</b> F11	Visual Basic editor
<b>Ctrl</b> O	Open file	<b>Alt</b> + Enter	Properties
<b>Ctrl</b> S	Save file	<b>Alt</b> + MB1	Pre-selection Navigator
<b>Ctrl</b> P	Print	<b>Ctrl</b> F11	Pre-selection Navigator
<b>Ctrl</b> Z	Undo	<b>Up/Down</b> or <b>Left/Right</b> arrow	Pre-selection Navigator
<b>Ctrl</b> Y	Redo		
<b>Ctrl</b> C	Copy	<b>Shift</b> + MB2	Local zoom and change of viewpoint
<b>Ctrl</b> V	Paste		
<b>Ctrl</b> X	Cut	<b>Shift</b> + manipulation with compass	Displacement respecting constraints
<b>Ctrl</b> U	Update		
<b>Ctrl</b> F	Find		



## Glossary

**Discretization:** is the process of dividing the model into a mesh of finite elements.

**Degrees of Freedom (DOF):** are the variables solved during analysis, which for a pure structural analysis are the nodal displacements and rotations.

**Generative Shape Design:** is the workbench that allows you to quickly model both simple and complex shapes, using wireframe and surface features.

**Part Design:** is the workbench dedicated for designing parts using the solid modeling approach.

**Sag:** is the perpendicular distance between the actual geometry and the Element.

**Course Meshing:** is the meshing carried out by elements whose size is relatively larger.

**Property:** is an attribute such as color or a name that can be assigned to any feature. All features can be customized in both appearance and function.

**Free Vibration Analysis:** is the analysis in which the eigen frequencies are computed by referencing only masses. (constraints are not taken into account)



-----

-----

-----

-----

-----

-----















## **User Companion CATIA | ENOVIA | DELMIA | SIMULIA | 3DVIA**

**Your everyday companion!**

Companion is an essential tool which allows you to continuously enhance your skills and optimize your performance with Dassault Systemes products – right at your desk! The Companion includes theory, demonstrations, exercises, and methodology recommendations that enable you to learn proven ways to perform your daily tasks. Every release the Companion is updated by Dassault Systemes experts to ensure that your knowledge remains current.

For more details please visit [www.3ds.com/education/](http://www.3ds.com/education/)



**Show them what you know!**

**Get Certified!**

Research shows, and industry experts agree, that an IT certification increases your credibility in the Information Technology workplace. It provides tangible evidence to show that you have the proficiency to provide a higher level of support to your employer. Are you ready to get certified and affirm the knowledge, skills, and experience you possess and gain a worldwide recognized credential leading to success?

For complete details please visit <http://www.pearsonvue.com/dassaultsystemes/>

