










Generative Assembly Structural Analysis

-  [Site Map](#)
-  [Preface](#)
-  [What's New?](#)
-  [Getting Started](#)
-  [Basic Tasks](#)
-  [Workbench Description](#)
-  [Index](#)



P1



P2



© Dassault Systèmes 1994-99. All rights reserved.

Site Map

[Preface](#)

[Using This Guide](#)

[More Information](#)

[What's new](#)

[Getting Started](#)

[Entering Analysis Connections Workbench](#)

[Creating Analysis Face Face Connection](#)

[Creating Analysis Welding Point Connection](#)

[Creating Property Connections](#)

[Computing the Static Case](#)

[Basic Tasks](#)

[Before You Begin](#)

[Analysis Connections](#)

[Creating Face Face Connections](#)

[Creating Distant Connections](#)

[Creating Welding Point Connections](#)

[Property Connections](#)

[Creating Fastened Connections](#)

[Creating Slider Connections](#)

[Creating Contact Connections](#)

[Creating Bolt Tightening Connections](#)

[Creating Virtual Rigid Bolt Tightening Connections](#)

[Creating Virtual Spring Bolt Tightening Connections](#)

[Creating Pressure Fitting Connections](#)

[Creating Rigid Connections](#)

[Creating Smooth Connections](#)

[Creating Spot Welding Connections](#)

[Workbench Description](#)

[Property Connections Toolbar](#)

[Analysis Connection Toolbar](#)

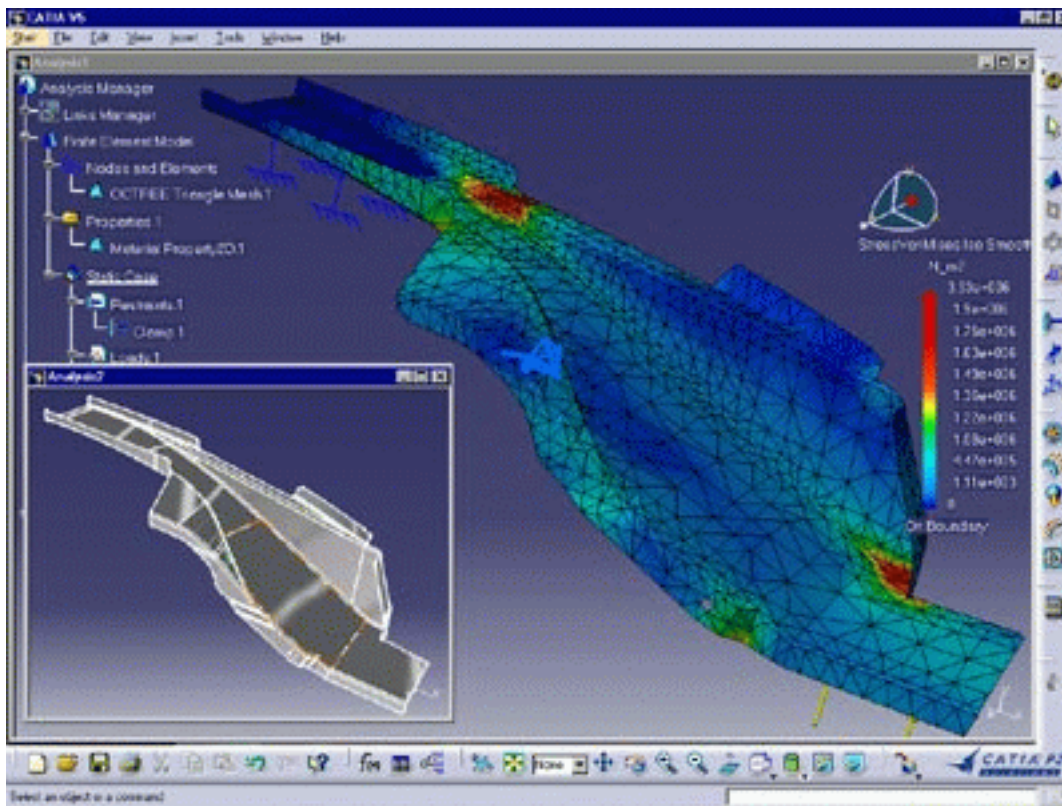
[Index](#)

Preface

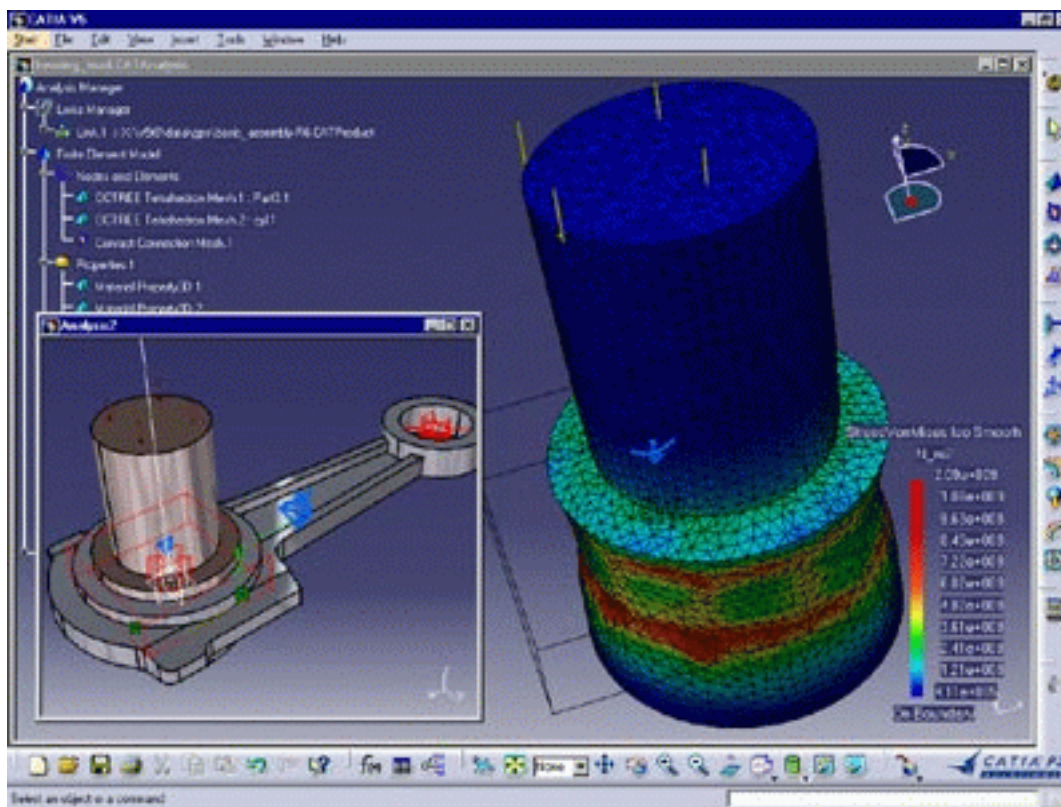
ELFINI Structural Analysis, Generative Part Structural Analysis and **Generative Assembly Structural Analysis Version 5** products allow you to rapidly perform first order mechanical analysis for 3D parts systems.

As a scalable line of products, **ELFINI Structural Analysis Version 5** along with **Generative Part Structural Analysis** and **Generative Assembly Structural Analysis** can be cooperatively used with other current or future companion products such as **Part Design, Assembly Design** and **Generative Drafting**. The widest application portfolio in the industry is also accessible through interoperability with *CATIA Solutions Version 4* to enable support of the full product development process from initial concept to product in operation.

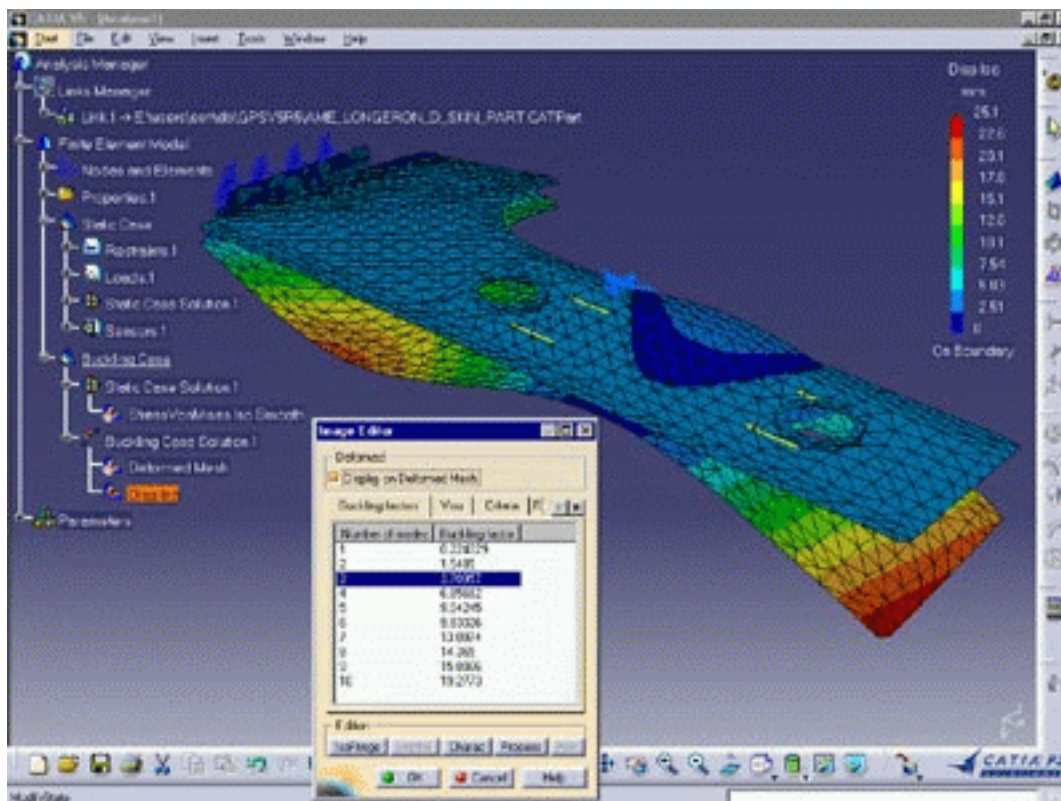
- The **Generative Part Structural Analysis** application is intended for the casual user. Indeed, its intuitive interface offers the possibility to obtain mechanical behavior information with very few interactions. The dialog boxes are self explanatory and require practically no methodology, all defining steps being commutative.



- The **Generative Assembly Structural Analysis** product has been designed as an integrated extension to **Generative Part Structural Analysis** enabling the study of the mechanical behavior of a whole assembly. The product has been conceived with the same "easy to learn" and "fun to use" ergonomics principles.



- The **ELFINI Structural Analysis** product is a natural extensions of both above mentioned products, fully based on the V5 architecture. It represents the basis of all future mechanical analysis developments.



The **ELFINI Structural Analysis** User's Guide has been designed to show you how to analyze a system consisting of single parts or of assemblies of parts, operating within a specified environment. There are several ways for undergoing a part to external actions. There are also several types of useful output information. This book aims at illustrating these various possibilities.

[Using This Guide](#)
[More Information](#)

Using This Guide

This book is intended for the user who needs to quickly become familiar with *Analysis Connections* Version 5. The user should be familiar with basic concepts such as document windows, standard and view toolbars.

To get the most out of this guide, we suggest you start reading and performing the step-by-step tutorial "[Getting Started](#)". This tutorial will show you how to analyze a part from scratch.

The "[Basic Tasks](#)" section presents the main capabilities of the product. Each individual task is carefully defined and explained in detail.



Where to Find More Information

Prior to reading this book, we recommend that you read:

- Generative Structural Analysis (GPS and EST) user's guide
- Infrastructure User's guide Version 5
- Part Design User's Guide
- [Conventions](#) chapter



Conventions

Certain conventions are used in CATIA, ENOVIA & DELMIA documentation to help you recognize and understand important concepts and specifications. The following text conventions may be used:

- The titles of CATIA documents *appear in this manner* throughout the text.
- File -> New identifies the commands to be used.

The use of the mouse differs according to the type of action you need to perform.

Use this

mouse button, whenever you read



Select (menus, commands, geometry in graphics area, ...)

Click (icons, dialog box buttons, tabs, selection of a location in the document window, ...)

Double-click

Shift-click

Ctrl-click

Check (check boxes)

Drag

Drag and drop (icons onto objects, objects onto objects)



Drag

Move



Right-click (to select contextual menu)

Graphic conventions are denoted as follows:



indicates the estimated time to accomplish a task.



indicates a target of a task.



indicates the prerequisites.



indicates the scenario of a task.



indicates tips



indicates a warning.



indicates information.



indicates the end of a task.



indicates functionalities that are new or enhanced with this Release.

Enhancements can also be identified by a blue-colored background in the left-hand margin.

Getting Started

This tutorial will guide you step-by-step through your first **Generative Assembly Structural Analysis** session, allowing you to get acquainted with the product. You just need to follow the instructions as you progress.

In this tutorial, you will

1.



create *Analysis Connections* (**Analysis Connections** workbench),

2.



add *properties* to these connections (**Generative Structural Analysis**

workbench) and then compute the CATAnalysis document.

[Entering Analysis Connections Workbench](#)

[Creating Analysis Face Face Connection](#)

[Creating Analysis Welding Point Connection](#)

[Creating Property Connections](#)

[Computing the Static Case](#)



These tasks should take about 15 minutes to complete.

Entering Analysis Connections Workbench



The **Analysis Connections** workbench of the **Generative Assembly Structural Analysis** product lets you add constraints to existing geometry. You can also add these constraints using **Product** workbench (Design mode). If so, the geometry will be over-constrained.

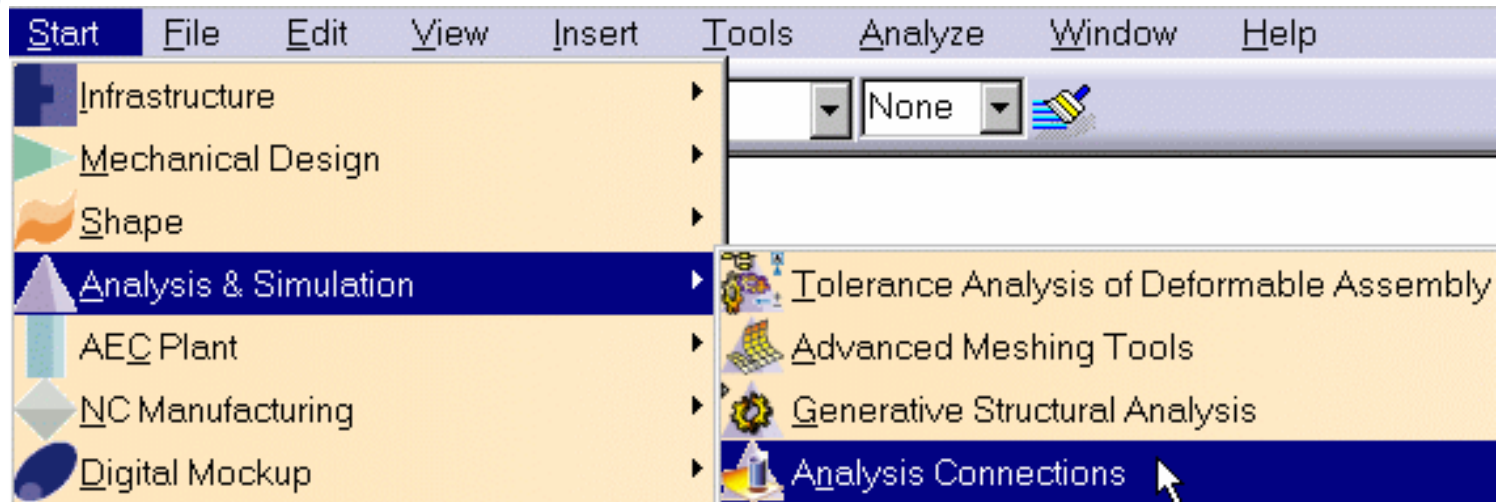


Open the [LandingGear.CATProduct](#) document from the sample directory.





1. Select Start-> Analysis & Simulation -> Analysis Connections from the menu bar.



Now that you entered the **Analysis Connections** workbench ...



... the following commands are available:



In other words, you can create Face Face, Distant or Welding point connections.

In this tutorial, you will create Connections (**Analysis Connections** workbench of the **Generative Assembly Structural Analysis** product), add property connections (**Generative Structural Analysis** workbench of the **Generative Part Structural Analysis** product) and then compute the CATAnalysis document.




Creating a Face Face Connection

Face Face Analysis Connections are **connections** used for connecting either faces that are **parallel to each others** or **concentric cylinders**, on an assembly model.



Remember that you entered the **Analysis Connections** workbench of the **Generative Assembly Structural Analysis** product.

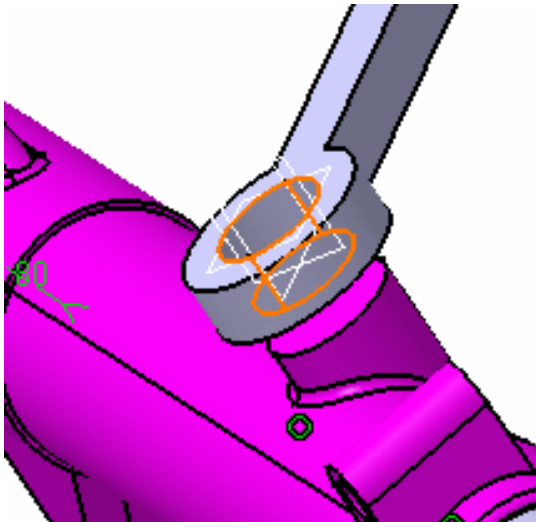
 This task shows you how to create a Face Face Connection.

 **1.** Select the Face Face Connection icon.

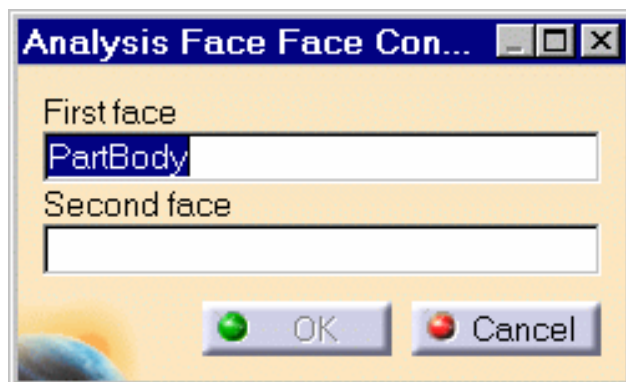
The Analysis Face Face Connection dialog box is displayed.

