



CATIA Training

Foils

[Student Notes:](#)

Generative Assembly Structural Analysis

**Version 5 Release 16
November 2005**

EDU-CAT-EN-GAS-FF-V5R16

Course Presentation

[Student Notes:](#)

Objectives of the course

This course covers tools for Structural Analysis on an assembly.
You will learn about (1) Different approaches in Generative Assembly Structural Analysis, (2) Creating analysis connections between assembly components, (3) Assigning connection properties to analysis connections and (4) To perform a global assembly analysis using the GPS tools.

Targeted audience

CATIA Designers

Prerequisites

V5F, GPF



Student Notes:

Table of Contents (1/2)

■ Introduction to GAS	5
◆ What is Generative Assembly Structural Analysis	6
◆ Assembly Analysis Overview	8
◆ Analysis Assembly Overview	9
◆ what is GAS	10
◆ Hypotheses Used for Analysis	11
◆ To sum Up	12
■ Analysis Connections	13
◆ What is Analysis Connection	14
◆ General Analysis Connection	20
◆ Defining Line Analysis Connections	24
◆ Defining Point Analysis Connections	31
◆ Defining Surface Analysis Connections	37
◆ Points to Points Analysis Connection	44
■ GAS Connections	48
◆ Face Face Connection Properties	49
◆ Distant Connection Properties	69
◆ Welding Connection Properties	86

Table of Contents (2/2)

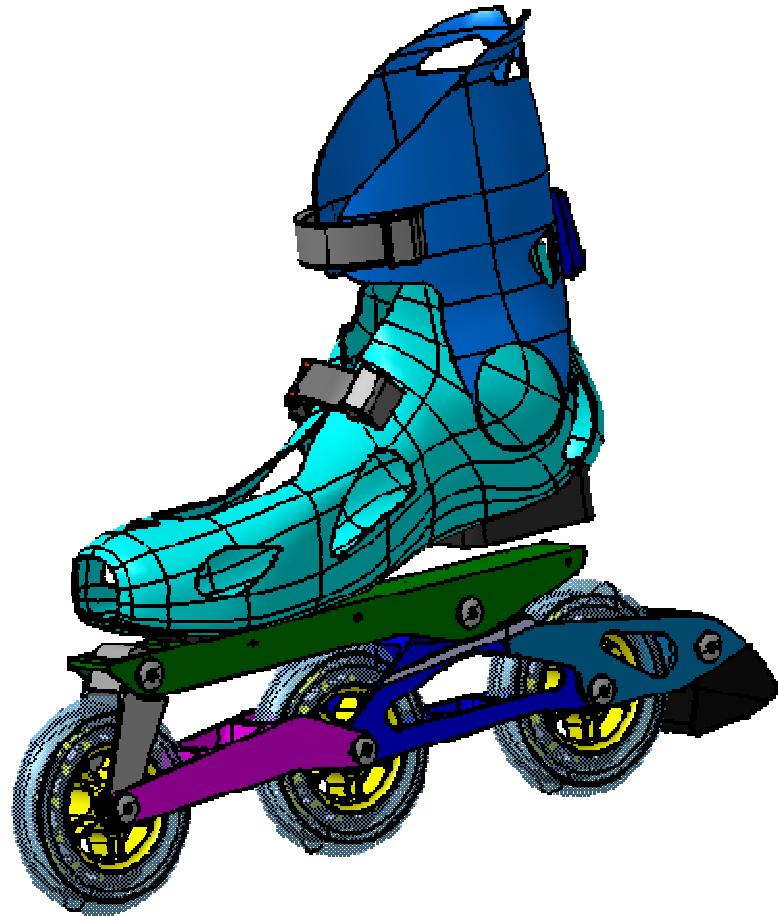
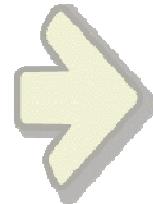
◆ Nodes to Nodes Connection Property	93
◆ To Sum Up	96
Analysis Assembly Management	97
◆ What is Analysis Assembly	98
◆ Why Use Analysis Assembly	99
◆ Analysis Assembly Process	100
◆ What is Shape Representation	101
◆ How to Attach Shape to Part	102
◆ What is Analysis Assembly 2D Viewer	104
◆ How to Use Analysis Assembly 2D Viewer	105
◆ Summary	106

Student Notes:

[Student Notes:](#)

Introduction to GAS

You will learn the rules necessary to a good understanding and use of GAS



[Student Notes:](#)

What is Generative Assembly Structural Analysis (1/2)

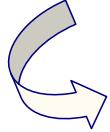
Generative Assembly Structural Analysis (GAS) is a tool to perform structural analysis of assembly. It provides features to model physical assembly into Finite Element Assembly.

While designing, structural analysis of individual parts is performed. These parts are commonly components of a product.

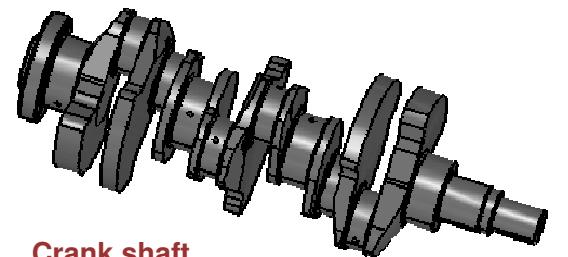
For example, analysis of crank shaft helps to understand structural, dynamic behavior under applied load, and to improve the design.



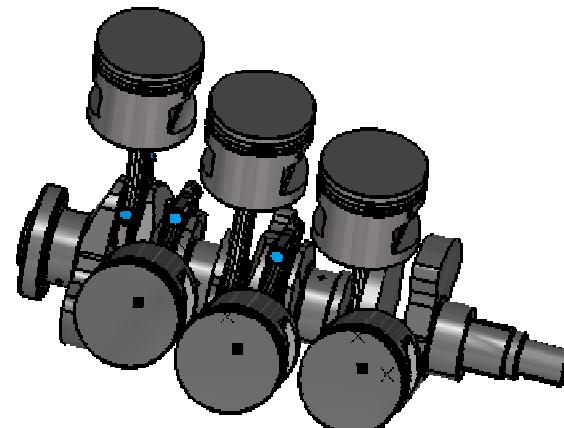
Crank shaft is one of the part of engine assembly which also contains connecting rod, pin, piston, bolts etc.



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



Crank shaft



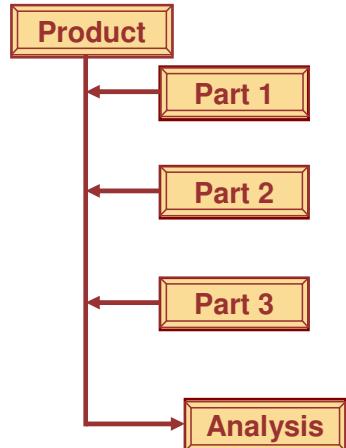
Crank shaft in engine assembly

Student Notes:

What is Generative Assembly Structural Analysis (2/2)

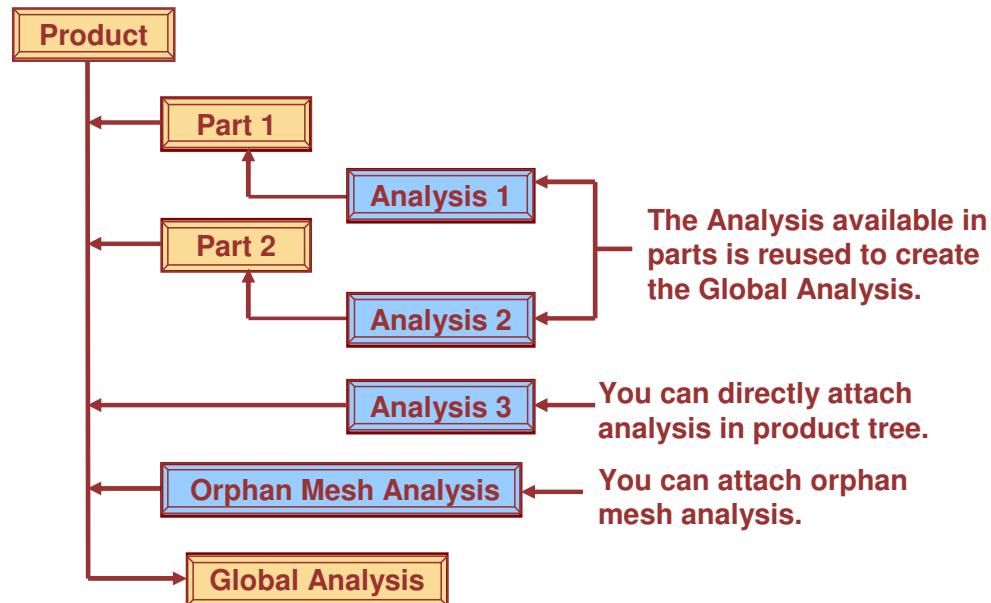
We have two basic approaches to perform Generative Assembly Structural analysis.

Assembly Analysis:



In this approach you create the assembly and analyze it.

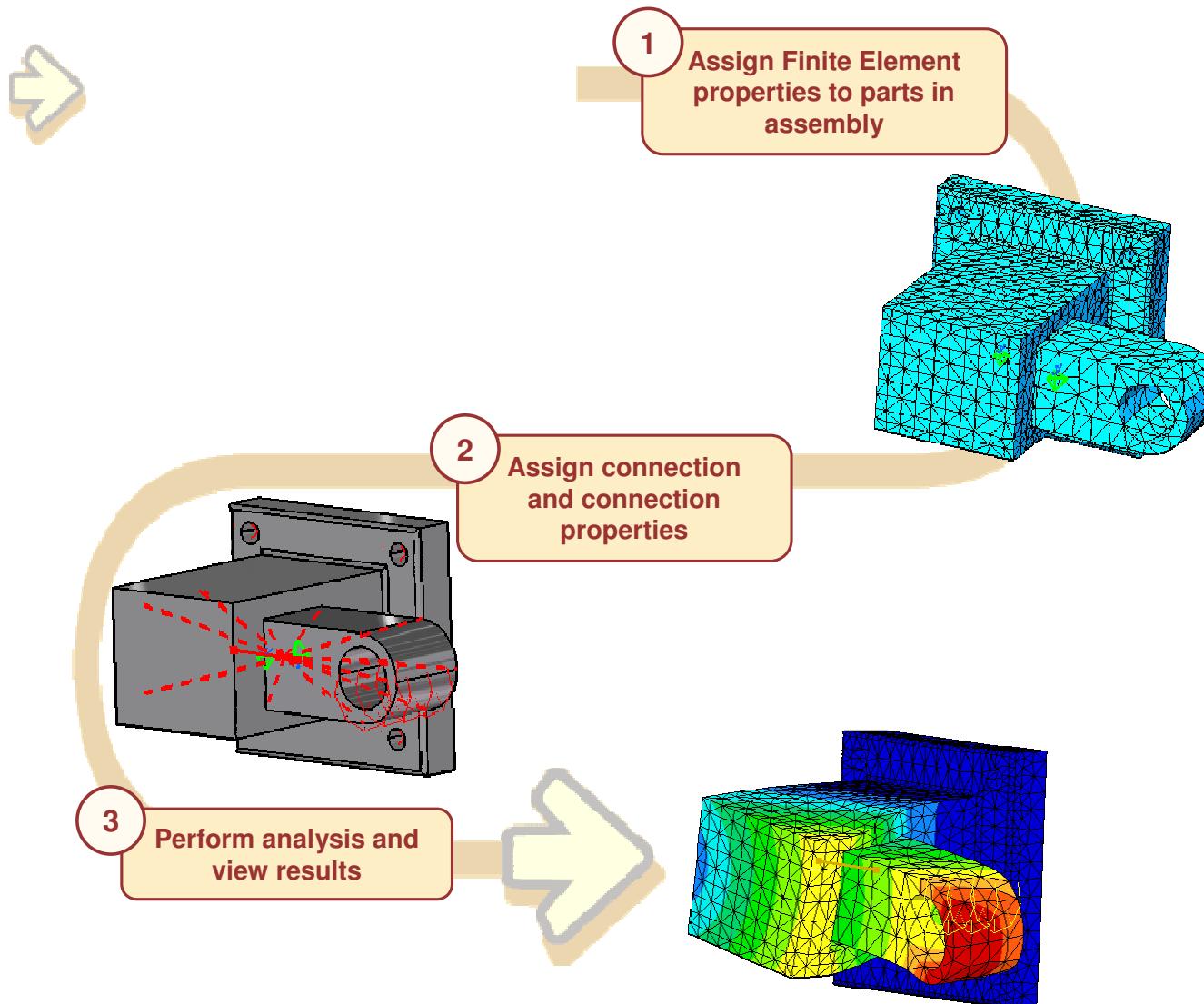
Analysis Assembly:



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.

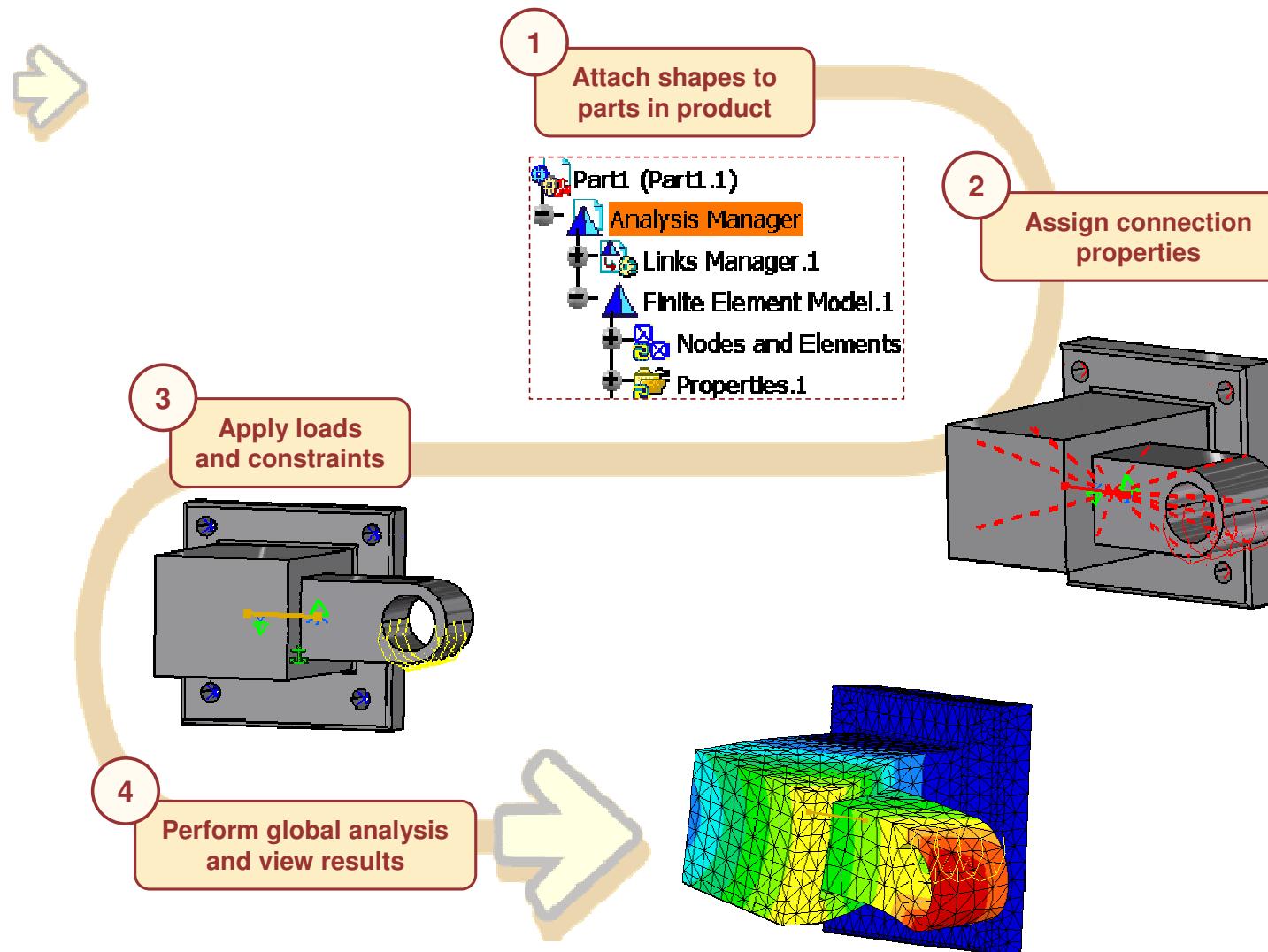
Assembly Analysis Overview

Student Notes:



Analysis Assembly Overview

[Student Notes:](#)



[Student Notes:](#)

What is GAS

The GAS license provides functionalities for analysis of assembly, through the GPS workbench. It allows you to define connections between assembly components and assign different types of connection properties to that connections to simulate the real connection behavior.

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

- Face/Face Connections properties



- Distant Connections properties



- Welding Connections properties



- GPSAeroConnection container



However, you must have previously defined 'Assembly constraints' or 'Analysis Connections' using GAS Workbench to be able to create the connections properties between the parts of assemblies. By the way, you must make sure that your assembly is not over-constraint.

You may also need to add assembly constraints at a given distance (this constraint goes through a virtual point) so that you can simulate a part that is not designed.

Student Notes:

Hypotheses Used for Analysis

When you work with the Analysis workbench, three types of hypotheses are made:

- 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 (being virtual or not) feature, then the problem is linear, that which means the displacement is a linear function of the load.

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

To Sum Up ...

[Student Notes:](#)

You have learned following things about GAS

- What is Generative Structural Assembly Analysis
- Generative Structural Assembly Analysis approaches
- Hypothesis used for Analysis

[Student Notes:](#)

Analysis Connections

In this lesson, you will see what are the different types of GAS Analysis Connections, necessary to define support for Analysis Connection properties.

- What is Analysis Connection
- General Analysis Connection
- Defining Line Analysis Connections
- Defining Point Analysis Connections
- Defining Surface Analysis Connections
- Points to Points Analysis Connection

Student Notes:

Analysis Connection

You will learn what is Analysis Connection and why it is required.

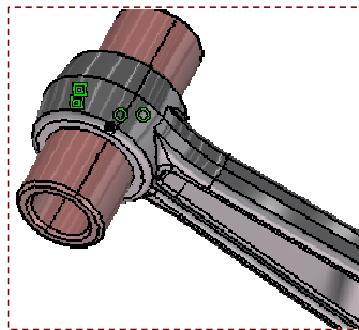
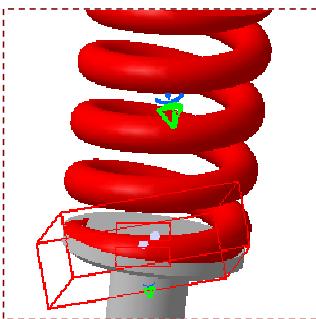


[Student Notes:](#)

Why Use Connections and Connection Properties

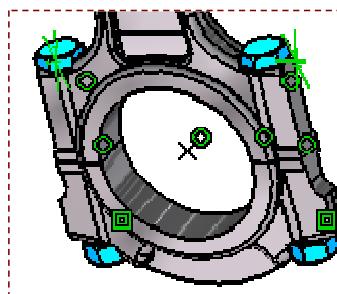
GAS has made conversion of physical assemblies to FE assemblies very easy by means of connection and their connection properties. Wide variety of connection types and connection properties are provided to model physical assembly connections.

When parts connected to each other, they transmit rotational and translational DOFs in well defined manner. In addition to this, connection itself have structural properties which needs to be taken into account.



For example, when two parts are connected through spring joint, the spring stiffness will also play a role. This stiffness will dictate the amount of displacement transferred from one part to another. This can also be viewed for welded or bolted assemblies.

Connection defines which parts in assembly are connected and connection property assigns related physical properties to those connections. You can also effectively utilize the constraints defined in assembly as support in connection properties.

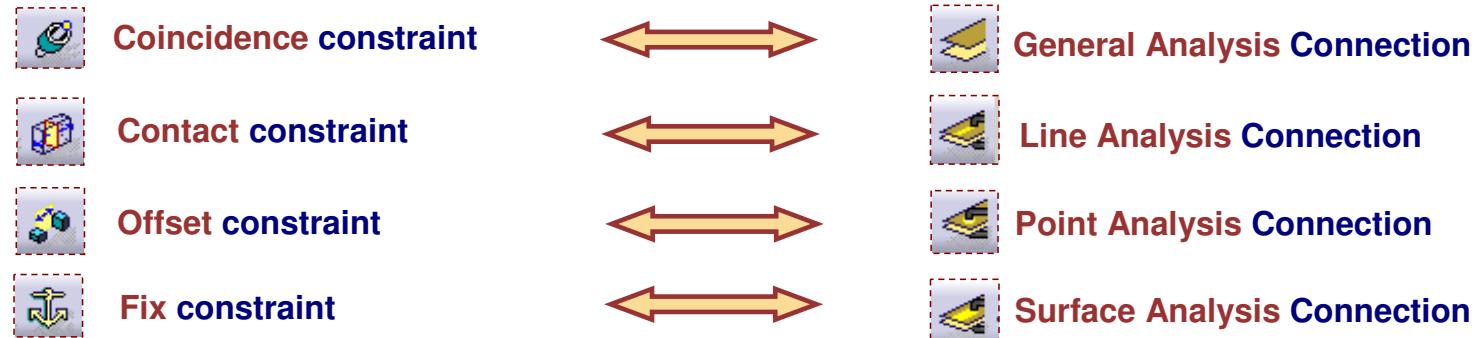


[Student Notes:](#)

Using Assembly Constraints for Analysis Connections

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

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



The following matrices show you, with respect to the connection type, what constraints are necessary for their creation. You will notice that, some connections (like rigid or smooth) can be applied on different kinds of constraints while others can be applied on a specific constraint only.

