Student Notes:

CATIA V5 Training

Foils

CATIA Product Design Expert

Version 5 Release 19
September 2008

EDU_CAT_EN_ASM_AF_V5R19

Student Notes:

About this course

Objectives of the course

Upon completion of this course, you will be able to:

- Modify CATIA options in order to optimize performance for large, complex designs

- Manage contextual links between product documents by using publications

- Create and use parameters to drive a product design

- Create sections to visualize internal product structure

- Create scenes and exploded views

- Generate annotations and bills of material for assembly drawings

Targeted audience

Mechanical Designers

Prerequisites

Students attending this course should have the knowledge of CATIA Product Design, CATIA Part Design

2 days

Student Notes:

Table of Contents (1/3)

Student Notes:

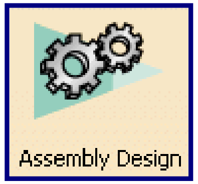
Table of Contents (2/3)

Student Notes:

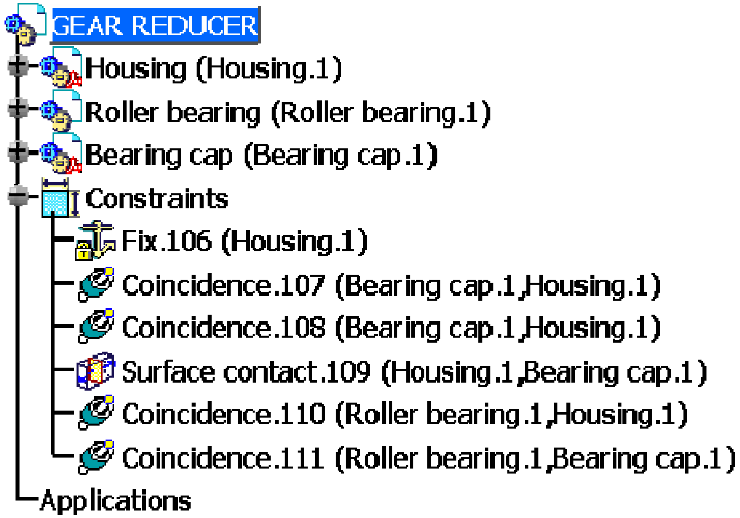
Table of Contents (3/3)

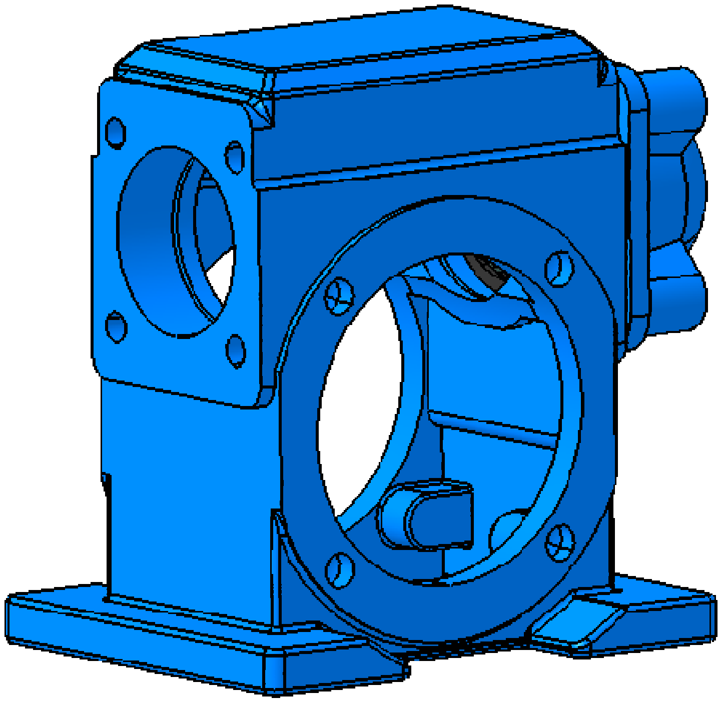
Student Notes:

Introduction to Product Design Expert (1/2)



In the Product Design course, you have learned how to assemble the components into a Product.



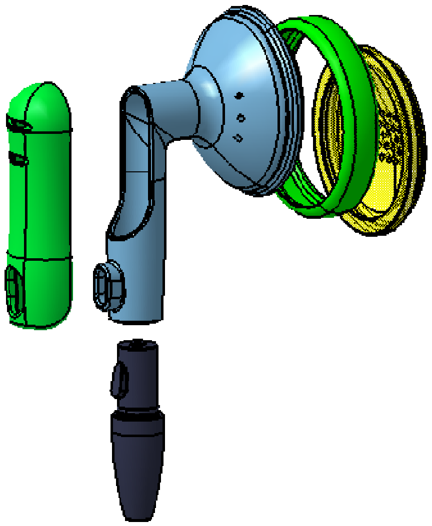


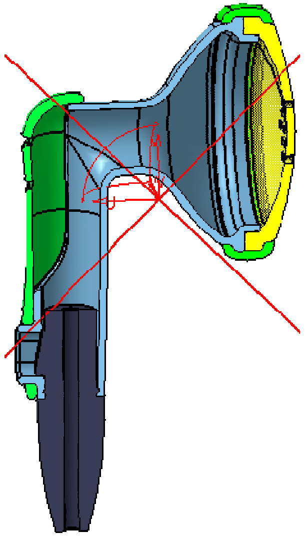
Student Notes:

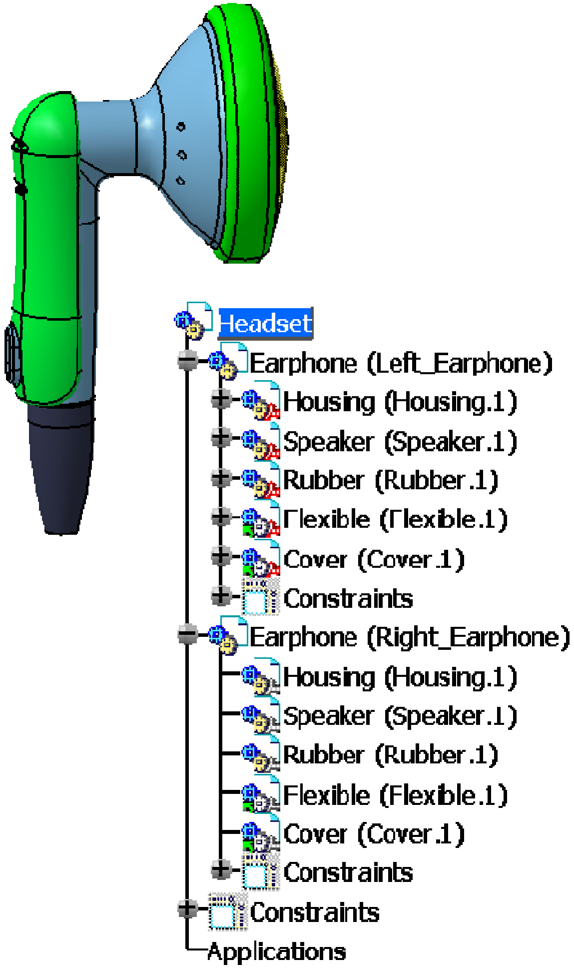
Introduction to Product Design Expert (2/2)

In the Product Design Expert course, you will learn how to improve your assemblies in a progressive approach:

- **Analyze the links between the components of a Product.**
- **Create the different components of the product with associative design using publication.**
- **Make some sub-assemblies flexible.**
- **Manage the representations of the product in order to improve the display performances of large assemblies.**
- **Analyze the product and prepare its drawing.**







Student Notes:

Managing a Product Structure

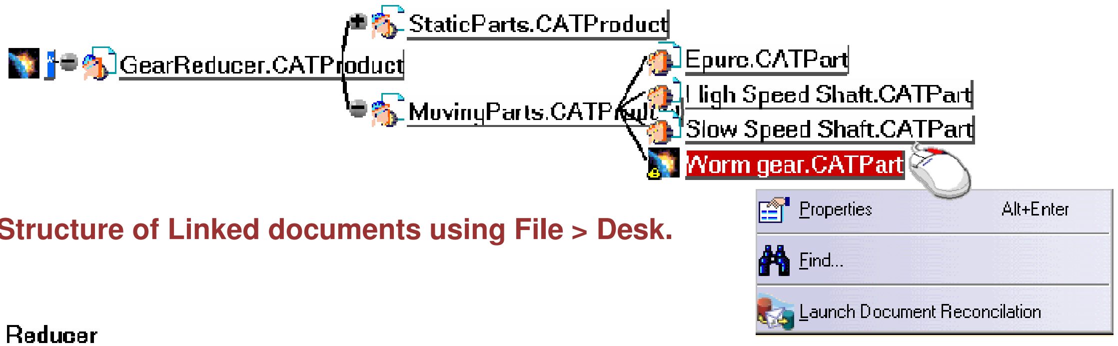
In this lesson, you will learn how to manage product structure of assembly documents.

- Introduction to Managing a Product Structure
- Managing Links Between Components
- Generate CATPart from CATProduct
- Recap Exercise: Bicycle Assembly
- To Sum Up

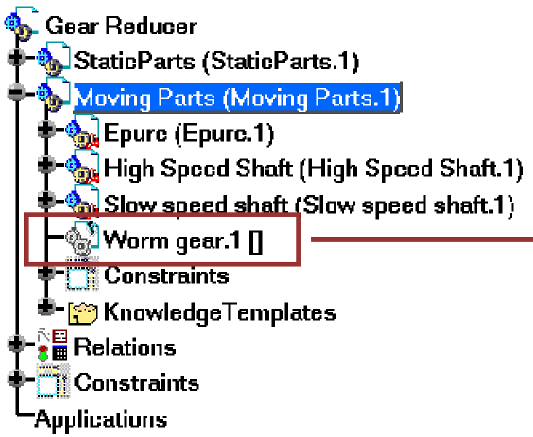
Introduction to Managing a Product Structure (1/2)

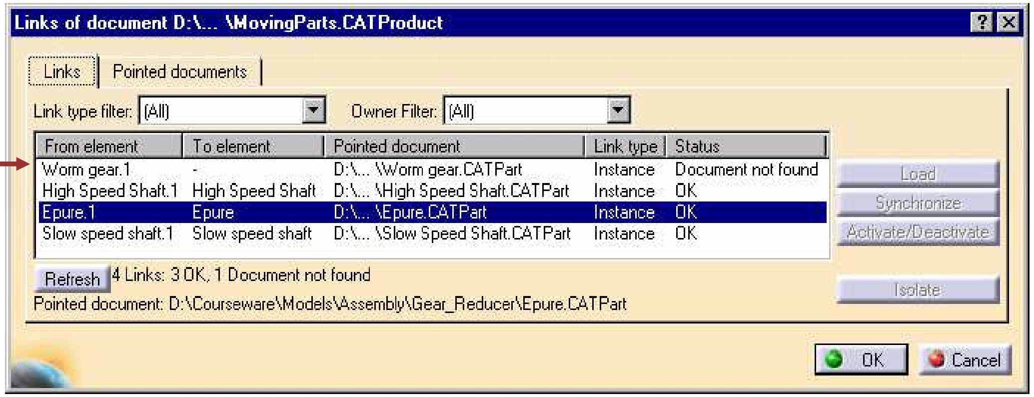
Student Notes:

You can manage a product structure of an assembly document using File > Desk or Edit > Links command. You can load / unload, activate / deactivate, replace , find documents using File > Desk or Edit > Links command.



Structure of Linked documents using File > Desk.



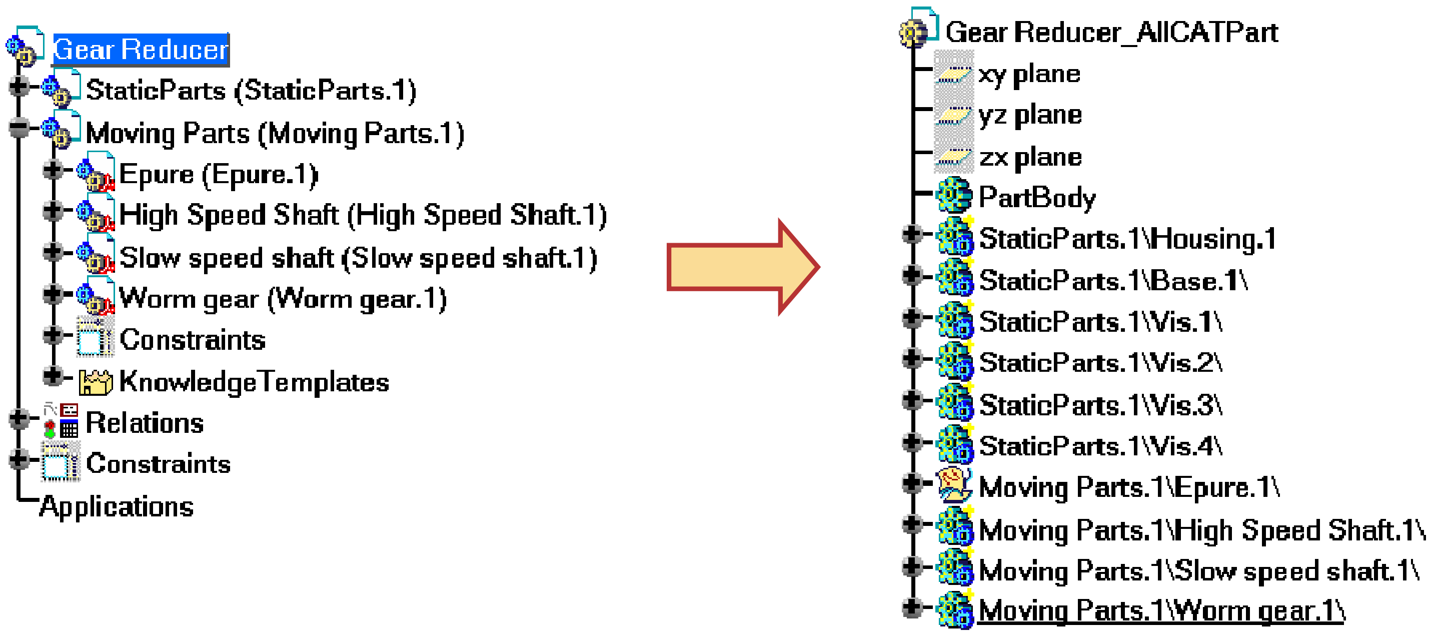


Links of selected component using Edit > Links.

Student Notes:

Introduction to Managing a Product Structure (2/2)

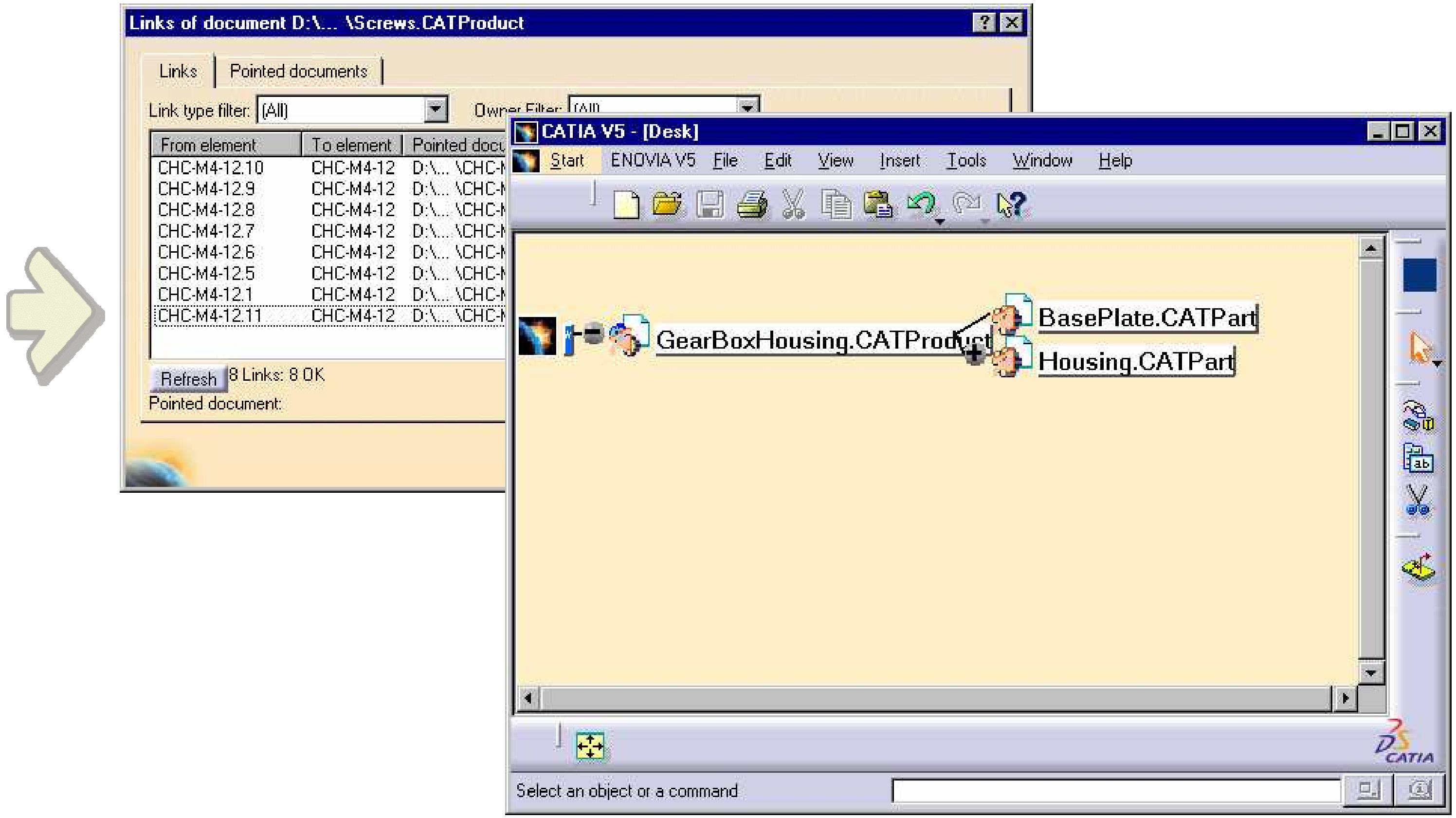
You can suppress the product design details by generating a single CATPart from a CATProduct using Tools > Generate CATPart from Product .



Student Notes:

Managing Links Between Components

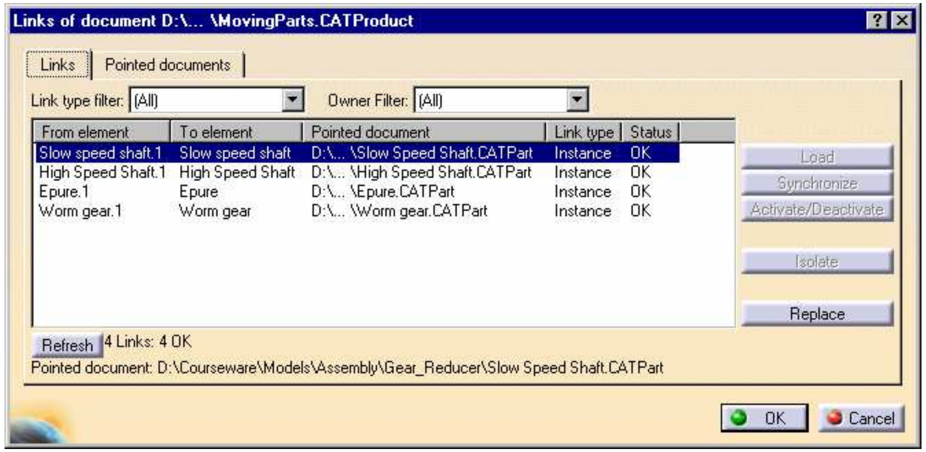
You will learn how to manage links between Components of a product.



Student Notes:

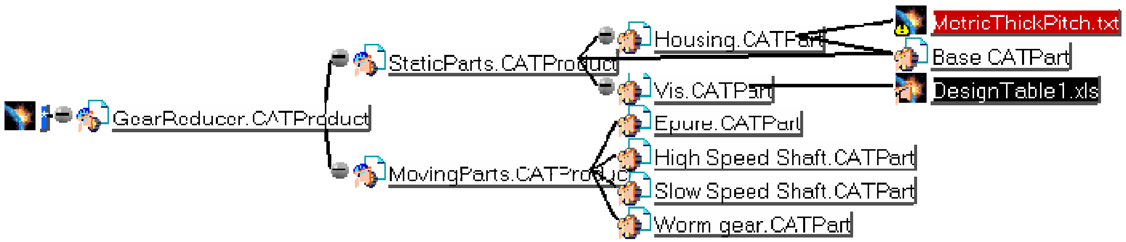
What are Links Between Components in a Product?

In an assembly document, "Links" are maintained between all related CATProducts, CATParts, cgr files and documents (.txt and .xls files) which are referred by Design tables, etc. These links can be seen from Edit > Links Menu or by using File > Desk command.



Edit Links Window displays linked documents

File > Desk command displays a graph showing linked documents



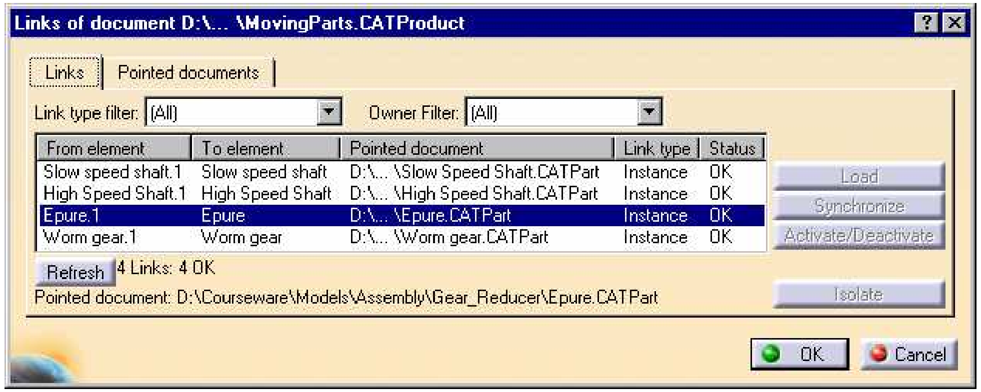
Student Notes:

Why Manage Links?

Using Edit Links and File Desk command, you can perform a number of tasks related to managing the product structure of a product.

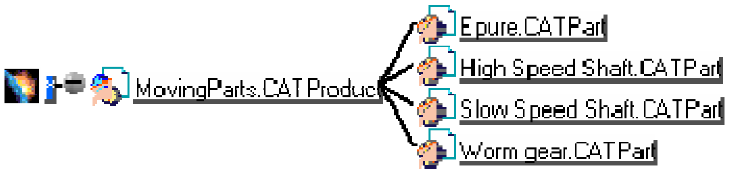
Using Edit Links, it is possible to :

- **Quickly analyze the broken links.**
- **Load / Unload individual components.**
- **Activate / Deactivate components.**
- **Isolate components.**
- **Replace components.**



Using File Desk, it is possible to :

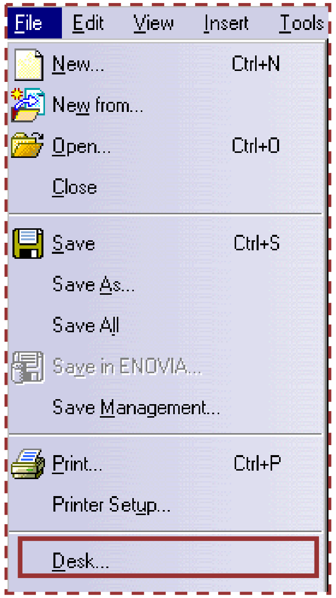
- **Visualize structure of linked components.**
- **Load / Unload individual components.**
- **See the links of CATProduct.**
- **View the properties of component.**
- **Find missing components and re establish links .**



Student Notes:

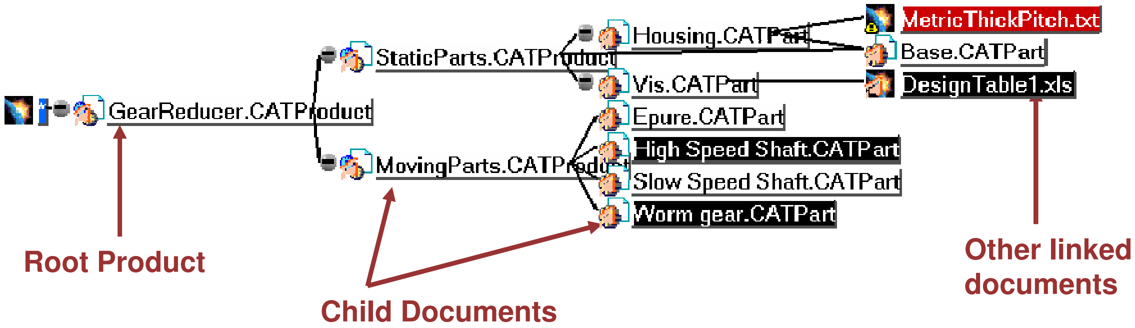
Accessing File Desk Command

You will see how to access File Desk command.



Open a CATProduct and Click on File > Desk.

A new window displays a tree with CATProduct at the root and its child documents. These documents are CATParts, CATProducts, V4 models, other documents such as Text files and Excel documents.



The colors used to identify the various document types are the following ones:

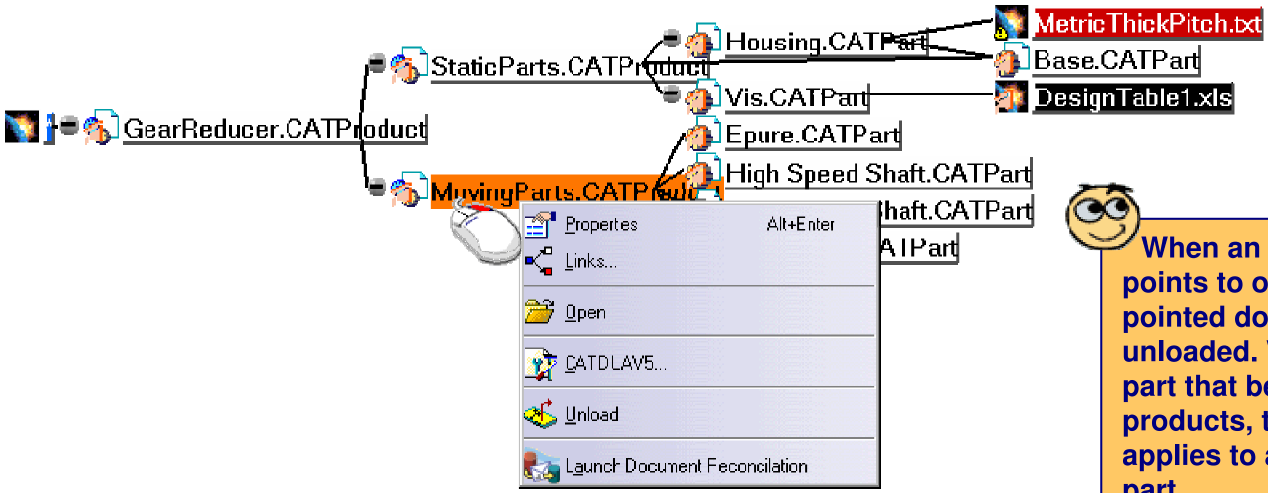
- **White for loaded documents**
- **Black for documents that are not loaded in the current session**
- **Red for documents that have not been found.**

Student Notes:

Managing Links of a Product Using File Desk (1/2)

You will learn how to use File Desk command to manage component links.

To access contextual menu, right click on a desired component in the tree. Using contextual menu, you can load / unload documents, search for missing components, view the properties and view links.



When an unloaded document points to other documents, the pointed documents are also unloaded. When unloading a part that belongs to several products, the Unload command applies to all the instances of the part.

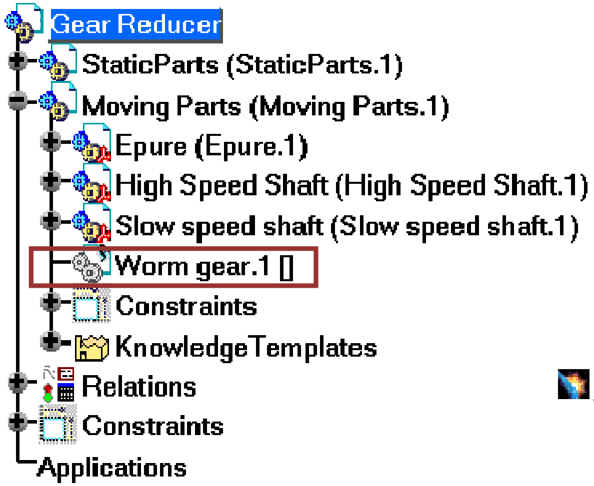
You can also use the Desk File Management toolbar to move, rename and delete documents. Renaming a file does not rename the part in specification tree.

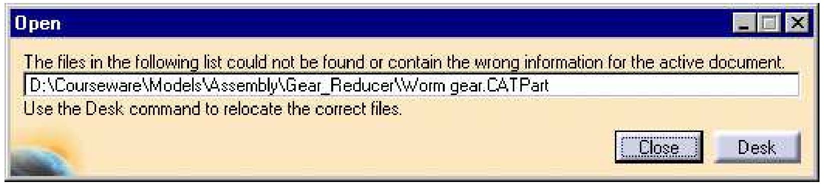
Student Notes:

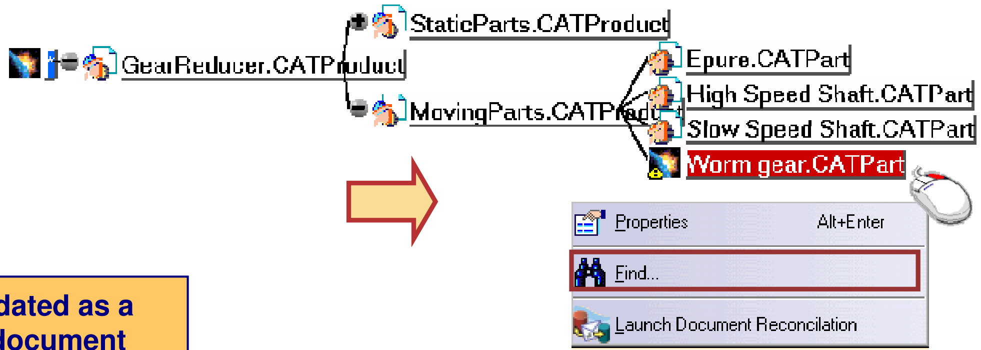
Managing Links of a Product Using File Desk (2/2)

When you open a document of which one or more links are invalid, the Open dialog box appears along with the document opened and a broken link icon appears in the specification tree.

Click on Desk to launch the File Desk command. From contextual menu of Red colored document, click on Find. This command launches a file browser to search and select the missing document. Selecting a missing document restores the broken link.



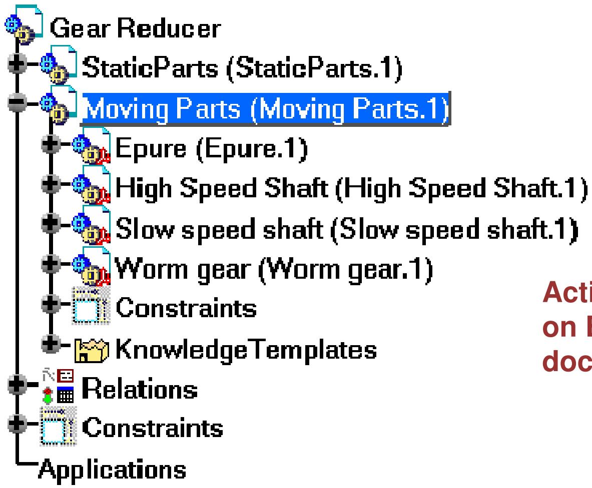




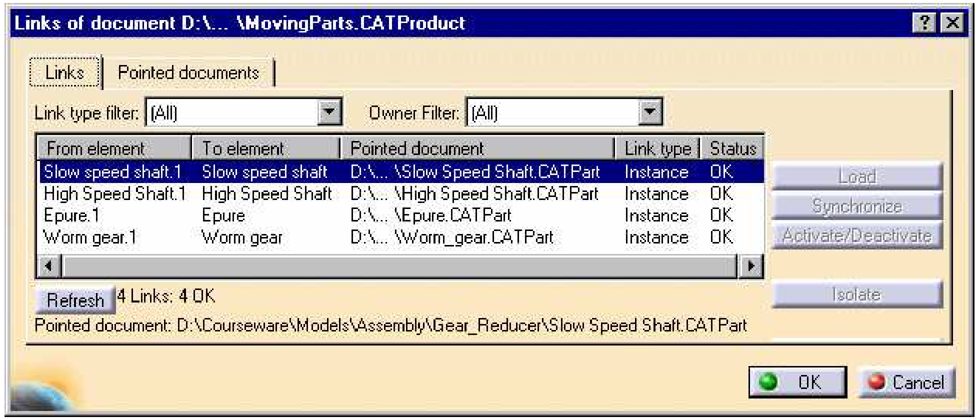
The specification tree will be updated as a result of loading or unloading of document and finding the missing document.

Accessing Edit Links Command

You will see how to access Links command.



Active Document is Moving Parts. Click on Edit > Links, to see the linked documents of this selected component

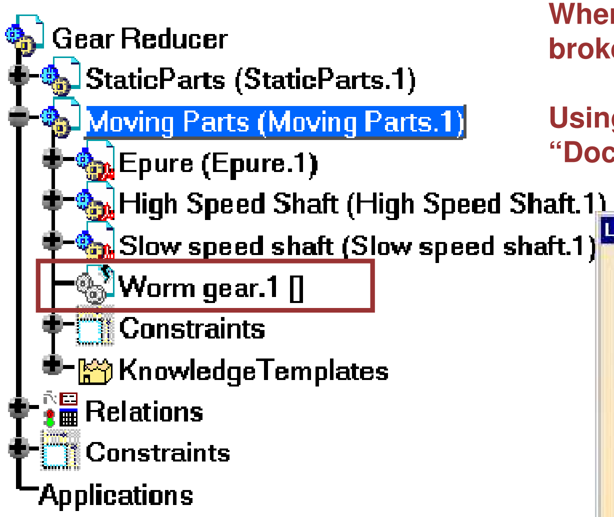


Student Notes:

Student Notes:

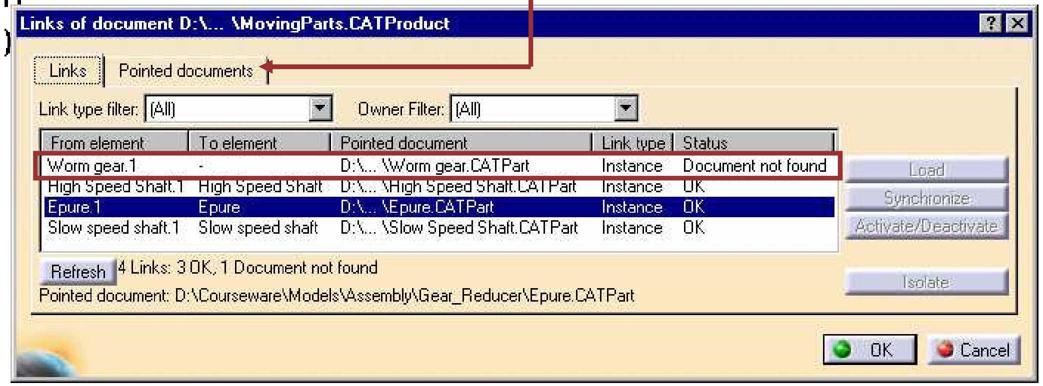
Managing Links of a Product Using Edit Links (1/2)

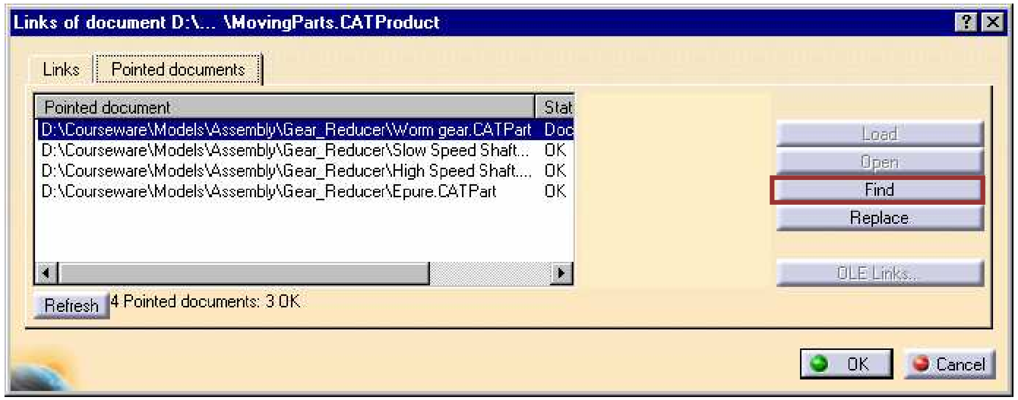
Using Edit Links, you can restore broken links of a selected component.



When you open a document and one or more of its links are invalid, a broken link icon appears in the specification tree.

Using Edit Links, you can see the status for the missing document as "Document not found". Click on Pointed Documents tab.



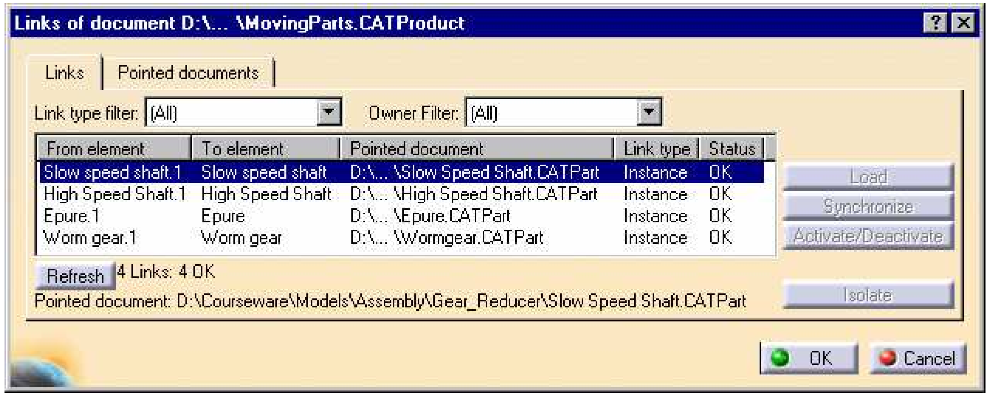


Select the document which is not found, and click on Find button to launch the file browser which will allow you to select the missing document.

Student Notes:

Managing Links of a Product Using Edit Links (2/2)

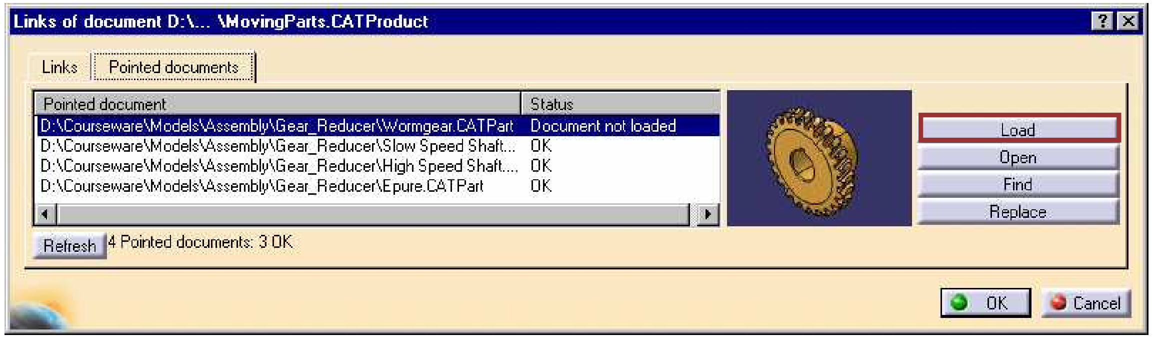
You can Activate / Deactivate a component from Links window. Deactivating a component will suppress the synchronization of linked elements during part update.



You can synchronize the links in the components using Synchronize command.

You can Load the document which is not loaded from the Pointed documents tab, by clicking on the Load button.

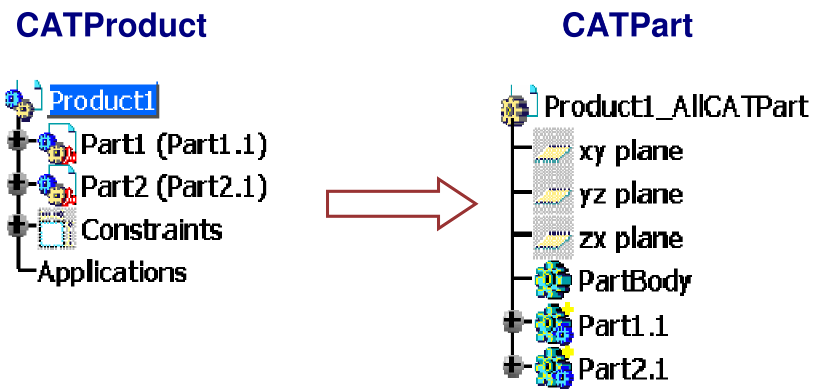
You can also Replace the document with another one.



Student Notes:

Generate CATPart from product

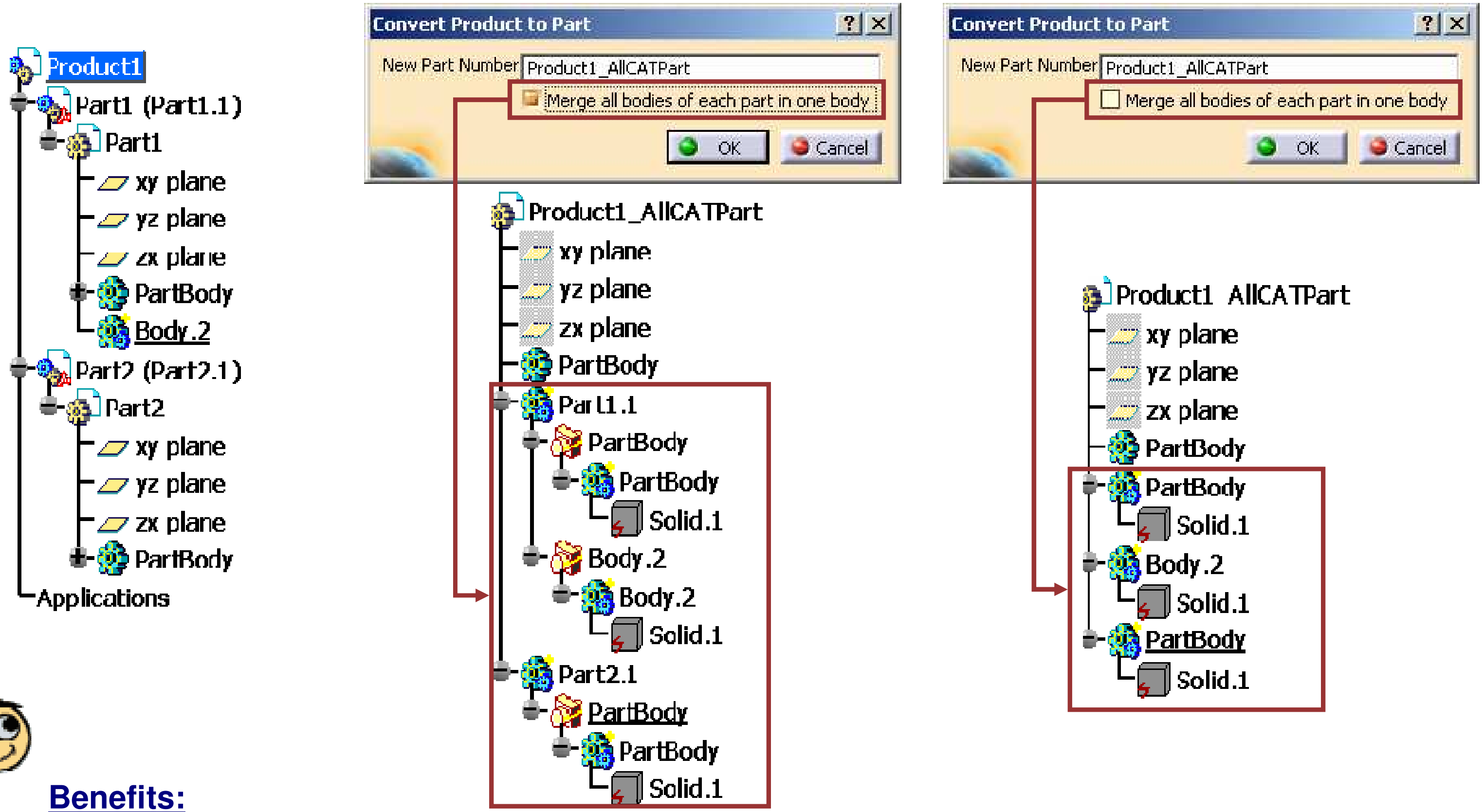
You will see how to generate a CATPart from a CATProduct



Student Notes:

What is Generating a CATPart from a product ?

This functionality creates a non associative result.



Benefits:

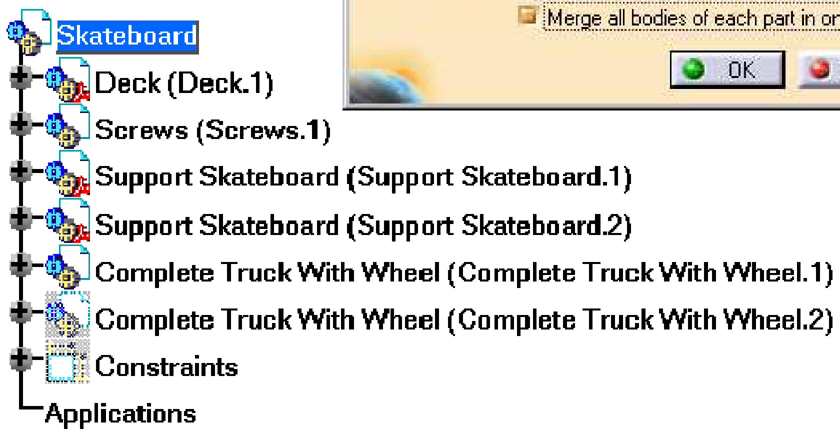
- **Band width requirement reduction**
- **Single part is seen as such in the BOM**
- **IP Protection: no view on internal design**

Generating a CATPart from a product (1/2)

Using "Generate CATPart from Product" menu under Tools menu, you can generate a singe CATPart from a CATProduct.