2. Select a first face on the assembly.

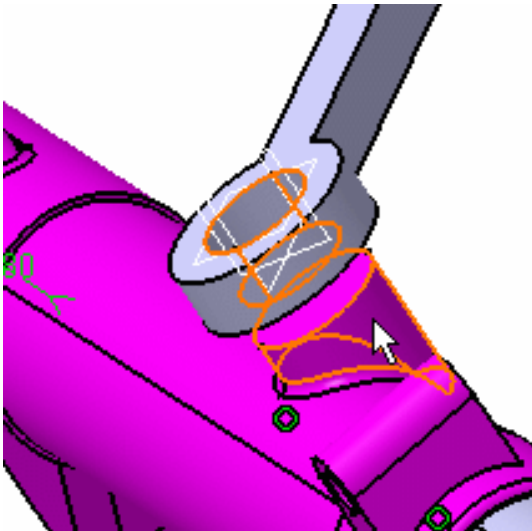
To manipulate the product more easily click with the middle mouse button on the product.



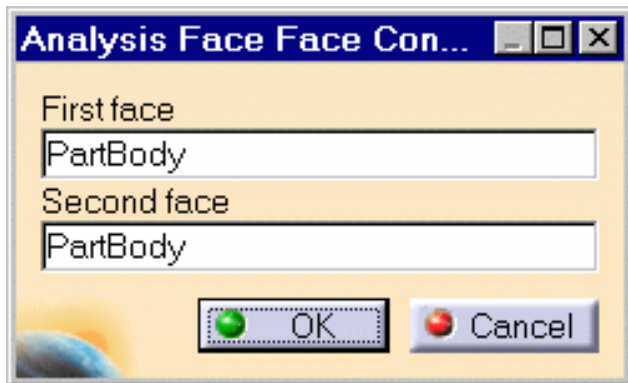
The Analysis Face Face Connection dialog box displays the selected face.



3. Select a second face on the assembly.

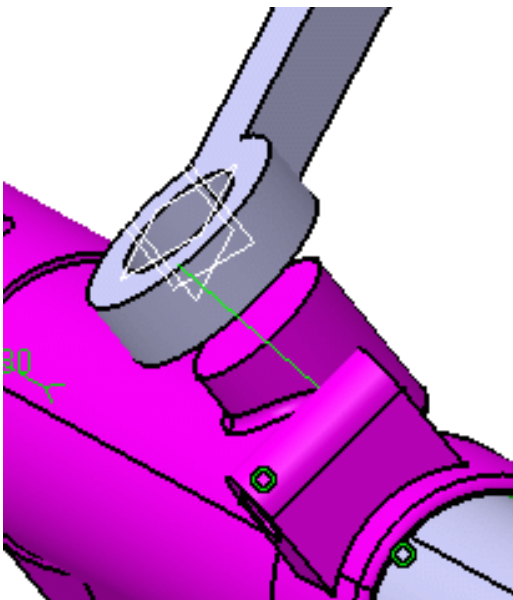


The Analysis Face Face Connections dialog box displays the selected faces.



4. Click OK in the Face Face dialog box.

A symbol appears on each connected face.



The Analysis Connection Manager object appears in the specification tree.





Creating a Welding Point Connection

Welding Point Analysis Connections are **connections** used for projecting welding points onto two faces, on an assembly model.



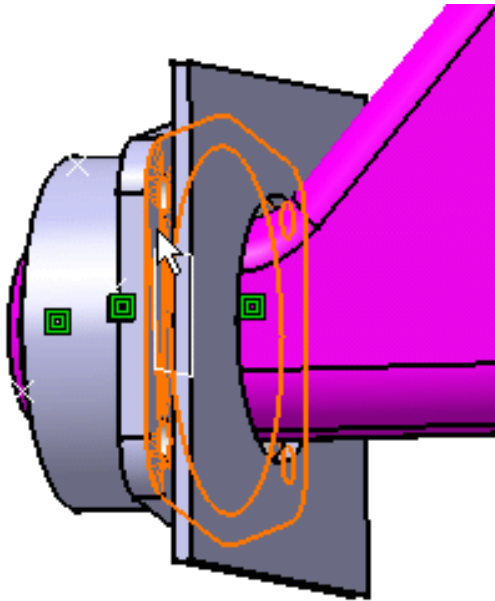
You are still in the **Analysis Connections** workbench of the **Generative Assembly Structural Analysis** product

 This task shows you how to create a Welding Point Connection.

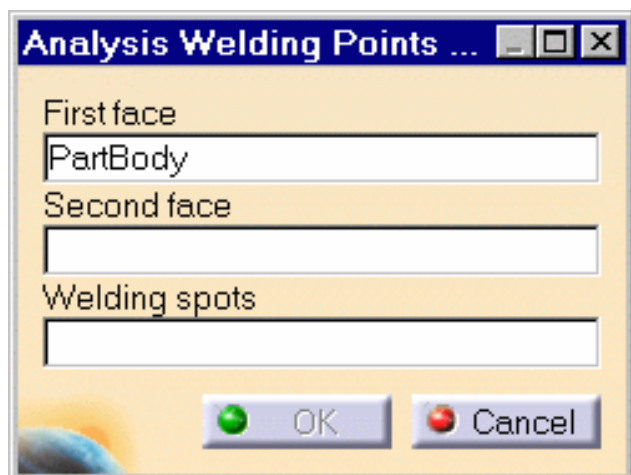
1. Select Start -> Analysis & Simulation -> Analysis Connections.
2. Select the Welding Point Connection  icon.

The Analysis Welding Points Connection dialog box is displayed.

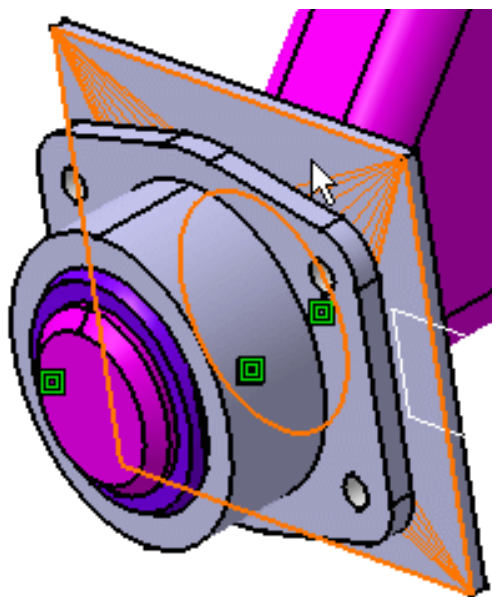
3. Select a first face on the assembly.



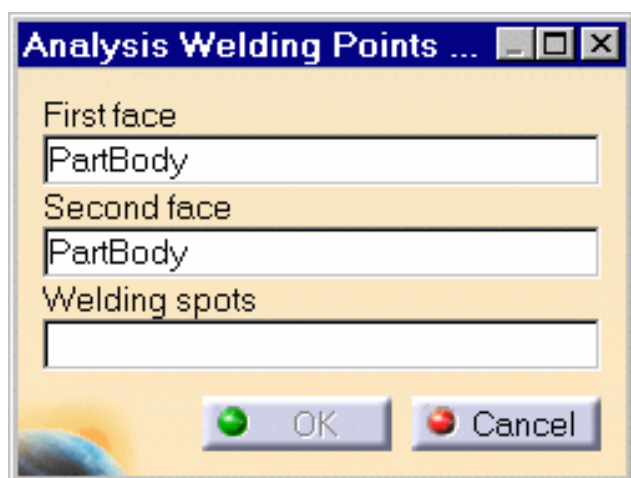
The Analysis Welding Points Connection dialog box appears as shown here:



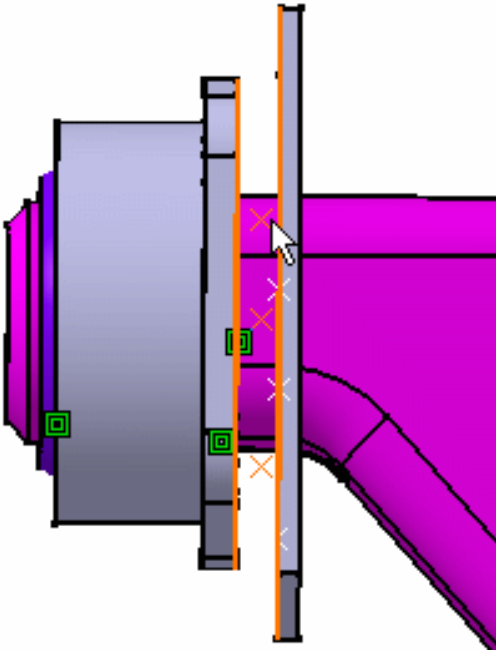
4. Select a second face on the assembly.



The Analysis Welding Points Connection dialog box appears as shown here:

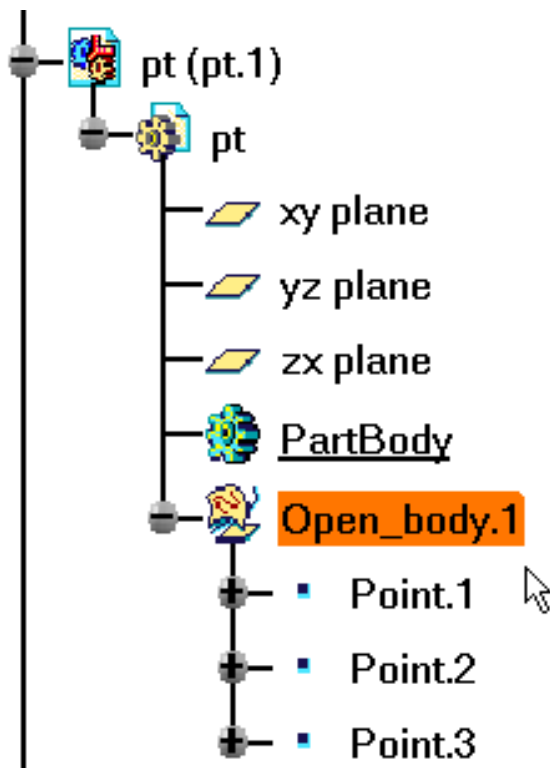


5. Multi-select a set of points in a single Open Body, on the product.

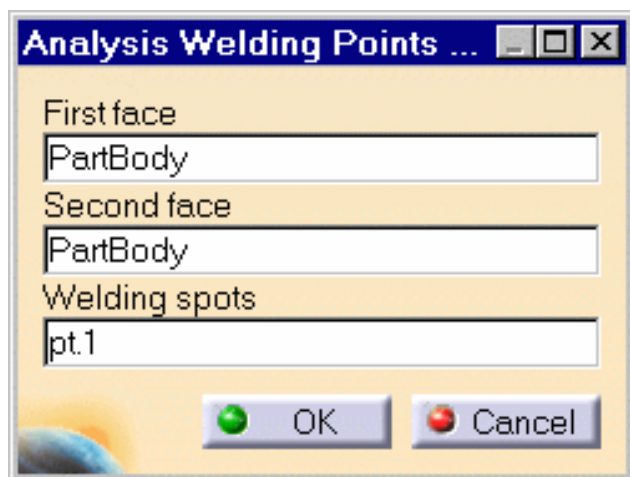


OR

5. Select a set of points in a single Open Body, on the specification tree. In this case select Open_body.1.

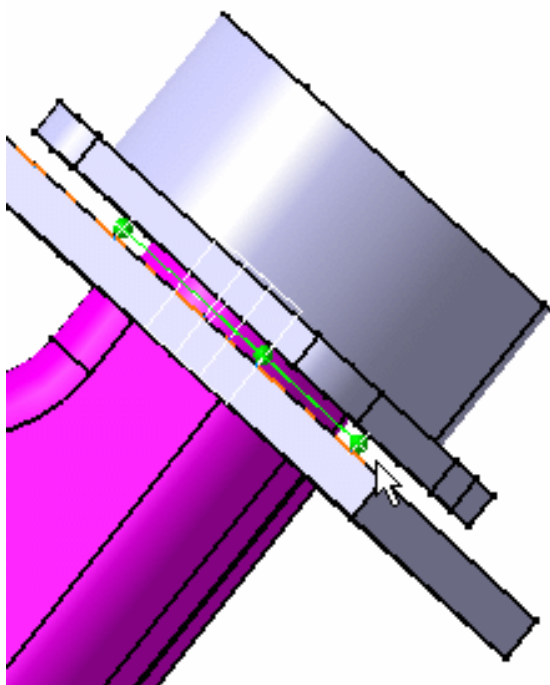


The Analysis Welding Points Connection dialog box appears as shown here:

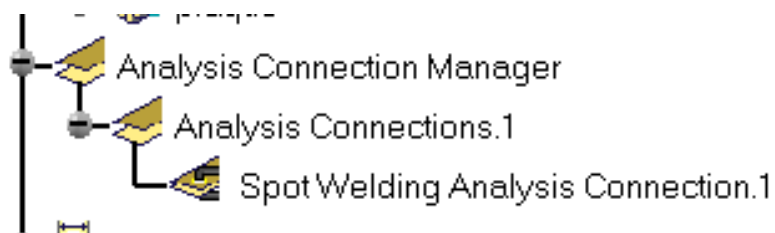


6. Click OK in the Analysis Welding Points Connection dialog box.

Symbols appear on the points connecting the faces.



Analysis Connection Manager appears in the specification tree.



Adding Property Connections

Property Connections are used to specify the boundary interaction between bodies in an assembled system. Once the geometric assembly positioning constraints are defined at the Product level, the user can specify the physical nature of the constraints.



You will now enter in the **Generative Structural Analysis** workbench of the **Generative Part Structural Analysis** product. For this, Select Start-> Analysis & Simulation -> Generative Structural Analysis from the menu bar and select the Static Analysis case and click OK in the New Analysis Case dialog box.



1. Select the desired Property Connection icon.

Connections



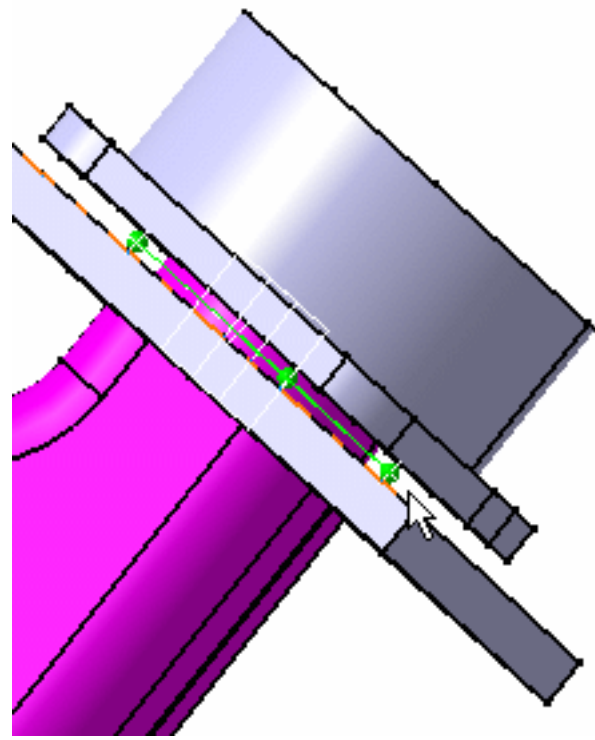
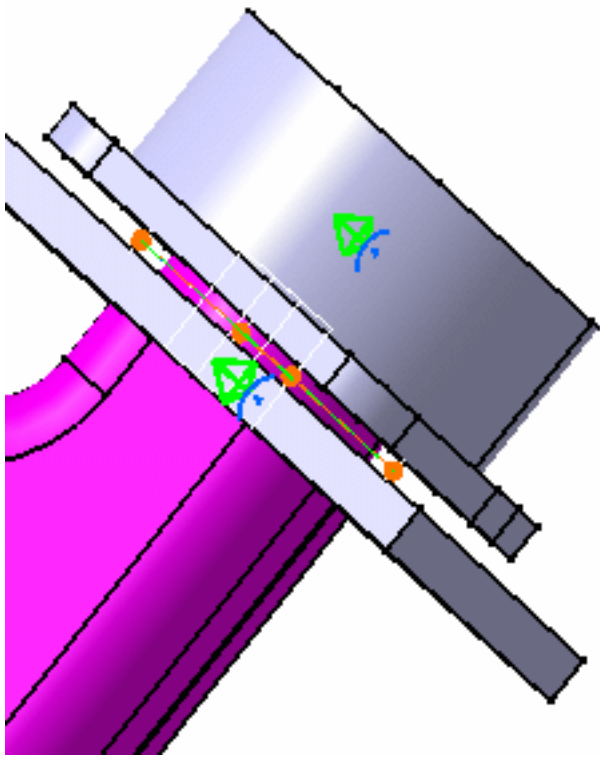
Advanced Connections



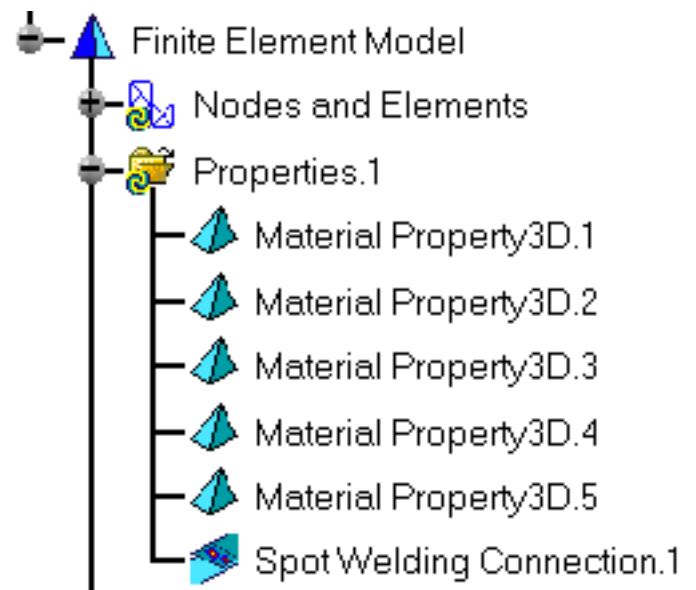
2. Select the previously created Analysis connections (either from the specification tree or on the geometry).




Selecting the **Spot Welding icon** and the **Welding Point connection:** You create this **Spot Welding Property connection:**



The Spot Welding Connection object appears in the specification tree.



 Repeat the same operation to create fastened Property connections on the Face-Face connection previously created.



Computing the Static Case

The Compute command is most often applied to Analysis Case Solutions (which are particular types of objects sets). In this case, it generates the analysis case solution, along with partial results for all objects involved in the definition of the Analysis Case.

The primary **Static Solution Computation** result consists of a **displacement vector** whose components represent the values of the system degree of freedom. This result can be further processed to produce other results such as stresses, reaction forces, etc.

The program can compute simultaneously several Solution objects sets, with optimal parallel computation whenever applicable.