Student Notes:

What Assembly Constraint to Use for What Connection

You will see which are the most appropriate constraints for each kind of connection

Connections	Point / Point	Point / Line	Point / Face	Line / Line	Line / Face	Face / Face
Slider				Contact Coincidence	Contact Coincidence	Contact Coincidence
Contact				Contact Coincidence	Contact Coincidence	Contact Coincidence
Fastened				Contact Coincidence	Contact Coincidence	Contact Coincidence
Fastened Spring				Contact Coincidence	Contact Coincidence	Contact Coincidence
Pressure Fitting				Contact Coincidence	Contact Coincidence	Contact Coincidence
Bolt Tightening				Contact Coincidence	Contact Coincidence	Contact Coincidence
Rigid			Contact Contact	Contact * Contact *	Contact * Contact *	Contact * Contact *
Smooth			Contact Contact	Contact * Contact *	Contact * Contact *	Contact * Contact *
Virtual Rigid Bolt Tightening			Contact Coincidence Contact Coincidence	Contact Coincidence Contact Coincidence	Contact Coincidence Contact Coincidence	Contact Coincidence Contact Coincidence
Virtual Spring Bolt Tightening			Coincidence Offset	Coincidence Offset	Coincidence Offset	Coincidence Offset
User-Defined		Contact	Contact	Contact	Contact	Contact

* with optional handler point

Student Notes:

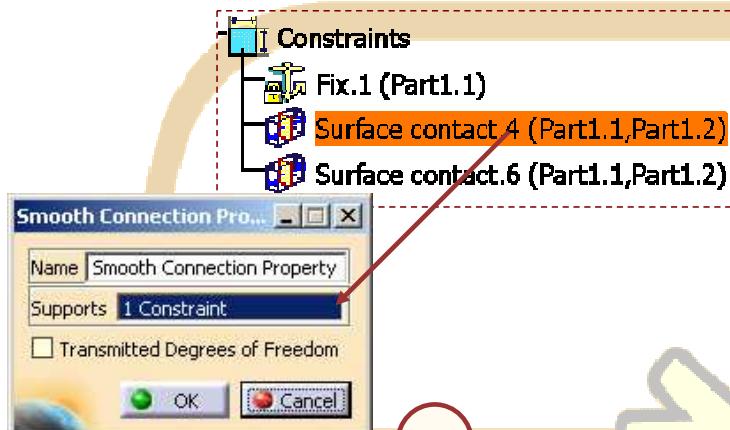
How to Use Assembly Constraints in Connection Property

You can use appropriate assembly constraints as support in connection property. It is not necessary to create Analysis Connection if corresponding Assembly constraint is available

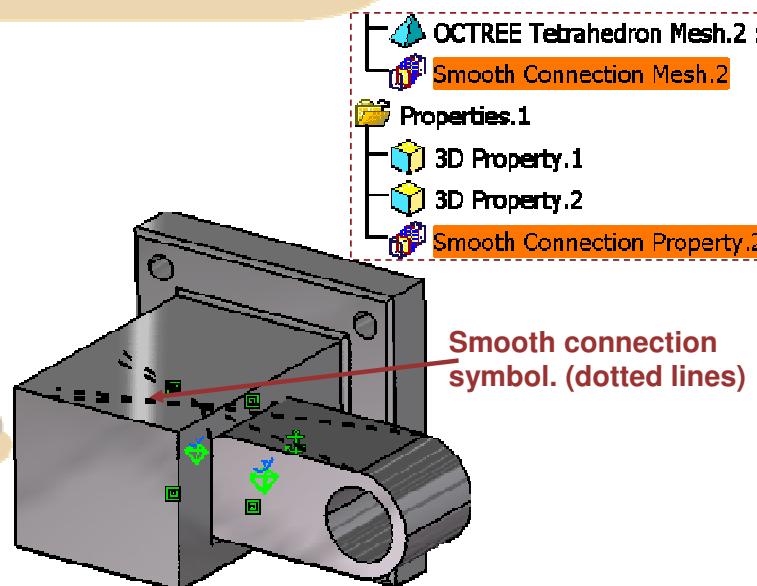
- 1 Click on Smooth Connection Property



- 2 Select Surface contact constraint as in supports



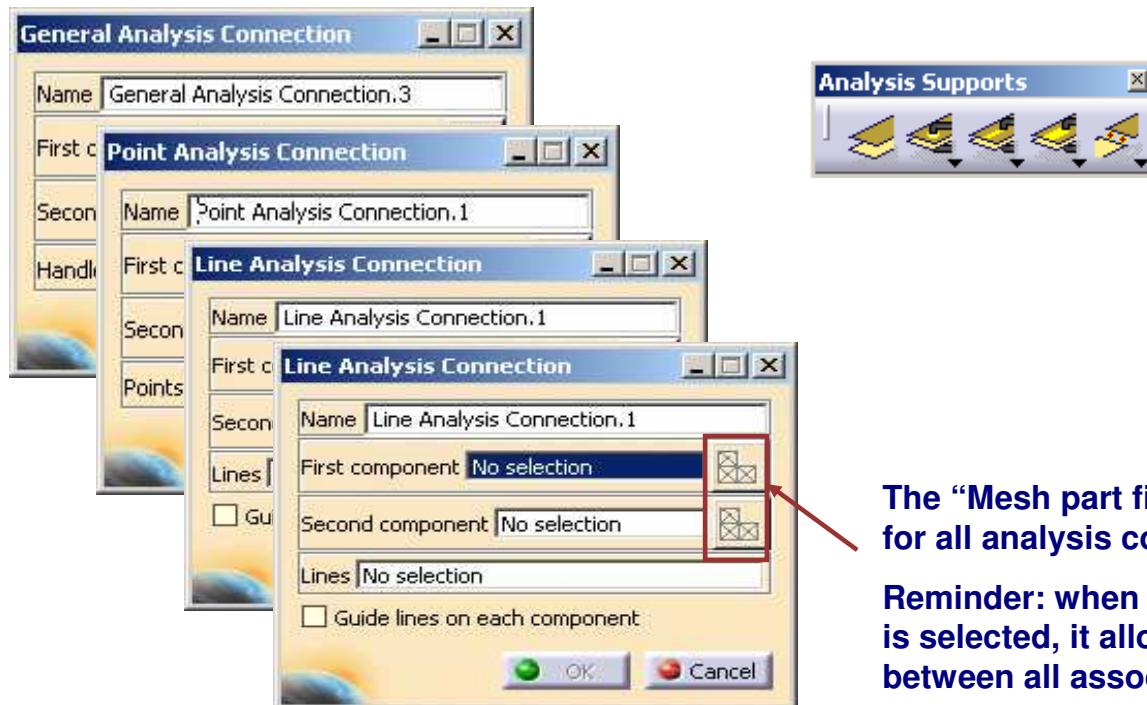
- 3 Click OK



[Student Notes:](#)

Common Tools

All “seam” connections (Point, Line, Surface connections) can be applied directly on mesh parts



The “Mesh part filter” tool is available for all analysis connections

Reminder: when a geometrical support is selected, it allows you to choose between all associated mesh parts

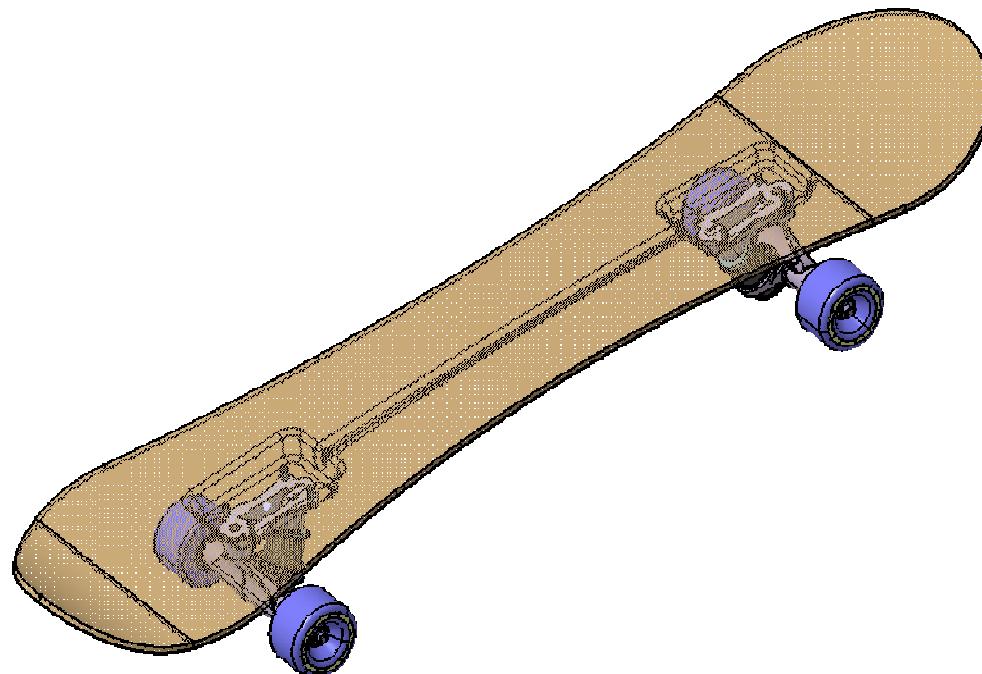
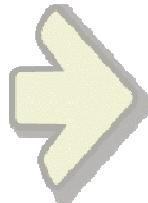


It enables connection definition for orphan meshes (imported meshes)

[Student Notes:](#)

General Analysis Connection

You will learn how to define a General Analysis Connection and when to use it



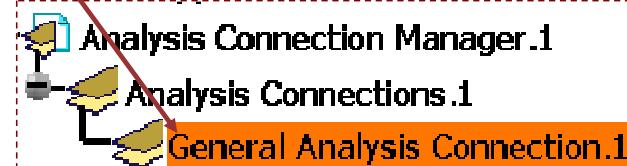
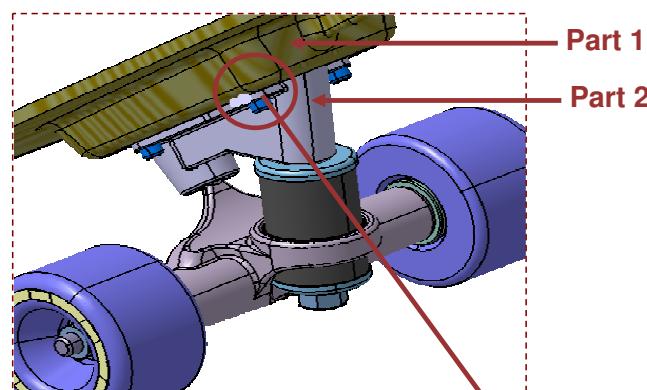
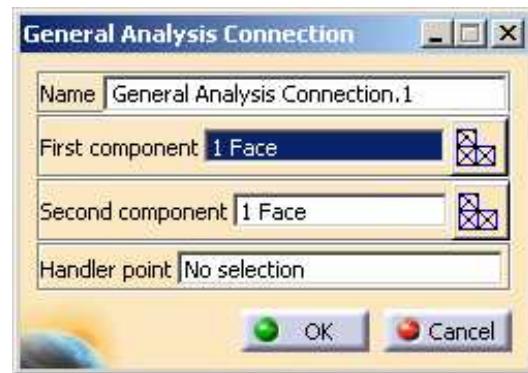
[Student Notes:](#)

What is a General Analysis Connection

General Analysis Connection is used for connecting any part from an assembly with or without point type geometrical elements.

The “General” connection is available in the “Analysis Connections” toolbar:

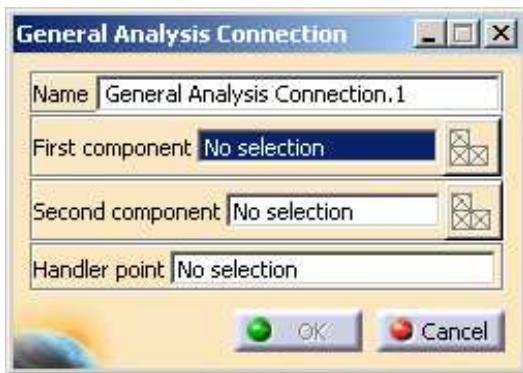
can be performed between any type of geometry



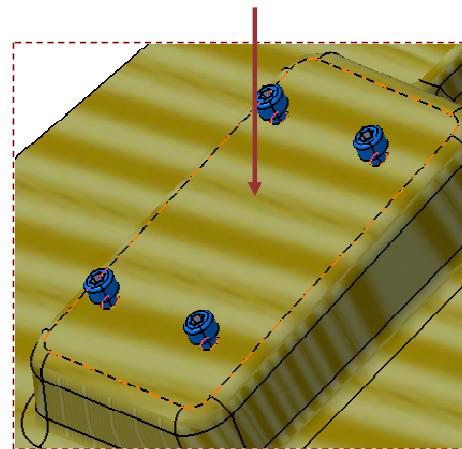
[Student Notes:](#)

How to apply a General Analysis Connection

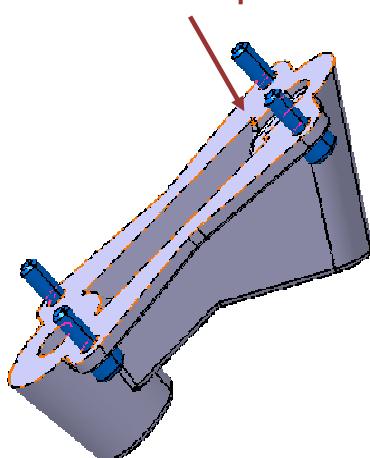
- ① Click on the “General Connection” icon



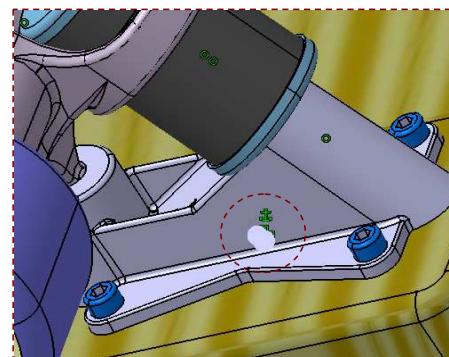
- ② Select the first component



- ③ Select the 2nd component



- ④ Click on Ok



Student Notes:

When to Use the General Analysis Connection

In most cases, a General Analysis Connection can be used as support for following connection properties.

Connections		Point / Point	Point / Line	Point / Face	Point / Mechanical Feature	Line / Line	Line / Face	Line / Mechanical Feature	Face / Face	Face / Mechanical Feature	Mechanical Feature / Mechanical Feature
Slider			Yes	Yes	Yes	Yes	Yes	Yes	Yes	Yes	Yes
Contact			Yes	Yes	Yes	Yes	Yes	Yes	Yes	Yes	Yes
Fastened			Yes	Yes	Yes	Yes	Yes	Yes	Yes	Yes	Yes
Fastened Spring			Yes	Yes	Yes	Yes	Yes	Yes	Yes	Yes	Yes
Pressure Fitting			Yes	Yes		Yes	Yes		Yes		
Bolt Tightening			Yes	Yes		Yes	Yes		Yes		
Rigid			Yes	Yes	Yes	Yes*	Yes*	Yes*	Yes*	Yes*	Yes*
Smooth			Yes	Yes	Yes	Yes*	Yes*	Yes*	Yes*	Yes*	Yes*
Virtual Rigid Bolt		Yes	Yes	Yes		Yes	Yes		Yes		
Virtual Spring Bolt Tightening		Yes	Yes	Yes		Yes	Yes		Yes		
User-Defined		Yes	Yes	Yes	Yes	Yes	Yes	Yes	Yes	Yes	Yes

Student Notes:

Defining Line Analysis Connections

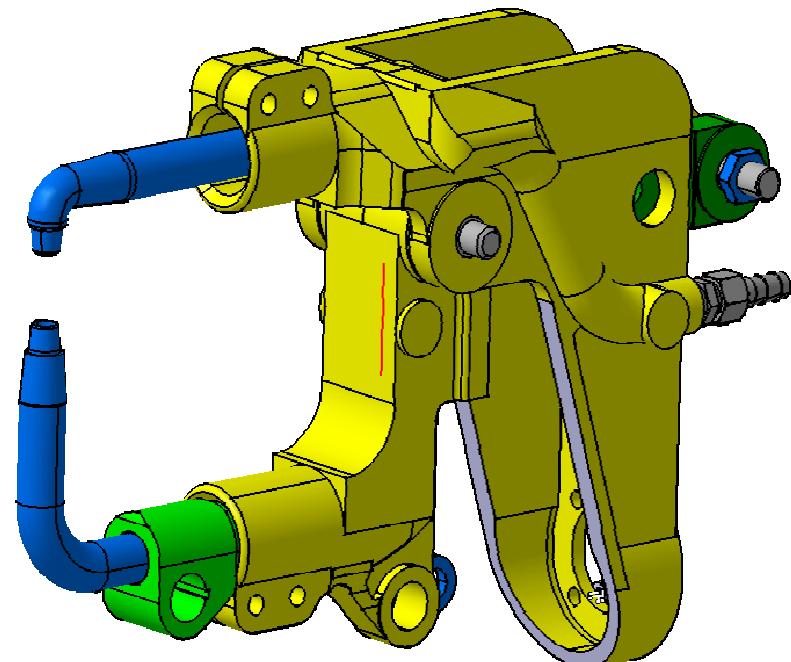
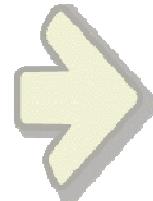
In this lesson, you will see what are the different kind of "Line Analysis Connections"

- Line Analysis Connection
- Line Analysis Connection within one Part

[Student Notes:](#)

Line Analysis Connection

You will learn how to define a Line Analysis connection and how to use it



What is Line Analysis Connection

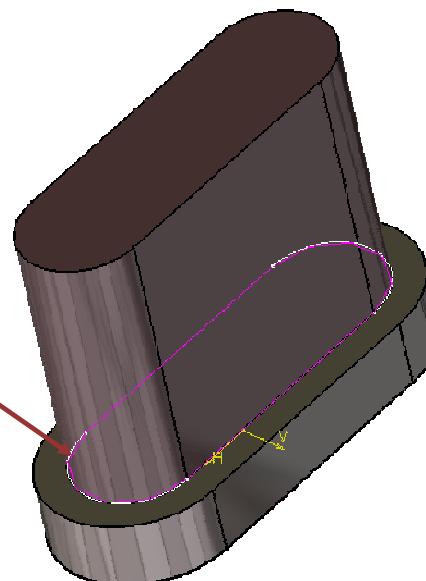
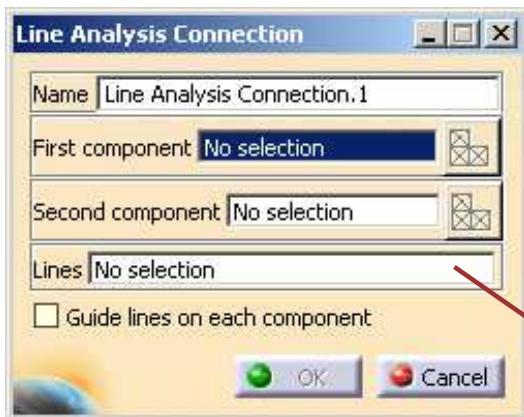


[Student Notes:](#)

Line Analysis Connections are used to simulate seam welding.

As you can see, the Line Analysis connection needs a line (previously created i.e from GSD).

This line is where the seam welding will be located.

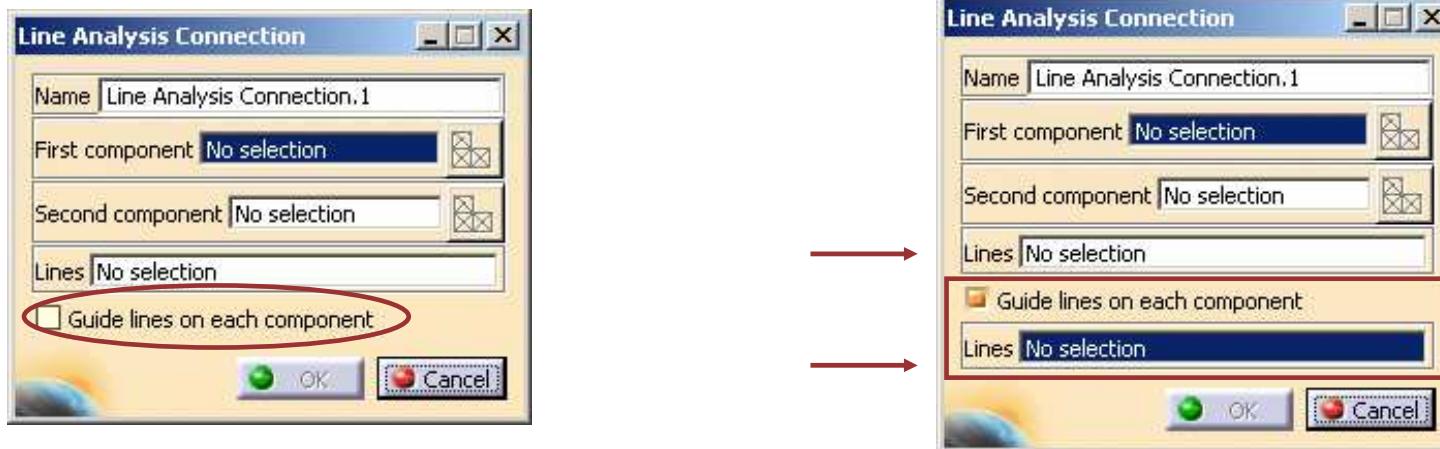


[Student Notes:](#)

Guide Lines

You can select guide lines on each components.

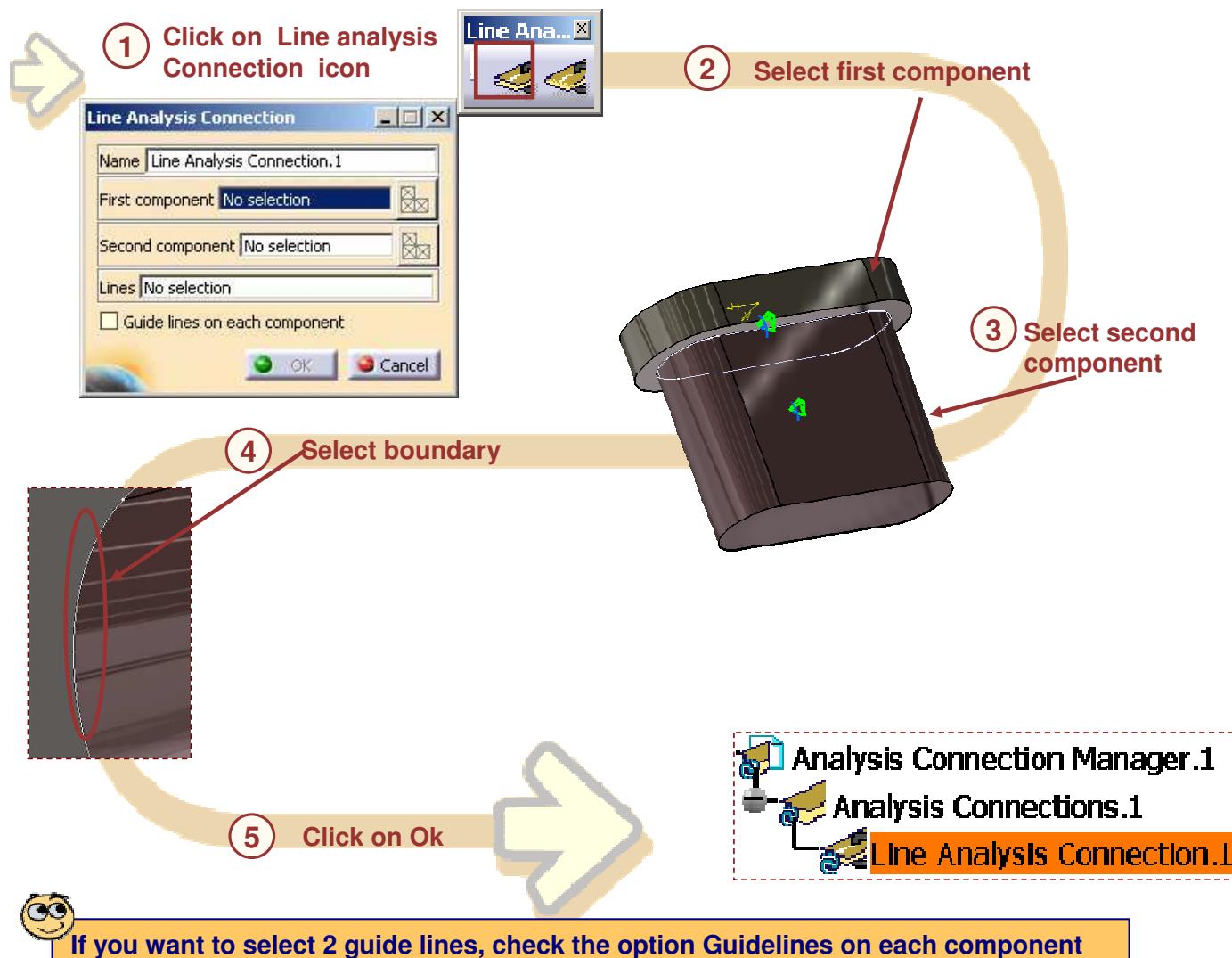
The option “Guide Lines on each component” allows you to guide the connection orientation by selecting two lines.



It will provide a better precision for seam weld orientation and enables welding on non-parallel parts

[Student Notes:](#)

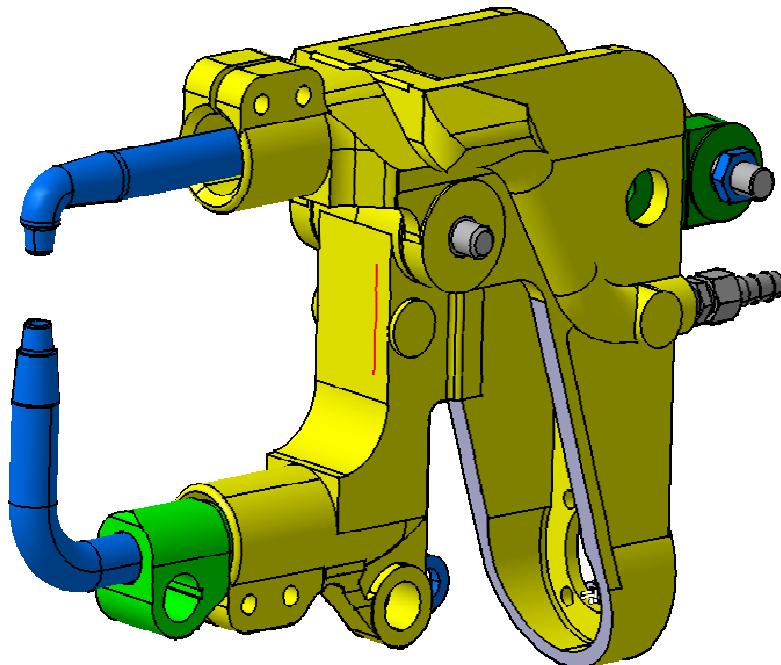
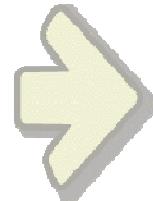
How to apply a Line Analysis Connection



[Student Notes:](#)

Line Analysis Connection within one Part

You will learn how to define a Line Analysis Connection within one Part and how to use it



Student Notes:

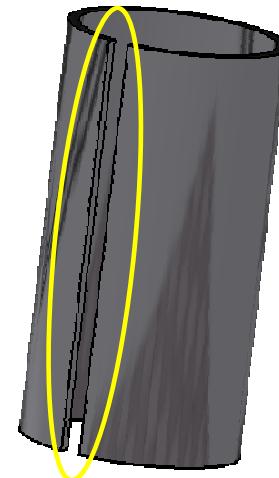
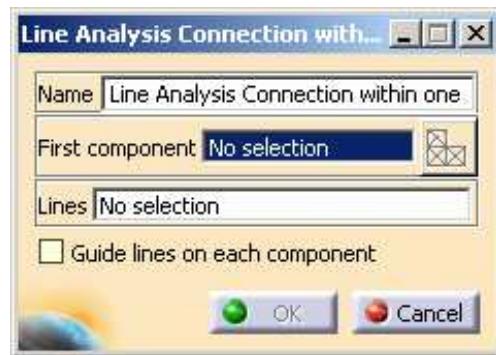
What about Line analysis Connection within one Part

You have to define this kind of analysis connection if you want to seam weld a part on itself.

