Lesson 6: Assembly Structural Analysis

In this lesson you will learn different approaches to analyze the assembly using assembly analysis connection properties between assembly components. In addition to this you will learn how to use virtual parts and distributed mass in Assembly Analysis.

Lesson content:

- **Case Study: Analysis Assembly of Drill Press Sub-Assembly**
- **Design Intent**
- **Assembly Structural Analysis**
- **Analysis Assembly**
- **Analysis Connections and Connection Properties**
- **Virtual Parts**
- **Distributed Mass**

Duration: **Approximately 0.25 days**

Case Study: Analysis Assembly of Drill Press Sub-Assembly

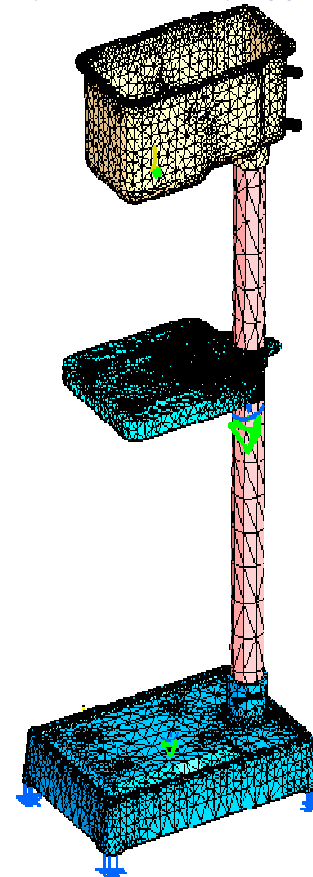The case study for this lesson is the Analysis Assembly of Drill Press Sub-Assembly. The focus of this case study is to create FE model of the Drill Press Sub-Assembly using Analysis Assembly approach

Introduction to FEA	GPS pre-processing	GPS Computation	GPS Post-processing	Mesh Refinement	**Assembly Structural Analysis**
Exercises	Exercises	Exercises	Exercises	Exercises	Exercises

(Case Study) Master Project
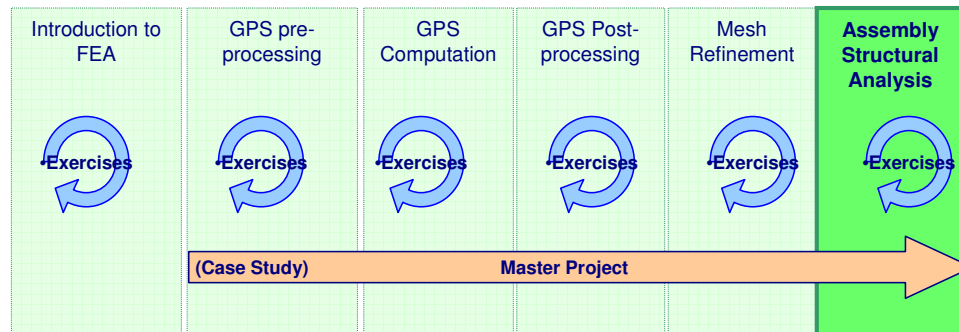
Drill Press Sub-Assembly Mesh Image

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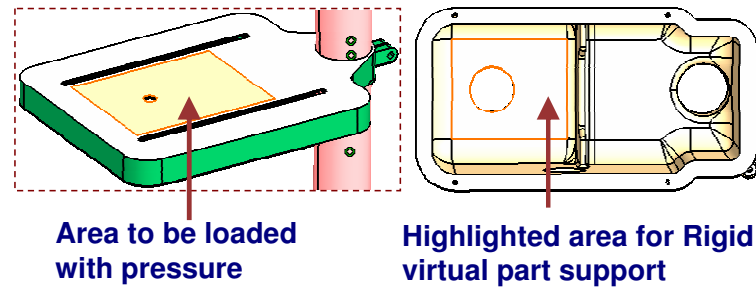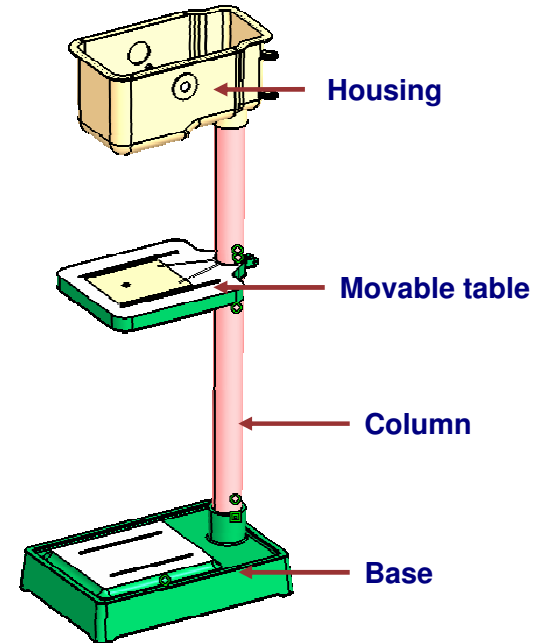
The analysis must be performed with the following parameters and component connections :

- ✓ 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.

Housing

Movable table

Column

Base

Area to be loaded with pressure

Highlighted area for Rigid virtual part support

Stages in the Process

The following steps are used to perform the case study:

- Learn different Assembly Structural Analysis approaches.
- Create Assembly connections.
- Apply Connection properties.
- Create a Virtual Part.
- Create a Distributed Mass.
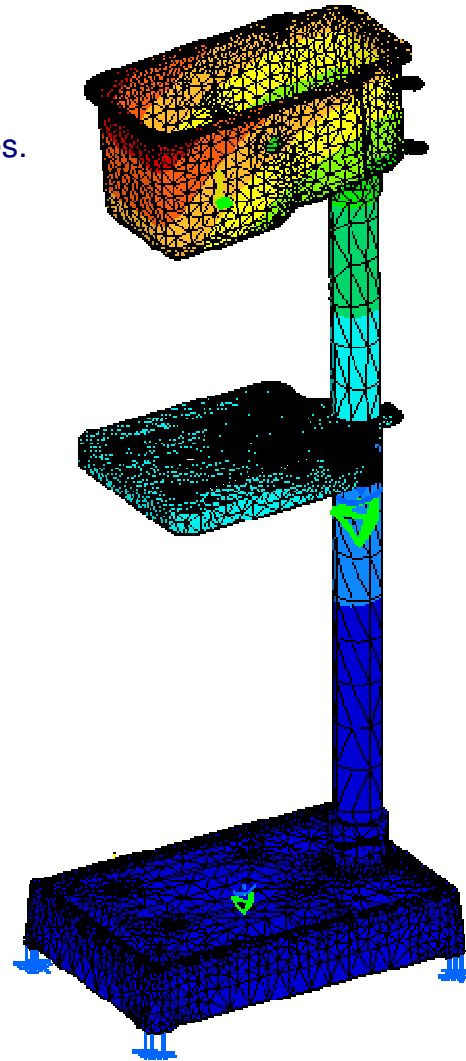
Step 1: Assembly Structural Analysis

You will learn about Generative Assembly Structural Analysis and the different approaches to perform Assembly Structural Analysis.

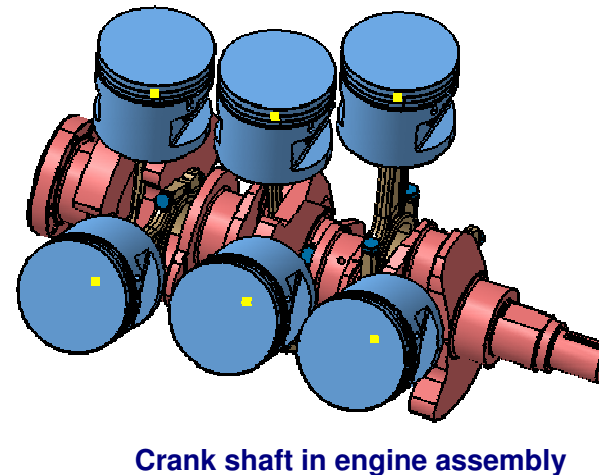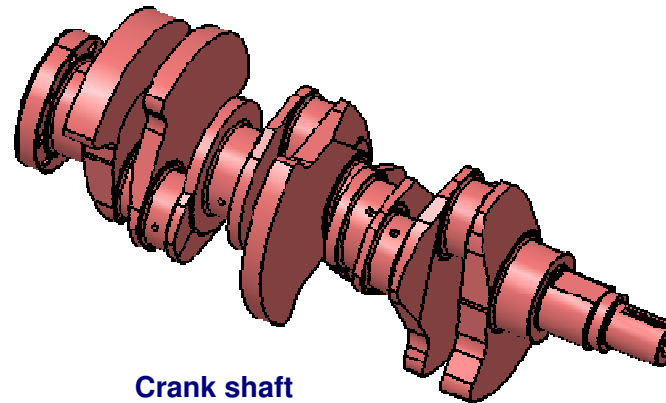**To study Generative Assembly Structural Analysis use the following steps:**

1. **Assembly Structural Analysis**
2. Analysis Assembly
3. Analysis Connections and Connection Properties
4. Virtual Parts
5. Distributed Mass

What is Assembly Structural Analysis

Assembly Structural Analysis is generally used to perform structural analysis of assemblies.
In this, physical assemblies are modeled using Finite Element Assemblies.

- While designing, structural analyses of individual parts are performed. These parts are commonly components of a product.

- For example, an analysis of a crankshaft produces the information about the structural, dynamic response of the component under an applied load, which can be used to improve the crankshaft design.

Crank shaft

- A crankshaft is one of the parts of engine assembly which also contains connecting rod, pin, piston, bolts etc.

- Once these individual parts are assembled, it is necessary to understand the structural and dynamic behavior of the assembly. Therefore, assembly analysis is required.

Crank shaft in engine assembly

What is Generative Assembly Structural Analysis (1/5)

The Generative Assembly Structural Analysis (GAS) license provides functionalities for analysis of assembly, through the Generative Structural Analysis workbench. It allows you to define the connections between the assembly components and assign different types of connection properties to these connections to simulate the real connection behavior.

In other words, 'GAS' allows you to define real constraints using connection properties. It lets you define four different kinds of connection properties:

- Face/Face Connection properties

- Distant Connection properties

- Welding Connection properties

- GPSAeroConnection container

To be able to create the connection properties, you must have previously defined 'Assembly constraints' or 'Analysis Connections' using GAS Workbench. You must ensure that your assembly is properly constrained (i.e., neither over-constrained nor under-constrained).
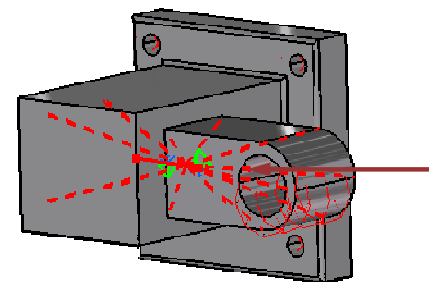
What is Generative Assembly Structural Analysis (2/5)

When you work with the Analysis workbench, the following assumptions are imposed:

- Small displacements (translation and rotation)
- Small strain
- Linear constitutive law: linear elasticity

Thus, if there is no contact feature (either virtual or real), no pressure fitting property and no bolt tightening feature (virtual or not), then the problem is linear which means that the displacement is a linear function of the load.

Rigid Connection property

Linear Problem

On the other hand, If there is at least one contact feature (either virtual or real) or pressure fitting property or bolt tightening feature (virtual or not), then the problem is nonlinear, which means that the displacement is a non linear function of the load.
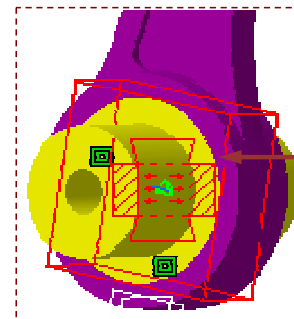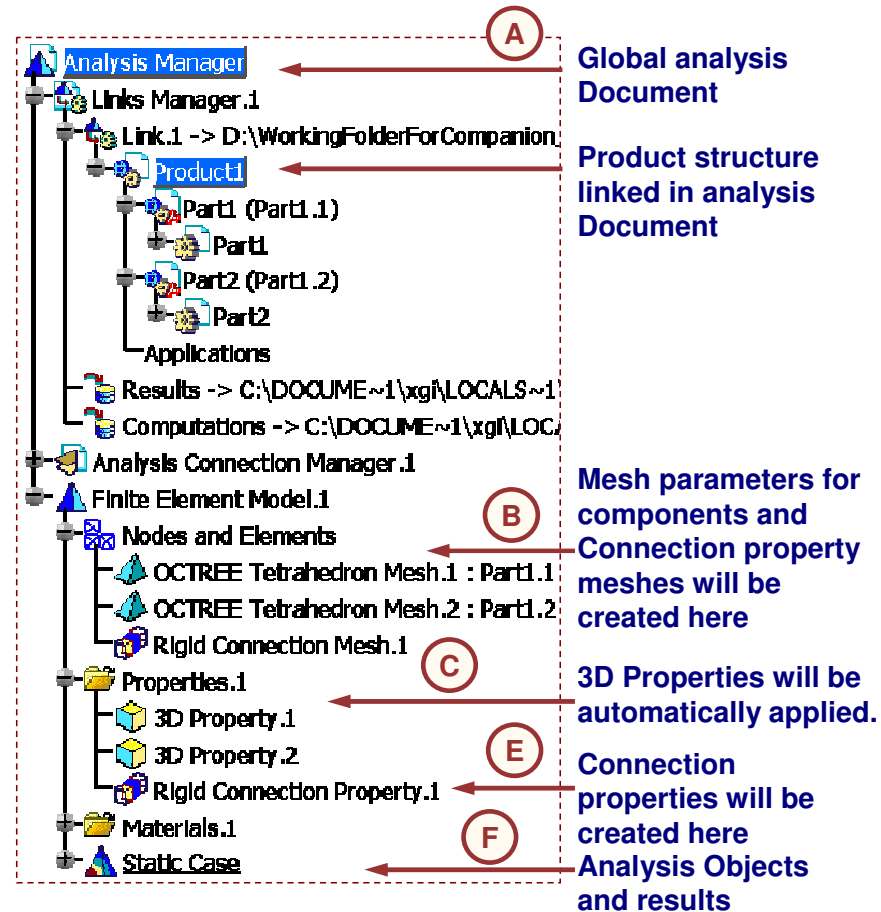

Pressure Fitting Connection property

Non-linear Problem

What is Generative Assembly Structural Analysis (3/5)

We have two basic approaches to perform Generative Assembly Structural analysis – Assembly Analysis and Analysis Assembly
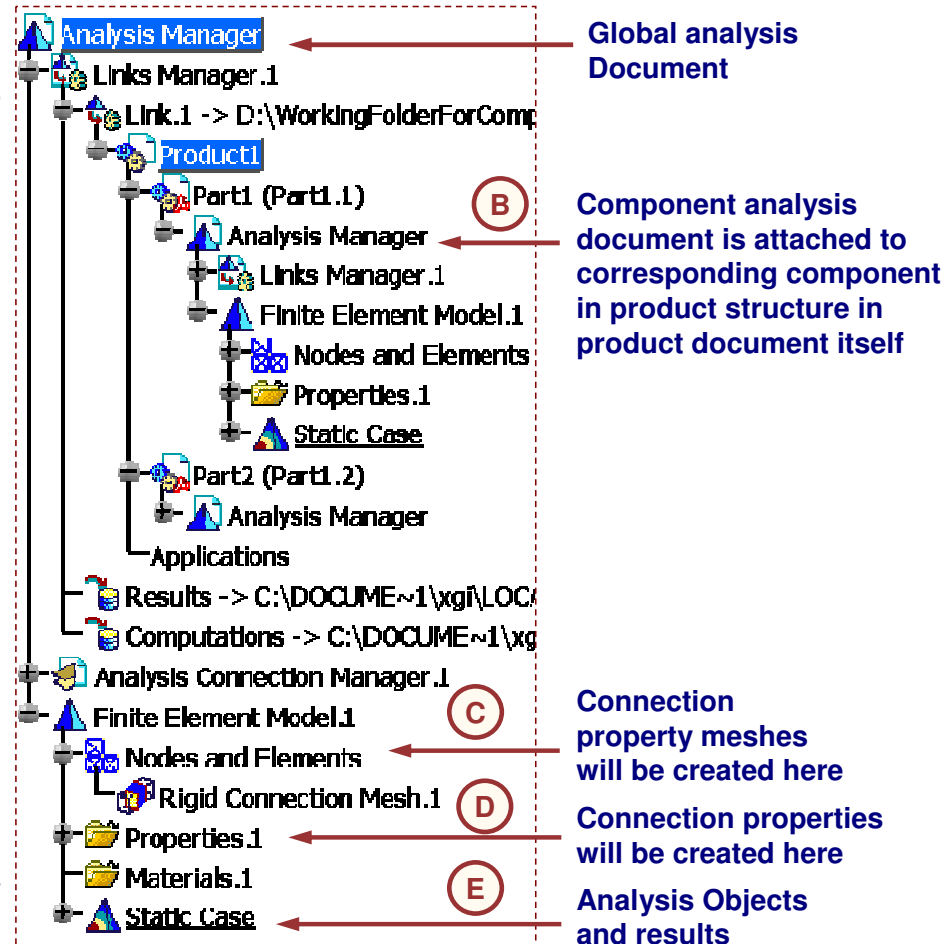
Assembly Analysis:

A. The product document is available. You will switch to analysis workbench to create analysis document which will be the only analysis document in this approach.