You are still in the **Generative Structural Analysis** workbench of the **Generative Part Structural Analysis** product.

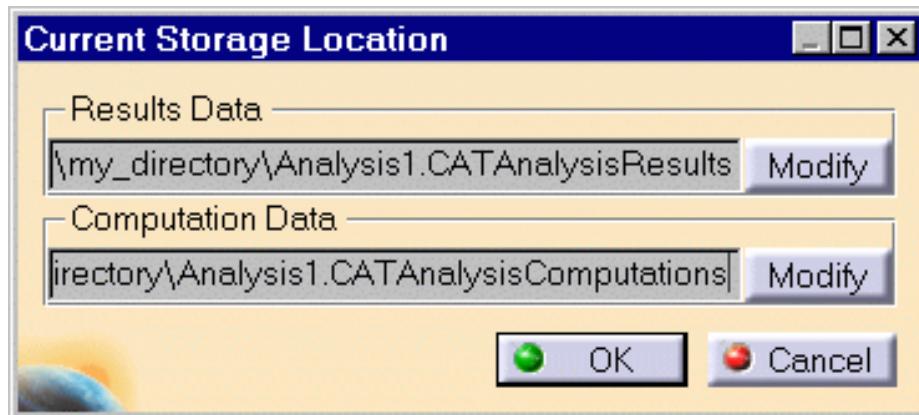
This task will show you how to compute the Static Case Solution of a Finite Element Model on which you previously created connections.



1. Click the Storage Location icon .


The Current Storage Location dialog box is displayed.

2. If needed, change the path of the Storage Result and/or Computation Data directory (Results Data and Computation Data fields) to another directory.



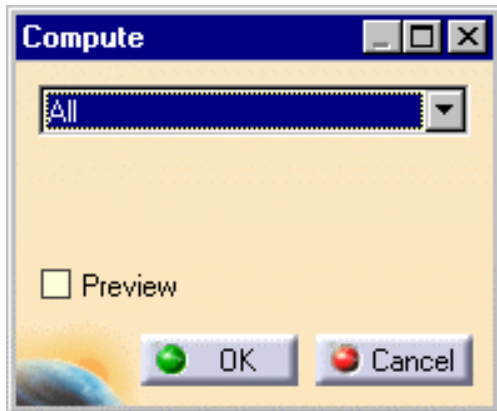
The Results and Computation Data are stored in one single file with given extensions:

- xxx.CATAnalysisResults
- xxx.CATAnalysisComputations

3. Click the Compute icon  .

The Compute dialog box appears.

The Elfini Storage Location dialog box disappears.



4. Select the All default value proposed for defining which are the objects sets to be updated.
5. Click OK in the Compute dialog box to launch the computation.

At the bottom right of the screen, a series of status messages (Meshing, Factorization, Solution) informs you of the degree of advancement of the computation process.



Basic Tasks

The basic tasks you will perform in the **Generative Assembly Structural Analysis** workbench are:

[Before You Begin](#)
[Analysis Connections](#)
[Property Connections](#)

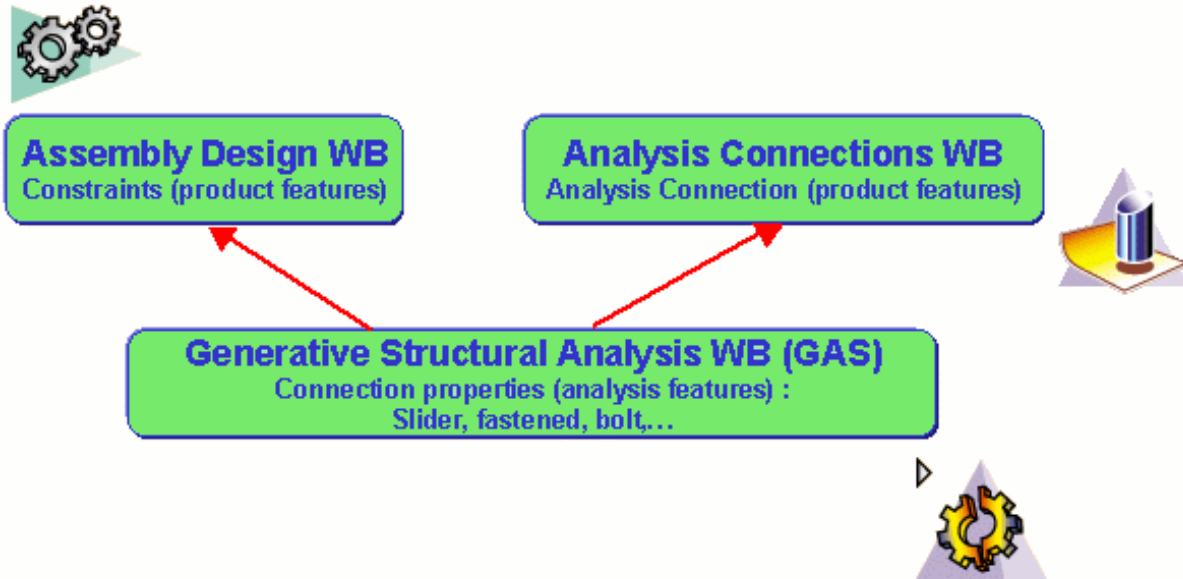
Before You Begin

Before you begin you should be familiar with the following basic concepts on:

- Analysis strategy
- Analysis Connections workbench



A Standalone Analysis Connection Workbench



When you create a connection, you generally first create an assembly constraint in the **Assembly Design** workbench. Then, in the **Generative Structural Analysis** workbench, you add a property on this constraint (fastened, pressure fitting, bolt and so forth). This property generates a Mesh part. The property is propagated to the above mentioned Mesh part.

Why you will use Analysis Connections workbench:

In some cases, **Assembly Design** constraints are not sufficient to modelize connections from an Analysis viewpoint:

1. In order to support properties, users often need to define too many constraints using the **Assembly Design** workbench, leading to over-constrained models that cannot be updated.
2. It is impossible to define constraints that are not positioning constraints but connection constraints (for example : connecting one face from one solid with one face from another solid, these faces being not necessarily coincident).
3. In the case of surface parts assemblies, it is not possible to define positioning constraints.
4. It is impossible to modelize welding spot connections.

In order to meet these different needs, a new workbench called **Analysis Connection** was added to the **Generative Assembly Design** product. **Analysis Connection** let you create all these Connections dedicated to Analysis modeling.



About Analysis Connections Workbench






















The **Analysis Connections** workbench allows you creating constraints that are not design constraints.

The **Analysis Connections** workbench let's you create face face, distance or welding constraints (called *connections*). As a user, you may need to add design face-to-face connections in order to create property constraints afterwards (**Generative Structural Analysis** workbench). As a result, a given number of these design constraints need to be de-activated so that the model is not over-constrained. You may also need to add design constraints at a given distance (this constraint goes through a virtual point) so that you can simulate a part that was not designed.










































What Type of Property For What Type of Connection ?




















1. For Connections Between Two Surfaces

Connection Property		 Assembly Design			 Analysis Connection		
		Coincidence	Contact	Offset	Face Face	Distant	Welding
Fastened							
Slider							
Contact							
Rigid							
Smooth							
Spot welding							

2. For Connections Between Two Parts

Connection Property		 Assembly Design			 Analysis Connection		
		Coincidence	Contact	Offset	Face Face	Distant	Welding
Fastened							
Slider							
Contact							
Rigid							
Smooth							
Bolt tightening							
Pressure fitting							
Virtual bolt tightening							
Virtual spring bolt tightening							
Spot welding							

3. For Connections Between a Part and a Surface

Connection Property		 Assembly Design			 Analysis Connection		
		Coincidence	Contact	Offset	Face Face	Distant	Welding
Fastened							
Slider							
Contact							
Rigid							
Smooth							
Spot welding							

Analysis Connections



Analysis Connections are assembly connections cannot be created from the **Generative Assembly Structural Analysis** workbench. They are some kind of constraints that are not design constraints. If needed, you will then be able to add [property connections](#) on this assembly.



[Creating Face Face Connections](#): connect either faces that are parallel to each others or concentric cylinders, on an assembly model.



[Creating Distant Connections](#): connect any part from an assembly with or without point type geometrical elements, on an assembly model.




[Creating Welding Points Connections](#): project welding points onto two parallel faces, on an assembly model.



Creating a Face Face Connection




Face Face Analysis connections are **connections** used for connecting either faces that are parallel to each others or concentric cylinders, on an assembly model.

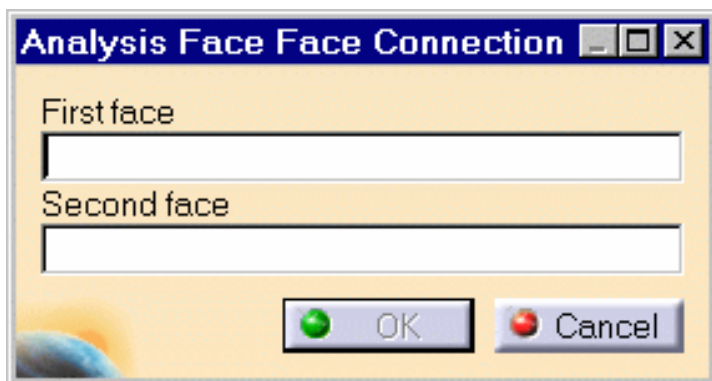
 This task shows you how to create a Face Face Connection.

 Open the [Sample18.CATAnalysis](#) document.

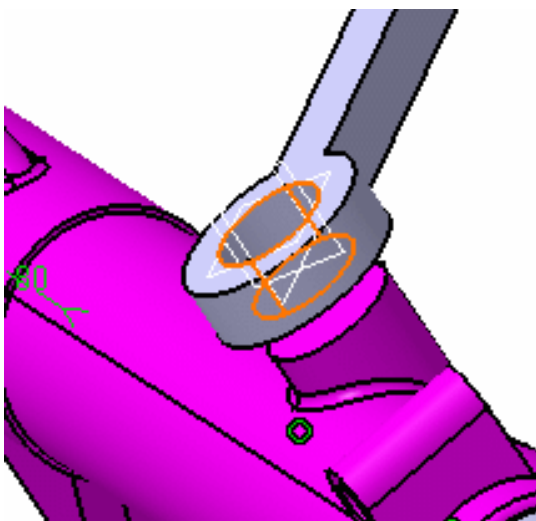
Make sure that you entered the **Analysis Connections** workbench of the **Generative Assembly Structural Analysis** product.

 1. Select the Face Face Connection icon.

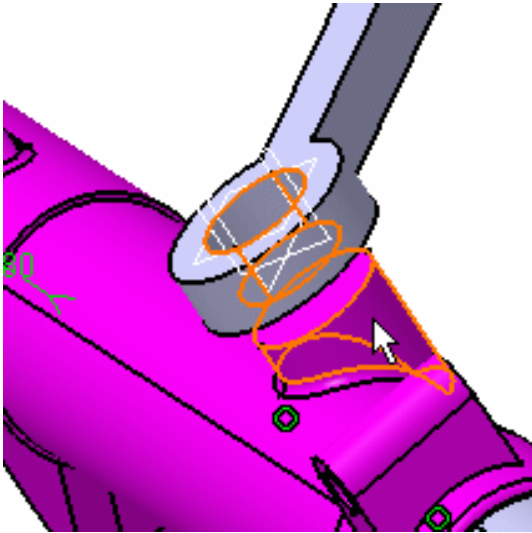
The Analysis Face Face Connections dialog box is displayed.



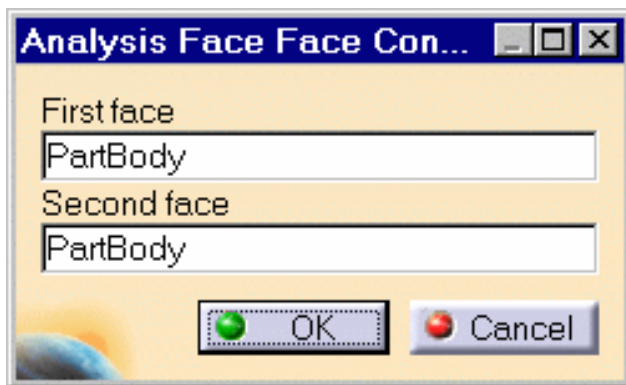
2. Select a first face on the assembly.



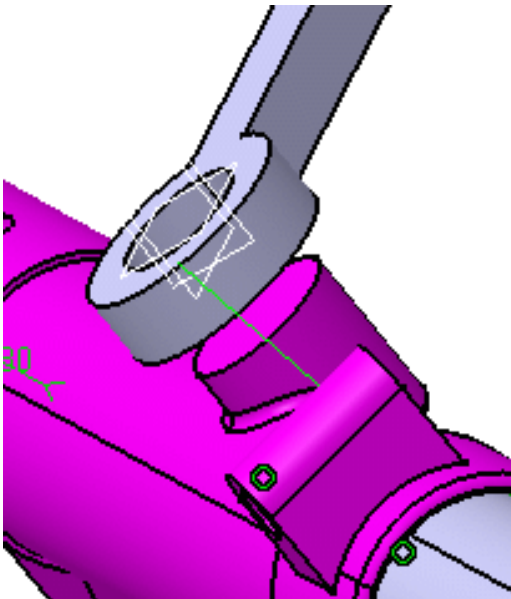
3. Select a second face on the assembly.



The Analysis Face Face Connection dialog box appears as shown here:



4. Click OK in the Analysis Face Face Connections dialog box.



- A symbol appears on each connected face.
- Analysis Connection Manager appears in the specification tree.

Creating a Distant Connection



Distant Analysis connections are **connections used for connecting any part from an assembly with or without point type geometrical elements, on an assembly model.**



This task shows you how to create a Distant Connection. This is very useful when you want to benefit from the representation of a part without actually designing this part.



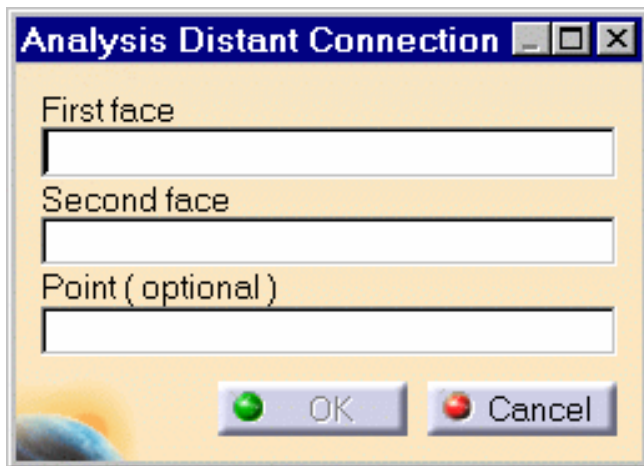
Open the [Sample02.CATProduct](#) document.

Make sure that you entered the **Analysis Connections** workbench of the **Generative Assembly Structural Analysis** product.

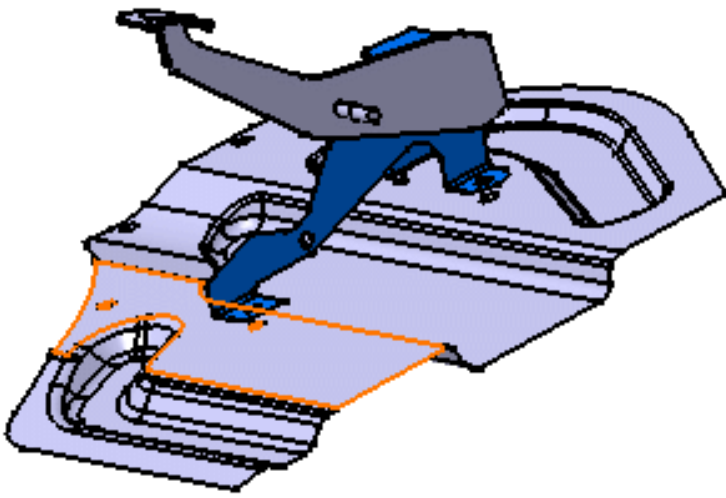


1. Select the Distant Connection  icon.

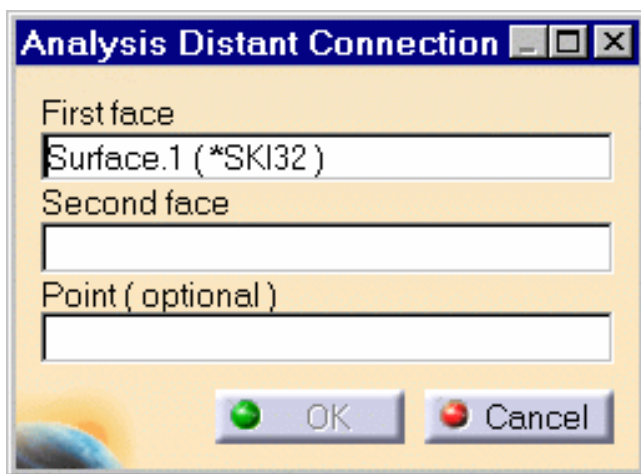
The Analysis Distant Connection dialog box is displayed.



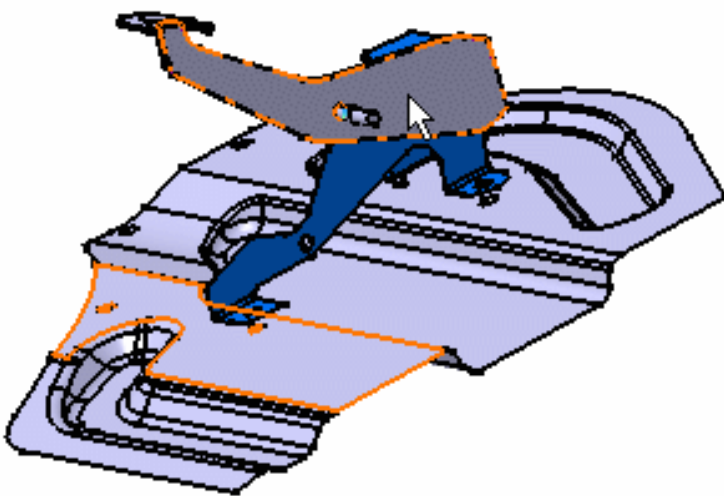
2. Select a first face on the assembly.