If your objective is to seam weld two different edges, you have to define previously a Point Analysis Connection with One Part. A standard Line Analysis Connection is not appropriate because it need two components to work properly.



To define a such design connection just proceed as if you wanted to define a Line Analysis Connection but select one component only



The gap of the cylinder was increased for a better display but is not representative of the reality

Defining Point Analysis Connections

In this lesson, you will see what are the different kind of "Point Analysis connections"

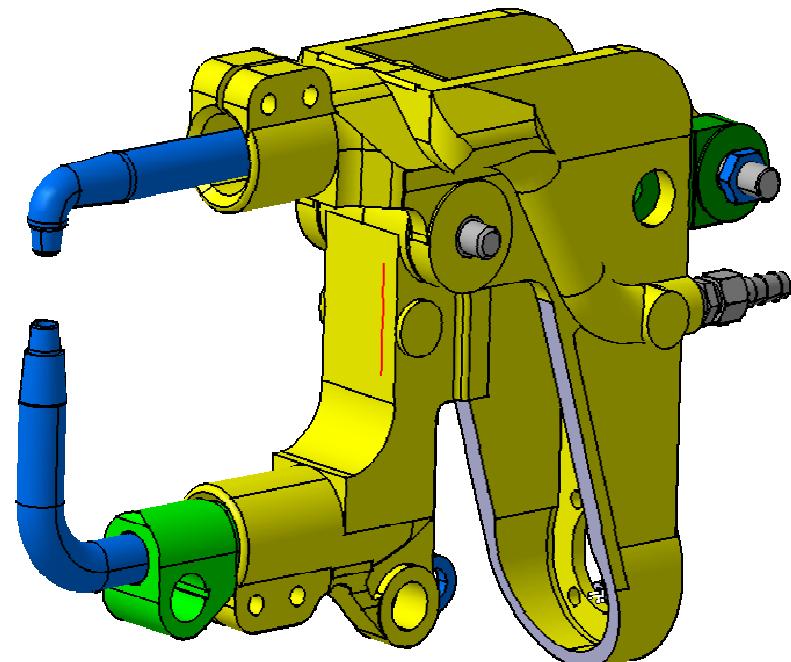
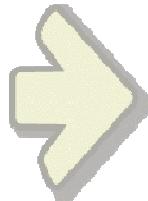
- Point Analysis Connections
- Point Analysis Connection within one Part

[Student Notes:](#)

[Student Notes:](#)

Point Analysis Connection

You will learn how to define a “Point Analysis” connection and how to use it



What is a Point Analysis Connection

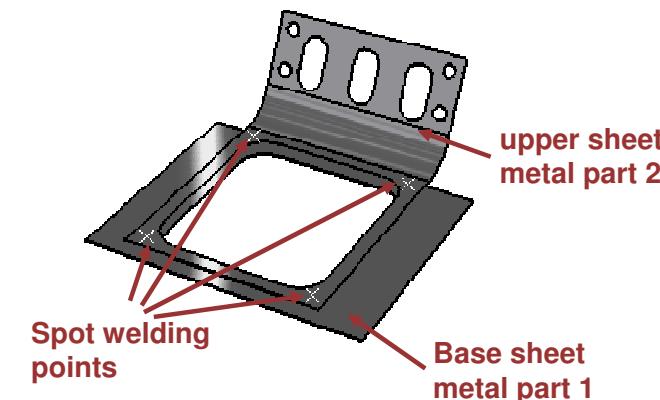


Point Analysis connections are used for projecting welding points onto parallel faces, on an assembly model.

Thus, you need to define a Point analysis connection to be able to use the “Spot welding” connection. Point analysis connection can be performed between any type of geometry



As you can see, the “Point Analysis” connection needs some points (previously created i.e from GSD). These points are the places where the spot welding will be applied.

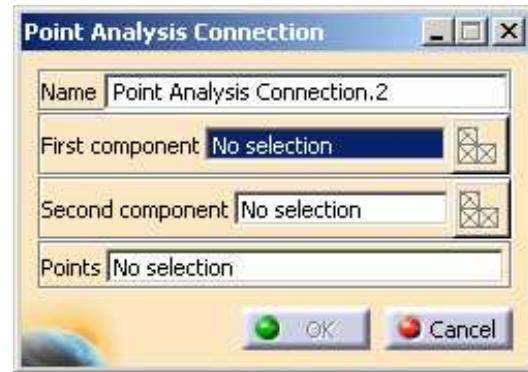


[Student Notes:](#)

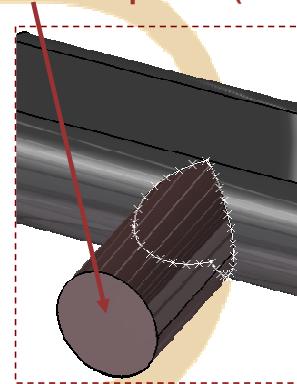
How to Apply a Point Analysis Connection



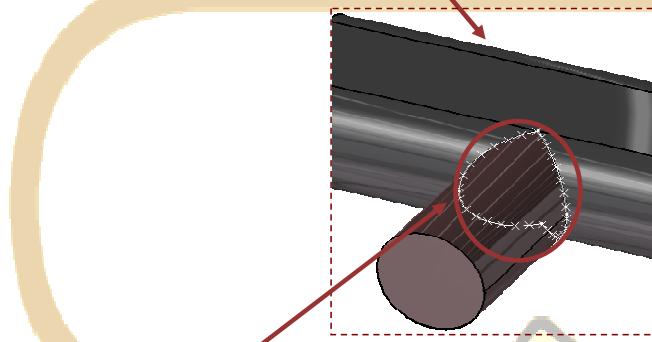
- ① Click on the Point Analysis connection icon



- ② Select the first component (i.e cylinder's face)



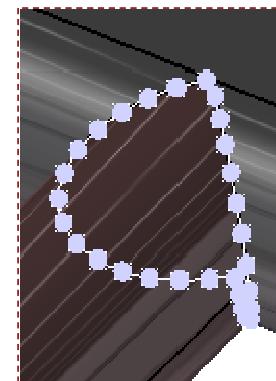
- ③ Select the 2nd component



- ④ Select the points



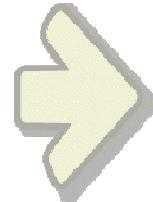
- ⑤ Click on Ok



Student Notes:

Point Analysis Connection within one Part

You will learn how to define a Point Analysis Connection With one Part and how to use it



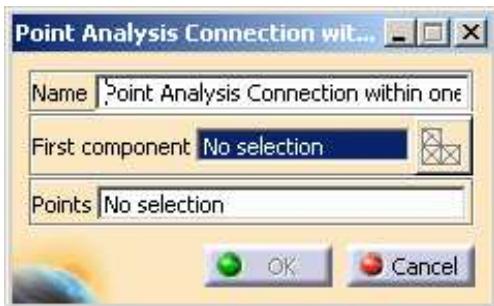
[Student Notes:](#)

What about Point Analysis Connection within one Part

This tool allows you to weld a part on itself only

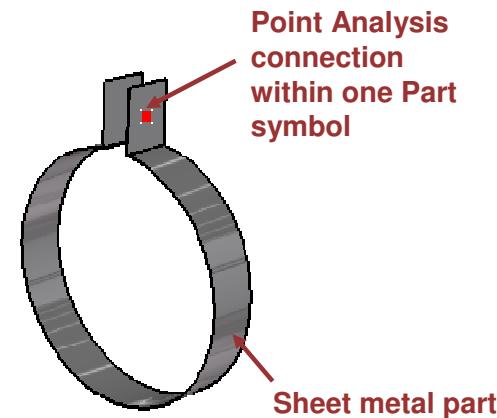


If your objective is to spot weld 2 different areas on a same part, you have to use previously this tool:



For example, with this tool you can define a spot welding between the 2 edges of the cylinder: They must have in common 1 or several points

To define a such design connection just proceed as if you wanted to define a 'Line Analysis Connection' but select one component only



Defining Surface Analysis Connections

In this lesson, you will see how to define Surface Analysis Connections

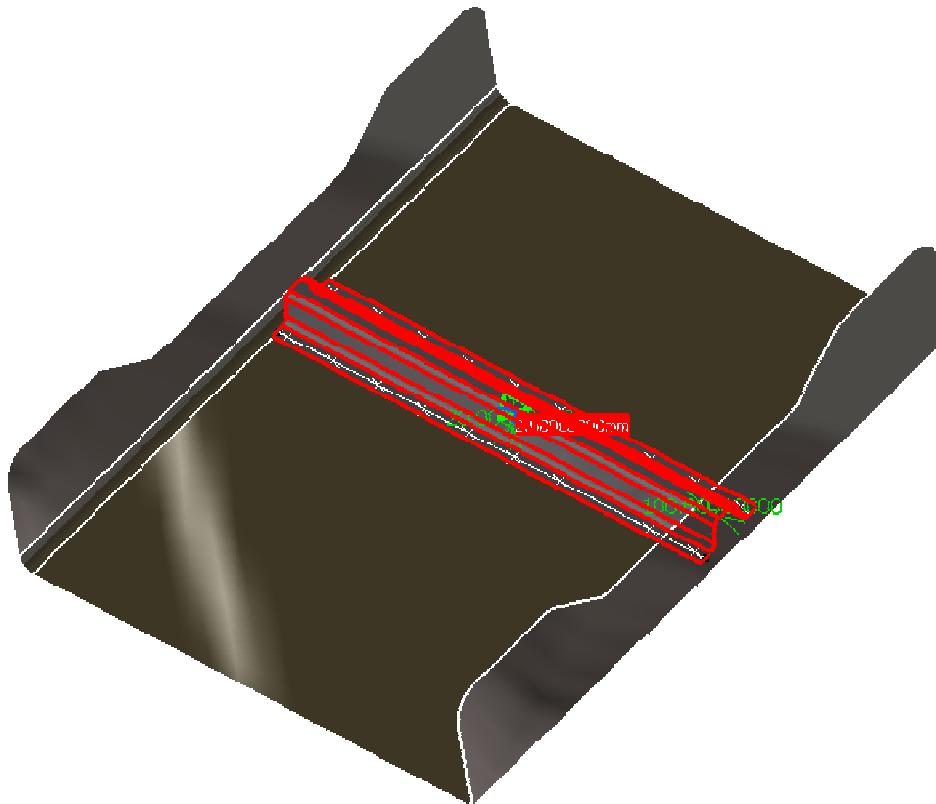
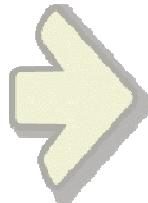
- Surface Analysis Connection
- Surface Analysis Connection within one Part

[Student Notes:](#)

Student Notes:

Surface Analysis Connection

You will see what is Surface Analysis Connection and how to define it

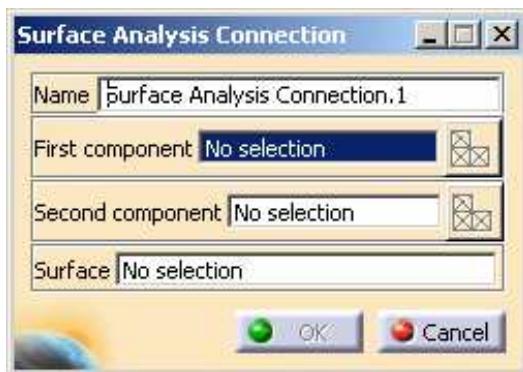


[Student Notes:](#)

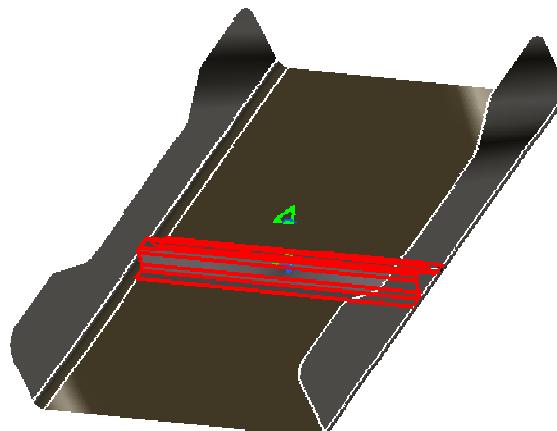
What is a Surface Analysis Connection

A Surface Analysis Connection allows you to connect two supports, controlled by an input surface

It can be used to define adhesive property connections made of hexahedron elements



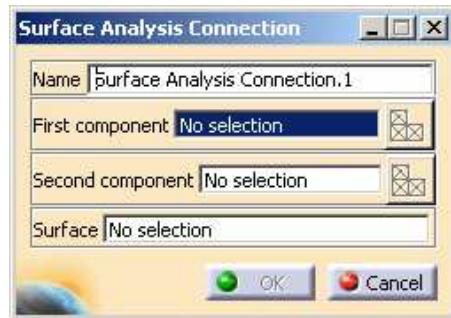
To define a Surface Analysis Connection you need to select two faces (one per component) and a common surface.



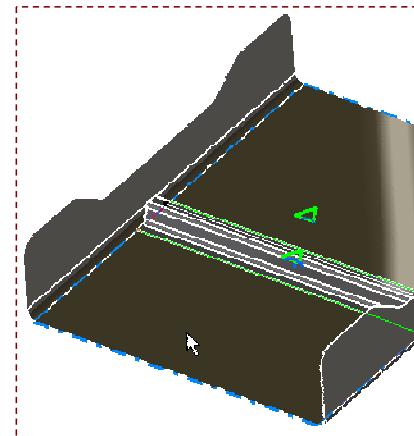
How to Define a Surface Analysis Connection (1/2)

Reminder: A Surface Analysis Connection allows you to connect two supports, controlled by an input surface

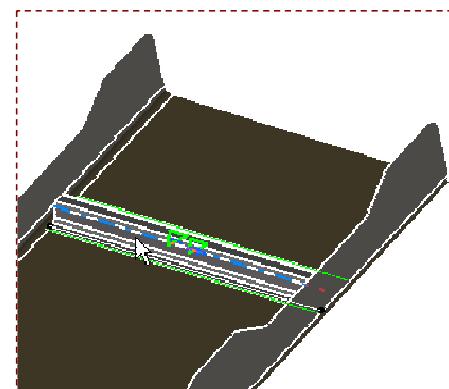
- ① Click on surface analysis connection



- ② Select first component



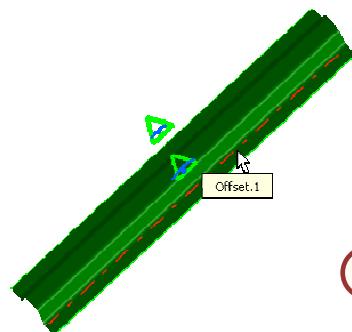
- ③ Select second component



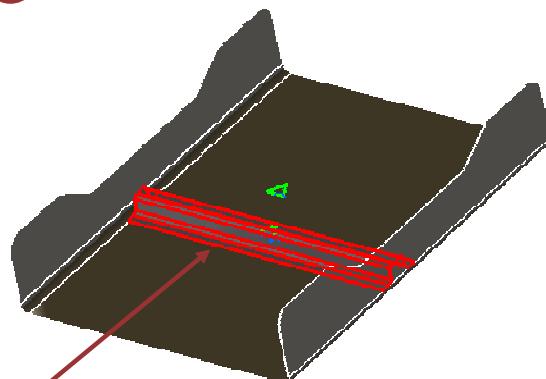
Student Notes:

How to Define a Surface Analysis Connection (2/2)

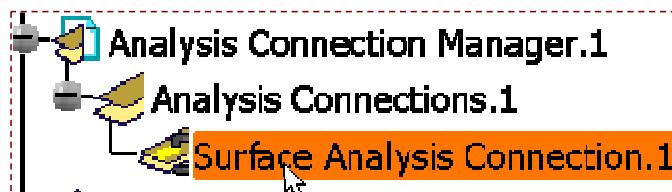
④ Select the surface



⑤ Click on Ok to validate



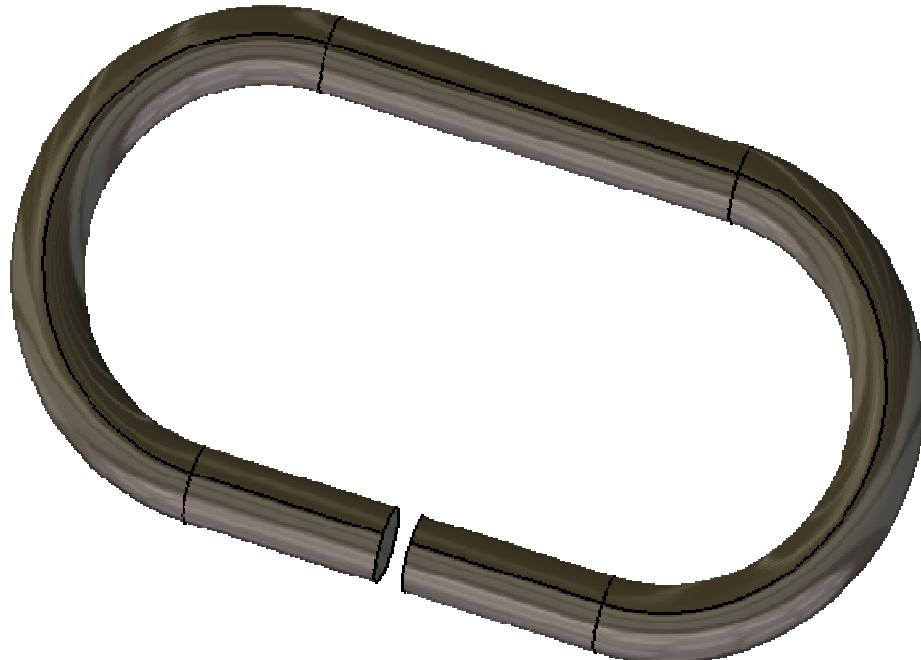
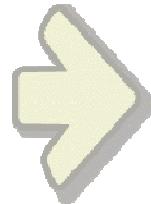
The surface analysis connection is symbolized by the red lines and accessible in the specification tree



Student Notes:

Surface Analysis Connection within One Part

You will see how to define a “Surface Analysis” connection with one Part



[Student Notes:](#)

What about Surface Analysis Connection within one Part

Surface analysis connections within one part are used for simulating welding surface onto parallel faces, belonging to the same part.

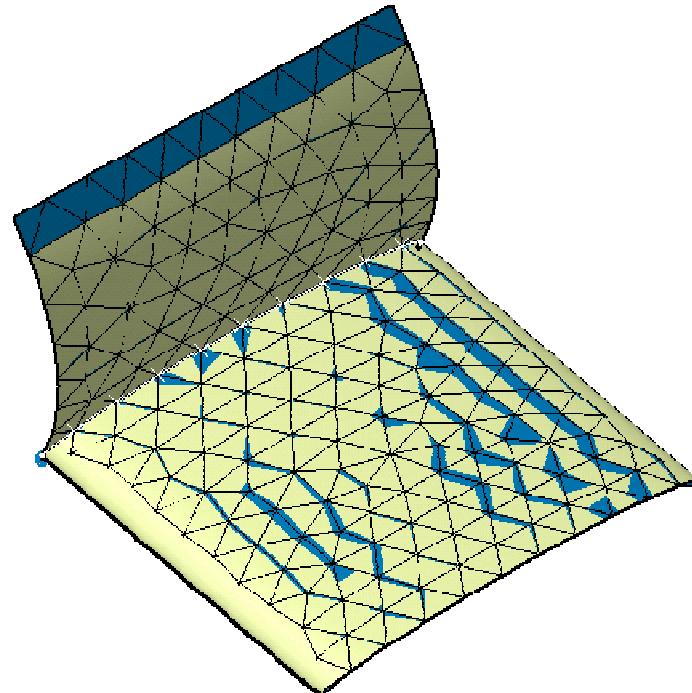
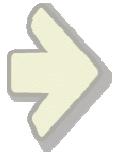
In the picture below you can see a chain link. It is not “closed” and you want to weld it.



Student Notes:

Points to Points Analysis Connection

A Points To Points Analysis Connection is introduced to establish Analysis connection between two mesh parts using points.

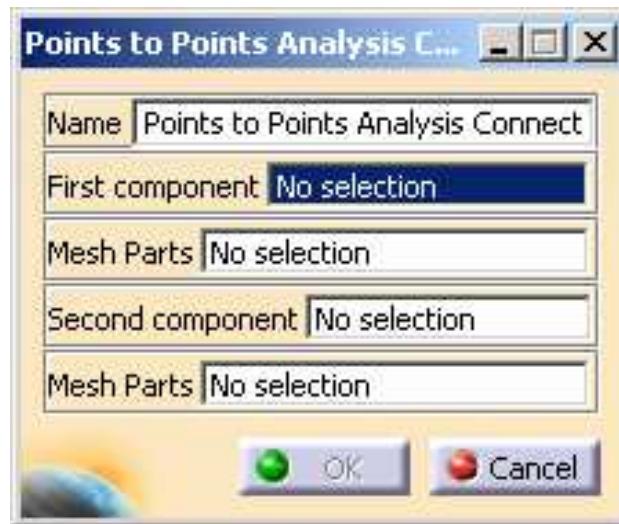


About Points to Points Analysis Connection



[Student Notes:](#)

Points to Points Analysis connection is introduced to connect part meshes to each other through point or set of points.



How to Apply the Points to Points Analysis Connection

Student Notes:

Points To Points Analysis Connection is used to connect two part meshes using set of points.

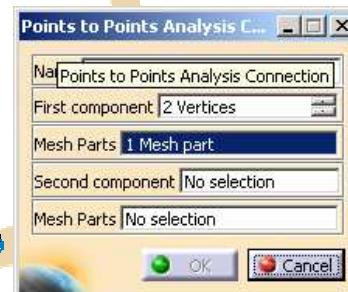
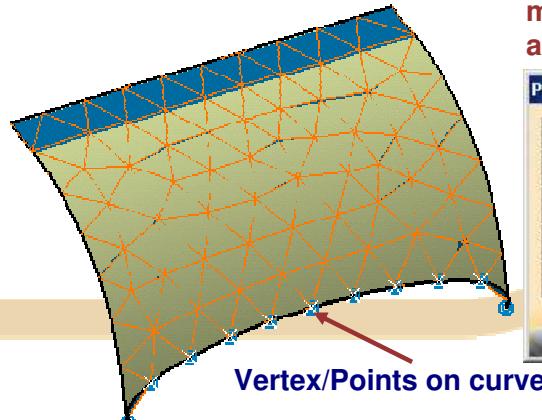


- 1 Select the 'Points to Points Connection' command



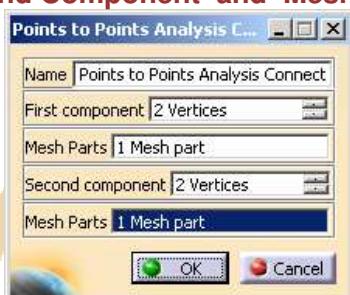
- 2

Select the vertex/points of part and corresponding mesh in 'First component' and 'Mesh Parts'

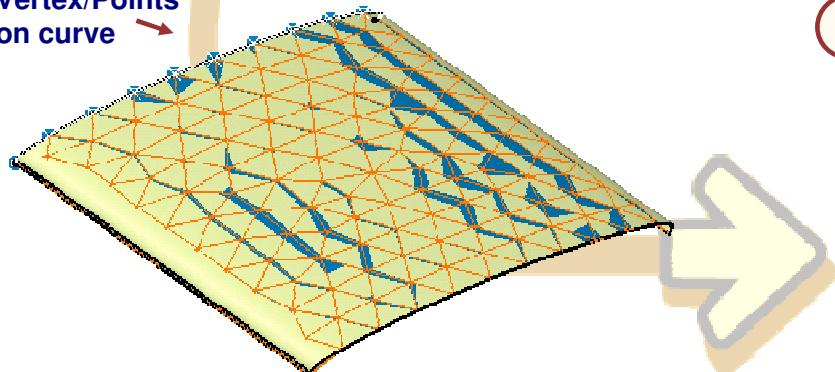


Vertex/Points on curve

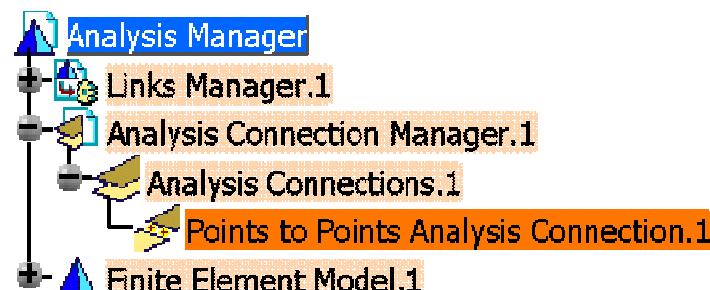
- 3 Select vertex/points of the part and the corresponding mesh in 'Second Component' and 'Mesh Parts'



Vertex/Points on curve



- 4 Click OK to confirm



To Sum Up ...