1 Click on Tools > Generate CATPart from Product menu

2 Select the root product, edit a new part number. Select the Merge all bodies in each Part in one Body option.



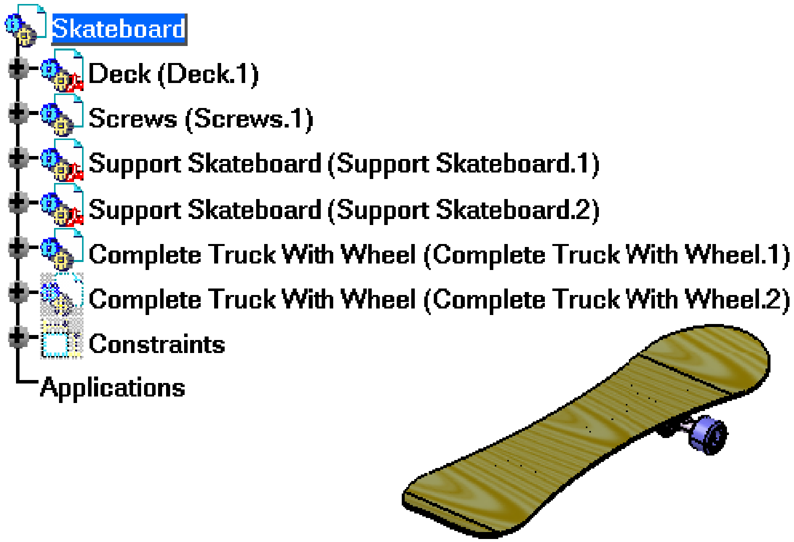
3 Click on OK.

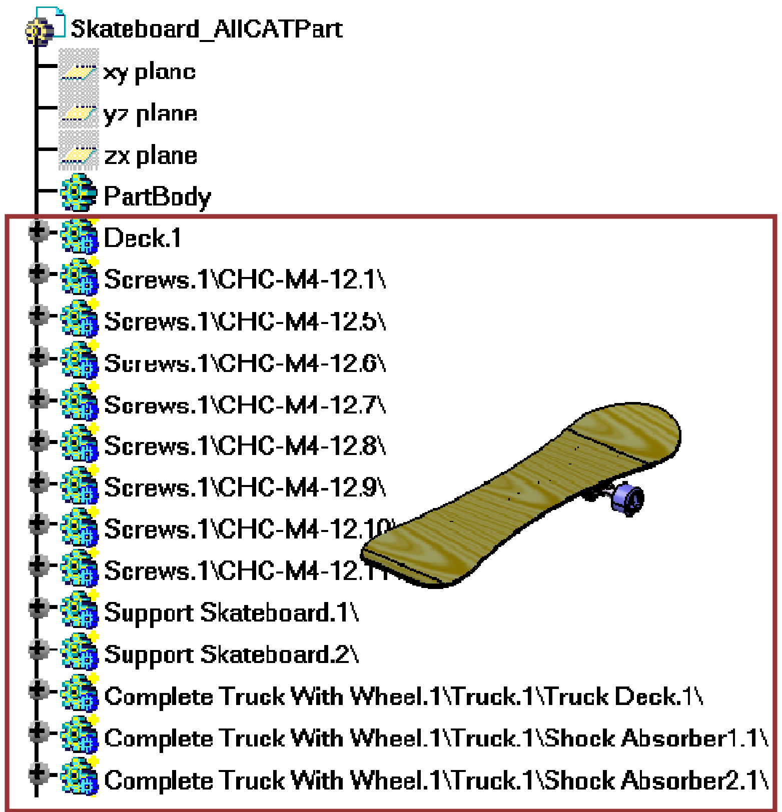
Student Notes:

Student Notes:

Generating a CATPart from a product (2/2)

4 A new CATPart is generated from CATProduct.





Check that the hidden component in the original CATProduct does not appear in the Final Part.

Components are renamed with the path of instance in reference product

Student Notes:

Bicycle Assembly

Recap Exercise: Managing Product Structure

15 min

In this exercise, you will add the missing document in the Bicycle assembly. You will modify the design of one of its components and generate a single CATPart for review and analysis.

In this process, you will use following functions in Assembly Design Workbench:

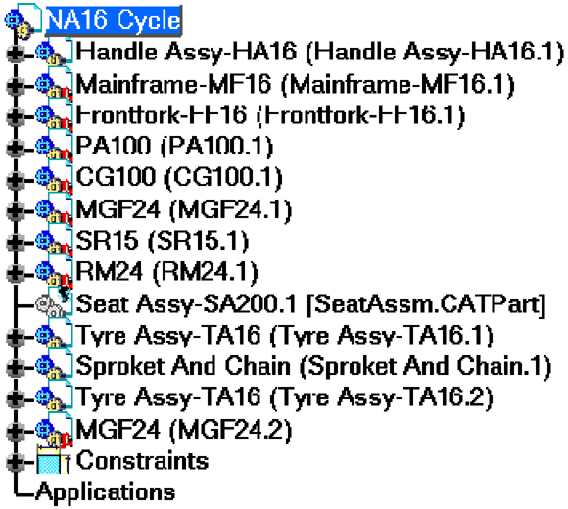
- Desk Command
- Edit > Links
- Generate CATPart from Product

Student Notes:

Do It Yourself (1/13)

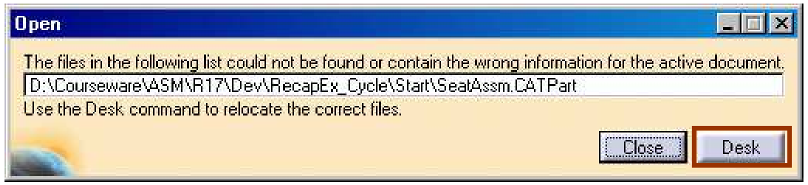
Use “NA16Cycle_Start.CATProduct” for this exercise.

- **Launch the Bicycle assembly in CATIA.**



The Seat of the cycle is missing, you will reroute the broken link to the new Seat component.

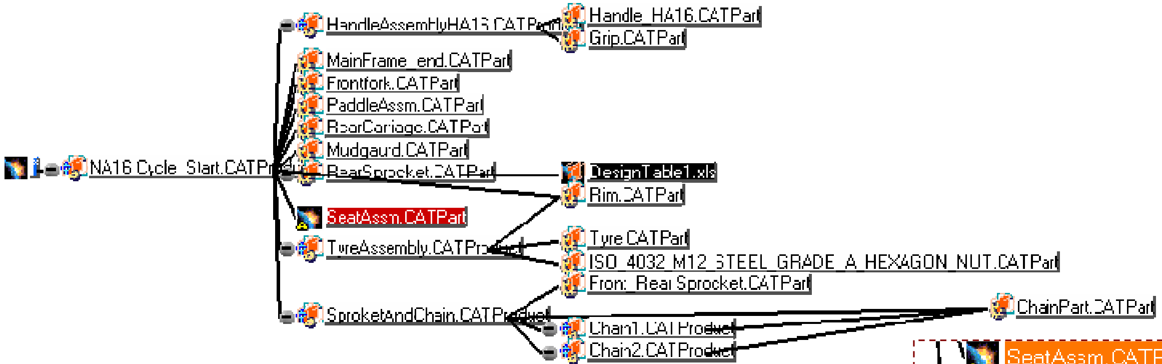
- **Access the Desk command.**

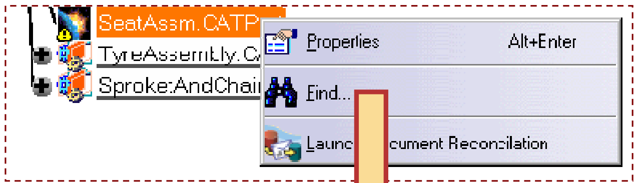


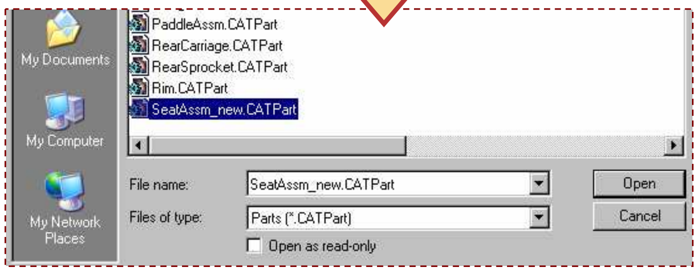
Student Notes:

Do It Yourself (2/13)

- Find the missing document using Desk window:
  - Select "'SeatAssm.CATPart"
  - Link this missing document to "SeatAssm_new.CATPart"
  - Close the Desk Window





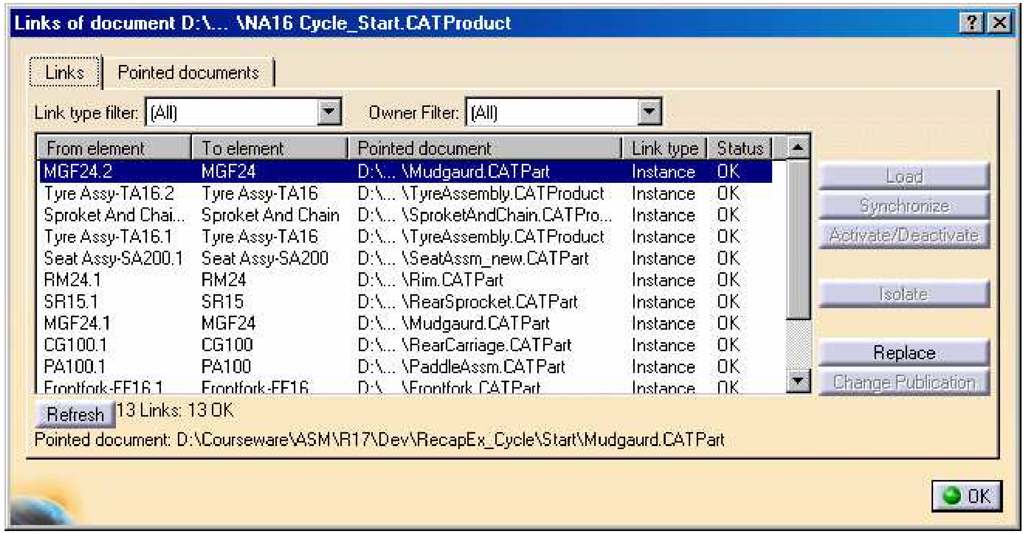


The missing Seat component is now located and linked to the assembly.

Student Notes:

Do It Yourself (3/13)

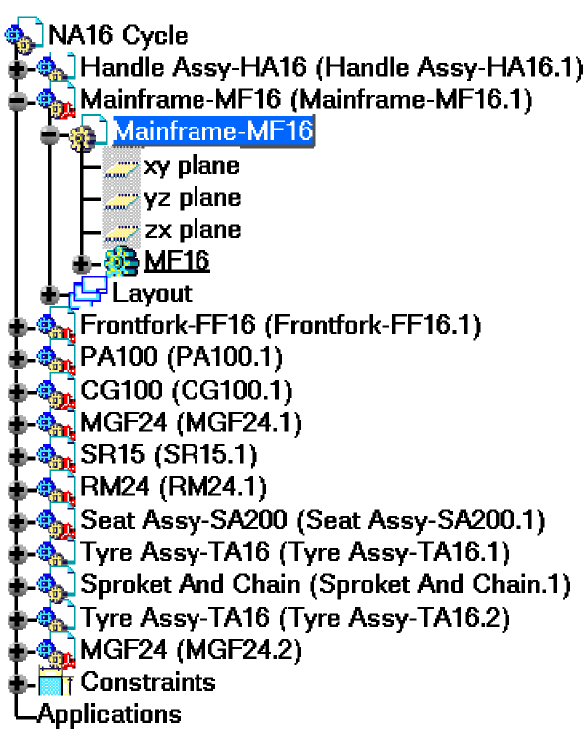
- **Check the links of NA16_Cycle_Start.CATProduct using “Edit > Links” and verify that status of all the linked documents is OK.**



Links panel displays links of selected assembly

Do It Yourself (4/13)

- **A design change is proposed to strengthen the “Mainframe-MF16” part.**
- **Edit the “Mainframe-MF16” part using Part Design Workbench.**



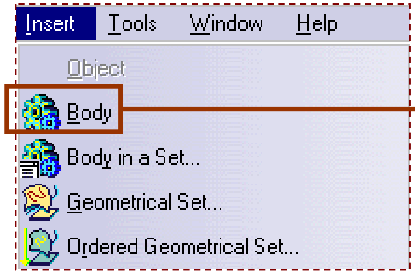
Design changes proposed

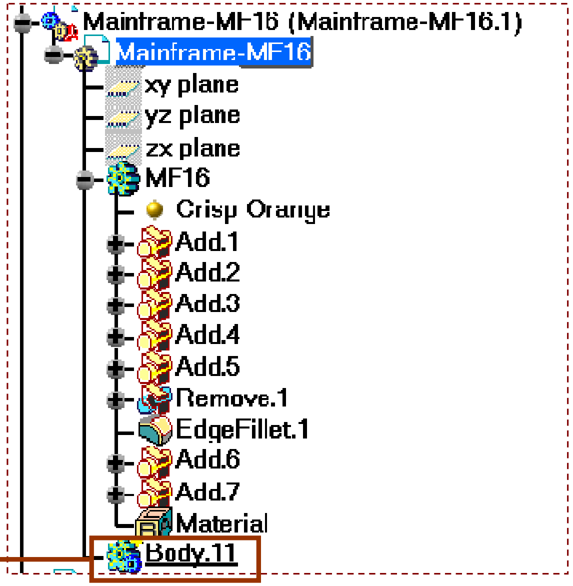
Student Notes:

Student Notes:

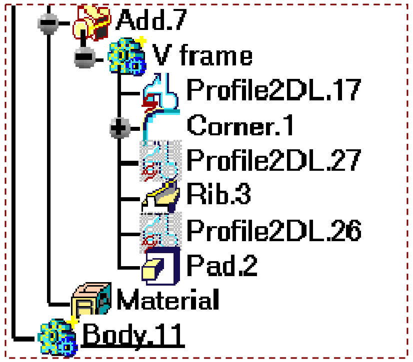
Do It Yourself (5/13)

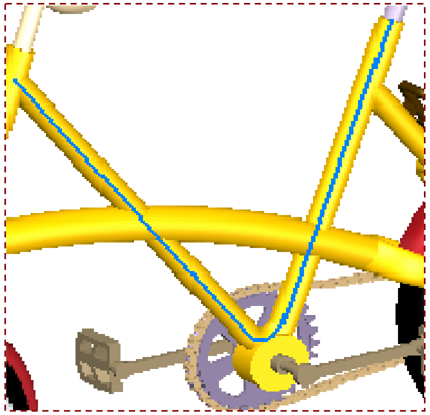
- Insert a new Body in “Mainframe-MF-16”





- Unhide “Profile2DL.17” from ‘V frame’ body





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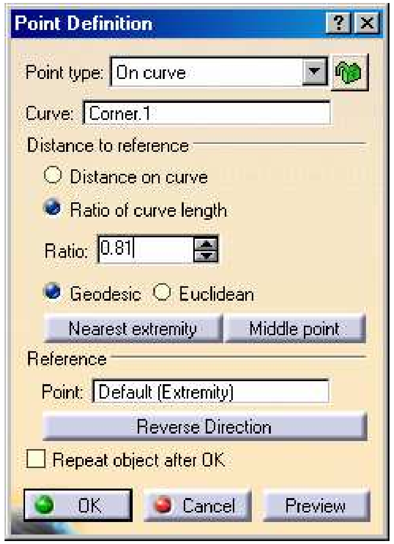
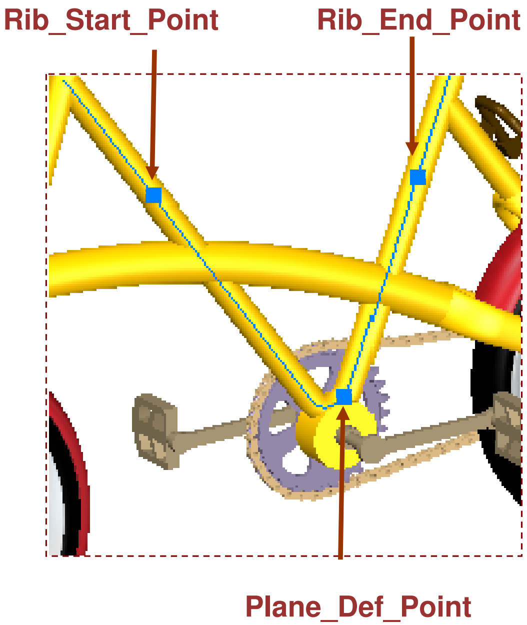
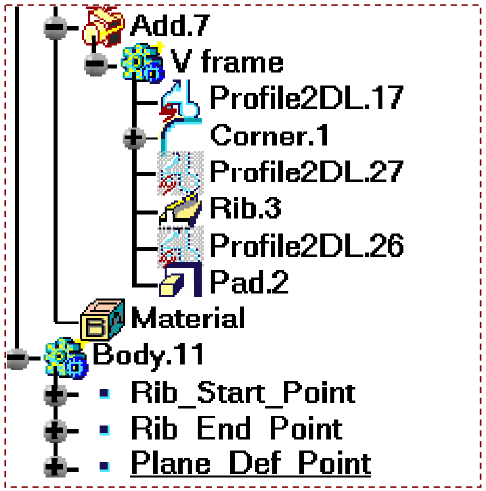
Student Notes:

Do It Yourself (6/13)

- **Create the following three points using ‘Point on Curve’ option and locate points by specifying the values of Ratio:**
  - **‘Rib_Start_Point’ at 0.81**
  - **‘Rib_End_Point’ at 0.18**
  - **‘Plane_Def_Point’ at 0.43**



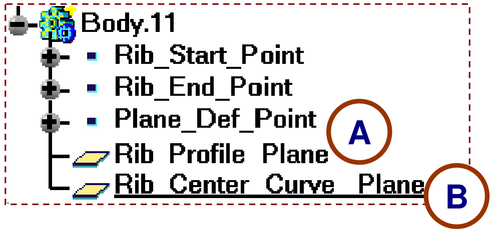


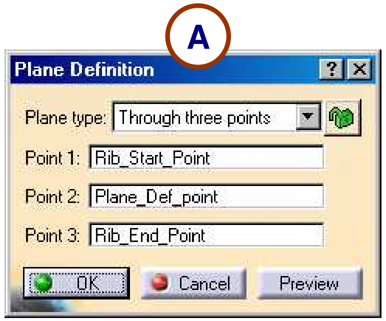


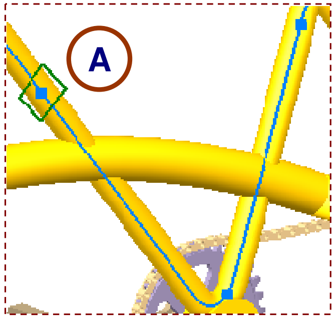
Student Notes:

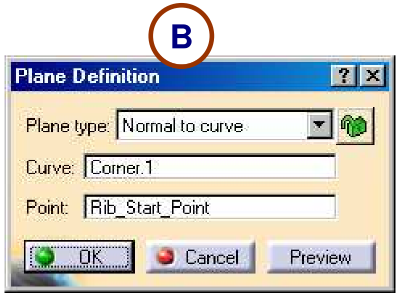
Do It Yourself (7/13)

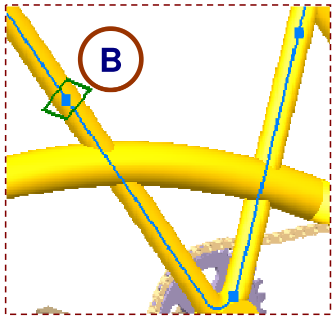
- Create the following planes as shown
  - 'Rib Profile Plane' offset from zx plane of Mainframe-MF16
  - 'Rib_Center_Curve _Plane' passing through the three points created in earlier step







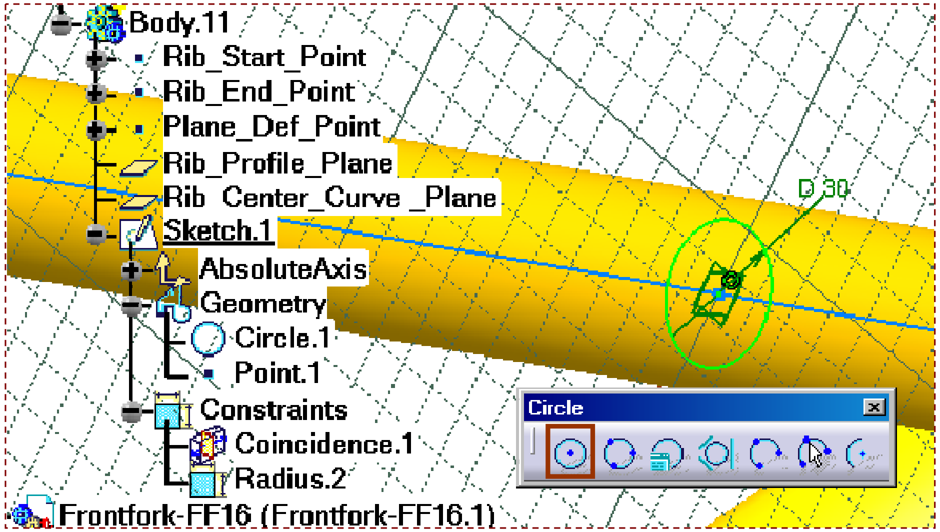




Student Notes:

Do It Yourself (8/13)

- **Sketch a circular profile of the rib with diameter=30mm on 'Rib_Profile_Plane' as sketch support**
  - **Create coincident constraint between the center of the circular profile and Rib_Start_Point.**

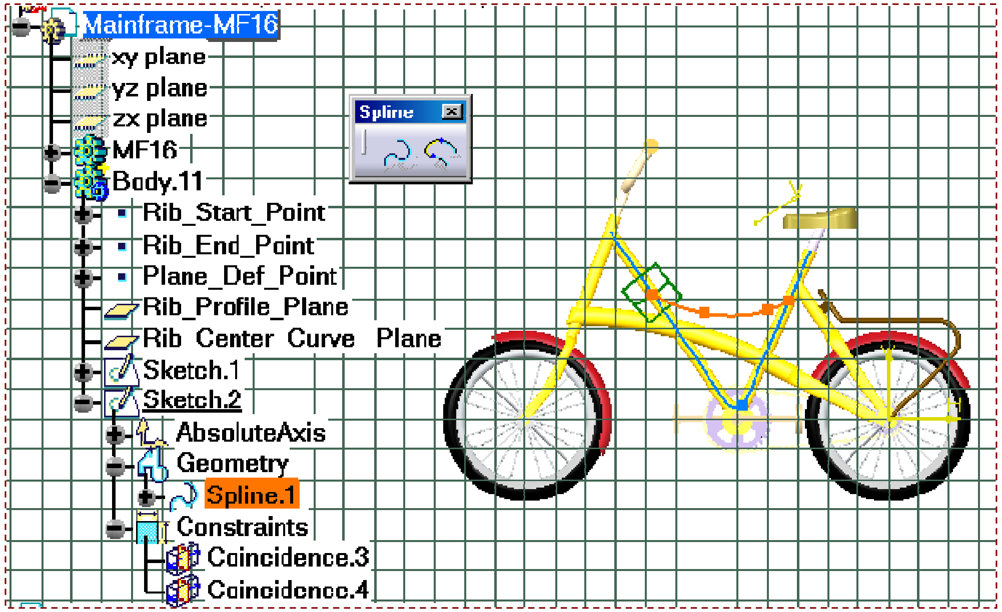


Sketch for Rib profile

Student Notes:

Do It Yourself (9/13)

- Sketch the center curve of the rib on 'Rib_Profile_Plane' using spline command.
  - Create coincident constraints between the starting point of the spline and 'Rib_Start_Point'
  - Create coincident constraints between the end point of the spline and 'Rib_End_Point

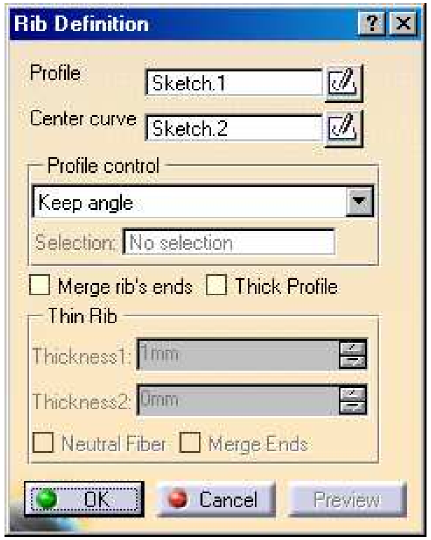


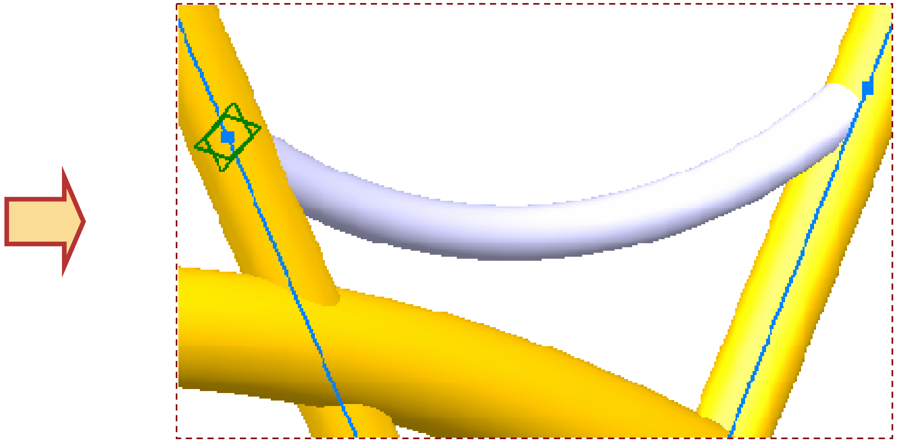
Sketch profile for Center curve

Student Notes:

Do It Yourself (10/13)

- Create a rib using the sketches created above.
  - Use the circular profile as Profile Sketch
  - Use the spline as Center curve



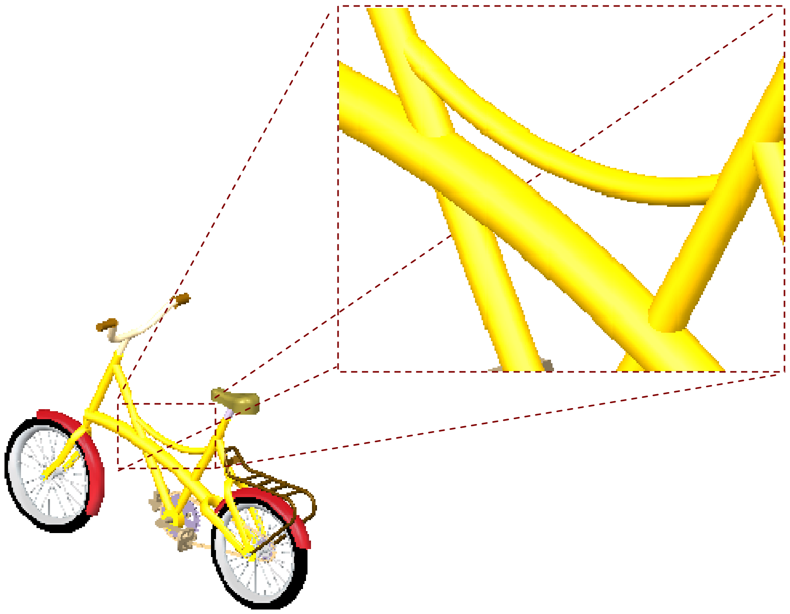


Design rib using sketches created in previous steps

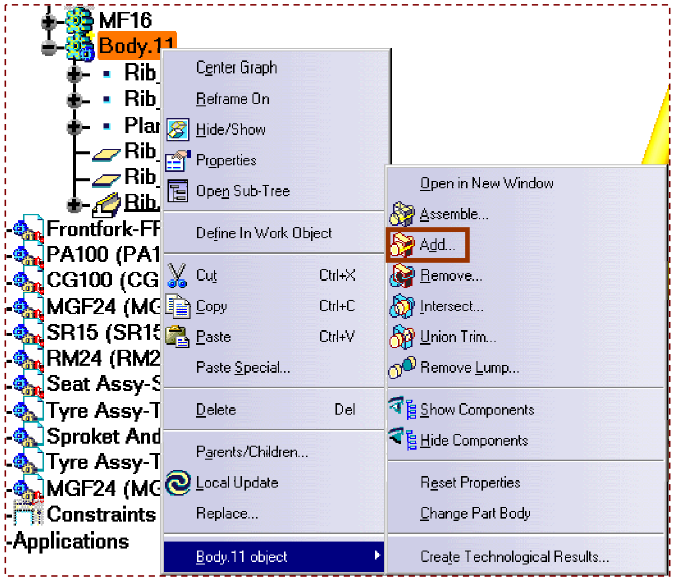
Do It Yourself (11/13)

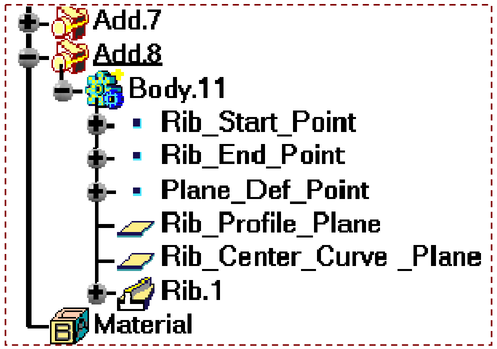
You can compare your result with "NA16Cycle_End.CATProduct".

- Using Contextual Menu, perform a Boolean operation 'Add' to add this new rib profile to the main body of the 'Mainframe-MF16'
- Hide "Profile2DL.17" from 'V frame' body
- Save the root assembly as 'NA16_Cycle_End.CATProduct"



Result of Merging the new body with existing Body



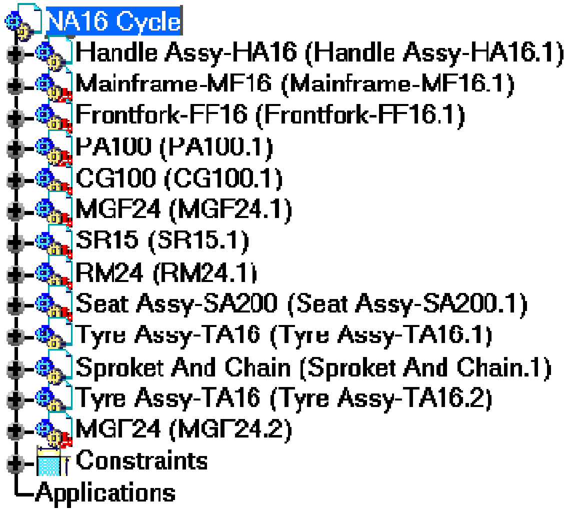


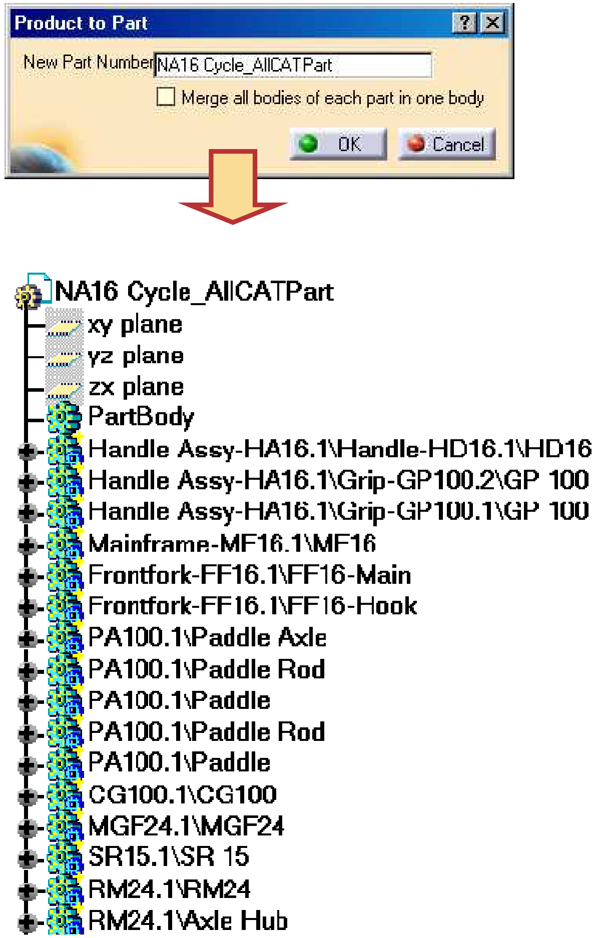
Student Notes:

Student Notes:

Do It Yourself (12/13)

- Create a single CATPart document of the bicycle
- Save the new CATPart.





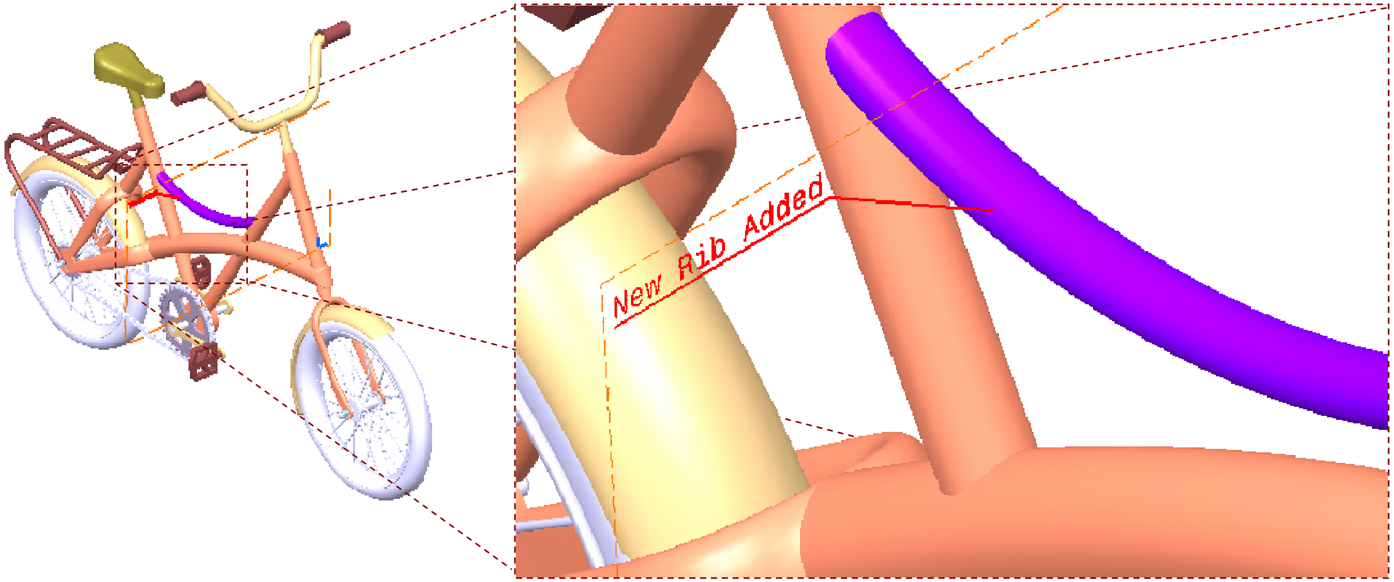
Design intent of the Bicycle is hidden
by generating a CATPart

Student Notes:

Do It Yourself (13/13)

You can compare your result with “NA16CycleAllCATPart.CATPart”.

- Annotate the newly designed rib as shown



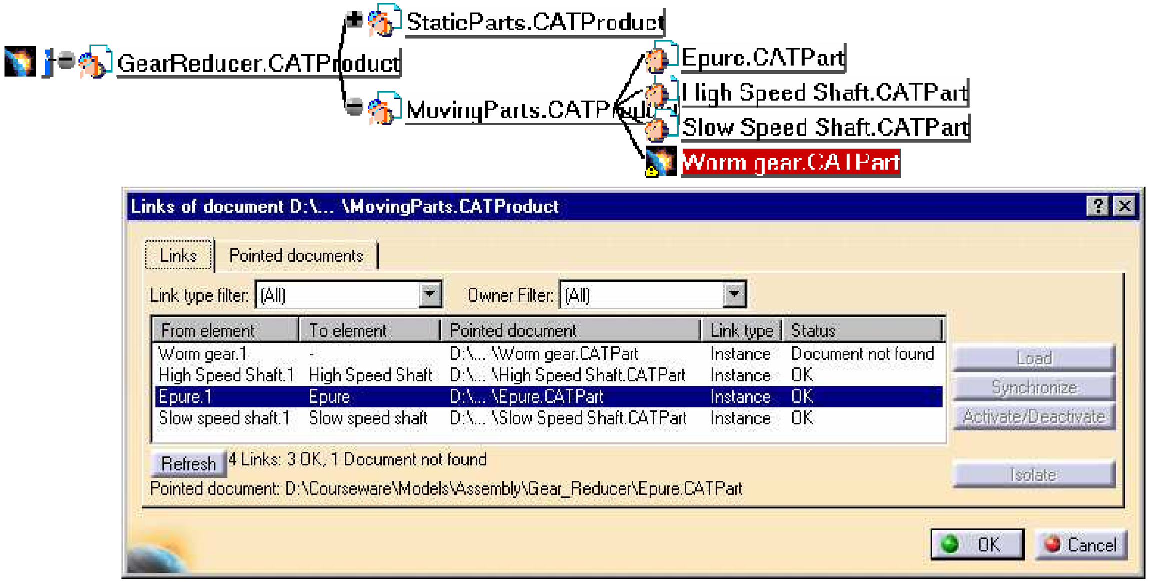
Annotations are added to show the modified design of the frame

Student Notes:

To Sum Up

In this lesson you have learned how to :

- **Manage Links between the components of a Product using File Desk and Edit Links command. You can Activate / Deactivate component , Load/Unload component, Synchronize links, Replace documents, Find missing documents.**



- **Generate a CATPart from a CATProduct. You can suppress all the design details of an assembly document and reduce the document size of the component by converting a CATProduct to CATPart.**

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

Designing and Managing Contextual Parts

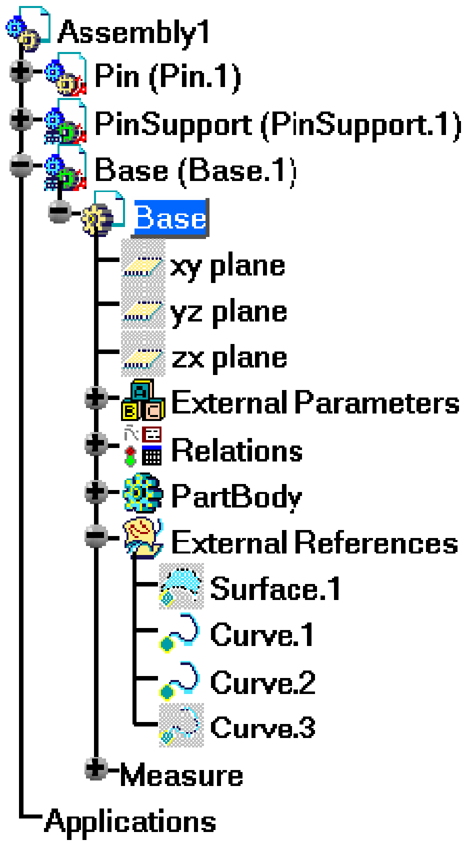
Designing Managing Contextual Parts

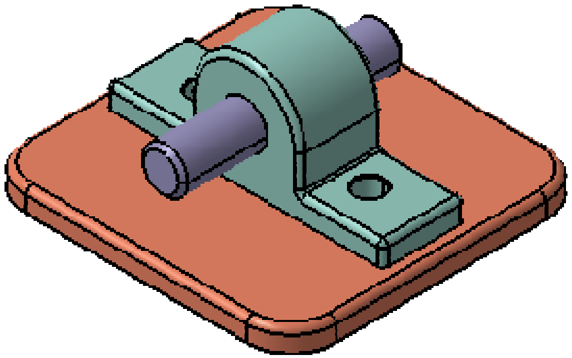
- Introduction to Design in Context
- Creating Contextual Parts
- Sketch and Design in Context
- Knowledgeware and Design in Context
- Editing Contextually-related Parts
- Creating Assembly Features
- Isolating Contextual Parts
- Analyzing Contextual Parts
- Deleting Contextually-related Components
- Saving Contextually-related Documents
- Recap Exercise: Earphone
- To Sum Up

Student Notes:

Introduction to Design in Context

You will learn what is design in context and its usefulness.

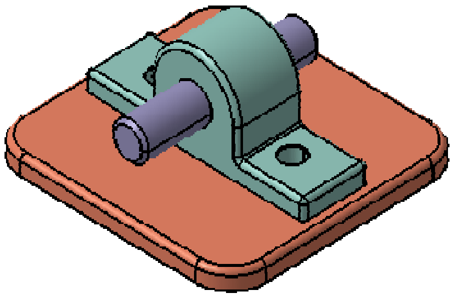




Student Notes:

What is Designing Contextual Parts?

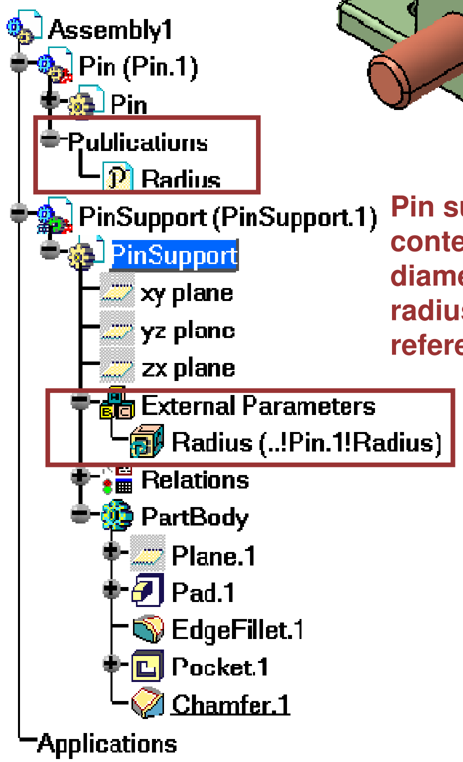
- An assembly is a CATProduct document containing components such as CATParts, CATProducts, V4 Models and Models from external source (IGES, STEP, VRML). The individual parts are positioned relative to each other and are constrained using assembly constraints. Assembly design also allows you to design parts contextually.
- In a CATProduct document you can create associative links between several parts. These links can be geometrical and are then referred to as “External references“ or parametrical and then are referred to as “External parameters” in the specification tree.
- A part is contextually designed if a part is using an external parameter or an external reference element for its definition.

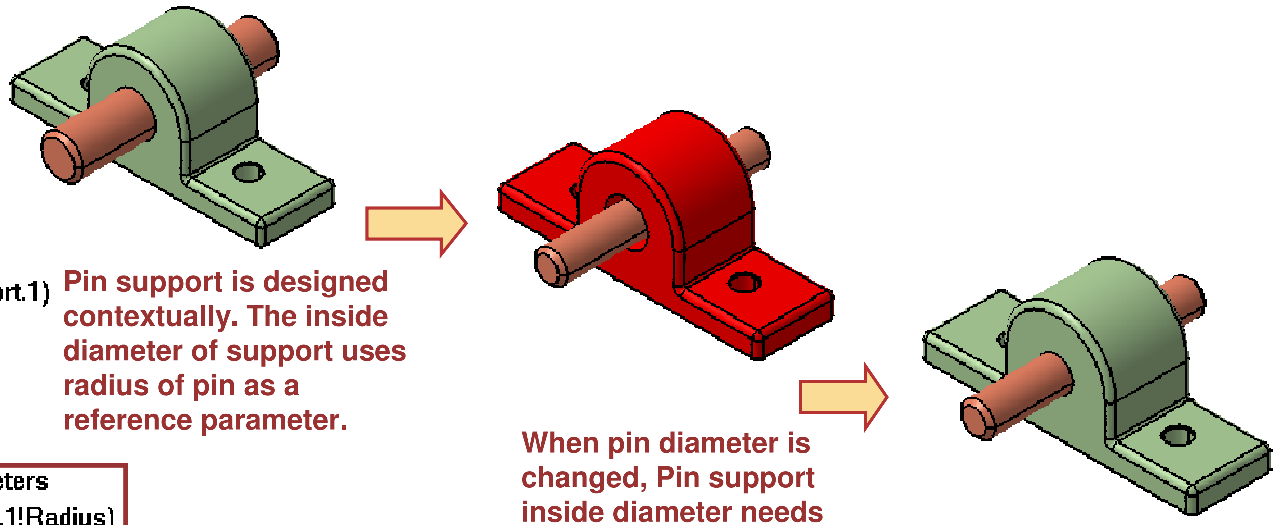


Student Notes:

Contextual Part using External Parameters

When a part refers to the parameters defined in another part, a contextual part using External Parameters is designed.





Pin support is designed contextually. The inside diameter of support uses radius of pin as a reference parameter.

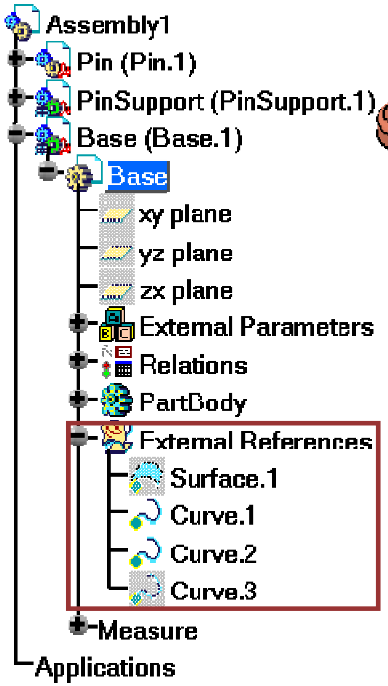
When pin diameter is changed, Pin support inside diameter needs to be updated, hence the part becomes red.

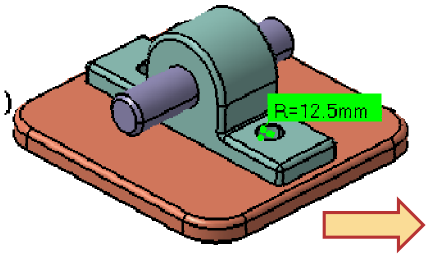
After updating an assembly, pin support inside diameter gets updated.

Student Notes:

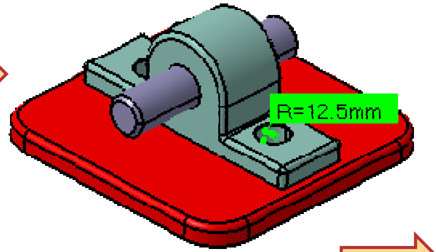
Contextual Part using External References

When a part refers to the geometrical elements in another part, a contextual part using External Reference Elements is designed.

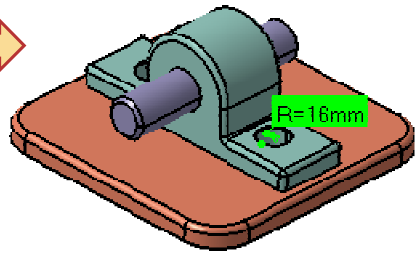




In the Base part, holes defined in the pin support are used to define the sketch for the pad.