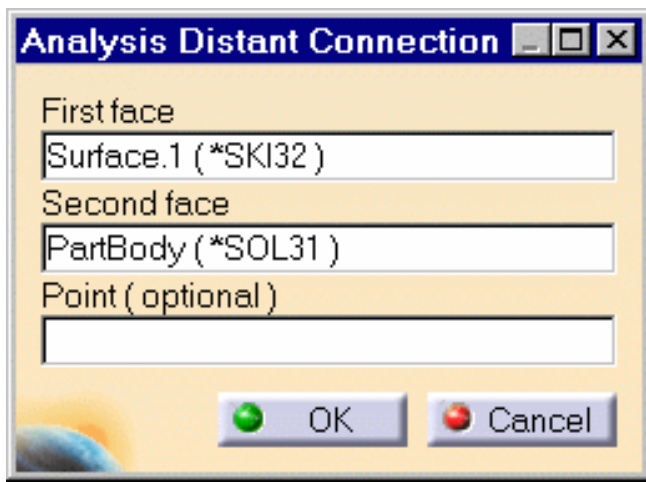


The Analysis Distant Connection dialog box appears as shown here.



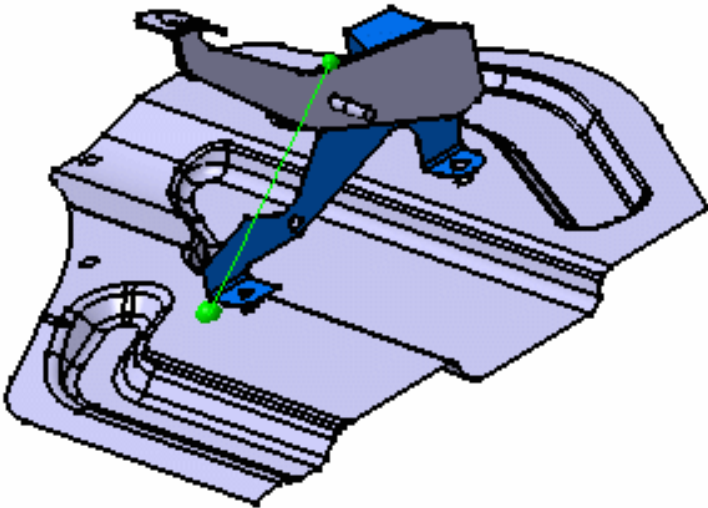
3. Select a second face on the assembly.





4. Click OK in the Analysis Distant Connection dialog box.

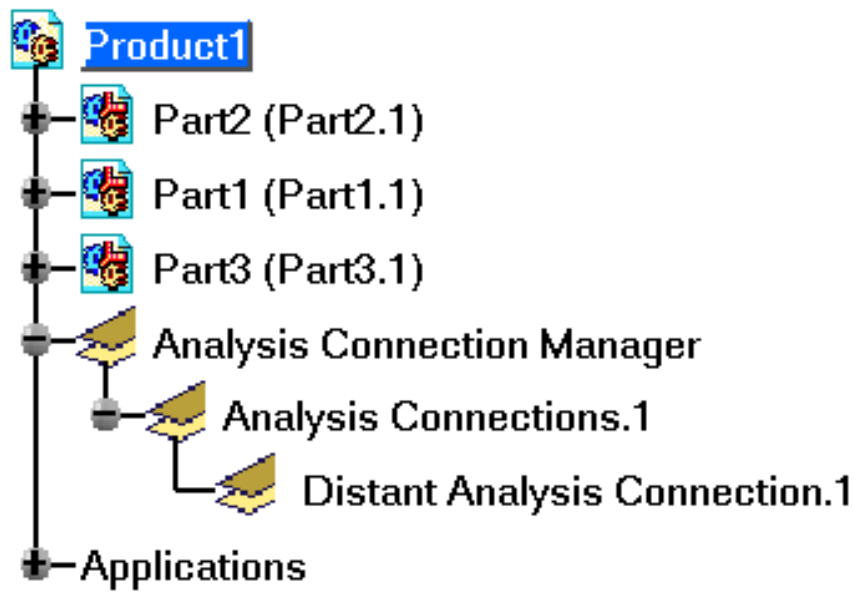
A virtual point is automatically created and the distance connection will go through this virtual point.



OR

4. If needed, select a point and then click OK in the Analysis Distant Connection dialog box.

The Distant Analysis Connection.1 object now appears in the specification tree.




- A symbol appears on each connected face.
- Analysis Connection Manager appears in the specification tree.




Creating a Welding Point Connection




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

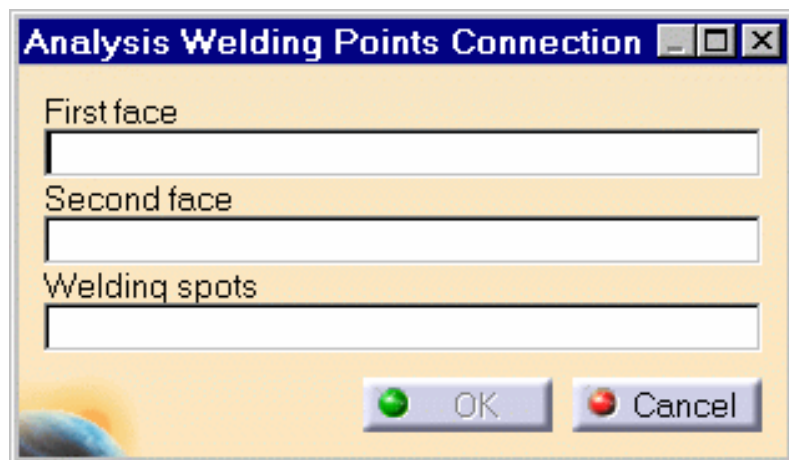
 This task shows you how to create a Welding Point Connection.

 Open the [Sample03.CATProduct](#) document: you created a set of 3D points on the assembly (Open Body).

Make sure that you entered the **Analysis Connections** workbench of the **Generative Assembly Structural Analysis** product.

 **1.** Select the Welding Point Connection  icon.

The Analysis Welding Points Connection dialog box is displayed.

The dialog box has a title bar with the text "Analysis Welding Points Connection" and standard window controls. It contains three text input fields labeled "First face", "Second face", and "Welding spots". At the bottom right, there are two buttons: "OK" with a green circular icon and "Cancel" with a red circular icon.

Analysis Welding Points Connection

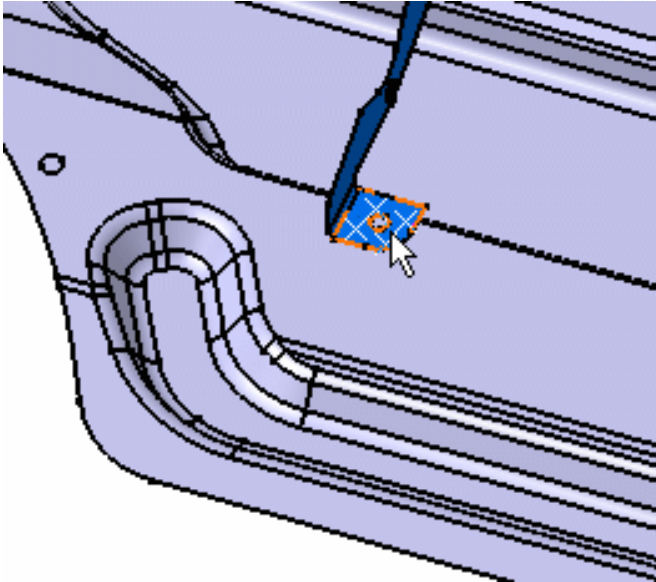
First face

Second face

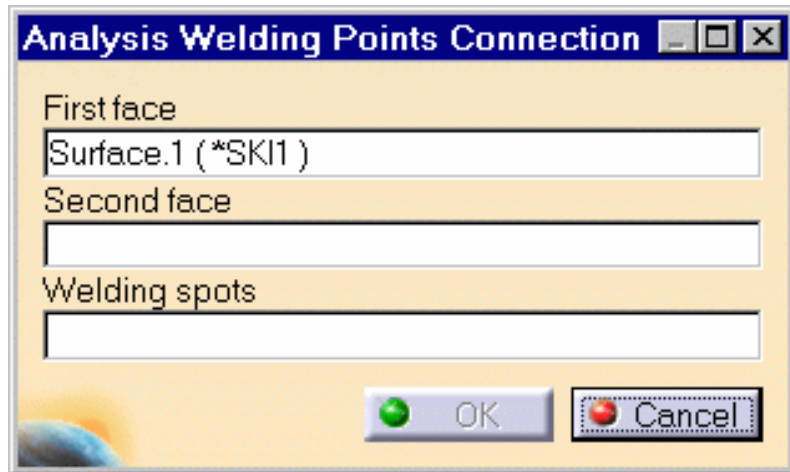
Welding spots

OK Cancel

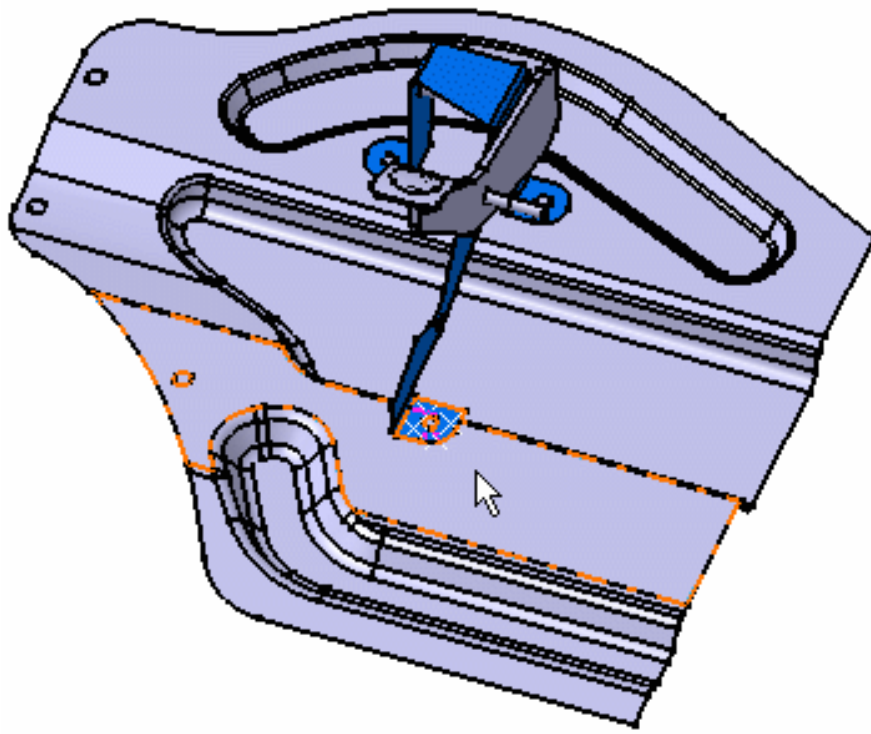
2. Select a first face on the assembly.



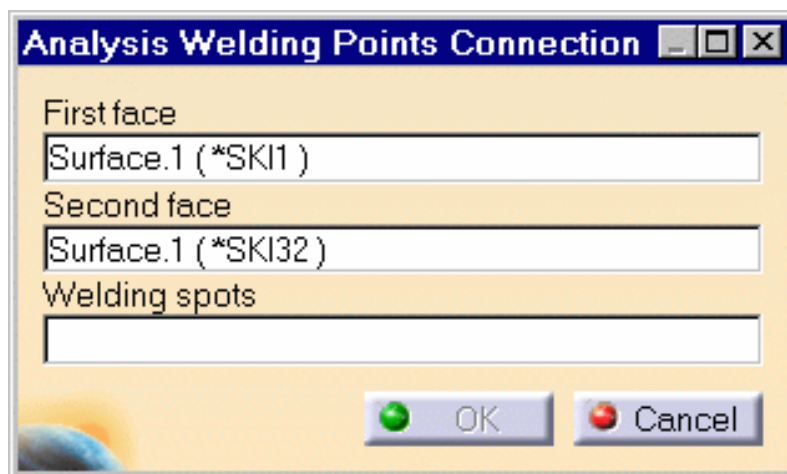
The Analysis Welding Points Connection dialog box is updated:



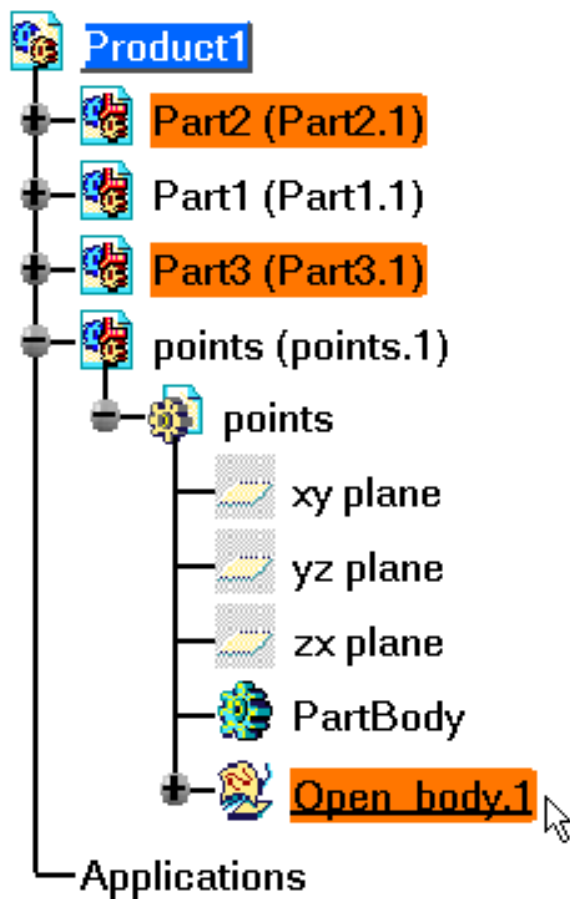
3. Select a second face on the assembly.



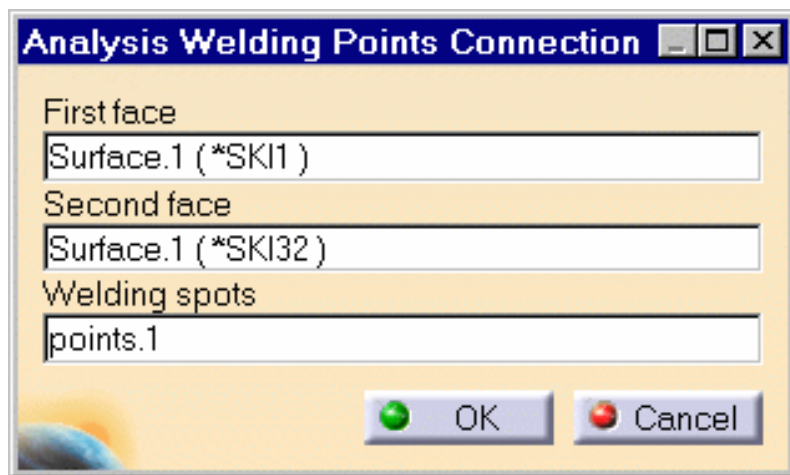
The Analysis Welding Points Connection dialog box is updated:



4. Multi-select a set of points in a single Open Body.

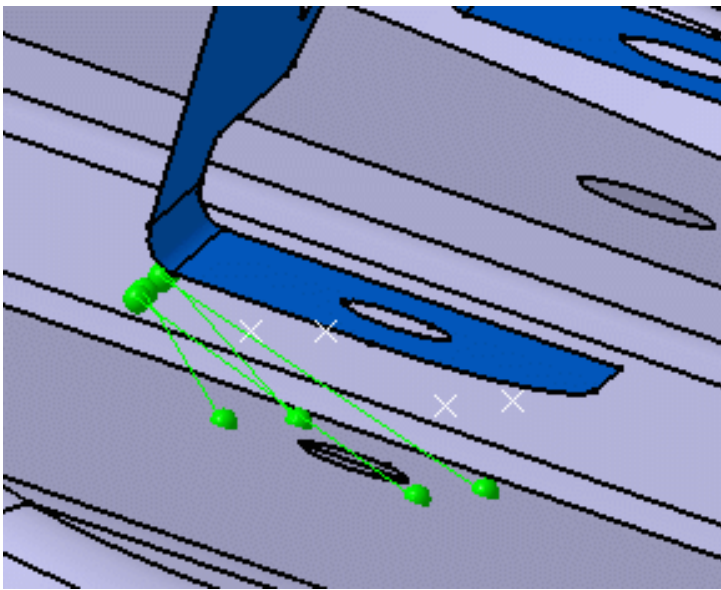


The Analysis Welding Points Connection dialog box is updated:

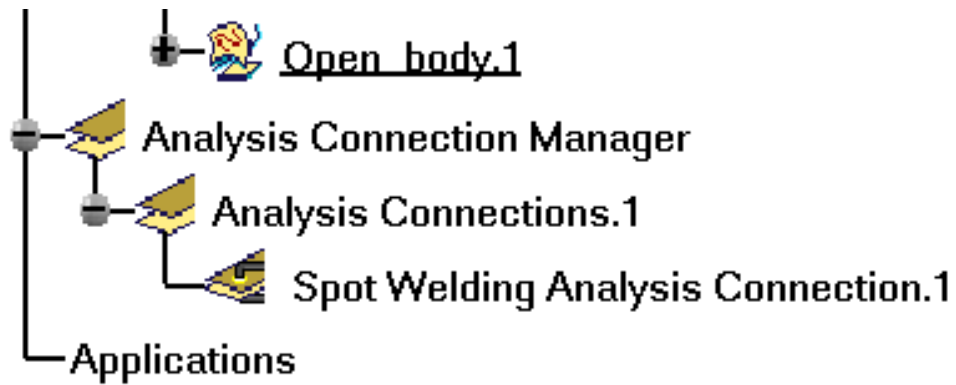


5. Click OK in the Analysis Welding Points Connection dialog box.

Symbols appear on each connected face.



The Spot Welding Analysis Connection.1 object now appears in the specification tree.



Property Connections



Property connections are assembly connections used to specify the boundary interaction between bodies in an assembled system. Once the geometric assembly positioning constraints are defined at the Product level, the user can specify the physical nature of the constraints.



[Creating Fastened Connections:](#) Fastens bodies together at their common interface.



[Creating Slider Connections:](#) Fastens bodies together at their common interface in the normal direction while allowing them to slide relative to each other in the tangential directions.



[Creating Contact Connections:](#) Prevents bodies from penetrating each other at a common interface



[Creating Pressure Fitting Connections:](#) Prevents bodies from penetrating each other at a common interface



[Creating Bolt Tightening Connections:](#) Prevents bodies from penetrating each other at a common interface



[Creating Virtual Rigid Bolt Tightening Connections:](#) Takes into account pre-tension in a bolt-tightened assembly in which the bolt is not included.



[Creating Virtual Spring Bolt Tightening Connections:](#) Specifies the boundary interaction between bodies in an assembled system.



[Creating Rigid Connections:](#) Fastens bodies together at a common rigid interface.



[Creating Smooth Connections:](#) Fastens bodies together at a common soft interface.



[Creating Spot Welding Connections:](#) Fastens bodies together at a common soft interface.