[Student Notes:](#)

You have seen CATIA V5 Tools for Analysis Connections

-  **What is Analysis connection**
-  **General Analysis Connection**
-  **Point Analysis Connection**
-  **Line Analysis Connection**
-  **Surface Analysis Connection**
-  **Points to Points Analysis Connection**

GAS Connections

[Student Notes:](#)

- Face Face Connection Properties
- Distant Connection Properties
- Welding Connection Properties
- Nodes to Nodes Connection Property
- To Sum Up

[Student Notes:](#)

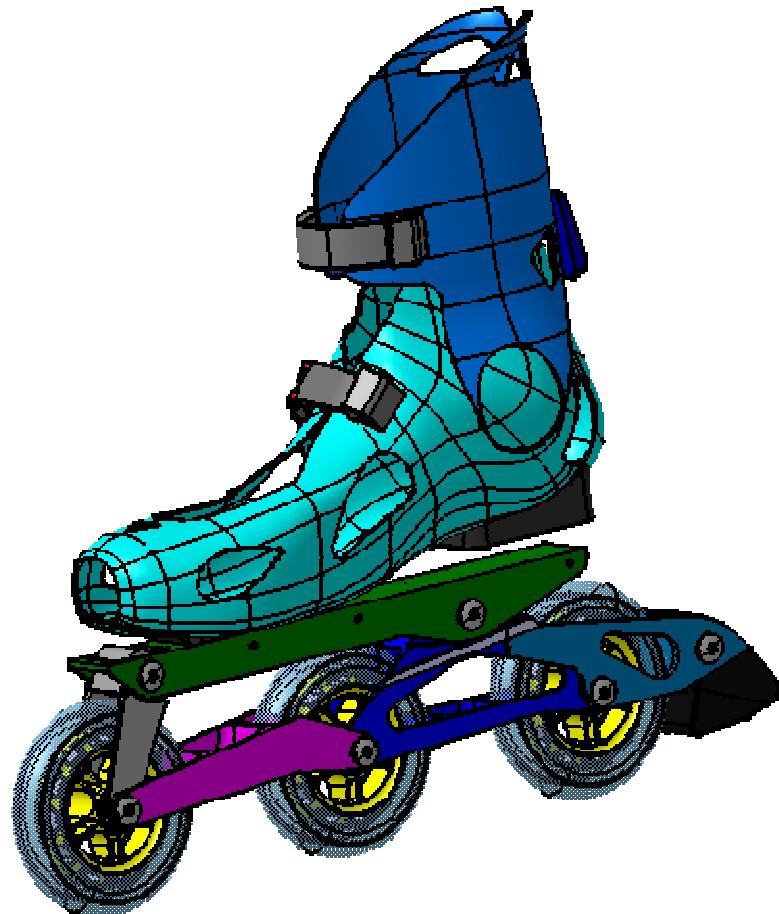
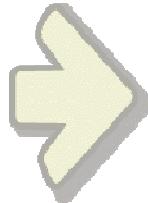
Face Face Connections

- Fastened Connection Property
- Fastened Spring Connection Property
- Contact Connection Property
- Slider Connection Property
- Pressure Fitting Connection Property
- Bolt Tightening Connection Property

[Student Notes:](#)

Fastened Connection Property

You will learn how to define a fastened connection property and when to use it



[Student Notes:](#)

What is a Fastened Connection Property

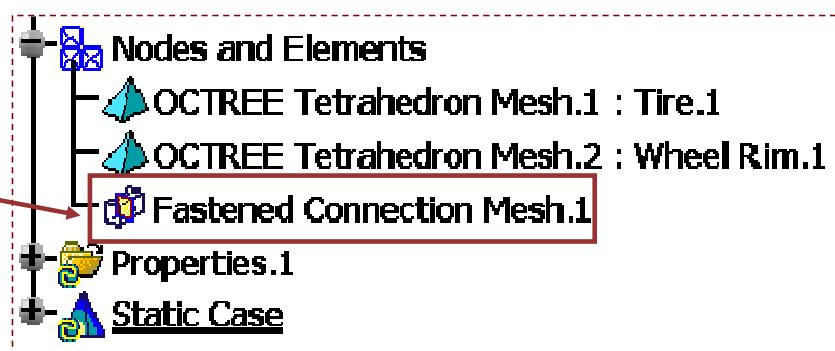
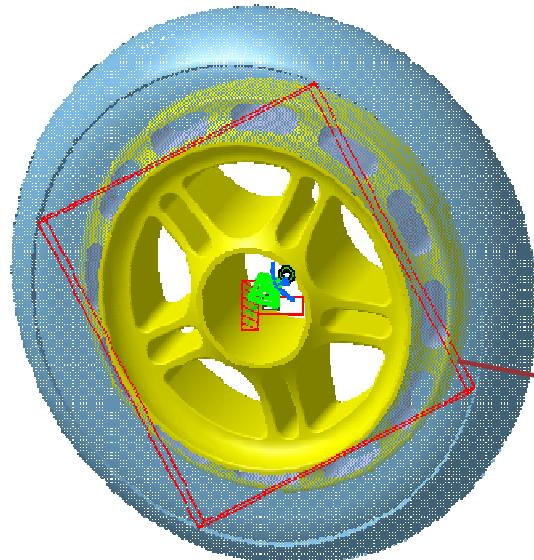


A Fastened Connection property is the link between two bodies which are fastened together at their common boundary.

From a finite element model point of view, this is equivalent to the situation where corresponding nodes of two compatible meshes are merged together. Consequently, two bodies will behave as if they were a single one. However, they can have different material properties.

Fastened Connection relations take into account elastic deformability of interfaces.

Example of a Fastened connection between a wheel and a tire:



[Student Notes:](#)

How to apply a Fastened Connection Property

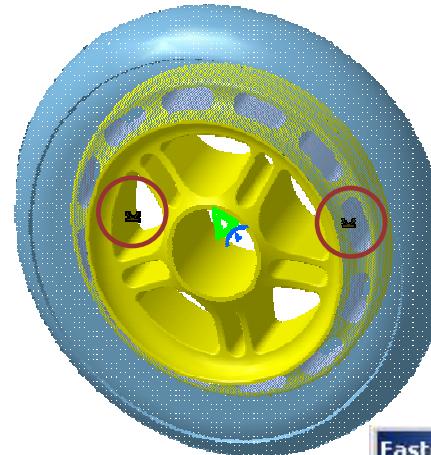


Before you begin, make sure you have defined a static case analysis and you have created the appropriate constraints

- ① Click on the “Fastened Connection ” icon

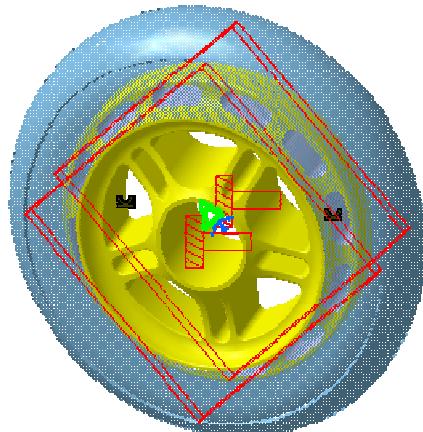


- ② Select the constraint previously created in the Assembly Design/Analysis Connection workbench



- ③ Click on Ok

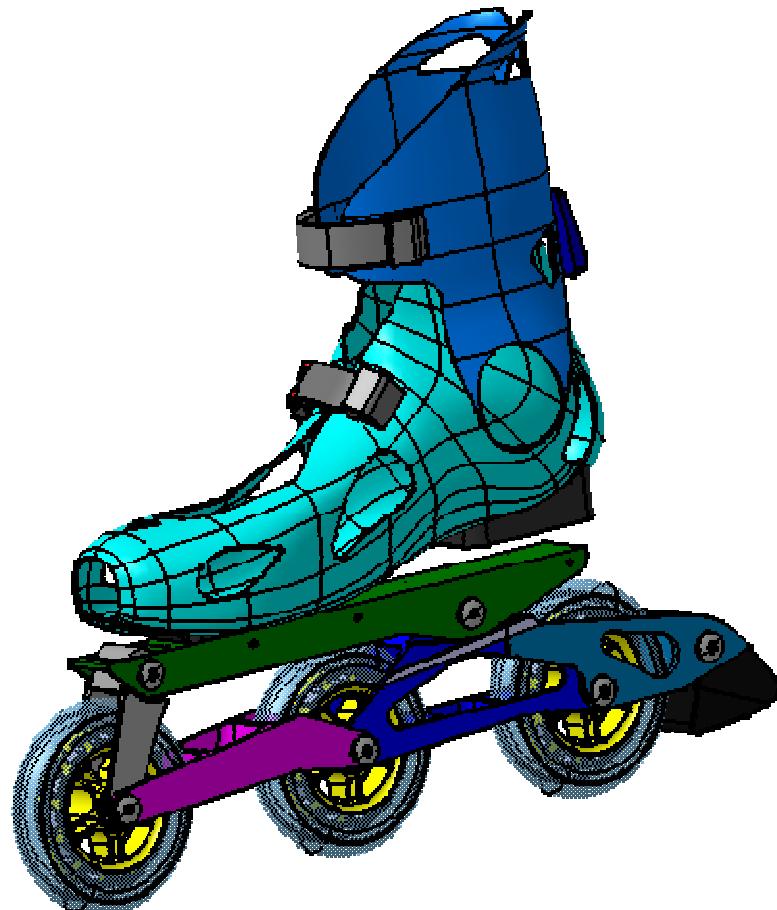
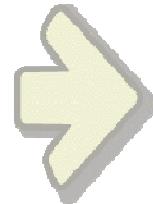
A symbol representing the Fastened Connection appears



Student Notes:

Fastened Spring Connection Property

You will learn how to define a fastened spring connection property and when to use it



What is a Fastened Spring Connection Property

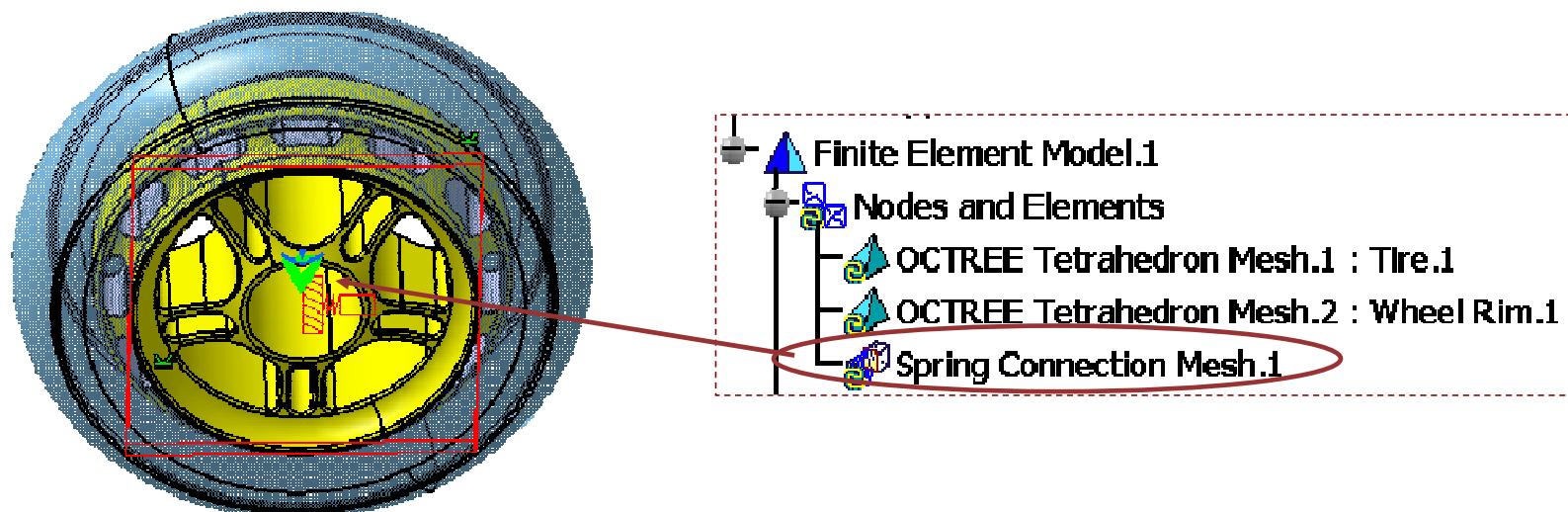


[Student Notes:](#)

A Fastened Spring Connection is an elastic link between two faces

From a finite element model viewpoint, this is equivalent to the situation where the corresponding nodes of two compatible meshes are merged together but, the rigidity is defined interactively. However, since bodies can be meshed independently, the Fastened Spring Connection is designed to handle incompatible meshes.

Example of a Fastened Spring Connection between a wheel and a tire:



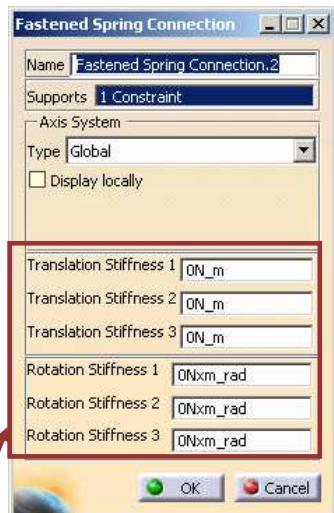
[Student Notes:](#)

How to Apply a Fastened Spring Connection Property



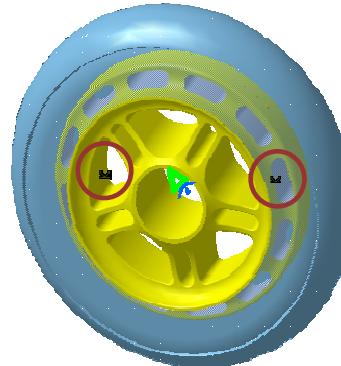
Before you begin, make sure you have defined a static case analysis and you have created the appropriate constraints

- 1 Click on the “Fastened Spring Connection” icon

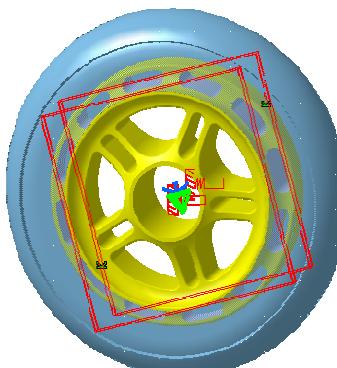


- 3 Customize the translation and rotation stiffness

- 2 Select the constraint previously created in the Assembly



- 4 Click on Ok

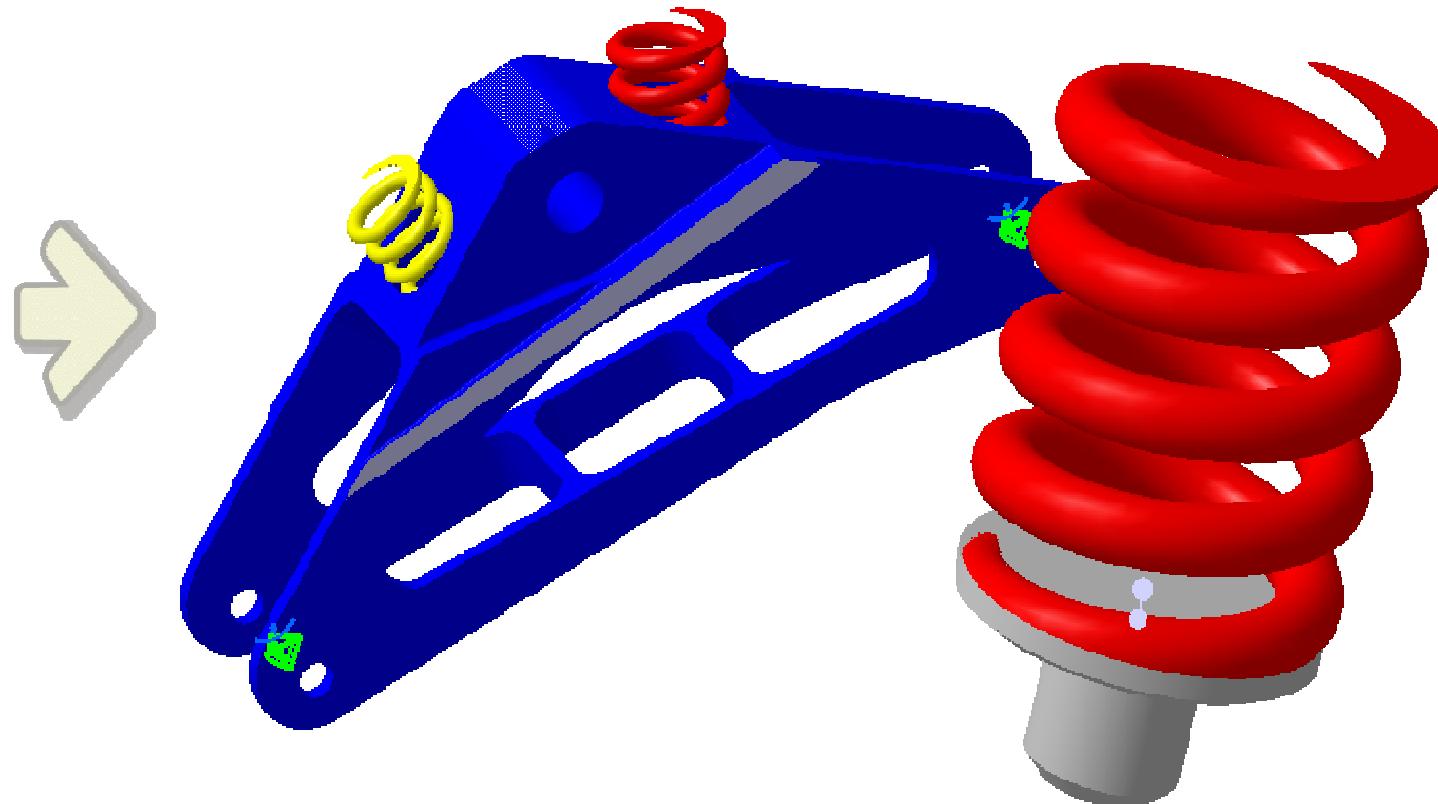


A symbol representing the Fastened Spring Connection appears

[Student Notes:](#)

Contact Connection Property

You will learn how to define a contact connection property and when to use it



[Student Notes:](#)

What is a Contact Connection Property

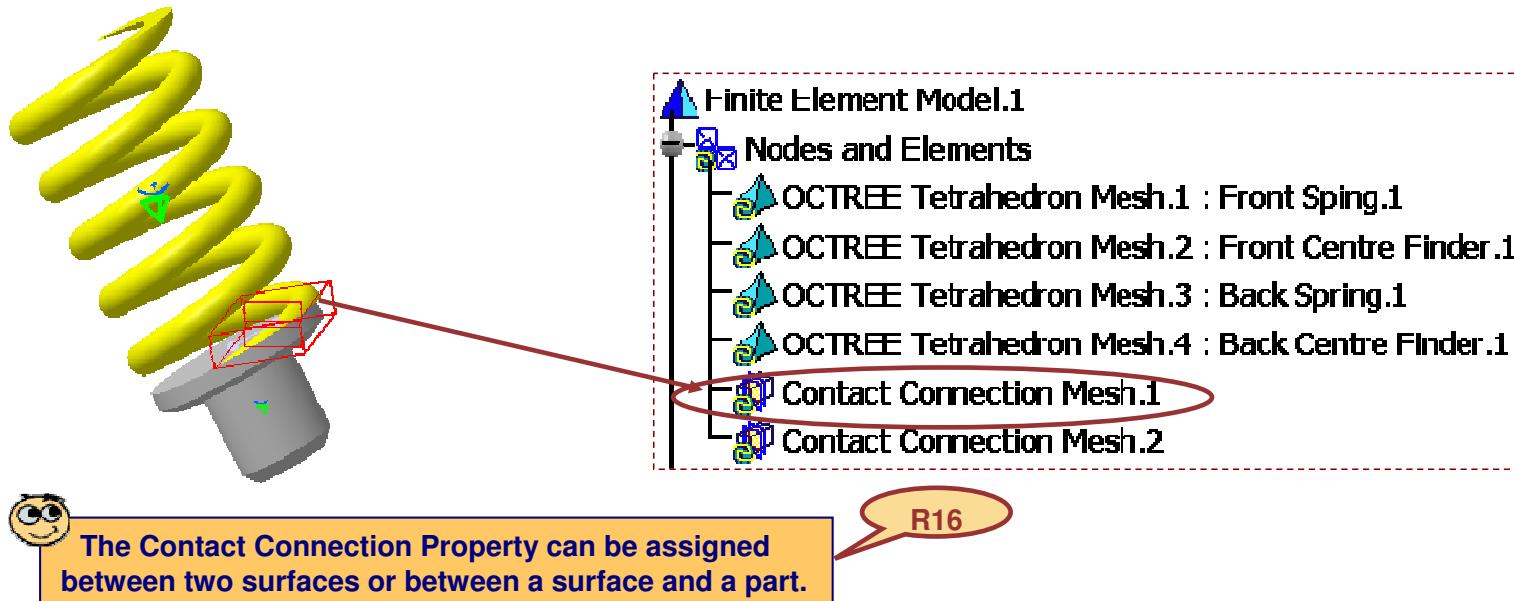


A Contact Connection is the link between two part bodies which are prevented from inter-penetrating at their common boundary

They will behave the same as if they were allowed to move arbitrarily relative to each other as long as they do not come into contact within a user-specified normal clearance. When they come into contact, they can still separate or slide relative to each other in the tangential plane, but they cannot reduce their relative normal clearance.

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

Example of a Contact Connection (applied on a Face/Face constraint):



[Student Notes:](#)

How to apply a Contact Connection Property



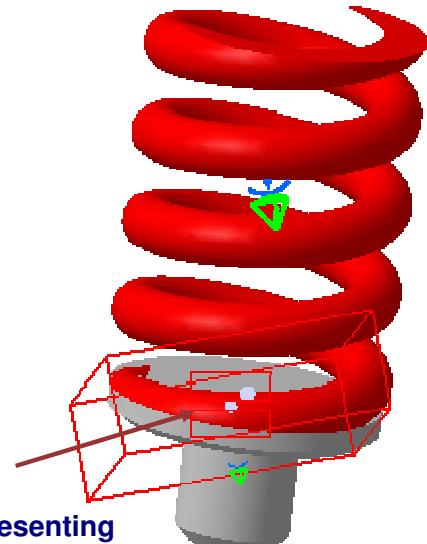
Before you begin, make sure you have defined a static case analysis and you have created the appropriate constraints

- 1 Click on the “Contact Connection ” icon

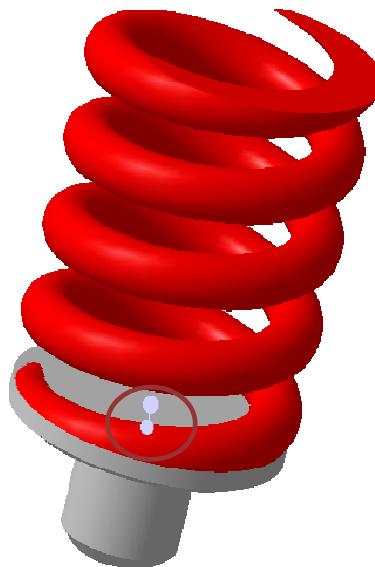


- 2 Select the constraint previously created in the Assembly Design/Analysis Connection workbench

- 3 Click on Ok



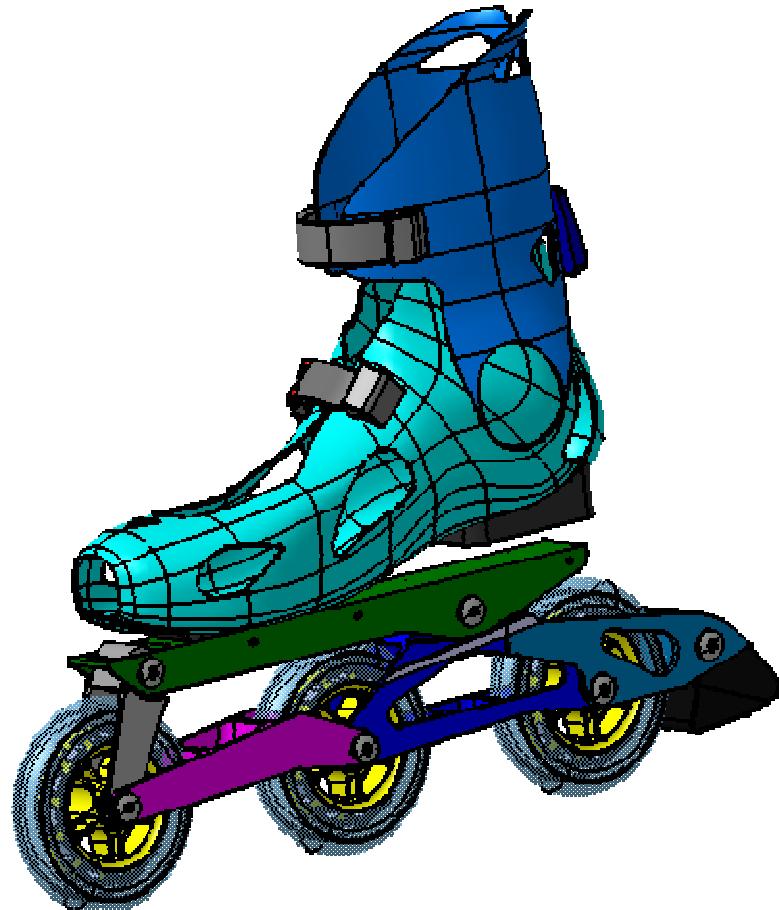
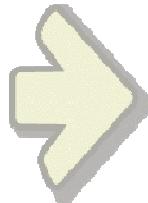
A symbol representing the contact Connection appears



[Student Notes:](#)

Slider Connection Property

You will learn how to define a slider connection property and when to use it



What is a Slider Connection Property

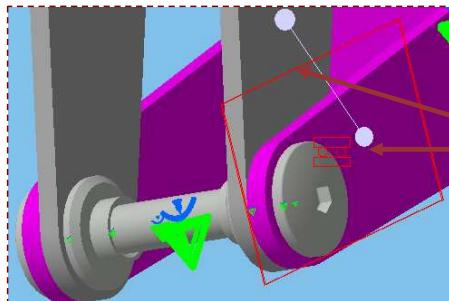
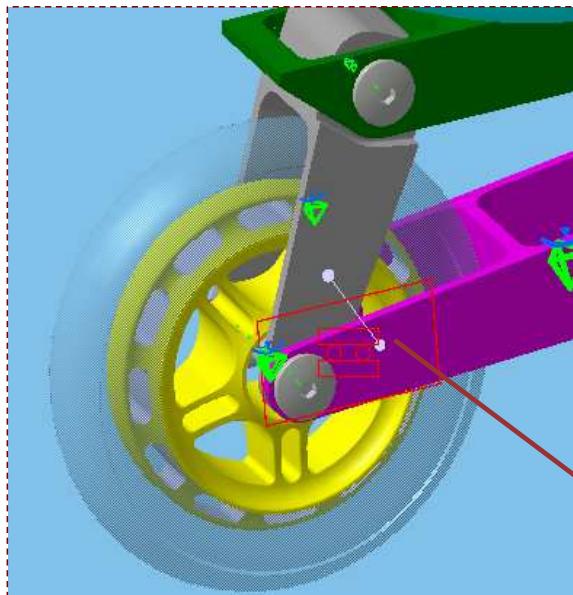


[Student Notes:](#)

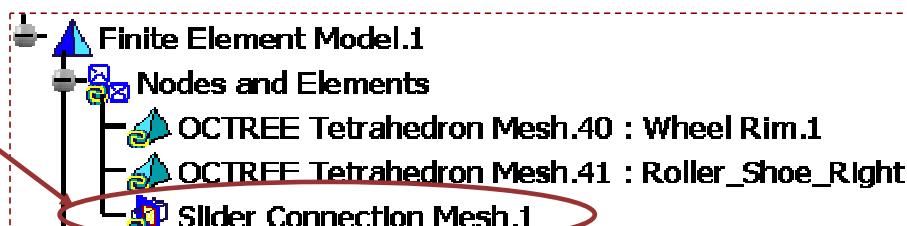
A Slider Connection is a link between two bodies which are constrained to move together in the local normal direction at their common boundary

These two bodies will behave the same as if they were allowed to slide relative to each other in the local tangential plane. The Slider Connection takes into account the elastic deformability of the interfaces.