B. The default OCTREE Tetrahedron Mesh parameters are automatically applied to 3D components in product structure.

C. In similar way, 3D Properties are automatically applied to 3D components.

D. For 1D or 2D components you will apply mesh parameters and properties manually.

E. You will assign Connection Properties between components either by creating analysis connections or by re-using existing assembly constraints.

F. Finally, analysis of the complete assembly is performed.

A — Global analysis Document

Product structure linked in analysis Document

B — Mesh parameters for components and Connection property meshes will be created here

C — 3D Properties will be automatically applied.

E — Connection properties will be created here

F — Analysis Objects and results

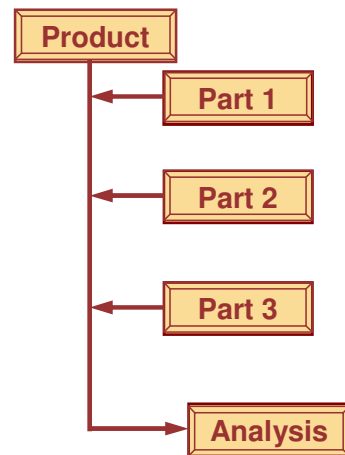# What is Generative Assembly Structural Analysis (4/5)

Analysis Assembly:

A.   The Product document may or may not be available. However you have individual component analysis documents.

B.   In case the product document is available, attach component analysis documents to corresponding components in product document.

C.   The component Mesh parameters and Property information is reused from these component analysis documents for the Global Assembly analysis. You can create mesh parameters and properties in the Global analysis document for components not having attached analysis documents.

D.   You will assign Connection Properties between components either by creating analysis connections or by re-using existing assembly constraints.

E.   Finally, analysis of the complete assembly is performed.

Global analysis Document

Component analysis document is attached to corresponding component in product structure in product document itself

Connection property meshes will be created here

Connection properties will be created here

Analysis Objects and results

What is Generative Assembly Structural Analysis (5/5)
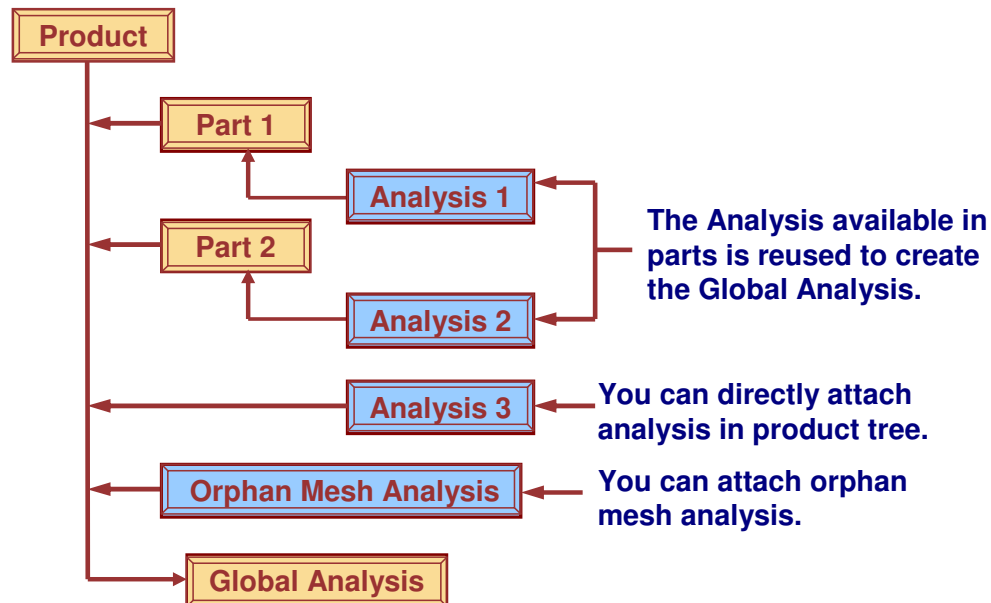
These two basic approaches to perform Generative Assembly Structural analysis are differentiated as follows.

Assembly Analysis:

Analysis Assembly:

Product

Part 1

Part 2

Part 3

Analysis

Product

Part 1

Analysis 1

Part 2

Analysis 2

The Analysis available in parts is reused to create the Global Analysis.

Analysis 3

You can directly attach analysis in product tree.

Orphan Mesh Analysis

You can attach orphan mesh analysis.

Global Analysis

In this approach you create the assembly and analyze it.

In this approach, analyses of individual parts in product are available. This analyses are assembled to form 'Analysis Assembly' and then final analysis called 'Global analysis' is performed.

Step 2: Analysis Assembly

You will learn about advantages of using Analysis Assembly approach and how to re-use existing analysis documents in approach.

To study Generative Assembly Structural Analysis use the following steps:

1. Assembly Structural Analysis
2. **Analysis Assembly**
3. Analysis Connections and Connection Properties
4. Virtual Parts
5. Distributed Mass

What is Analysis Assembly

In this approach, analyses of individual parts in a product are available. These analyses are assembled to form an 'Analysis Assembly' and then a final 'Global Analysis' is performed for the product.

While following this approach you have the option of using an available product with assembly constraints. In case a product is not available, you can create a product document and directly attach individual part analysis files in the product document. These analysis files must be computed with at least 'Mesh only' option, so that it contains Mesh and complete FE property information .

The Analysis Assembly approach has following advantages:

- It uses already meshed individual parts and imported orphan mesh parts effectively

- When a single part is used in multiple assemblies, you need to mesh that part only once.

- It enables concurrent engineering of FE Analysis. It is possible to mesh individual parts in an assembly simultaneously by different users at different locations.

- It reduces the time required to analyze large assemblies.

- It facilitates management of analysis data.
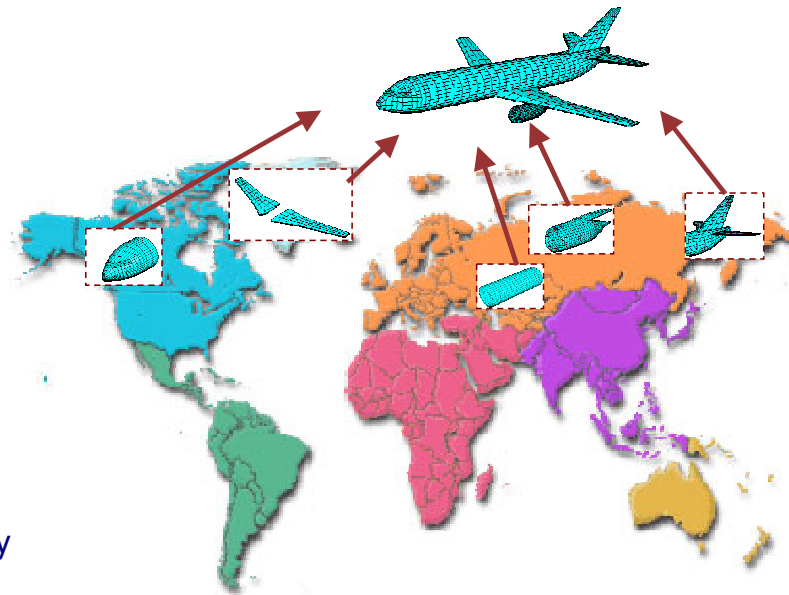
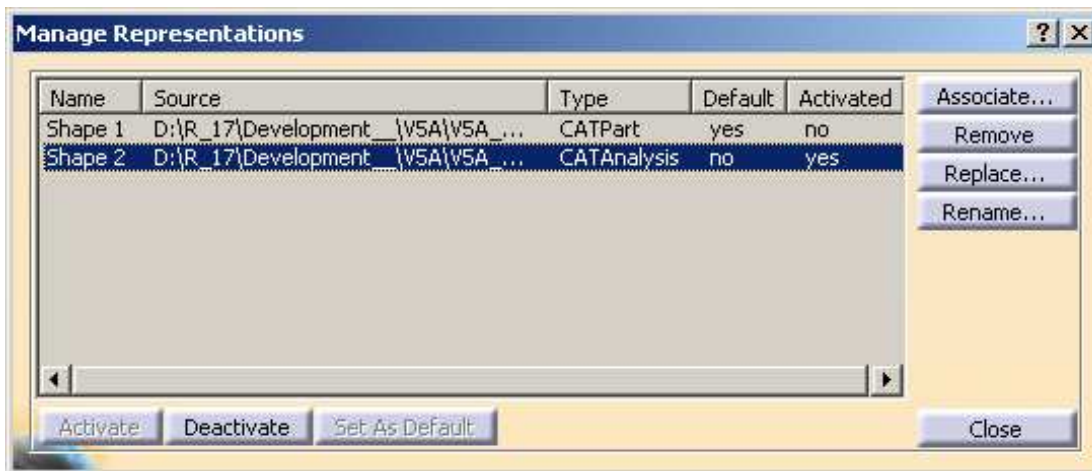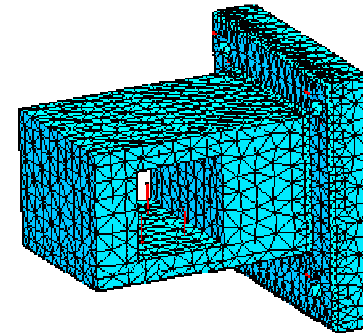# What is Shape Representation

The analysis document is an alternate shape representation of the Part document. Attaching the FE analysis document to a part is known as attaching shape representation to part. It is defined by using the 'Manage Shape Representation' option in the product's contextual menu.

- For a given part there can be more than one shape representation.
- You can attach any number of shape representations to a part.
- At a time only one shape can be active.

Part

Shape Representation

Manage Representations

Name	Source	Type	Default	Activated
Shape 1	D:\R_17\Development_\V5A\V5A ...	CATPart	yes	no
Shape 2	D:\R_17\Development_\V5A\V5A ...	CATAnalysis	no	yes

Associate...
Remove
Replace...
Rename...

Activate   Deactivate   Set As Default   Close
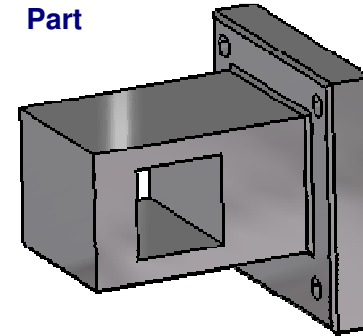
Attaching Shape to Part (1/2)

1. Click on the Part contextual Menu **Representations -> Manage Representations.**
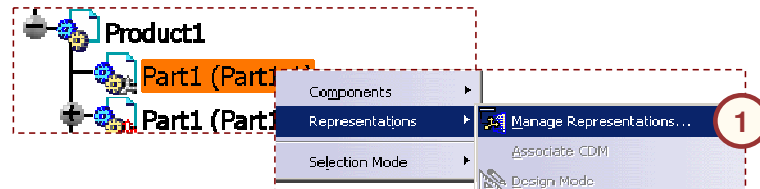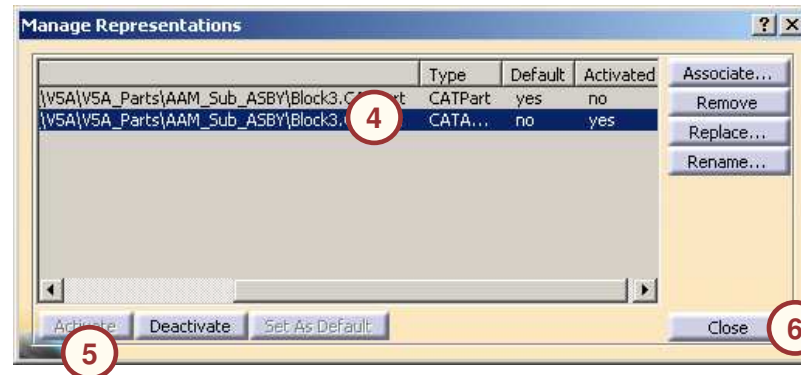

2. Click **Associate** button.
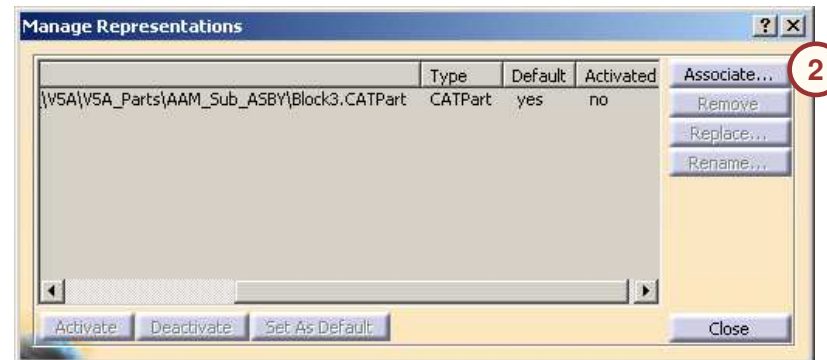
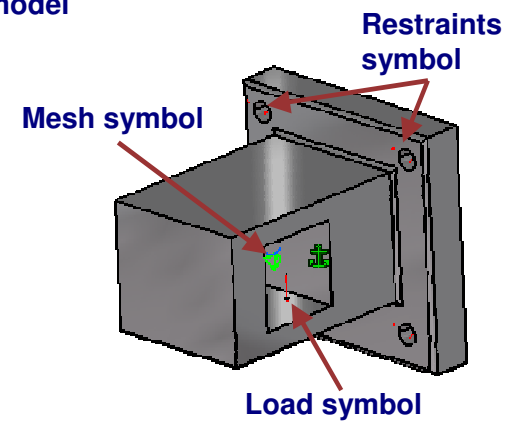3. Select desired CATAnalysis File (shape) related to that part.
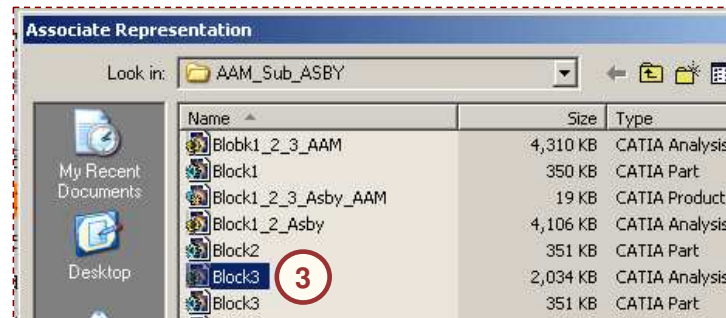
Attaching Shape to Part (2/2)

4. Select the same file in Manage Representation window.

5. Click on **Activate** button.

6. Click **Close** button.

Manage Representations

	Type	Default	Activated
\V5A\V5A_Parts\AAM_Sub_ASBY\Block3.C...rt	CATPart	yes	no
\V5A\V5A_Parts\AAM_Sub_ASBY\Block3.(	CATA...	no	yes

Associate...
Remove
Replace...
Rename...

Activate   Deactivate   Set As Default   Close
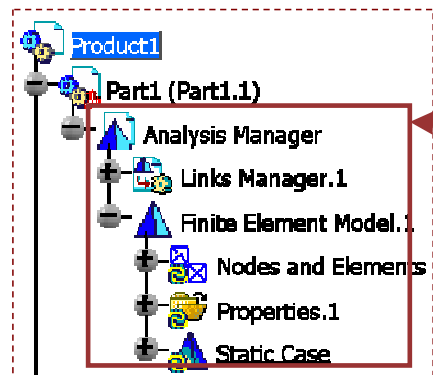
Mesh and FE properties information of part attached to part model

- Product1
  - Part1 (Part1.1)
    - Analysis Manager
      - Links Manager.1
      - Finite Element Model.1.
        - Nodes and Elements
        - Properties.1
        - Static Case

Restraints symbol

Mesh symbol

Load symbol

How to Use Analysis Assembly 2D Viewer

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. These changes in the Analysis Assembly document are updated using the Analysis Assembly 2D viewer.