If the hole diameter changes, hole diameter of Base needs update, hence the part becomes red.

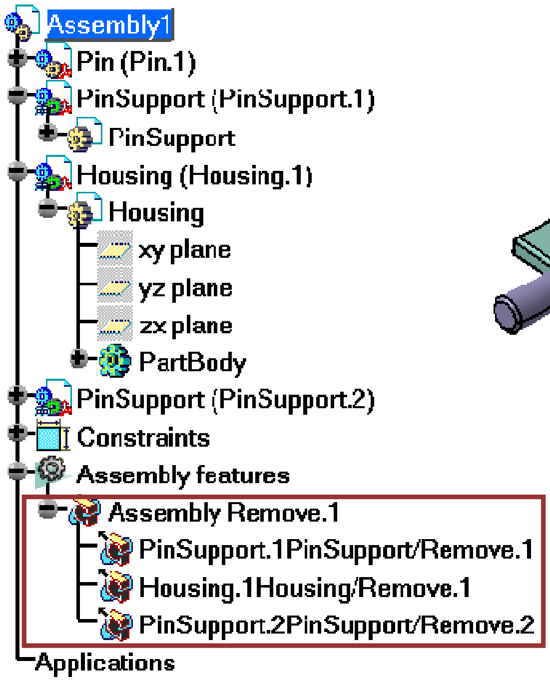


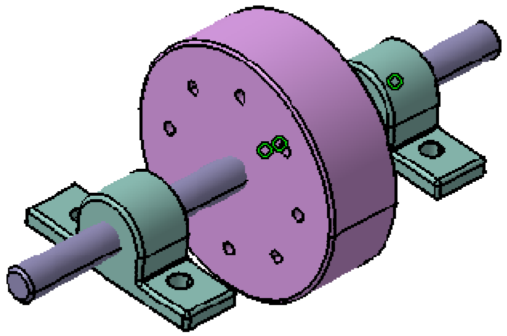
After updating an assembly, hole diameter in base part gets updated.

Student Notes:

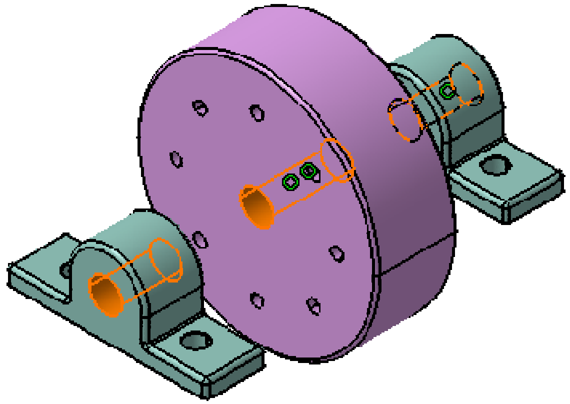
Contextual Parts using Assembly Features

A contextual link is also created when Assembly Remove feature is created using an existing part.





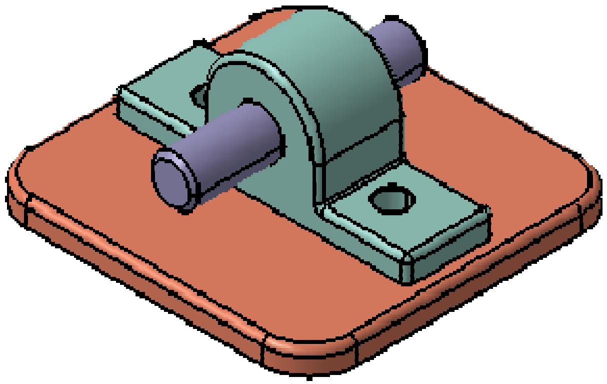
In this example, pin is used to create an Assembly Remove feature in the two Pin Supports and Housing.

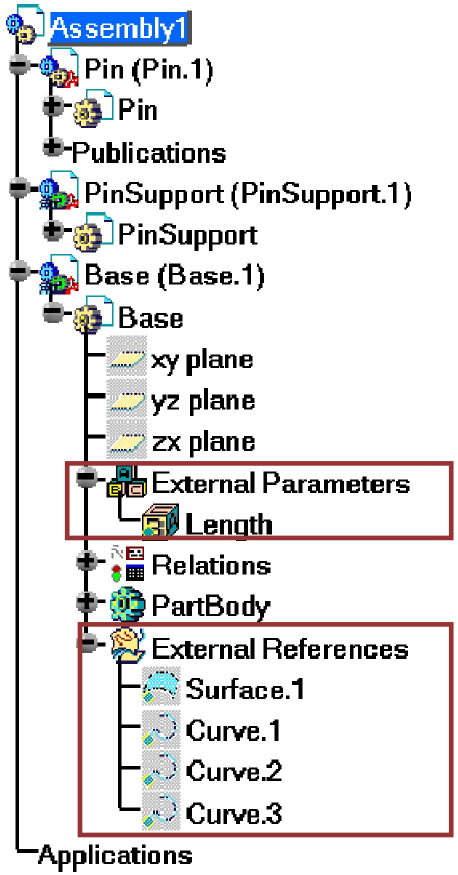


Any changes in Pin like diameter or shape will affect the Hole dimensions in the Housing and Pin Supports.

Creating Contextual Parts

You will learn how to design parts that are contextual, or geometrically driven by other components.



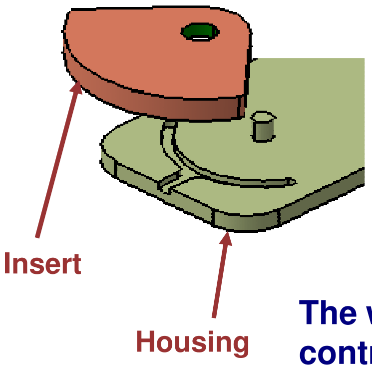


Student Notes:

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

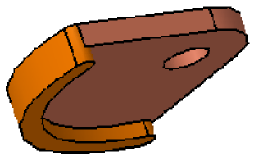
What are Contextual Parts?

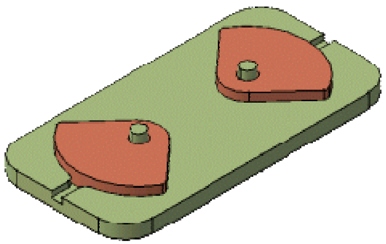
Contextual parts are parts having geometry driven by other components. Changing geometry in another component can automatically cause changes to a contextual part.

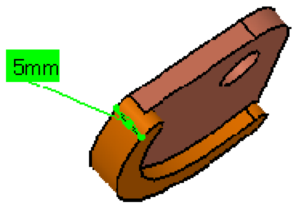


Rounded edge and Hole of insert are <u>contextually</u> concentric with the pin in the green component. The sketch of the rounded edge and hole are explicitly constrained to be concentric with the pin.

The width of the highlighted “rib” is <u>contextually</u> controlled by the edges of the “slot” in the green component. The sketch of the brown “rib” was projected from the edges of the “slot”.

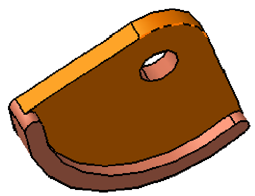






The depth of the “rib” is <u>contextually</u> controlled by the depth of the “slot”. The depth of the “rib” is defined as up-to-plane of the slot bottom.

The bottom face of the insert <u>contextually</u> rests on the top face of the green component. The insert sketch has green face as sketch support.



Student Notes:

Why to use Design in Context?

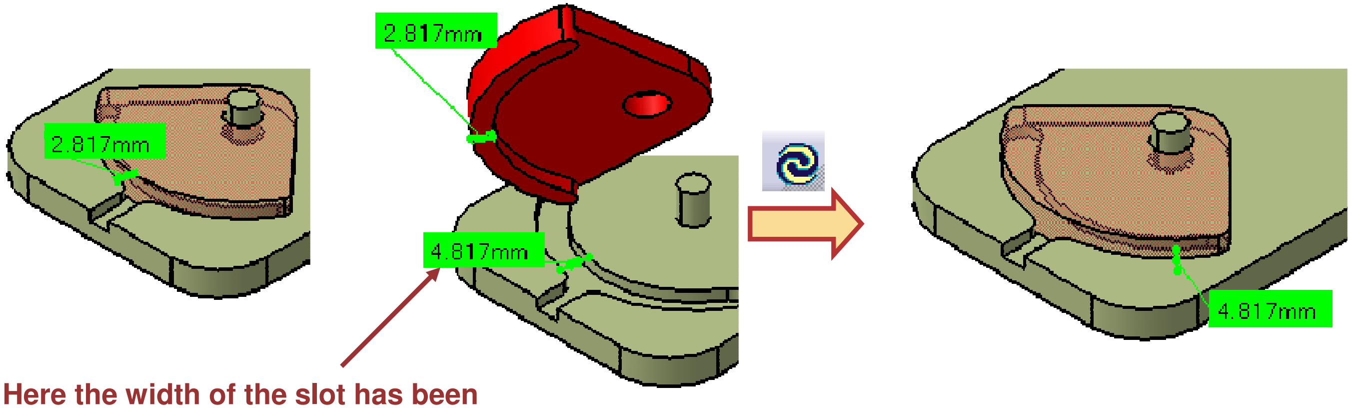
Designing in Context helps Concurrent Engineering Design. It has following benefits:

- **Reuse existing Geometry : In order to facilitate design, you can reuse any geometrical element defined in other part to define a contextual part. For instance, you can reuse an existing sketch in another part instead of recreating it. You can also reuse geometrical entity (point, line, curve, plane or a surface).**
- **Reuse Parameters: You can reuse parameters defined in one part to define a contextual part.**
- **Automatic Update of assembly in case of changes in design : When designing in context, the contextual part is automatically updated when the geometry of the referenced part changes. You don't have to edit the contextual part to change its design.**

Student Notes:

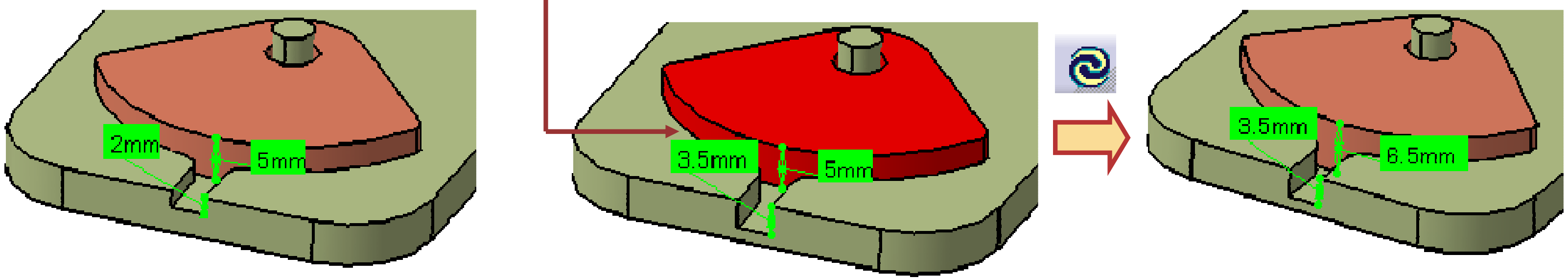
Examples Of Contextual Parts in Action (1/2)

Here are some examples of how contextual parts can be driven by changes to other parts.



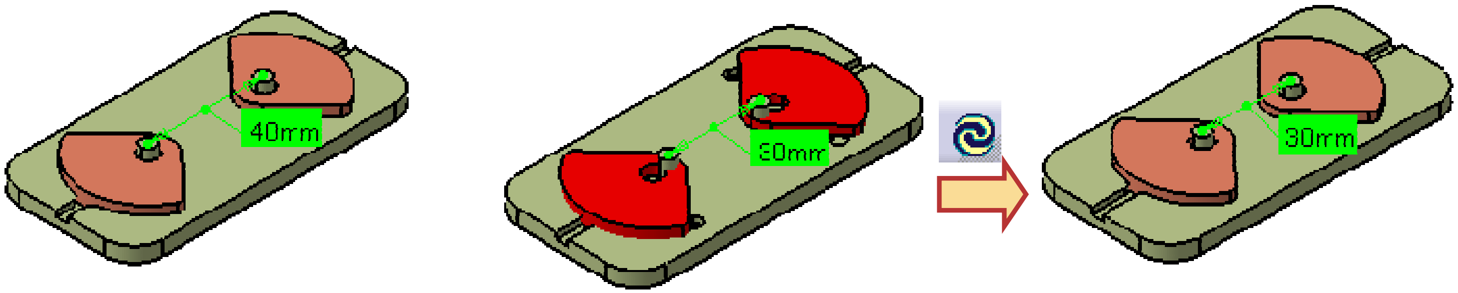
Here the width of the slot has been changed. Notice how the width of the "rib" is driven by the edges of the slot.

Here the depth of the slot has been changed. Notice how the depth of the "rib" is driven by the depth of the slot.



Examples Of Contextual Parts in Action (2/2)

Here are some examples of how contextual parts can be driven by changes to other parts.



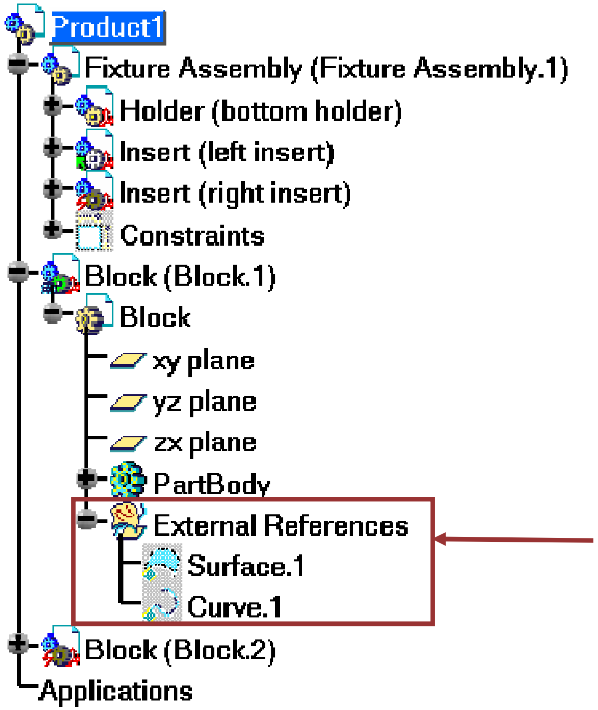
Here the location of the pin has been changed. Notice how the location of the hole is driven by the location of the pin.

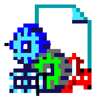
Student Notes:

Student Notes:

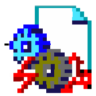
Contextual Parts in the Tree

The tree indicates whether a part is contextual and therefore has External References to other components.

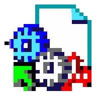




The green gear signifies the "original" instance of a part that is contextual (driven by another part).



The brown gear signifies the second or subsequent instance of a part that is contextual. It identifies a part out of is context.



The white gear signifies the "original" instance of a part defined in context of an intermediate document.

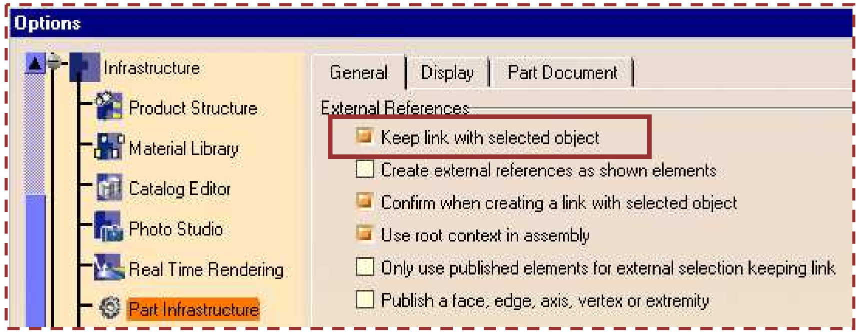
External geometry is copied from driving parts to contextual parts that are being driven. The copies are organized in the External References branch of the part.

Student Notes:

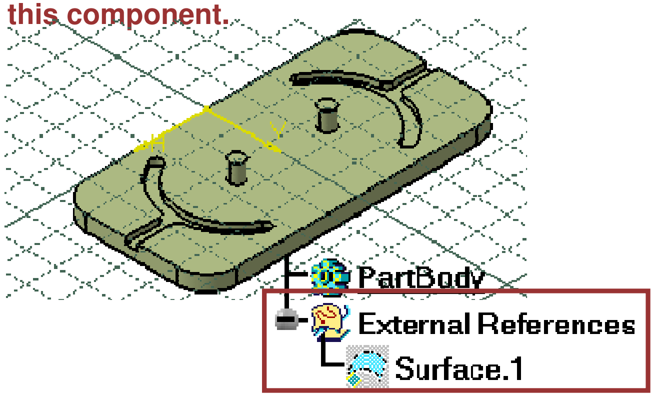
Creating Contextual Elements

Contextual elements can be created while designing sketches and features in-context.

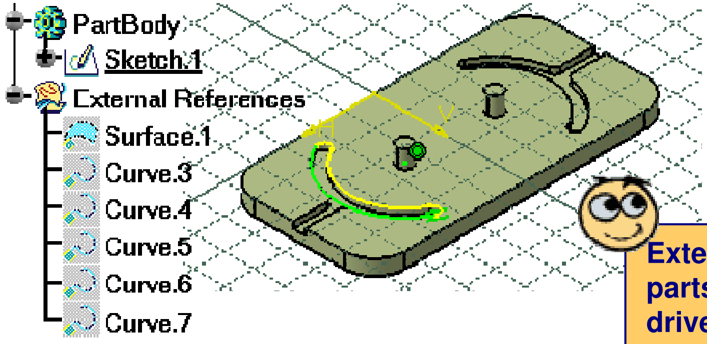
1 Turn ON the option Keep Link with Selected Object



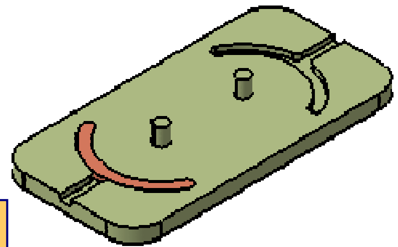
2 Click on Sketcher icon and select the face of green component to link the sketched face with this component.



3 Project edges of the slot onto the sketch plane and complete the sketch. Constrain sketch elements to edges of other components.



4 Create a pad using this sketch and limit it upto the groove depth. This pad is now contextually designed.

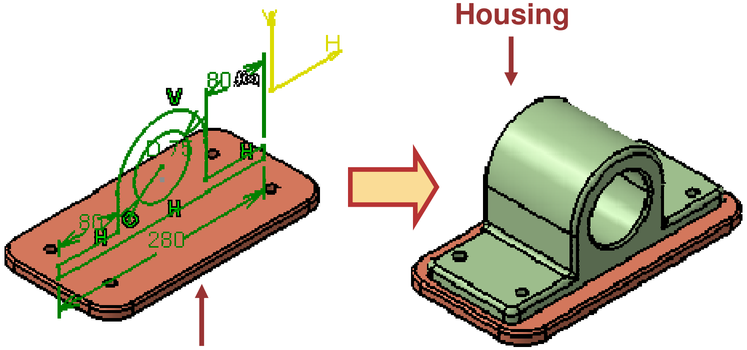


External geometry is copied from driving parts to contextual parts that are being driven. The copies are organized in the External References branch of the part.

Student Notes:

Constraining Contextual Instances Of Parts (1/2)

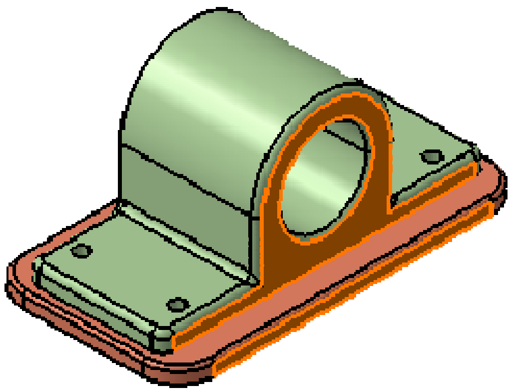
Assembly constraints are forbidden when there is a potential conflict between geometric and assembly constraints. Assembly constraints are always forbidden when any element in a sketch is associative.



Here the Housing component is sketched on a plane defined in Base Plate component. Also the sketch is constrained using the edges of the Base Plate component. The pad's sketch has external links to the Base Plate.

Case 1 : Geometrical Constraints and Assembly Constraints in conflict :

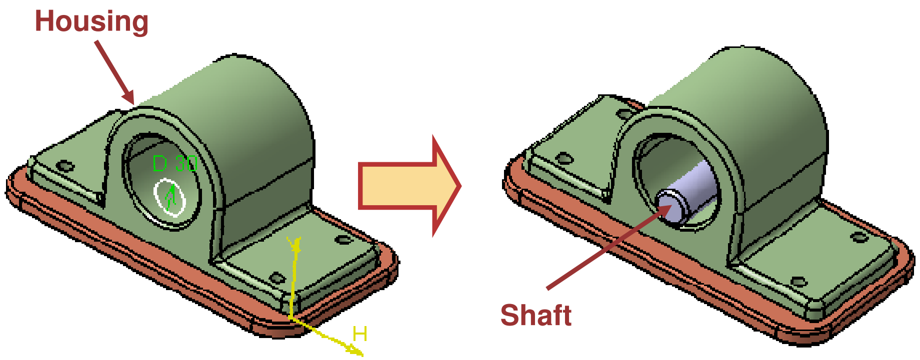
The offset constraint is forbidden because it would cause a potential conflict between the sketch and assembly constraint .



Attempt to define an assembly offset constraint between highlighted faces leads to the following warning message.

Constraining Contextual Instances Of Parts (2/2)

Case 2 : Geometrical Constraints and Assembly Constraints are not in conflict



Here the sketch of the Shaft is designed using face of the housing part. The Shaft has external link to the Housing part.

Here shaft is <u>NOT</u> concentric with respect to the Housing pocket.

An Assembly Coincidence constraint is permitted between the axis of Shaft and the axis of Housing part as there is NO conflict between the geometrical and assembly constraint.

Shaft and Housing are concentric now.

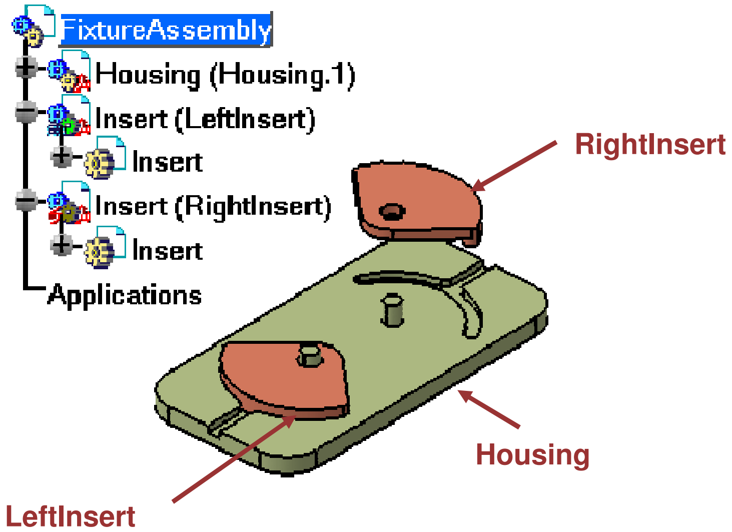
Defining an assembly coincidence constraint

Student Notes:

Student Notes:

Constraining Non-Contextual Instances Of Parts

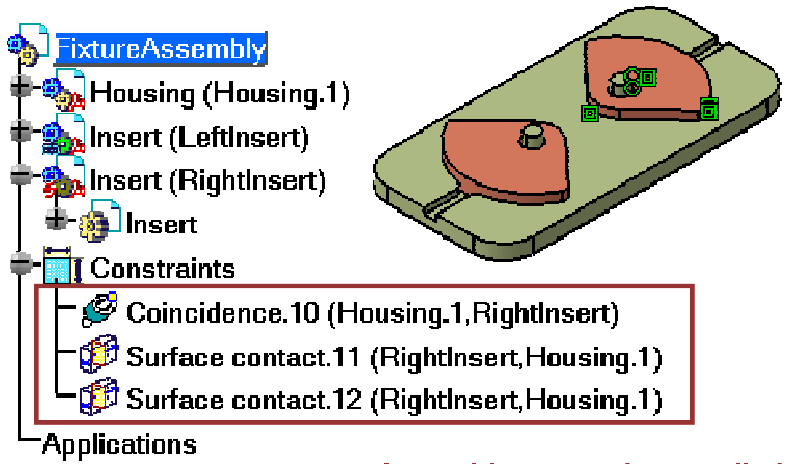
Assembly constraints can be used when there is no conflict between assembly and geometry constraints. Non contextual parts can be constrained using assembly constraints as these parts.



RightInsert is a copy of Insert part. It is a Non Contextual Instance and not designed in context. It can be positioned using assembly constraints because no geometric elements of the part were contextually defined within this instance of the part.

While constraining contextual parts, you cannot use geometrical elements that have external references to other parts as parents (immediate or not).

You can use geometrical elements that have no link with external geometry or parameter. These geometrical elements can be for instance: xy, zx and zy planes, a point build with coordinates, a line defined with an angle from Z axis, etc.

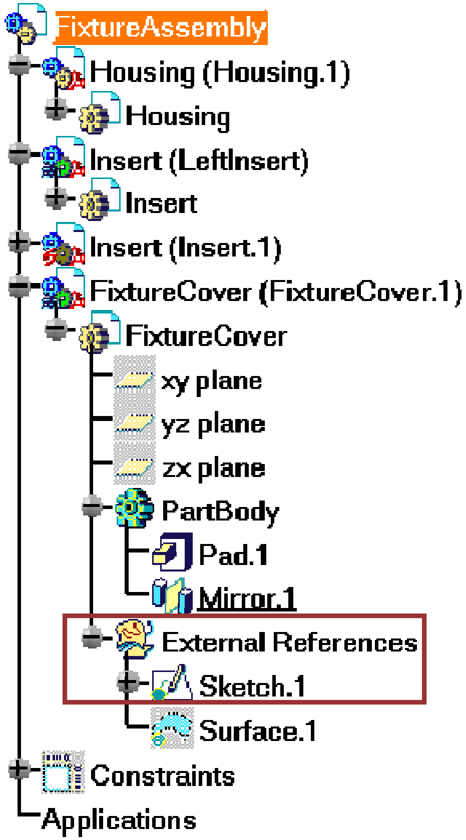


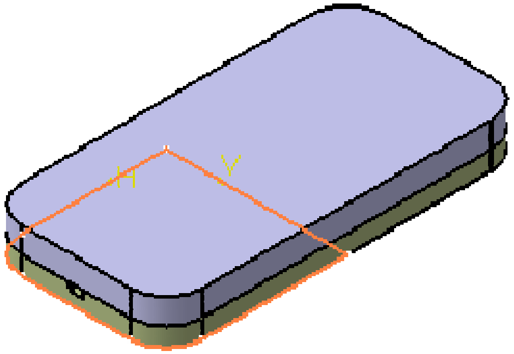
Assembly constraints applied to Non-Contextual Instance

Student Notes:

Sketch and Design in Context

You will learn how to use sketches of other parts in the assembly to design parts in context .



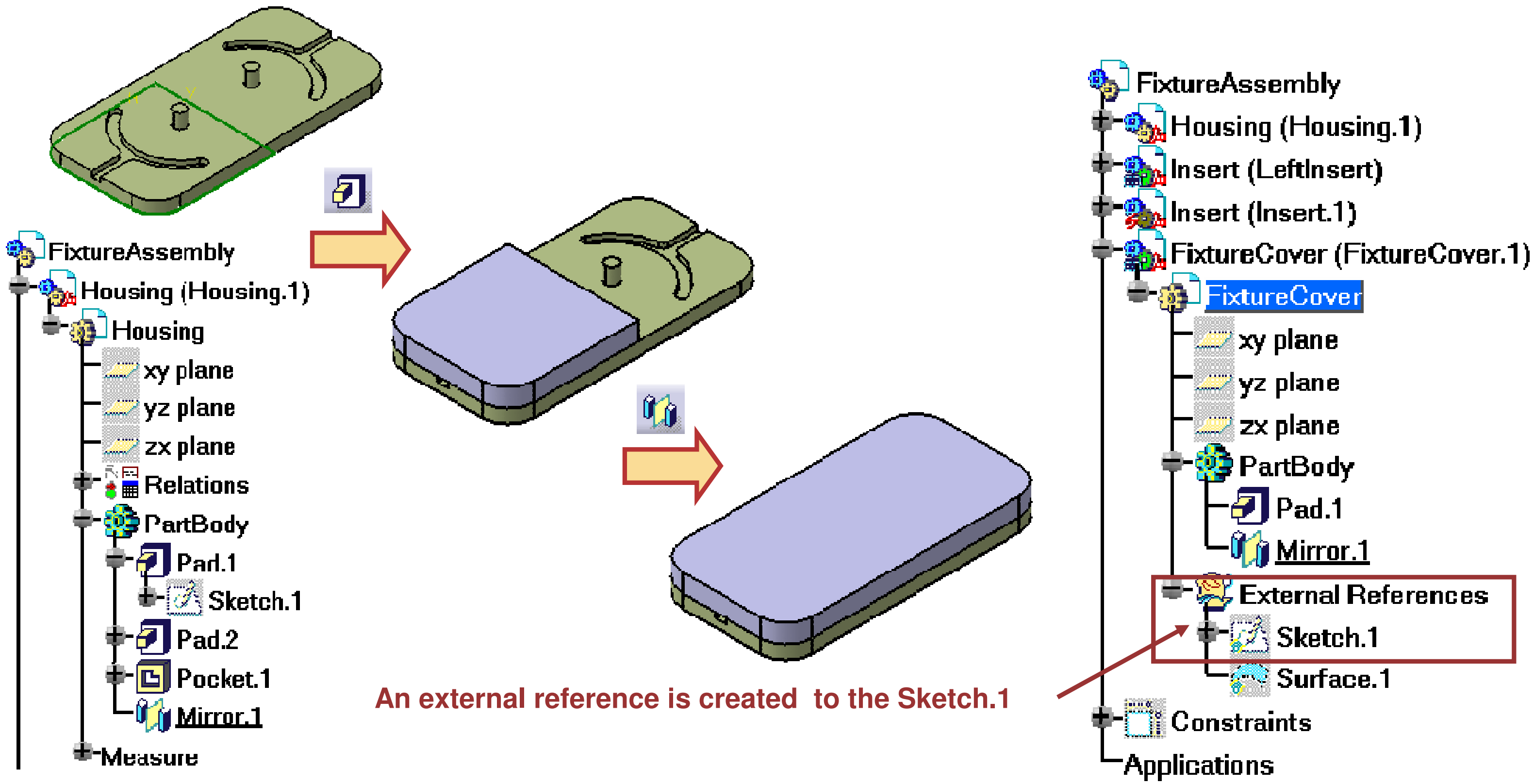


Student Notes:

What is using Sketch in Context ?

It is possible to reuse a sketch created in one part to define another part. It means that the two parts share the same sketch. If sketch in the original instance is modified, the geometry of contextual part is also modified.

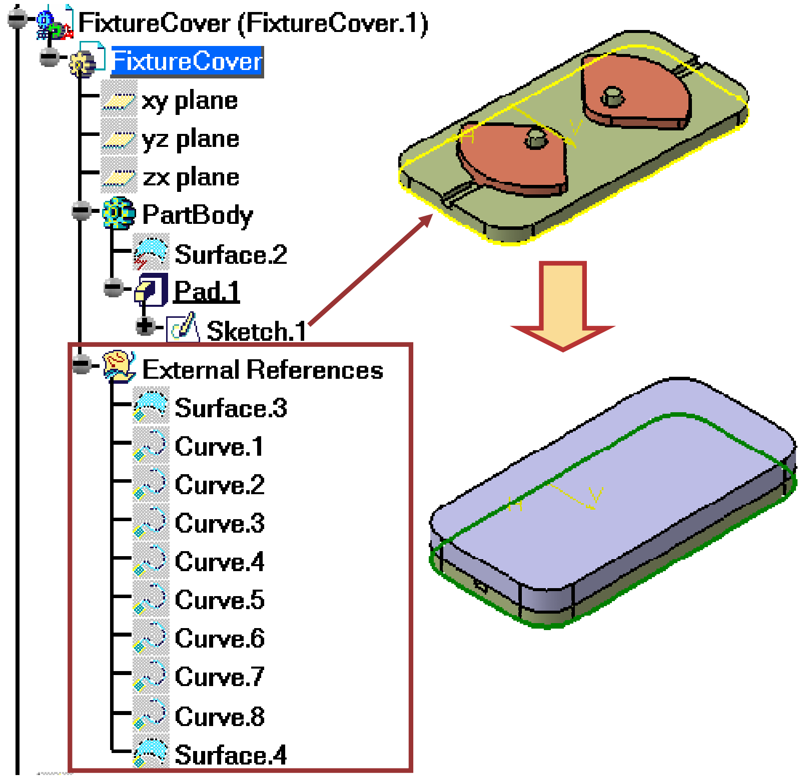
In this example, the pad of the Fixture Cover reuses a sketch.1 of Housing part. Hence Fixture Cover is contextually linked to Housing part.

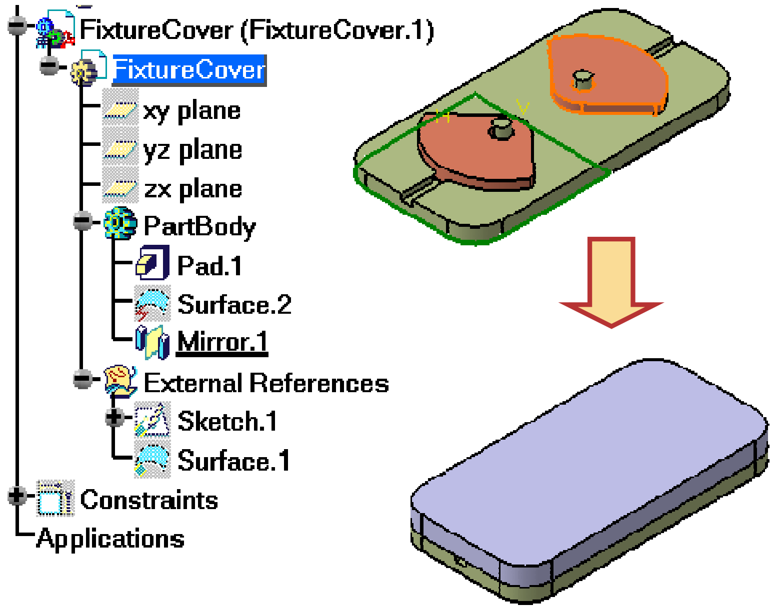


Student Notes:

Why Using Sketch in Context ?

It can be much more useful to select same sketch to define two different parts than using projections of edges of one part to define the other .





In the first case, edges are projected from green part into the sketch of the other. A lot of external references are created which need to synchronized everytime when sketch changes.

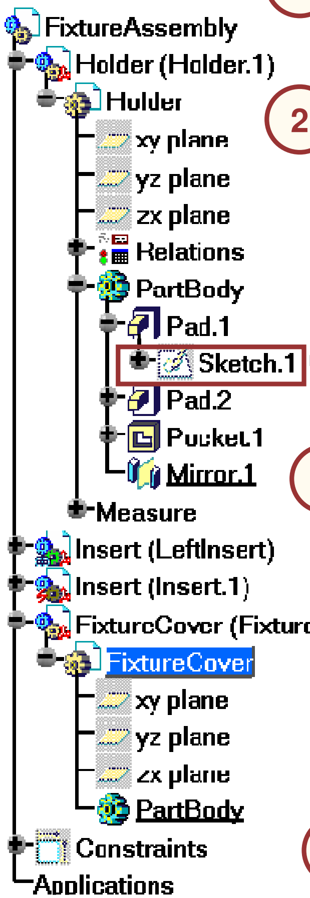
In the second case, the sketch of Housing part is directly used to create the pad of the Fixture Cover.Now there is only one external reference to synchronize. Hence updation is faster.

Student Notes:

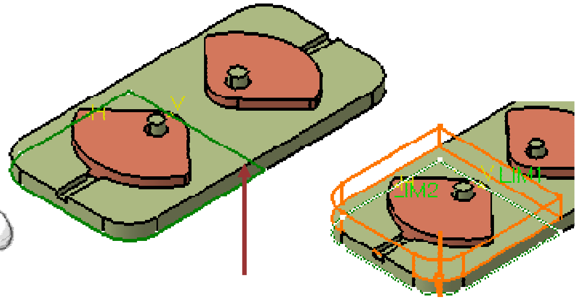
Using a Sketch As an External Reference (1/2)

Simply use the sketch of another part to design the new part and take care to keep link with selected object.

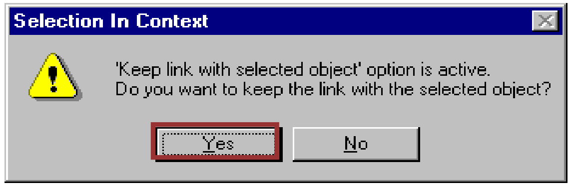
1 Edit the Part in which you want to create a pad or another Sketch Based feature.



2 Click on Pad icon. 

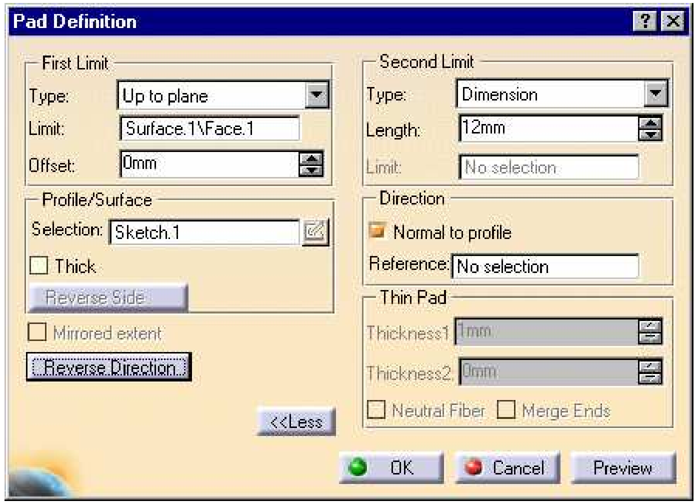


3 Select Sketch in another part as profile

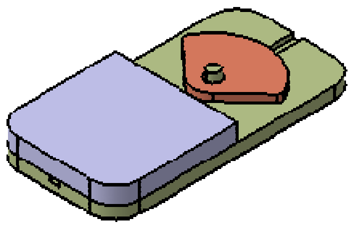


4 Click on "Yes" to confirm your choice.

5 Define the pad first limit as top surface and second limit as shown. Click on Reverse direction.



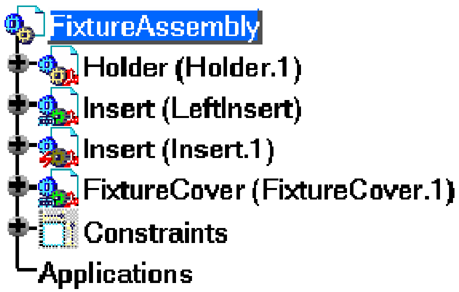
6 Click on OK.

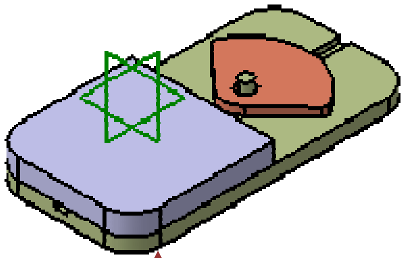


Student Notes:

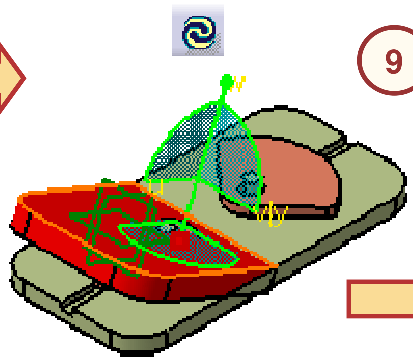
Using a Sketch As an External Reference (2/2)

7 Activate the root assembly and try to move the newly created component

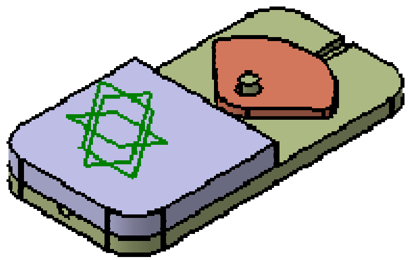




8 Click on Update.



9 You will see that position of the component relative to the original sketch impacts its geometry



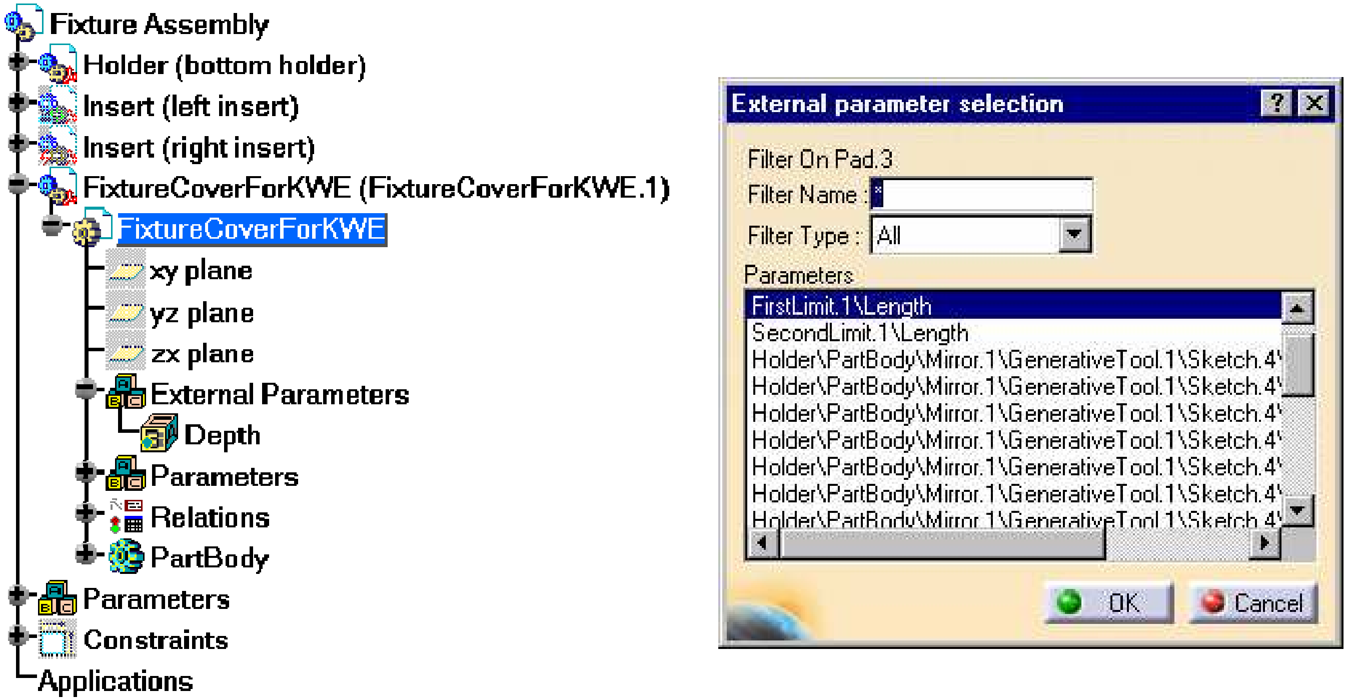
Reference planes of “FixtureCover”

Relative positions of Pad (linked to the external reference“Sketch1”) and reference planes of the part have changed. “Sketch1” remain an exact copy of “MasterSketch”.

Student Notes:

Knowledgeware and Design in Context

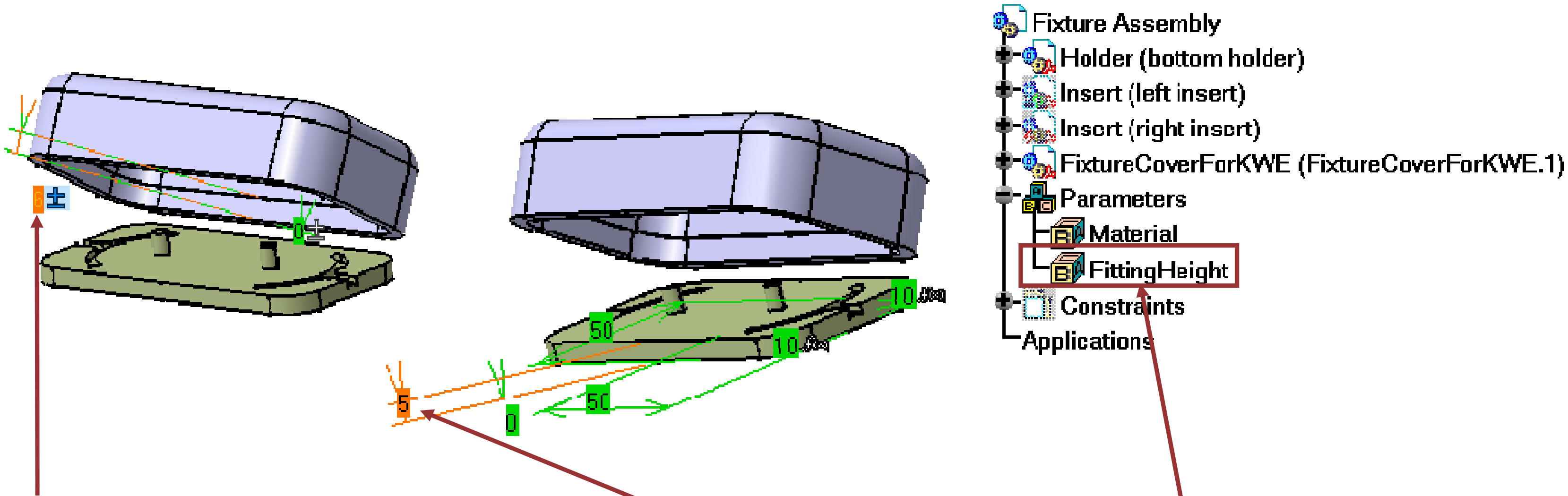
You will learn how to use parameters of other parts in the assembly to design parts in context .



Student Notes:

Why to use Parameters in Context ?

During assembly design, you can have parameters of a part driven by parameters of another part of the assembly or by parameters of the assembly itself .