Example of a Slider Connection (applied on a Face/Face constraint):



The Face/Face constraint was defined between the outer gray surface and the inner pink surface



[Student Notes:](#)

How to Apply a Slider Connection Property

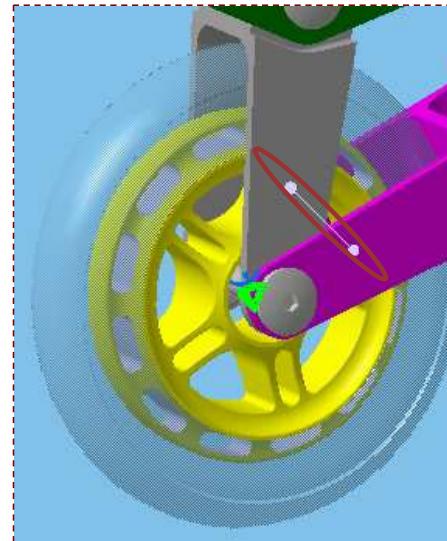


Before you begin, make sure you have defined a static case analysis and you have created the appropriate constraints

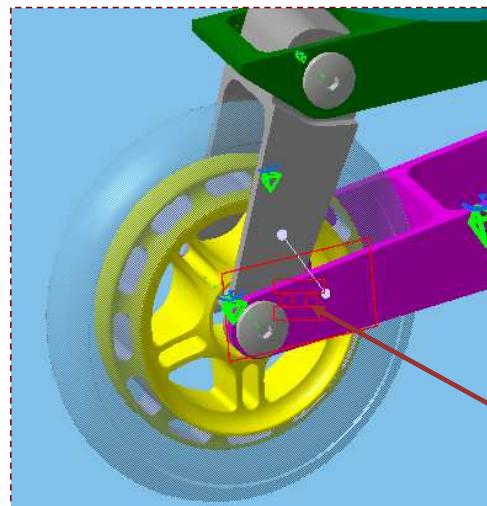
- 1 Click on the “Slider Connection ” icon



- 2 Select the constraint previously created in the Assembly Design/Analysis Connection workbench



- 3 Click on Ok

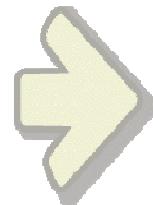


A symbol representing the Slider Connection appears

[Student Notes:](#)

Pressure Fitting Connection Property

You will learn how to define a pressure fitting connection property and when to use it



Student Notes:

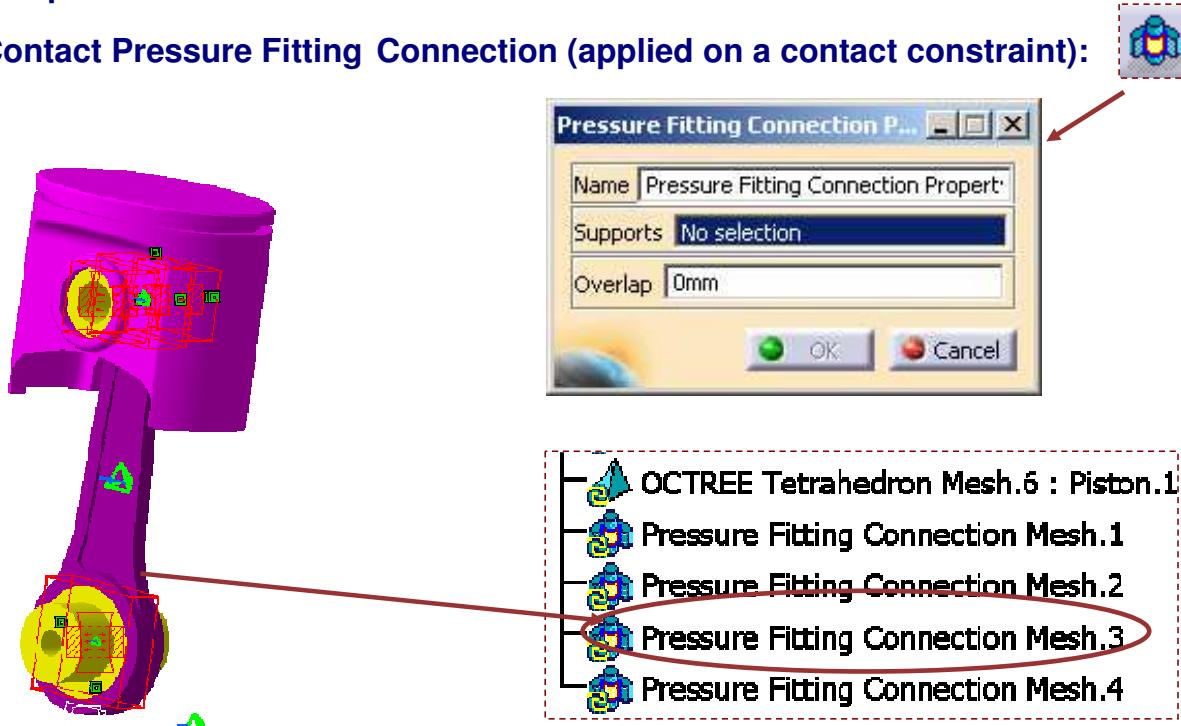
What is a Pressure Fitting Connection Property

A Pressure Fitting Connection is a link between two parts which are assembled in a Pressure Fitting configuration, which means there are interferences or overlaps between them

Along the surface normal, the connection behaves as a contact connection with negative clearance value (positive overlap). The difference lies in the tangential directions where both parts are linked together.

The Pressure Fitting Connection relations take into account the elastic deformability of the interfaces and handle incompatible meshes

Example of a Contact Pressure Fitting Connection (applied on a contact constraint):



[Student Notes:](#)

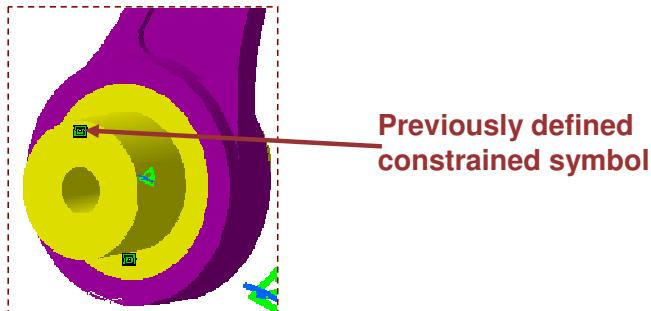
How to apply a Pressure Fitting Connection Property

Before you begin, make sure you have defined a static case analysis and you have created the appropriate constraints

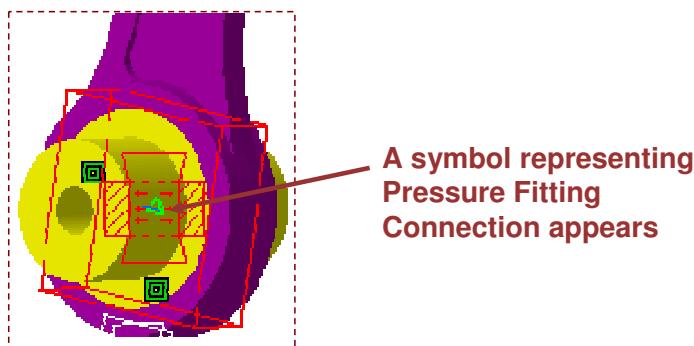
- 1 Click on the Pressure Fitting Connection icon



- 2 Select the constraint previously created in the Assembly Design/Analysis Connection workbench



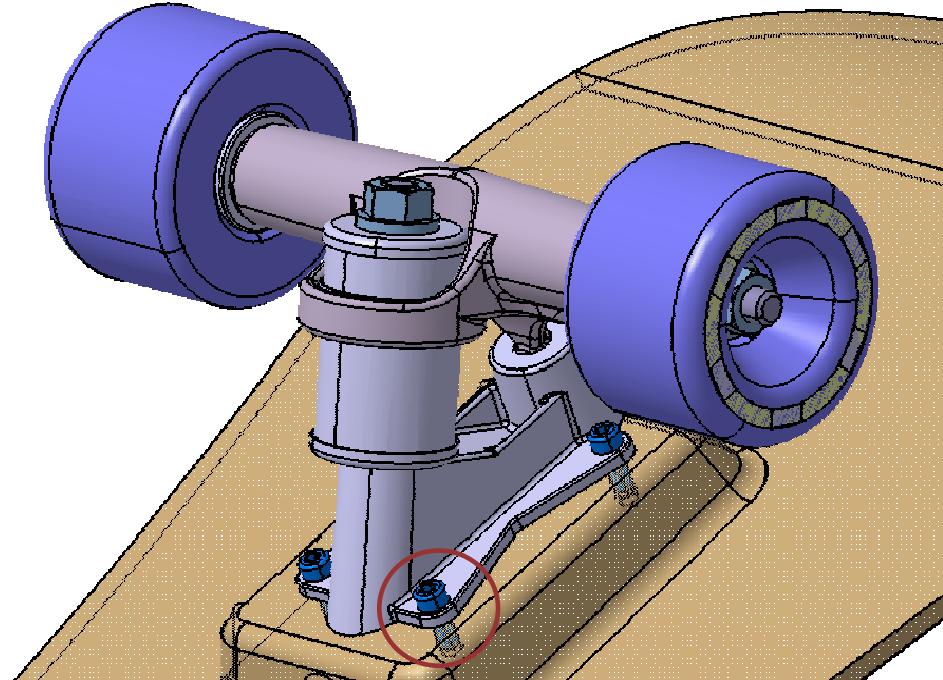
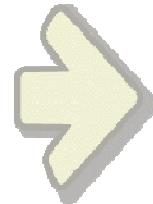
- 3 Click on Ok



[Student Notes:](#)

Bolt Tightening Connection Property

You will learn how to define a Bolt Tightening Connection Property and when to use it



[Student Notes:](#)

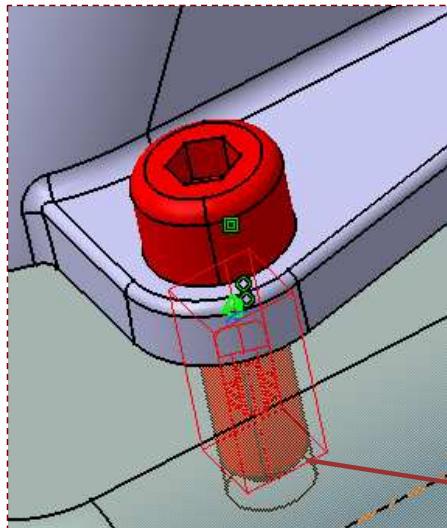
What is a Bolt Tightening Connection Property

A Bolt Tightening Connection is a connection that takes into account pre-tension in bolt-tightened assemblies

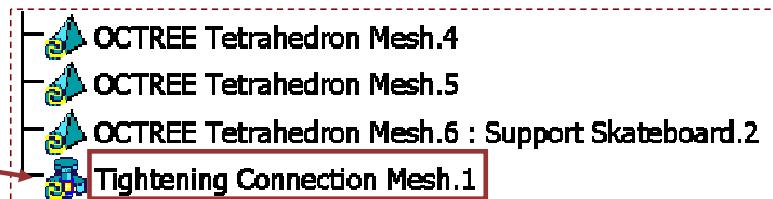
The computation is carried out according to the two-step:

- Firstly, the model is submitted to tension forces relative to bolt tightening by applying opposite forces respectively on the bolt thread and on the support tapping
- Then, the relative displacement of these two surfaces (obtained in the first step) is imposed while the model is submitted to user loads

During these two steps, the bolt and the support displacements are linked in the direction normal to the bolt axis.



Bolt Tightening Connection requires a ‘surface contact’ assembly constraint between the bolt thread and the bolt support tapping. These surfaces must be coincident



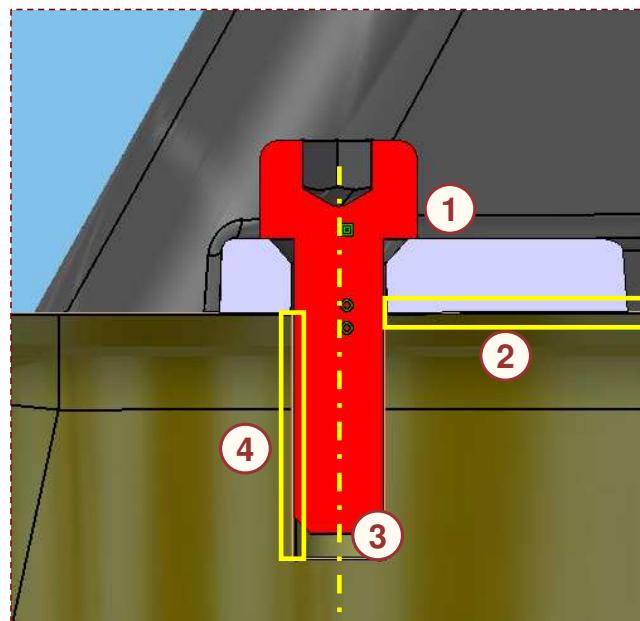
[Student Notes:](#)

How to apply a Bolt Tightening Connection Property (1/2)

Before you begin, make sure that all the needed assembly constraints were created.

In this section view, you can see the screw, the skate board's truck and the board. To be able to define a “Bolt Tightening” connection some constraints must have been previously defined:

1. “Surface Contact” constraint between the screw and the truck
2. “Surface Contact” constraint between the truck and the board
3. “Coincident” constraint between the screw and the holes board axis
4. “Surface Contact” constraint between the outer surface of the screw and the inner surface of the board hole.



[Student Notes:](#)

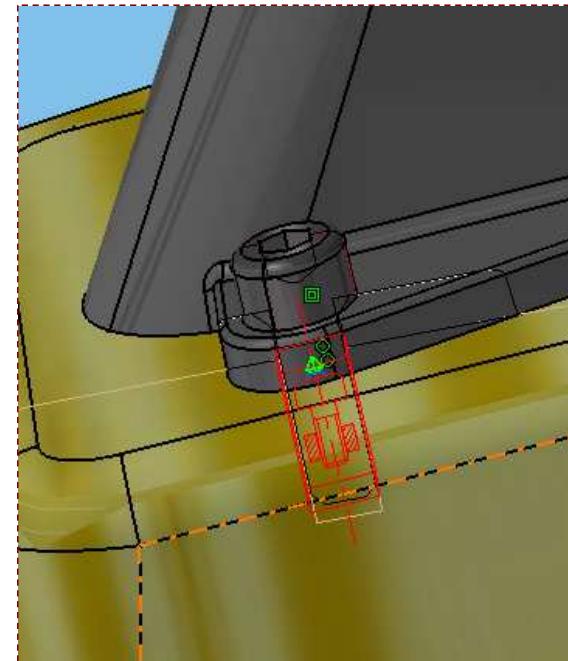
How to apply a Bolt Tightening Connection Property (2/2)

Before you begin, make sure that all the needed assembly constraints were created

- ① Click on the “Bolt Tightening” icon



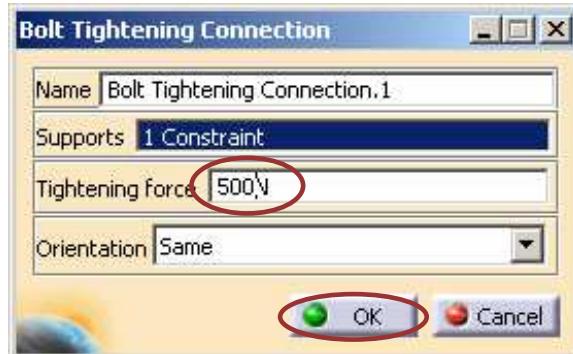
A symbol representing the Bolt Tightening Connection appears



- ② Select the “Surface contact” constraint previously created (the one between the screw and the board’s hole).

- ③ Enter a force value

And click on Ok



Distant Connections

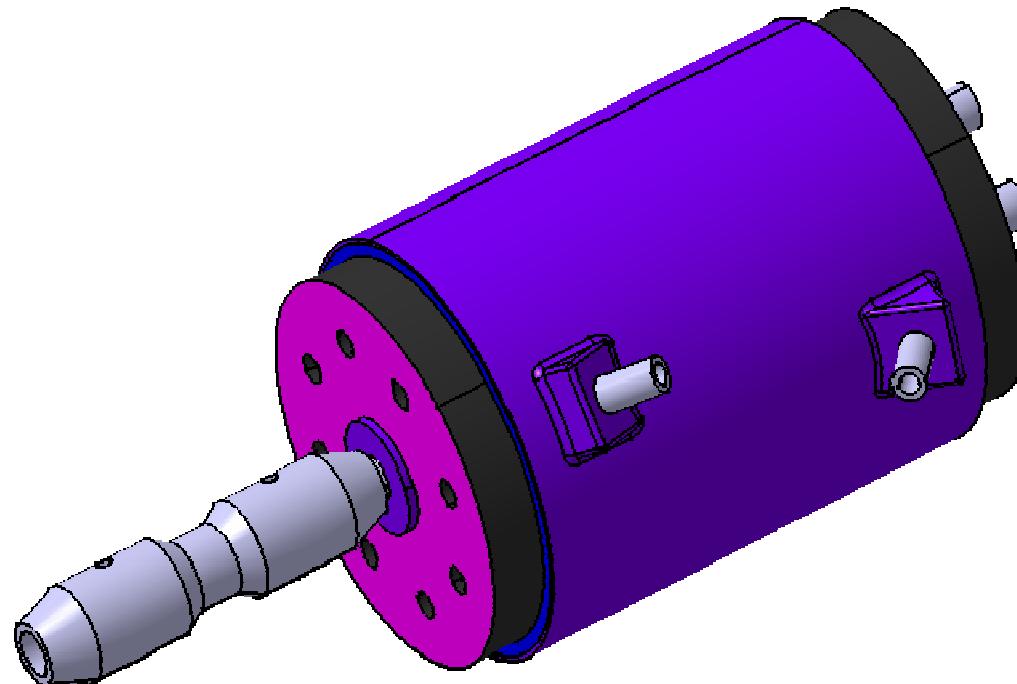
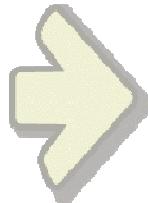
[Student Notes:](#)

- Rigid Connection Property
- Smooth Connection Property
- Virtual Bolt Tightening Connection Property
- Virtual Spring Bolt Tightening Connection Property
- User-defined Connection Property

[Student Notes:](#)

Rigid Connection Property

You will learn how to define a Rigid connection property and when to use it



What is a Rigid Connection Property

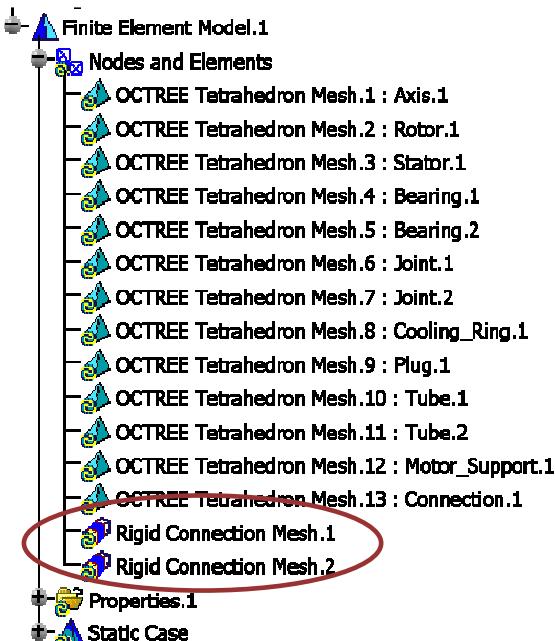
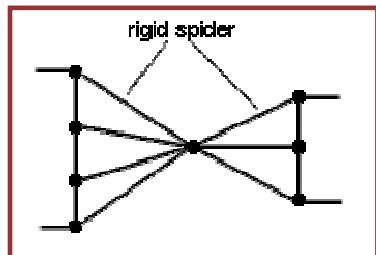
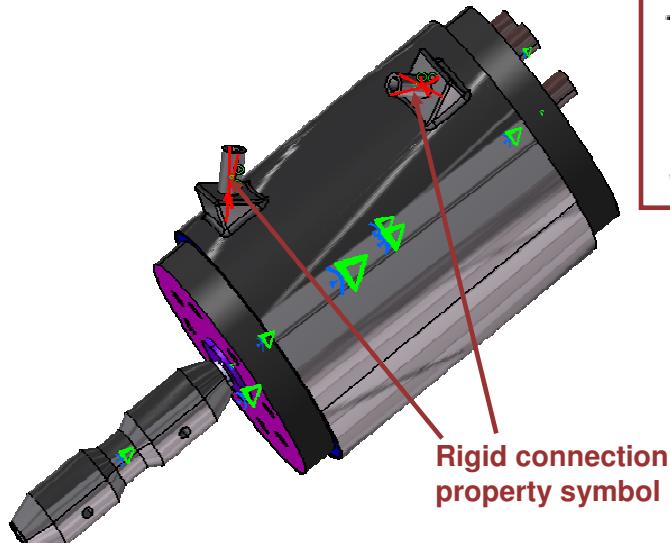


[Student Notes:](#)

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 is created at the midpoint between centroids of the two systems of points represented by the nodes of the two meshes. This node is connected by a rig-beam element to each node of the first and of the second meshes.