1. Double-click Global **Analysis Manager** which is shown highlighted

2. Click on **Analysis Assembly 2D viewer** icon

3. Click on **Synchronize** button. The assembled analysis definition is updated.
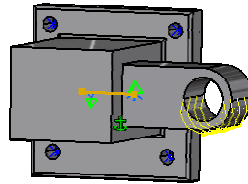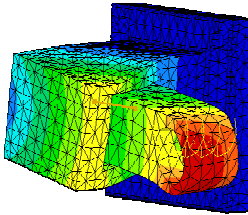
4. Click **OK.**
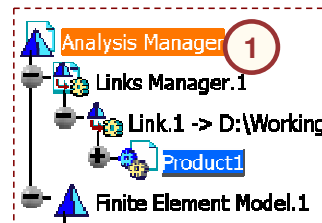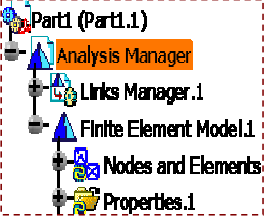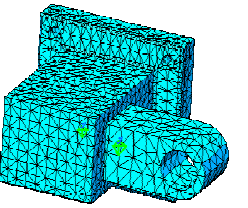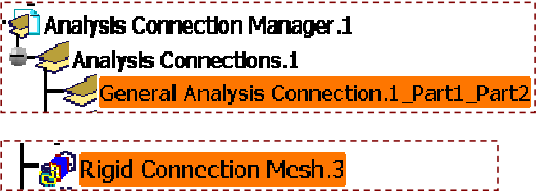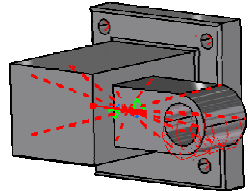
Red color shows newly attached shape is current shape
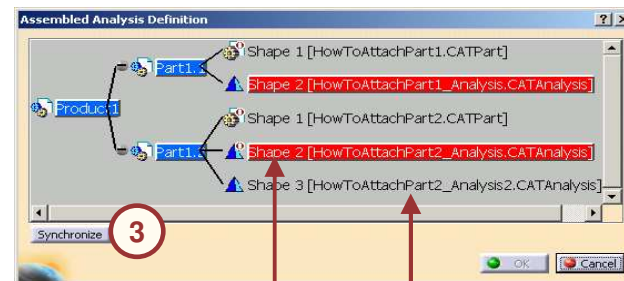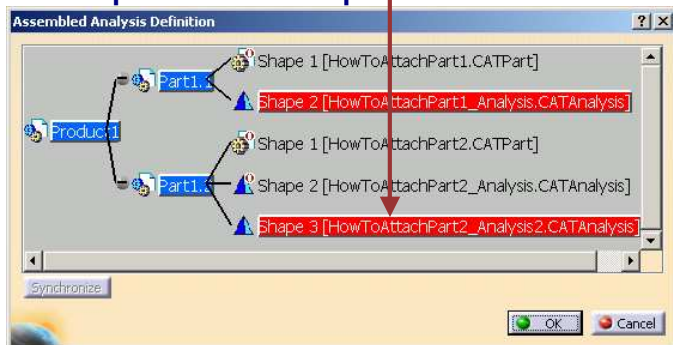
Earlier attached shape

Newly attached shape to same part to be updated

Analysis Assembly Overview

| Step No.	General Process	FEA Process Steps in GPS Workbench
1	**Attach shapes to parts in product**	Part1 (Part1.1) Analysis Manager Links Manager.1 Finite Element Model.1 Nodes and Elements Properties.1
2	**Assign connection and connection properties**	Analysis Connection Manager.1 Analysis Connections.1 General Analysis Connection.1_Part1_Part2 Rigid Connection Mesh.3
3	**Apply loads and constraints**	Static Case Restraints.1 Loads.1
4	**Perform global analysis and view results**	Static Case Solution.1 Translational displacement magnitude.1

Exercise 6A

Recap Exercise

15 min

In this exercise you will see which nodes are added to the specification tree in Assembly Structural Analysis. You will also learn the difference between the Assembly Analysis Approach and the Analysis Assembly Approach. Detailed instructions are provided for this exercise.

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

- **Identify different nodes in the specification tree for the analysis of an assembly**

- **Differentiate between specification tree structures of Assembly Analysis Approach and Analysis Assembly Approach**
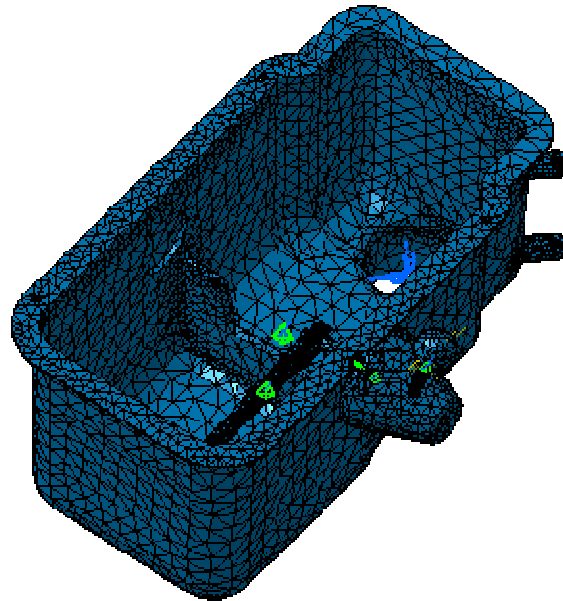
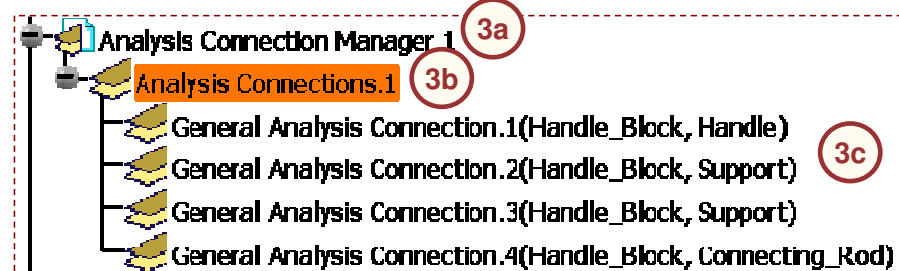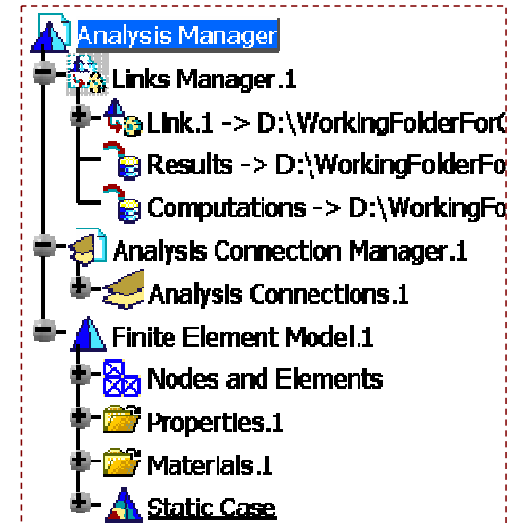# Exercise 6A (1/4)

1. **Open the Analysis Document.**
   - V5A_Ex_6A_Assembly_Analysis.CATAnalysis.
     a. This is the analysis document created by Assembly Analysis Approach (Traditional Approach)

2. **Observe the Assembly Structure.**
   - Use the Specification tree.
     a. Expand the nodes under the specification tree.

3. **Observe the different Analysis connections**
   - Use connection manager in the specification tree.
     a. Expand the node *Analysis Connection Manager.1*
     b. Expand *Analysis Connections.1*
     c. Observe the General analysis connections created between the assembly components.
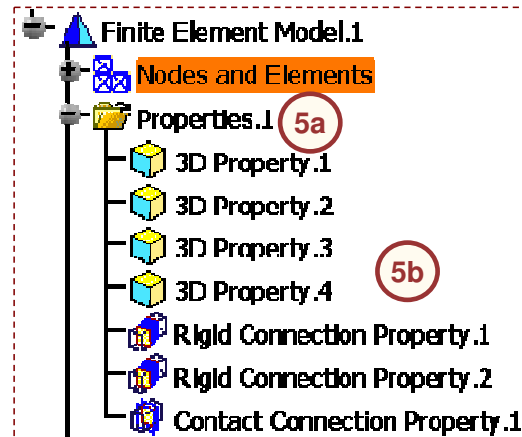
Exercise 6A (2/4)

4.   Observe the Mesh Information.

- Use the *Nodes and Elements.*

    a.   In the node *Finite Element model.1*, expand the *Nodes and Elements.*

    b.   It includes mesh information related to different components of the Assembly.

    c.   This also includes the Connection meshes created when the analysis connections are used between Assembly components.
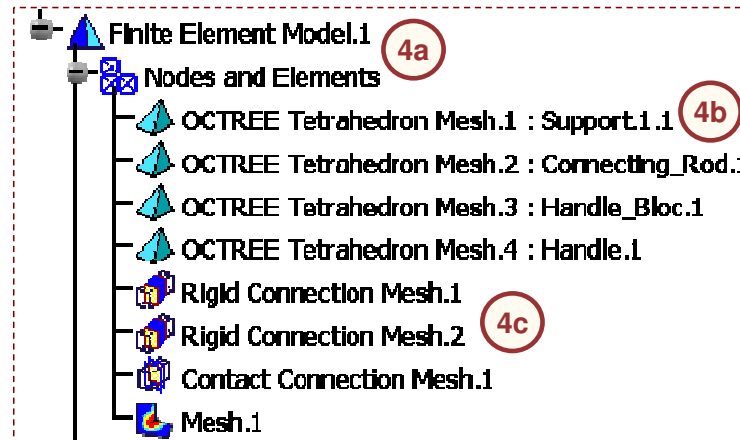
5.   Observe the properties.

- Use the *Properties* node.

    a.   In the node *Finite Element model.1*, expand the *Properties.1* node.

    b.   It contains the 3D properties of the components and also the connection properties applied to the connections within the components.
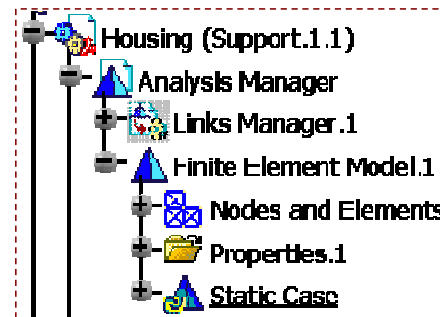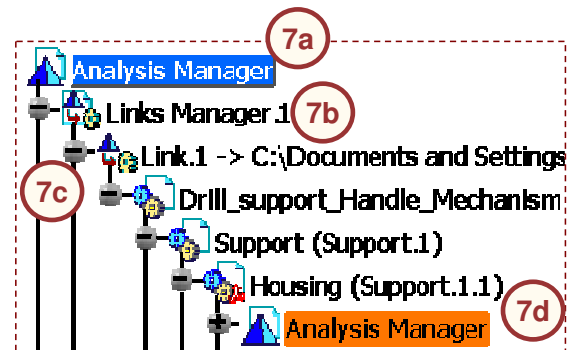
Exercise 6A (3/4)

6.   Open the Analysis Document.

- V5A_Ex_6A_Analysis_Assembly.CATAnalysis.

  a.   This document is created by Analysis Assembly Approach.

  b.   This Global Analysis Document contains the assembly of component level sub-analysis documents.

7.   Observe the Assembly Structure.

- Use the Specification tree.

  a.   Double click the *Analysis Manager*.

  b.   Expand the *Links Manager.1*.

  c.   Expand the *Link.1*. Model information is present at this level. When you expand Drill_Support_Handle_mechanism, you will see complete assembly structure.

  d.   In the node for every component in the assembly, there is one Analysis Manager. Sub-analysis level information is present at this link. You can observe this for every component.
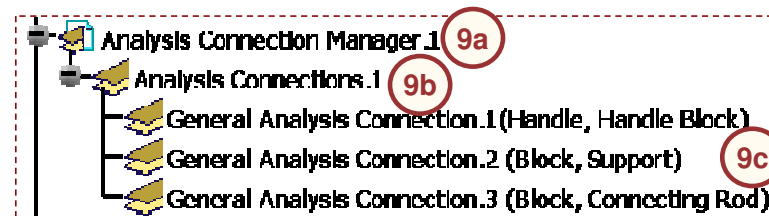
Exercise 6A (4/4)

8.   Observe the Mesh information.

- Use *Nodes and Elements.*
  - a.   Expand the *Finite Element Model.1.*
  - b.   Expand *Nodes and Elements.*
  - c.   This includes the Connection Meshes created when the analysis connections are used between Assembly components.
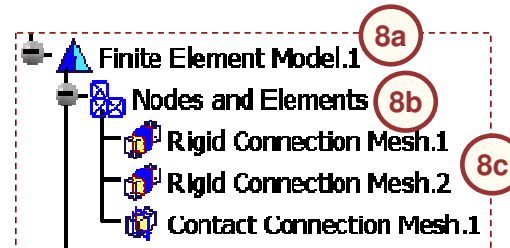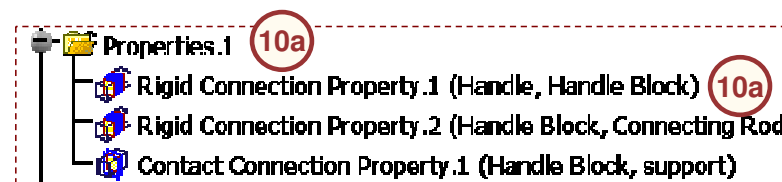
9.   Observe the different Analysis connections

- Use Analysis Connection Manager in the specification tree.
  - a.   Expand the node *Analysis Connection Manager.1*
  - b.   Expand *Analysis Connections.1*
  - c.   Observe the *General Analysis Connections* created between the assembly components.

10.   Observe the properties.

- Use the *Properties* node.
  - a.   Under the *Finite Element model.1*, expand the *Properties* node.
  - b.   It contains the information about connection properties created in global analysis document.

Finite Element Model.1 (8a)
  Nodes and Elements (8b)
    Rigid Connection Mesh.1
    Rigid Connection Mesh.2 (8c)
    Contact Connection Mesh.1

Analysis Connection Manager.1 (9a)
  Analysis Connections.1 (9b)
    General Analysis Connection.1 (Handle, Handle Block)
    General Analysis Connection.2 (Block, Support) (9c)
    General Analysis Connection.3 (Block, Connecting Rod)

Properties.1 (10a)
  Rigid Connection Property.1 (Handle, Handle Block) (10a)
  Rigid Connection Property.2 (Handle Block, Connecting Rod)
  Contact Connection Property.1 (Handle Block, support)

Exercise 6A: Recap

✓ **Identify different nodes in the specification tree for Analysis of Assembly.**

✓ **Differentiate between specification tree structures of Assembly Analysis Approach and Analysis Assembly Approach.**
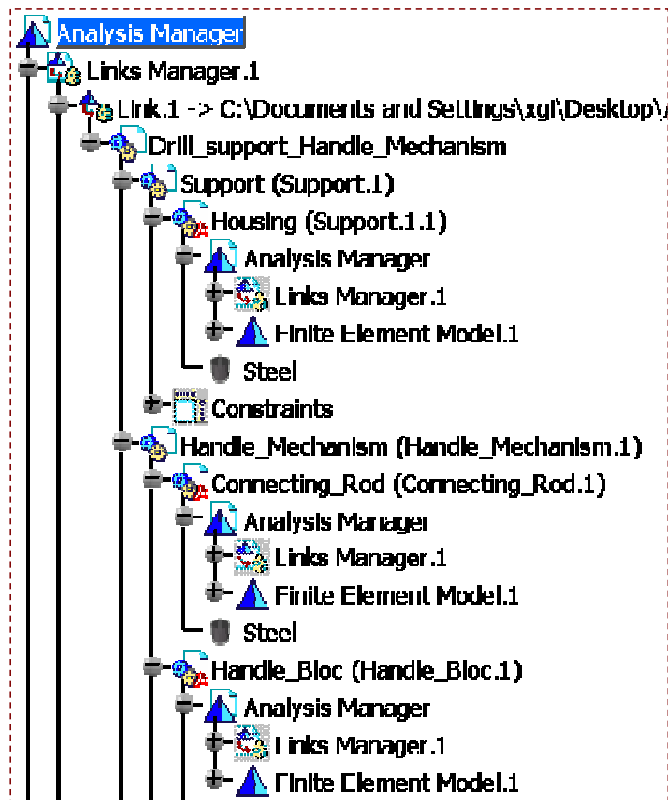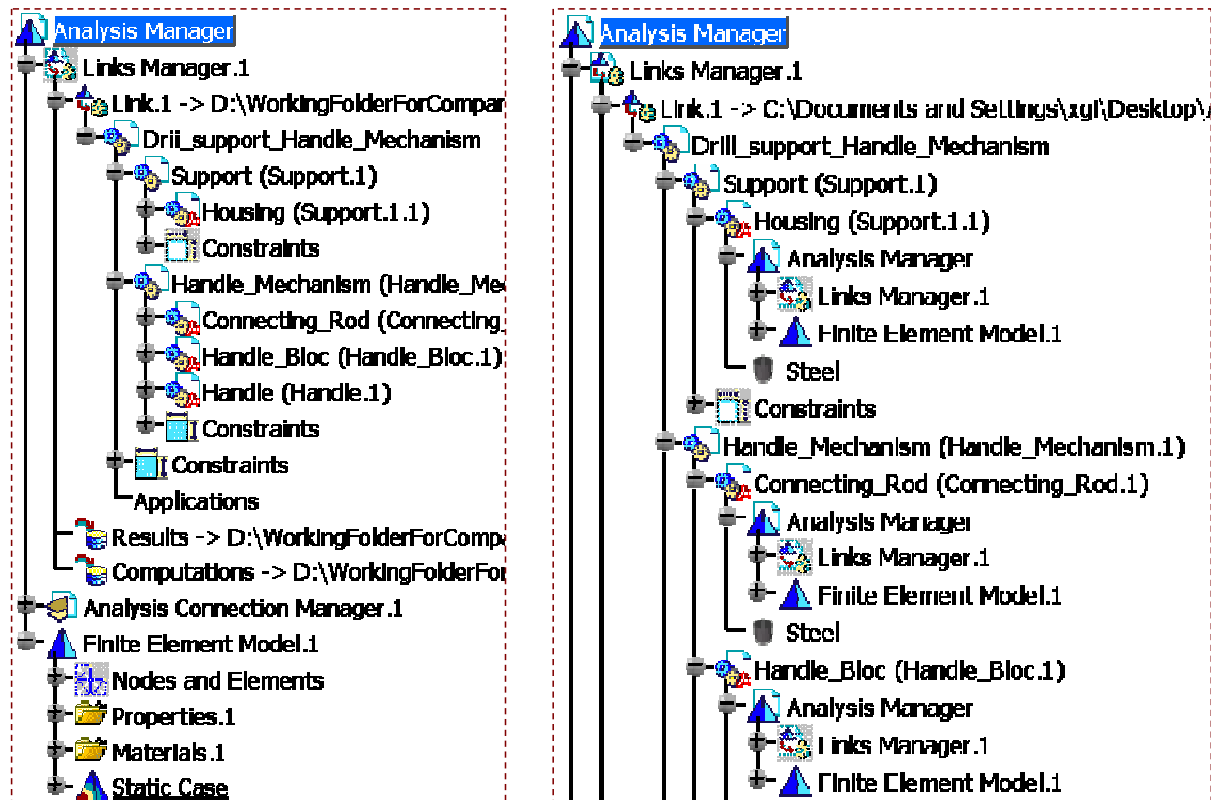
Step 3: Analysis Connections and Connection Properties

You will learn what is Analysis Connection and how to assign different connection properties to connect two components in assembly.