In this case, we would like this parameter concerning FixtureCoverForKWE component to be equal to …

…this other parameter concerning Holder component which himself could be equal to…

…this user parameter of the root assembly

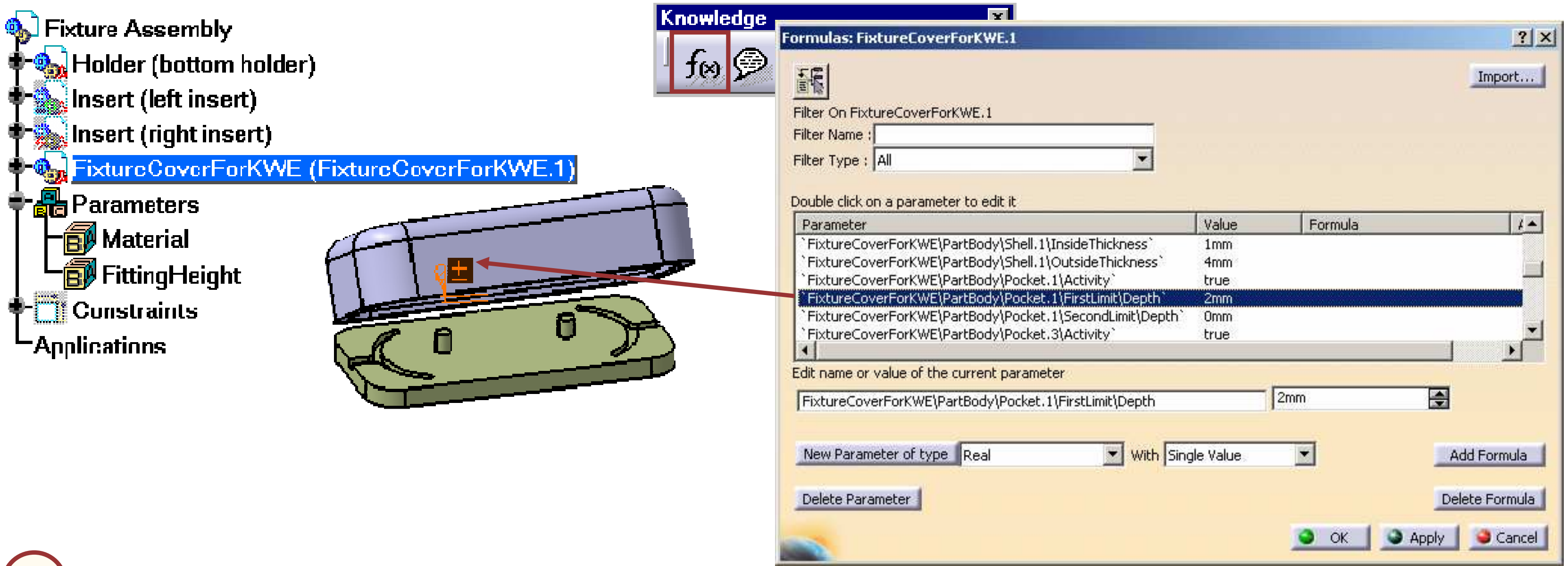
Student Notes:

Linking Parameters Of Two Parts in the Assembly (1/3)

Creating a relation involving parameter of another part is possible, a linked copy of this parameter will be created under External Parameter node.

1 Edit the part on which you want to create a relation.

2 Select Formula icon . Formula editor panel is opened. Select the parameter you want to drive.



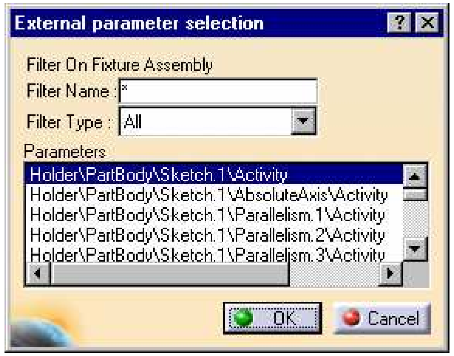
3 Click on "Add formula".

4 Editing the formula, First select the other part in geometry so CATIA will know that you want to select a parameter outside the active part.

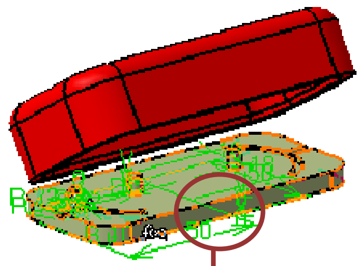
Student Notes:

Linking Parameters Of Two Parts in the Assembly (2/3)

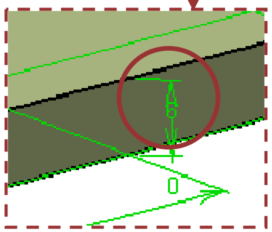
5 The External parameter selection box has appeared.



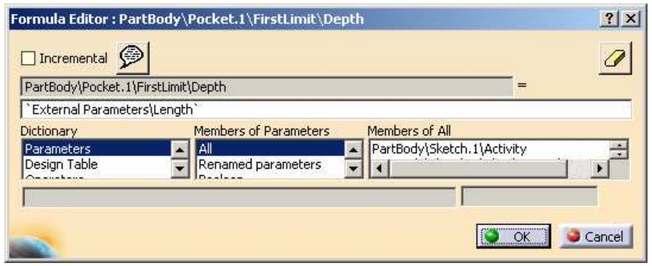
6 Click on a feature of the selected part to focus parameters filter on it and make the parameters appear in 3D.



7 Select driving parameter.



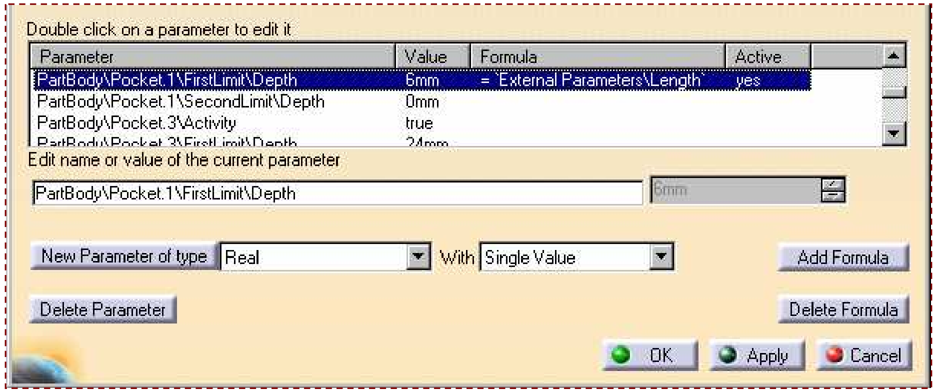
8 The parameter is selected in the dialog box. Click OK to confirm.



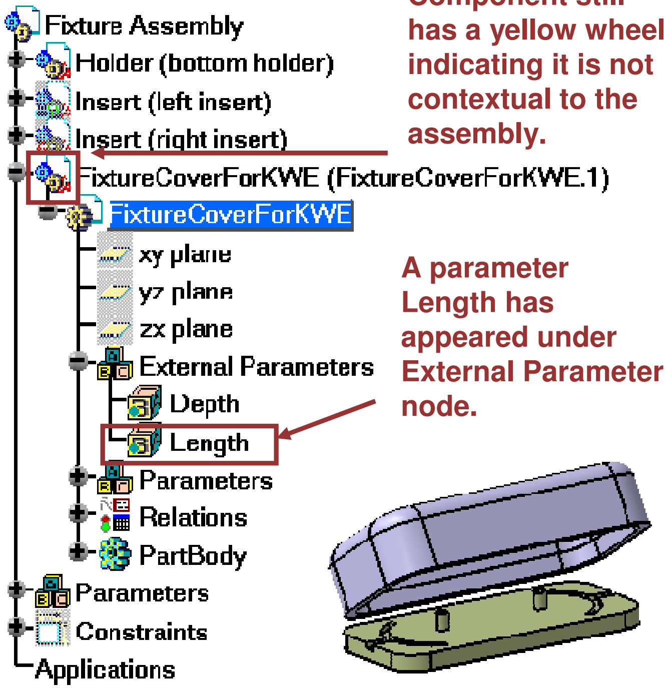
Student Notes:

Linking Parameters Of Two Parts in the Assembly (3/3)

9 **Click OK to confirm Parameter edition.**



10 **Here is the result**



Component still has a yellow wheel indicating it is not contextual to the assembly.

A parameter Length has appeared under External Parameter node.

11 **Parent and Children box of the External Parameter «Length» displays the link to the parameter and indicates its owner document.**

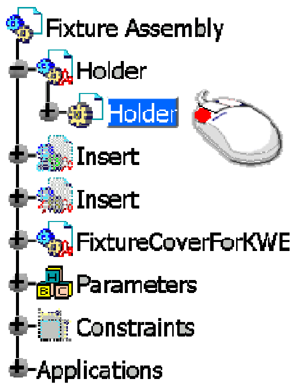
Created formula is under Relations node.

Student Notes:

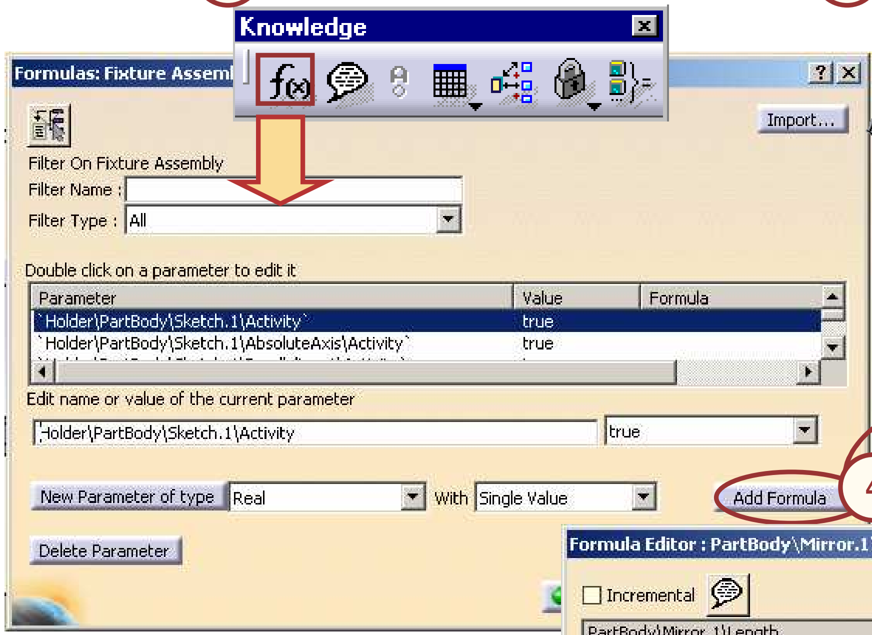
Using a Parameter Of the Assembly to Design a Part (1/3)

You can also drive a parameter of a Part of the Assembly with a parameter of the Assembly itself.

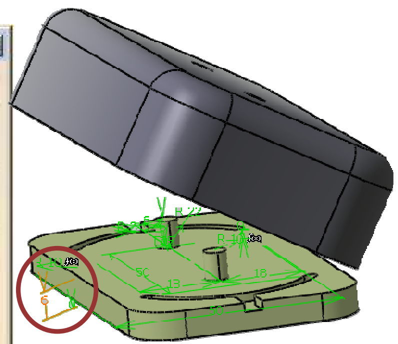
1 Edit the Part in which you want to create a relation



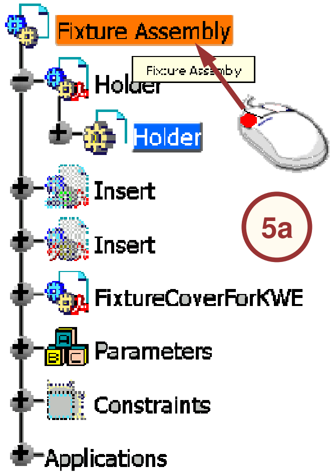
2 Click on Formula icon



3 Select the parameter you want to be driven



4 Click on

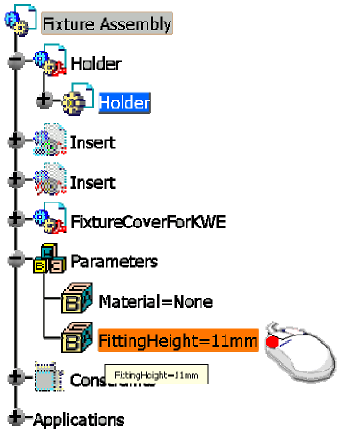


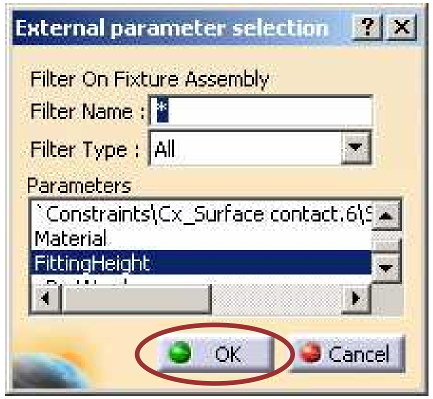
5a First select the root node of the assembly so CATIA will know you want to select a parameter outside the active part

5 Editing the formula, select a Parameter in the root assembly

Student Notes:

Using a Parameter Of the Assembly to Design a Part (2/3)

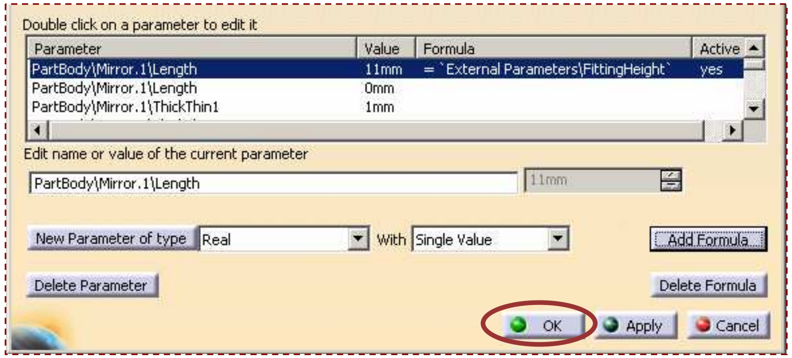




5b The External parameter selection box has appeared

5c Select driving parameter

5d Validate the External Parameter selection



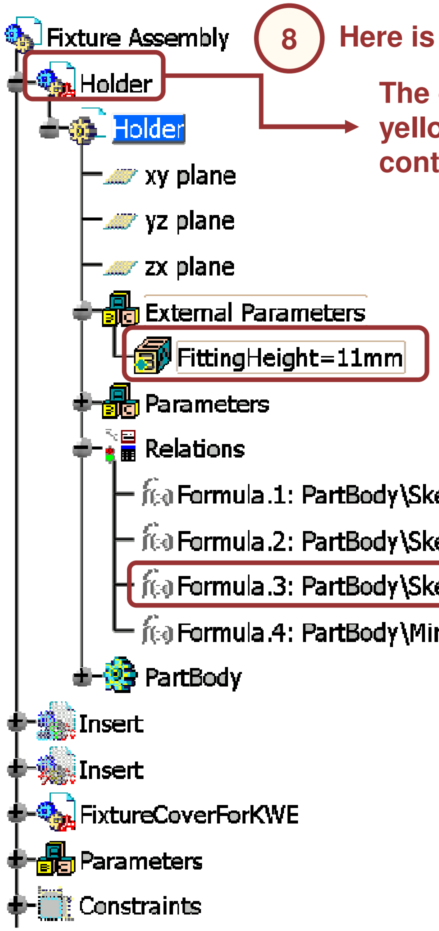
6 Validate the formula

7 Validate the Parameter edition

Student Notes:

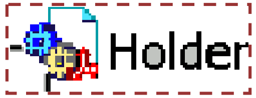
Using a Parameter Of the Assembly to Design a Part (3/3)

The component using an external parameter in the assembly becomes contextual.



8 Here is the result

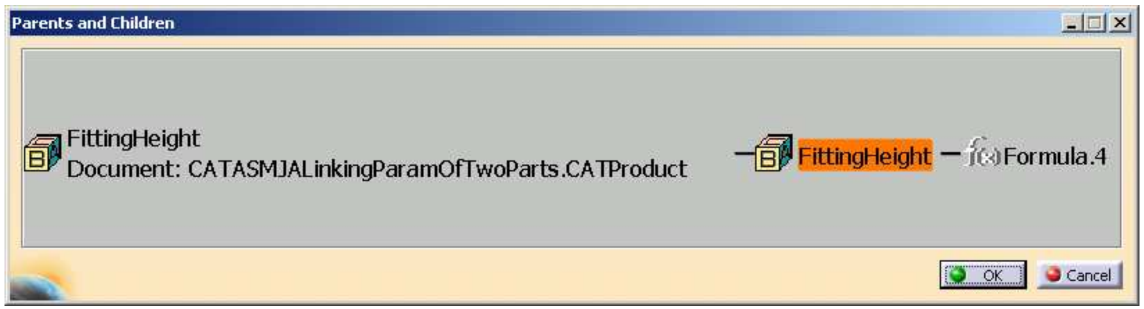
The component still have a yellow wheel indicating it is not contextual to the assembly



A parameter with green light (indicating synchronization with external document) has appeared under External Parameters node

Created formula is under Relations node

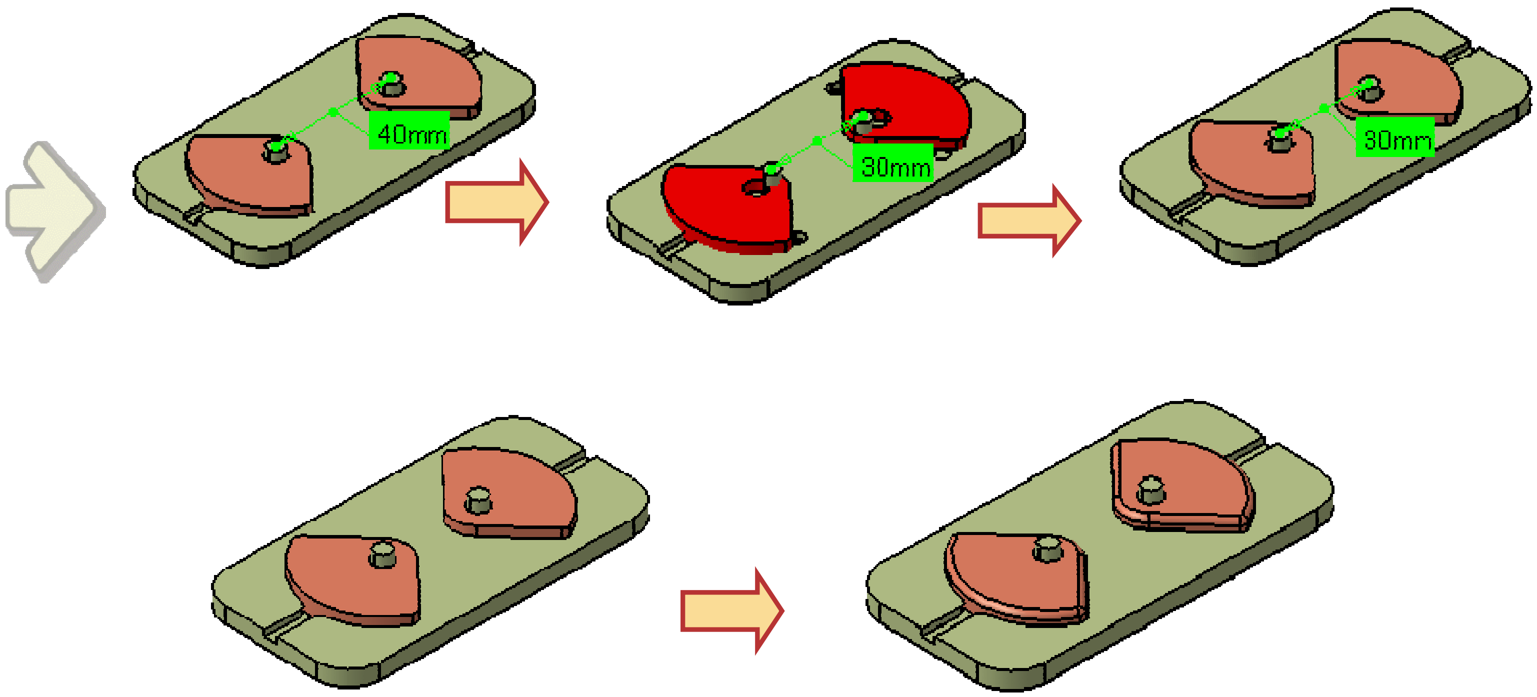
9 Parent and Children box of «FittinHeight» parameter displays the link to the external Parameter and indicates its owner document



Student Notes:

Editing Contextually-related Parts

You will learn how to edit contextual parts

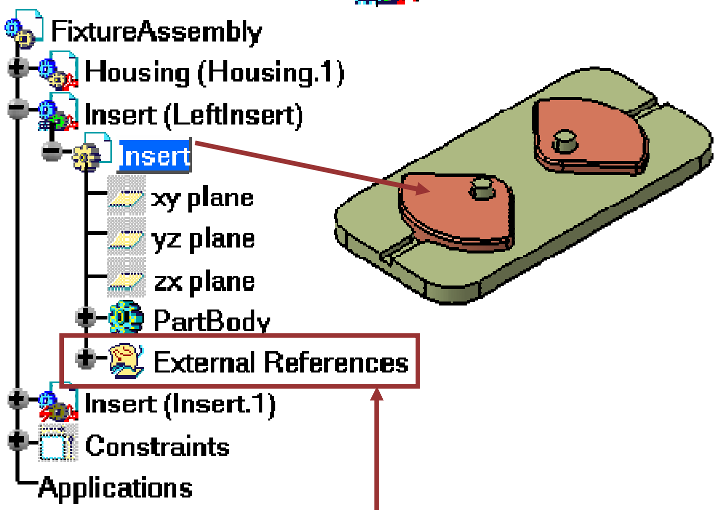


Student Notes:

What Is Editing Contextually-related Parts ?

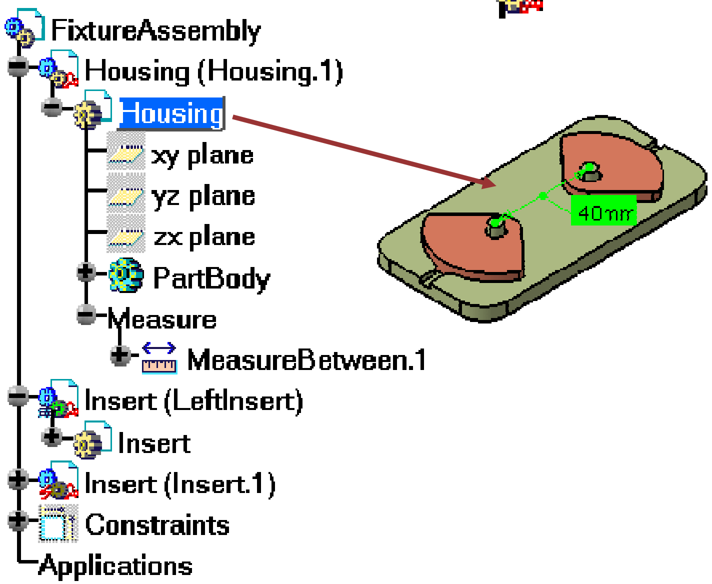
With regard to editing parts, there are two notions to consider: editing contextual parts that have external references and are therefore driven; and editing parts that drive contextual parts.

Editing Driven Part



Here we are editing a contextual part that has External References and is therefore <u>driven</u> by geometry in other components.

Editing Driving Part



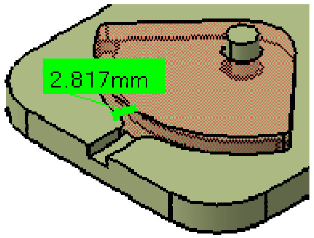
Here we are editing a part that <u>drives</u> geometry in other parts that are contextual.

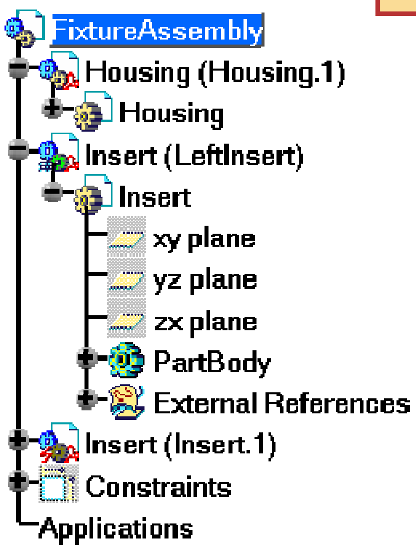
Student Notes:

What Is Editing Driving Parts ?

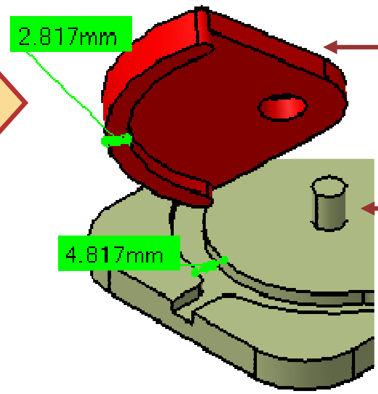
Editing a part that drives other contextual parts will cause changes to geometry in the other parts.

Housing (Driving Part)



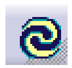


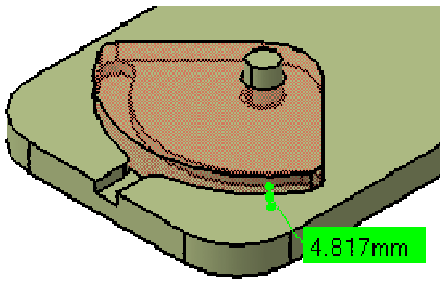
In this example Housing is a Driving Part. It drives the geometry of Insert Part.



Insert (Driven Part)

Housing (Driving Part)



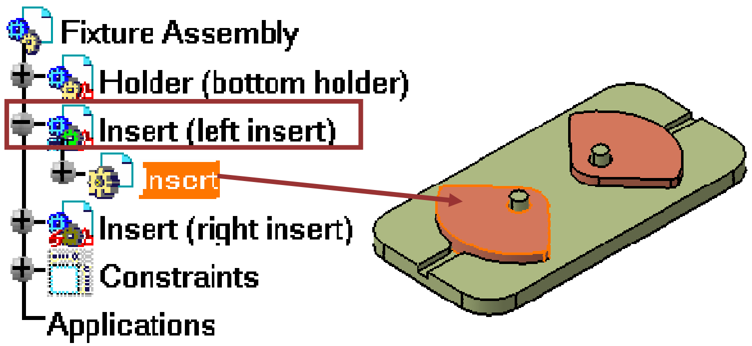


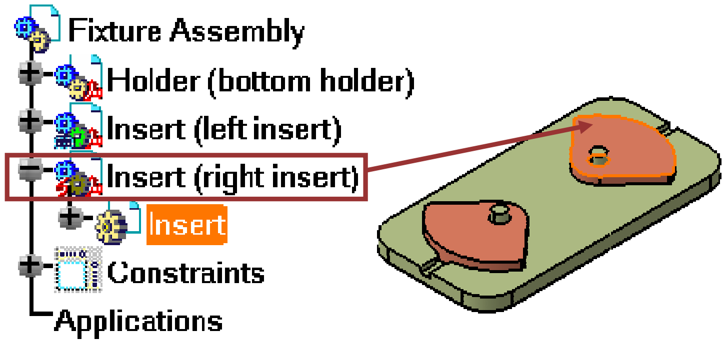
Here the width of the slot in the green part has been changed. The width of the "rib" of the insert is driven by the edges of the slot.

Student Notes:

What Is Editing Contextual Parts ?

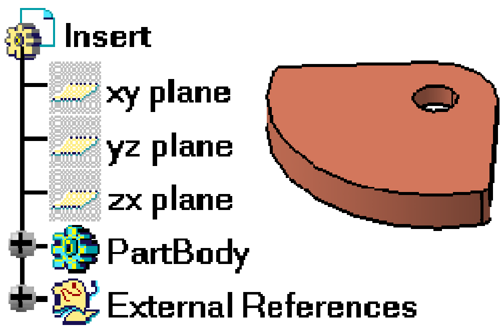
Parts that are contextual (driven) by other components can be edited within or outside the context of the assembly in which the contextual elements were defined.





You can edit the <u>"original" instance</u> of a contextual part because often many of the contextual elements were probably defined here.

However, you can also edit a contextual part via instances of the part that are <u>not the "original" instance</u>. This can be useful when defining new contextual elements that are dependant on the position of an instance that is not the "original" instance.



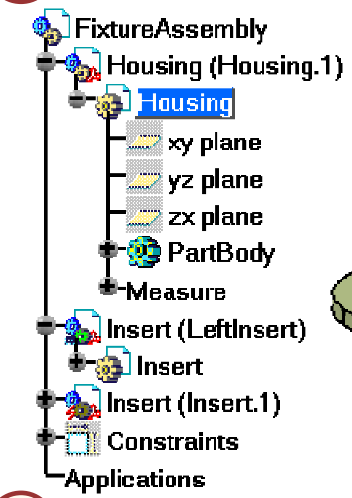
You can also edit contextual parts <u>without opening the assembly</u>, but contextual elements cannot be completely updated because the context (assembly and components) in which the contextual elements were defined is not available.

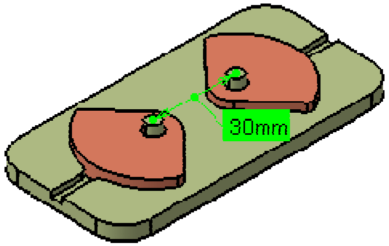
Student Notes:

Editing Driving Parts

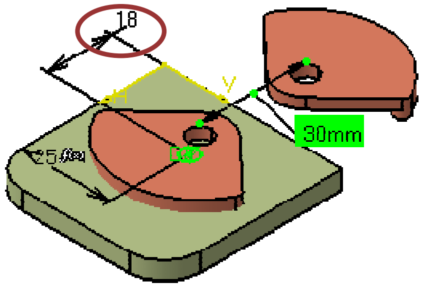
After editing driving parts you will have to update contextual (driven) parts in-context of the assembly.

1 Double-click the driving part to be edited

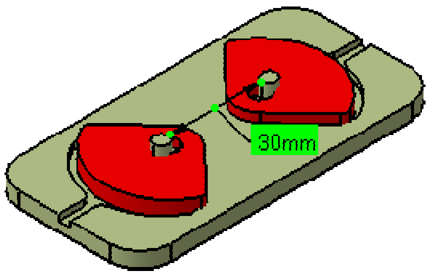




2 Edit the driving part – Change the pad offset distance to 18 mm.

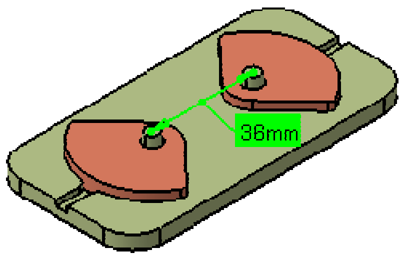


3 Exit the sketcher and Activate the Root Product.



4 Update the assembly to update the contextual part.

You can edit driving parts outside the context of the assembly, but the assembly must be opened to fully update contextual parts because contextual elements can be updated only in the context in which they were defined.

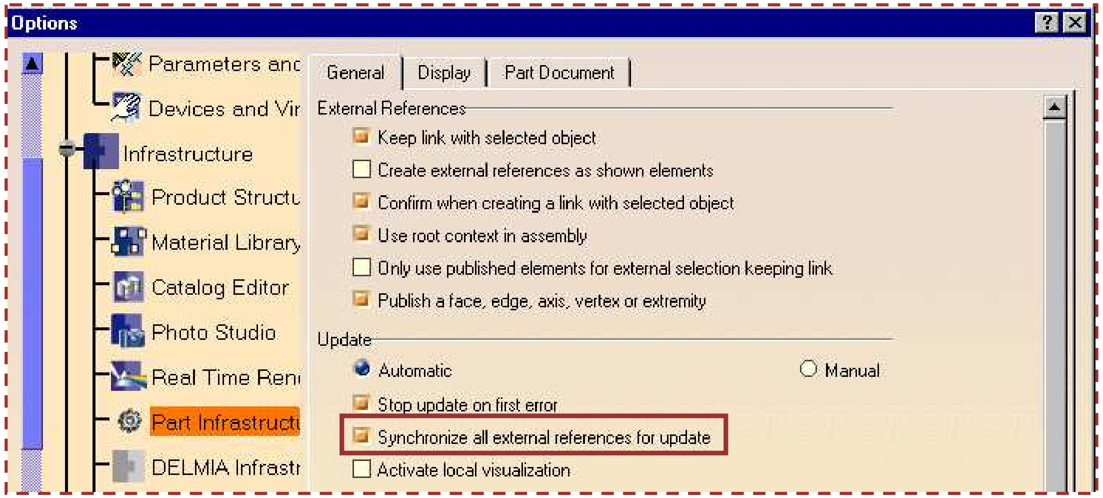


Student Notes:

Automatically Synchronizing Changes when Editing Driving Parts

You can set an option to synchronize all contextual elements when simply pressing Update.

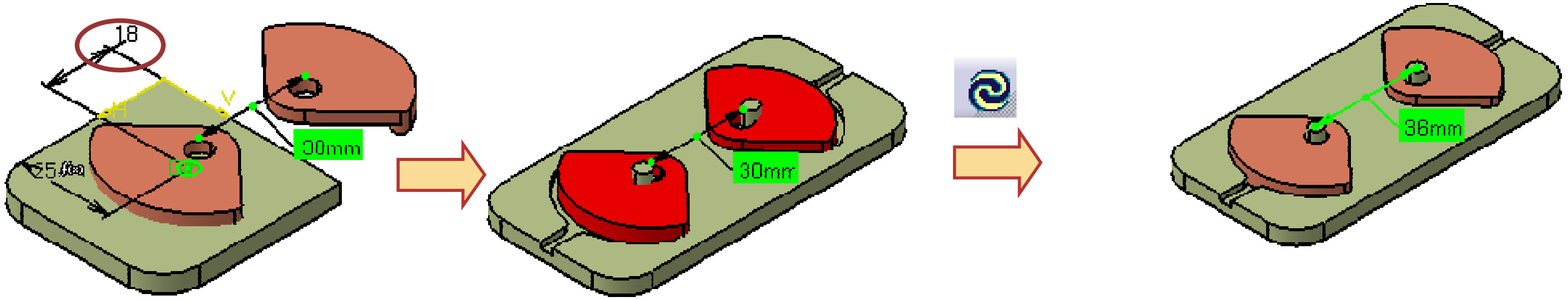
1 Turn ON the option Synchronize All External References for Update



All contextual elements in driven parts are synchronized with driving parts by simply pressing Update.

2 Edit the driving part. The location of pad in Housing is changed.

3 Update contextual parts that are driven by the edited part.

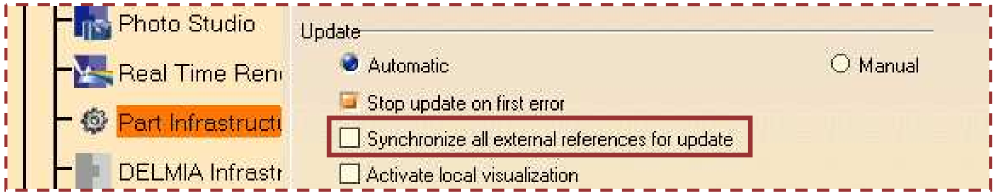


Student Notes:

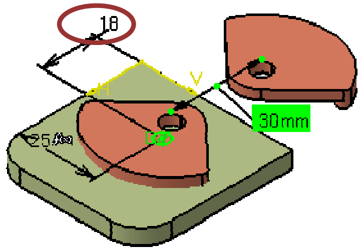
Manually Synchronizing Changes when Editing Contextual Parts (1/2)

You can set an option to synchronize individual contextual elements.

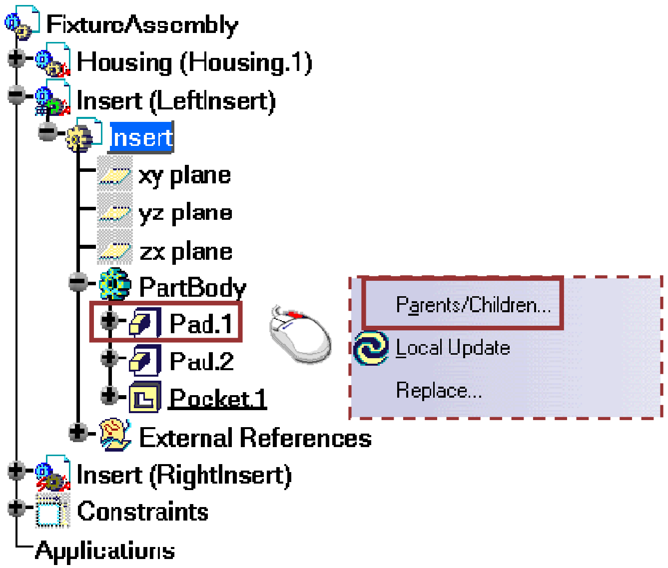
1 Turn OFF the option Synchronize All External References for Update



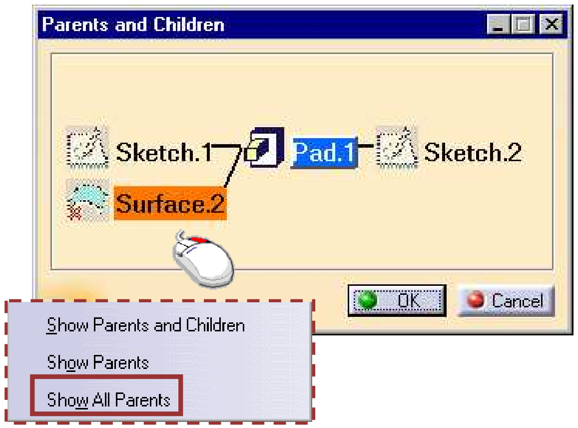
2 Edit the driving part.



3 Right-click the feature to be updated in the contextual part and select Parent/Children…

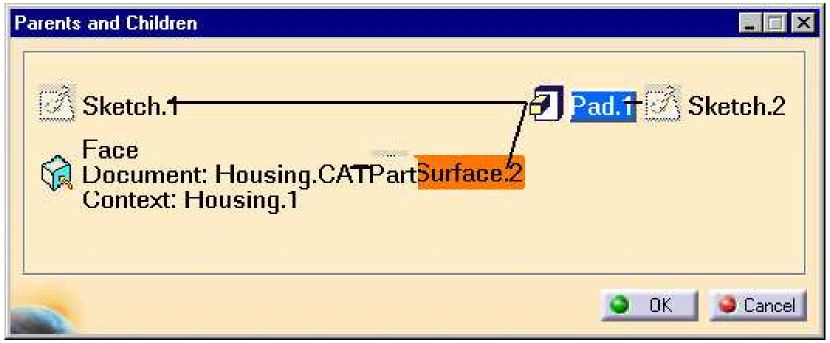


4 Right-click the node of interest and select Show All Parents to see External References.

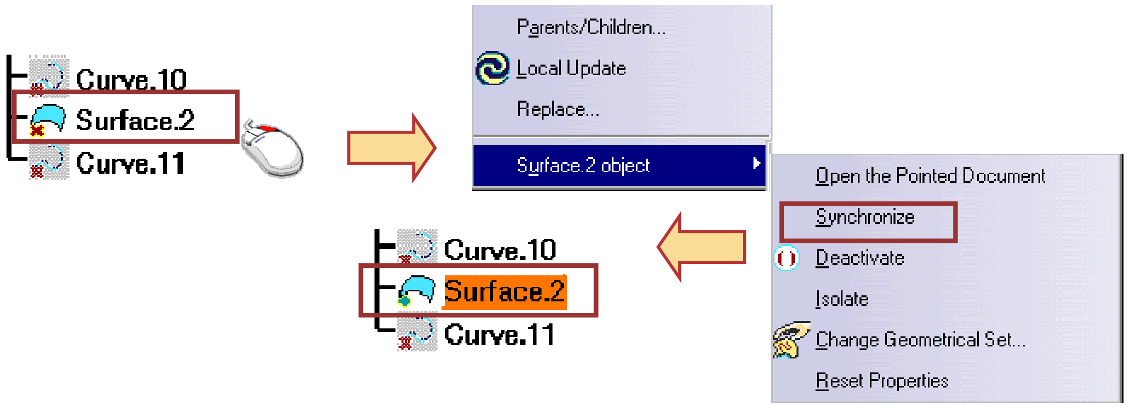


Manually Synchronizing Changes when Editing Contextual Parts (2/2)

5 **All parents of Surface.2 are displayed in the Parents and Children window.**



6 **Right-click the node of interest in the specification tree and from selected Object Menu, click on Synchronize.**



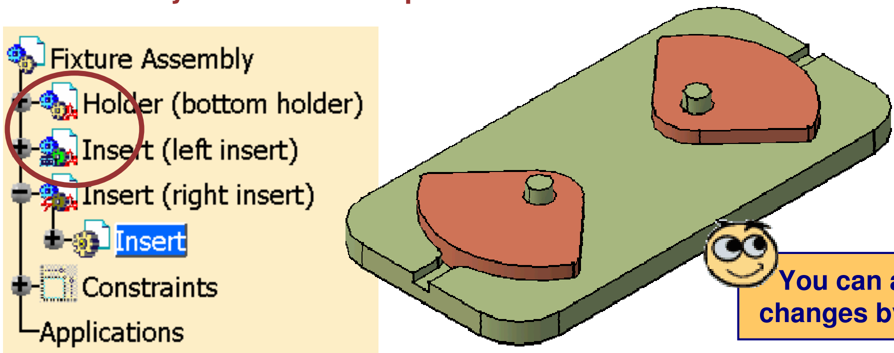
Student Notes:

Student Notes:

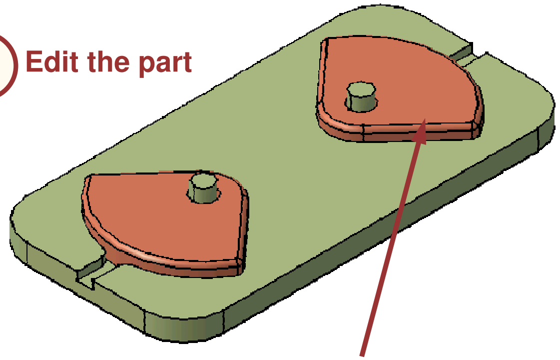
Editing Contextual Parts

When editing contextual parts pay close attention to the creation and edition of contextual elements.

1 To edit or create non-contextual elements, double-click any instance of the part



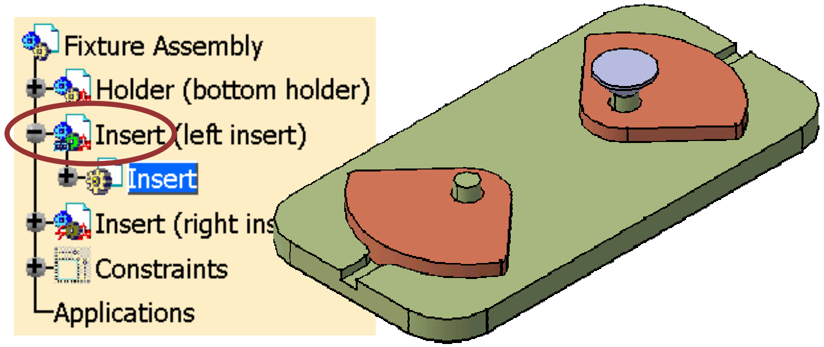
2 Edit the part



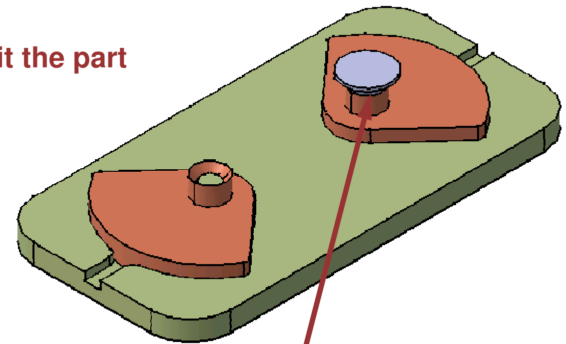
You can also make non-contextual changes by opening only the CATPart.

Added fillets

1 To edit or create a contextual element, double-click the instance of the part in which the contextual elements is defined (or will be defined)



2 Edit the part



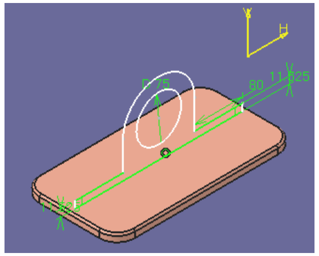
Added a pad and limited it up-to-surface of the light blue component. This contextual element had to be defined in the "right insert" instance in order to reference the near-by blue component.

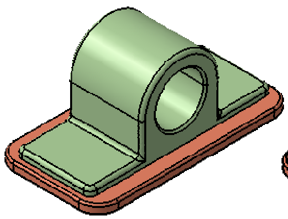
Fully Constraining Contextual Parts

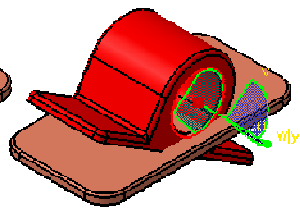
Student Notes:

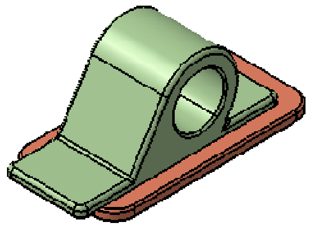
It is important to fully constrain contextual parts to avoid unintentional distortion of geometry.

In this example, Housing part is contextually designed and has external references to the geometrical elements of the Base.







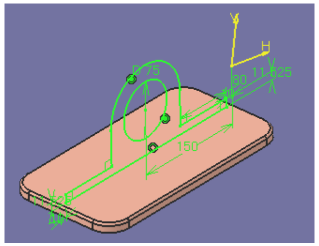


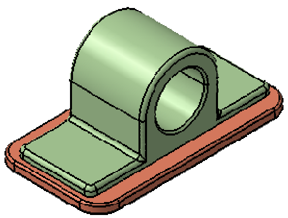
The sketch of the pad is not fully constrained.

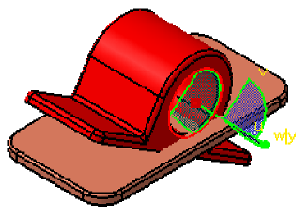
The housing part is rotated by a small amount.

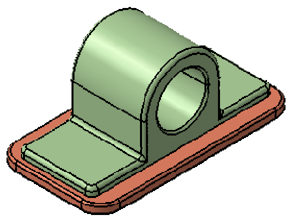
The geometry of pad is distorted.

On updating the Housing part, the contextual sketch is projected back onto the large green part. But the geometry of the housing part is distorted.









The sketch of the pad is Fully constrainted.

The housing part is rotated by a small amount.

The housing part is updated without any distortion.