Creating Fastened Connections



A *Fastened Connection* is **the link between two bodies which are fastened together at their common boundary**, and will behave as if they were a single body. From a finite element model viewpoint, this is equivalent to the situation where the corresponding nodes of two compatible meshes are merged together. However, since bodies can be meshed independently, the Fastened Connection is designed to handle incompatible meshes.

The Fastened Connection relations take into account the elastic deformability of the interfaces.

The program proceeds as follows:

- each node of the finer surface mesh is projected parallel to the local outer normal of the first surface onto the second surface mesh.
- a projection point is located whenever possible at the intercept of the projection direction with the second surface mesh (extrapolated at the face boundary by roughly half an element width).
- if a projection point exists, the start node is connected by a kinematical spider element to all nodes of the element face on which the projection point has landed.
- a set of join-type relations (involving interpolation using element shape functions and a rig-beam relations) is computed between the start node degree of freedom and the connected nodes degree of freedom.

Thus, the Fastened Connection generates at most as many spider kinematical elements as there are nodes on the finer surface mesh for which a projection onto the opposite surface mesh exists.



This task shows how to create a Fastened Connection between two parts.



You can use the [sample16.CATAnalysis](#) document: you applied constraints to the assembly (**Assembly Design** workbench).

Make sure you enter the **Generative Structural Analysis** workbench of the **Generative Part Structural Analysis** product and create a Finite Element Model containing a Static Analysis Case from this assembly.



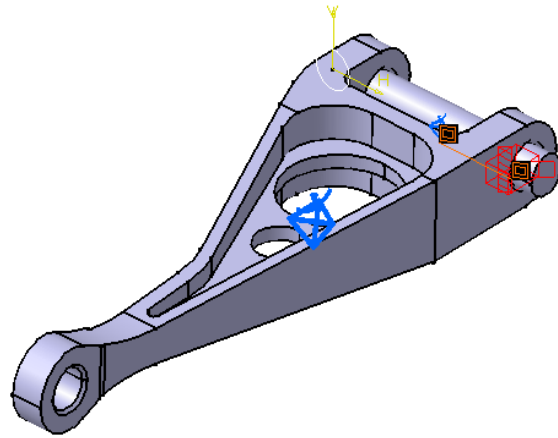
1. Click the Fastened Connection icon



The Fastened Connection dialog box is displayed.



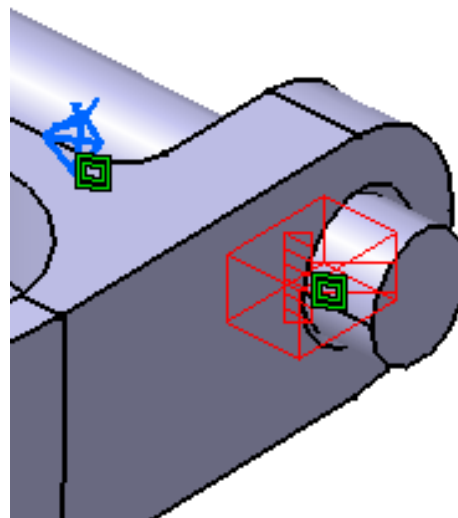
2. Select the assembly constraint previously created in the Assembly Design workbench.



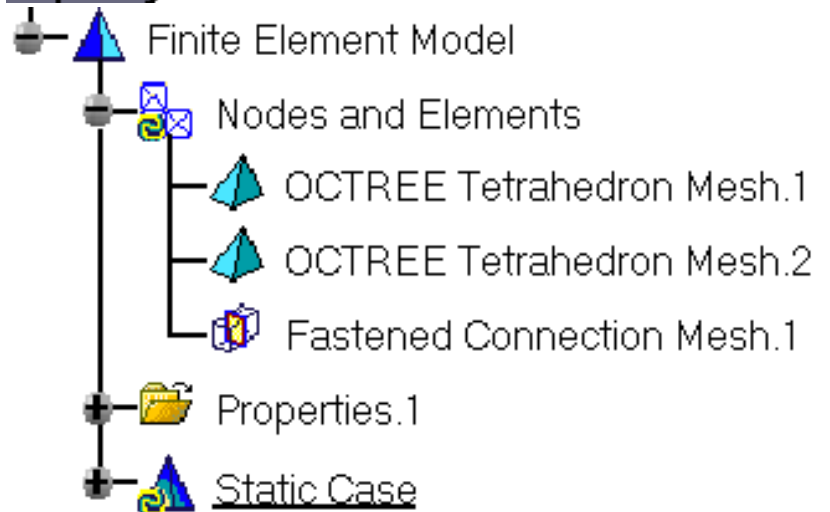
The only allowed constraint type is Contact between surfaces.



3. Click OK to create the Fastened Connection.
A symbol representing the Fastened Connection is visualized on the corresponding faces.



A Fastened Connection object appears in the features tree under the active Nodes and Elements objects set.





- The Finite Element Model contains two Mesh objects, one for each part of the assembly.
- The sizes of the two meshes are different as can be seen by comparing the Mesh Size symbols.



Creating Slider Connections



A *Slider Connection* is **the I two bodies which are constrained to move together in the local normal direction at their common boundary**, and will behave as if they were allowed to slide relative to each other in the local tangential plane. Since bodies can be meshed independently, the Slider Connection is designed to handle incompatible meshes.

The Slider Connection relations take into account the elastic deformability of the interfaces.

The program proceeds as follows:

- each node of the finer surface mesh is projected parallel to the local outer normal of the first surface onto the second surface mesh.
- a projection point is located whenever possible at the intercept of the projection direction with the second surface mesh (extrapolated at the face boundary by roughly half an element width).
- if a projection point exists, the start node is connected by a kinematical spider element to all nodes of the element face on which the projection point has landed.
- a set of join-type relations (involving interpolation using element shape functions and a rig-beam relations) is computed between the start node degrees of freedom and the connected nodes degrees of freedom.
- these relations are projected on the local normal direction yielding a single scalar relation between the start node degrees of freedom and the connected nodes degrees of freedom.

Thus, the Slider Connection generates at most as many spider kinematical elements as there are nodes on the finer surface mesh for which a projection onto the opposite surface mesh exists.



This task shows how to create a Slider Connection between two parts.



You can use the [sample16.CATAnalysis](#) document: you applied constraints to the assembly (**Assembly Design** workbench).

Make sure you enter the **Generative Structural Analysis** workbench of the **Generative Part Structural Analysis** product and create a Finite Element Model containing a Static Analysis Case from this assembly.



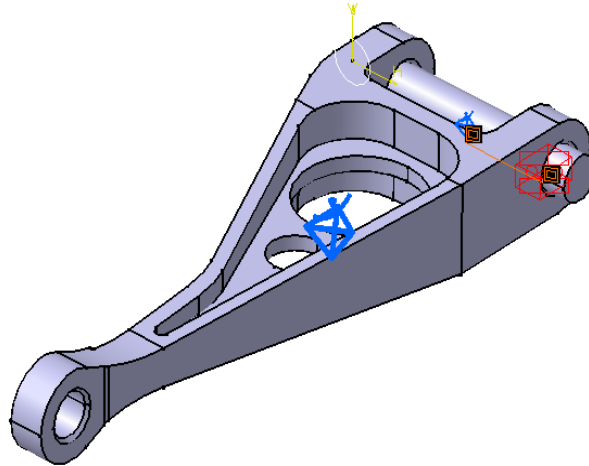
1. Click the Slider Connection icon



The Slider Connection dialog box is displayed.



2. Select an assembly constraint previously created in the Assembly Design workbench.

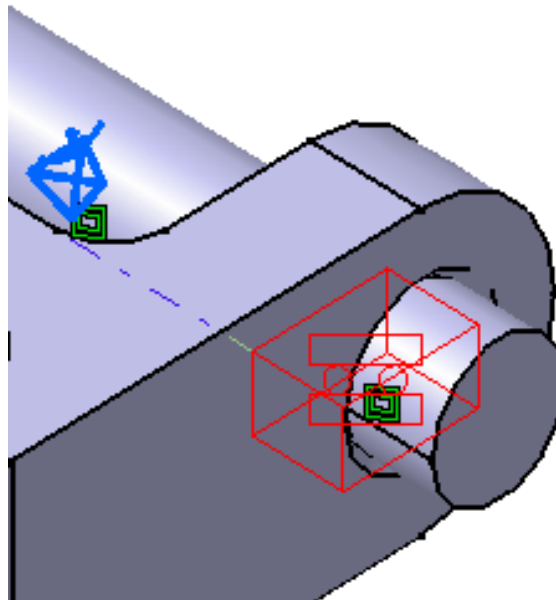


The only allowed constraint type is Contact between surfaces.

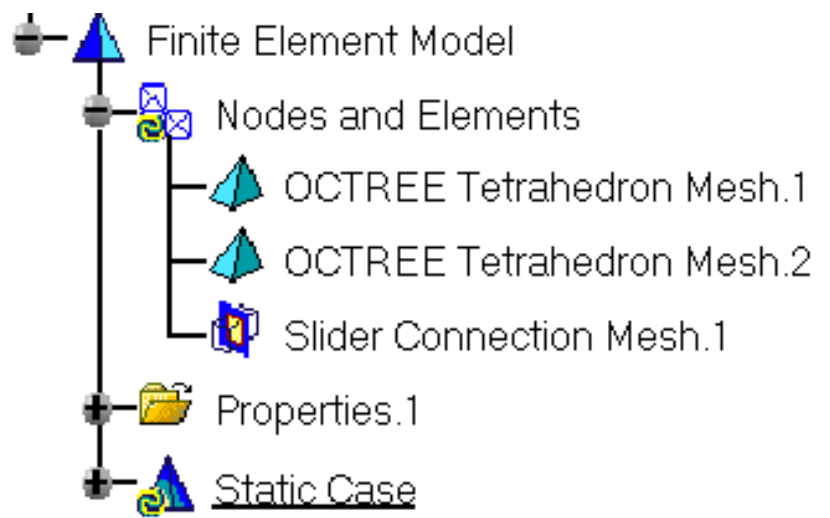


3. Click OK in the Slider Connection dialog box to create the Slider Connection.

A symbol representing the Slider Connection is visualized on the corresponding faces.



A Slider Connection object appears in the features tree under the active Nodes and Elements objects set.



- The Finite Element Model contains two Mesh objects, one for each part of the assembly.
- The sizes of the two meshes are different as can be seen by comparing the Mesh Size symbols.



Creating Contact Connections



A *Contact Connection* is **the link between two part bodies which are prevented from inter-penetrating at their common boundary**, and will behave 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. Since part bodies can be meshed independently, the Contact Connection is designed to handle incompatible meshes.

The Contact Connection relations take into account the elastic deformability of the interfaces.

The Contact **cannot be generated** on two surfaces or between a surface and a part.

The program proceeds as follows:

- each node of the finer body surface mesh is projected parallel to the local outer normal of the first surface onto the second surface mesh.
- a projection point is located whenever possible at the intercept of the projection direction with the second body surface mesh (extrapolated at the face boundary by roughly half an element width).
- if a projection point exists, the start node is connected by a node-to-face element with contact property to all nodes of the element face on which the projection point has landed.
- a set of join-type relations (involving interpolation using element shape functions) is computed between the projection point degrees of freedom and the degrees of freedom of the element face nodes (the projection point virtual degrees of freedom are eliminated in the process).
- rigid body kinematical relations are computed between the start node and the projection node.
- after the elimination of the projection point degrees of freedom, a contact relation is generated by projecting these relations in the local normal direction yielding a single scalar inequality between the start node degrees of freedom and the degrees of freedom of the element face nodes, with a right-hand side equal to the user-defined clearance.

Thus, the Contact Connection generates at most as many node-to-face elements with contact property as there are nodes on the finer surface mesh for which a projection onto the opposite surface mesh exists.



This task shows how to create a Contact Connection between two parts.



Open the [sample16.CATAnalysis](#) document: you applied constraints to the assembly (**Assembly Design** workbench).

Make sure you enter the **Generative Structural Analysis** workbench of the **Generative Part Structural Analysis** product and create a Finite Element Model containing a Static Analysis Case from this assembly.



1. Click the Contact

Connection icon .

The Contact Connection dialog box is displayed.

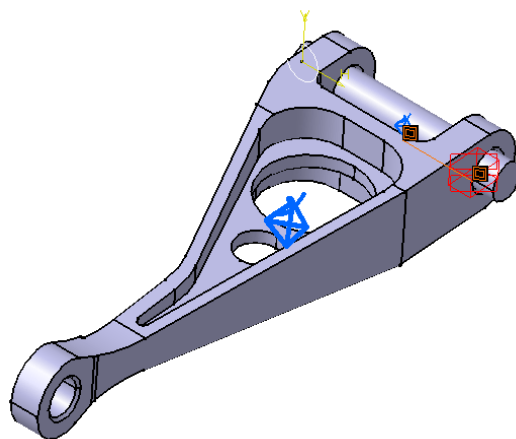


The Clearance field can be used to enter an algebraic value for the maximum allowed normal clearance reduction:

- a positive Clearance value (used to model a known gap between the surfaces) means that the surfaces can still come closer until they come in contact.
- a negative Clearance value (used for instance to model a press-fitted clamp between the surfaces) means that the surfaces are already too close, and the program will have to push them apart.
- the default value used for the Clearance represents the actual geometric spacing between surfaces.

2. Optionally modify the default value of the Clearance parameter.

3. Select an assembly constraint previously created in the **Assembly Design workbench.**

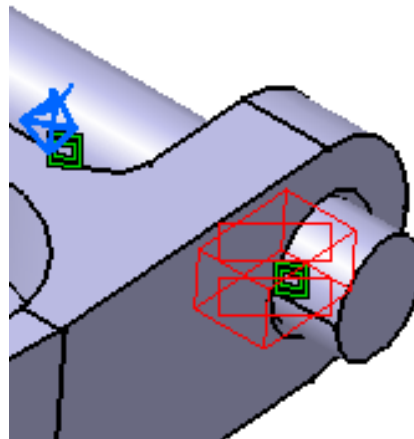


The only allowed constraint type is Contact between body surfaces.

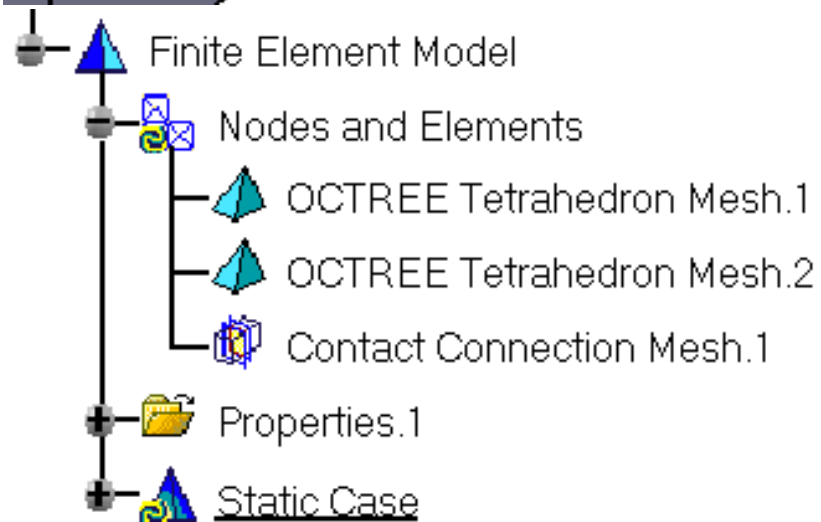


4. Click OK in the Contact Connection dialog box to create the Contact Connection.

A symbol representing the Contact Connection is visualized on the corresponding faces.



A Contact Connection object appears in the specification tree under the active Nodes and Elements objects set.



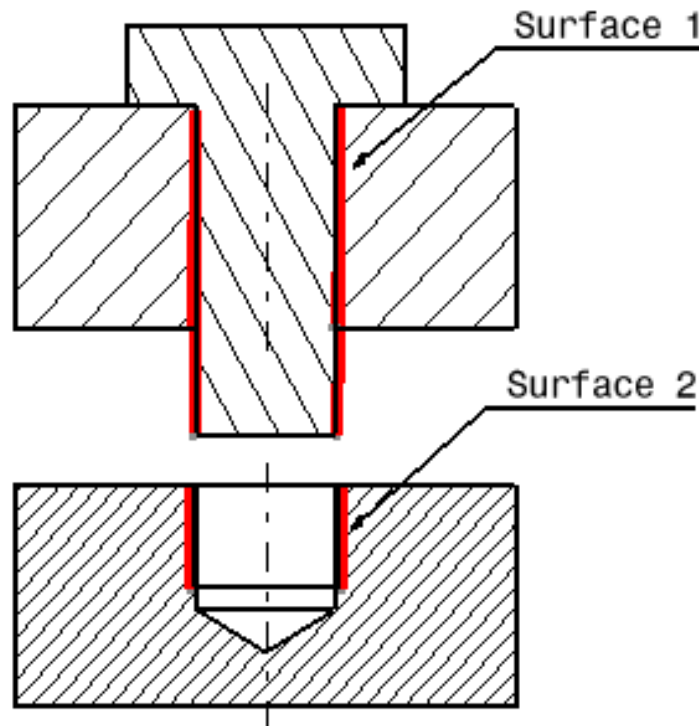
- The Finite Element Model contains two Mesh objects, one for each part of the assembly.
- The sizes of the two meshes are different as can be seen by comparing the Mesh Size symbols.



Creating Bolt Tightening Connections



As a support, the *Bolt Tightening Connection* requires a surface contact assembly constraint between the bolt thread and the bolt support tapping. Note that both these surfaces should be coincident.



Bolt tightening connection

Surface 1 and surface 2 are supports for the assembly constraint of surface contact type.

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 traditional approach. In the first step of the computation, the model is submitted to tension forces relative to bolt tightening by applying opposite forces on the bolt thread and on the support tapping, respectively. Then, in the second step, 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. Since bodies can be meshed independently, the Bolt Tightening Connection is designed to handle incompatible meshes.

The Contact Connection relations take into account the elastic deformability of the interfaces.

The program proceeds as follows:

- each node of the finer surface mesh is projected parallel to the local outer normal of the first surface onto the second surface mesh.

- a projection point is located whenever possible at the intercept of the projection direction with the second surface mesh (extrapolated at the face boundary by roughly half an element width).
- if a projection point exists, the start node is connected by a node-to-face element with Bolt Tightening property to all nodes of the element face on which the projection point has landed.
- a set of join-type relations (involving interpolation using element shape functions) is computed between the degrees of freedom of the start node and the degrees of freedom of element face nodes (the projection point virtual degrees of freedom are eliminated in the process).
- rigid body kinematical relations are computed between the start node and the nodes of element face.
- after the elimination of the projection point degrees of freedom, these relations are rotated in a coordinate frame the third vector of which is aligned with the tension direction (bolt axis).
- two scalar equality relations are generated in the first two directions of the coordinate frame, in order to link the displacement of the start node and the nodes of the element face in the plane normal to the bolt axis.
- a cable relation (the reverse of a contact relation) is generated between the start node and the nodes of element face in the third direction, generating an inequality.