To study Generative Assembly Structural Analysis use the following steps:

1. Assembly Structural Analysis
2. Analysis Assembly
3. **Analysis Connections and Connection Properties**
4. Virtual Parts
5. Distributed Mass

Why Use Connections and Connection Properties

- In the finite element model, you must define how the parts of an assembly interact with each other. GAS connections are an easy way for you to define interactions between parts without modeling all the physical details of the joint. A wide variety of connection types and connection properties are provided to model physical assembly connections.

- When parts are connected to each other, the relationships between the translational and rotational degrees of freedom at the connection are well defined. The characteristics of these relationships are determined by the structural properties of the connection itself.

- For example, when two parts are connected with a spring joint, the spring stiffness plays a role in how forces are transmitted from one part to the other. This stiffness will dictate how the connected parts move relative to each other.

- You can utilize the constraints defined in assembly in order to define connection physical properties.
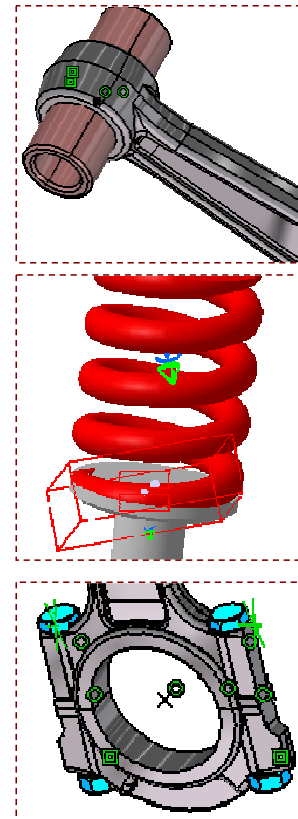
Using Assembly Constraints for Analysis Connections

You can use either assembly constraints or a corresponding analysis connection in order to define a connection property. Thus, if assembly constraints are already defined then there is no need to create an analysis connections as support for creating connection property.

You need to create an analysis connection if an assembly constraint is not available for a required joint. You will see which are the most appropriate constraints for each kind of connection.

| Constraints	Equivalent Analysis Connection
Coincidence constraint	General Analysis Connection
Contact constraint	Line Analysis Connection
Offset constraint	Point Analysis Connection
Fix constraint	Surface Analysis Connection

Applying General Analysis Connection

General Analysis Connection is used for connecting parts of an assembly, with or without handler point. This operation can be performed between any type of geometry.

Follow the steps given below to create a General Analysis Connection between a piston pin and connecting rod:

1. Click on the **General Analysis Connection** icon.

2. Select the outer surface of the Pin which is in contact with Rod as **First component.**

3. Select the inner surface of the Rod in contact with Pin as **Second component.**

4. Click **OK.**
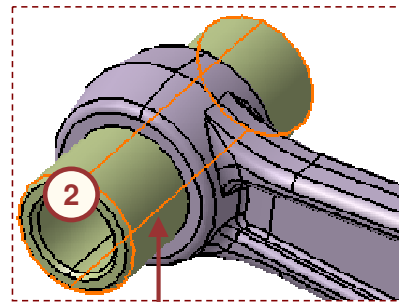
First Component as Outer surface of Pin
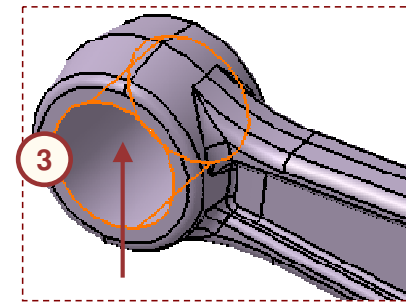
Second Component as inner surface of Rod
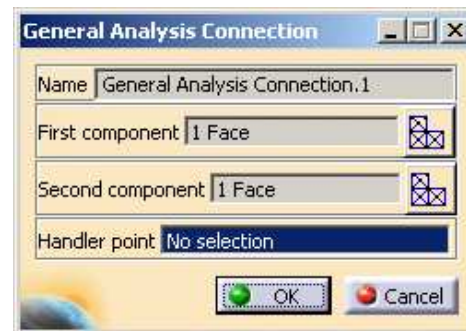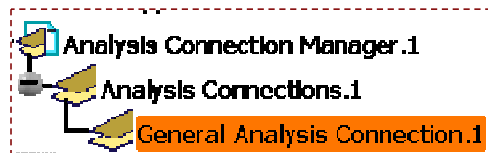
Other Analysis Connections

There are other types of Analysis connections under Analysis Support toolbar. These connections are used for the different purposes as shown in the table.

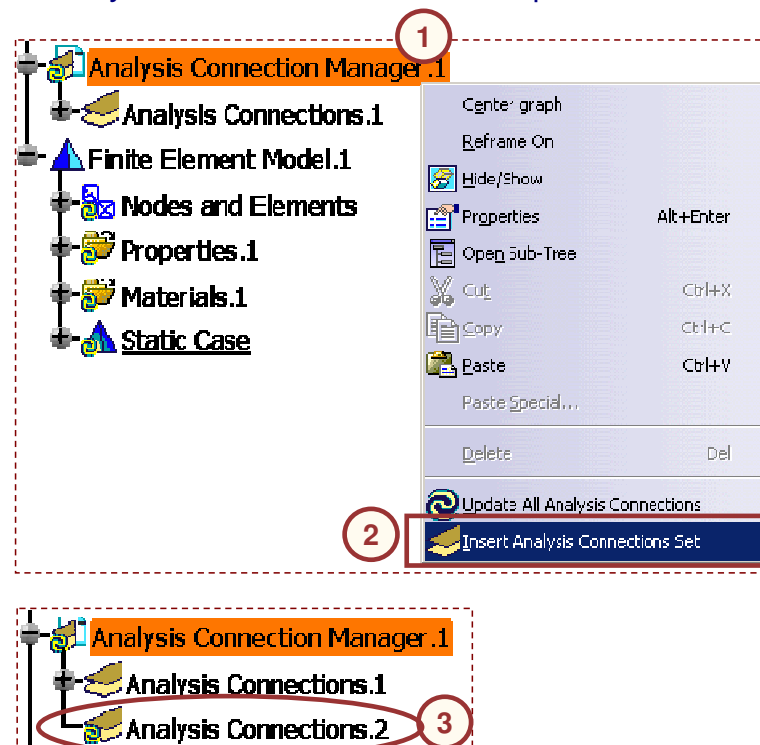| Analysis Connection	Use
Point Analysis Connection	Used for projecting welding points onto parallel faces, on an assembly model
Point Analysis Connection within one Part	Used for projecting welding points onto parallel faces of the same part
Line Analysis Connection	Used for simulating welding seam onto parallel faces, on an assembly model
Line Analysis Connection within one Part	Used for simulating welding seam onto parallel faces of the same part
Surface Analysis Connection	Used for simulating welding surface onto parallel faces, on an assembly model
Surface Analysis Connection within one Part	Used for simulating welding seam onto parallel faces of the same part
Points to Points Analysis Connection	Used to connect two sub-meshes
Point Analysis Interface	Used to create point analysis interface

Copyright DASSAULT SYSTEMES

Creating Set of Analysis Connections

You can create different sets of analysis connections under the Analysis Connection Manager. A set can contain different kinds of analysis connections. It is possible to group the analysis connections as per user convenience.

By default one set of analysis connections is present under the node Analysis Connection Manager. You can insert extra sets as per requirement and create analysis connection under the required set.

1. Right click on the *Analysis Connection Manager* node in the specification tree.

2. Select **Insert Analysis Connections Set** from the contextual menu.

3. Set of Analysis connections get added under the Analysis Connection Manager.

This Contextual menu is available only if at least one analysis connection is created.

Creating Analysis Connection under Specific Set

You can create an analysis connection under the required set of connections.

1. Click on the **General Analysis Connection** icon.

2. Select the set of analysis connections under which you want to create the analysis connection. (Select *Analysis connection.2*).

3. The General Analysis Connection dialog box appears. Name the analysis connection as General Analysis Connection Moving_Jaw and Body.

4. Select the First component and second component as per requirement.

5. Click **OK**. General Analysis connection appears under the chosen set of analysis connections
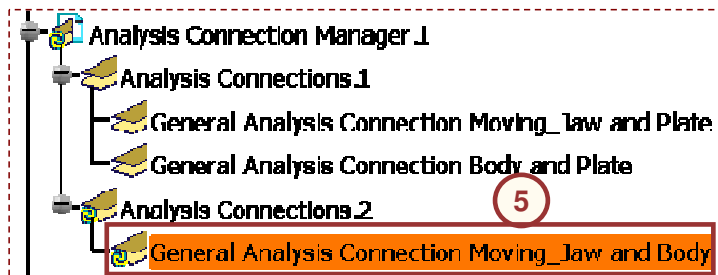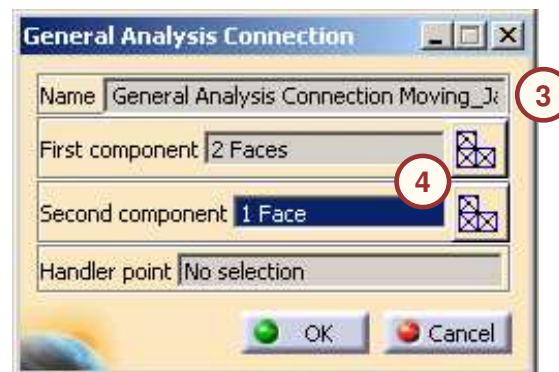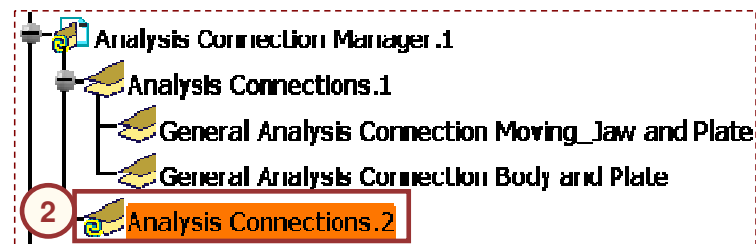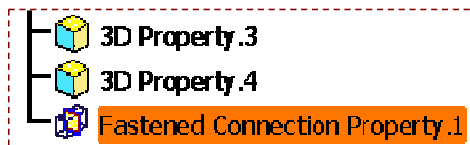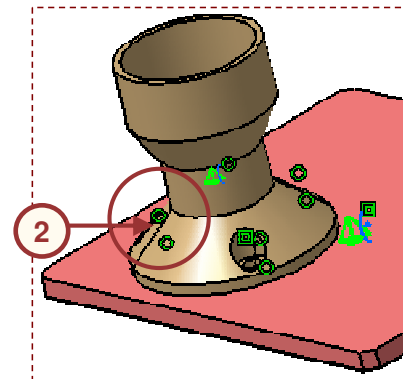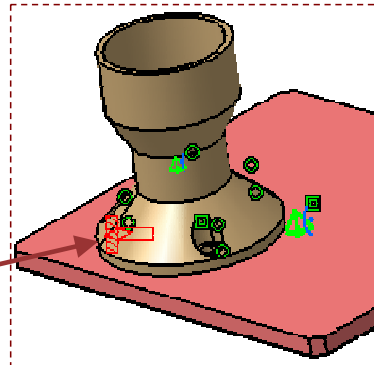
Applying a Fastened Connection Property

A Fastened Connection property is the link between two bodies which are fastened together at their common boundary. The two bodies behaves as if they were a single one, however they can have different material properties. A Fastened Connection takes into account the elastic deformability of the interface.

Follow the steps given bellow to apply fastened Connection Property.

1. Click on the **Fastened Connection** icon.

2. Select the constraint previously created in Product Design workbench or appropriate Analysis Connection in **Supports** field.

   Generally you can select one of the constraint from contact, coincidence and offset constraints, or a General Analysis Connection of any type except point/point type, as a support.

3. Click **OK.**

A symbol representing the Fastened Connection Property appears
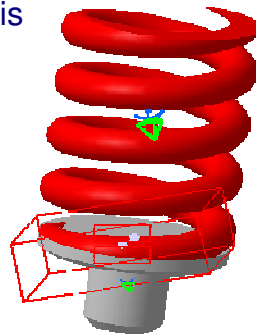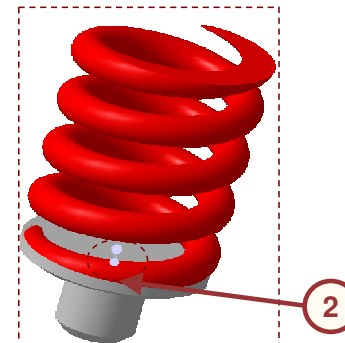
Applying Contact Connection Property

A Contact Connection is the link between the 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 into contact or within a user-specified normal clearance. When they come into contact or are at the clearance distance, they can still separate or slide relative to each other in the tangential plane, but they cannot come any closer than the user-specified clearance.

The Contact Connection is designed to handle the incompatible meshes and take into account the elastic deformability of the interfaces.

Follow the steps given bellow to apply Contact Connection Property.

1. Click the **Contact Connection** icon.

2. Select the constraint previously created in the Assembly Design/Analysis Connection workbench, for **Supports.**

   Generally you can select one of the constraint from contact, coincidence and offset constraints, or a General Analysis Connection of any type except point/point type, as a support.

3. Click **OK.**

A symbol representing the contact Connection appears
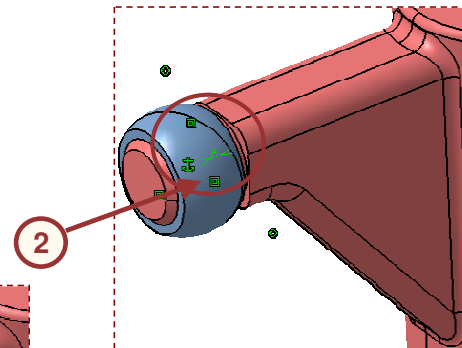
Applying Rigid Connection Property

A Rigid Connection is the link between two bodies which are stiffened and fastened together at their common boundary, and will behave as if their interface was infinitely rigid.

The Rigid Connection relations do not take into account the elastic deformability of the interfaces.

A central node, created at the midpoint between centroids of the two systems of points represented by the nodes of the two meshes, is connected by a rigid beam element to each node of the first and of the second meshes.

Follow the steps given bellow to apply Rigid Connection Property.

1. Click the **Rigid Connection** icon.

2. Select an assembly constraint : 'coincidence'

   Generally you can select one of the constraint from contact, coincidence and offset constraints, or a General Analysis Connection of any type, as a support.

3. Check the **Transmitted Degrees of Freedom** option to transmit some required Degrees of Freedom to the distant connection, if needed.