[Student Notes:](#)

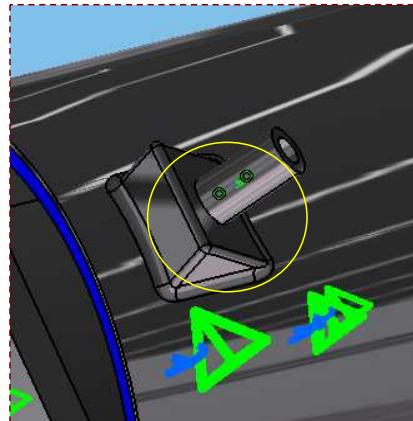
How to apply a Rigid Connection Property

Before you begin, make sure that all the needed assembly constraints were created

- ① Click on the “Rigid Connection ” icon



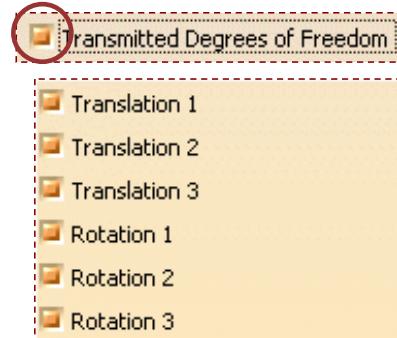
- ② Select an assembly constraint i.e “coincidence”



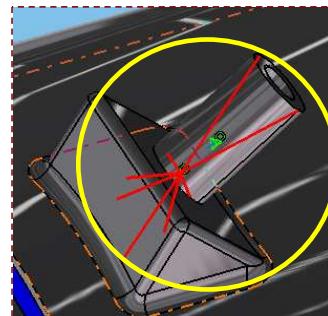
The product below is a small scale model Brushless motor.

- ③ Check the Transmitted Degrees of Freedom option.

If needed, you can also transmit some degree of freedom to the distant connection



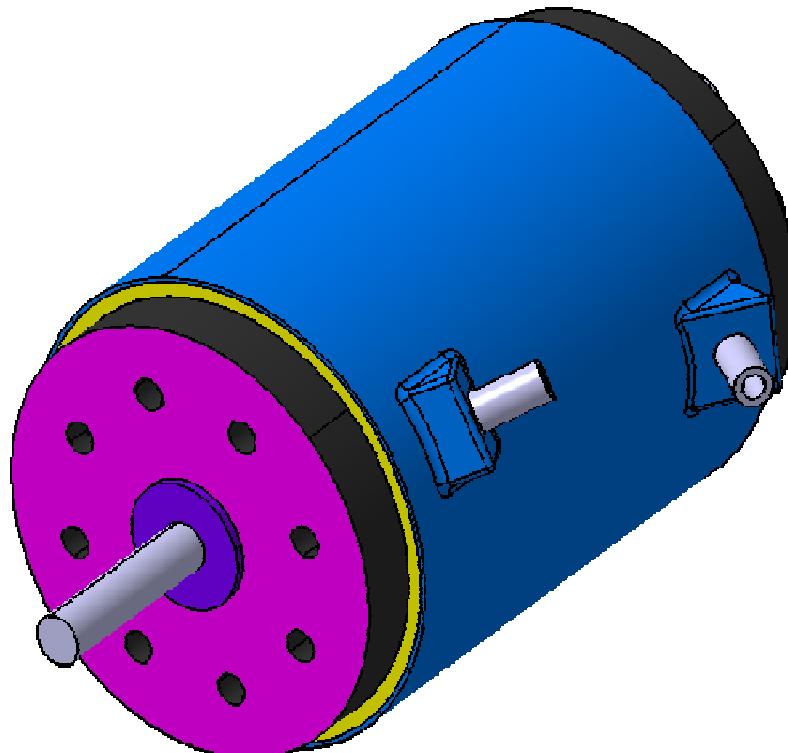
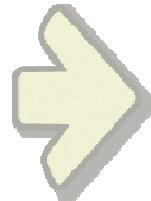
- ④ Click on Ok



Student Notes:

Smooth Connection Property

You will learn how to define a Smooth connection property and when to use it



[Student Notes:](#)

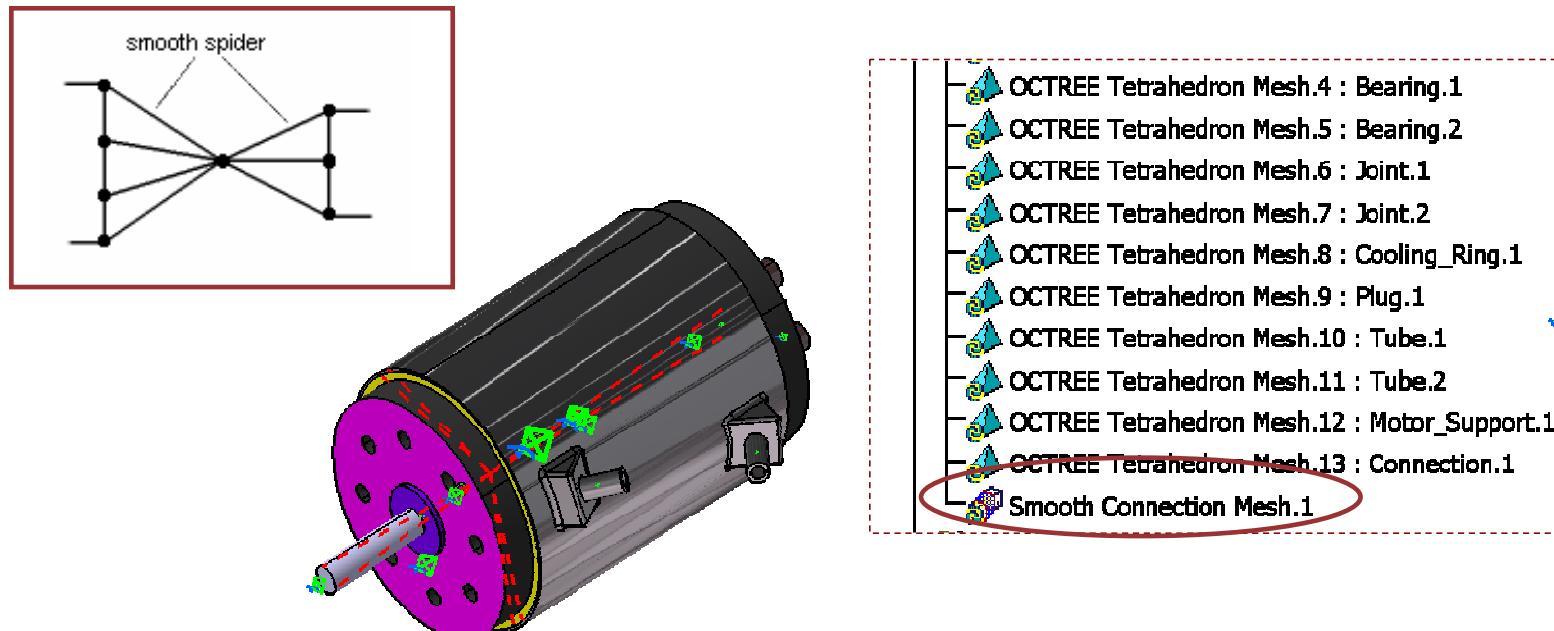
What is a Smooth Connection Property



A Smooth Connection is the link between two bodies which are fastened together at their common boundary, and will behave approximately as if their interface was soft

The Smooth Connection relations take approximately into account the elastic deformability of the interfaces

A central node is created at the midpoint between centroids of the two systems of points represented by the nodes of the two meshes. This node is connected by two spider elements to all nodes of the first and of the second meshes.



[Student Notes:](#)

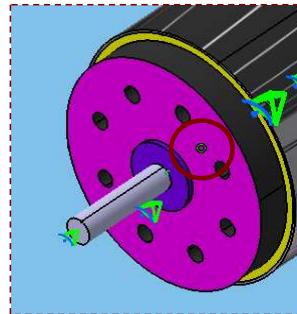
How to apply a Smooth Connection Property

Before you begin, make sure that all the needed assembly constraints were created

- ① Click on the “Smooth Connection ” icon 

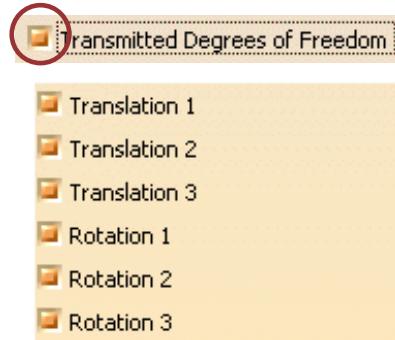


- ② Select an assembly constraint i.e “coincidence”

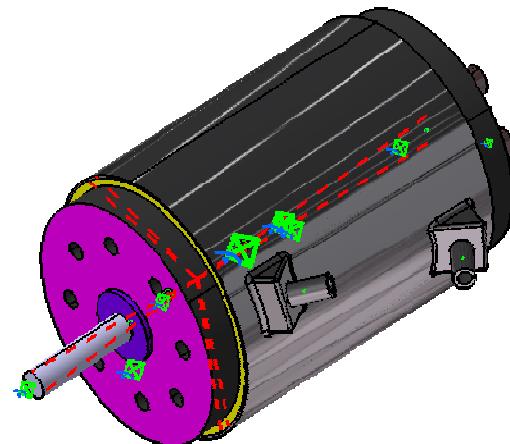


The product below
is a small scale
model Brushless
motor

- ③ Optional: Check the “Transmitted Degrees of Freedom” option to transmit required DOF to the distant connection



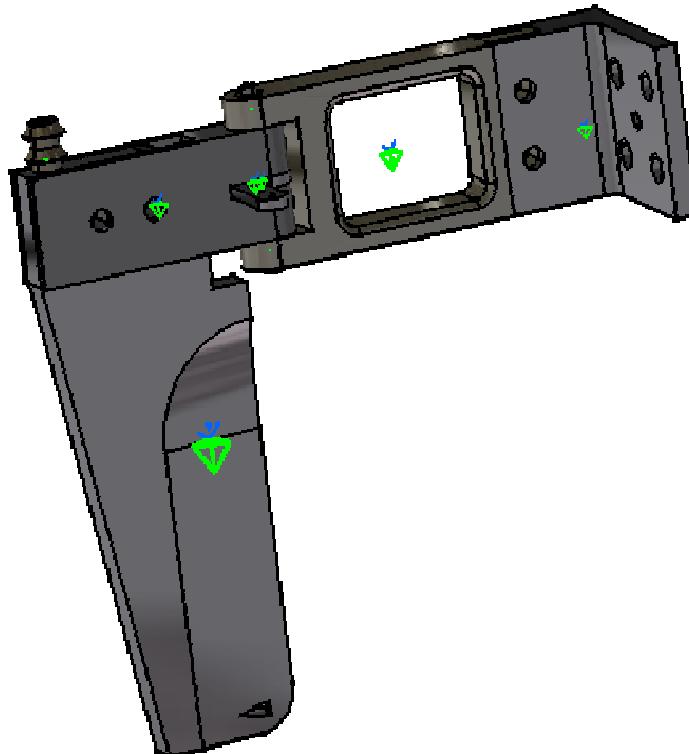
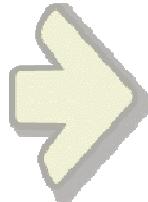
- ④ Click on Ok



Student Notes:

Virtual Bolt Tightening Connection Property

You will learn how to define a Virtual Bolt Tightening Connection Property and when to use it



Student Notes:

What is a Virtual Bolt Tightening Connection Property

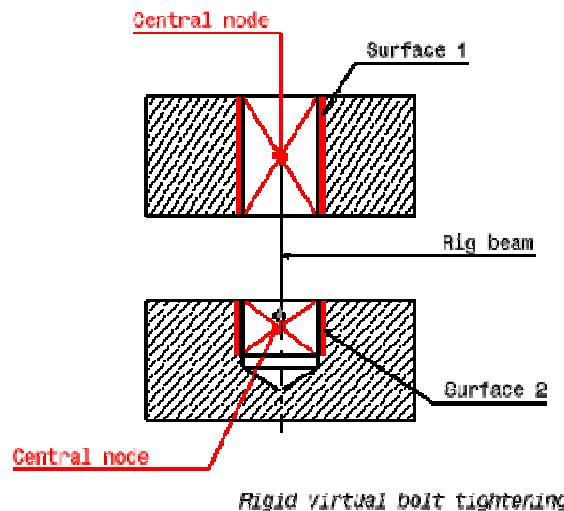


A Virtual Rigid Bolt Tightening Connection is a connection that takes into account pre-tension in a bolt-tightened assembly in which the bolt is not included

The computation is carried out according to the two-step:

- The model is submitted to tension forces relative to bolt tightening by applying opposite forces respectively on the first surface (S1) and the second surface (S2) of the assembly constraint
- Then, the relative displacement of these two surfaces (obtained in the first step) is imposed while the model is submitted to user loads

During these two steps, the rotations of both surfaces and the translations perpendicular to the coincidence constraint axis are linked together, while taking into account the elastic deformability of the surfaces



Once the geometric assembly **positioning constraints** are defined at the Product level (Assembly or Analysis Connection workbenches), the user can specify the physical nature of the constraints: **Virtual Rigid Bolt Tightening Connection**

When creating this connection, all the positioning constraints can be selected



You can define this property even when analysis connection is defined between two points. In this case, the bolt tightening orientation will be defined by the direction between these two points.

R16

[Student Notes:](#)

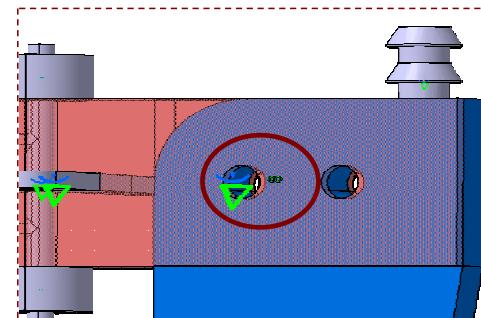
Applying a Virtual Bolt Tightening Connection Property

Before you begin, make sure that all the needed assembly constraints were created

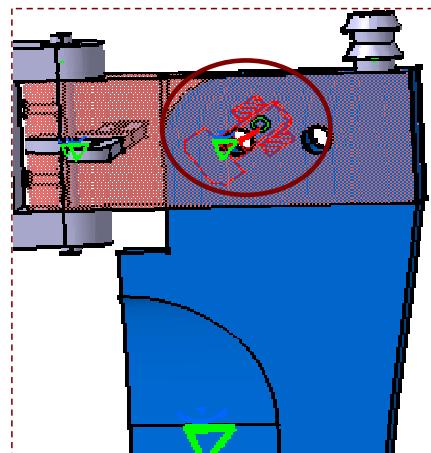
- ① Click on the “Virtual Bolt Tightening ” icon



- ② Select an assembly constraint
i.e “coincidence”



- ③ Enter a tightening force
value and click on Ok

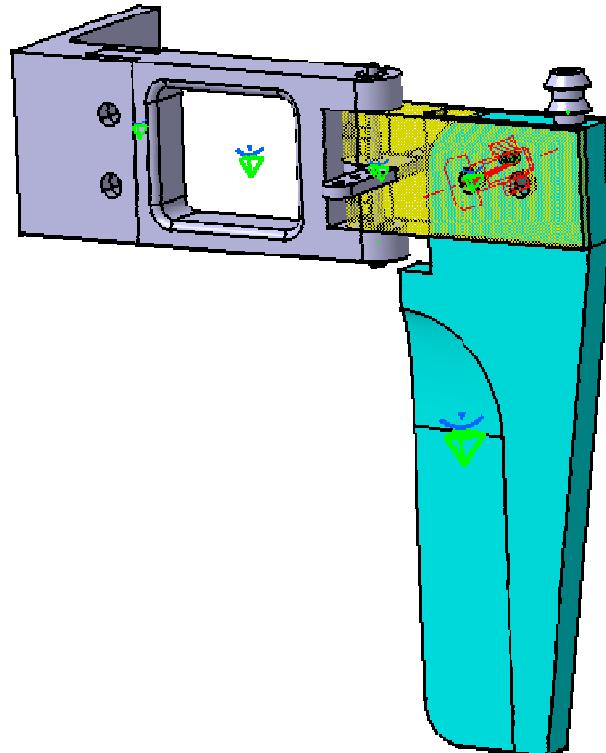
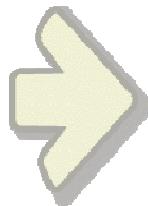


The product below
is a small scale
model off- shore's
rudder

[Student Notes:](#)

Virtual Spring Bolt Tightening Connection

You will learn how to define a Virtual Spring Bolt Tightening Connection Property and when to use it



[Student Notes:](#)

What is a Virtual Spring Bolt Tightening Connection Property

A Virtual Spring Bolt Tightening Connection is a connection that takes into account pre-tension in a bolt-tightened assembly in which the bolt is not included

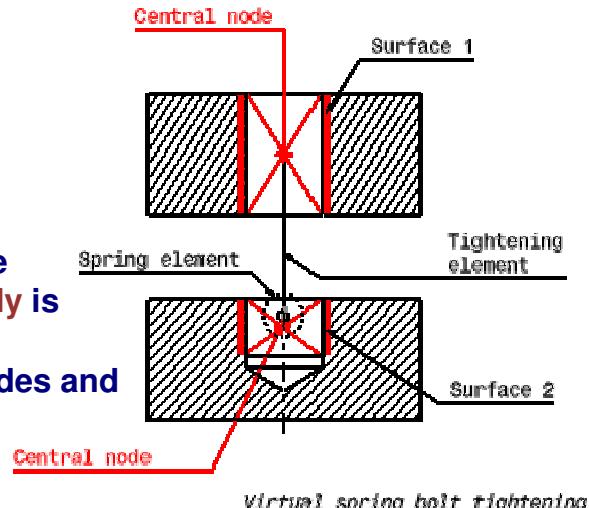


The computation is carried out according to the two-step:

- the model is submitted to tension forces relative to bolt tightening by applying opposite forces respectively on the first surface (S1) and the second surface (S2) of the assembly constraint
- Then, the relative displacement of these two surfaces (obtained in the first step) is imposed while the model is submitted to user loads

During these two steps, the rotations of both surfaces and the translations perpendicular to the coincidence constraint axis are linked together, while taking into account the elastic deformability of the surfaces

- A **central node** is created at the centroid of each surface
- For each surface/central node couple, a set of mean rigid body is generated to link the **average displacement** of the central nodes and the nodes of the surface



- The **first central node** is linked to the duplicate of the **second central node** using a **tightening element**

- The **second central node** is linked to its duplicate using a **spring element** the characteristics of which are defined by the user



You can define this property even when analysis connection is defined between two points. In this case, the bolt tightening orientation will be defined by the direction between these two points.

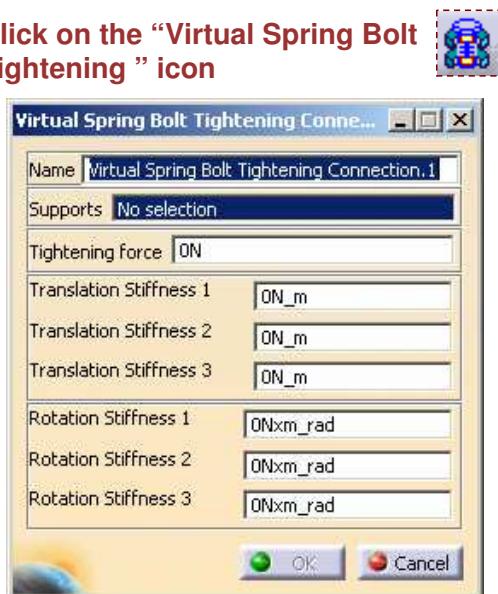
R16

[Student Notes:](#)

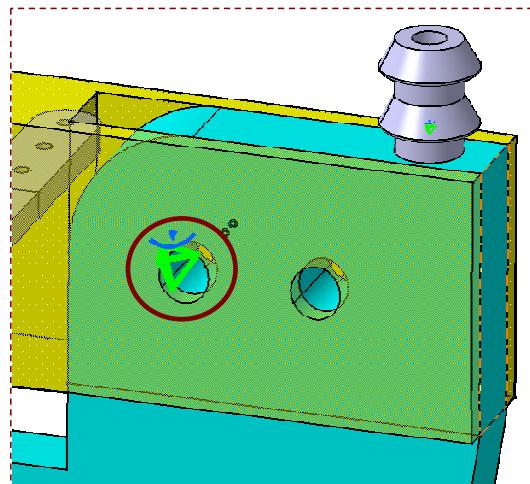
Applying a Virtual Spring Bolt Tightening Connection

Before you begin, make sure that all the needed assembly constraints were created

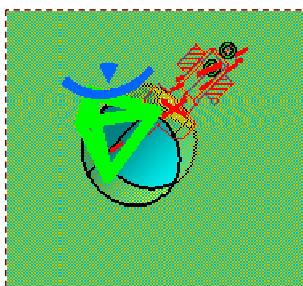
- ① Click on the “Virtual Spring Bolt Tightening” icon



- ② Select an assembly constraint i.e “coincidence”



- ③ Enter the value of the force and stiffness parameters and click Ok

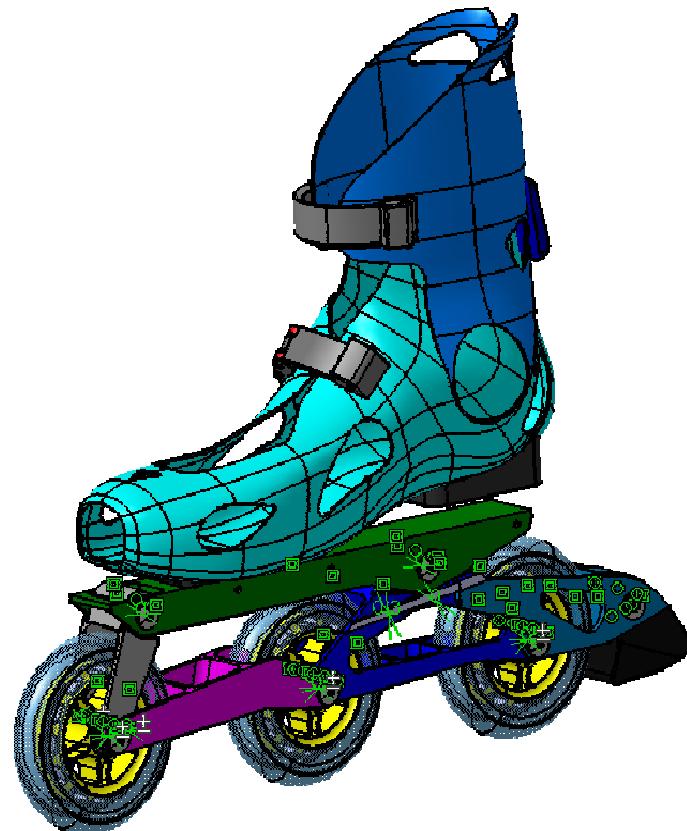
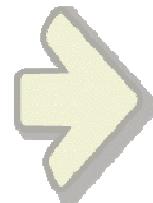


The product below is a small scale model off- shore's rudder

[Student Notes:](#)

User-defined Connection Property

You will learn how to define a User-defined Connection Property and when to use it



[Student Notes:](#)

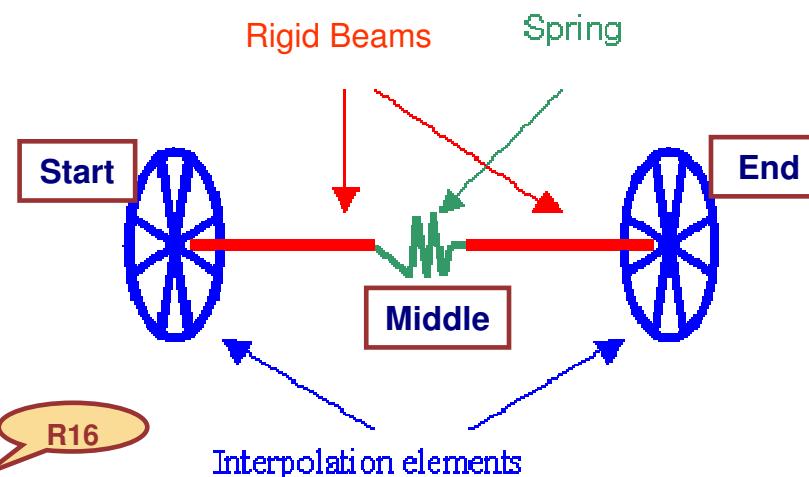
What is a User-defined Connection Property



User-Defined Connection Property allow you to specify the types of elements as well as their associated properties included inside a distant connection

The User-Defined Connection Property is the tool that allows you to define any type of constraints. All the connections you have seen so far are a particular case of this tool

You have to specify the nature of 3 connections at 3 different places: Start/Middle/End



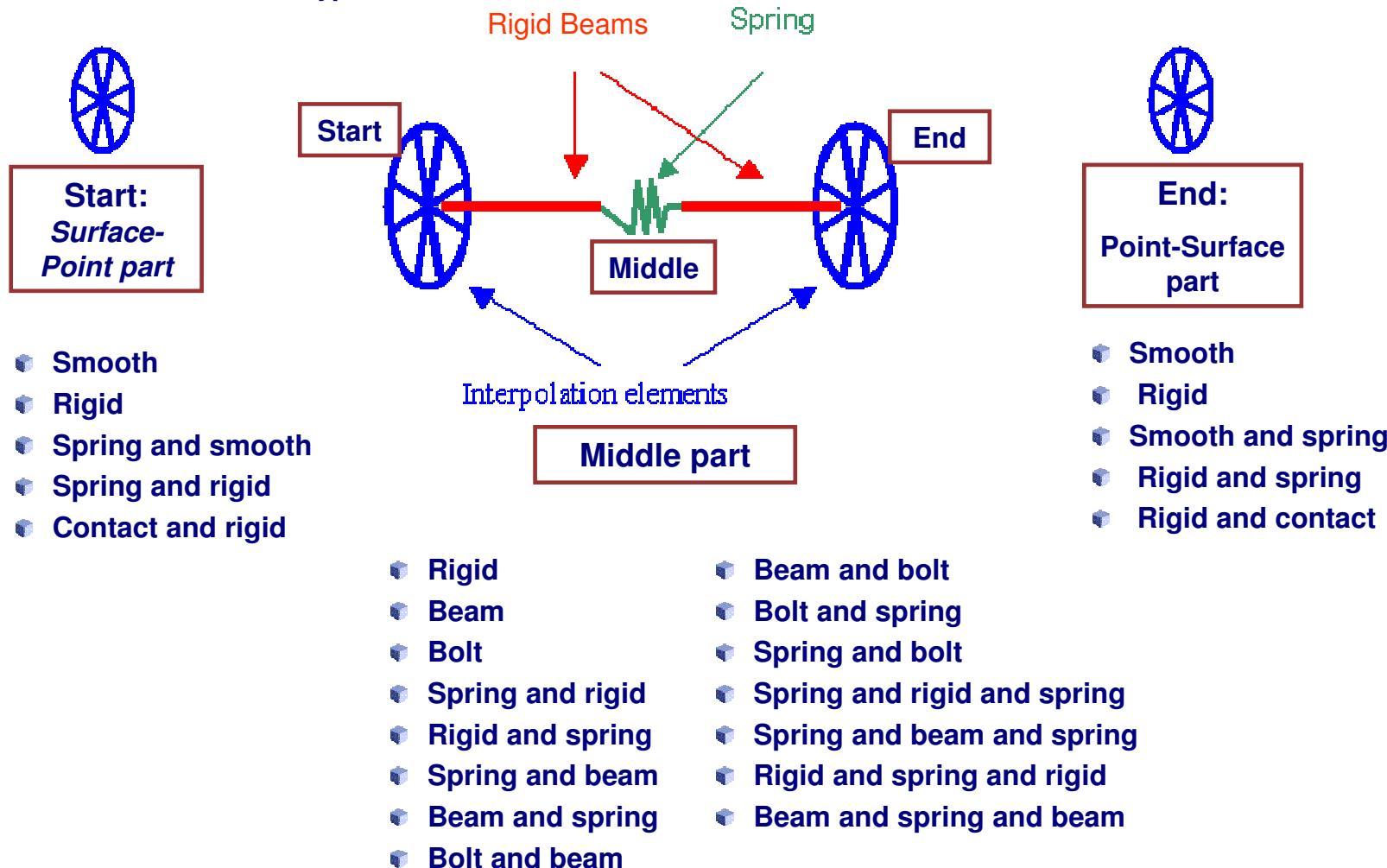
R16
You can define this property even when analysis connection is defined between two points. In this case, the bolt tightening orientation will be defined by the direction between these two points.