Thus, the Bolt Tightening Connection generates at most as many node-to-face elements with Bolt Tightening property as there are nodes on the finer surface mesh for which a projection onto the opposite surface mesh exists.



This task shows how to create a Bolt Tightening Connection between two parts.



Open the [sample05.CATAnalysis](#) document: you applied constraints to the assembly (**Assembly Design** workbench).

Make sure you enter the **Generative Structural Analysis** workbench of the **Generative Part Structural Analysis** product and create a Finite Element Model from the assembly prior to beginning this task.



1. Click the Bolt Tightening Connection icon



The Bolt Tightening dialog box is displayed.

Bolt Tightening

Name: Bolt Tightening.2

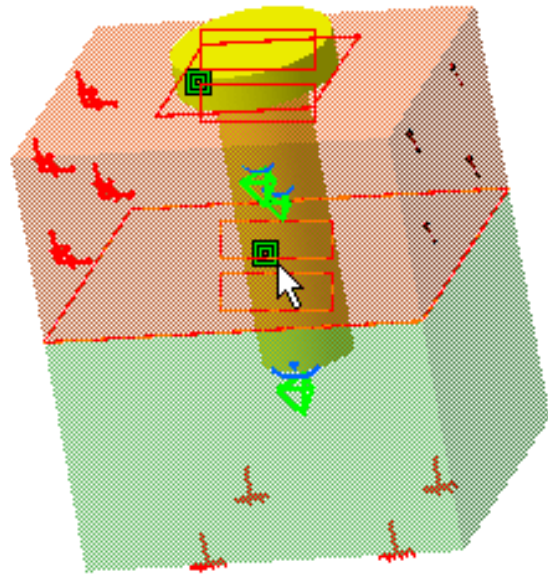
Supports: No selection

Tightening force: 500N

Orientation: Same

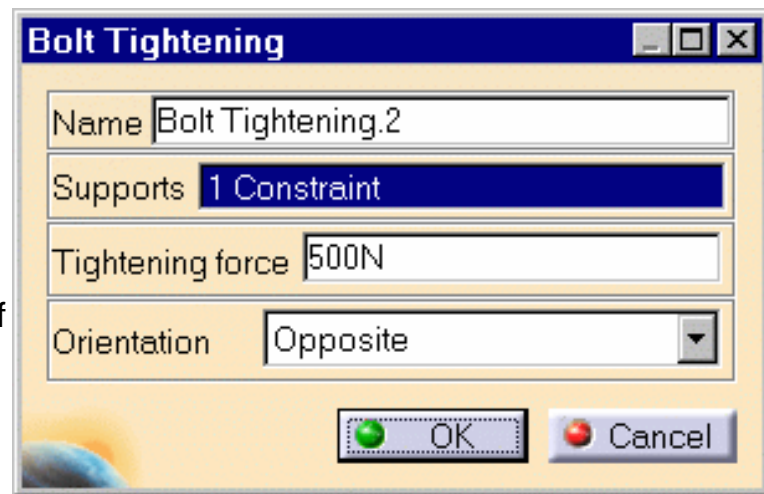
OK Cancel

2. Select a constraint on the assembly previously created in the **Assembly Design** workbench.



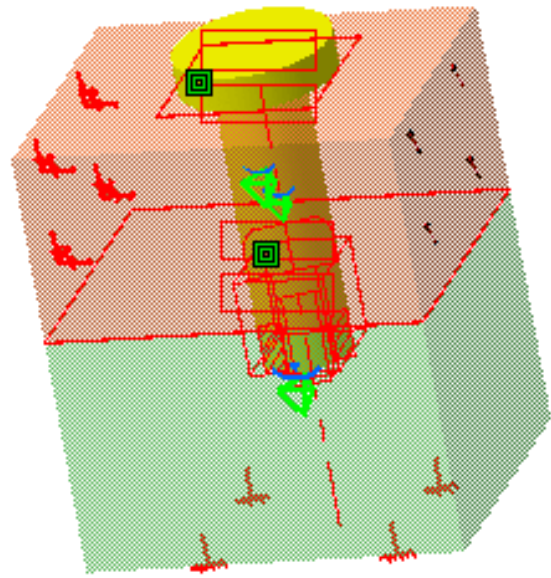
3. Optionally modify the default value of the force and orientation parameters.

Choose either the same or the opposite orientation so that the graphic representation of the bolt tightening connection matches the bolt direction.

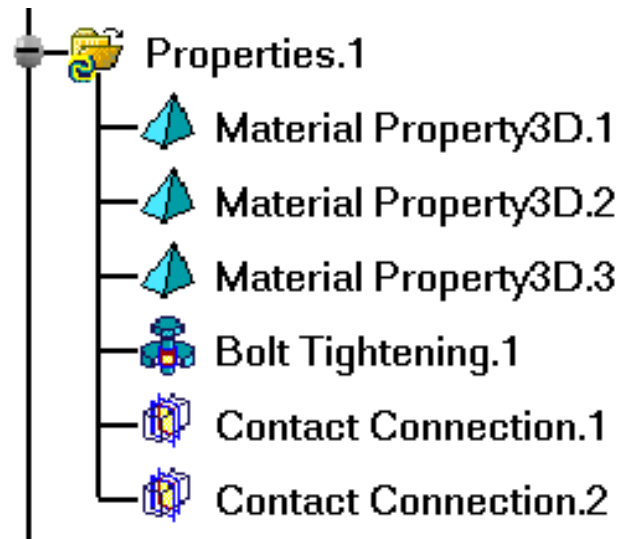


4. Click OK in the Bolt Tightening dialog box to create the Bolt Tightening Connection.

A symbol representing the Bolt Tightening Connection is visualized on the corresponding faces.



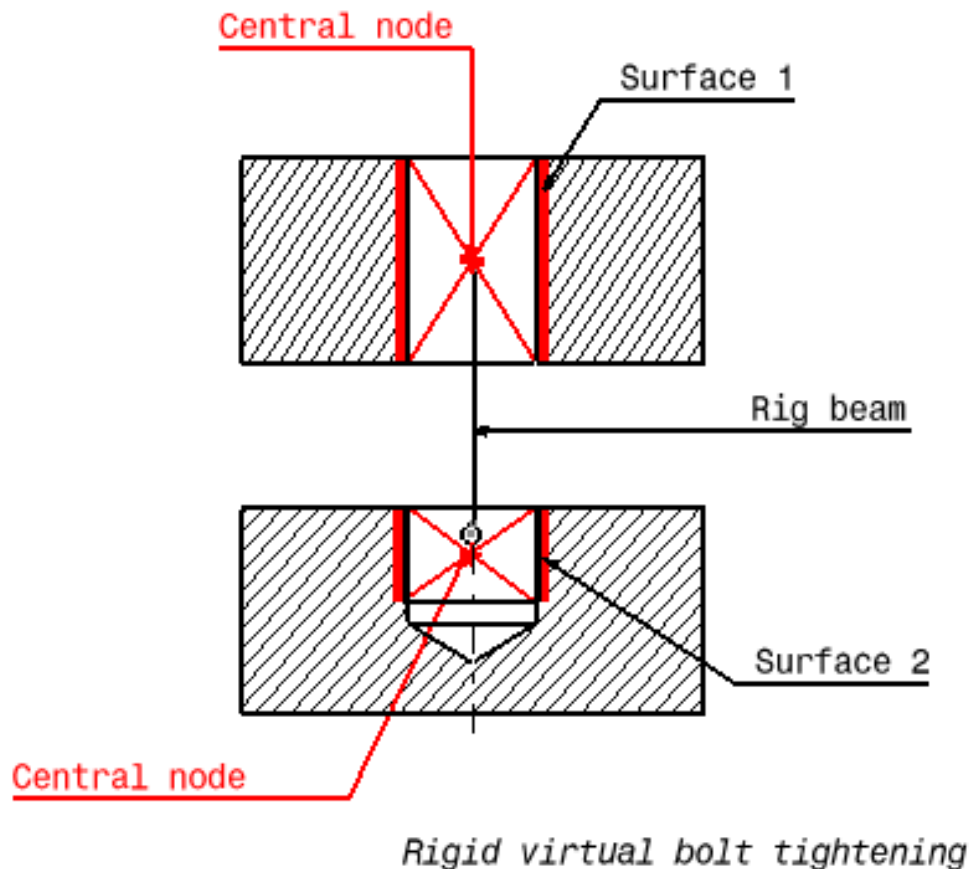
Bolt Tightening.1 Connection object appears in the specification tree under Properties.1.



Creating Virtual Rigid Bolt Tightening Connections



Virtual Rigid Bolt Tightening Connections are used to specify the boundary interaction between bodies in an assembled system. Once the geometric assembly positioning constraints are defined at the Product level, the user can specify the physical nature of the constraints. When creating this connection, both the coincidence constraints and the **Analysis Connections** workbench constraints can be selected.



Surface 1 and surface 2 are supports for the assembly constraints of coincidence type.

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 traditional approach. In the first step of the computation, the model is submitted to tension forces relative to bolt tightening by applying opposite forces on the first surface (S1) and the second surface (S2) of the assembly constraint, respectively. Then, in the second step, 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. Since bodies can be meshed independently, the Virtual Rigid Bolt Tightening Connection is designed to handle incompatible

meshes.

The program proceeds as follows:

- a central node is created at the centroid of each surface of the assembly constraint referenced as the support.
 - for each surface/central node couple, a set of mean (constr-n) relations is generated to link the average displacement of the central node and the nodes of the surface.
 - the first central node is linked rigidly to the duplicata of the second central node.
 - the second central node is linked rigidly to its duplicata except for the translation in the direction of the coincidence constraint.
 - in the direction of the coincidence constraint, a cable relation (the reverse of a contact relation) is generated between translation degrees of freedom of the second central node and its duplicata.
-



This task shows how to create a Virtual Rigid Bolt Tightening Connection between two parts.



Open the [sample11.CATAnalysis](#) document: you applied constraints to the assembly (**Assembly Design** workbench).

Make sure you enter the **Generative Structural Analysis** workbench of the **Generative Part Structural Analysis** product and create a Finite Element Model from the assembly prior to beginning this task.

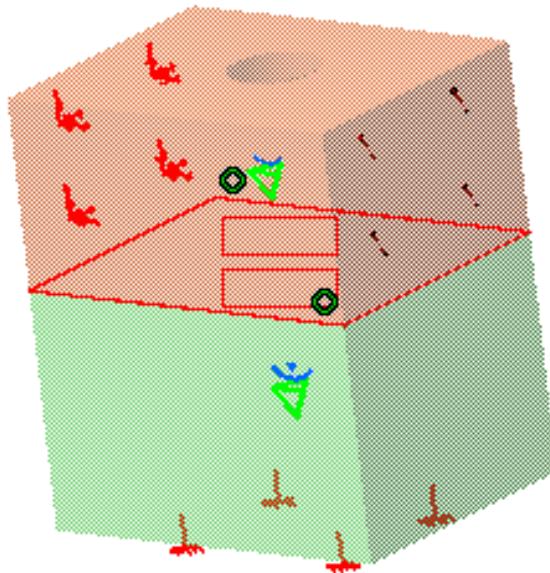


1. Click the Virtual Rigid Bolt Tightening Connection icon

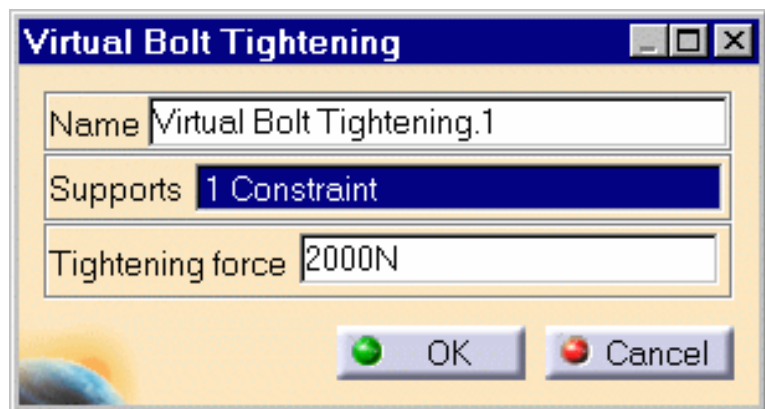


The Virtual Rigid Bolt Tightening dialog box is displayed.

2. Select a constraint on the assembly previously created in the **Assembly Design** workbench.

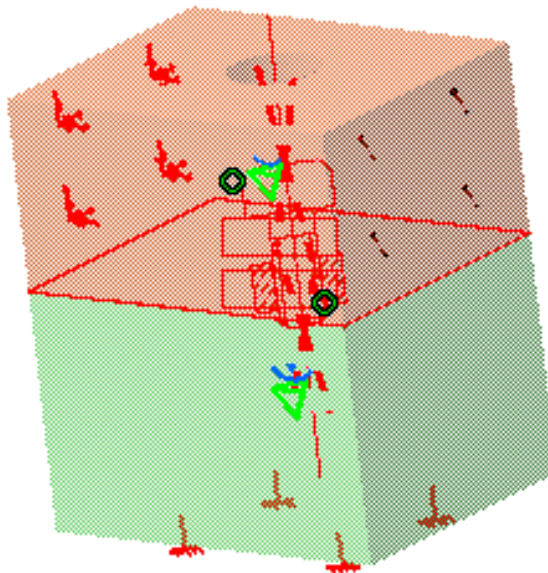


3. Optionally modify the default value of the force and orientation parameters.

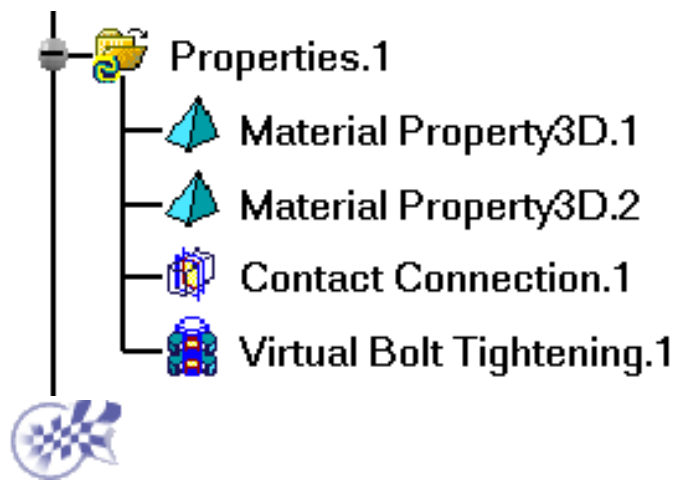


4. Click OK in the Virtual Rigid Bolt Tightening dialog box to create the Virtual Rigid Bolt Tightening Connection.

A symbol representing the Virtual Rigid Bolt Tightening Connection is visualized on the corresponding faces.



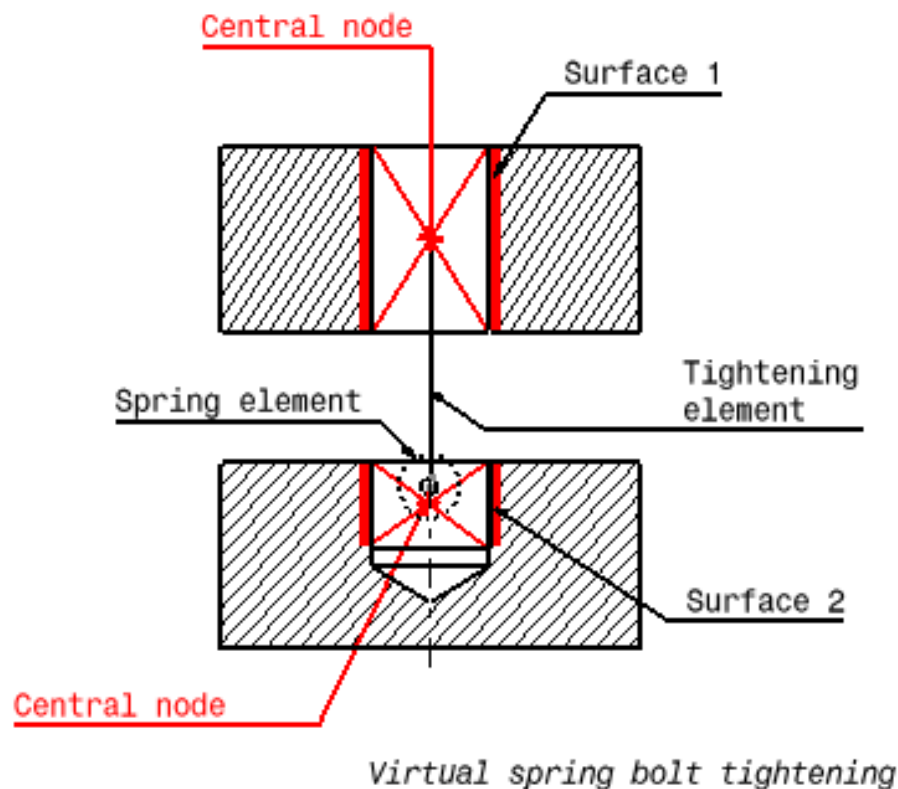
Virtual Bolt Tightening.1 Connection object appears in the specification tree under Properties.1.



Creating Virtual Spring Bolt Tightening Connections



Virtual Spring Bolt Tightening Connections are used to specify the boundary interaction between bodies in an assembled system. Once the geometric assembly positioning constraints are defined at the Product level, the user can specify the physical nature of the constraints. When creating this connection, both the coincidence constraints and the **Analysis Connections** workbench constraints can be selected.



Surface 1 and surface 2 are supports for the assembly constraints of coincidence type.

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 traditional approach. In the first step of the computation, the model is submitted to tension forces relative to bolt tightening by applying opposite forces on the first surface (S1) and the second surface (S2) of the assembly constraint, respectively. Then, in the second step, the relative displacement of these two surfaces (obtained in the first step) is imposed while the model is submitted to user loads. The Virtual Spring Bolt Tightening Connection takes into account the elastic deformability of the surfaces and since bodies can be meshed independently, the Virtual Spring Bolt Tightening Connection is designed to handle incompatible meshes.

The program proceeds as follows:

- a central node is created at the centroid of each surface of the assembly constraint referenced as the support.
- for each surface/central node couple, a set of mean rigid body (constr-n) relations 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 duplicata of the second central node using a tightening element. This element generates:
 - a set of equality relations linking both nodes according to the rigid body motion except for the translation in the direction of the element.
 - a cable inequality relation (the reverse of a contact element) in the direction of the element. This cable relation is used to enforce the relative displacement of both surfaces at the second step of the computation.
- the second central node is linked to its duplicata using a spring element the characteristics of which are defined by the user.