4. Click **OK**.
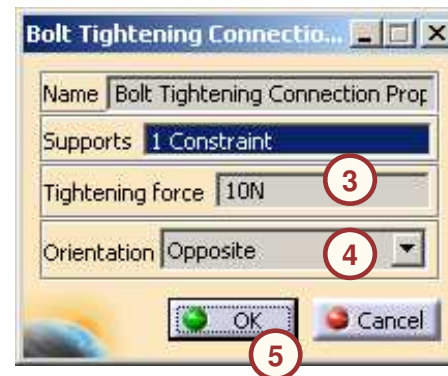
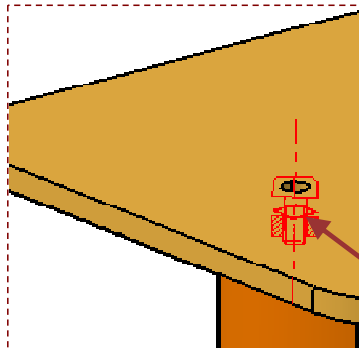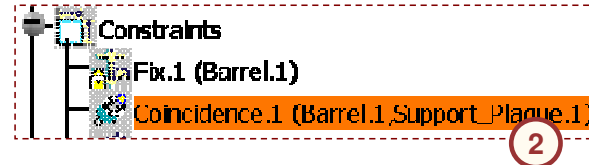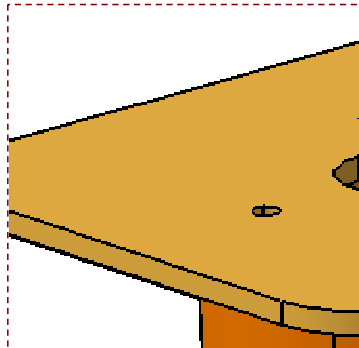**A symbol representing the Rigid Connection Property appears**
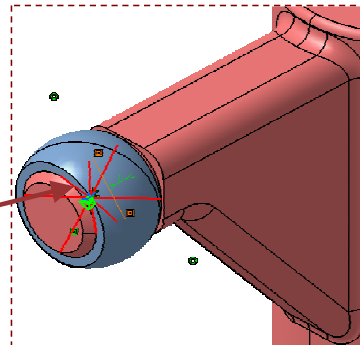
Applying Bolt Tightening Connection Property

The Bolt Tightening Connection is a connection that takes into account pre-tension in a bolted assembly.

Follow the steps given bellow to apply Bolt Tightened Connection Property.

1. Click the **Bolt Tightening Connection Property** icon.

2. Select the Surface **contact constraint** previously created (the one between the screw and the board's hole).

    Generally you can select one of the constraint from contact, coincidence and offset constraints, or a General Analysis Connection of any type, as a support.

3. Enter the value of **Tightening force.**

4. Select the **Orientation** as **Same** or **Opposite** as per requirement.

5. Click **OK**.

Face Face Connection Prope... ⊠

①

Constraints
Fix.1 (Barrel.1)
Coincidence.1 (Barrel.1,Support_Plaque.1) ②

Bolt Tightening Connectio... ⊟ □ ⊠
Name | Bolt Tightening Connection Prop
Supports | 1 Constraint
Tightening force | 10N ③
Orientation | Opposite ④ ▼
OK | Cancel ⑤

Properties.1
3D Property.1
3D Property.2
Bolt Tightening Connection Property.1

A symbol representing the Bolt Tightening Connection Property appears
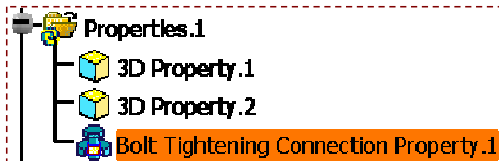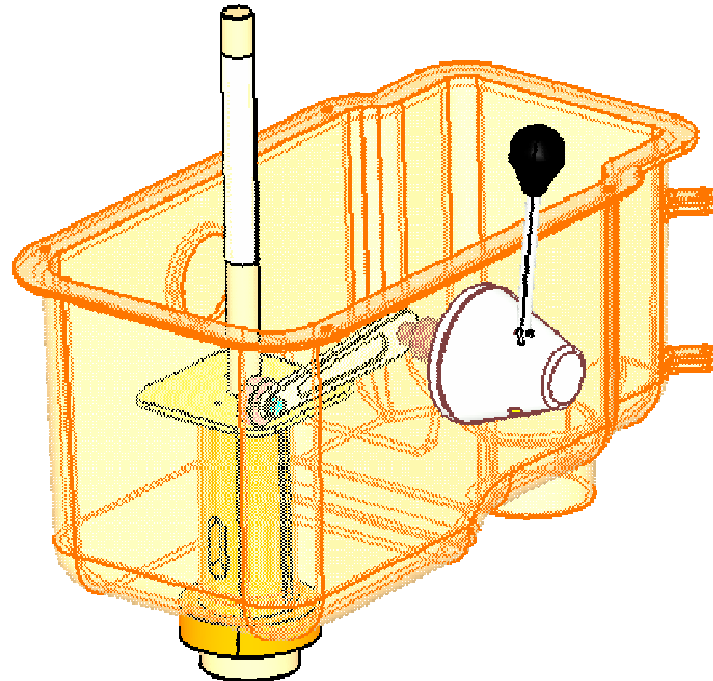
Exercise 6B

Recap Exercise

30 min

In this exercise, you will use Analysis Connections and Analysis Connection Properties. Detailed instructions are provided for this exercise.

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

- **Create General Analysis Connections**

- **Create Analysis Connection Properties using General Analysis Connections**

- **Create Analysis Connection Properties using Assembly Constraints**
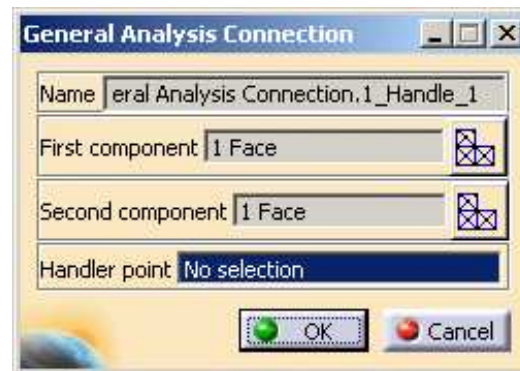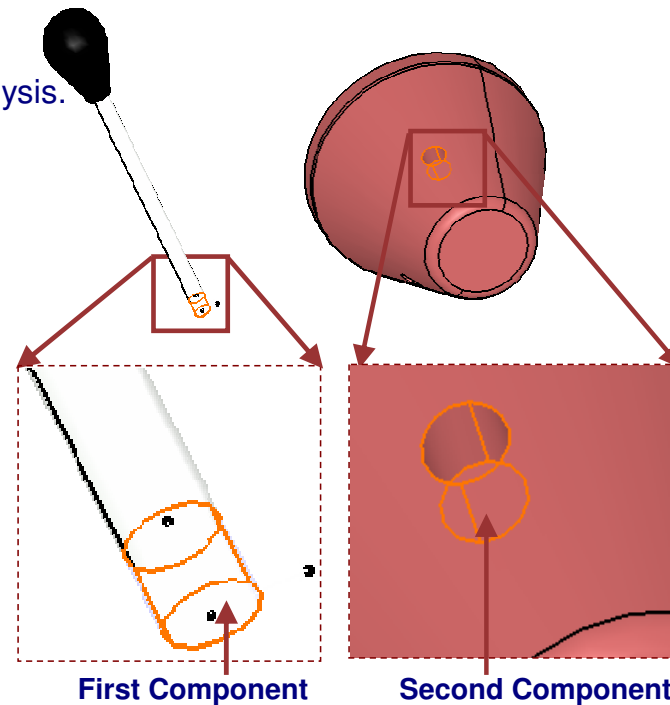
Exercise 6B (1/11)

1. **Open a Analysis Document.**

   ▪ V5A_Ex_6B_Analysis_Connections_Start.CATAnalysis.

2. **Create Assembly Analysis Connection between Handle and Handle Block.**

   ▪ Use General Analysis Connection for Handle and Handle Block.

   a. In **Analysis Connections** toolbar, click on **General Analysis Connection**. Name the Connection as General Analysis Connection.1_Handle_1 in **Name** field.

   b. Select the surface of *Handle.1* which is in contact with Handle Block as **First Component** (Hide the *Handle_Bloc(Handle_Bloc.1)* to select the exact surface.)

   c. Select the cylindrical surface of hole in Block surface as shown in **Second Component** (Hide the *Handle(Handle.1)* to select the exact surface).

   d. Click **OK**.

First Component   **Second Component**

Exercise 6B (2/11)

3. **Assign Connection Property to the created connection.**

   ▪ Assign Rigid Connection Property to General Analysis Connection.

     e. In **Connection Properties** toolbar, click **Rigid Connection Property** icon.

     f. In **Name** field enter **Rigid Connection Property Handle and Handle Block.**

     g. Select **Supports** field, then in *Analysis Connection Manager.1 > Analysis Connections.1*, select *General Analysis Connection.1_Handle_1*.

     h. Click **OK**.
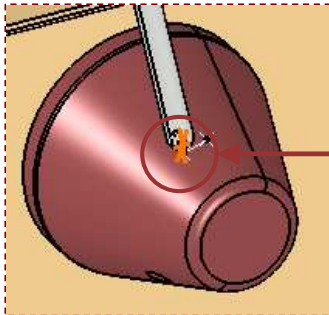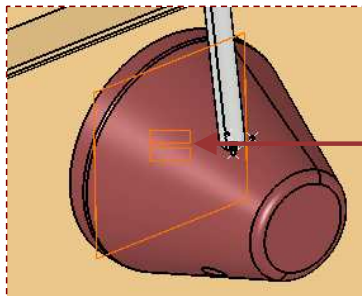
A symbol representing the Rigid Connection Property

Exercise 6B (3/11)

4. **Create Contact Connection between Handle Block and Housing.**

- Use Surface Contact assembly constraint to apply Contact Connection Property.

  a. In **Face To Face Connection Properties** toolbar, click on **Contact Connection Property** icon.

  b. In **Name** field enter **Contact Connection Property Handle Block and Housing.**

  c. Select **Supports** field, then click the *Surface Contact.12* Assembly constraint from assembly level constraints as shown.
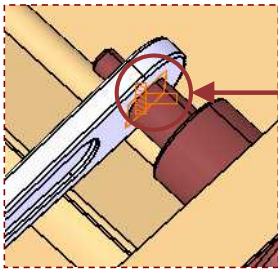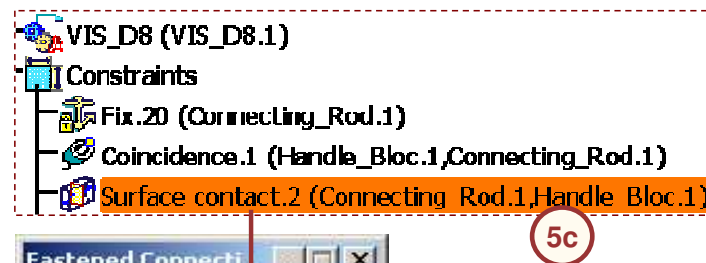
  d. Click **OK**.

A symbol representing the Contact Connection Property

Exercise 6B (4/11)

5. **Create Fastened Connection between Handle Block and Connection Rod.**

   ▪ Use Surface Contact assembly constraint to apply Fastened Connection Property.

   a. In **Face To Face Connection Properties** toolbar, click on **Fastened Connection Property** icon.

   b. In **Name** field enter *Fastened Connection Property Handle Block and Connection Rod.*

   c. Select **Supports** field click the *Surface Contact.2* Assembly constraint in *Handle_Mechanism* sub-assembly in product structure.
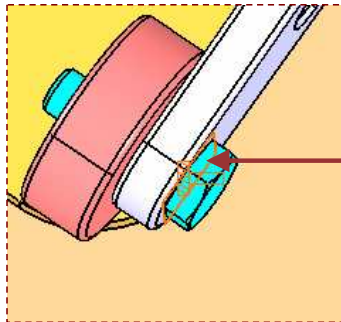
   d. Click **OK**.

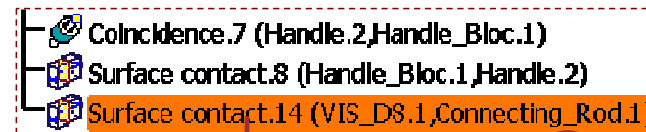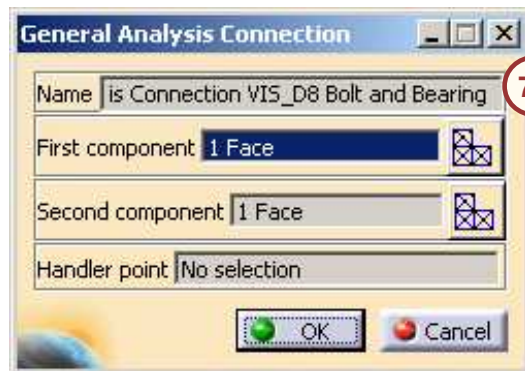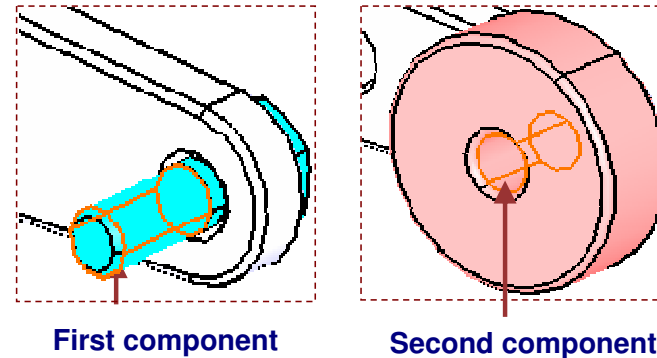**A symbol representing the Fastened Connection Property**

Exercise 6B (5/11)

6. **Create Fastened Connection between Connection Rod and VIS_D8 Bolt.**

   ▪ Use Surface Contact assembly constraint to apply Fastened Connection Property.

   a. In **Face To Face Connection Properties** toolbar, click on **Fastened Connection Property** icon.

   b. In **Name** field enter *Fastened Connection Property Connection Rod and VIS_D8 Bolt.*

   c. In **Supports** field click the *Surface Contact.14* Assembly constraint in *Handle_Mechanism* sub-assembly in product structure.

   d. Click **OK**.

Coincidence.7 (Handle.2,Handle_Bloc.1)
Surface contact.8 (Handle_Bloc.1,Handle.2)
Surface contact.14 (VIS_D8.1,Connecting_Rod.1)   (6c)

Fastened Connecti...
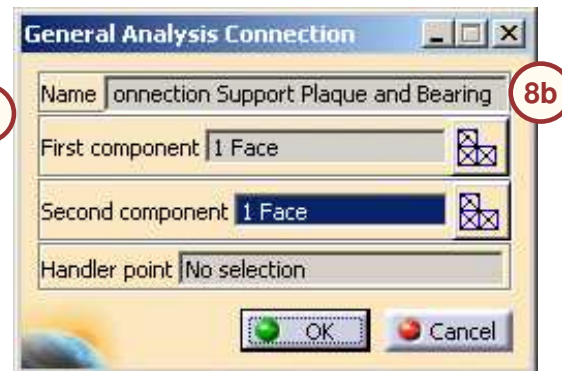Name | n Rod and VIS_D8 Bolt   (6b)
Supports | 1 Constraint   (6c)
OK   Cancel

Rigid Connection Property Handle and Handle Block
Contact Connection Property Handle Block and Housing
Fastened Connection Property Handle Block and Connection Rod
Fastened Connection Property Connection Rod and VIS_D8 Bolt

A symbol representing the Fastened Connection Property

Exercise 6B (6/11)

7. **Create Fastened Connection between VIS_D8 Bolt and Bearing.**

   ▪ Use General Analysis Connection to apply Fastened Connection Property.

   a. Create a **General Analysis Connection** between VIS_D8 bold upper surface and Bearing inner surface.

   b. Name the connection as *General Analysis Connection VIS_D8 Bolt and Bearing.*

   c. Create **Fastened Connection Property** with name as *Fastened Connection Property VIS_D8 Bolt and Bearing.*

   d. Assign **Fasten Connection Property** with **General Analysis Connection** as support.

   e. Click **OK**.

First component   **Second component**
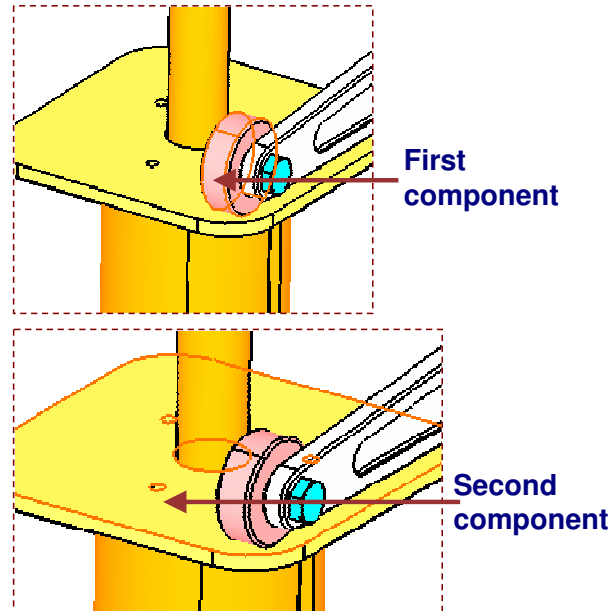

Rigid Connection Property Handle and Handle Block
Contact Connection Property Handle Block and Housing
Fastened Connection Property Handle Block and Connection Rod
Fastened Connection Property Connection Rod and VIS_D8 Bolt
Fastened Connection Property VIS_D8 Bolt and Bearing

General Analysis Connection

Name | is Connection VIS_D8 Bolt and Bearing   **7b**

First component | 1 Face

Second component | 1 Face

Handler point | No selection

OK   Cancel

Fastened Connecti...

7c Name | IS_D8 Bolt and Bearing

Supports | 1 Analysis connectio **7d**

OK   Cancel

Exercise 6B (7/11)

8. **Create Contact Connection between Support Plaque and Bearing.**

- Use General Analysis Connection to apply Contact Connection Property.

  a. Create a **General Analysis Connection** between Support Plaque upper surface and Bearing outer surface.

  b. Name connection as *General Analysis Connection Support Plaque and Bearing.*

  c. Create **Contact Connection Property** with **Name** as *Contact Connection Property Support Plaque and Bearing.*

  d. Assign **Contact Connection Property** with **General Analysis Connection** as support.

  e. Click **OK**.