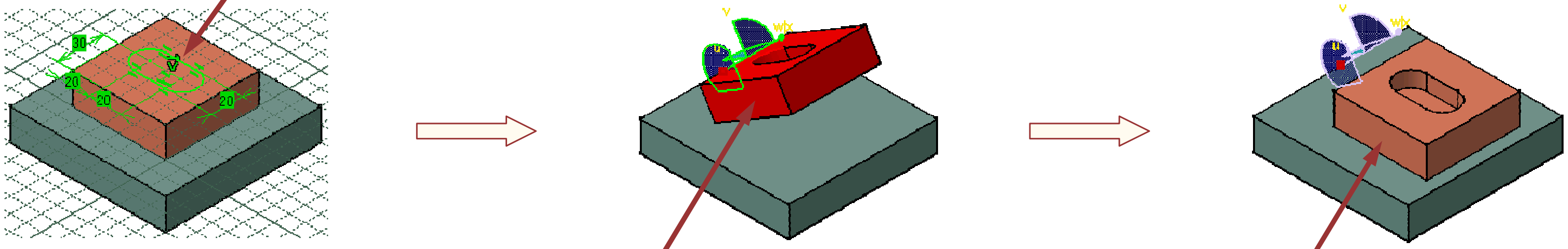
Fully constraining the housing part ensures that it maintains the expected location relative to the small brown pad.

Student Notes:

Fixing-in-Space Contextual Parts

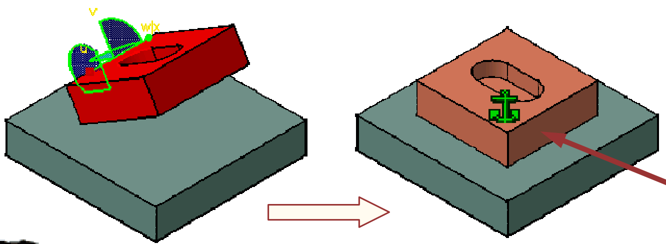
Moving a component can unintentionally cause geometry to move within a contextual part.

Here the slot is fully constrained to the pad



Suppose a component is temporarily moved

Updating the small brown part projects the contextual sketch back onto the large green part. But the pad appears to be in the wrong location.



To avoid unintentionally moving geometry in contextual parts, ensure that components are in their "assembled" position before updating contextual parts.

To make this easy:

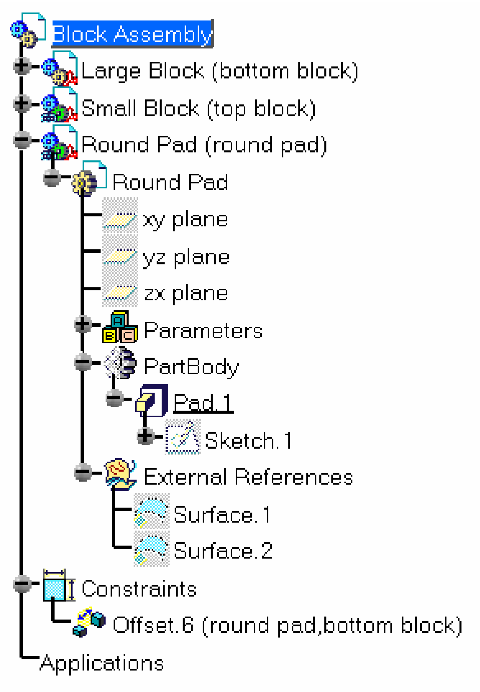
- Fix-in-space components
- Update the assembly to move components back in position before updating contextual parts

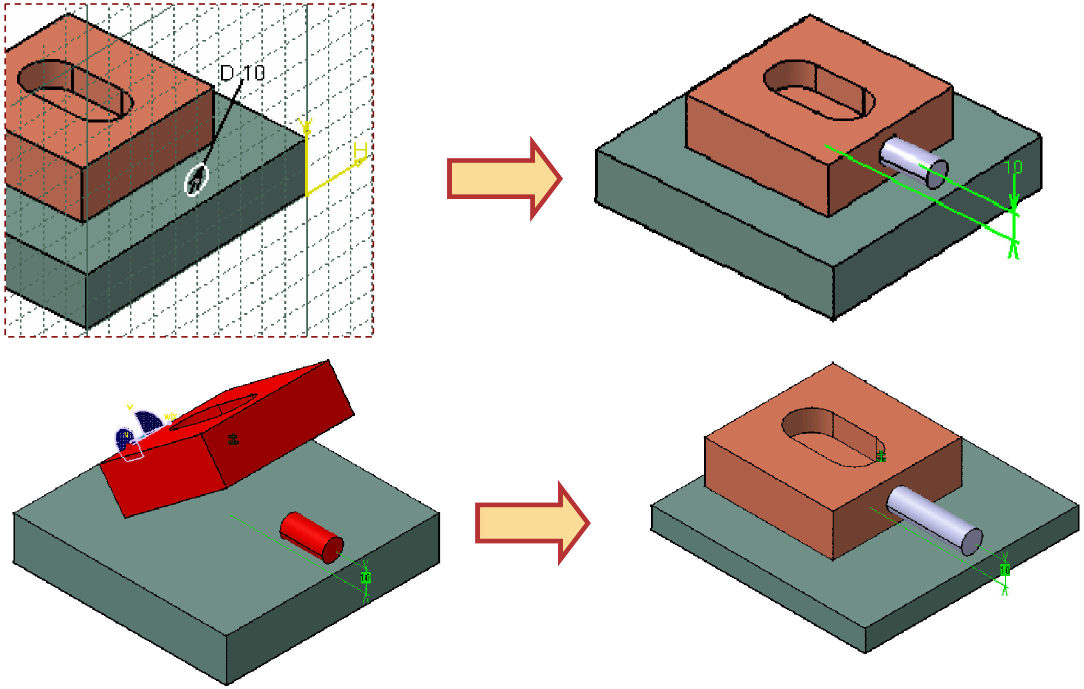
Unintentionally moving geometry in contextual parts may have adverse affects on scenes and drawings.

Student Notes:

Do It Yourself

Product used: CATASMAssociativeBlock_Assembly_A4.CATProduct



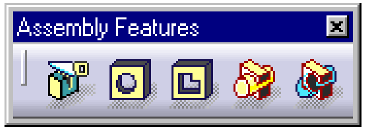


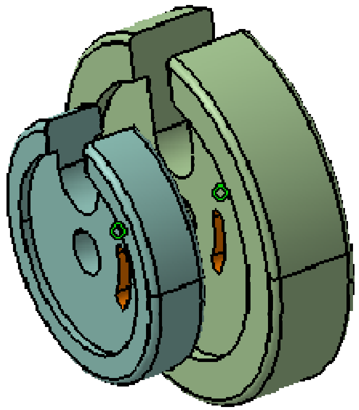
- Create a sketch for round pad of 10 mm diameter on the face of green block so that the sketch is contextually related to the green component
- Limit the round pad up-to-plane of the brown component so that the length of the round pad is contextually related to the face of the brown component
- Define an assembly offset constraint between the large green pad and the axis of the round pad
- Temporarily rotate the small brown part and update the assembly. The position of brown part is restored.
- Decrease the thickness of the large green pad and move the small brown pad.

Student Notes:

Creating Assembly Features

You will learn how to Create Assembly Features



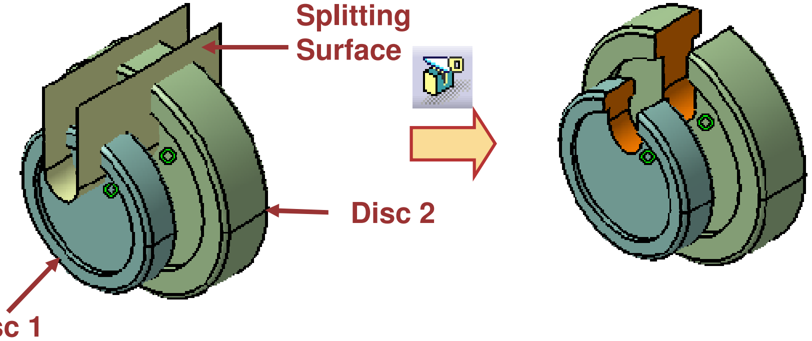


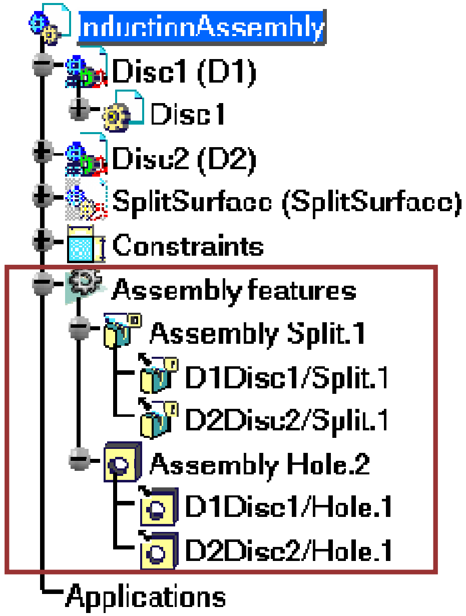
What Are Assembly Features ? (1/3)

Assembly features are features that are applied not only to a single part (in Part Design Workbench) but to a set of several parts of an assembly.

Following are different assembly features: Split, Hole, Pocket, Add and Remove.

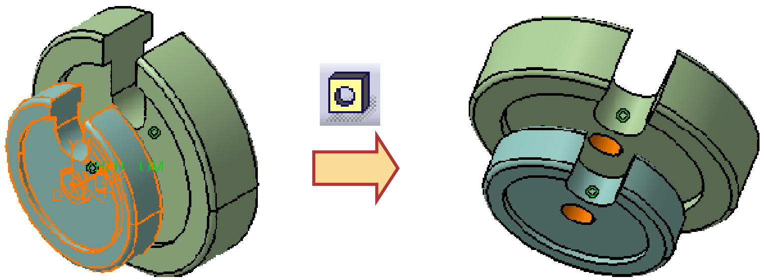
Split





Split : This operation splits one or more parts with the splitting surface in a single instance.

Hole



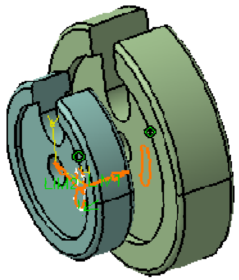
Hole: This operation creates hole passing through multiple parts in a single instance.

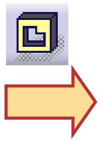
Student Notes:

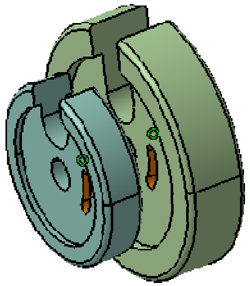
Student Notes:

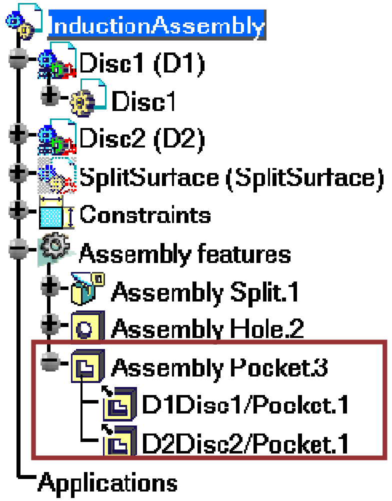
What Are Assembly Features ? (2/3)

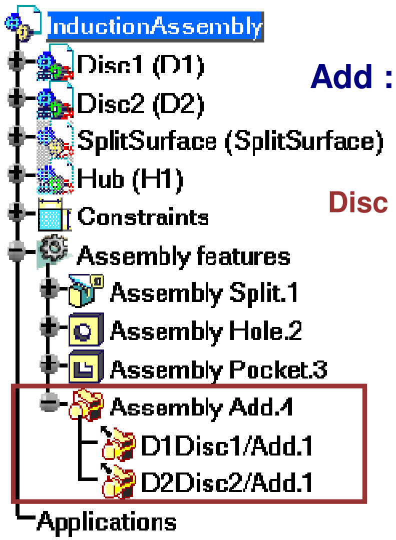
Pocket : This operation creates pockets in multiple parts in a single instance.







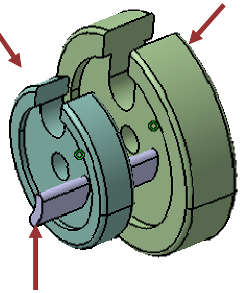




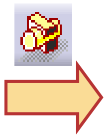
Add : This operation adds a part to multiple parts in a single instance.

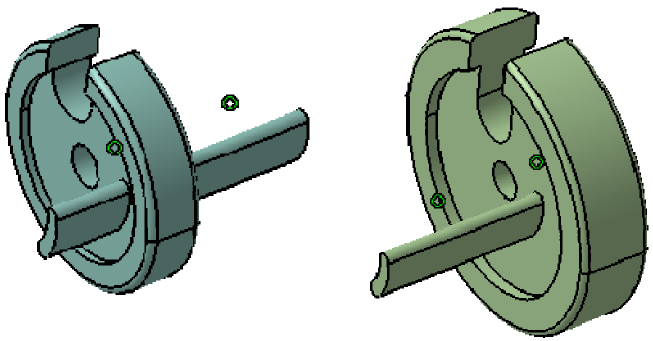
Disc 1

Disc 2



Body being added

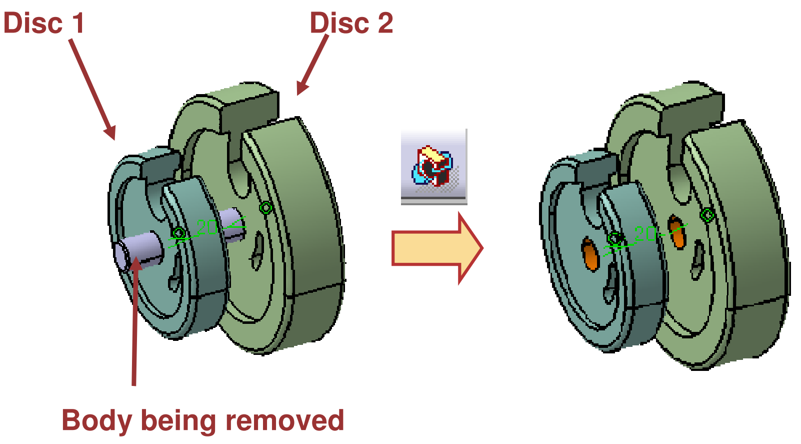


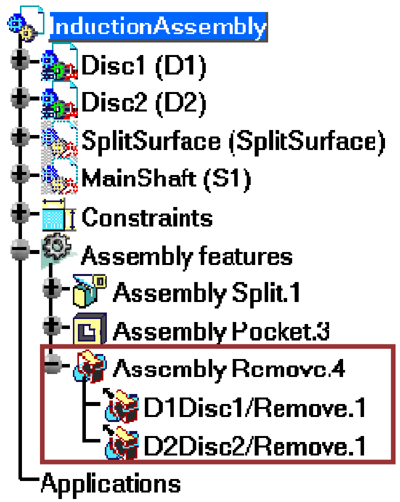


Student Notes:

What Are Assembly Features ? (3/3)

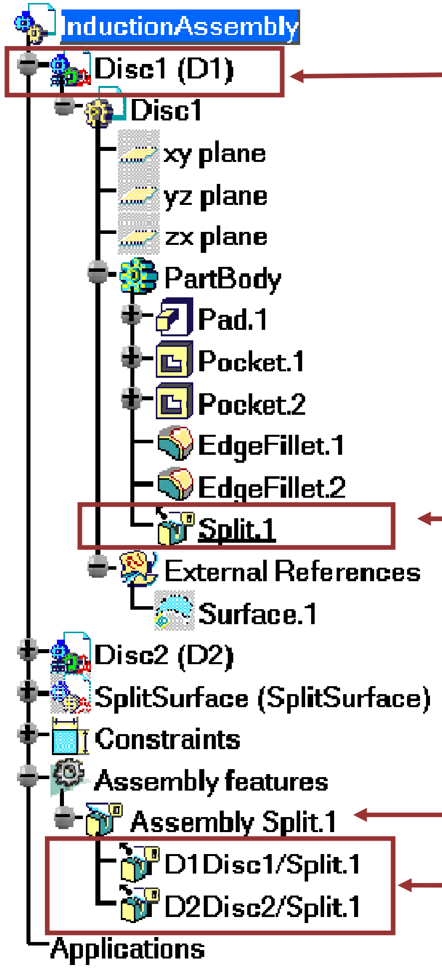
Remove : This operation removes material from all affected parts using the geometry of a part used in Remove operation in a single instance.



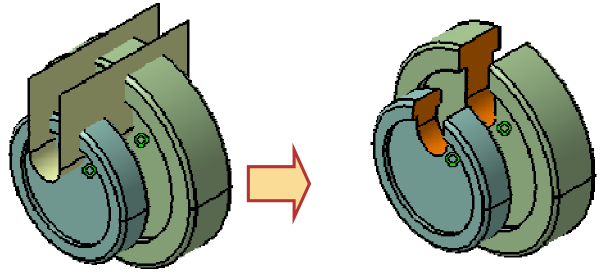


What Are Affected Parts ?

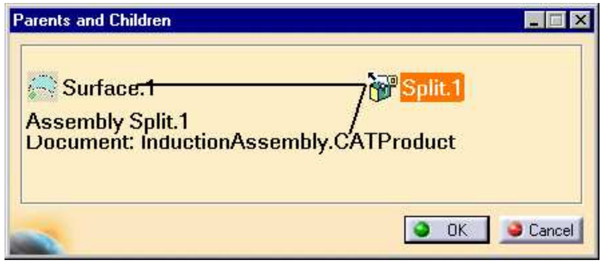
Affected parts are parts of the assembly that will be involved in the assembly feature



Affected parts become contextually linked



Linked feature is created in affected part



Creation and edition at the level of the assembly

List of affected parts and linked features in them

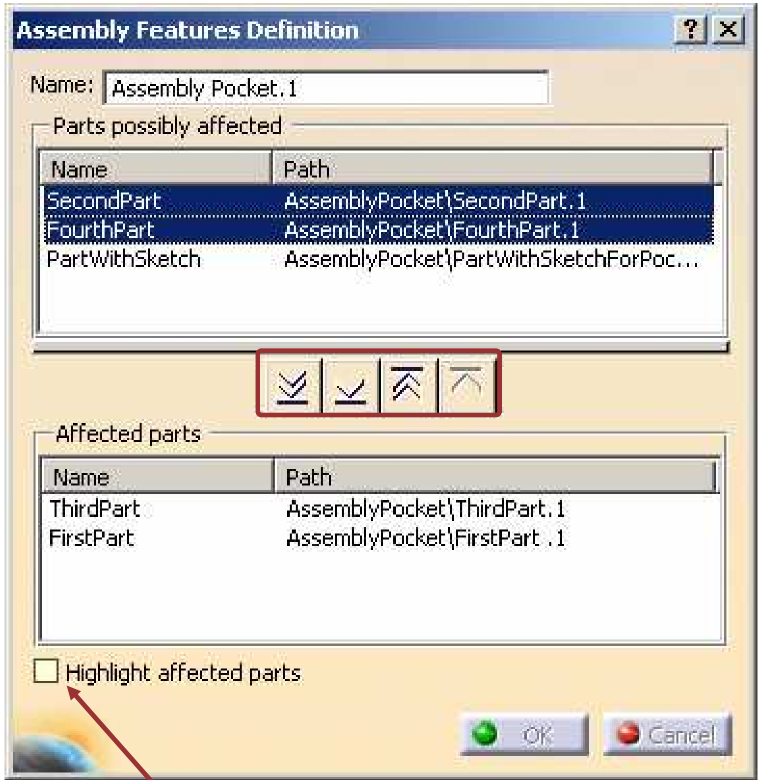
You can see that split is refering to Surface.1 which is a Split surface.

Student Notes:

Student Notes:

Specifying Affected Parts

Whatever assembly feature you want to create, you have to specify affected parts in the first appearing dialog box (Assembly Feature Definition dialog box).



Parts of the assembly that are not yet affected by the assembly feature. Select them by clicking them while using Ctrl and Shift keys

Parts of the assembly that will be affected by the assembly feature. Select them by clicking them while using Ctrl and Shift keys

This option allows you to highlight in geometry the parts that will be affected

Affecting Tools:

	Move all parts of upper field into lower field
	Move all parts of lower field into upper field
	Move selected parts of upper field into lower field
	Move selected parts of lower field into upper field

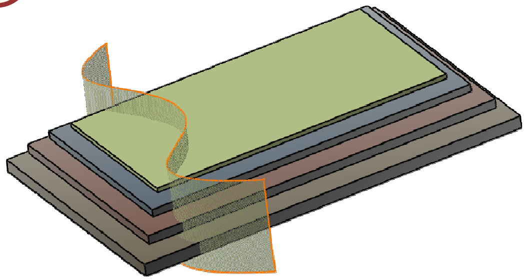
Student Notes:

Assembly Split

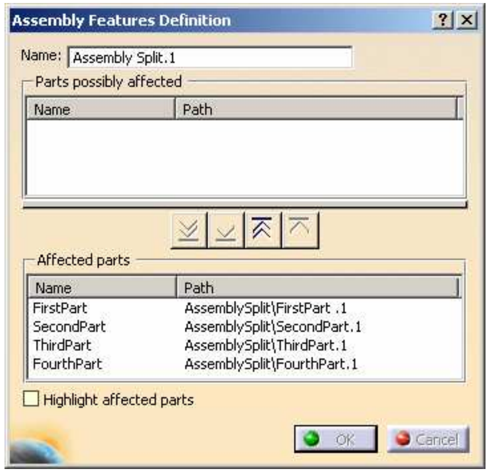
You need a surface or a plane to make a split, this surface can belong to one of the affected parts or not.

1 Click on Split icon 

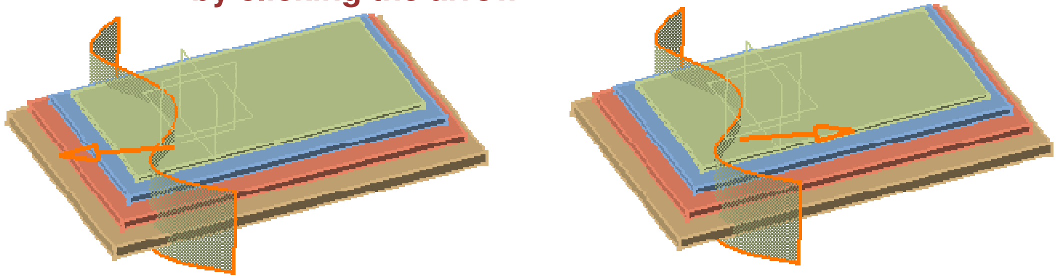
2 Select the splitting surface



3 Specify affected Parts

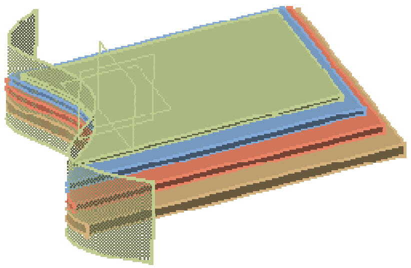


4 Select orientation of the split by clicking the arrow



5 Validate the command

6 Affected parts are splitted



Student Notes:

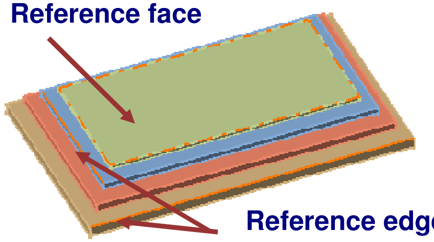
Assembly Hole

When creating an assembly hole, you will create a sketch that will belong to the part containing the reference plane

1 Click on the Hole icon.

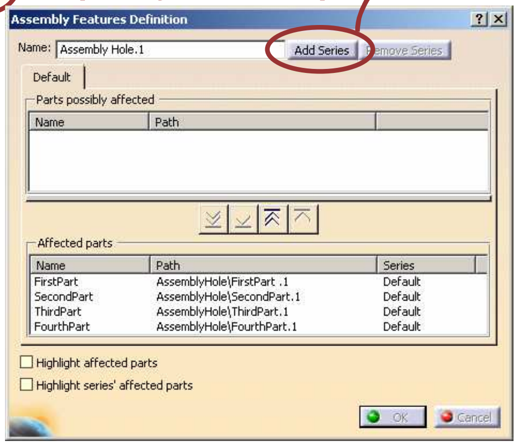
2 Select reference edges and surface for the Hole

Reference face



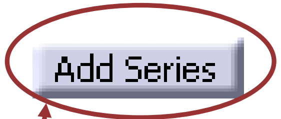
Reference edges

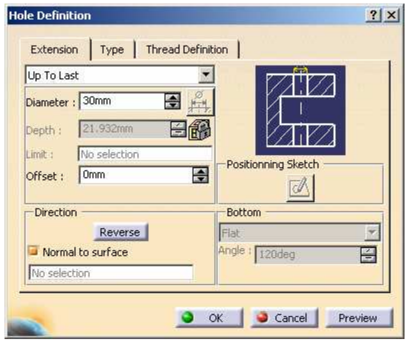
3a Specify affected parts



4 In case of checked options about contextual design, "links" can be created between elements

5 Specify hole parameters values and types and validate

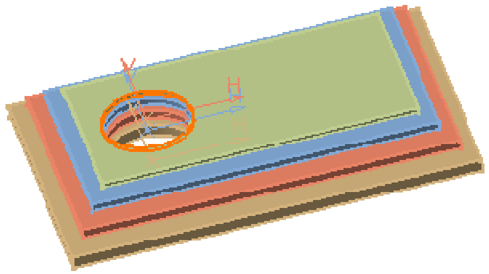




The option "Add Series" allows you to define different hole's specifications for each affected parts.

See next page for more information.

6 The hole goes through all affected parts

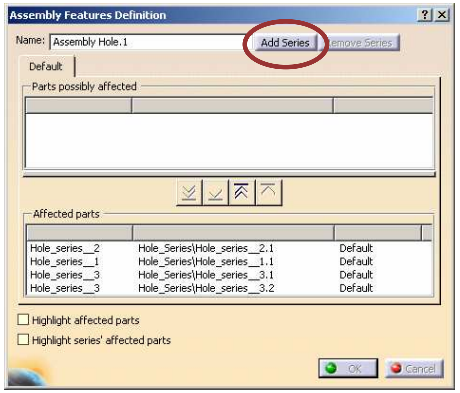


Student Notes:

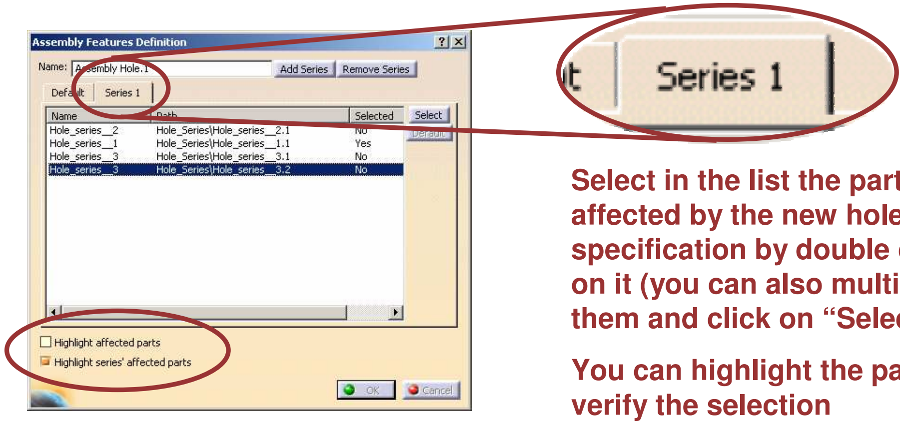
Assembly Hole Using Series

When creating an assembly hole, you can define different shapes of holes going thru parts of a product within the same assembly feature

1 It is intended that default specifications are well defined. Click on "Add Series"



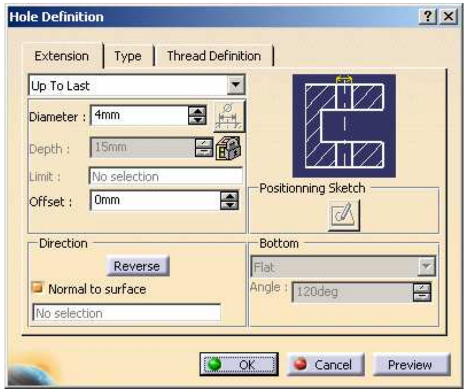
2 A new tab named "Series 1" is created



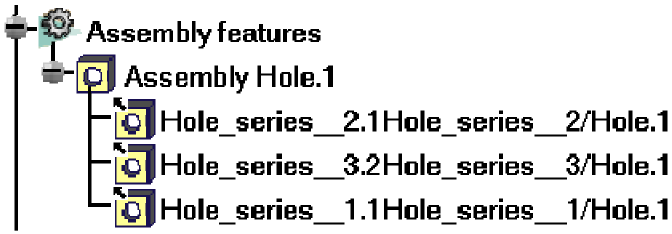
Select in the list the parts to be affected by the new hole specification by double clicking on it (you can also multi select them and click on "Select").

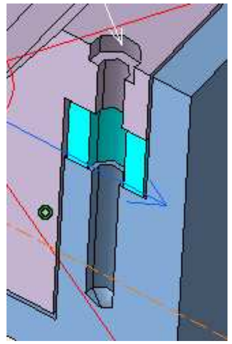
You can highlight the part to verify the selection

3 Define the new hole specification with "Hole definition" dialog box



4 Return to step 1 to add other series. After validation specification tree contain the Assembly hole feature which can be modified in Assembly Design Workbench.





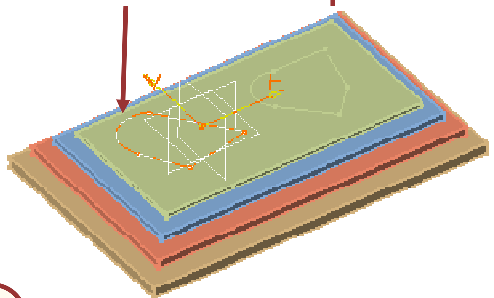
Student Notes:

Assembly Pocket

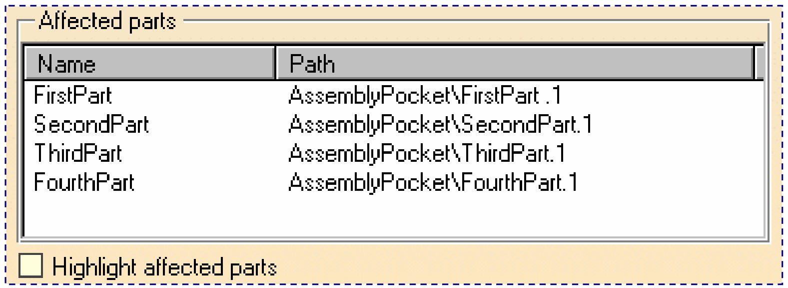
Pocket is a Sketch based feature that requires an existing sketch. This sketch can belong to one of the affected parts or not.

1 Click on Pocket icon

2 Select the sketch which will be used to make a pocket



3 Specify affected parts

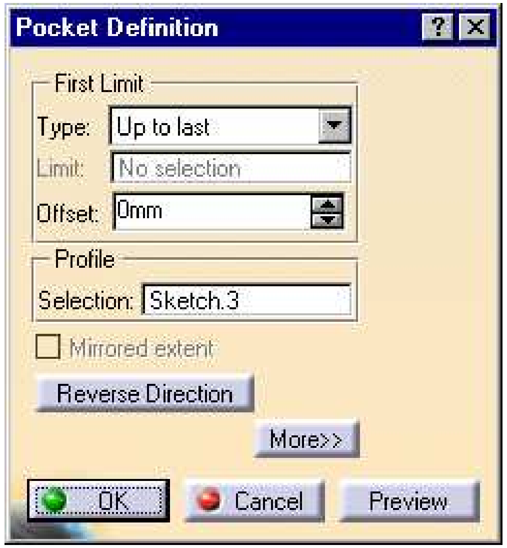


Affected parts

Name	Path
FirstPart	AssemblyPocket\FirstPart.1
SecondPart	AssemblyPocket\SecondPart.1
ThirdPart	AssemblyPocket\ThirdPart.1
FourthPart	AssemblyPocket\FourthPart.1

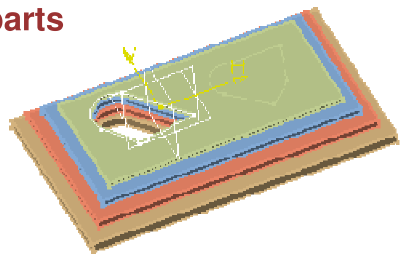
Highlight affected parts

4 Specify pocket parameter values and types



5 Validate the command

6 The created pocket goes through all affected parts



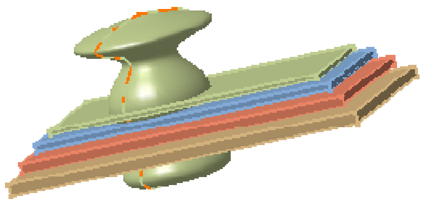
Student Notes:

Adding a Body to an Assembly

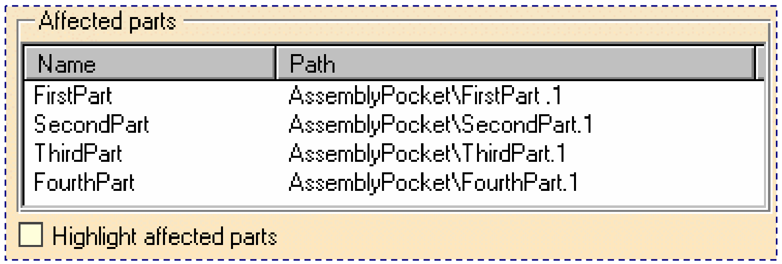
The body you want to add to several parts can belong to one of these parts. It can even be its Partbody.

1 Click on Add icon

2 Select the body to add



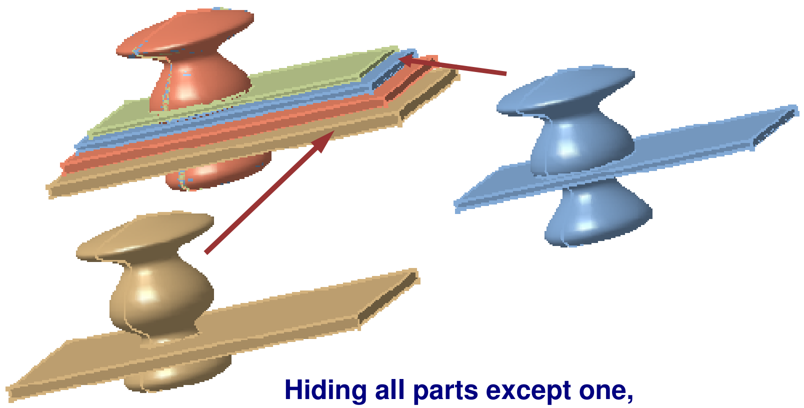
3 Specify affected parts



Name	Path
FirstPart	AssemblyPocket\FirstPart .1
SecondPart	AssemblyPocket\SecondPart.1
ThirdPart	AssemblyPocket\ThirdPart.1
FourthPart	AssemblyPocket\FourthPart.1

4 Validate

5 A linked copy of the body is added to each affected part



Hiding all parts except one, you will see that there is an added body to it

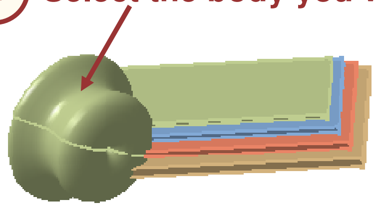
Student Notes:

Removing a Body from an Assembly

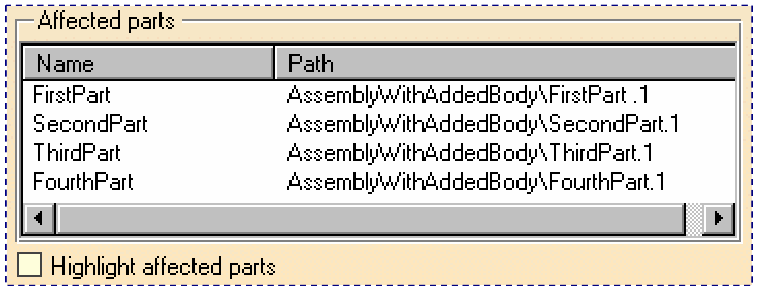
The body you want to remove from several parts can belong to one of these parts. It can even be its whole PartBody

1 Click on Remove icon

2 Select the body you want to remove



3 Specify affected parts



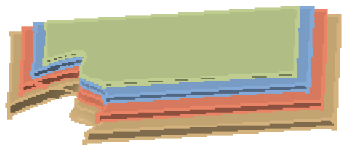
Affected parts

Name	Path
FirstPart	AssemblyWithAddedBody\FirstPart .1
SecondPart	AssemblyWithAddedBody\SecondPart.1
ThirdPart	AssemblyWithAddedBody\ThirdPart.1
FourthPart	AssemblyWithAddedBody\FourthPart.1

☐ Highlight affected parts

4 Validate the command

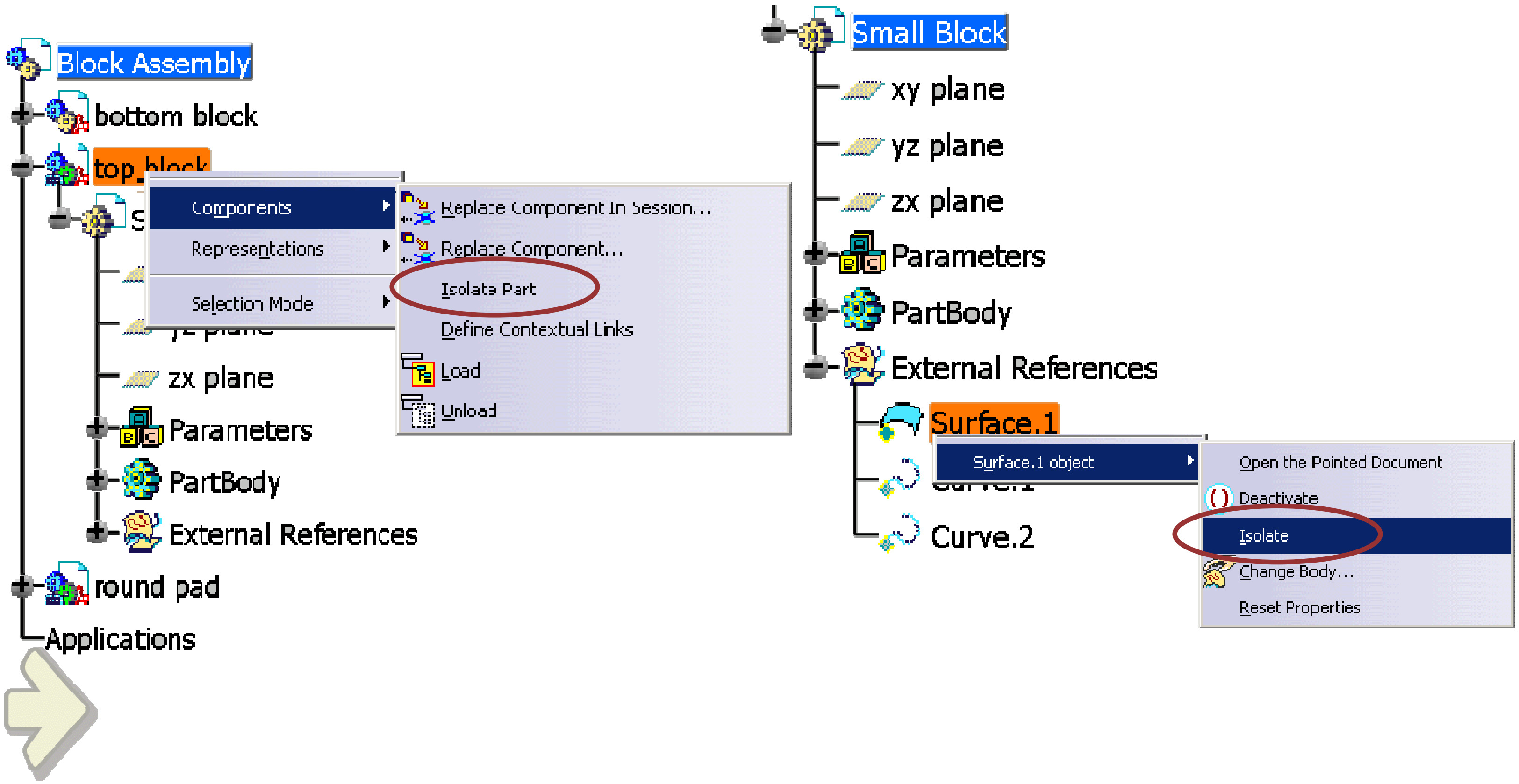
6 A linked copy of the body has been removed from each affected part



Student Notes:

Isolating Contextual Parts

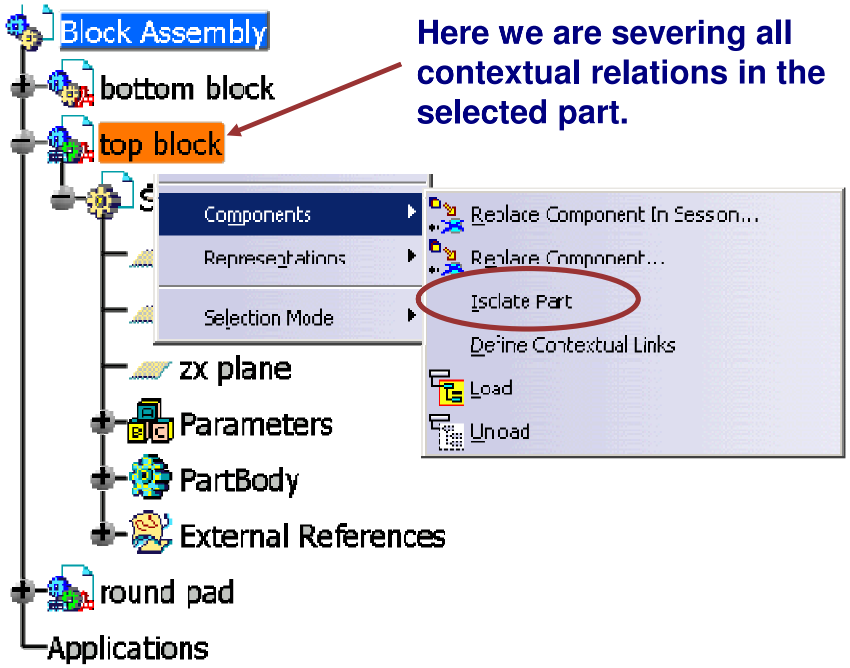
You will learn how to severe the contextual relationships between driving and driven parts.



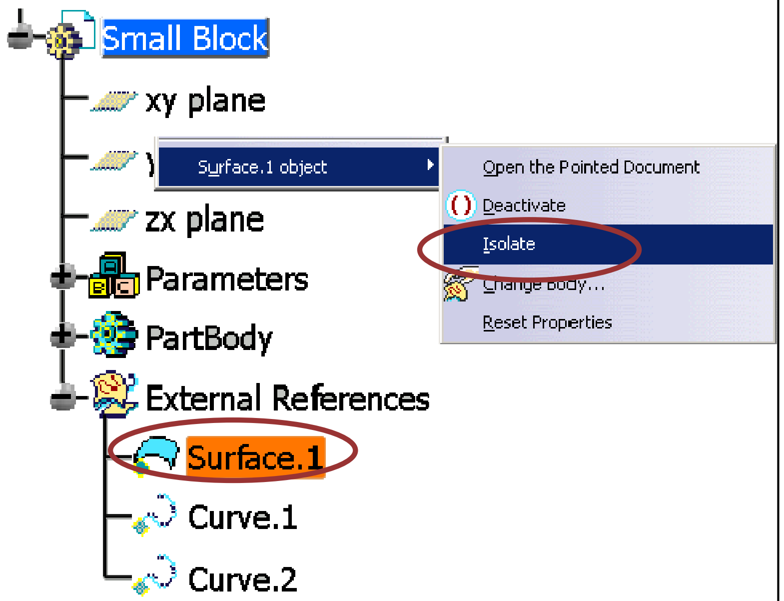
Student Notes:

What Is Isolating Contextual Parts ?

At times you may want to severe the contextual relationships between driving and driven parts.



Here we are severing all contextual relations in the selected part.



Here we are severing an individual contextual relation while leaving other contextual relations intact

Student Notes:

Why Isolate Contextual Parts ?

Contextual links may be severed because :

- **The part is being released and you want to avoid inadvertent changes.**
- **The design is stable and you no longer have a need to drive changes from one part to other parts.**
- **You inadvertently deleted the assembly and/or components that define the context for contextual elements.**

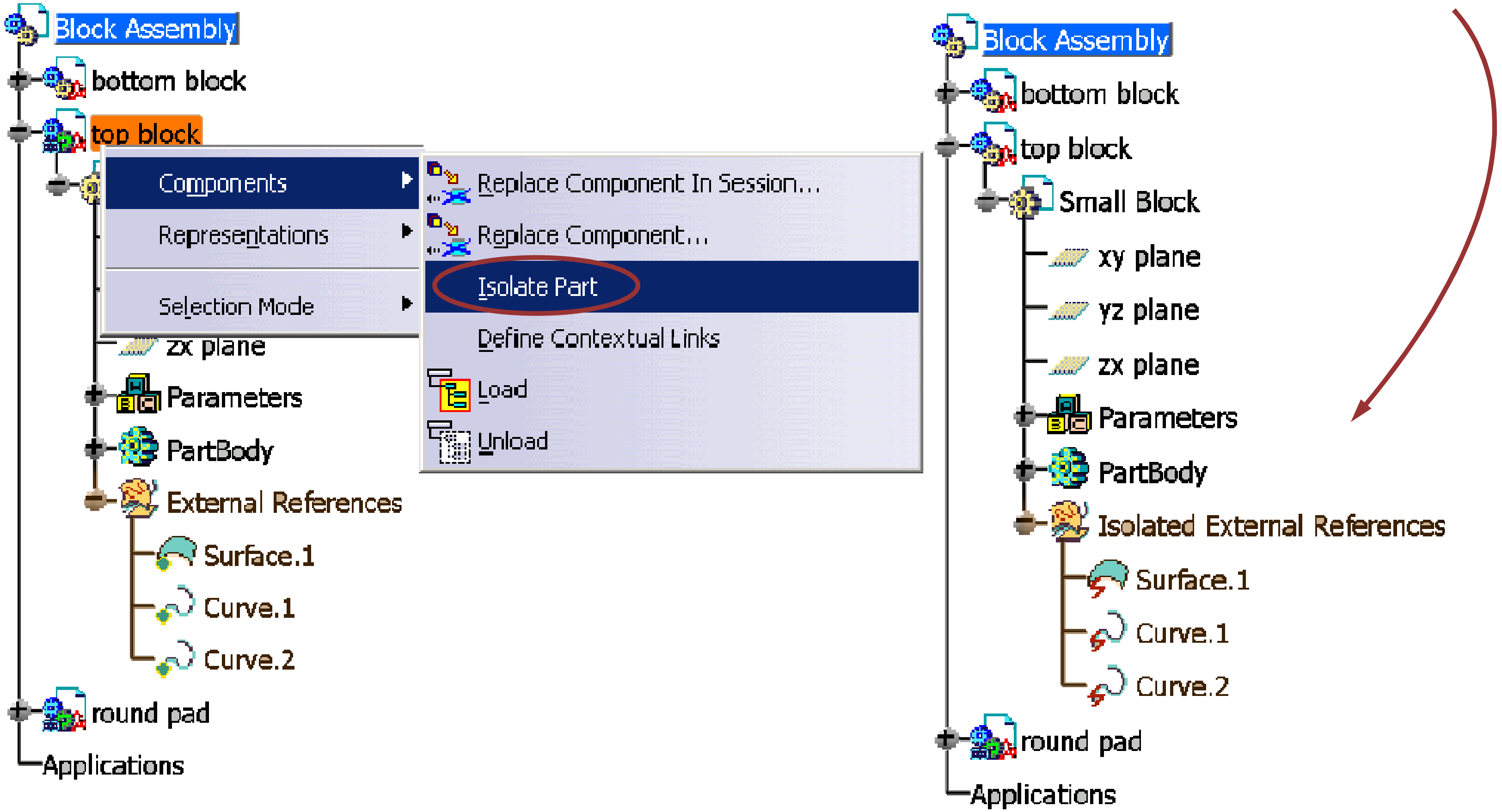
Student Notes:

Isolating All Elements in Contextual Parts

Isolating a part severs the contextual relationship with the driving components so that changes to driving parts no longer cause changes to the part that was formerly driven.