This task shows how to create a Virtual Spring Bolt Tightening Connection between two parts.



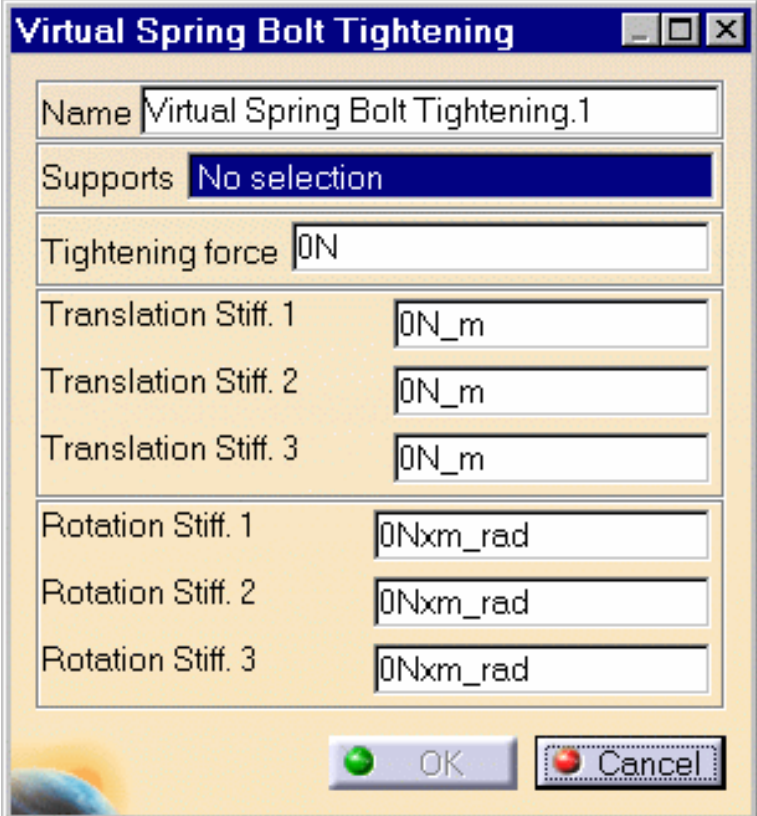
You can use the [sample14.CATAnalysis](#) document: you applied constraints to the assembly (**Assembly Design** workbench).



Make sure you enter the **Generative Structural Analysis** workbench of the **Generative Part Structural Analysis** product and create a Finite Element Model from the assembly prior to beginning this task.



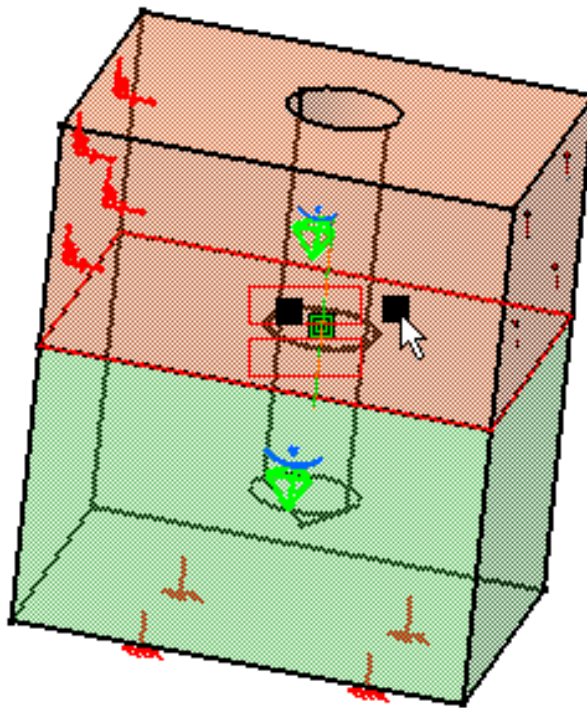
1. Click the Virtual Spring Bolt Tightening Connection icon .

The Virtual Spring Bolt Tightening dialog box is displayed.



Virtual Spring Bolt Tightening	
Name	Virtual Spring Bolt Tightening.1
Supports	No selection
Tightening force	0N
Translation Stiff. 1	0N_m
Translation Stiff. 2	0N_m
Translation Stiff. 3	0N_m
Rotation Stiff. 1	0Nxm_rad
Rotation Stiff. 2	0Nxm_rad
Rotation Stiff. 3	0Nxm_rad
 	

2. Select a constraint on the assembly previously created in the **Assembly Design** workbench.



3. Optionally modify the default value of the force and stiffness parameters.

Virtual Spring Bolt Tightening

Name: Virtual Spring Bolt Tightening.1

Supports: 1 Constraint

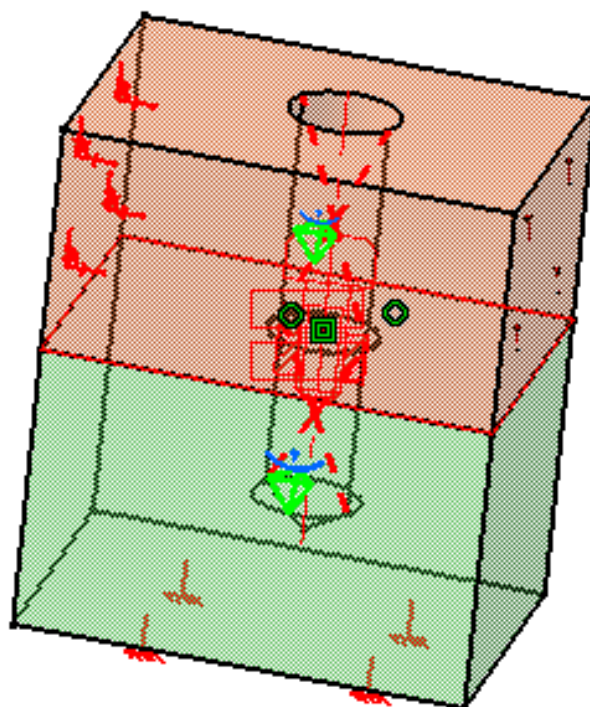
Tightening force: 500N

Translation Stiff. 1	6e+008N_m
Translation Stiff. 2	1e+006N_m
Translation Stiff. 3	1e+006N_m
Rotation Stiff. 1	1e+006Nxm_rad
Rotation Stiff. 2	1e+006Nxm_rad
Rotation Stiff. 3	1e+006Nxm_rad

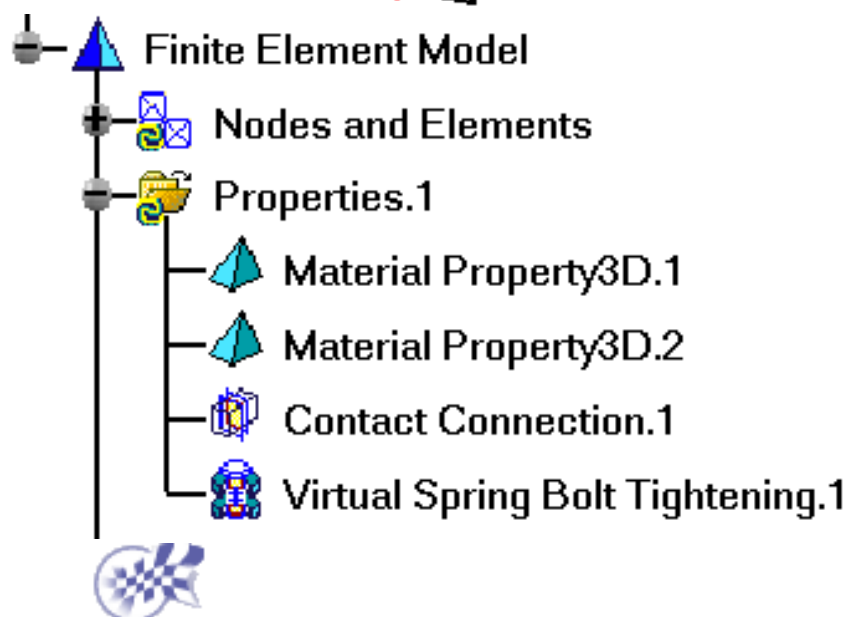
OK Cancel

4. Click OK in the Virtual Spring Bolt Tightening dialog box to create the Virtual Spring Bolt Tightening Connection.

A symbol representing the Virtual Spring Bolt Tightening Connection is visualized on the corresponding faces.



Virtual Spring Bolt Tightening.1
Connection object appears in the
specification tree under
Properties.1.





Creating Pressure Fitting Connections



The *Pressure Fitting Connection* uses assembly surface contact constraint as a support. A Pressure Fitting Connection is **the link between two bodies which are assembled in a Pressure Fitting configuration**, more precisely when there are interferences or overlaps between both parts. 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. Since bodies can be meshed independently, the Pressure Fitting Connection is designed to handle incompatible meshes.

The Pressure Fitting Connection relations take into account the elastic deformability of the interfaces.

The program proceeds as follows:

- each node of the finer surface mesh is projected parallel to the local outer normal of the first surface onto the second surface mesh.
- a projection point is located whenever possible at the intercept of the projection direction with the second surface mesh (extrapolated at the face boundary by roughly half an element width).
- if a projection point exists, the start node is connected by a node-to-face element with contact property to all nodes of the element face on which the projection point has landed.
- a set of join-type relations (involving interpolation using element shape functions) is computed between the degrees of freedom of the start node and the degrees of freedom of element face nodes (the projection point virtual degrees of freedom are eliminated in the process).
- rigid body kinematical relations are computed between the start node and the nodes of element face.
- these relations are rotated in a coordinate system the third vector of which corresponds to the normal of the local surface.
- after the elimination of the projection point degrees of freedom, a pressure fitting relation is generated between the start node and the projected node, transforming the scalar equality relation into an inequality relation with a right-hand side equal to the minus user-defined overlap.
- two scalar equality relations are generated in the tangential plane to link the tangential displacement of the start node and its projection.

Thus, the Pressure Fitting Connection generates at most as many node-to-face elements with Pressure Fitting property as there are nodes on the finer surface mesh for which a projection onto the opposite surface mesh exists.




This task shows how to create a Pressure Fitting Connection between two parts.



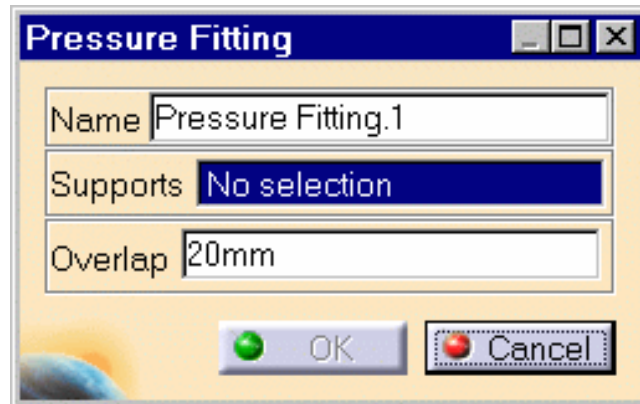
Open the [sample12.CATAnalysis](#) document: you applied constraints to the assembly (**Assembly Design** workbench).

Make sure you enter the **Generative Structural Analysis** workbench of the **Generative Part Structural Analysis** product and create a Finite Element Model from the assembly prior to beginning this task.



1. Click the Pressure Fitting Connection icon .

The Pressure Fitting dialog box is displayed.

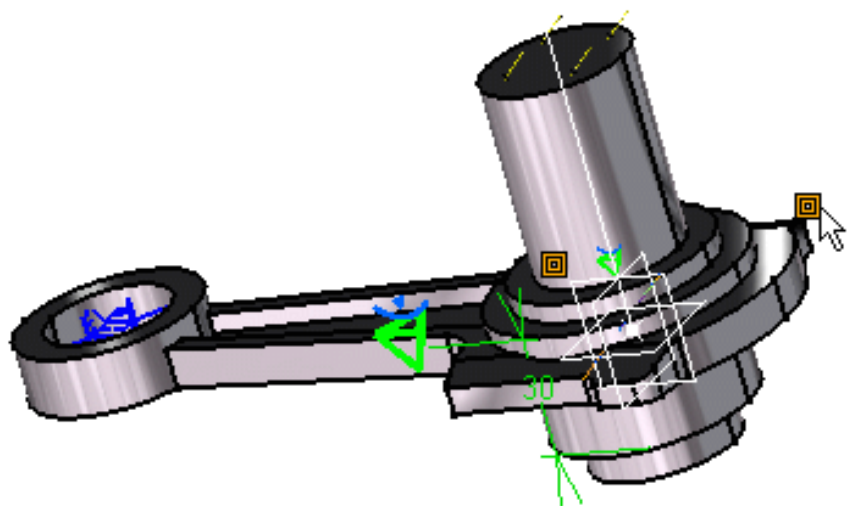


The Overlap field can be used to enter an algebraic value for the maximum allowed normal clearance reduction:

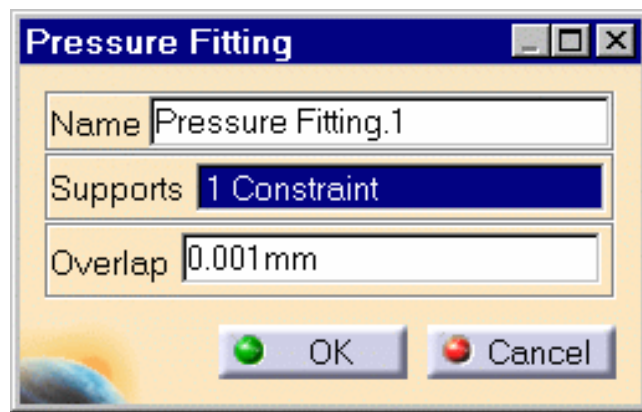
- a positive Overlap value (used for instance to model a press-fitted clamp between the surfaces) means that the surfaces are already too close, and the program will have to push them apart.
- a negative Overlap value (used to model a known gap between the surfaces) means that the surfaces can still come closer until they come in contact.
- the default value used for the Overlap represents the actual geometric spacing between surfaces.

The overlap indicates the interference between both parts. It is intended to be positive.

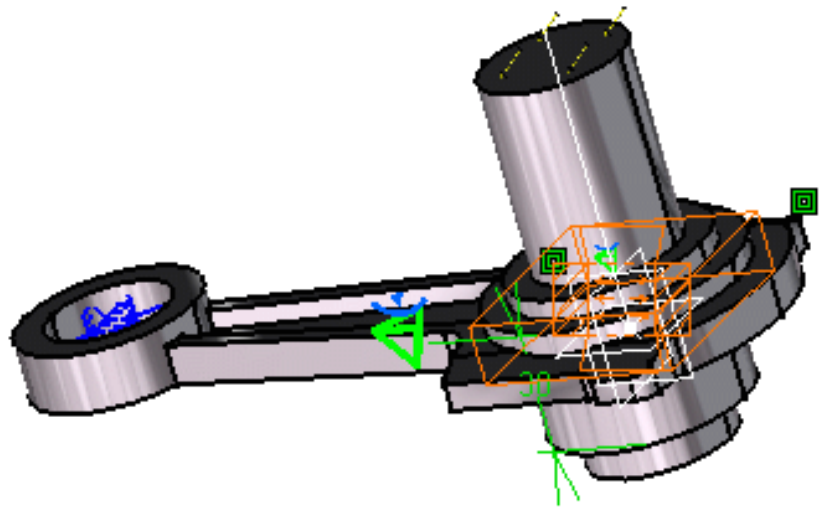
2. Select an assembly constraint previously created in the Assembly Design workbench.



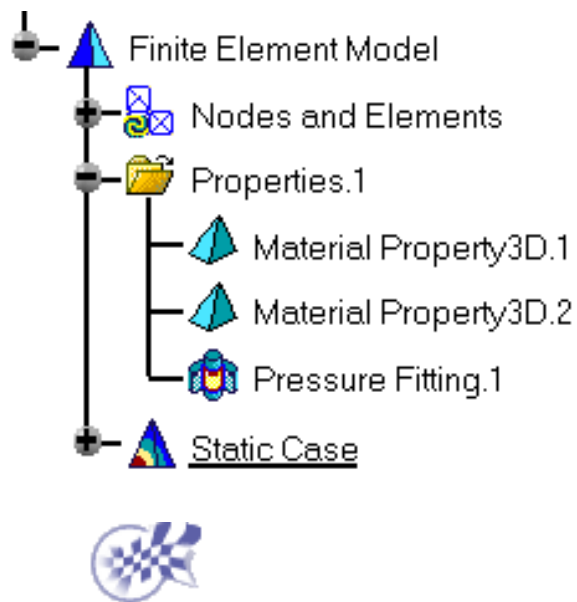
3. Optionally modify the default value of the overlap parameter. In this case, enter 0.001mm.



4. Click OK to create the Pressure Fitting Connection. A symbol representing the Pressure Fitting Connection is visualized on the corresponding faces.



Pressure Fitting.1 Connection object appears in the specification tree under Properties.1.



Creating Rigid Connections



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. Since bodies can be meshed independently, the Rigid Connection is designed to handle incompatible meshes.

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

The program proceeds as follows:

- a central node is created at the midpoint between the 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.
- a set of rig-beam relations is generated between the central node degree of freedom and the connected nodes degree of freedom.

Thus, the Rigid Connection generates as many rig-beam kinematical elements as there are nodes on the two surface meshes.



This task shows how to create a Rigid Connection between two parts.



Open the [sample17.CATAnalysis](#) document: you applied constraints to the assembly (**Assembly Design** workbench).

Make sure you enter the **Generative Structural Analysis** workbench of the **Generative Part Structural Analysis** product and create a Finite Element Model containing a Static Analysis Case from this assembly.



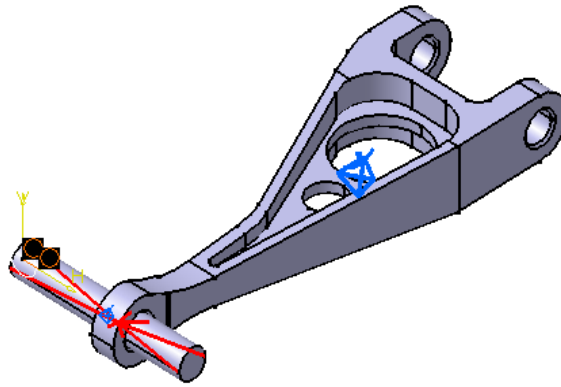
1. Click the Rigid Connection icon



The Rigid Connection dialog box is displayed.