First component

Second component

Exercise 6B (8/11)

9. **Create Contact Connection between Support Plaque and Barrel.**

   ▪ Use Surface Contact assembly constraint to apply Contact Connection Property.

      a. In **Face To Face Connection Properties** toolbar, click on **Contact Connection Property** icon.

      b. Name Connection Property as *Contact Connection Property Support Plaque and Barrel.*

      c. In **Supports** field select the *Surface Contact.13* in *Drill support* sub-assembly constraint as shown in image.
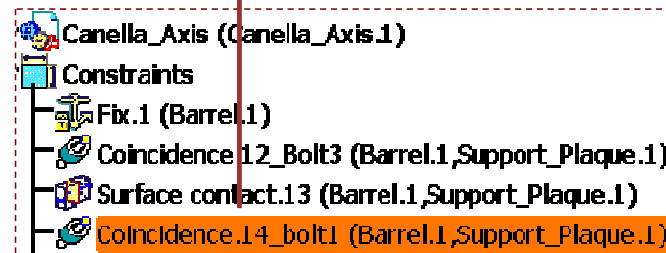
      d. Click **OK**.
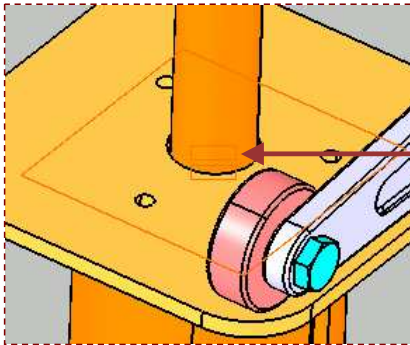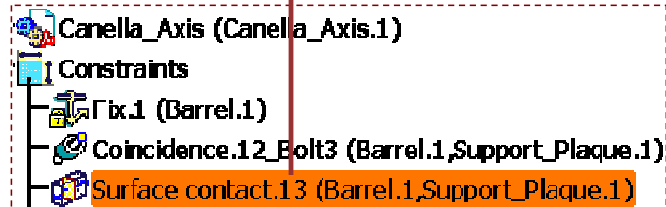
**A symbol representing the Contact Connection Property**

Exercise 6B (9/11)

10. Create Fastened Connection between Support Plaque and Barrel Holes.

- Use coincidence assembly constraint to apply Fastened Connection Property.

  a. In **Face To Face Connection Properties** toolbar, click on **Fastened Connection Property** icon.

  b. Name Connection Property as *Fastened Connection Property Support Plaque and Barrel Hole1.*

  c. In **Supports** field, click the *Coincidence.14_bolt1* Assembly constraint from *Drill Support sub-assembly.*

  d. Click **OK**.

- In the similar way, create Fastened Connection Property using *Coincidence.15_bolt2* and *Coincidence.12_bolt3.*
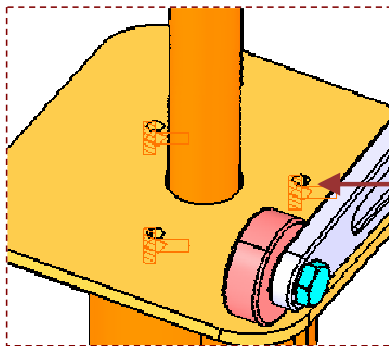
A symbol representing the Fastened Connection Property
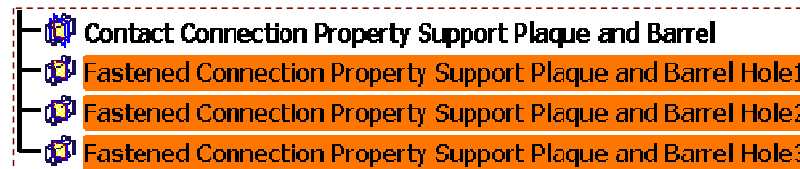
Exercise 6B (10/11)

11. **Create Contact Connection between Barrel and Support Housing.**

   ■ Use Surface Contact assembly constraint to apply Contact Connection Property.

   a. In **Face To Face Connection Properties** toolbar, click on **Contact Connection Property**.

   b. Name the Connection Property as *Contact Connection Property Barrel and Support Housing*

   c. In the **Supports** field, click the *Surface Contact.34* Assembly constraint from Assembly level constraints.

   d. Click **OK**.
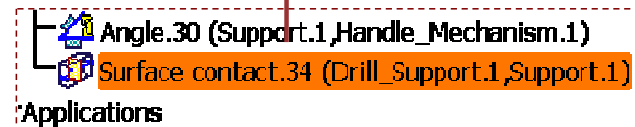
Exercise 6B (11/11)

12. **Create Rigid Connection between Support Plaque and Candela Axis.**

   ▪ Use General Analysis Connection to apply Rigid Connection Property.

   a. Create a **General Analysis Connection** between Support Plaque and Candela Axis as shown.

   b. Name connection as *General Analysis Connection Support Plaque and Candela Axis.*

   c. Create **Rigid Connection Property** with **Name** as *Rigid Connection Property Support Plaque and Candela Axis.*

   d. Assign **Rigid Connection Property** with **General Analysis Connection** as support.
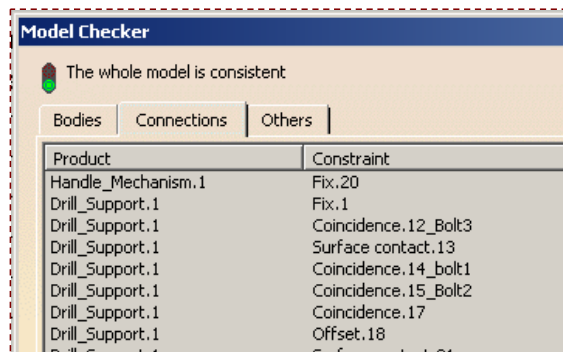
   e. Click **OK**.
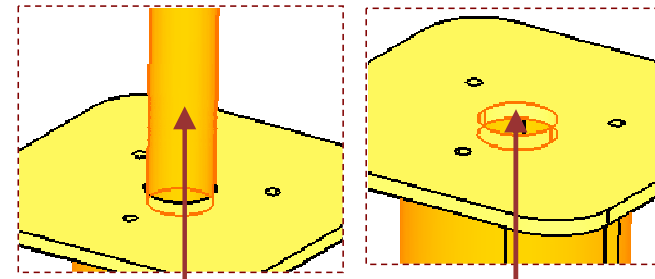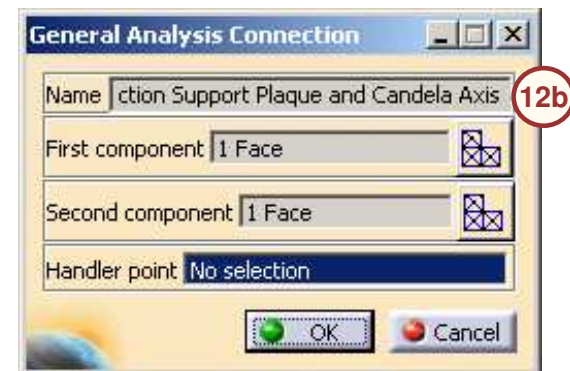
13. **Check All Connections using Model Checker.**

   ▪ Use Model Checker **Connections** Tab.

First component   **Second component**

Exercise 6B: Recap

✓ **Create General Analysis Connections.**

✓ **Create Analysis Connection Properties using General Analysis Connections.**

✓ **Create Analysis Connection Properties using Assembly Constraints.**

OCTREE Tetrahedron Mesh.10 : Support_Plaque.1
OCTREE Tetrahedron Mesh.11 : Canella_Axis.1
Rigid Connection Mesh.1
Contact Connection Mesh.1
Fastened Connection Mesh.1
Fastened Connection Mesh.2
Fastened Connection Mesh.3
Contact Connection Mesh.2
Contact Connection Mesh.3
Fastened Connection Mesh.4
Fastened Connection Mesh.5
Fastened Connection Mesh.6
Contact Connection Mesh.4
Rigid Connection Mesh.2
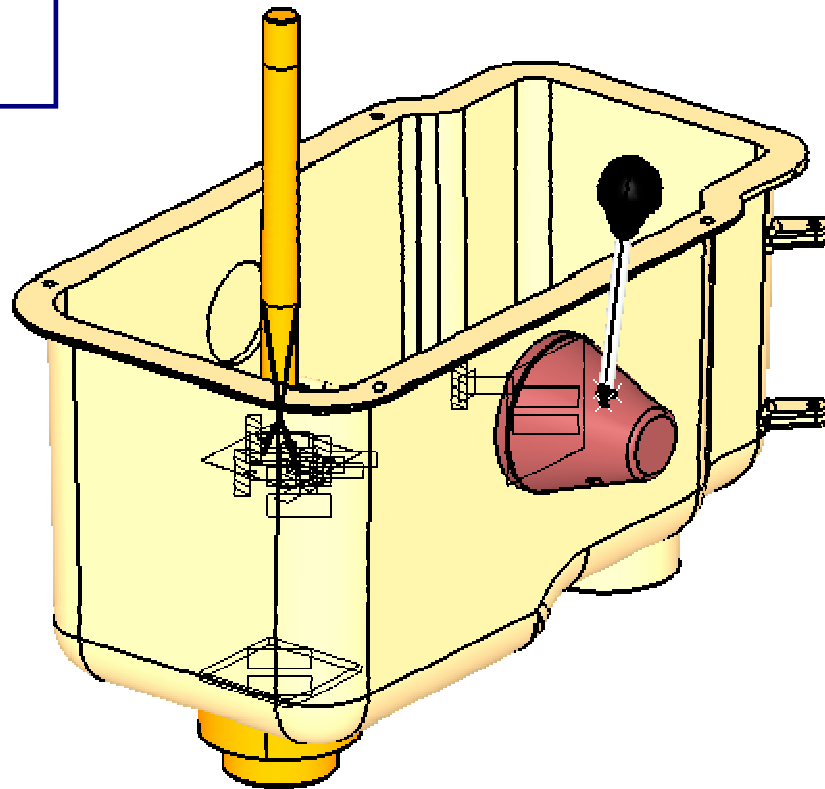
Step 4: Virtual Parts

You will learn what are virtual parts and how to use them.

To study Generative Assembly Structural Analysis use the following steps:

1. Assembly Structural Analysis
2. Analysis Assembly
3. Analysis Connections and Connection Properties
4. **Virtual Parts**
5. Distributed Mass

What is Virtual Part (1/2)

Virtual Parts represent bodies for which no geometry model is available, but which play a role in the structural analysis of single part or assembly systems. Virtual Parts are connected to component in assembly and used to transmit displacement at a distance. Following are different types of Virtual Parts.

| Virtual Part	Icon	Description
Rigid Virtual Part		**Rigid body acts as a mass-less rigid object which stiffly transmit actions (masses, restraints, loads), while locally stiffening the deformable part to which it is attached. It does not take into account the elastic deformability of the parts to which it is attached.**
Smooth Virtual Part		**Rigid body acts as a mass-less rigid object which softly transmit actions, without stiffening the deformable part to which it is attached. It does approximately take into account the elastic deformability of the parts to which it is attached.**

What is Virtual Part (2/2)

| Virtual Part	Icon	Description
Contact Virtual Part		**Rigid body acts as a mass-less rigid object which transmit actions, while preventing from body inter-penetration and thus without stiffening the deformable part to which it is attached. It does take into account the elastic deformability of the parts to which it is attached.**
Rigid Spring Virtual Part		**Elastic body acts as a 6 DOF spring in series with a mass-less rigid body which will stiffly transmit actions, while stiffening the deformable part to which it is attached. It does not take into account the elastic deformability of the parts to which it is attached.**
Smooth Spring Virtual Part		**Elastic body acts as a 6 DOF spring in series with a mass-less rigid body which will softly transmit actions, without stiffening the deformable part to which it is attached. It does approximately take into account the elastic deformability of the parts to which it is attached.**

Creating Rigid Virtual Part

You will see how to create a rigid virtual part to transmit the action.

Follow the steps given below to create Rigid Virtual Part.

1. Click on **Rigid Virtual Part** icon in **Virtual Parts** toolbar.
2. Select the required face in the **support** field.
3. In the **Handler** field select the Handler point through which the action will transmit.
4. Click **OK.**
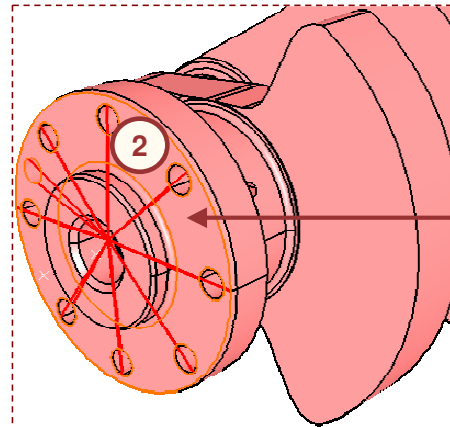
Select Face as support
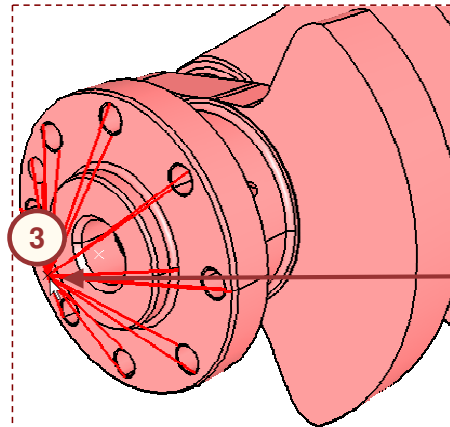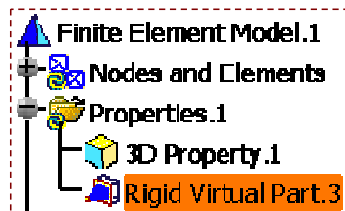
Select Handler point if required

Step 5: Distributed Mass

You will learn what is a distributed mass and how to use it.

To study Generative Assembly Structural Analysis use the following steps:

1. Assembly Structural Analysis
2. Analysis Assembly
3. Analysis Connections and Connection Properties
4. Virtual Parts
5. **Distributed Mass**

What is a Distributed Mass

The Distributed Mass is used to represent purely inertial systems such as additional equipment.
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 of the virtual part. To consider the weight of the virtual part, acceleration is coupled with the mass.

The distribution behavior depends on the selected support.

| Selected Support	Distribution Behavior
Punctual geometries OR Spatial groups	The mass is equally distributed along each node of the selection
1D geometries/groups	The mass is translated along equivalent Line Mass Density
2D geometries/groups	The mass is translated to equivalent Surface Mass Density
Virtual parts	The total mass is concentrated to Handler point of the virtual part

Creating Distributed Mass

You will see how to create a Distributed Mass.

1. Click on the **Distributed Mass** icon.

2. Select the handler point as **Supports.**

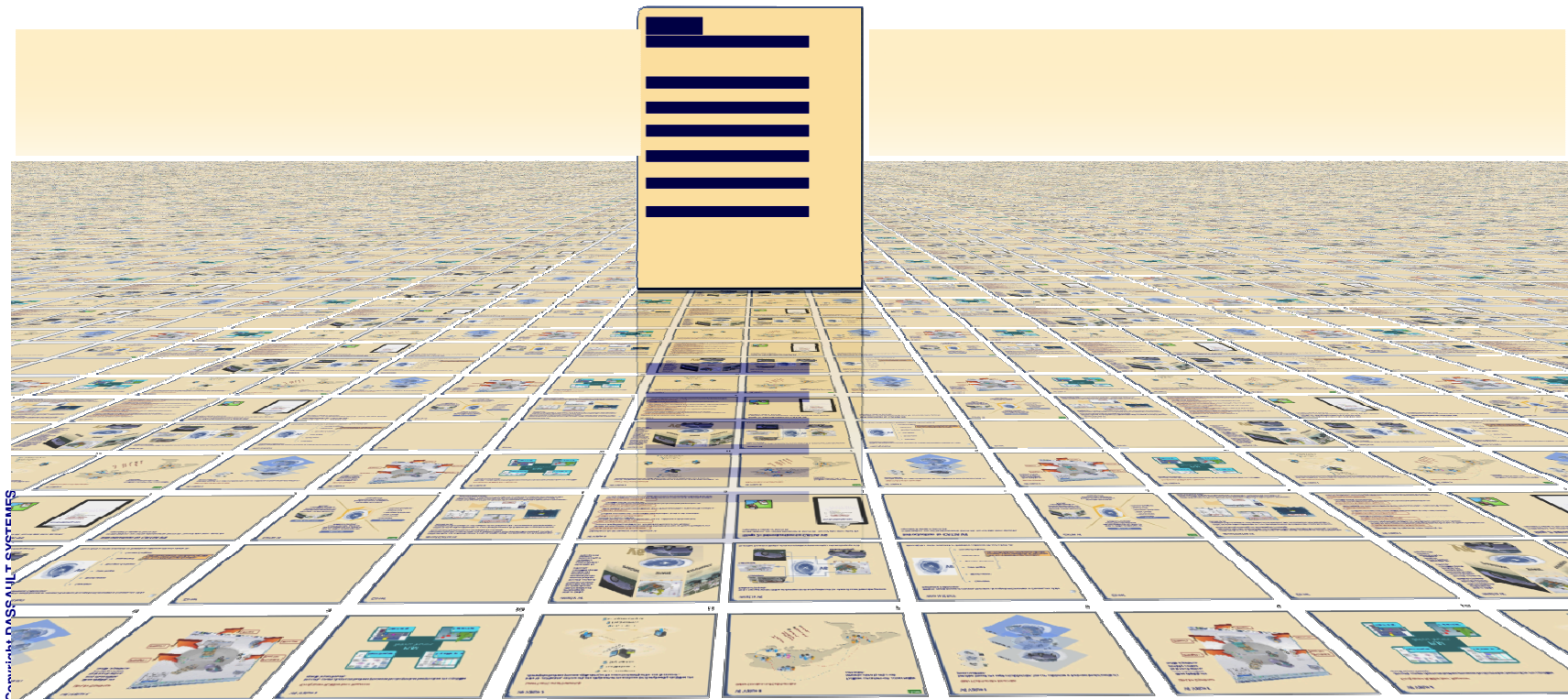3. Enter the value of **Mass**.