1 **Right-click the part to be isolated and select Isolate Part**

2 **The node "External References" becomes "Isolated External References"**

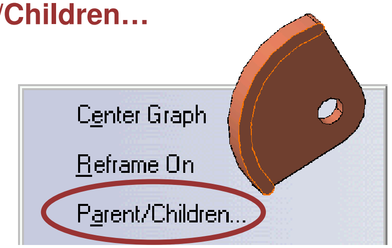


Student Notes:

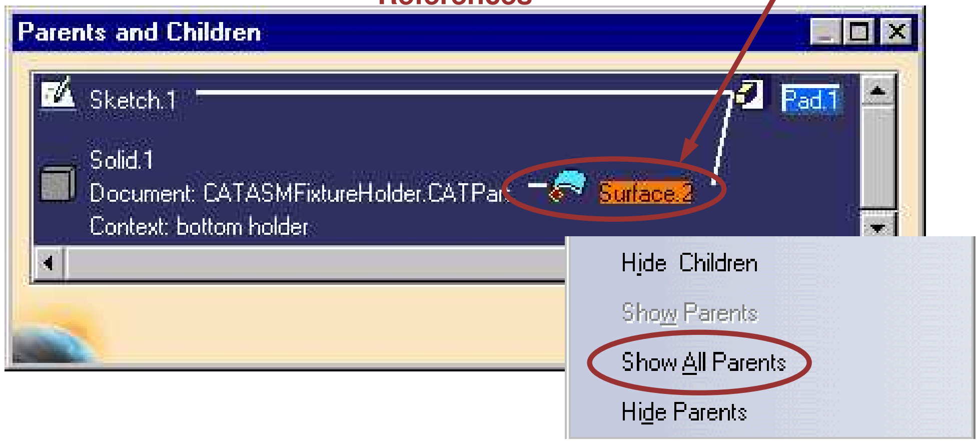
Isolating Individual Elements in Contextual Parts

You can isolate several individual contextual elements so that some elements remain driven.

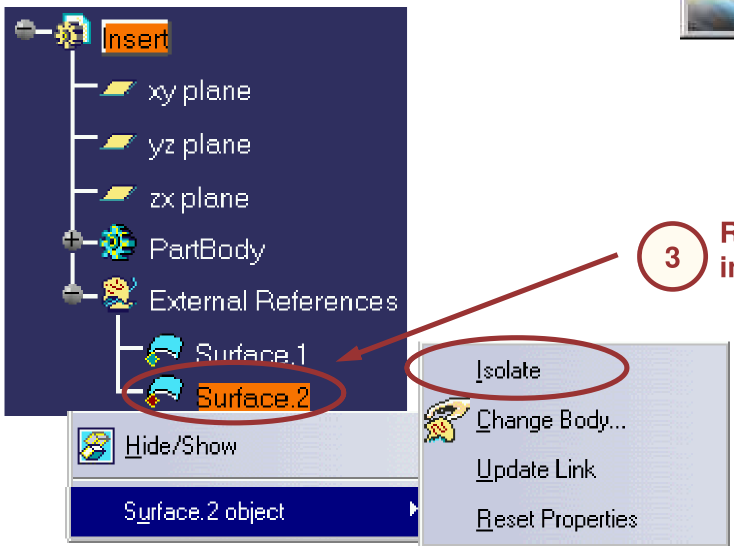
1 Right-click the feature to be isolated in the contextual part and select Parent/Children…



2 Right-click the node of interest and select Show All Parents to see External References



3 Right-click the External Reference of interest and select Isolate



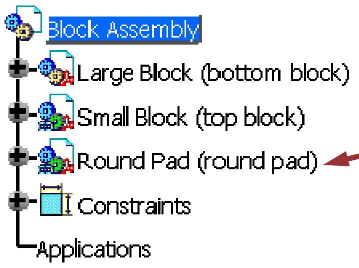
Student Notes:

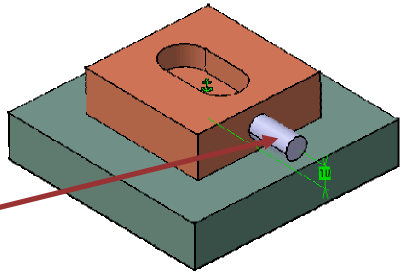
Do It Yourself

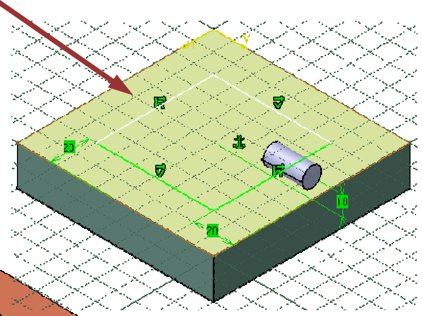
CATASMAssociativeBlock_Assembly_A4.CATProduct

1 Isolate the round pad part so that its length does not change when the top block moves.

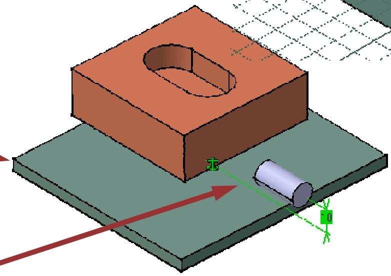
2 Isolate the surface on which the top block's pad was sketched so that the top block does not move when the thickness of the bottom block decreases.







3 Decrease the thickness of the green pad. This will result in a space between the bottom block and top block.



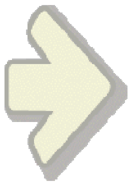
4 Edit the sketch of the brown pad to increase the distance between bottom block and top block. This will result in a space between the round pad and top block.

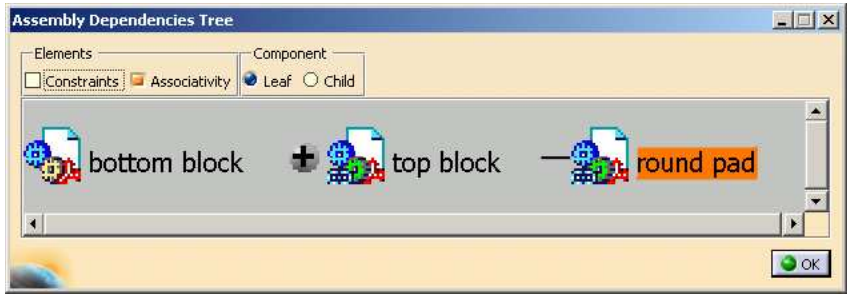
- Isolate the round pad part.
- Isolate the surface on which the top block's pad was sketched.
- Decrease the thickness of the bottom block's green pad.
- Increase the distance between the bottom block and top block.

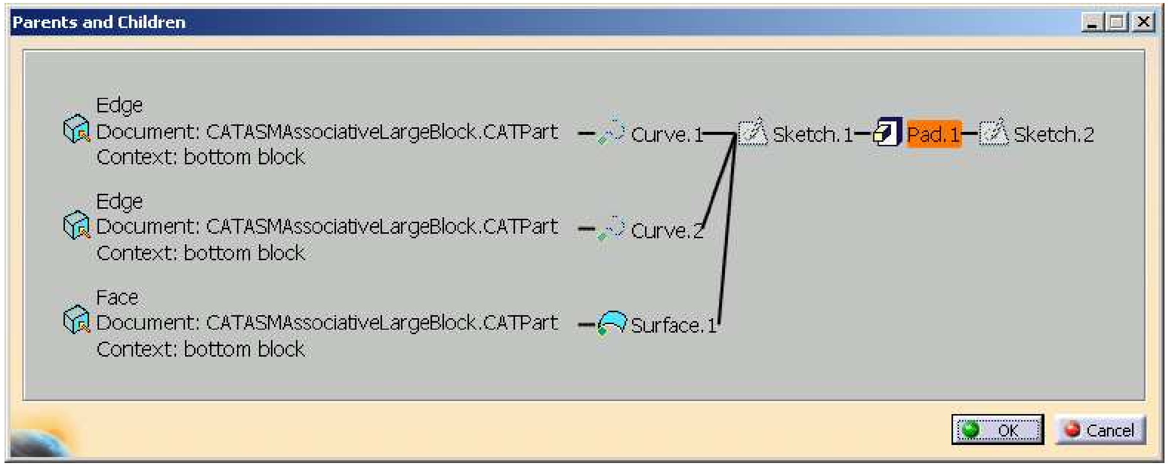
Student Notes:

Analyzing Contextual Parts

You will learn how to inquiry about the relationships between driving and driven parts.



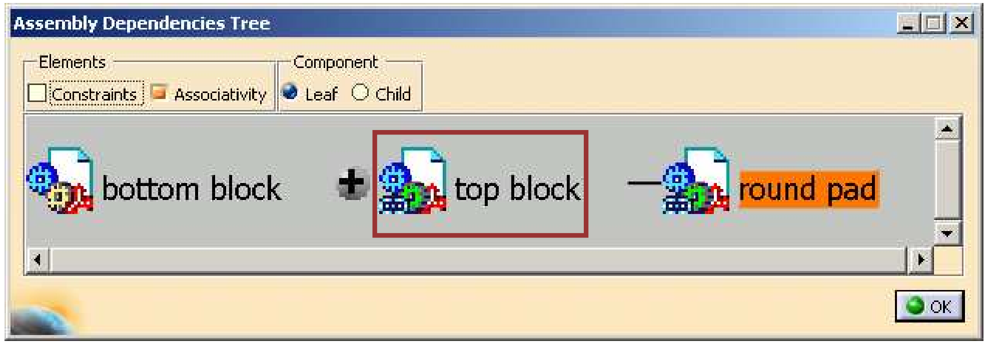




Student Notes:

What Is Analyzing Contextual Parts ?

To help understand contextual parts, inquiries can be made about relationships between driving and driven components, elements, and documents.

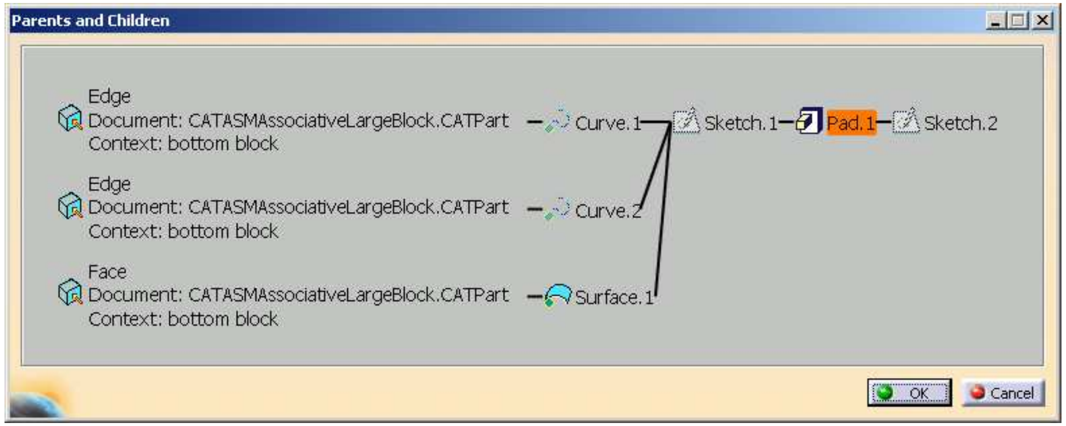


Here we are inquiring about the relationships between driving and driven components.

In this case the "top block" component is a contextual part that is driven by the "bottom block" component. In turn, the "top block" drives the "round pad" component which is another contextual part.

Here we are inquiring about the relationships between driving and driven elements and documents.

In this case sketch.1 has some External References to Pad.1 in the "bottom block" instance of another CATPart.

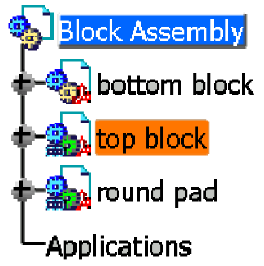


Student Notes:

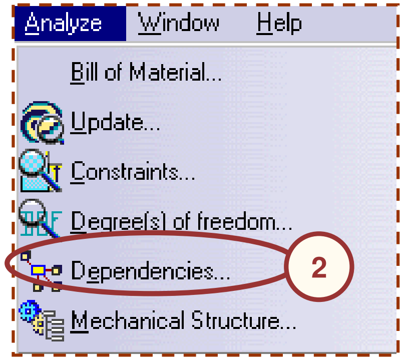
Inquiring About Parent and Child Components

Analyze dependencies to understand the relationships between driving and driven components.

1 **Select the component to be analyzed**

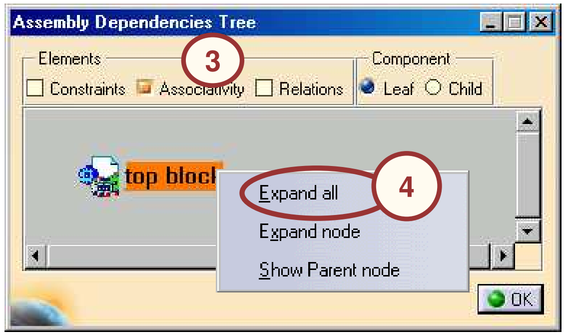


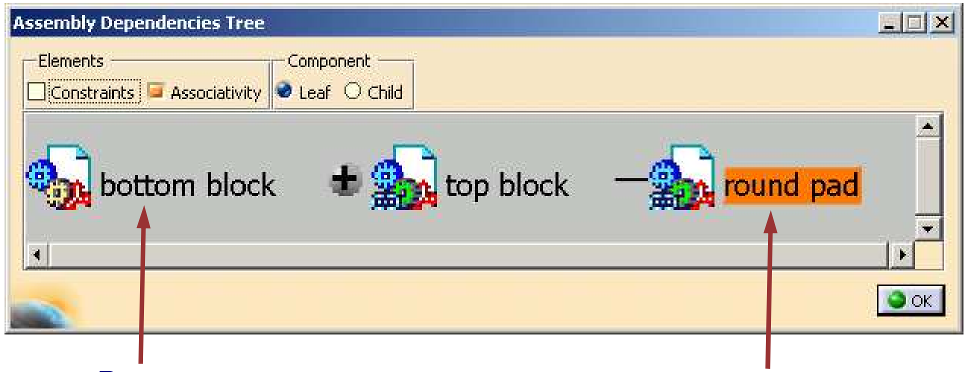
2 **Select Analyze > Dependencies**



3 **Activate the Associativity button and deactivate the Constraints button**

4 **Right-click and select Expand All to show the parents and children**





Parent

Child

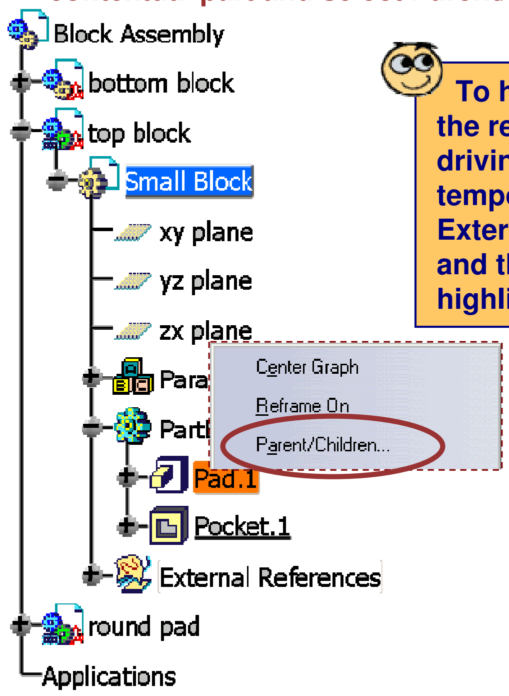
- **Parents drive the part being analyzed**
- **Children are driven by the component being analyzed**

Student Notes:

Inquiring About Parent and Child Elements & Documents

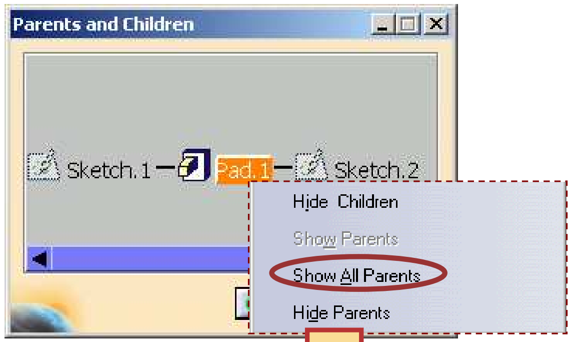
View parent/children to understand the relationships to external elements and documents.

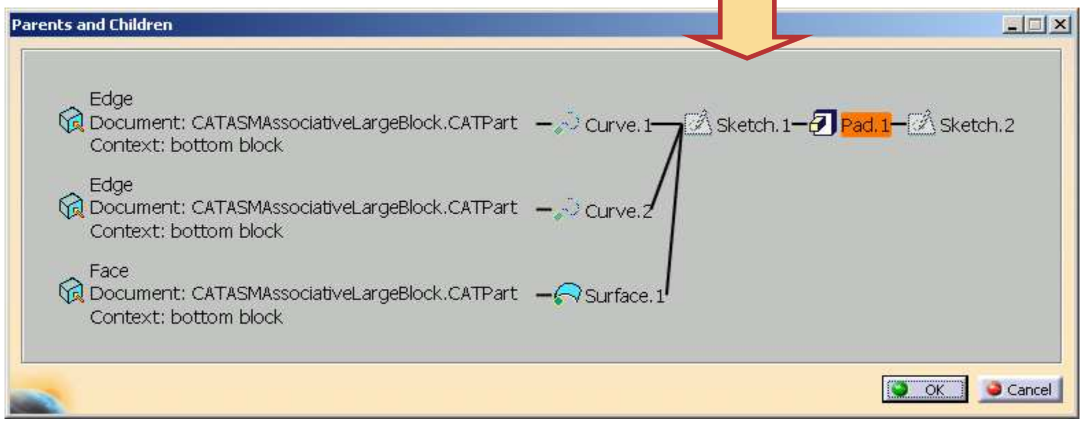
1 Right-click the feature to be analyzed in the contextual part and select Parent/Children…



To help you graphically see the relationship between driving and driven elements, temporarily show (un-hide) External Reference elements and then select elements to highlight them.

2 Right-click on the node of interest and select Show All Parents to see External Reference elements and documents

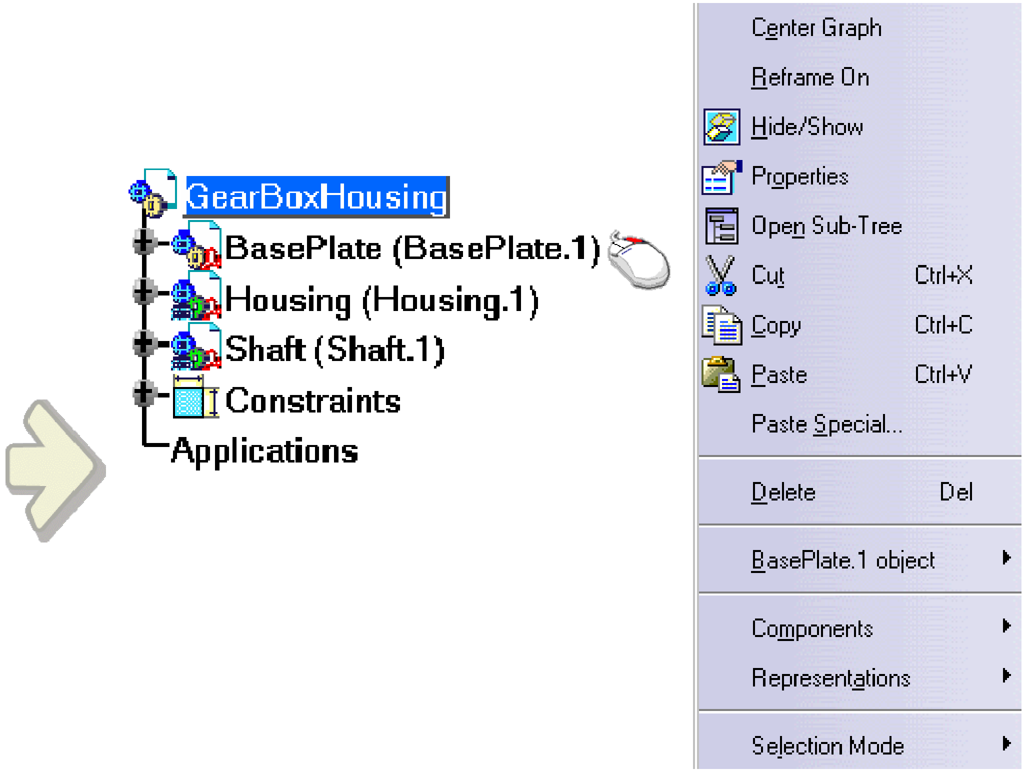




Student Notes:

Deleting Contextually-related Components

You will learn how to delete components that are contextual or that drive contextual parts.

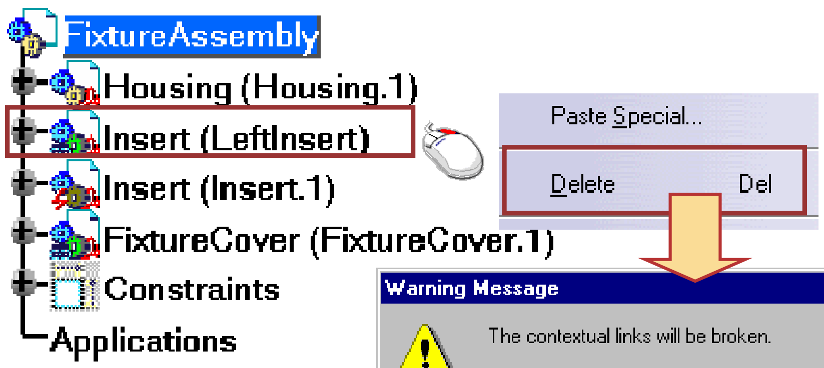


Student Notes:

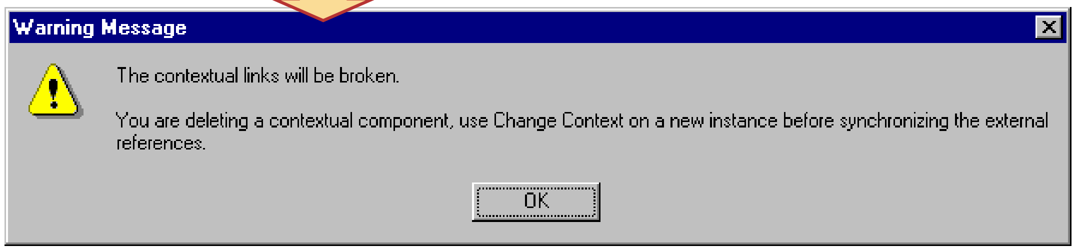
What is Deleting Contextually-related Components? (1/2)

Additional options are available for managing data when deleting components that drive contextual parts or when deleting contextual components.

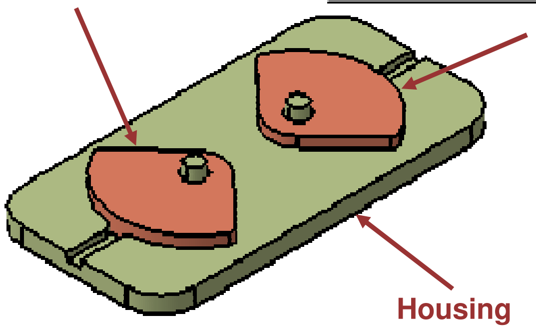
Deleting a driven Part:



Here we are deleting the "original" instance of a contextual part. We are warned to establish another component as the new "original" instance.



Left Insert (Original Instance)



Right Insert (Copy of Original Instance)

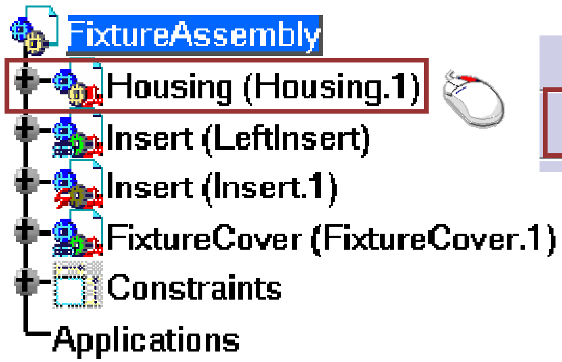
Housing

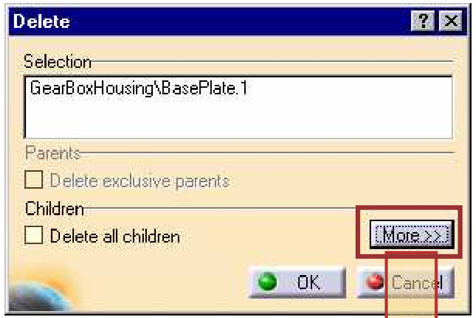
You can change context by using "Define Contextual Links" from contextual menu and define contextual links for a part which is not in context.

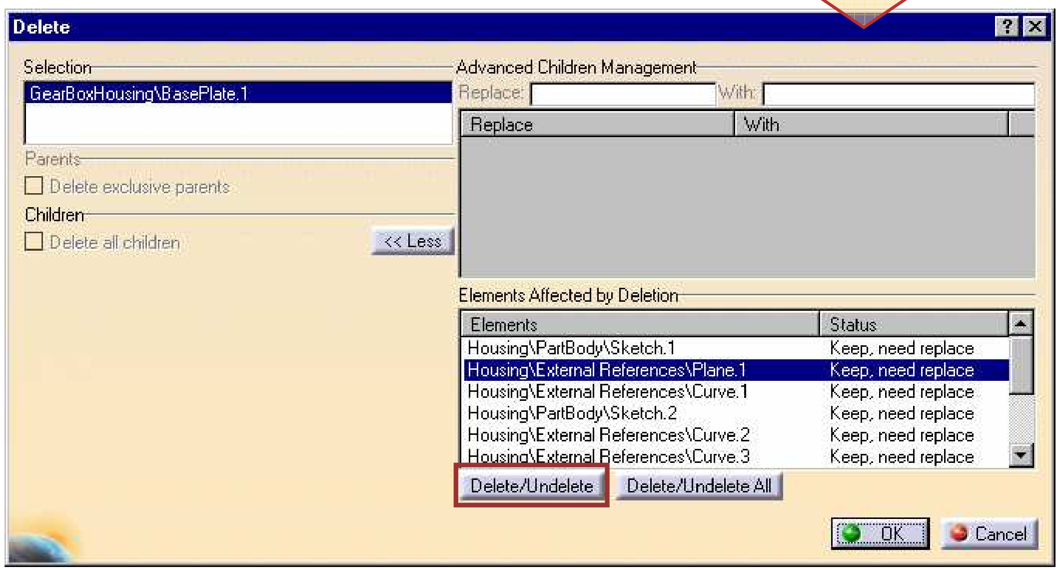
Student Notes:

What is Deleting Contextually-related Components? (2/2)

Deleting a Driving Part







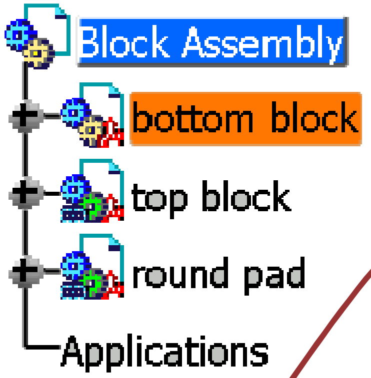
Here we are deleting a component that drives a contextual part. We have the option to delete the contextual components that are driven by the component being deleted.

Student Notes:

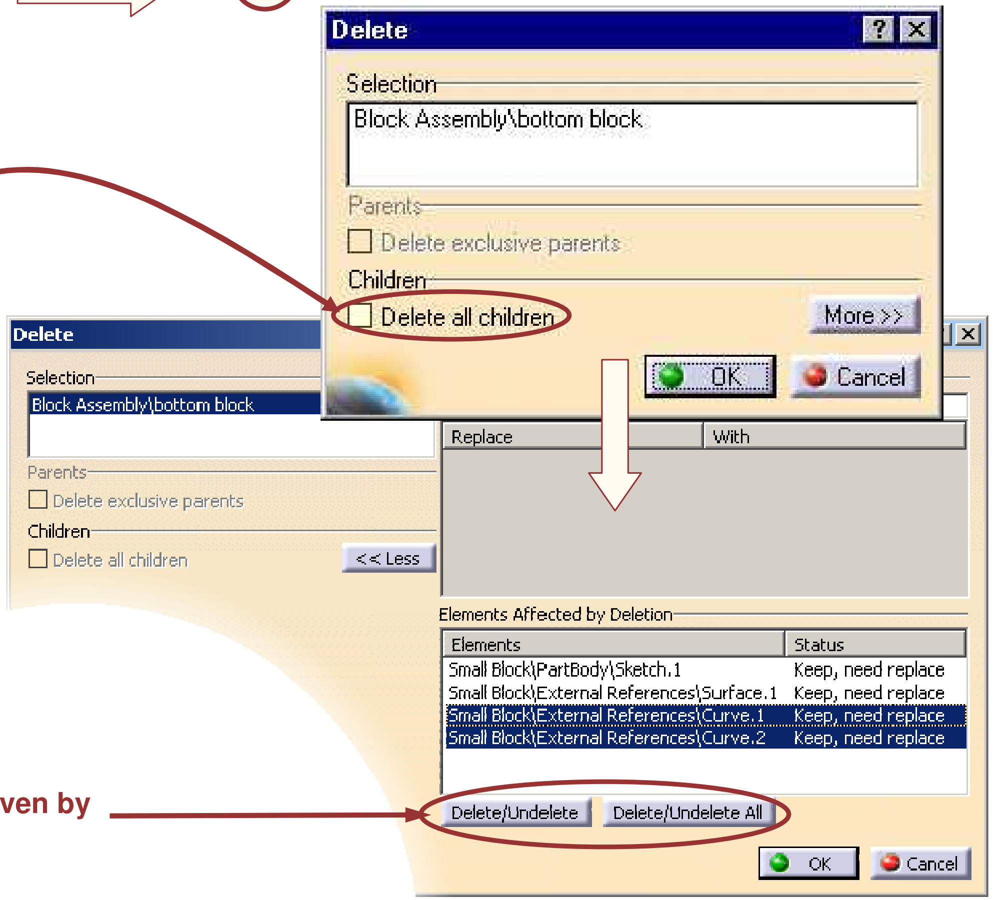
Deleting Driving Components

When deleting a component you will be optionally able to delete any contextual components that it drives because contextual components are children of driving components.

1 Select the component to be deleted and press <DELETE>



2 Press More >>



Note that "Delete all children" is not checked.

3 Specify whether or not to delete:

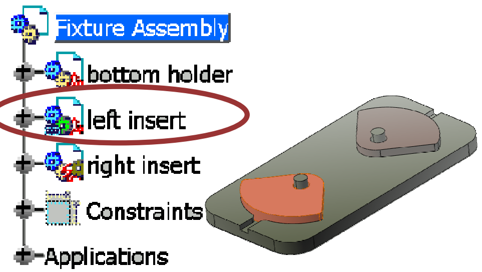
- Assembly constraints if any
- Contextual components that are driven by the component being deleted if any

Student Notes:

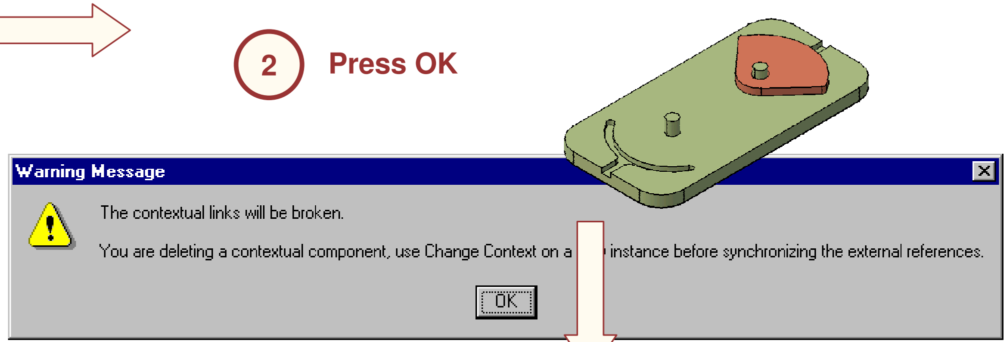
Deleting Contextual Components

After deleting a contextual component you will have to isolate any remaining instances or establish one as the new “original” contextual component.

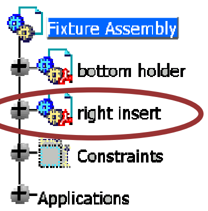
1 Select the component to be deleted and press <DELETE>



2 Press OK

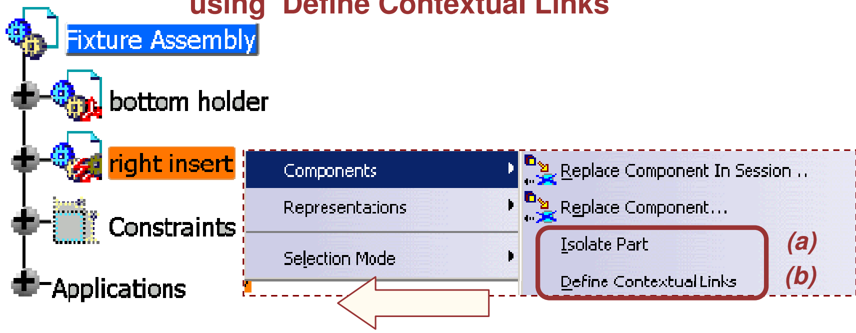


3a

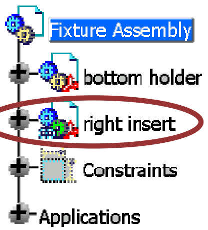


If “Isolate Part” is selected, there are no more links between this component and the other components in the product in which it is instantiated.

3 Isolate remaining instances of the contextual part or define them in context using ‘Define Contextual Links’



3b



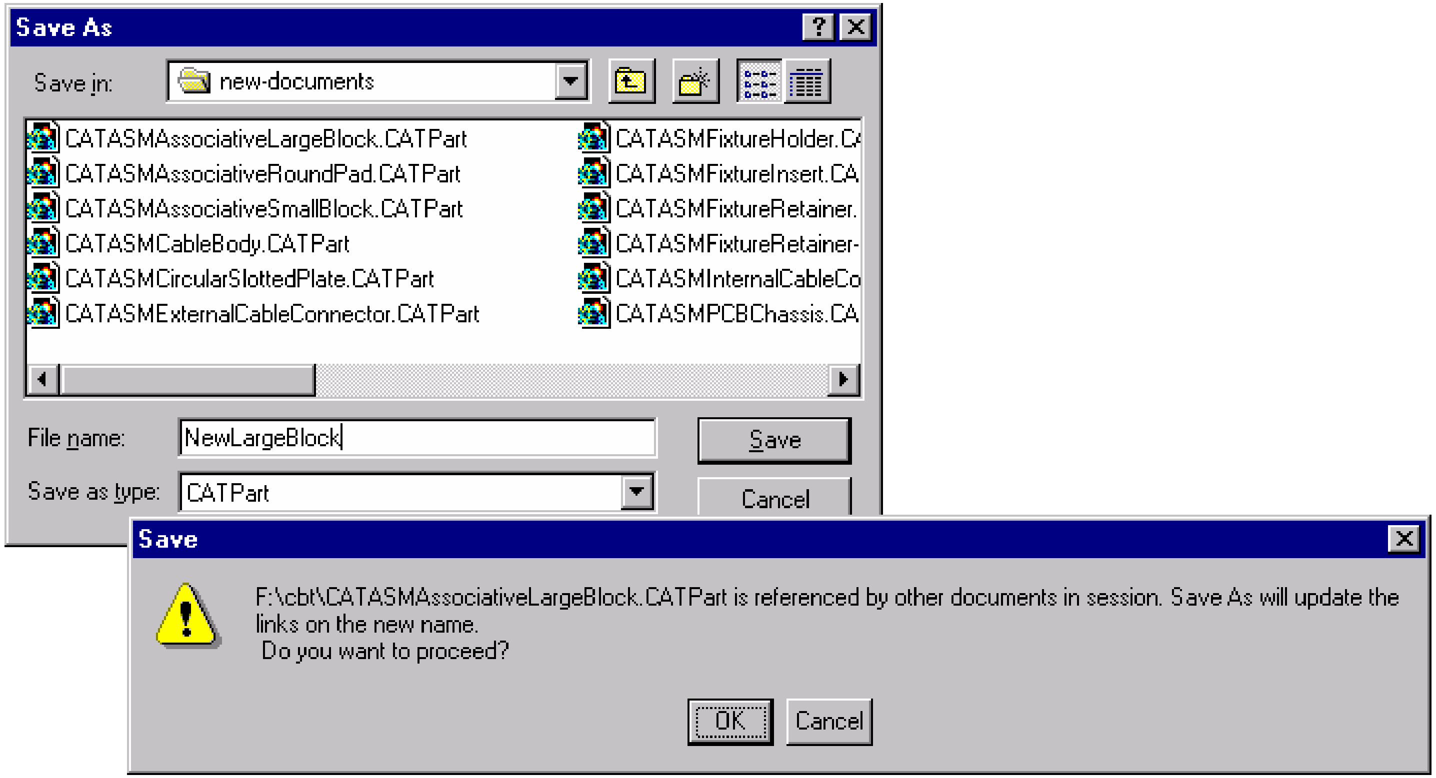
If “Define Contextual Links” is selected, the part becomes as the first instance of a contextual part.

Be careful: regarding on how the part is designed, the result will not be positive

Student Notes:

Saving Contextually-related Documents

You will learn how to save documents that are explicitly or implicitly related to contextual CATParts.



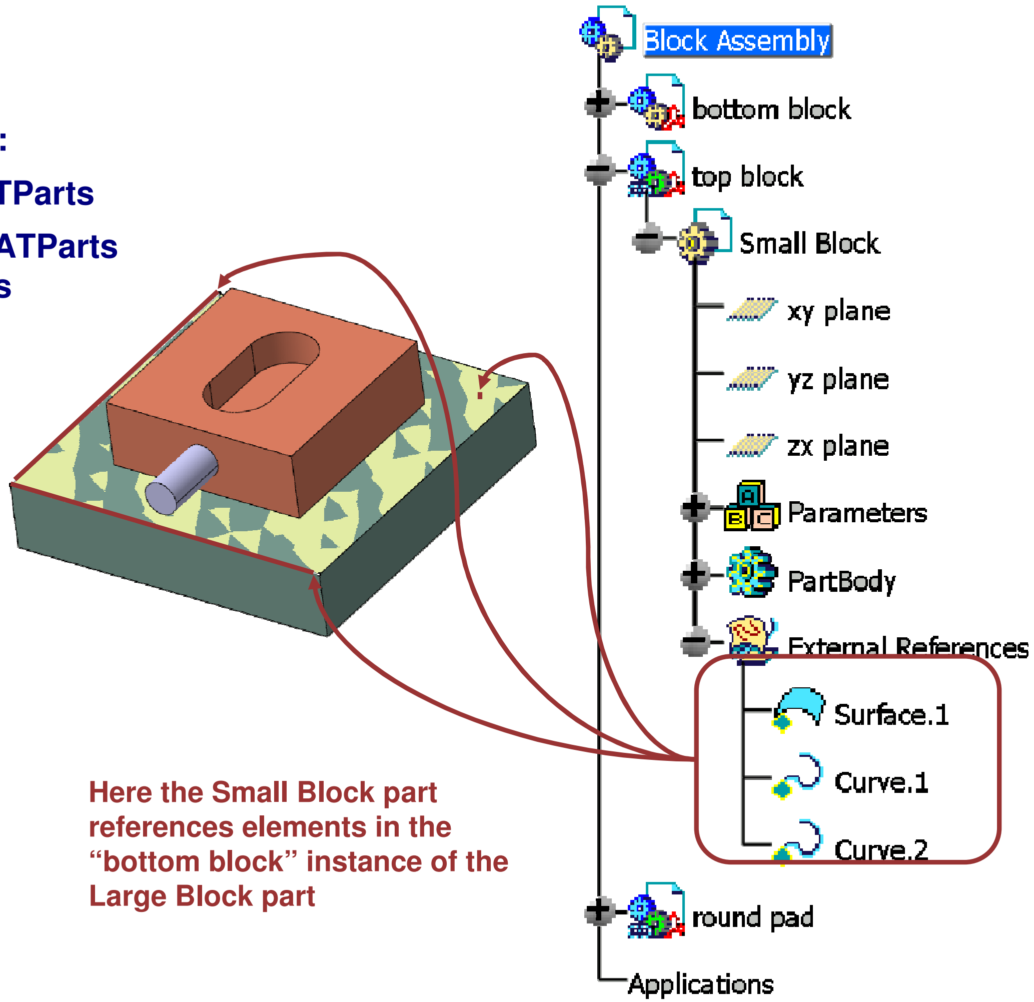
Student Notes:

What Is Saving Contextually-related Documents ?

Special attention is required when saving documents that are explicitly or implicitly related to contextual parts.

Contextual parts reference:

- **Elements in driving CATParts**
- **Specific instances of CATParts in specific CATProducts**



Saving one document with a new file name may require that a related document also be saved

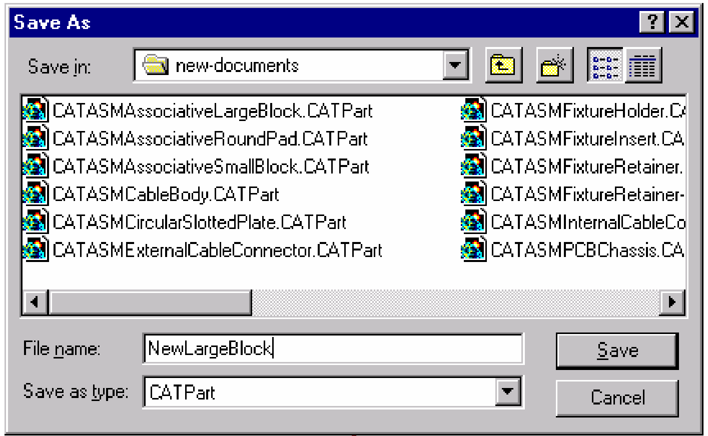
Here the Small Block part references elements in the "bottom block" instance of the Large Block part

Student Notes:

Saving Driving CATParts

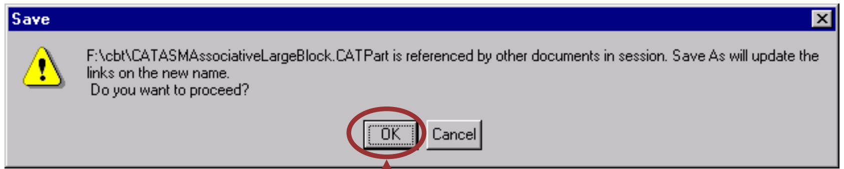
After saving a driving CATPart with a new file name, you will also need to save the driven CATParts and the parent CATProduct because they reference to the driving CATPart file name.

1. Open the assembly that references the CATPart to be Saved As



2. Click on the menu "Save As…" to save the driving CATPart

3. Click OK to proceed with the save



4. Save contextual CATParts that are driven by the CATPart that was Saved As

5. Save the CATProduct that is the parent of the CATPart that was Saved As

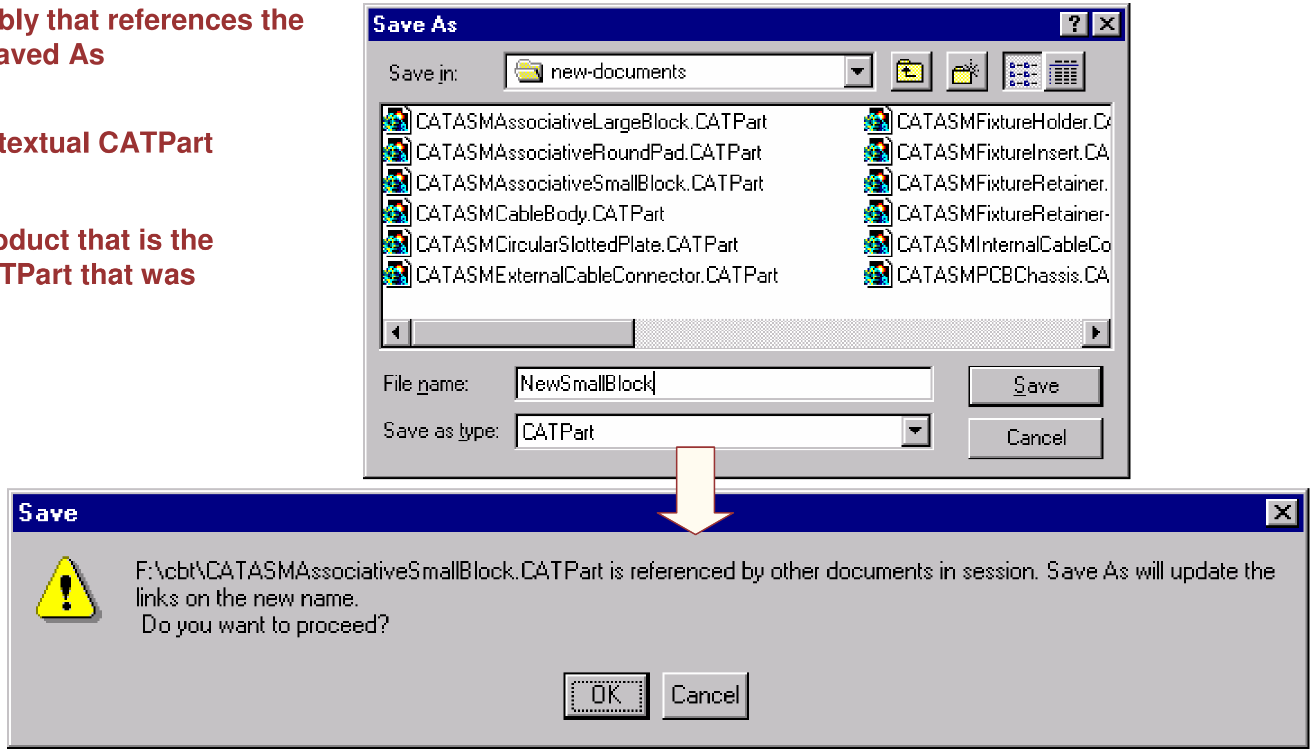
You might find it more convenient to use "Save Management…"

Student Notes:

Saving Contextual CATParts

After saving a contextual CATPart with a new file name you will also need to save the parent CATProduct because it references the contextual CATPart file name.