[Student Notes:](#)

What are the Possible Combinations



You will define which types of element will be featured in the connection

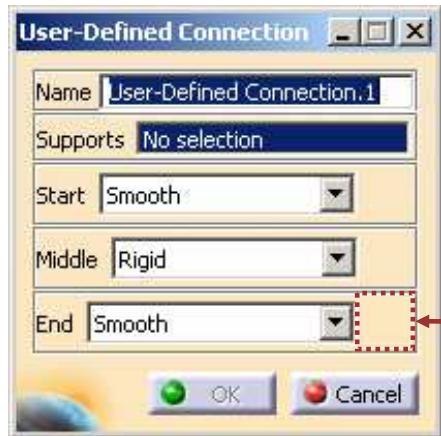


[Student Notes:](#)

How to apply a “User Defined Distant“ Connection

Before you begin, make sure that all the needed assembly constraints were created

- 1 Click on the “User Defined Distant Connection” icon



③

Define the types of the elements to be featured in the connection: Start, Middle and End

Depending on the “Start” connection, Middle and End elements will be proposed to you

④

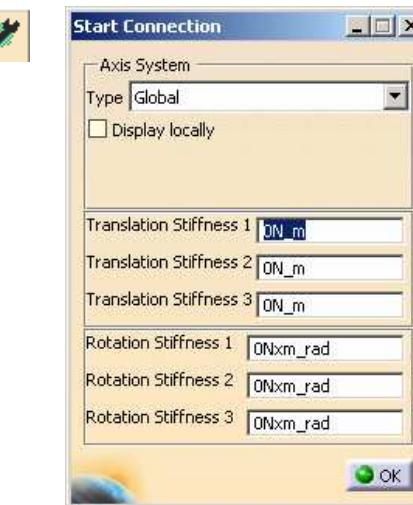
Click on the icon below if available and enter the expected data

⑤

Click on Ok

Depending on the selected combination type, the appropriate properties will be proposed

- 2 Select the constraint or Analysis connection to be used as support, either from the specification tree or from the assembly



Welding Connections

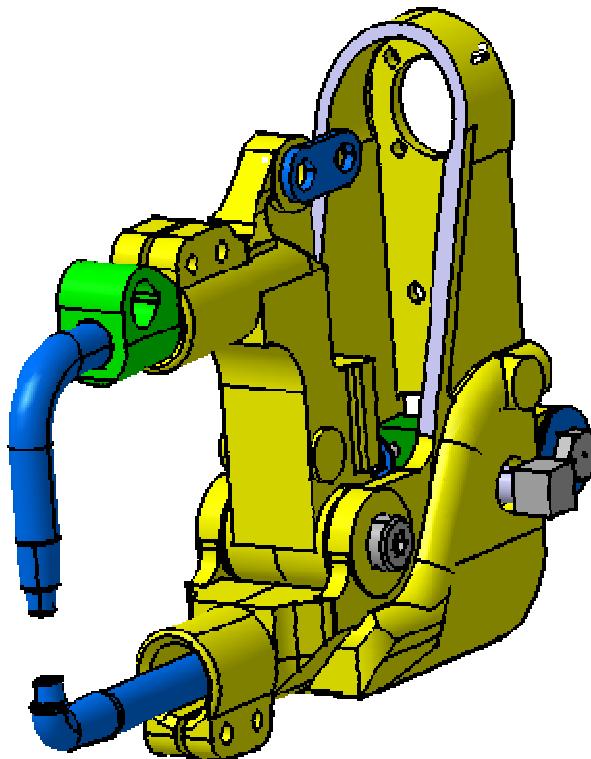
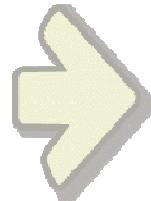
[Student Notes:](#)

- Spot Welding Connection Property
- Seam Weld Connection Property

[Student Notes:](#)

Spot Welding Connection Property

You will learn how to define a Spot Welding connection property and when to use it



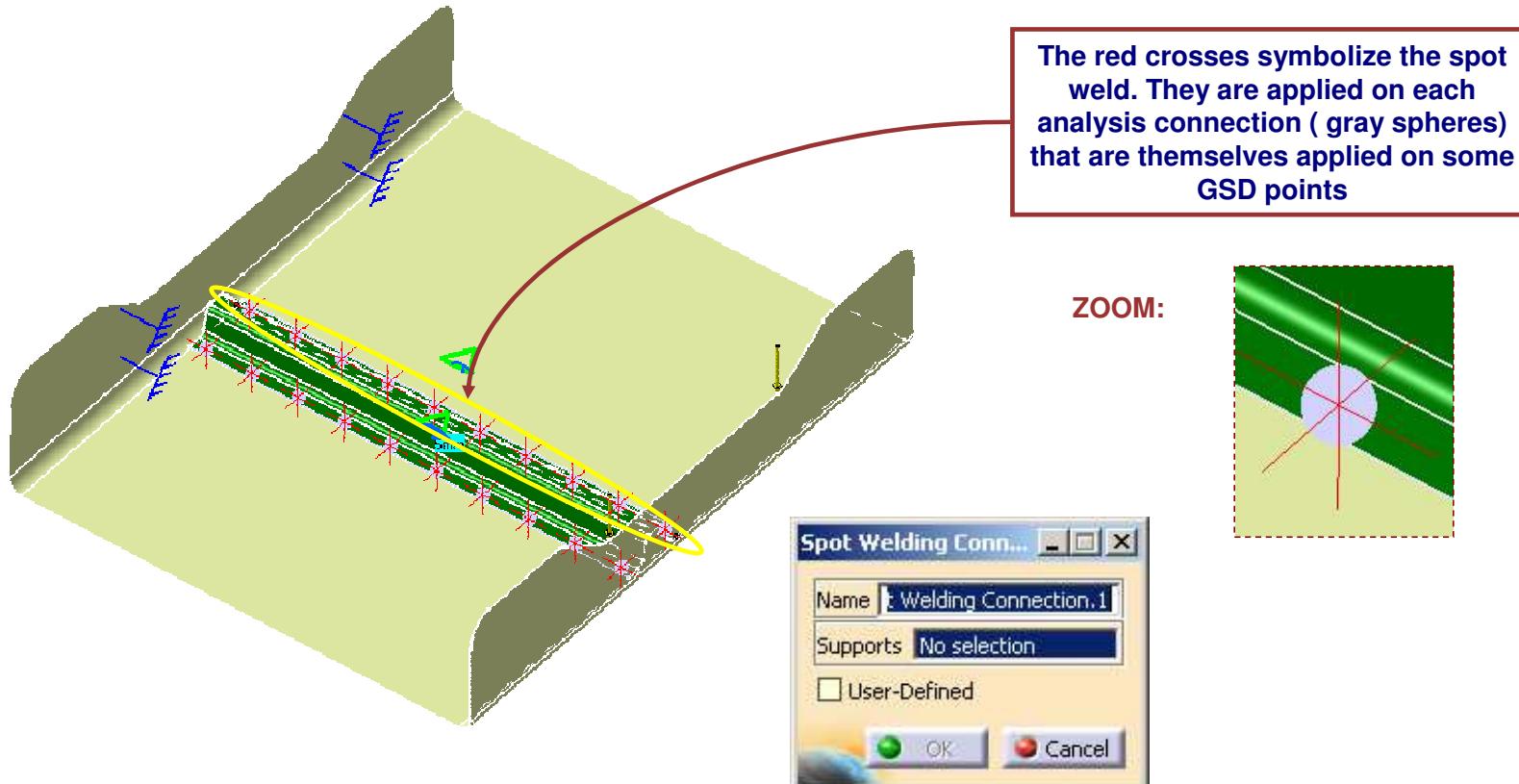
[Student Notes:](#)

What is a Spot Welding Connection Property



This type of connections allows to simulate “spot welding” between two parts.

Spot Welding Connections need as support, some Point Analysis Connections , which need themselves some points to be applied on.



[Student Notes:](#)

How to apply a Spot Welding Connection Property

Before you begin, make sure that all the needed assembly constraints were created

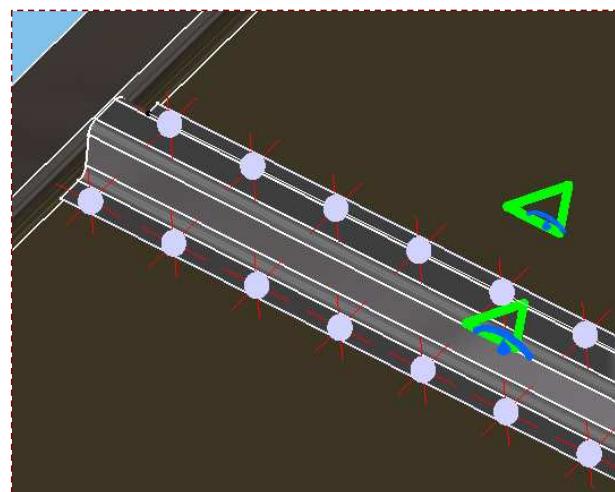
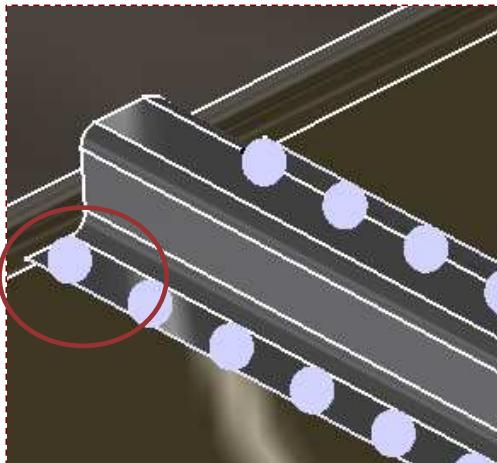
- 1 Click on the “Spot Welding Connection” icon 



- 3 Click on Ok

Below is what you obtain: You can see the red crosses on the analysis connections

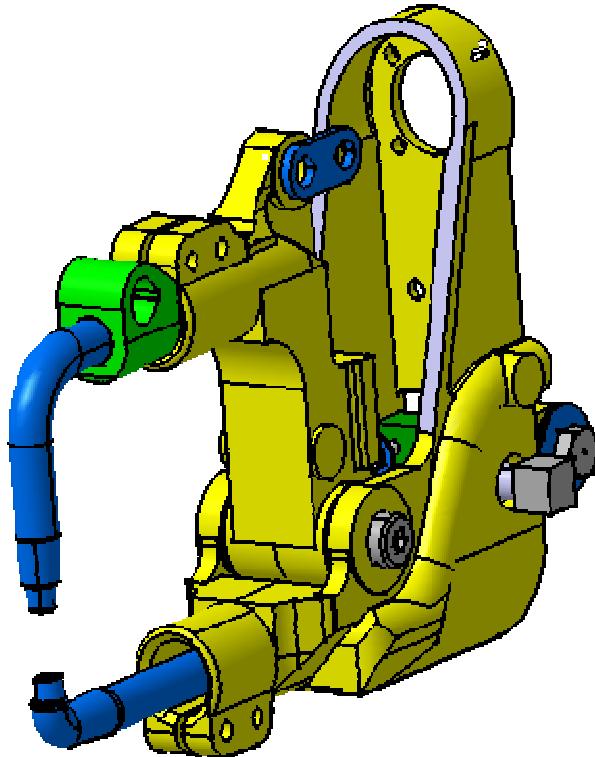
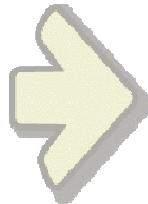
- 2 Select the support (spot welding analysis connection) to be assigned to spot welding connections and Type.



[Student Notes:](#)

Seam Welding Connection Property

You will learn how to define a Seam Welding Connection Property and when to use it



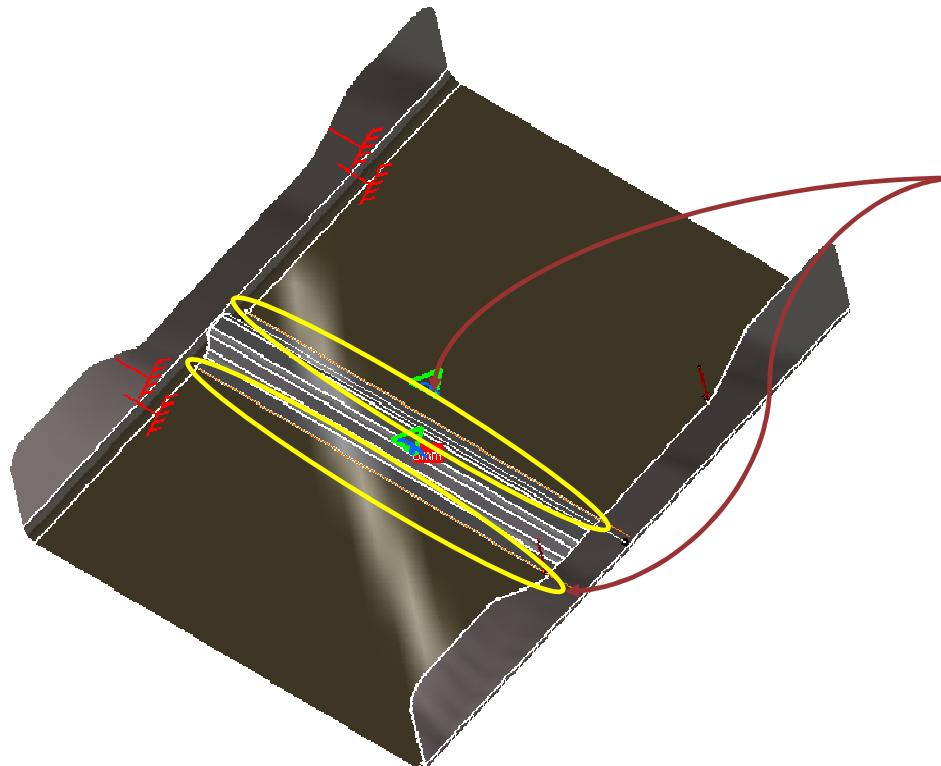
What is a Seam Welding Connection Property



[Student Notes:](#)

This type of connections allows to simulate Seam welding between two parts.

Seam Weld Connection Property need Seam Weld Analysis Connections as support, which need themselves some lines to be applied on



[Student Notes:](#)

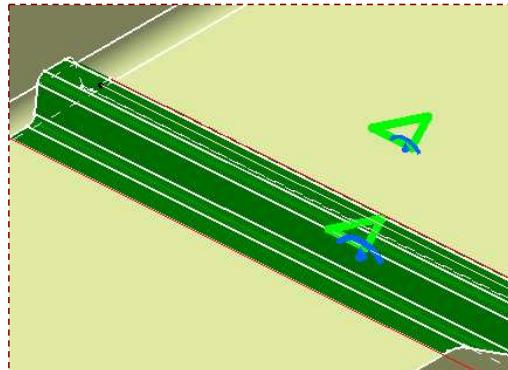
How to apply a Seam Welding Connection Property

Before you begin, make sure that all the needed assembly constraints were created

- ① Click on the Seam Welding Connection Property icon



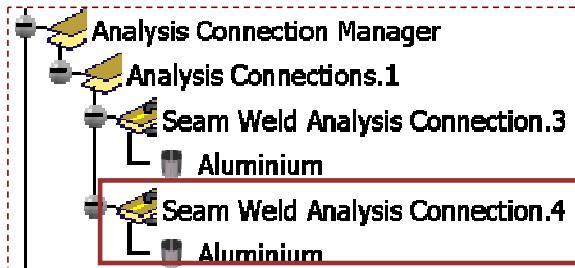
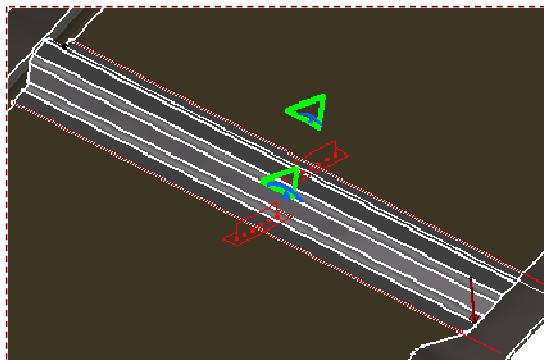
- ② Select a supports : Analysis Connection and Type



The red lines are
two Seam weld
Analysis connection

- ③ Click on Ok

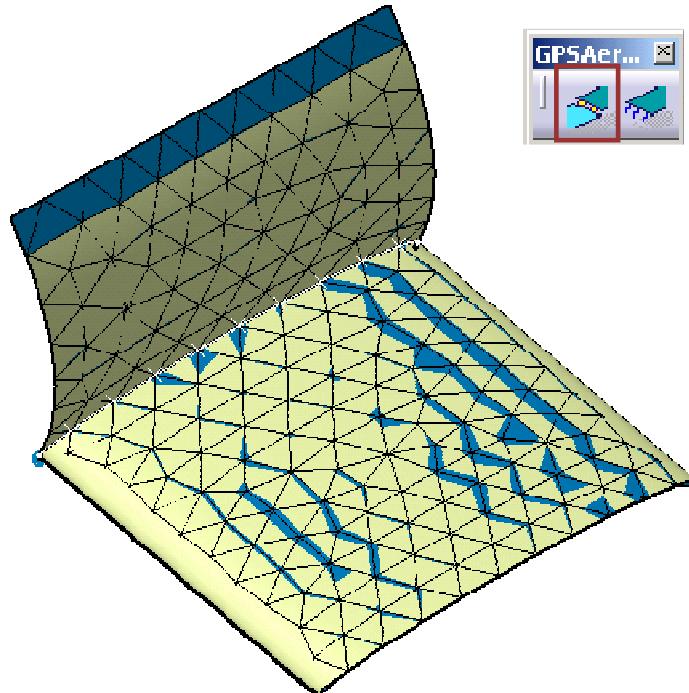
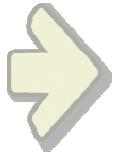
Below is the result:



[Student Notes:](#)

Nodes to Nodes Connection Property

The Nodes to Nodes Connection Property has been introduced to transmit DOFs between two mesh parts using the Points to Points Analysis Connection.



Student Notes:

What is Nodes to Nodes Connection Property

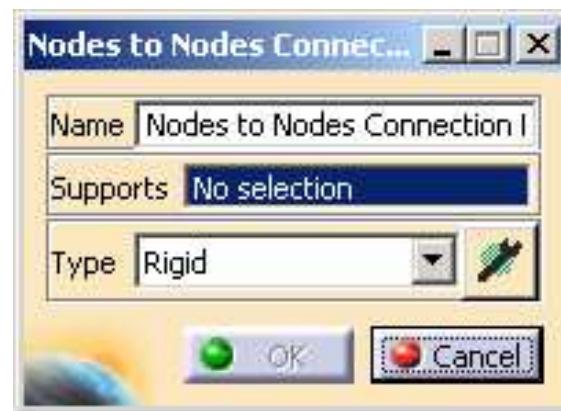
The 'Nodes to Nodes Connection Property' is applied on point to point connection. This helps to transmit the Degrees Of Freedom (DOFs) between mesh parts in Analysis Assembly.

Two types of physical properties are available.

- Rigid
- Coincidence

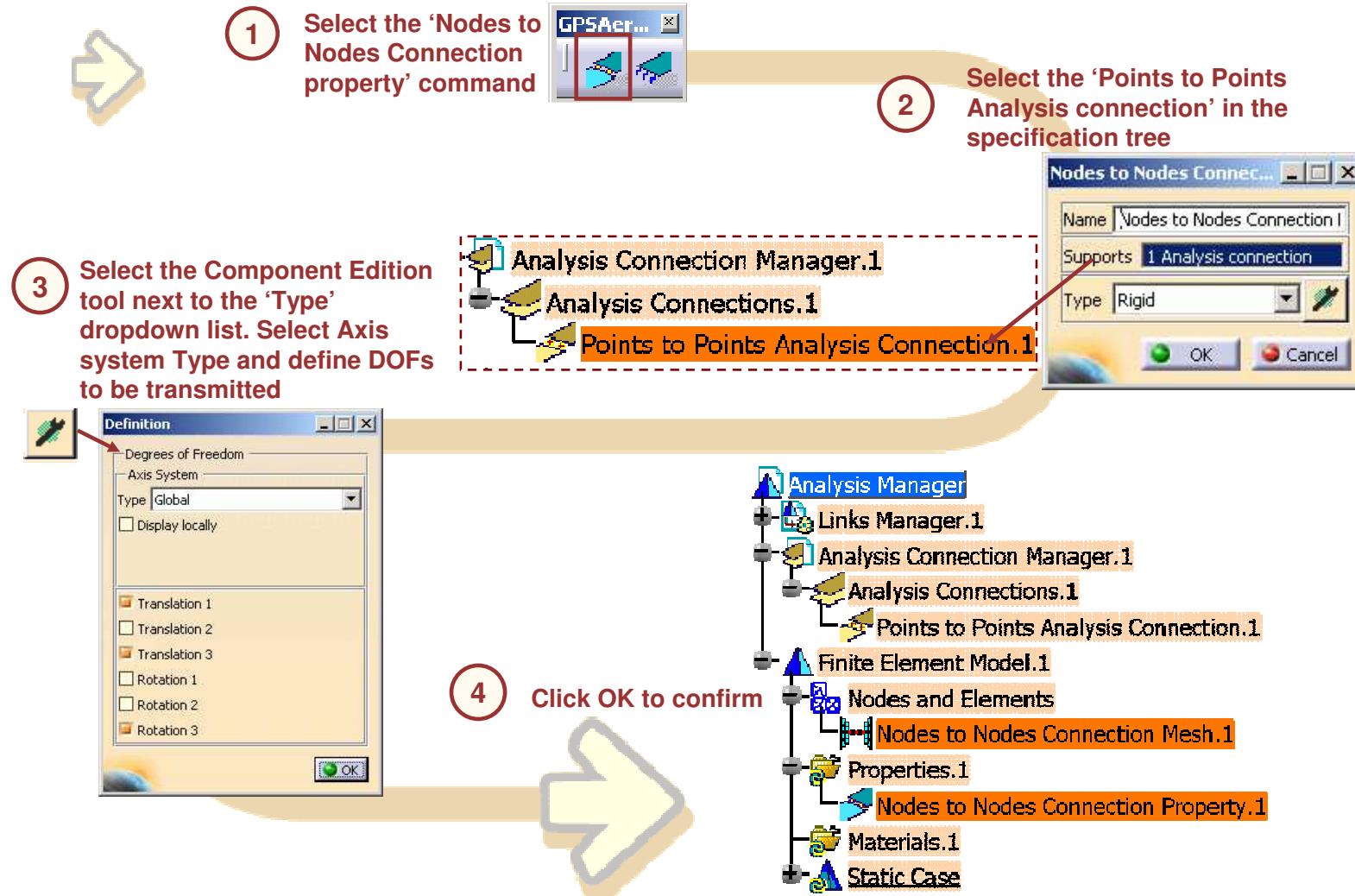
Both properties have the option to release DOFs.