4. Click **OK.**
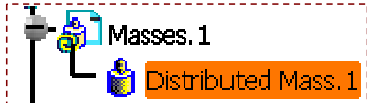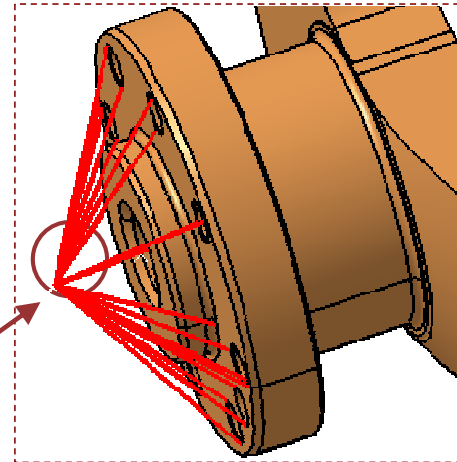
A symbol representing the Distributed Mass appears
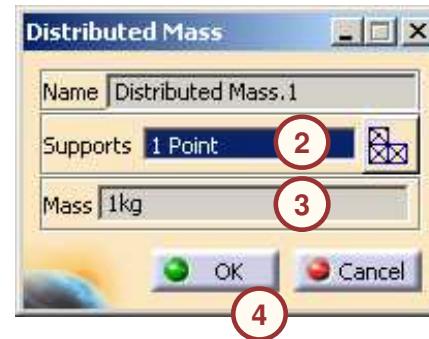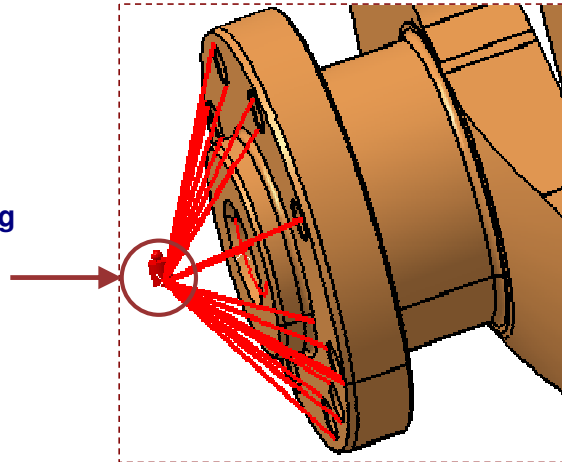
Student Notes:

To Sum Up

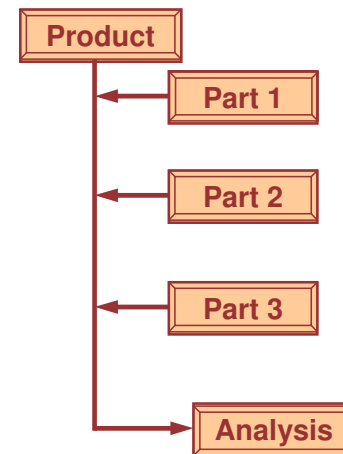In the following slides you will find a summary of the topics covered in this lesson.

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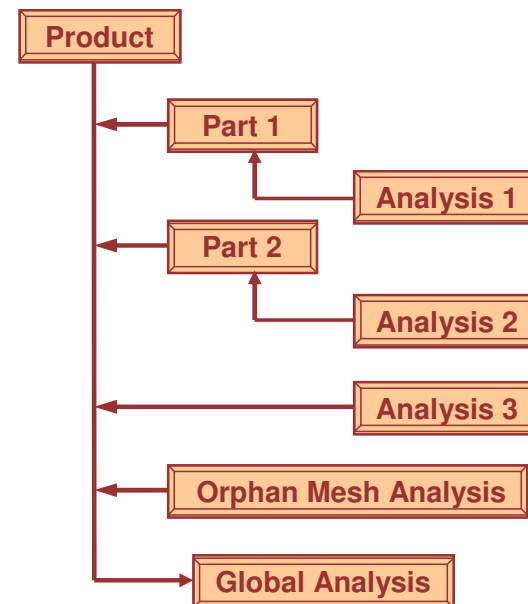
To simulate the real connection behavior you can define connections between different assembly components and assign different types of connection properties to these connections, using GAS license through Generative Structural Analysis workbench. You can also use 'assembly constraints' as a support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

Product

Part 1

Part 2

Part 3

Analysis

Assembly Analysis Approach

Product

Part 1

Analysis 1

Part 2

Analysis 2

Analysis 3

Orphan Mesh Analysis

Global Analysis

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.
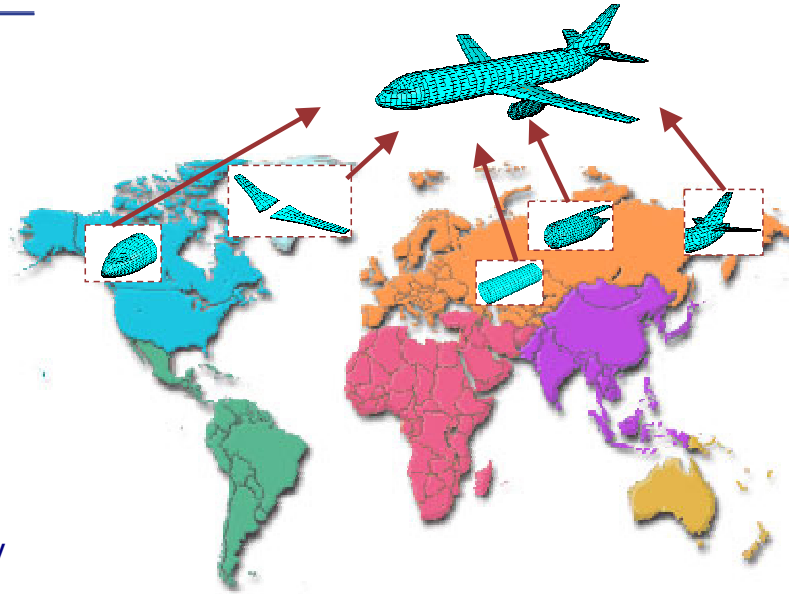
Analysis Connections and Connection Properties

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.
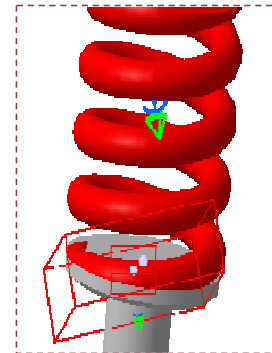
Virtual Parts

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
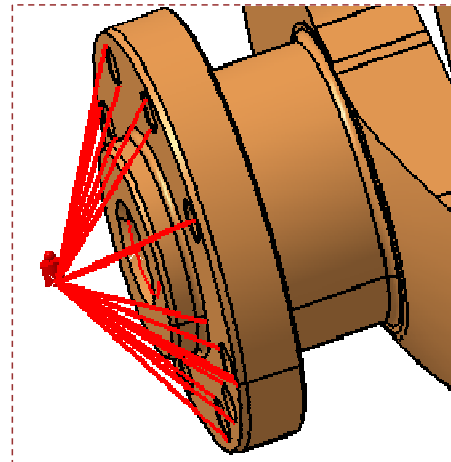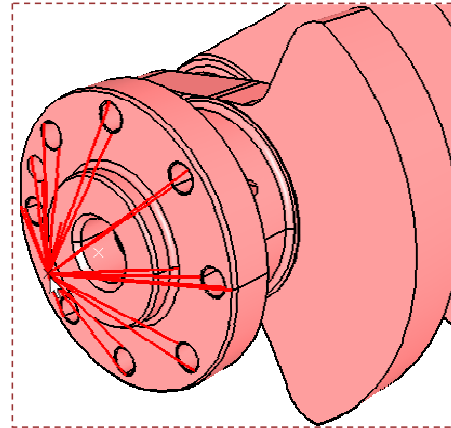
Distributed Mass

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.

1 ──────────── An... ⊠

Analysis Supports

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

2 ──────────── Analysis Supports ⊠

Face Face Connection Properties

3 **Fastened Connection Property:** creates a link between two bodies which are fastened together at their common boundary.

Face Face Connection Prope... ⊠

4 ────────────

3 ────────────

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.
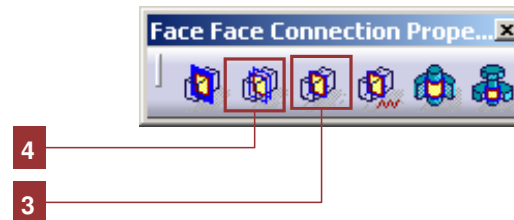
Main Tools (2/2)

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.
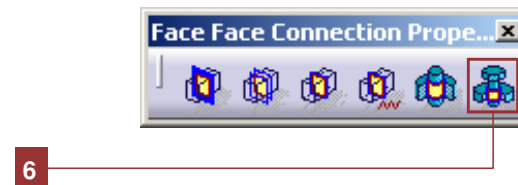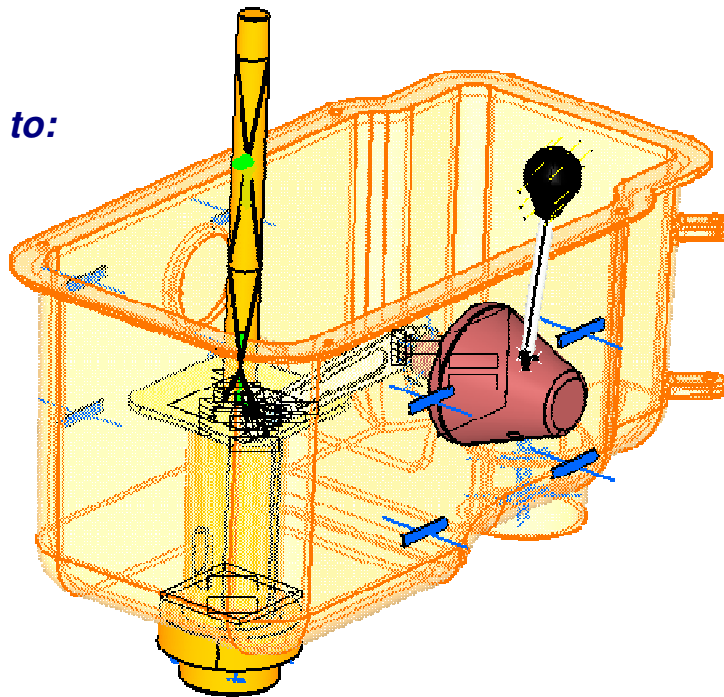
Exercise 6C

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.
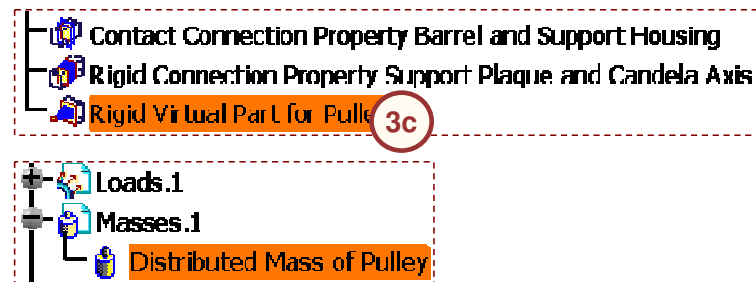   d. Click **OK**.

Virtual Part symbol
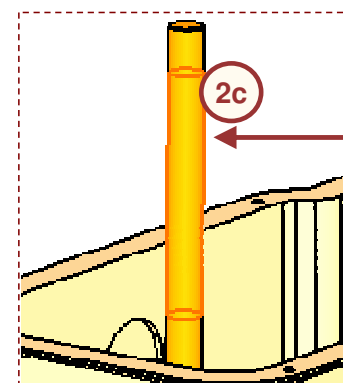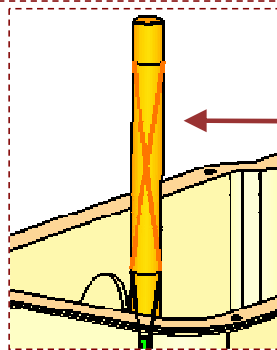
Select face as Support

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.
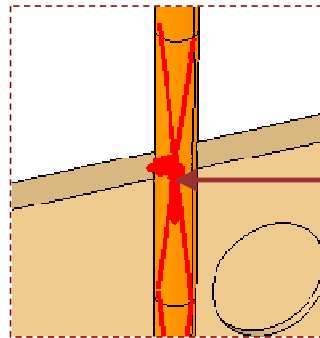   d. Enter *1 Kg* in **Mass** field.
     e. Click **OK**.
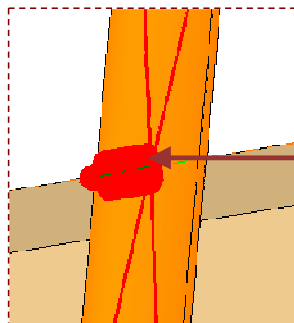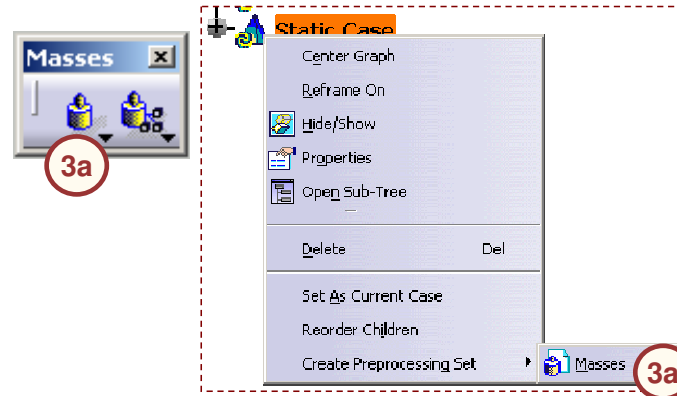
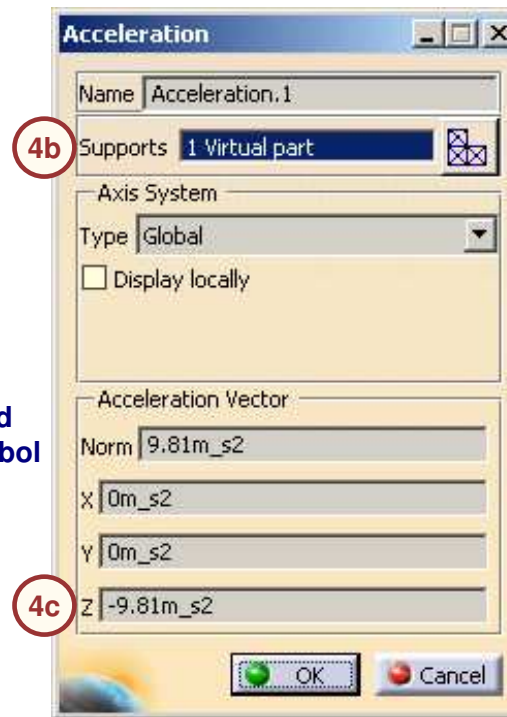**Distributed Mass symbol**

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_s2* in **Z** field.

    d.   Click **OK**.