1. Open the assembly that references the CATPart to be Saved As
2. Save As the contextual CATPart
3. Save the CATProduct that is the parent of the CATPart that was Saved As



4. Click OK to proceed with the save

You might find it more convenient to use "Save Management…"

Student Notes:

Saving Parent CATProducts

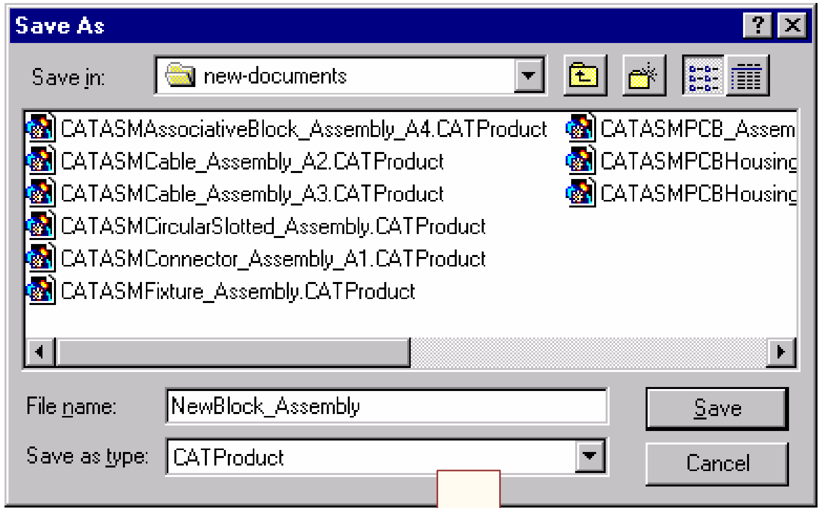
After saving a CATProduct with a new file name you will also need to save contextual CATParts that were defined in-context of the CATProduct because these CATParts reference the CATProduct file name.

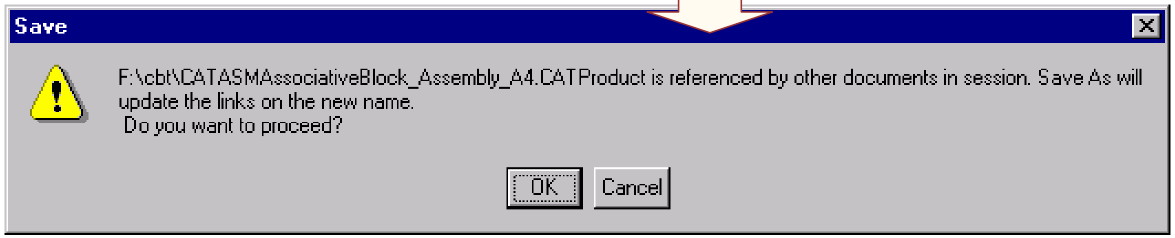
1. **Open the assembly that is to be Saved As** 

2. **Save As the CATProduct**

3. **Save contextual CATParts that were defined in-context of the CATProduct that was Saved As** 

4. **Click OK to proceed with the save**





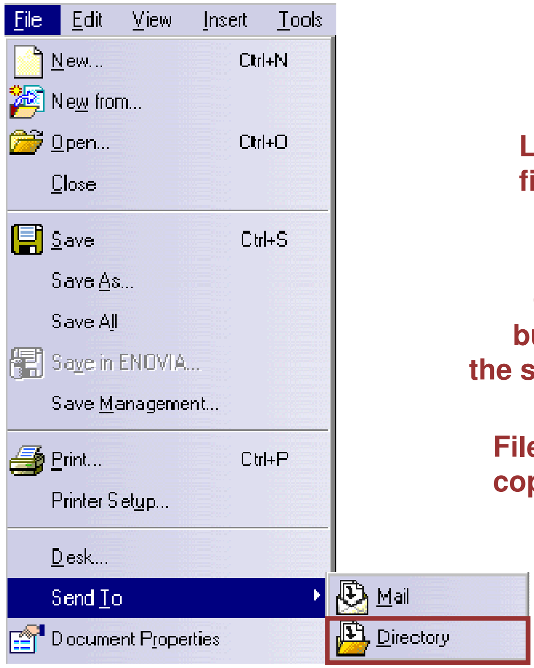
You might find it more convenient to use Save Management

Student Notes:

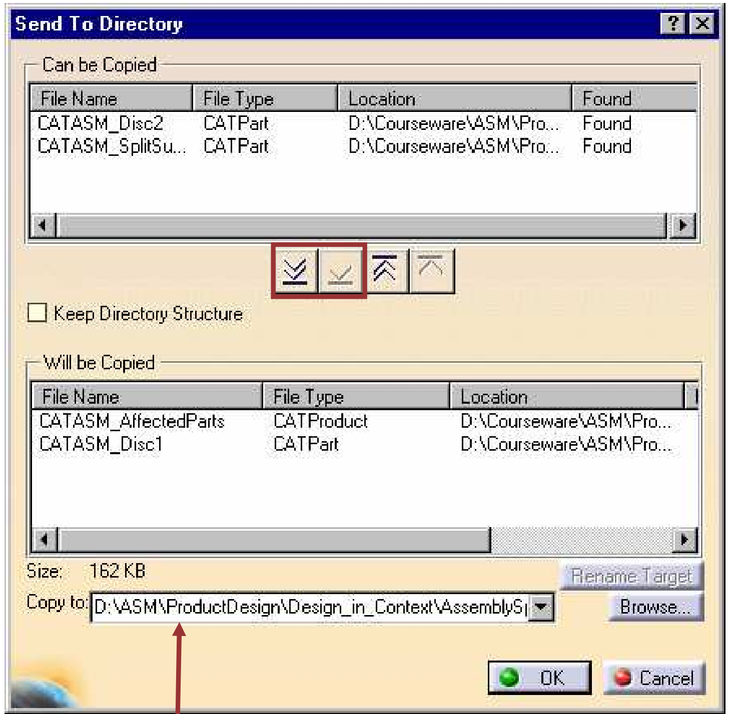
Copying CATProducts Using Send to Directory

You can save the CATProduct and related contextual parts using File > Send to > Directory, to create another copy of the CATProduct with all related files.

1 **Click on File > Send to > Directory from File Menu.**



2 **Select the files to be copied and specify the destination folder where the files will be copied.**



List of available files for copy

Click on these buttons to copy the selected file(s).

Files selected for copy

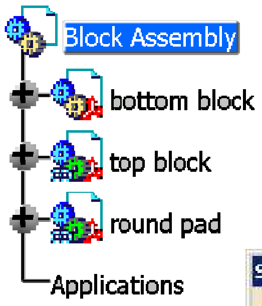
Destination folder

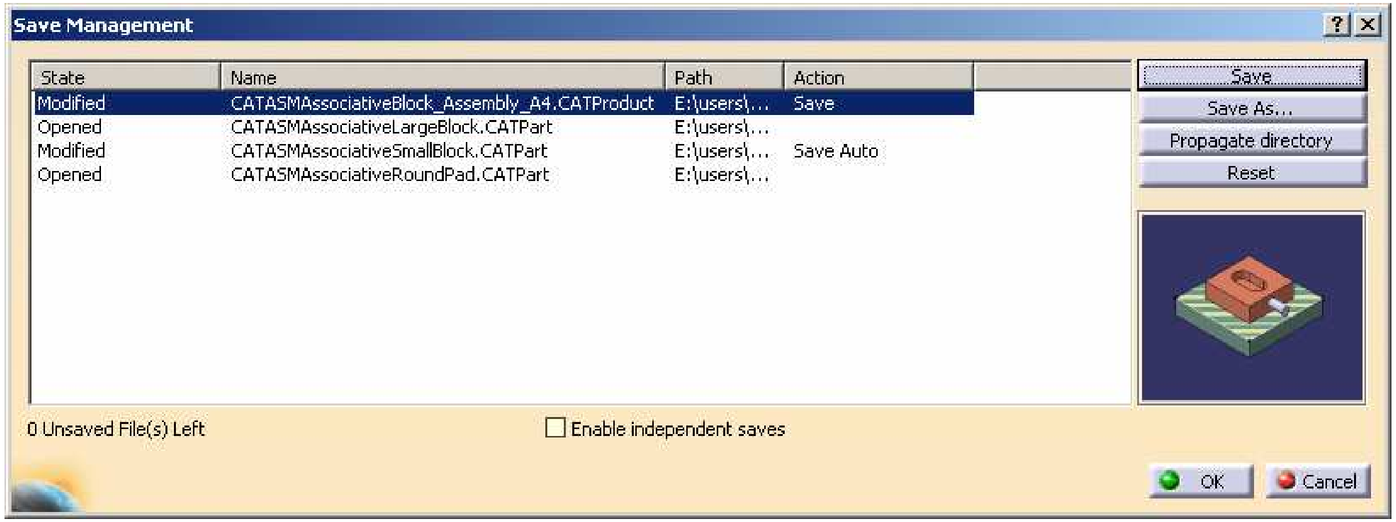
You can copy all files in the source by clicking on "Copy all files" button. 

Student Notes:

Do It Yourself

Product used: CATASMAssociativeBlock_Assembly_A4.CATProduct





1- Save the Large Block CATPart with a new file name. The Small Block will have to be saved too.
2- Save the Block Assembly with a new file name.
3- Close and re-open the CATProduct.
4- View the links in each document using Edit Links to ensure they are correct.

Student Notes:

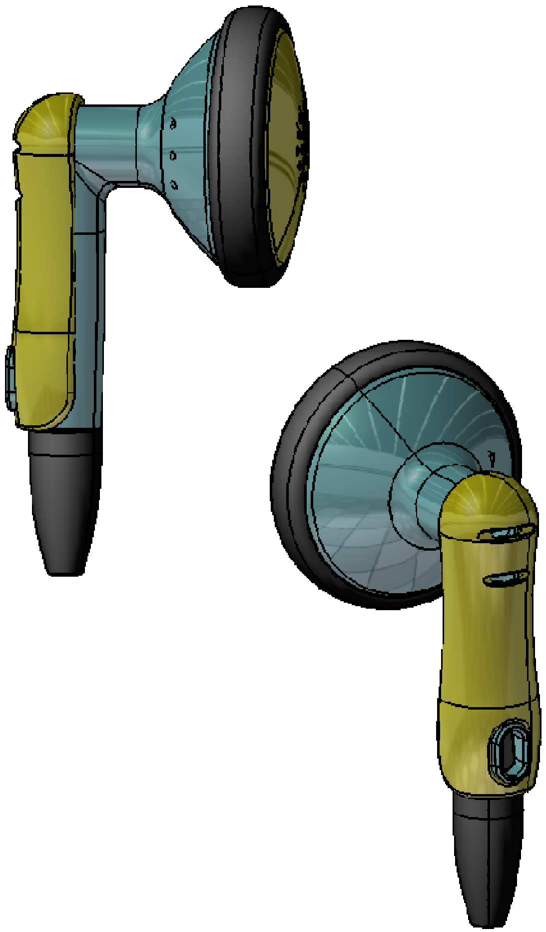
Design in Context

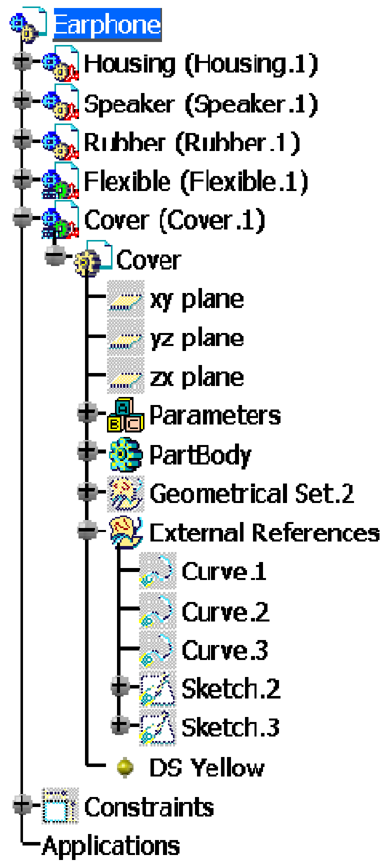
Recap Exercise: Earphone

20 min

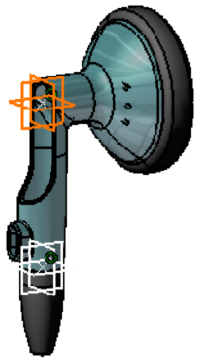
In this exercise, you will:

- Create and position a new part in a Product.
- Design this new part in context:
  - Reusing external elements.
  - Reusing sketches of another part to create sketch-based features.
  - Creating new individual features.

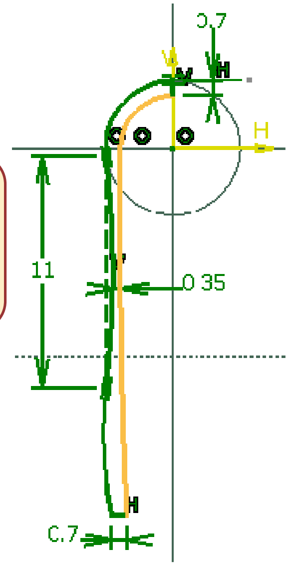




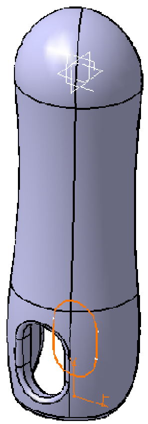
Design Process



1 Insert a new Part in an existing product and position it.

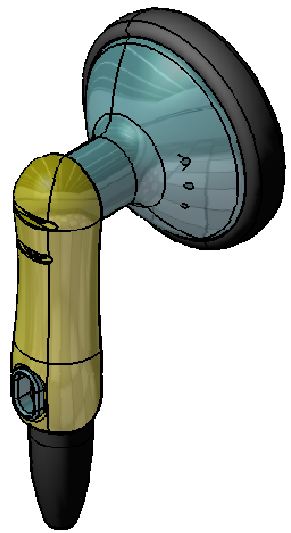


2 Design the first feature by projecting external elements in the sketch.



3 Use external sketches to create sketch-based features.

4 Finalize the design of the component.



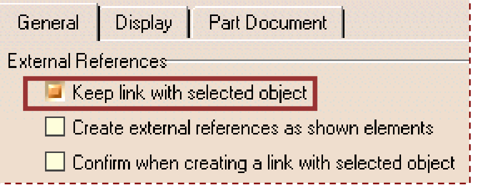
Student Notes:

Student Notes:

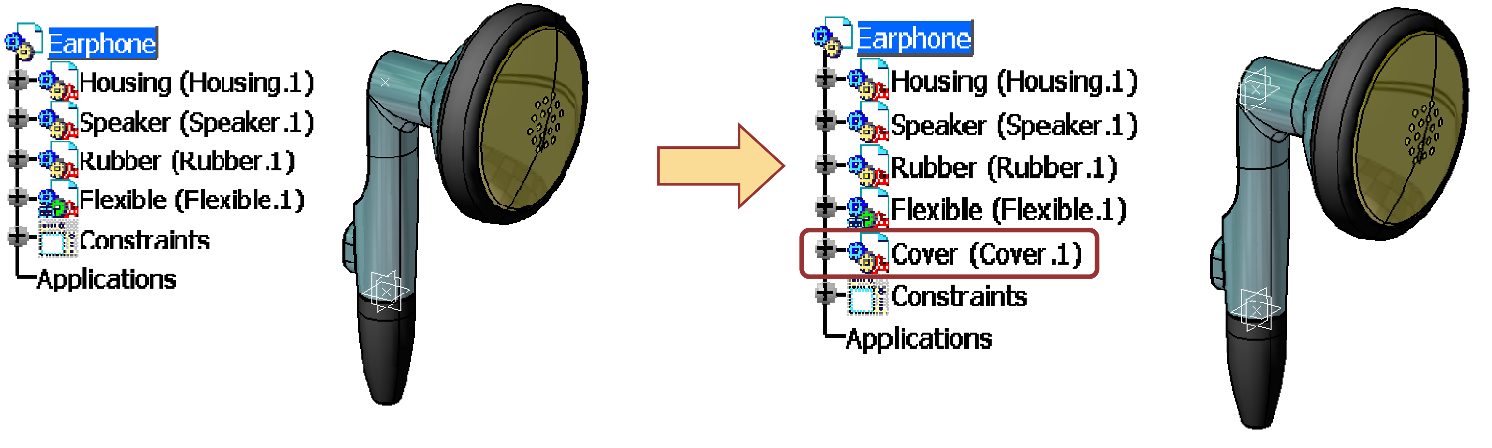
Do It Yourself : Step 1

Product used: Earphone_start.CATProduct

- Ensure that “Keep link with selected objects” option is activated in Tools > Options > Infrastructure > Part Infrastructure.



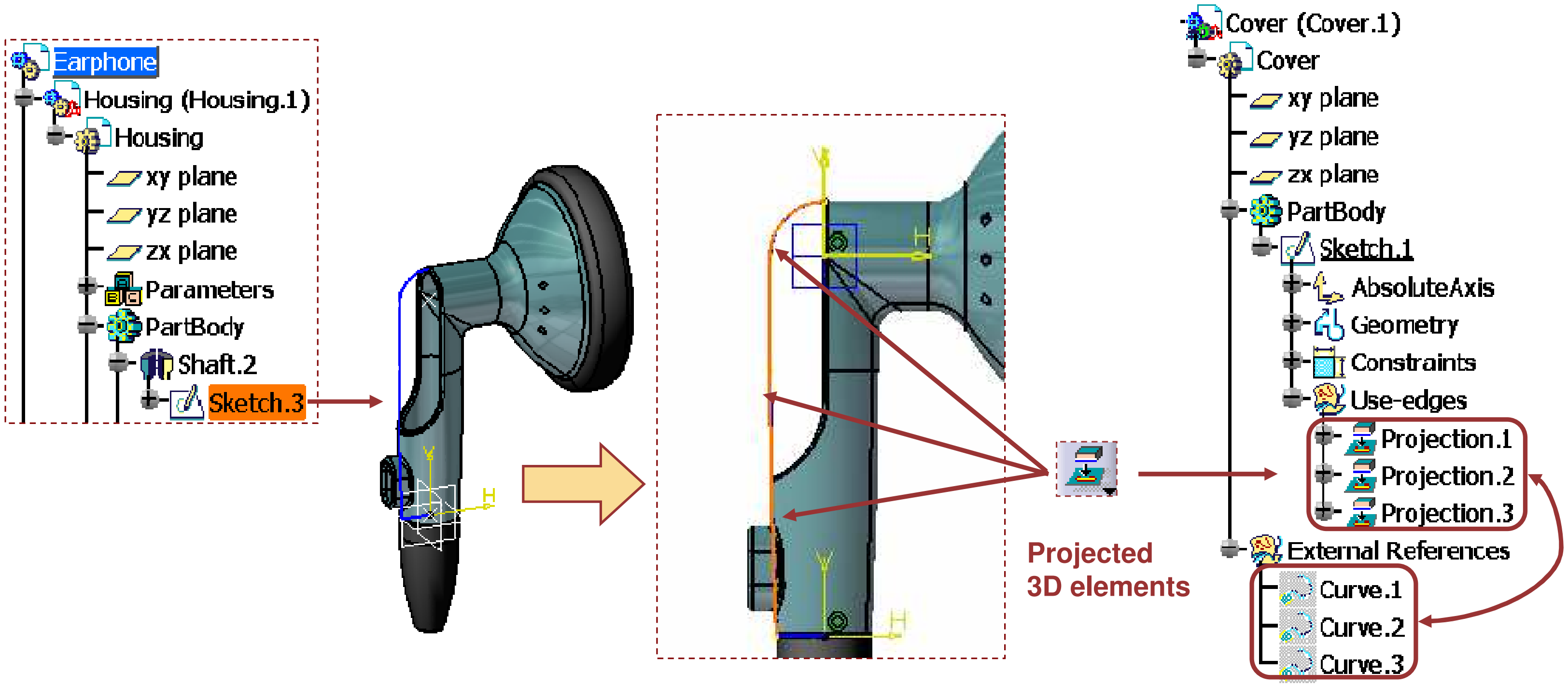
- In the Earphone Product, create a new part called “Cover”.
- Position the new part using reference elements of Housing component in order to center it on Bend_Point.
  - Create a coincidence between “Bend_Point” of Housing and xy plane of Cover
  - Create a coincidence between yz plane of Housing and yz plane of Cover
  - Create a coincidence between zx plane of Housing and zx plane of Cover
- Now that the component is constrained in the product, you can start to design it.



Student Notes:

Do It Yourself : Step 2 (1/3)

- In Housing component show “Sketch.3”.
- In Cover component, create a new sketch on zx plane.
- Project only the three elements of “Sketch.3” in this new sketch. Make sure the link is kept with Housing Component.



Student Notes:

Do It Yourself : Step 2 (2/3)

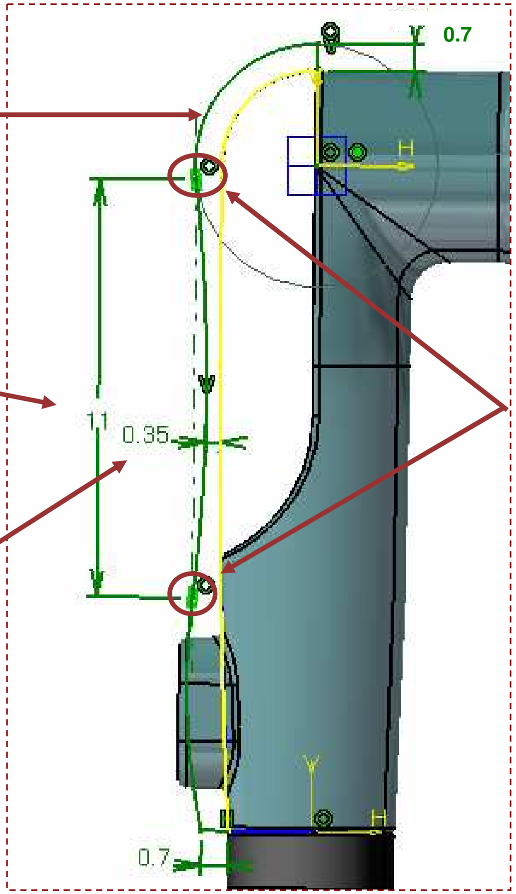
- Sketch the outer profile of the 'Cover' as shown and constrain the sketch as shown.
- Exit the sketcher.

Outer profile of cover is concentric with inner profile

Distance between two tangency points = 11mm

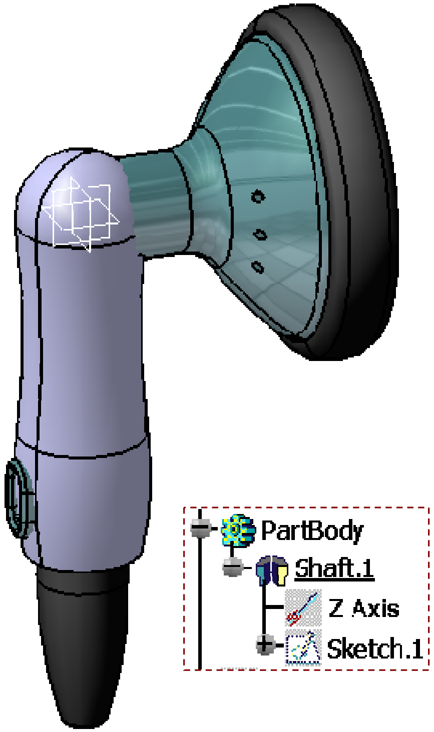
Minimum Distance between outer profile and inner profile is 0.35 mm

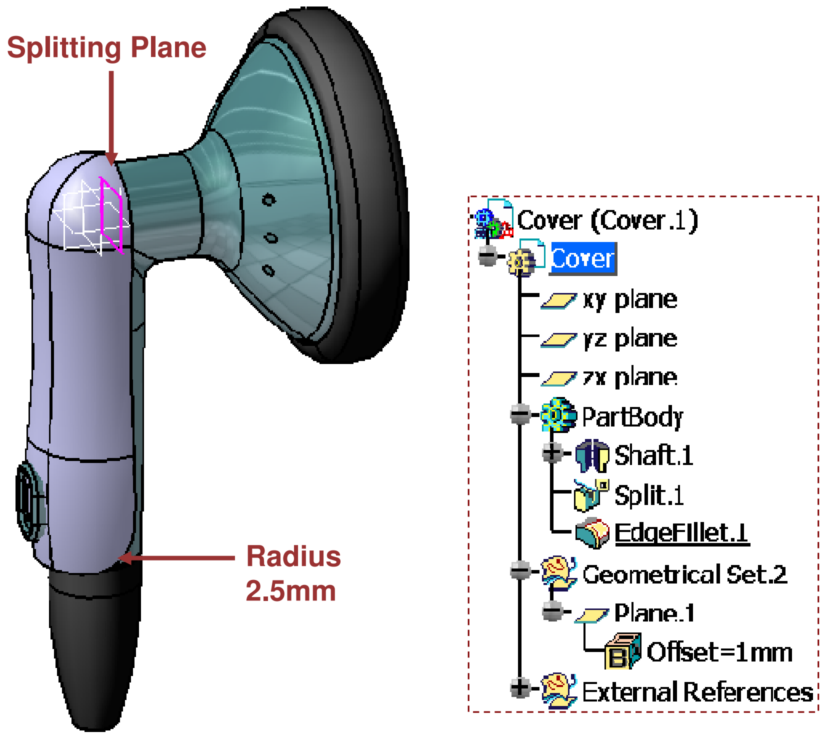
Tangency points are coincident with the vertical line



Do It Yourself : Step 2 (3/3)

- Create a complete Shaft around Z Axis with the sketch previously created.
- Create a new plane defined with an offset of 1mm from yz plane. Reverse its direction if necessary.
- Use this plane to split the current solid. Keep the biggest part of the solid.
- Define an edge fillet of Radius 2.5mm on the two corners of the Cover.



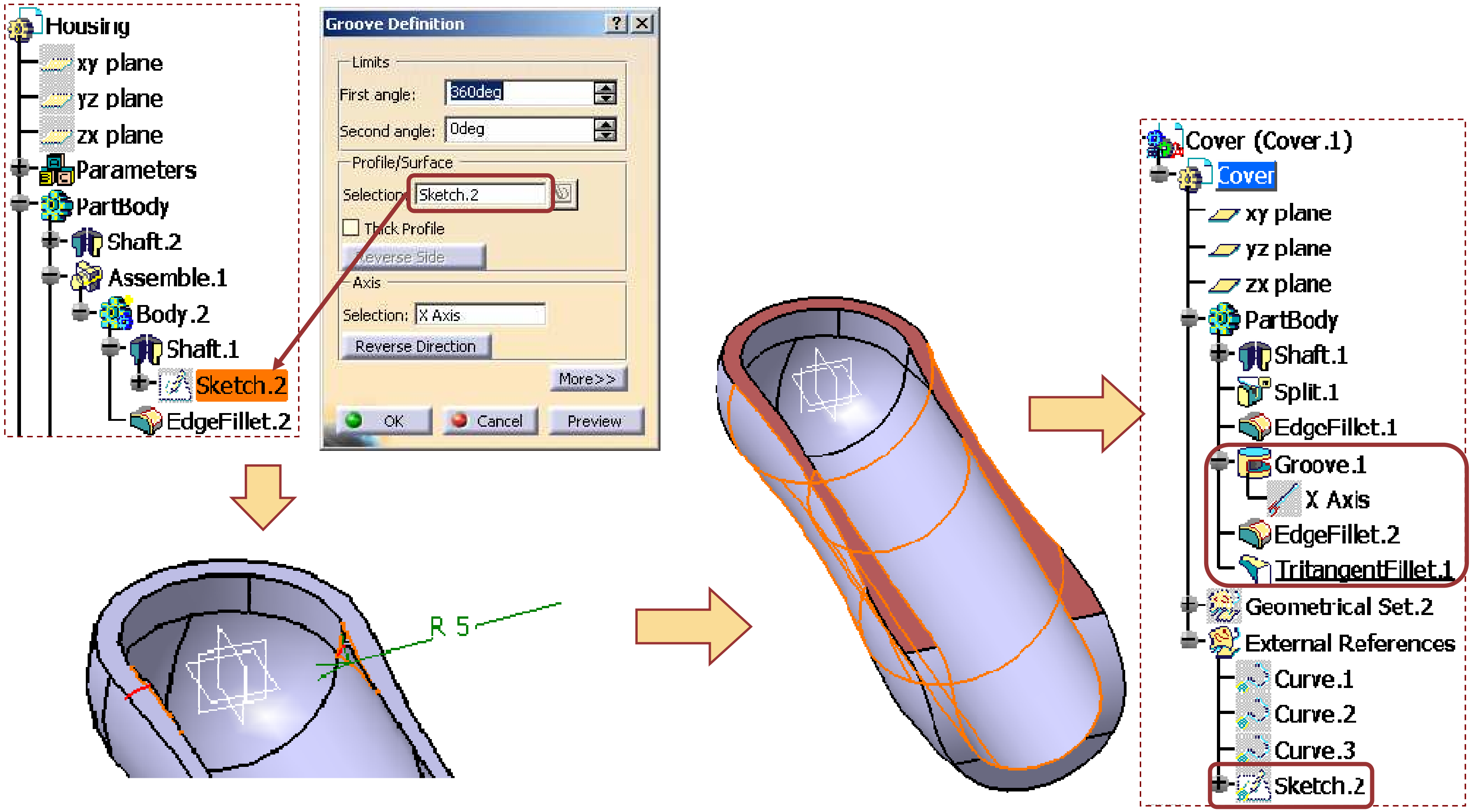


Student Notes:

Student Notes:

Do It Yourself : Step 3 (1/2)

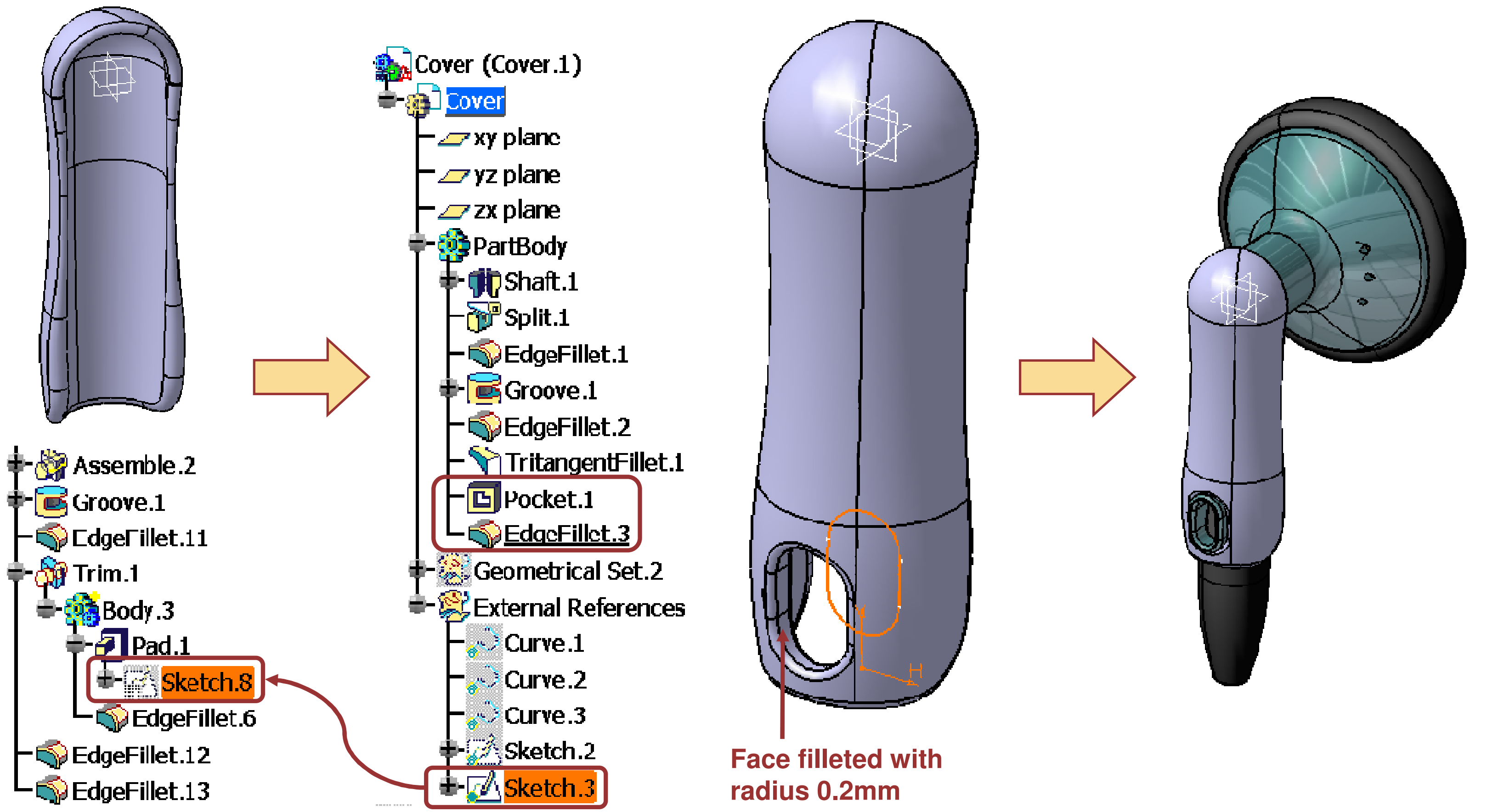
- Create a groove reusing “Sketch.2” (with link) from Housing component.
- Create two edge fillets of 5mm to smooth the edges let by the groove.
- Create a tritangent fillet in order to remove the planar face.



Student Notes:

Do It Yourself : Step 3 (2/2)

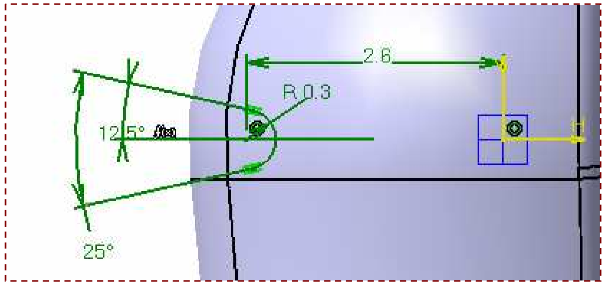
- Create a Pocket (up to last) reusing “Sketch.8” (with link) from Housing component.
- Fillet the inner face of the pocket (Radius = 0.2mm).

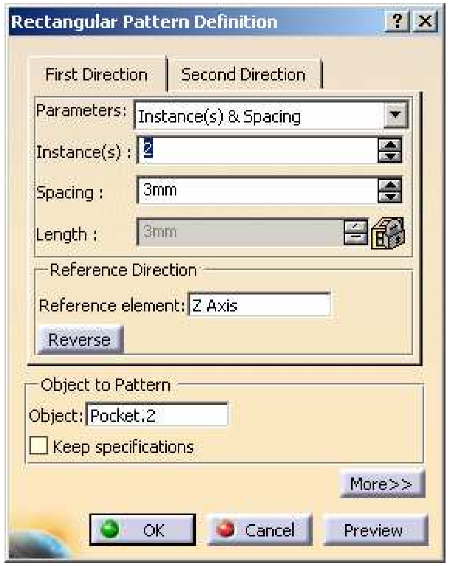


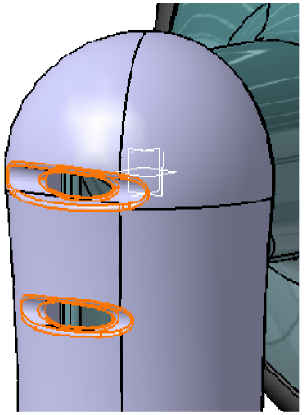
Student Notes:

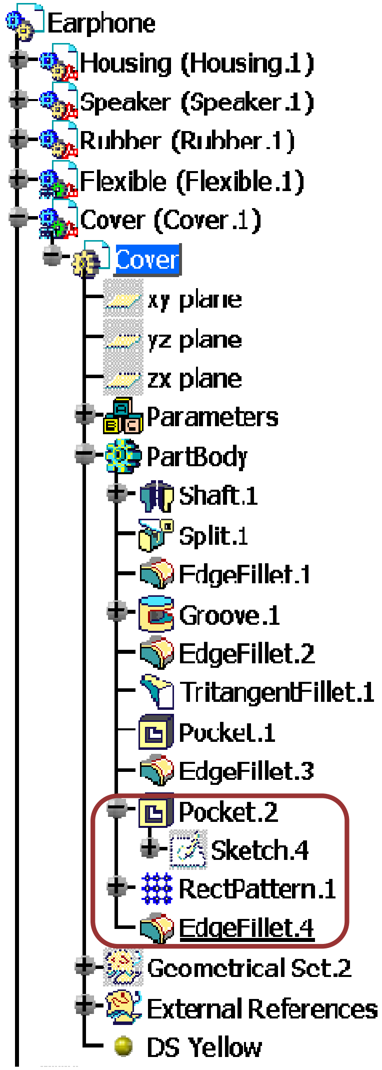
Do It Yourself : Step 4

- Create a new sketch on zx plane. See details below.
- Create a mirrored extent pocket with this sketch.
- Create a rectangular pattern in order to duplicate the pocket.
- Fillet both pockets (Radius 0.1mm)
- You can apply the material of your choice (Painting for instance) on Cover component.





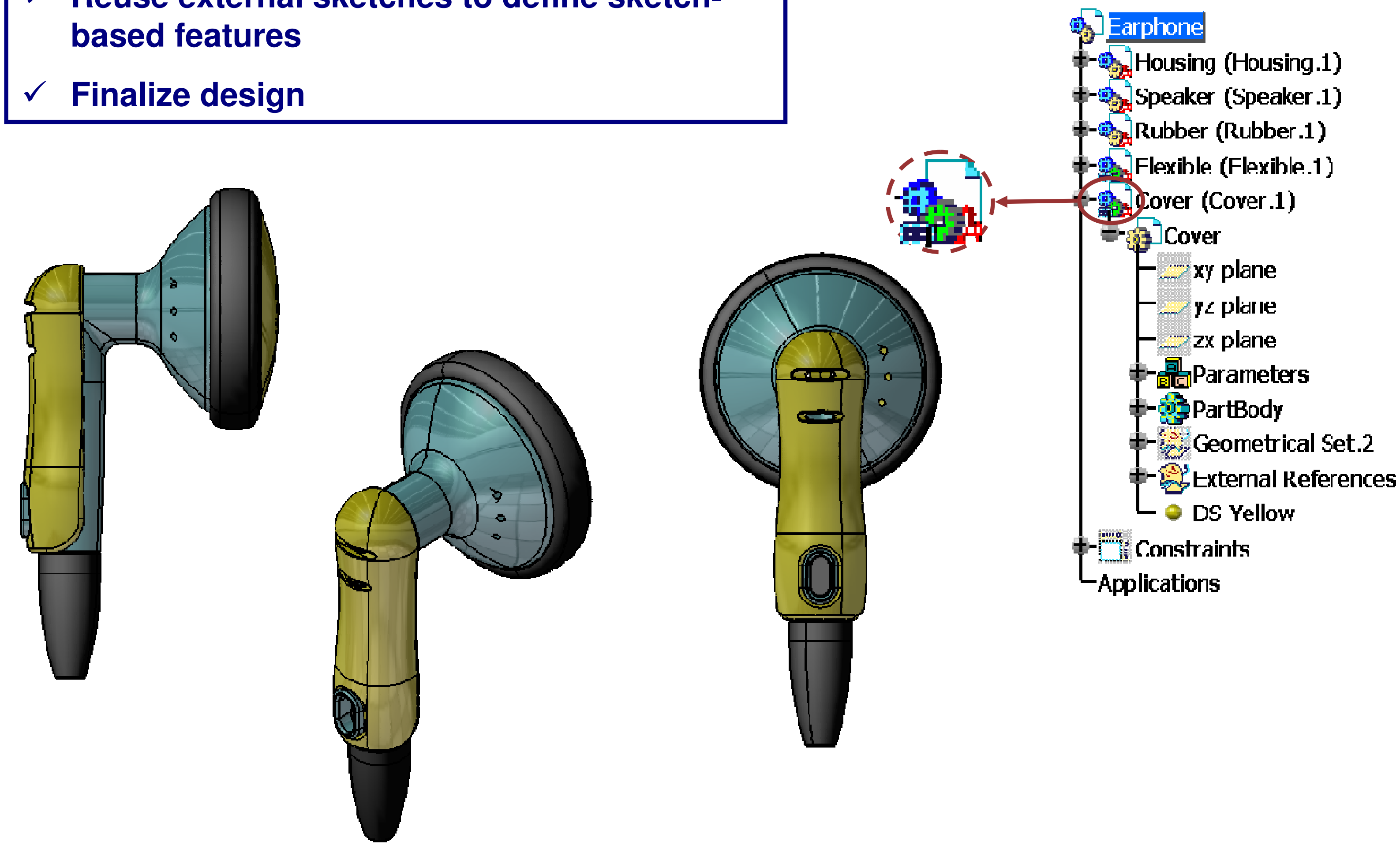




Student Notes:

Recap

- ✓ **Project 3D external elements in a sketch**
- ✓ **Reuse external sketches to define sketch-based features**
- ✓ **Finalize design**



Student Notes:

To Sum Up

In this lesson you have seen how to design contextual parts in an assembly. You have also seen how to:

- **Create Contextual Parts : Contextual parts can be created by reusing external geometrical elements and / or published parameters of other parts.**
- **Edit Contextual Parts : Editing a driving part impacts the changes in the contextual parts whereas editing driven part impacts only the contextual part and its non contextual instances .**
- **Create Assembly Features: You have seen how to create Assembly Split, Hole, Add and Remove features.**
- **Manage Contextual Parts: You have seen how to isolate, save, delete and analyze contextual parts.**

Student Notes:

Creating and Using Published Geometry

In this lesson you will learn what is published geometry and in which conditions it can be used

- Introduction to Publishing Geometry
- Creating Published Geometry
- Using Published Geometry
- Replacing Published Components
- Recap Exercise: Webcam
- To Sum Up

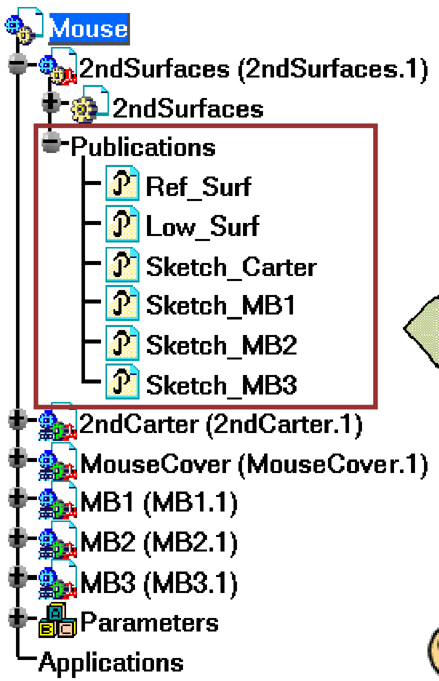
Student Notes:

Introduction to Publishing Geometry

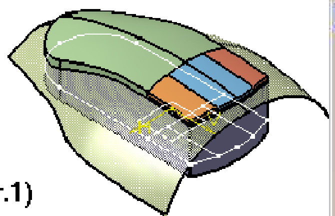
Publishing geometrical elements is the process of making geometrical features available to different users. This operation is very useful when working in assembly design context.

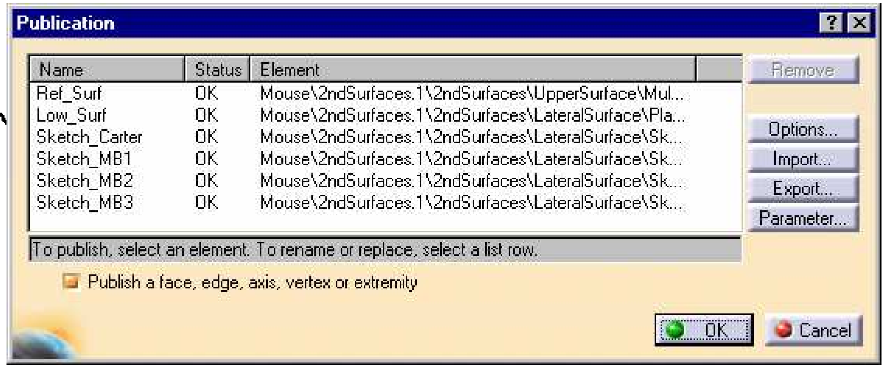
Publishing Geometry is also useful while constraining various components of an assembly.

It is useful when we replace components especially when the component being replaced is involved in assembly constraint or drives other component.



In this example, the pad of part 2ndCarter is limited by published elements Ref_Surf – a top surface and Low_Surf – bottom plane. Hence this pad is contextually linked to 2ndSurfaces.