Once the property is applied it will create Points to Points Connection Mesh.



[Student Notes:](#)

How to Apply the Nodes to Nodes Connection Property



To Sum Up ...

Student Notes:

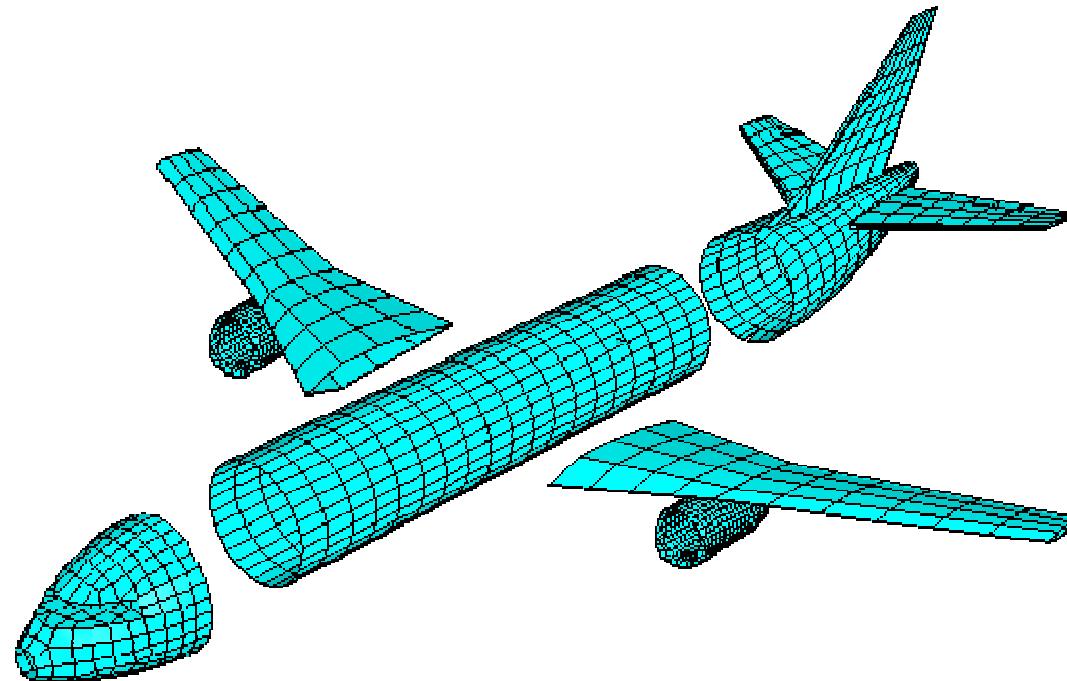
You have seen CATIA V5 GAS connections:

- How to define Face Face Connection Properties
- How to define Distant Connection Properties
- How to define Welding Connection Properties
- How to define Nodes to Nodes Connection Properties

Student Notes:

Analysis Assembly Management

You will learn how to use the technique of Analysis Assembly to reuse analysis created on parts.



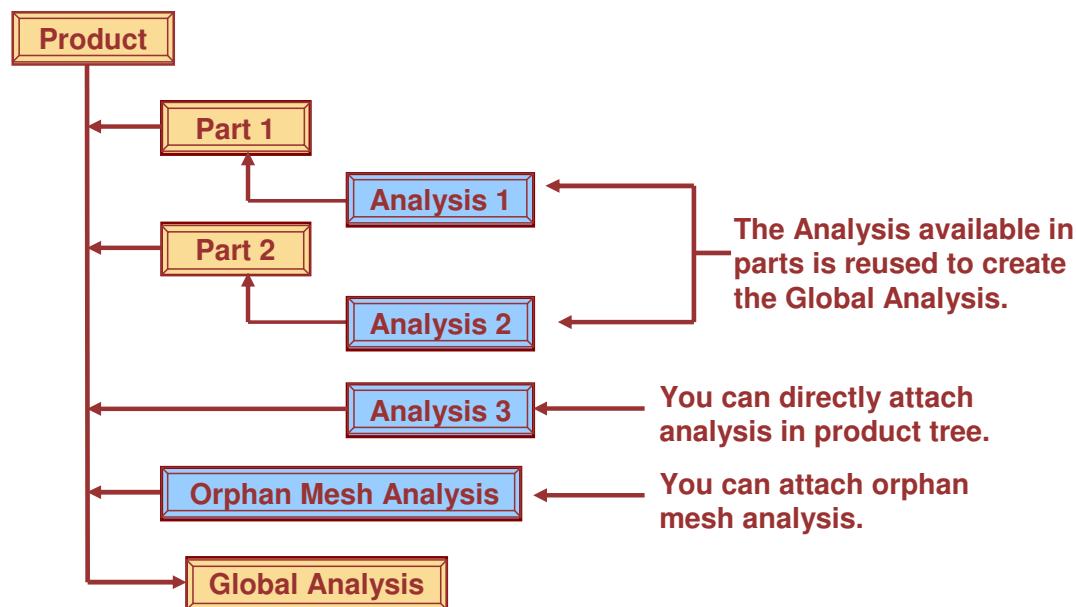
[Student Notes:](#)

What is Analysis Assembly

In this approach, analyses of individual parts in 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 one with all the necessary parts. Individual part analysis files are then attached to the corresponding parts in both the cases.

These analysis files must be computed with at least 'Mesh only' option ,so that it contains Mesh and complete FE property information .

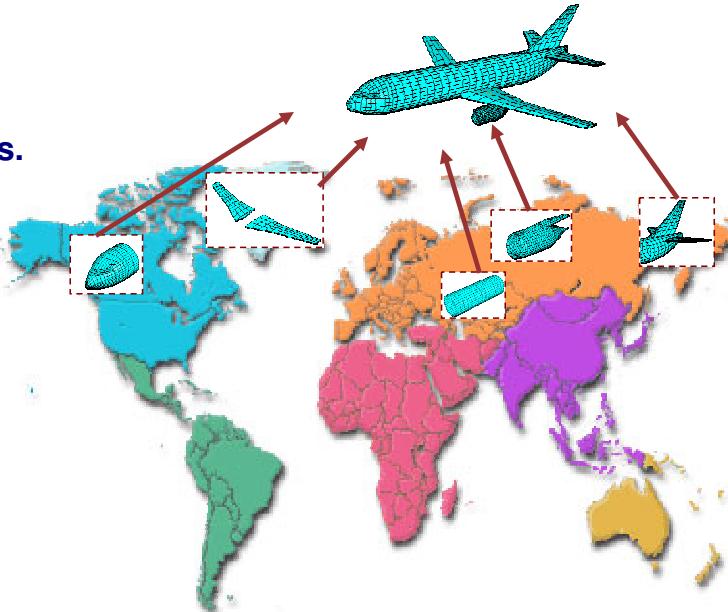


[Student Notes:](#)

Why Use Analysis Assembly

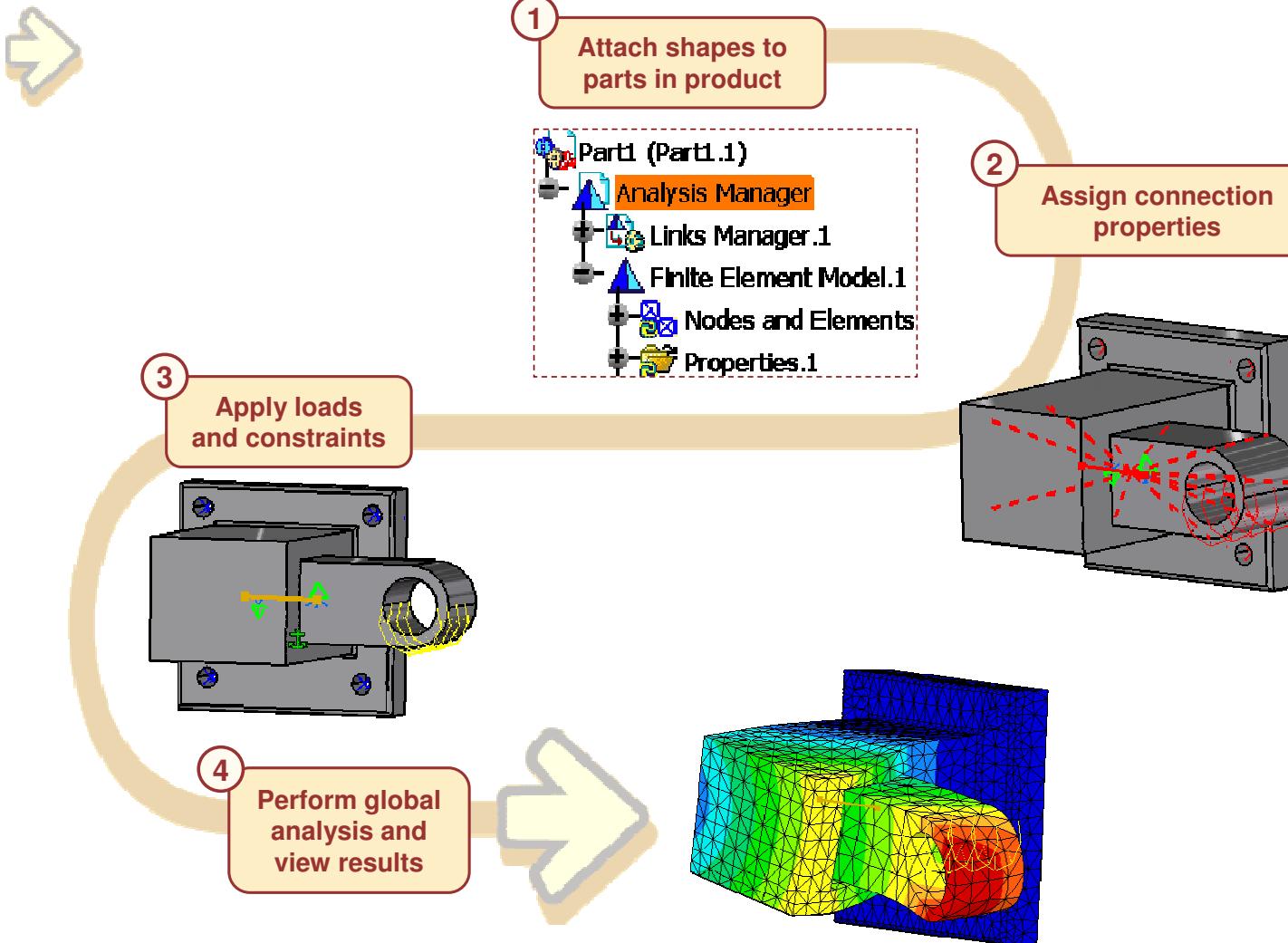
Earlier to the availability of this function, analyst will have to mesh a part twice first in individual part context and then in assembly context. The approach of Analysis Assembly has following advantages:

- It utilizes already meshed individual parts and imported orphan mesh parts effectively.
- When a single part is used in multiple assemblies, you have to mesh part only once.
- It enables concurrent engineering of FE Analysis. It is possible to mesh individual parts in assembly simultaneously by different users at different locations.
- It reduces time required to analyze large assemblies.
- It facilitates management of analysis data.



Analysis Assembly Process

[Student Notes:](#)

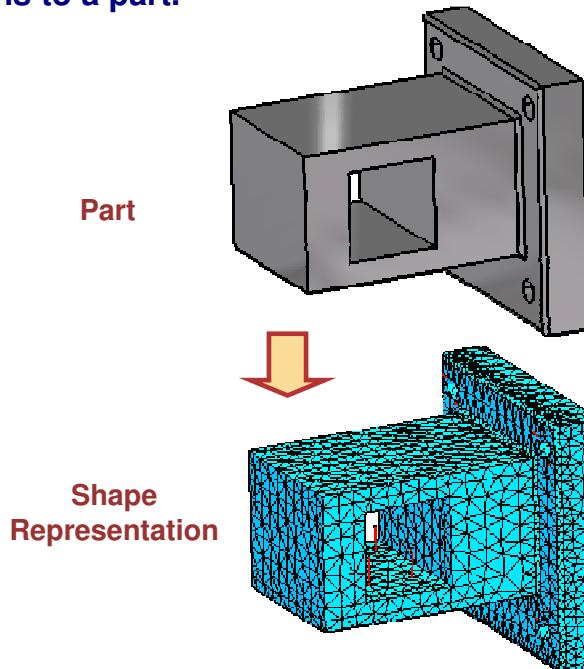
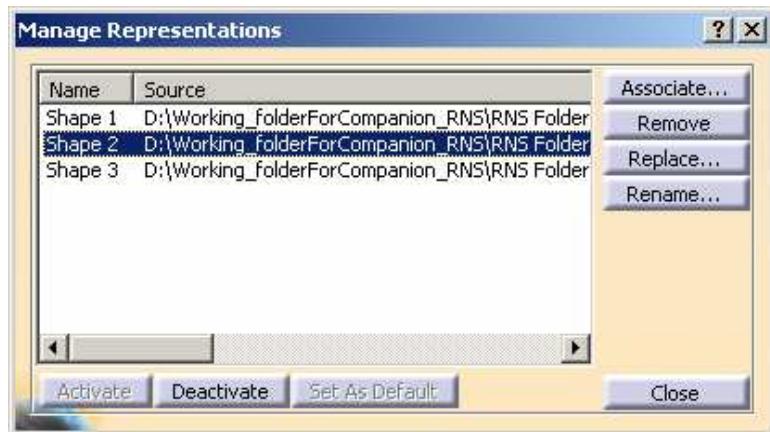


Student Notes:

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.



[Student Notes:](#)

How to Attach Shape to Part (1/2)

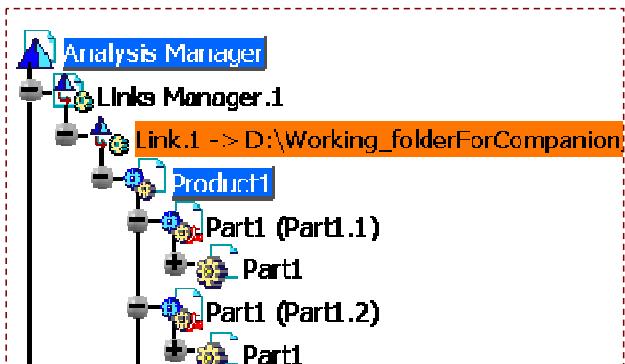
- 1 Open product document. Go to 'Analysis & simulation' workbench



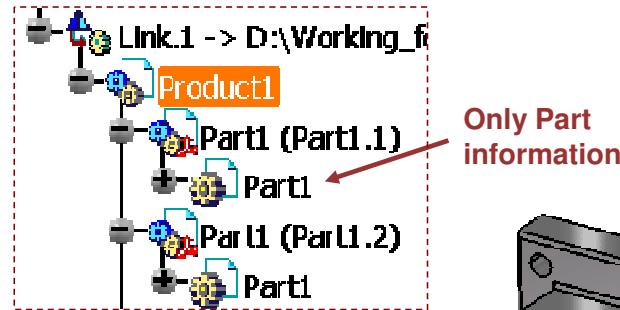
- 2 Double Click on Links Manager



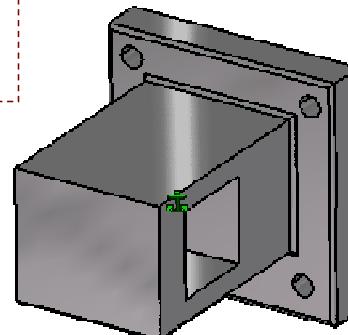
- 3 Double Click on Link to get Product tree



- 4 Double Click on Product



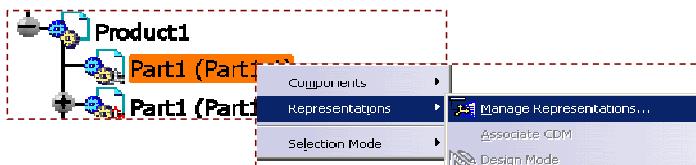
Part Model



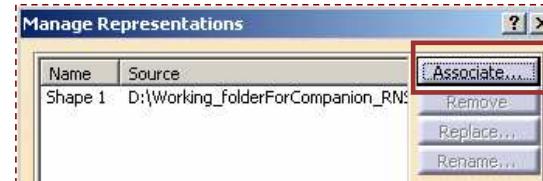
Student Notes:

How to Attach Shape to Part (2/2)

- 5** Click on Product contextual Menu Representations ->Manage Representations



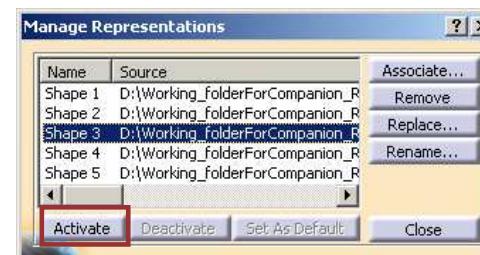
- 6** Click on Associate Button



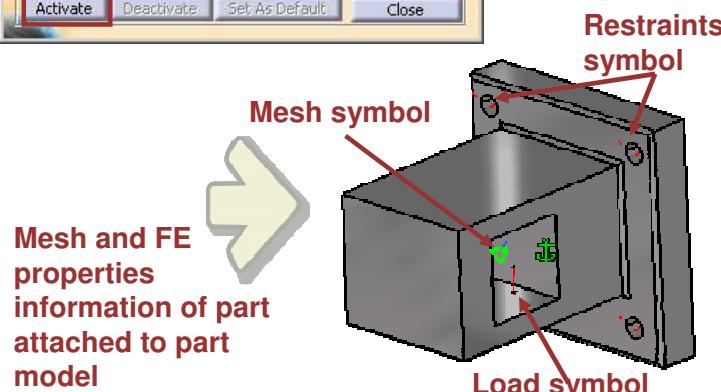
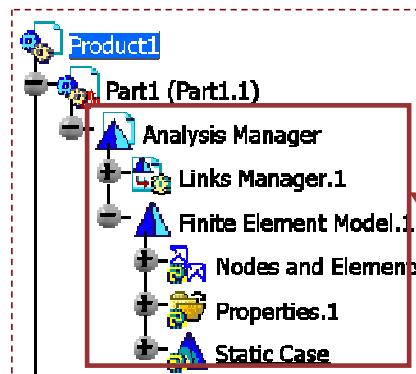
- 7** Select desired CATAnalysis File(shape) related to that part



- 8** Click on the same file in Manage Representation window and press activate button



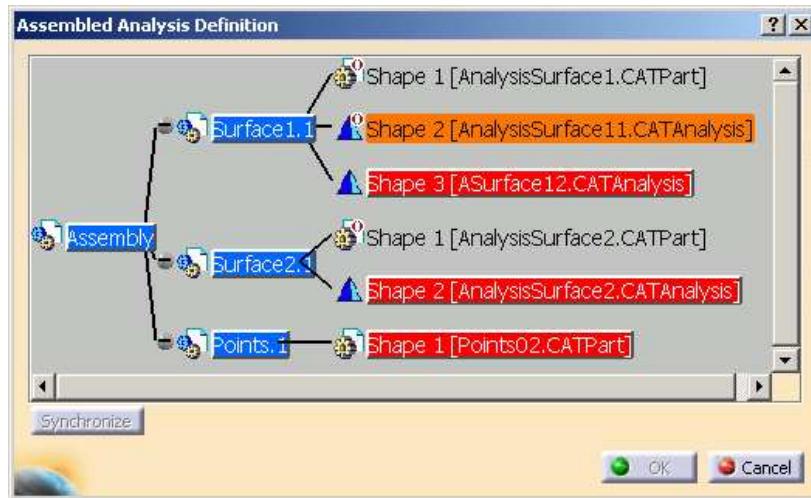
- 9** Click Close button



[Student Notes:](#)

What is Analysis Assembly 2D Viewer

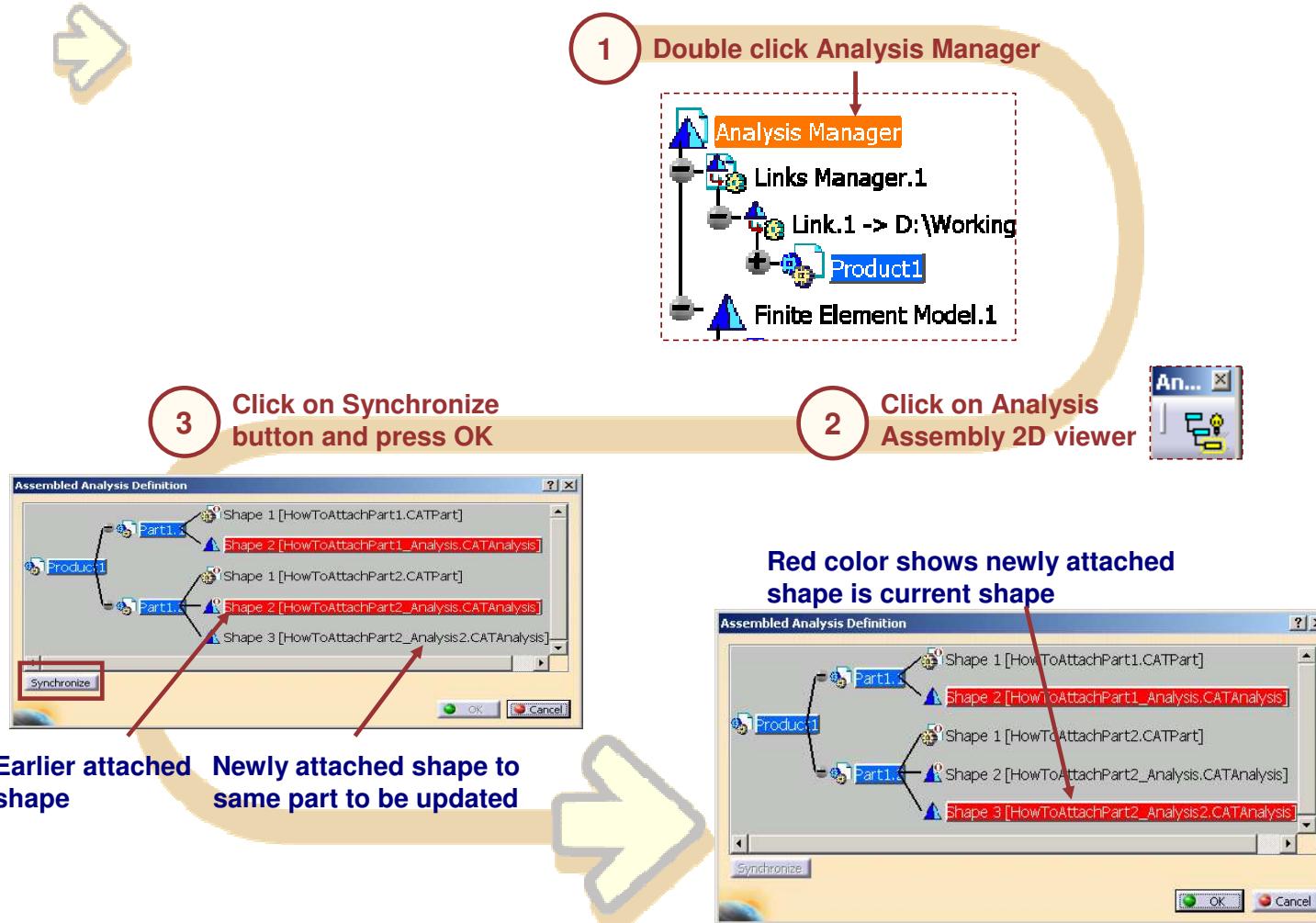
Analysis Assembly 2D Viewer enables you to add or remove a shape, activate or deactivate an existing shape, and add or remove a product component in Analysis Assembly. These changes in Analysis Assembly document is updated using Analysis Assembly 2D viewer.



[Student Notes:](#)

How to Use Analysis Assembly 2D Viewer

Analysis Assembly 2D viewer is used when shapes are activated, deactivated or part is added, removed from analysis Assembly tree.



To Sum Up ...

[Student Notes:](#)

You have seen Analysis Assembly Management:

- **What is Analysis Assembly**
- **Why use Analysis Assembly**
- **Analysis Assembly Process**
- **What is Shape Representation**
- **What is Analysis Assembly 2D Viewer**

To Sum Up ...

[Student Notes:](#)

You have seen CATIA V5 Generative Assembly Structural Analysis:

- **Introduction to GAS**
- **How to define Analysis Connections**
- **How to define Face Face Connection Properties**
- **How to define Distant Connections Properties**
- **How to define Welding Connections Properties**
- **How to define Nodes to Nodes Connection Property**
- **Analysis Assembly Management**