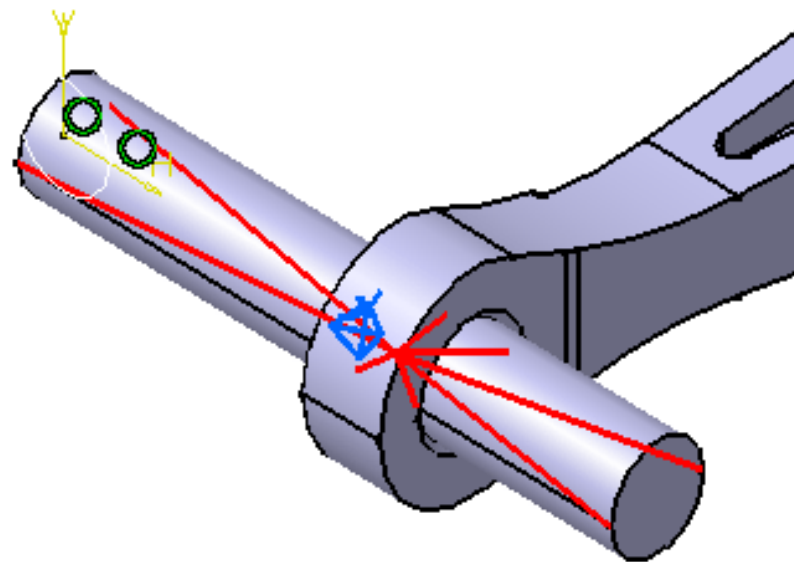
2. Select the Coincidence assembly constraint previously created in the **Assembly Design** workbench.



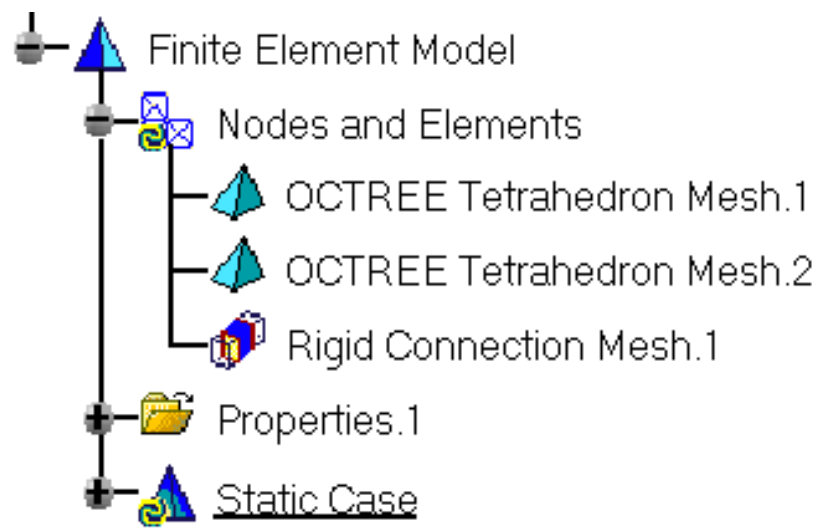
The allowed assembly constraint types are Coincidence, Offset and Angle.



3. Click OK in the Rigid Connection dialog box to create the Rigid Connection.
A symbol representing the Rigid Connection is visualized on the corresponding faces.



A Rigid Connection object appears in the features tree under the active Nodes and Elements objects set.



- The Finite Element Model contains two Mesh objects, one for each part of the assembly.
- The sizes of the two meshes are different as can be seen by comparing the Mesh Size symbols.



Creating Smooth Connections



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. Since bodies can be meshed independently, the Smooth Connection is designed to handle incompatible meshes.

The Smooth Connection relations take approximately into account the elastic deformability of the interfaces. The approximation is based on a least squares fit of a slave node degree of freedom rigidly linked to the master nodes (element shape functions are ignored).

The program proceeds as follows:

- a central node is created at the midpoint between the 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.
- a set of mean (constr-n) relations is generated between the central node degree of freedom and the connected nodes degree of freedom.

Thus, the Smooth Connection generates two spider kinematical elements.



This task shows how to create a Smooth Connection between two parts.



Open the [sample17.CATAnalysis](#) document: you applied constraints to the assembly (**Assembly Design** workbench).

Make sure you enter the **Generative Structural Analysis** workbench of the **Generative Part Structural Analysis** product and create a Finite Element Model containing a Static Analysis Case from this assembly.



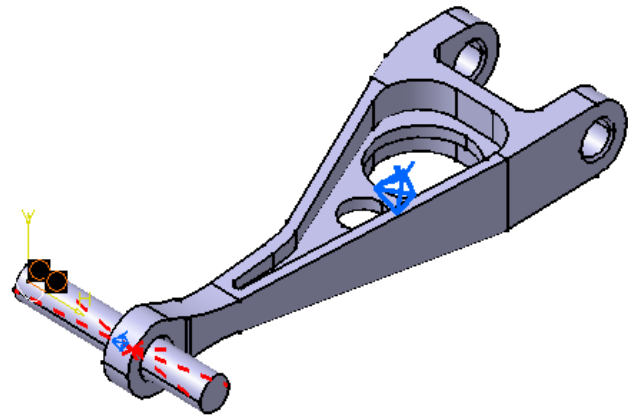
1. Click the Smooth Connection icon



The Smooth Rigid dialog box is displayed.



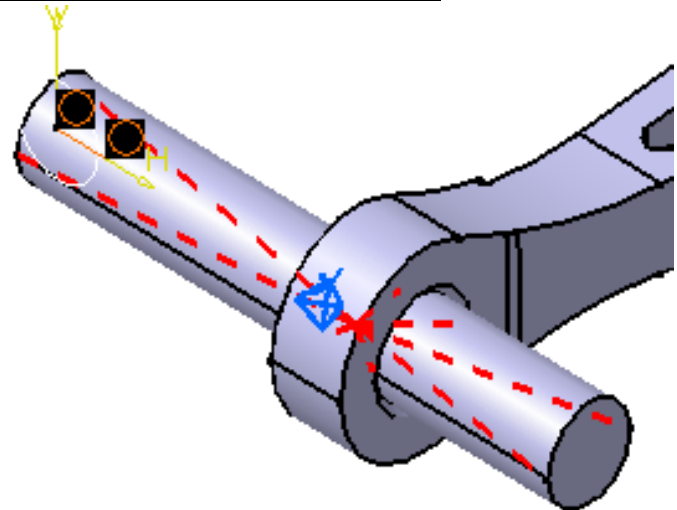
2. Select the Coincidence assembly constraint previously created in the Assembly Design workbench.



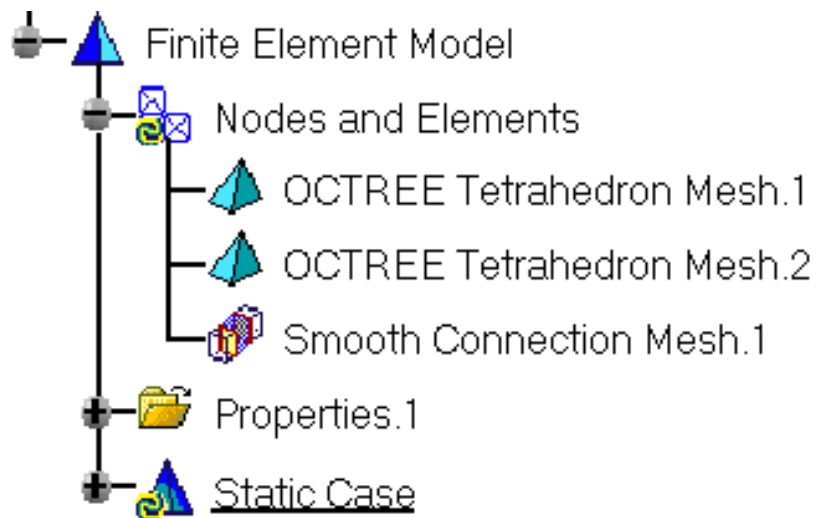
The allowed assembly constraint types are Coincidence, Offset and Angle.



3. Click OK to create the Smooth Connection.
A symbol representing the Smooth Connection is visualized on the corresponding faces.



A Smooth Connection object appears in the features tree under the active Nodes and Elements objects set.



- The Finite Element Model contains two Mesh objects, one for each part of the assembly.
- The sizes of the two meshes are different as can be seen by comparing the Mesh Size symbols.





Creating Spot Welding Connections



A *Spot Welding Connection* is **the link between two bodies, using analysis welding point connections.**



This task shows how to create a Spot Welding Connection between two parts.



Open [sample18.CATAnalysis](#) sample.

Make sure you enter the **Generative Structural Analysis** workbench of the **Generative Part Structural Analysis** product and create a Finite Element Model containing a Static Analysis Case from this assembly.

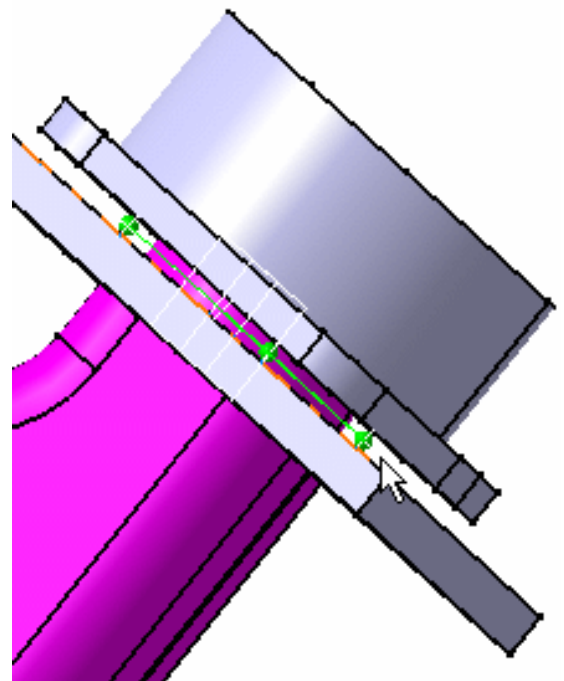


1. Click the Spot Welding Connection icon .

The Spot Welding Connection dialog box is displayed.



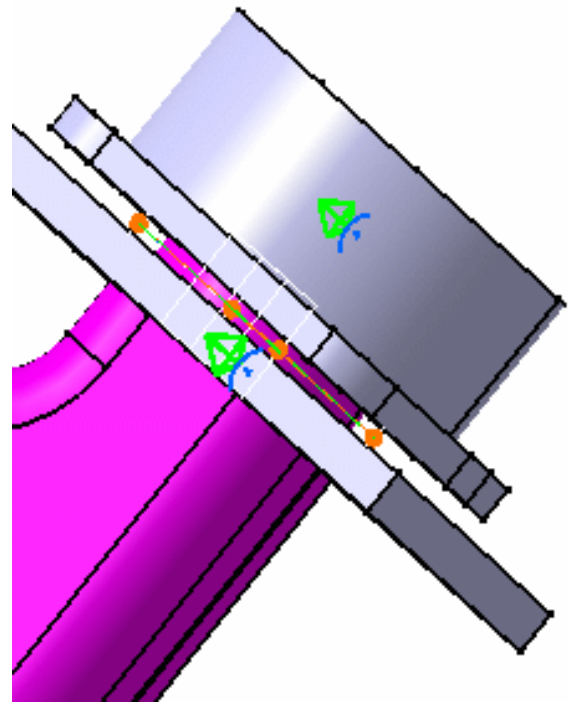
2. Select the Supports to be assigned spot welding connections. In this case, select the Analysis welding points (Joint Body) that were previously created.



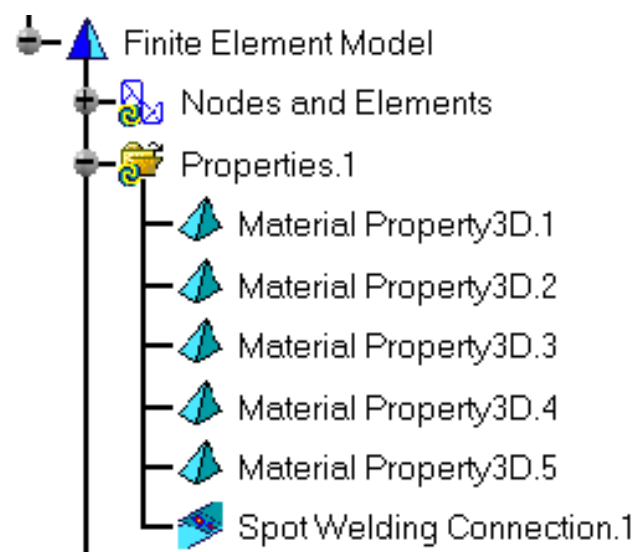
The Spot Welding Connection dialog box is updated.



3. Click OK in the Spot Welding Connection dialog box



A Spot Welding Connection object appears in the specification tree under the Properties objects set.





Workbench Description

This section contains the description of the icons and menus which are specific to the **Generative Assembly Structural Analysis** product, either:

- from the **Generative Part Structural Analysis** workbench



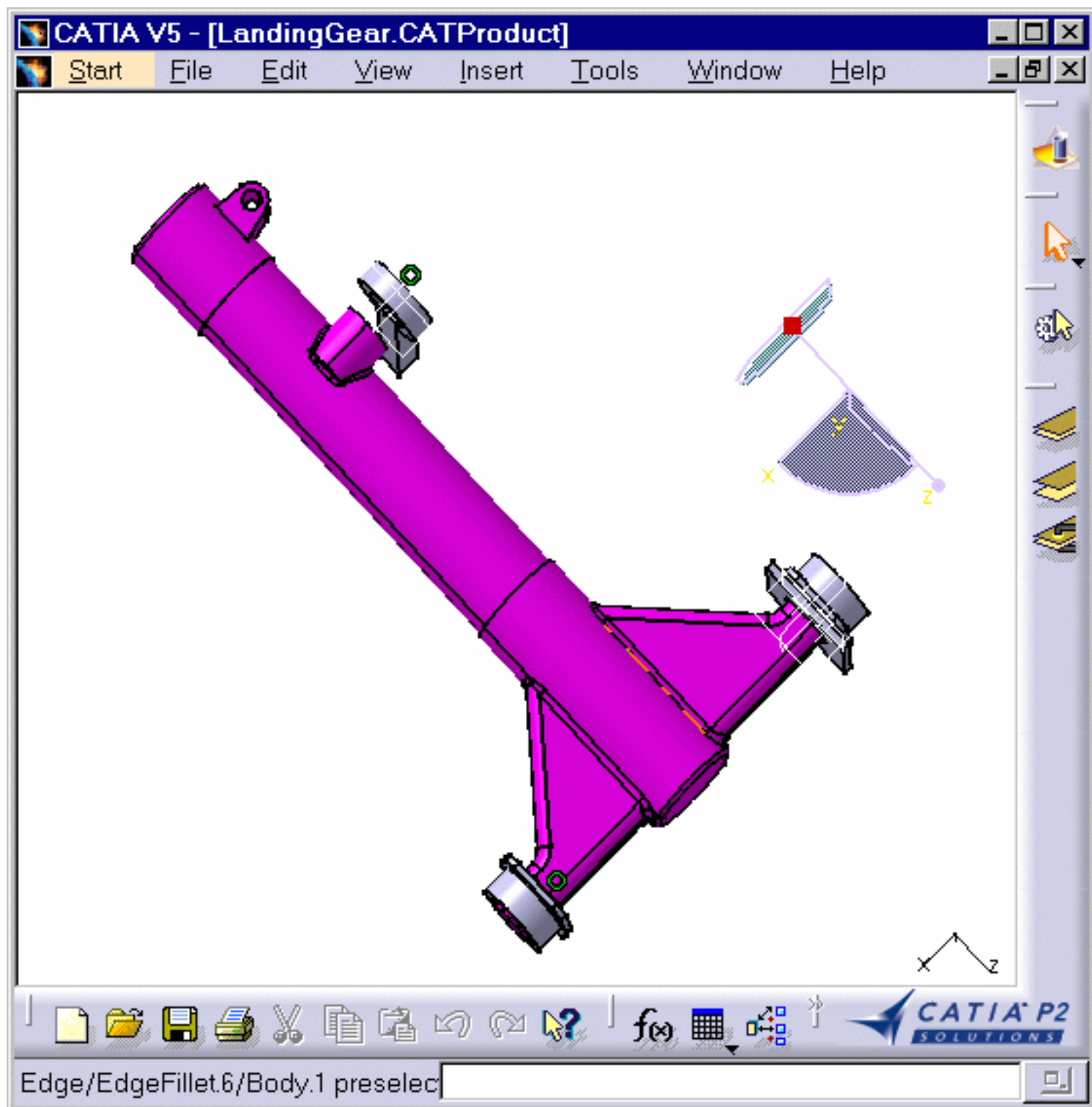
or

- from the **Analysis Connections** workbench



The menu items can be found on the top border of your analysis document.

The icons are grouped in toolbars, which can be found on the right and bottom border of your analysis document.



[Property Connections Toolbar](#)
[Analysis Connection Toolbar](#)

Property Connections Toolbar



The Mesh Specification toolbar (**Generative Assembly Structural Analysis** workbench) gives access to the following tools for defining Property Connections and Advanced Connections:



Connections



See [Fastened Connections](#)



See [Slider Connections](#)



See [Contact Connections](#)



See [Rigid Connections](#)



See [Smooth Connections](#)

Advanced Connections



See [Pressure Fitting Connections](#)



See [Bolt Tightening Connections](#)



See [Virtual Rigid Bolt Tightening Connections](#)



See [Virtual Spring Bolt Tightening Connections](#)



See [Spot Welding Connections](#)

Analysis Connections Toolbar



The Analysis Connections toolbar (**Analysis Connections** workbench) gives you direct access to the following tools for defining Analysis Connections:



Analysis Connections



See [Face Face Connections](#)



See [Distant Connections](#)



See [Welding Points Connections](#)




Index

A


Analysis connections

distant connection_ 

face face connection 


welding point connection 

B


bolt tightening connection 


C


connections


bolt tightening connection 

contact connection 

distant Analysis connection_ 

face face Analysis connection 

fastened connection 


pressure fitting connection 


rigid connection 

rigid virtual bolt tightening 

slider connection 

smooth connection 


spot welding connection 

virtual spring bolt tightening 

welding point Analysis connection 


contact connection 

F


fastened connection 

face-to-face connection 

P


pressure fitting connection 

property connections


bolt tightening connection 

contact connection 

fastened connection 


pressure fitting connection 

rigid connection 


rigid virtual bolt tightening 

slider connection 

smooth connection 

virtual spring bolt tightening 

R

rigid connection 


rigid virtual bolt tightening connection 

virtual spring bolt tightening connection 

S

slider connection 

smooth connection 

spot welding connection 

W

welding point Analysis connection 

workbench