Distributed Mass symbol
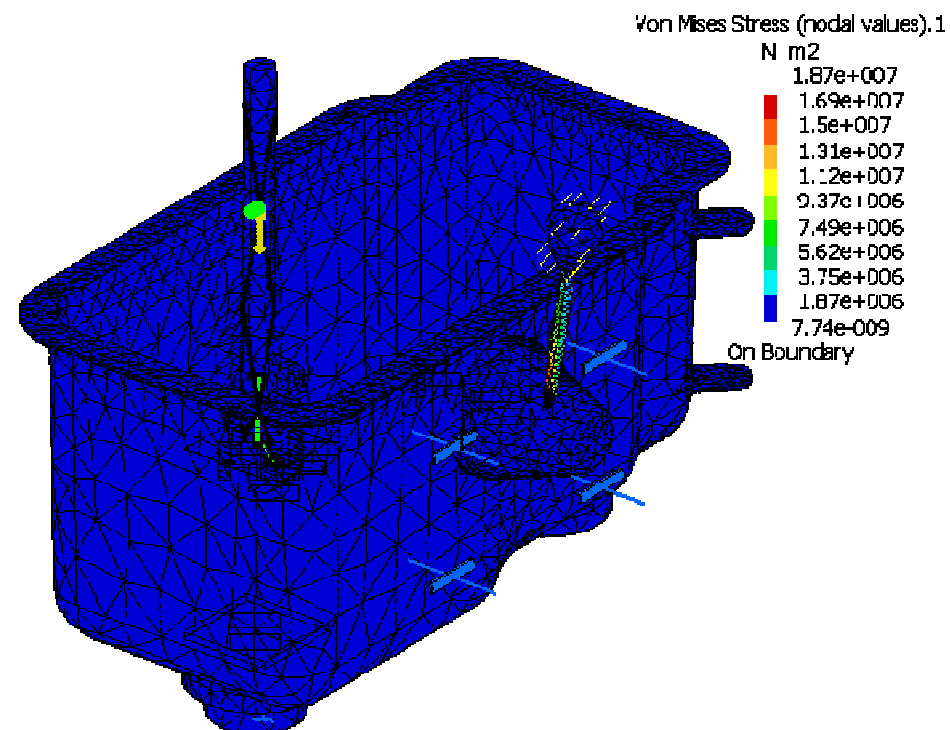
5.   Compute the Model and visualize results.

-   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.**

Von Mises Stress (nodal values).1
N_m2
1.87e+007
1.69e+007
1.5e+007
1.31e+007
1.12e+007
9.37e+006
7.49e+006
5.62e+006
3.75e+006
1.87e+006
7.74e-009
On Boundary

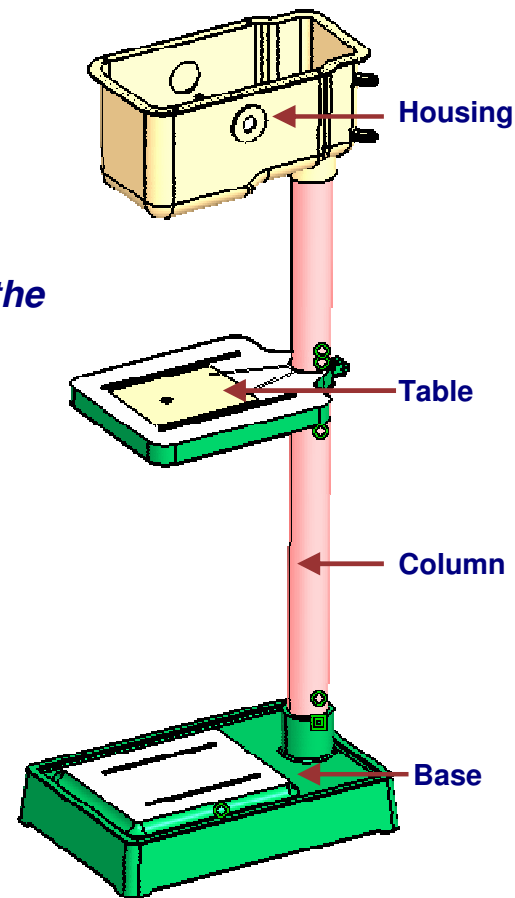# Case Study: Analysis Assembly of Drill Press Sub-Assembly

Recap Exercise

30 min

You will practice what you learned, by completing the case study model. In this exercise, you will analysis Drill Press Sub-assembly using Analysis Assembly approach. Recall the design intent of this model:

✓ Attach the Shapes (Analysis Documents)
✓ Create Assembly connections
✓ Apply Virtual Part
✓ Apply Additional Mass
✓ Apply the Load
✓ Apply Boundary condition
✓ Compute the assembly
✓ Display the results

Housing

Table

Column

Base

Case Study: Introduction

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 Drill Press Sub-Assembly using Analysis Assembly approach. You will perform following steps in order to achieve this.

1. Attach the Shapes (Analysis Documents).
2. Create Assembly connections.
3. Apply Virtual Part.
4. Apply Additional Mass.
5. Apply the Load.
6. Apply Boundary condition.
7. Compute the assembly.
8. Display the results.
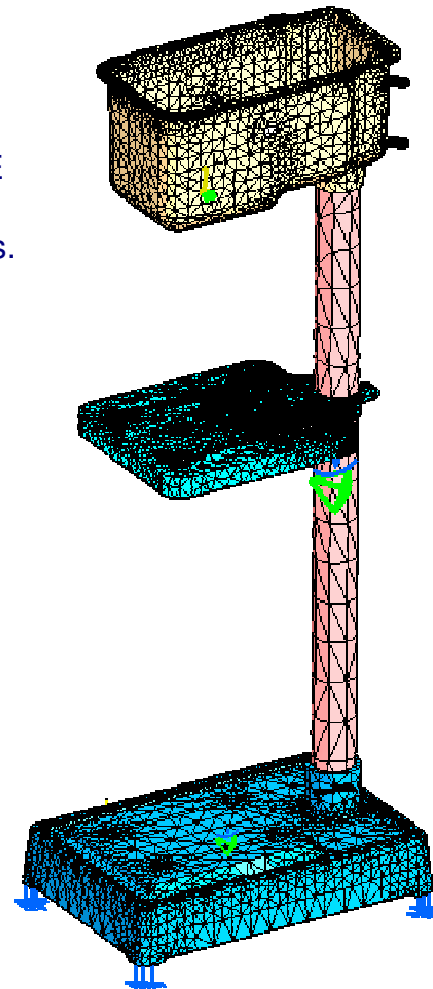
Drill press Sub-Assembly Mesh Image

Design Intent (1/2)

✓ In this case study you will follow Analysis Assembly approach to analyze assembly. There are four components in this assembly.

- Base

- Column

- Table (Movable Table)

- Housing
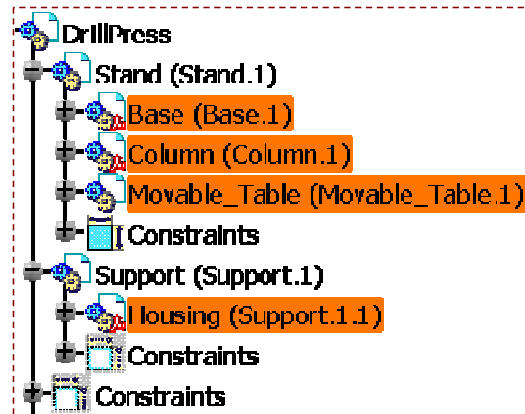
✓ The Component level analysis documents for Table and Housing are available. You will reuse these documents in this assembly. For other components mesh parameters and property will be automatically applied when you will switch to GPS workbench.

- Attach corresponding Analysis Documents for Table and Housing components in Product document.

- Switch to GPS workbench.
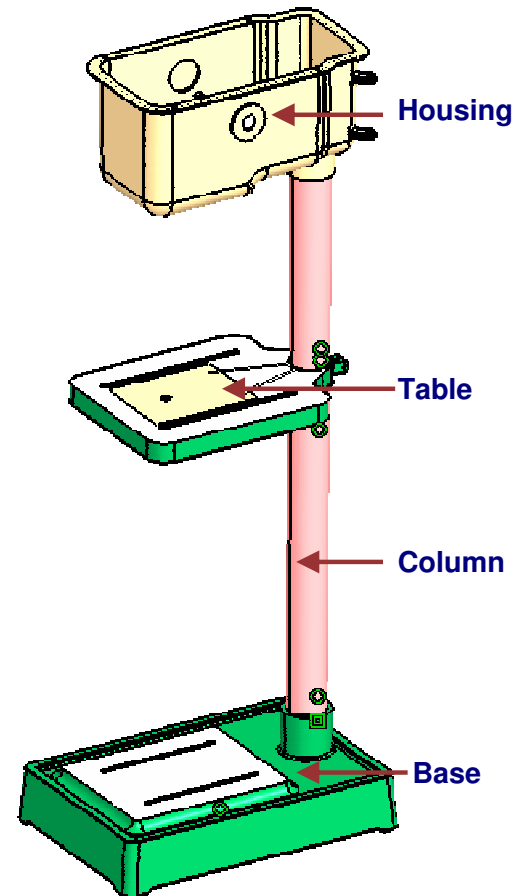
Design Intent (2/2)

✓ The components will be connected to each other in following manner.

- ▪ Rigid connection between Base and Column.

- ▪ Fastened Connection between Column and Table.

- ▪ Rigid Connection between Column and Housing.

✓ The Table is loaded with uniform pressure of 1N per square meter in area shown.

✓ The highlighted area in housing has mass load of 10 Kg to simulate the mass load of subassembly mounted on this surface as support. You will use Rigid Virtual part as support for mass.

- ▪ Apply this mass Load using Rigid virtual part on highlighted area.

- ▪ Convert this mass into force using gravity load.

✓ Check and Compute the FE model.

✓ Fix the bottom of the base.

✓ Compute and view the results.
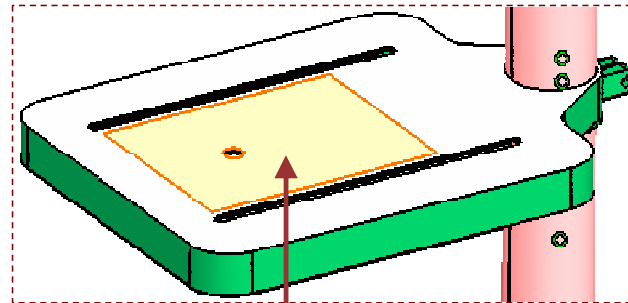
**Area to be loaded with pressure**
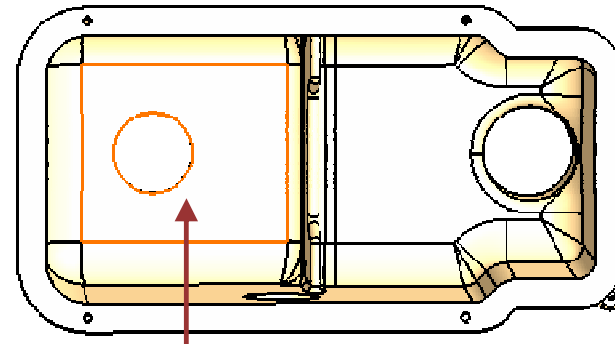
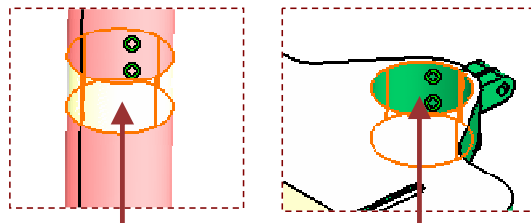**Highlighted area for Rigid virtual part support**

Do It Yourself (1/4)

The following steps offer hints to guide you through
the Analysis Assembly of Drill Press Sub-Assembly.

1. **Open an Analysis Document**
   DrillPress_Sub_Assembly_Analysis_Assembly_Start.CATProduct.

2. **Attach Existing analysis documents to the corresponding**
   components in DrillPress Sub-Assembly.
   - Attach Table_Sub_Analysis.CATAnalysis to Table component
   - Attach Support_Sub_Analysis.CATAnalysis to Housing
     component.
   - Switch to GPS workbench.

3. **Create Assembly Connections.**
   - Create Rigid connection between Base and
     Column using Contact surface constraint
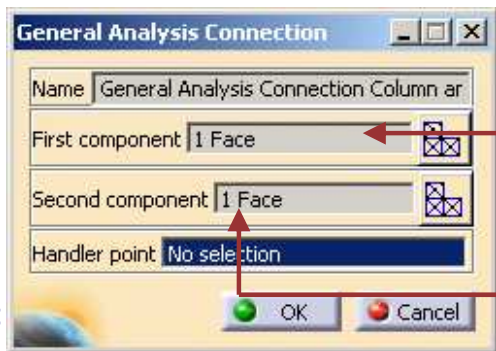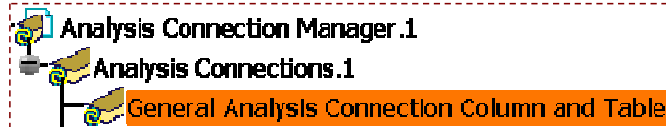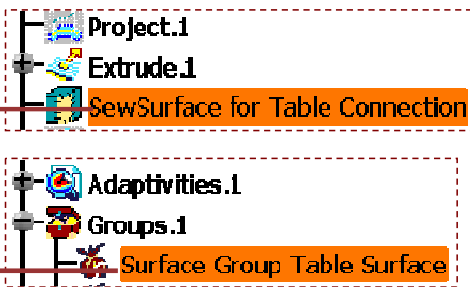   between Base and Column.
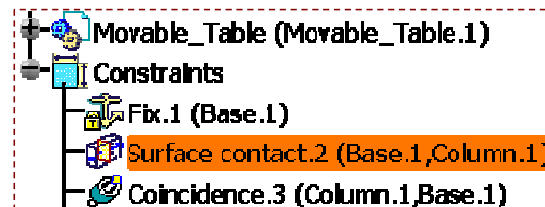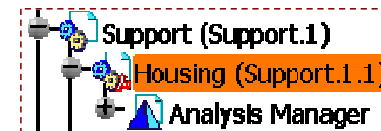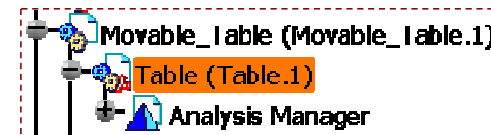
Do It Yourself (2/4)

- Create Fastened Connection between Column and Table. For this, create General Analysis Connection.

First component   **Second component**

Second component

- Create Rigid Connection between Column and Housing. For this, create General Analysis Connection. Select components from the specification tree


First component   **Second component**

Do It Yourself (3/4)

4. **Model drill mechanism sub-assembly mass using virtual part and additional mass.**

   - Double-click on Analysis Manager to activate Analysis Link in Housing(Support.1.1). The Analysis Manager will be highlighted in blue color.

   - Double-click on Global Analysis Manager to return to Global Analysis Manager. The Analysis Manager will be highlighted in blue color.

   - Create distributed Mass of 10 Kg with Rigid Virtual Part as support.
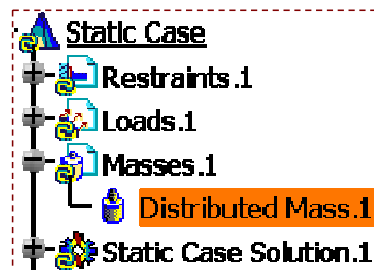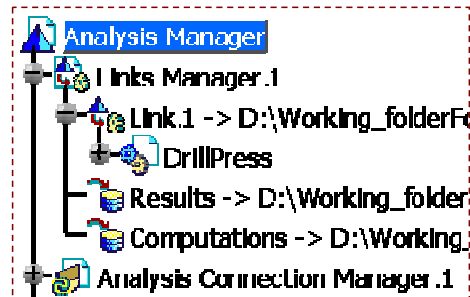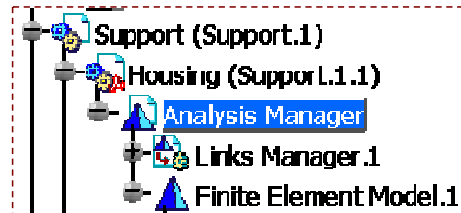
Do It Yourself (4/4)

5. **Apply Loads.**
    ▪ Apply Gravity Load to Distributed Mass with 9.81 m_s2 Acceleration in Z- direction.
    ▪ Apply 1 N_m2 Pressure on Drill Table surface shown in image.
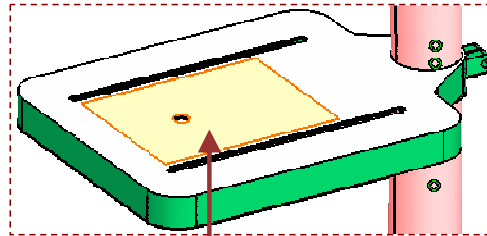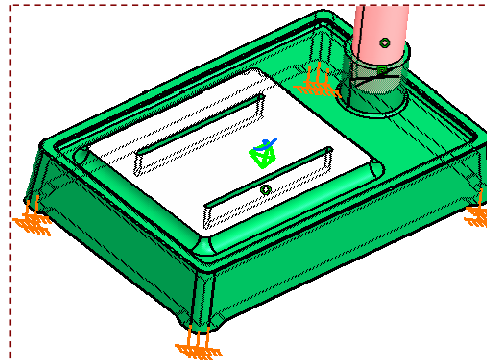
Area to be loaded with pressure

6. **Apply Boundary Condition.**
    ▪ Clamp the bottom of the Base.

7. **Check the Model.**

8. **Compute analysis.**

9. **View the results.**

Case Study: Analysis Assembly of Drill Press Sub-Assembly Recap

✓ **Attach the Shapes (Analysis Documents).**

✓ **Create Assembly connections.**

✓ **Apply Virtual Part.**

✓ **Apply Additional Mass.**

✓ **Apply the Load.**

✓ **Apply Boundary condition.**

✓ **Compute the assembly.**

✓ **Display the results.**

Translational displacement: magnitude.1
mm
0.204
0.183
0.163
0.143
0.122
0.102
0.0815
0.0611
0.0408
0.0204
0
On Boundary