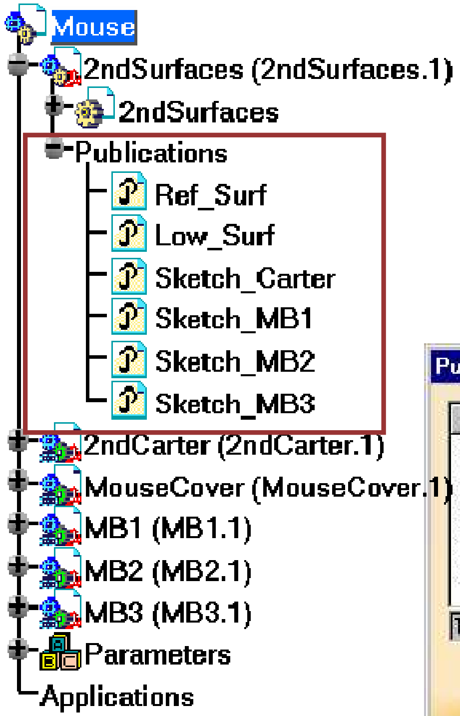


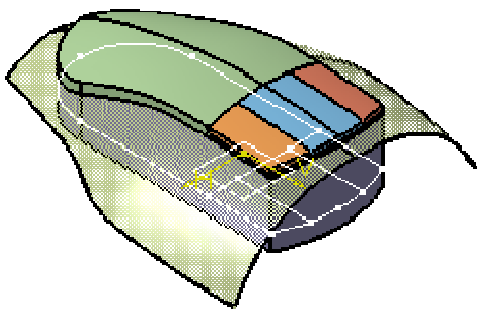


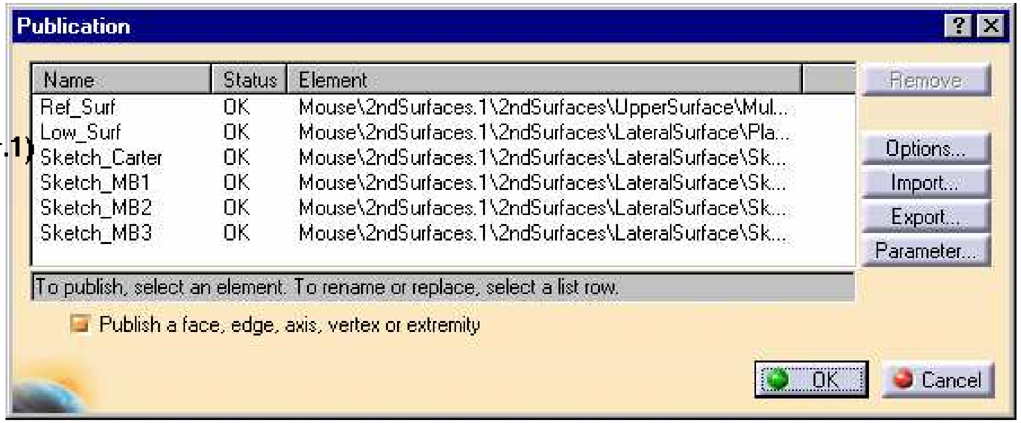
In this lesson, we will learn how to publish geometry, but you can publish parameters as well.

Creating Published Geometry

You will learn how to publish geometric elements of components







Student Notes:

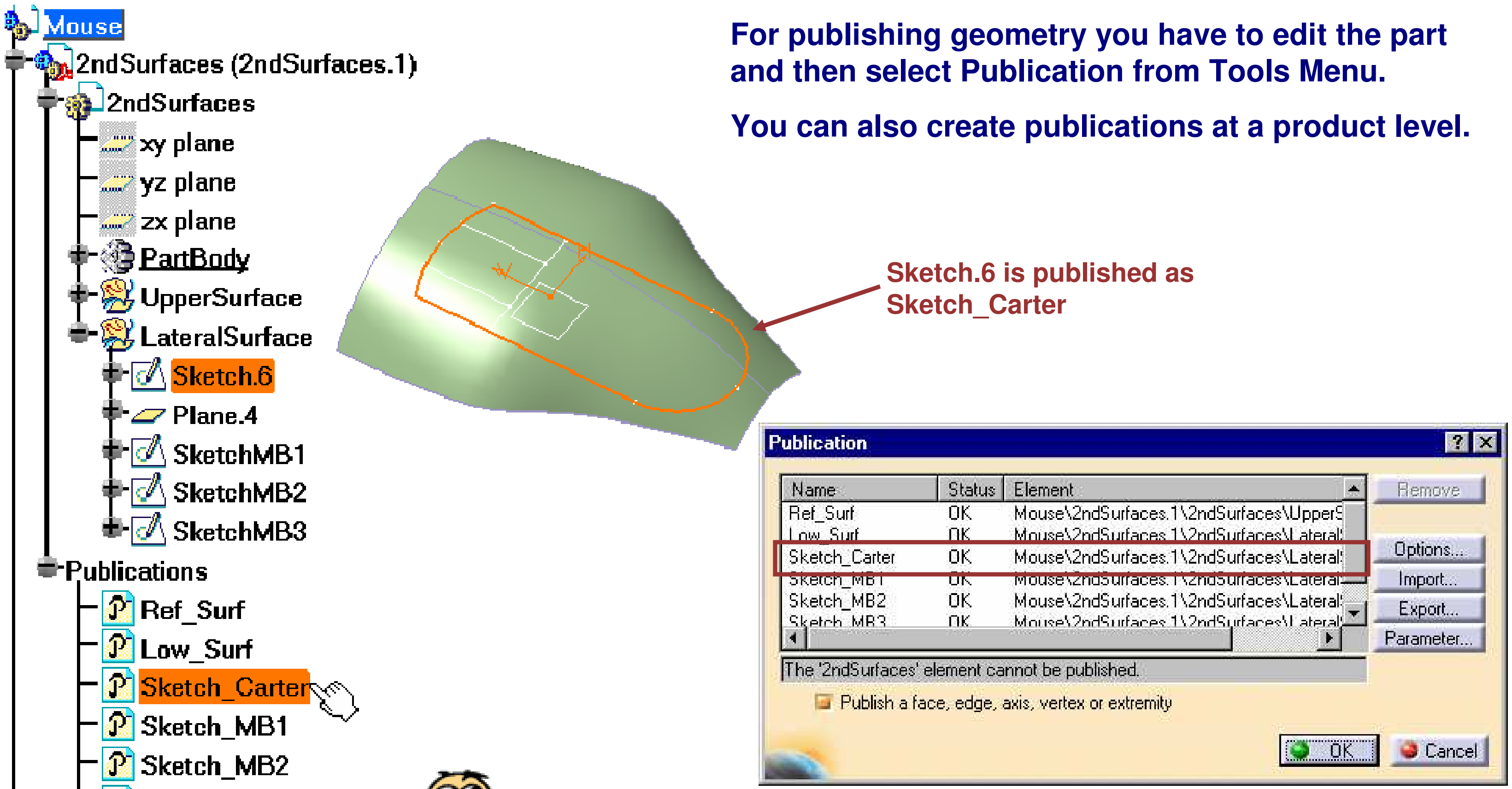
Student Notes:

What is Publishing Geometry ?

Publishing geometry of a component means associating a name to it so it will be recognized by other documents.

For publishing geometry you have to edit the part and then select Publication from Tools Menu.

You can also create publications at a product level.



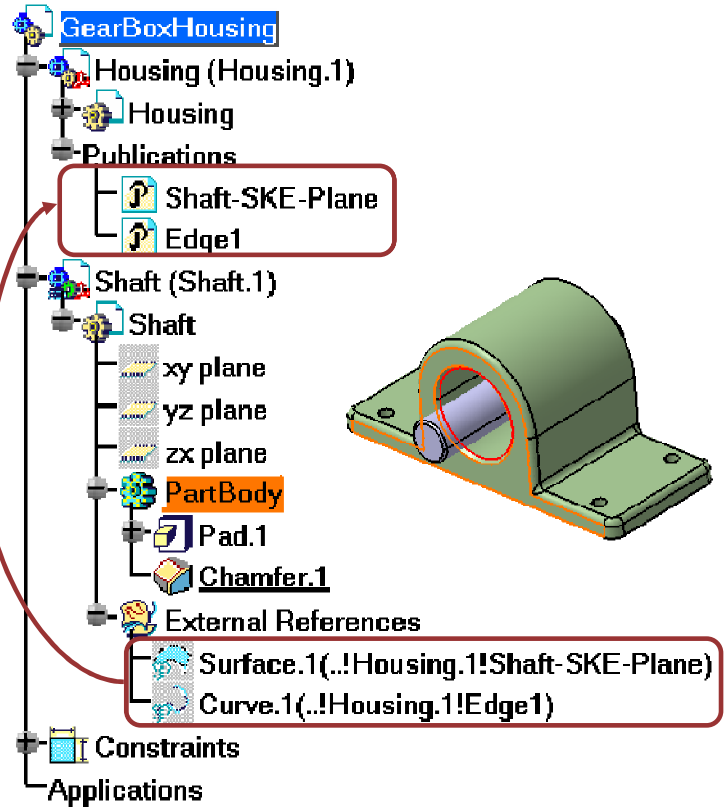
The published elements are visible only in specification tree. You can select the pointed geometrical element by selecting the publication in the specification tree.

Student Notes:

Why Publish Geometry ?

Publishing geometry has many benefits:

- **To label geometry and give it a name that can easily be recognizable (particularly in case of publishing edges, faces, etc.)**
- **To make some particular geometry easily accessible from one part of the tree.**
- **An option allows you to select as external reference only the published elements. In this case, Publication permits you to pre-select the elements that you allow to use as external references.**
- **To ease replacement of one component of the assembly by another. Published elements that have the same names are automatically reconnected, as you would have to reconnect them all one by one if they were not published.**



What Kind Of Geometry Can Be Published ?

Many types of geometries can published :

- **Wireframe features (Points, Lines, Curves, Planes)**
- **Whole sketches**
- **Bodies (Part Body, other body)**
- **Part Design features (pad, pocket, hole,etc.)**
- **Generative Shape Design Features (Extrudes Surfaces, Offsets, Joins etc.)**
- **Free Style Design features (Planar Patches, Curves etc.)**
- **Sub Elements of all Geometrical Elements (Faces, Edges, Vertices etc.)**

Student Notes:

Copyright DASSAULT SYSTEMES

Student Notes:

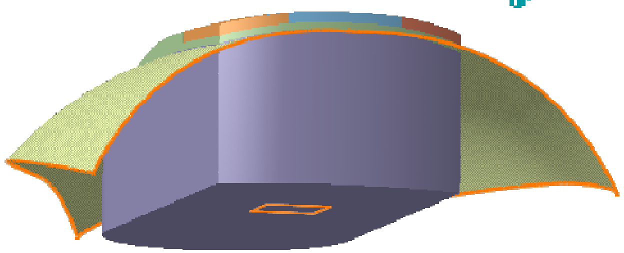
Published Elements in the Tree

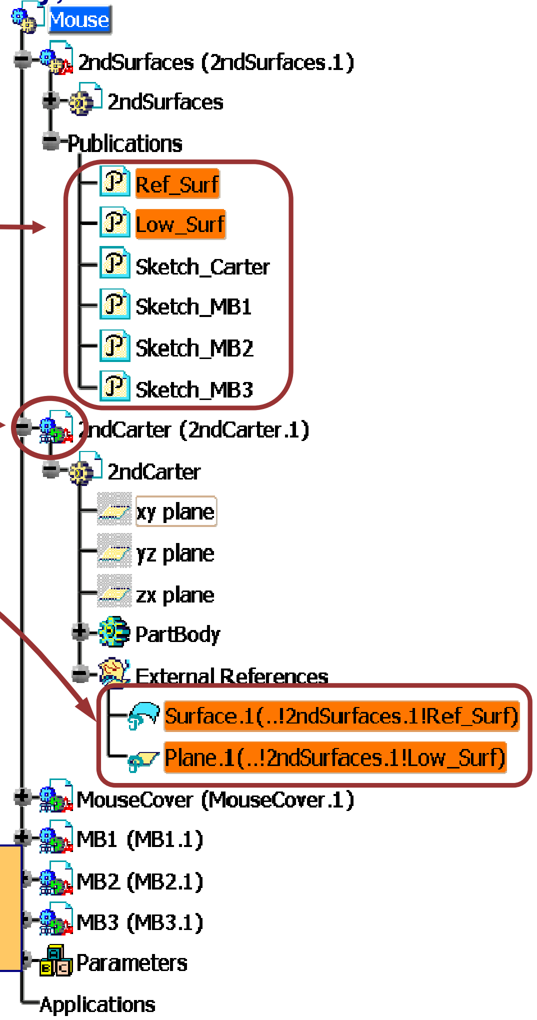
The tree displays names of published elements of a component under its Publication node. When an external reference is connected to a published geometry, it is shown also.

Here are the published elements of the component "2ndSurfaces"

The green gear signifies the "original" instance of a part that is contextual (driven by another part)

Copies of external geometry that are synchronized with published geometry are signaled with the capital P.





Capital P will be green when the link to external geometry is updated ( )
and be replaced by this symbol when not synchronized ( )

Student Notes:

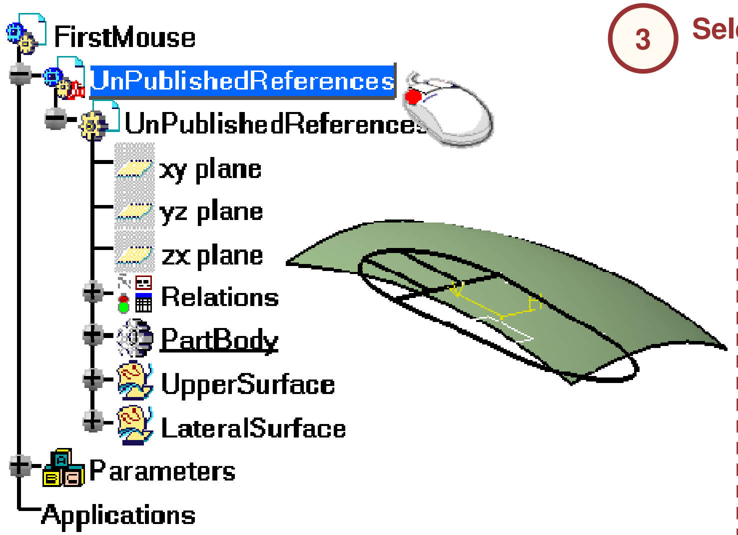
Publishing Geometry (1/3)

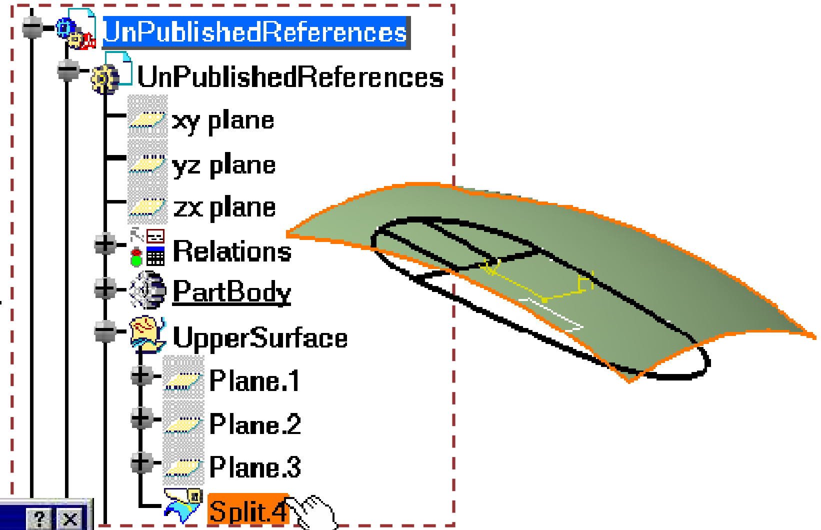
Publication will concern the active component and is available both in Assembly Design Workbench and Part Design Workbench

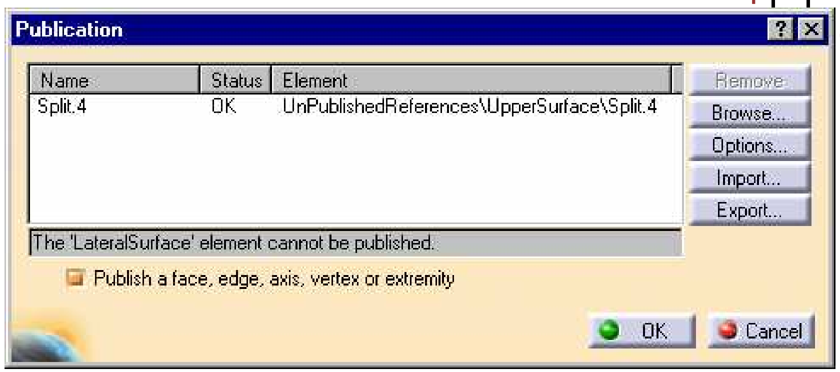
1 Activate the component in which you want to publish geometry

2 Select "Publication…" in Tools Menu

3 Select geometric element you want to publish

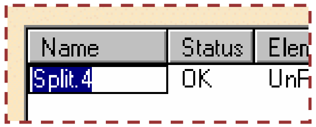






4 As soon as selected, the element is added in the list of published geometry.

To modify its published name, select its row then its name cell



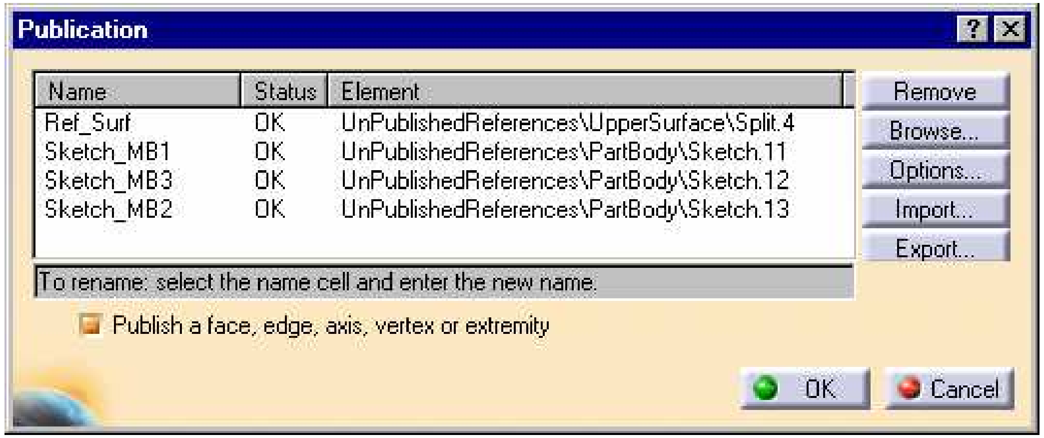
Student Notes:

Publishing Geometry (2/3)

You can publish as many elements as you want.

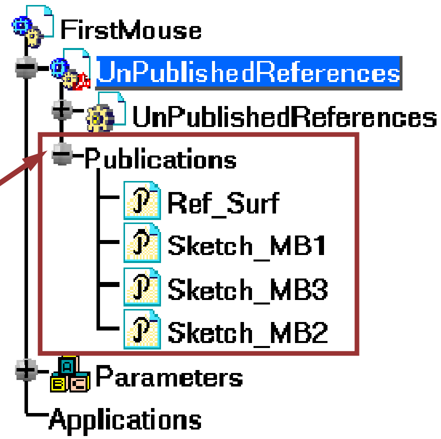
5 Key the name you want to associate to the selected element

6 Repeat step 3 to 5 to publish other elements



7 Click on OK to validate

It is not mandatory to publish all the geometry in one shot, you can come back later to the Publication of the component and add some other published geometry

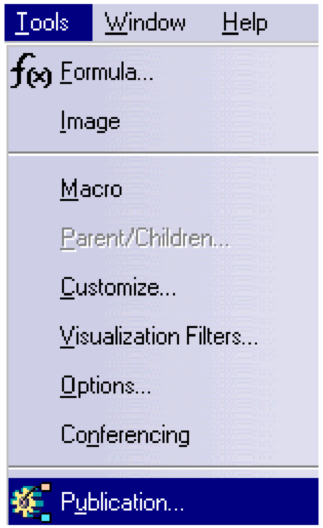


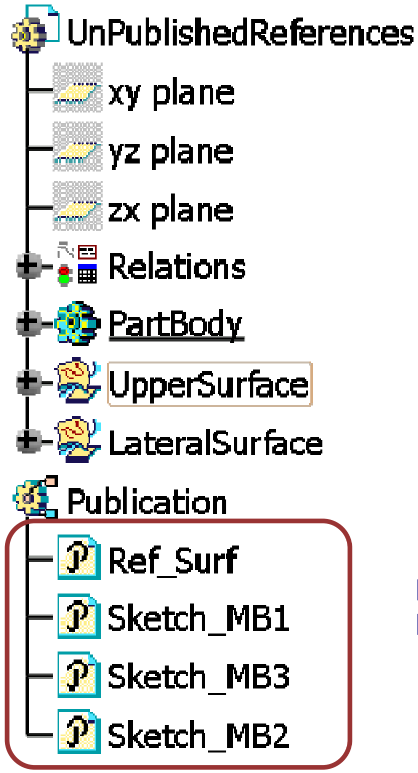
Published geometry is displayed under Publication node of the component

Publishing Geometry (3/3)

Concerning part components, you can as well publish their geometry from Part Design Workbench as from Assembly Design Workbench

Part Design Workbench Tools Menu





Published Geometry in Part Specification tree

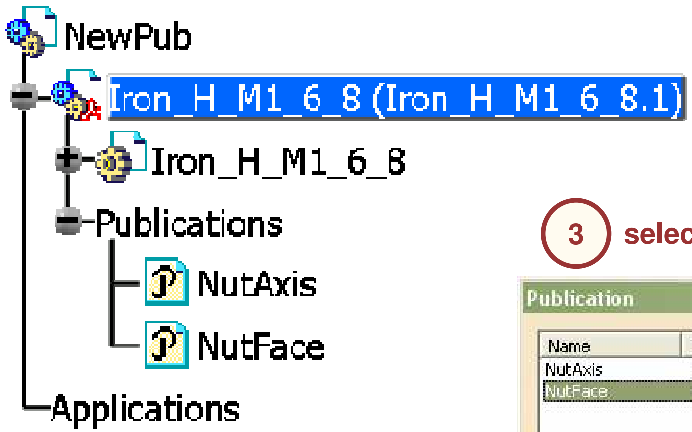
Student Notes:

Student Notes:

Changing a Published Element

You can also replace the published element by another one

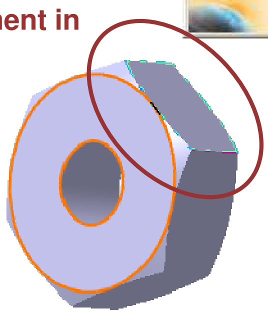
1 Edit the Part or component on which you want to change the published geometry



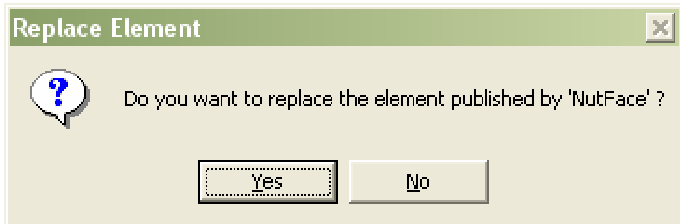
2 Select “Publication” in Tools Menu

3 select the row

4 select the new element in geometry



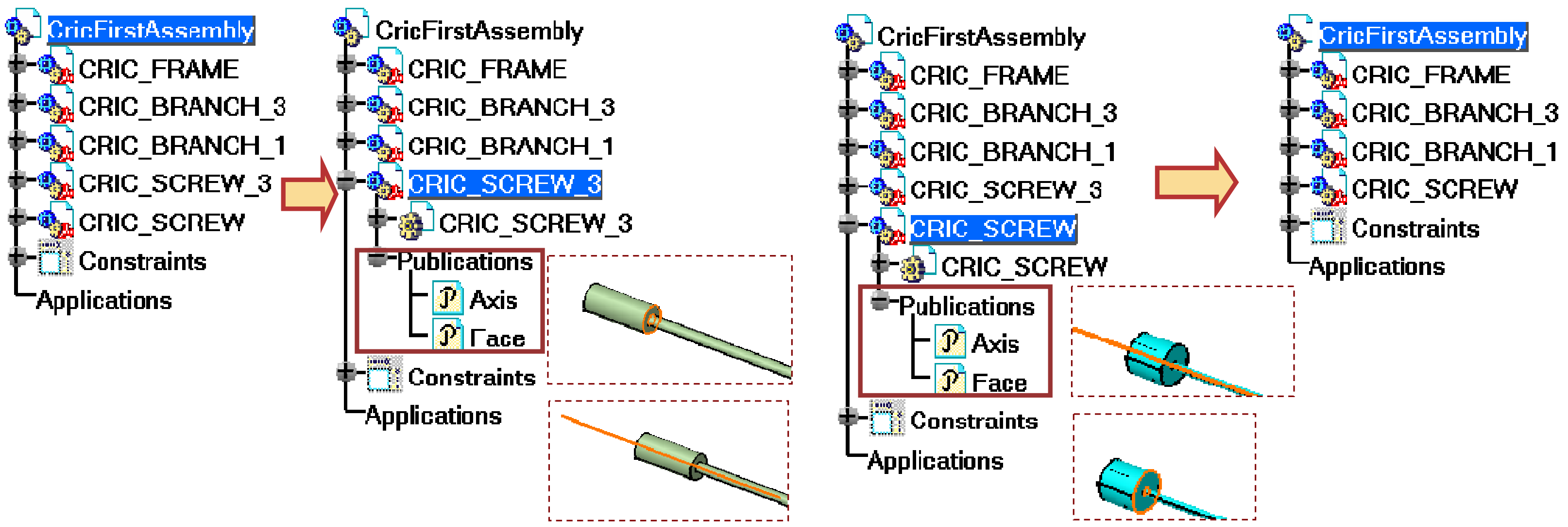
Click “Yes” to validate



Student Notes:

Do It Yourself (1/2)

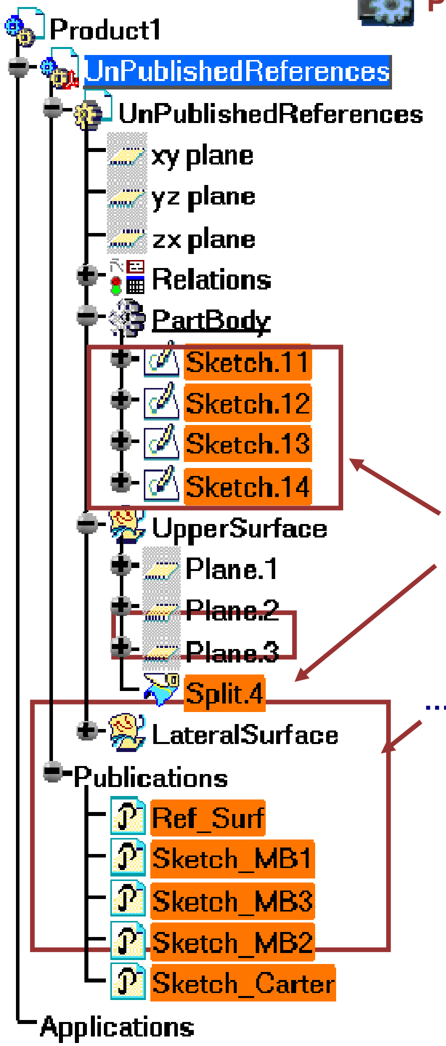
Documents used: "CATASMJAPublishingGeometryDoIt.CATProduct", "CRIC_SCREW3.CATPart", "CRIC_SCREW1.CATPart"



- **In "CATASMJAPublishingGeometryDoIt.CATProduct", insert "CRIC_SCREW3.CATPart" and "CRIC_SCREW1.CATPart"**
- **Activate "CRIC_SCREW_3" component and publish two of its geometric elements as below and save the active component as "PUB_CRIC_SCREW3.CATPart"**
- **Activate "CRIC_SCREW" component and publish two of its geometric elements as opposite and save the active component as "PUB_CRIC_SCREW1.CATPart"**
- **Remove "CRIC_SCREW_3" from the root assembly and save the product under the name of your choice**

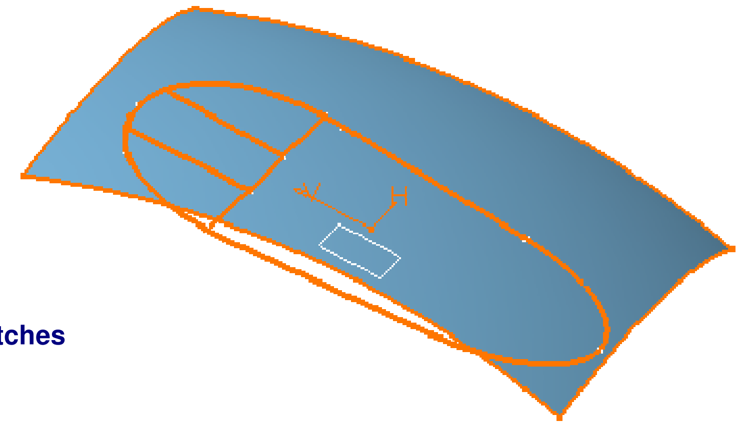
Do It Yourself (2/2)

Parts used: "UnPublishedReferences.CATPart"



Those highlighted sketches and feature have to be published under…

…those names



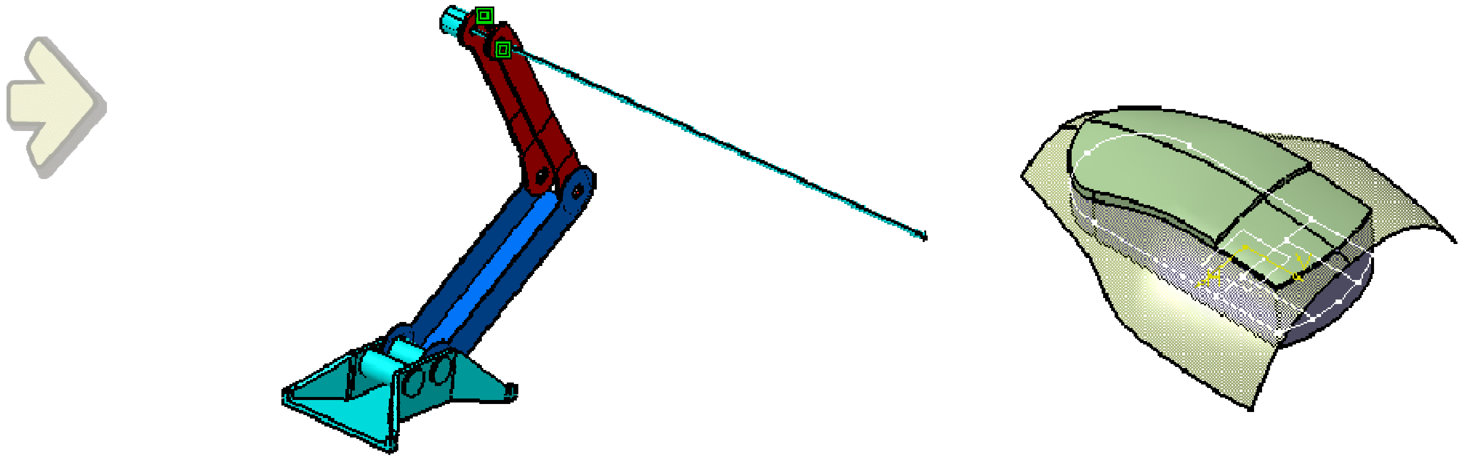
- **Insert "UnPublishedReferences.CATPart" in an empty CATProduct**
- **Activate "UnpublishedReferences" Component and Publish its geometry as shown above**
- **Save the Active Document as "2ndPublishedReferences.CATPart"**
- **You can close the Product without saving it because it is useless now**

Student Notes:

Student Notes:

Using Published Geometry

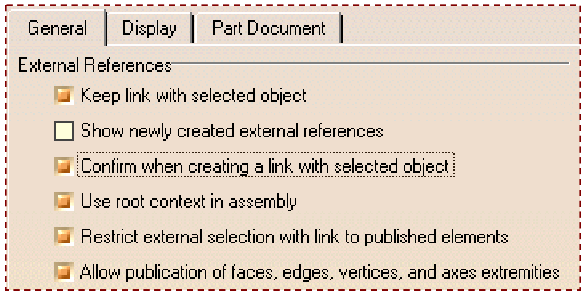
You will learn in which cases you can use published geometry.

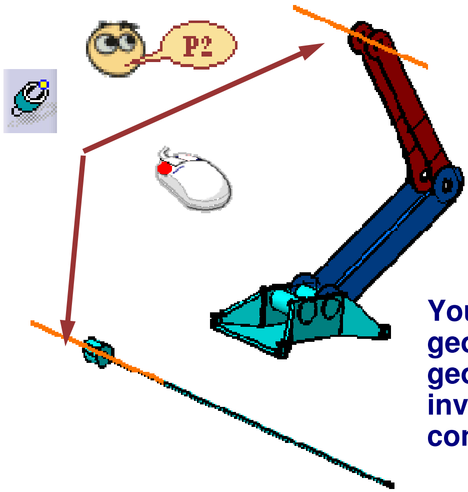


Student Notes:

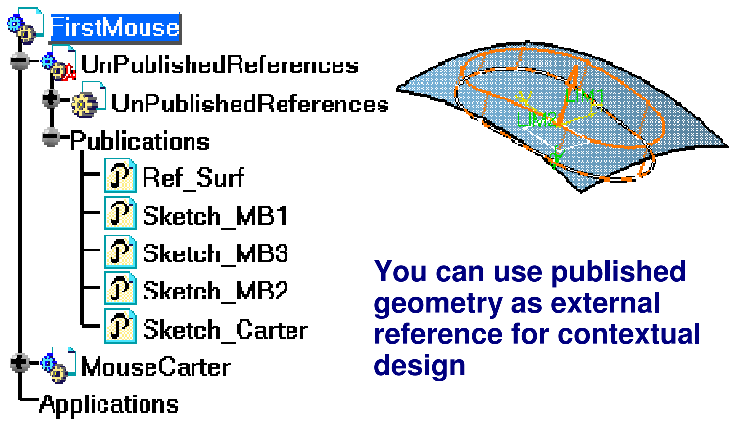
When Can You Use Published Geometry ?

Published geometry can be used in any command that requires geometric elements. It means in assembly constraint edition and design in context. But publishing geometry is especially useful when you replace components which are used in assembly constraints or contextual design.





You can use published geometry to specify geometric elements involved in assembly constraints



You can use published geometry as external reference for contextual design

In this case, we had the setting “Use published geometry of any level” activated and could only select published geometry to define the coincidence constraint.

In this case, to design MouseCarter Part , we used two published elements of PublishedReferences component : Sketch_Carter and Ref_Surf.

User Setting : Use Published Geometry to Constrain (1/3)

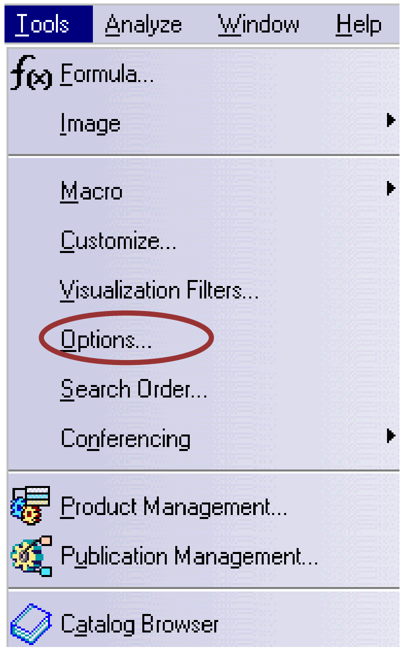
There is a setting that prevents from using other geometry than the published one when creating Assembly Constraints.

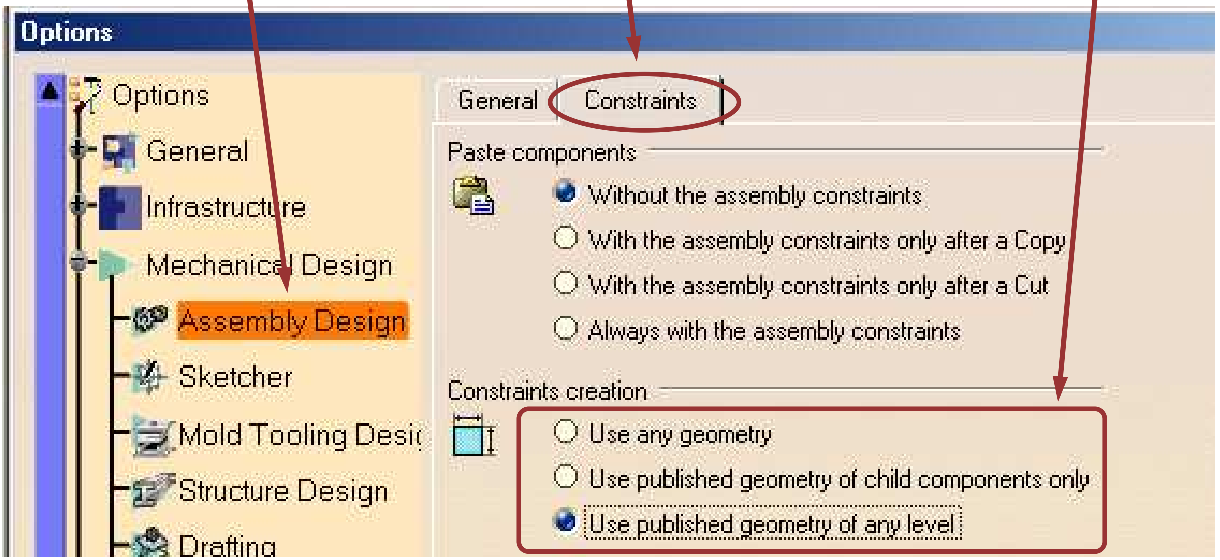
1 **Select Options... from the Tools menu**

2 **Select the "Assembly Design" branch under "Mechanical Design" node**

3 **Select Constraints tab**

4 **Activate one of the "Use published geometry" options**





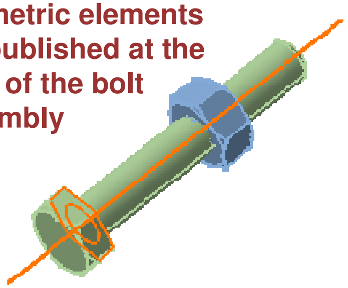
Student Notes:

Student Notes:

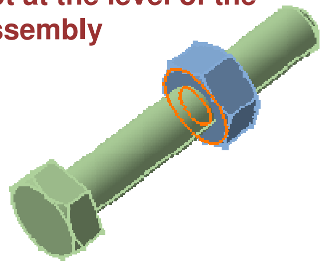
User Setting : Use Published Geometry to Constrain (2/3)

When imposing the use of published geometry, you can choose between two behaviors.

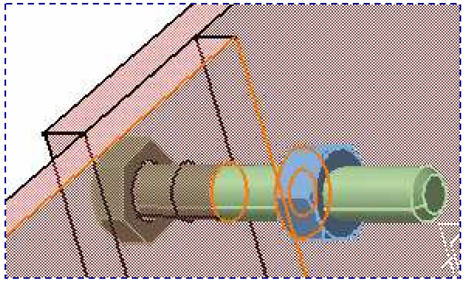
Only these two geometric elements are published at the level of the bolt assembly



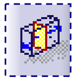
This face is published at the level of the nut component and not at the level of the bolt assembly



We have inserted the bolt assembly in another assembly also containing two plates

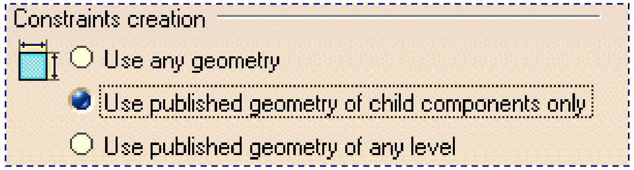


We have selected these two faces in order to put a contact constraint between them

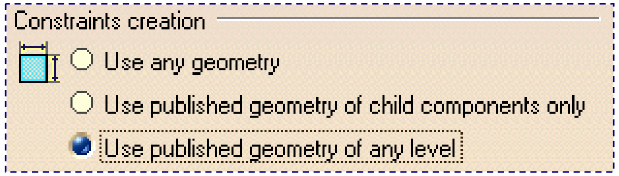


The behavior of CATIA won't be the same whether the active option is…

…This one…



…or This one.

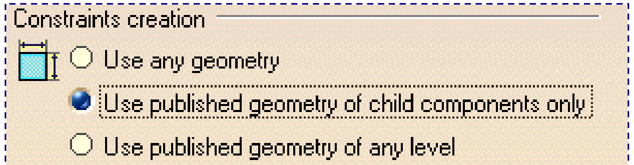


Student Notes:

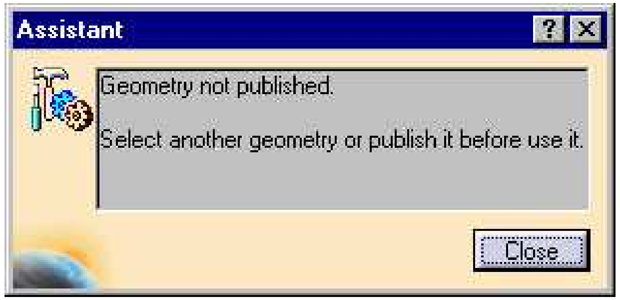
User Setting : Use Published Geometry to Constrain (3/3)

When imposing the use of published geometry, you can choose between two behaviors

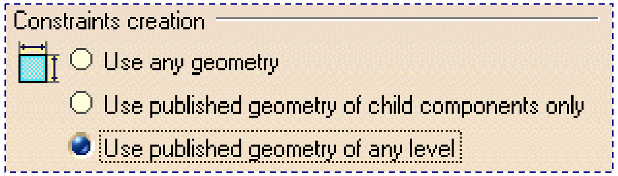
With this option…



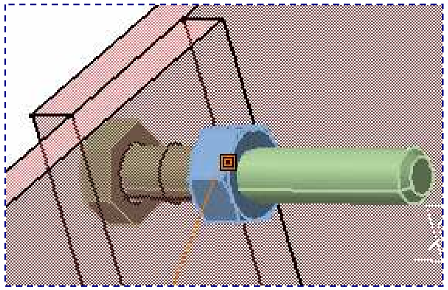
… It will not be possible to constrain because the face of the nut is not published at the required level



With this option…



… Contact constraint will be created because the face of the nut is published at least at a sub-level



Student Notes:

Using Published Geometry in Assembly Constraints (1/2)

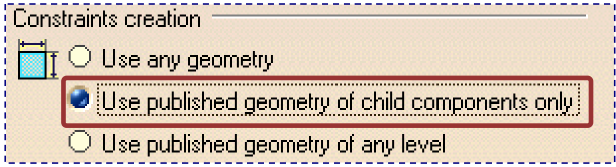
You can select published geometry to define assembly constraints between two components. There are two cases:

A. All elements involved in the constraint are published.

B. At least one element involved in the constraint is not published.

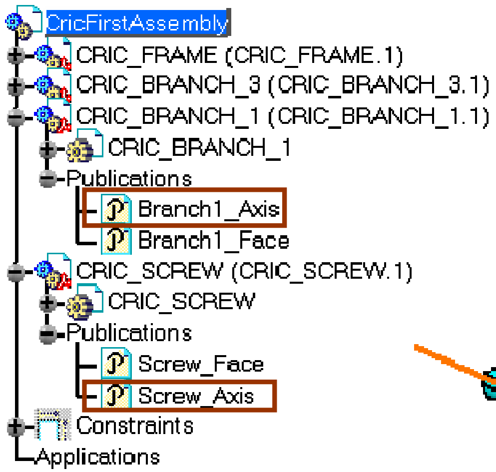
Case A: All elements published

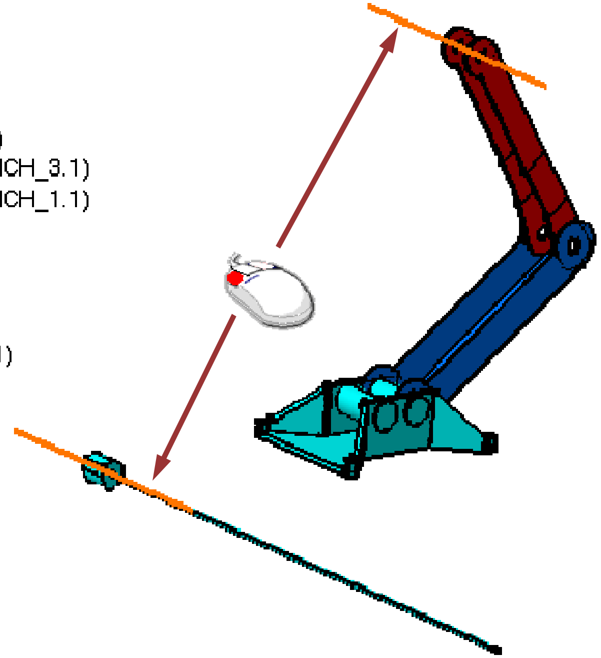
1 You can activate one of the “use Published geometry”options



2 Select your constraint 

3 Select Elements directly in geometry or from Publications in the specification tree.



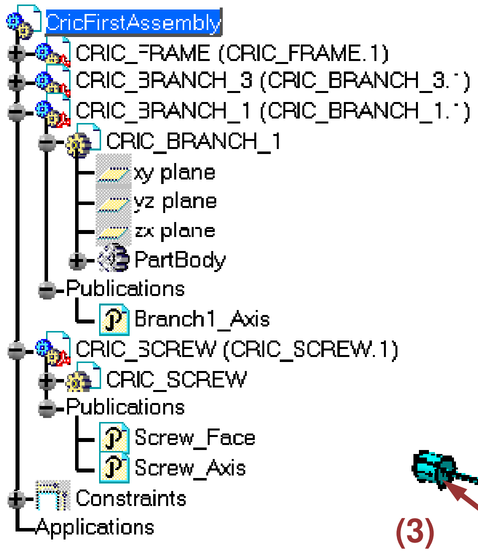


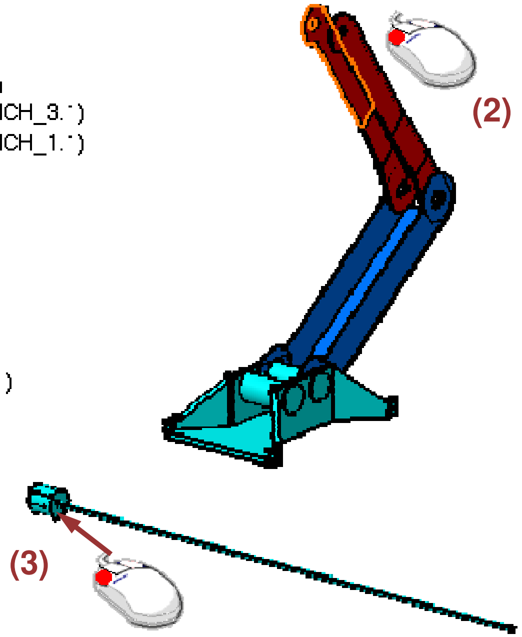
Student Notes:

Using Published Geometry in Assembly Constraints (2/2)

Case B: At least one of the elements is not published

1. Select your constraint
2. Select non published elements in geometry
3. Select published element under publication node in the tree or in the geometry





Student Notes:

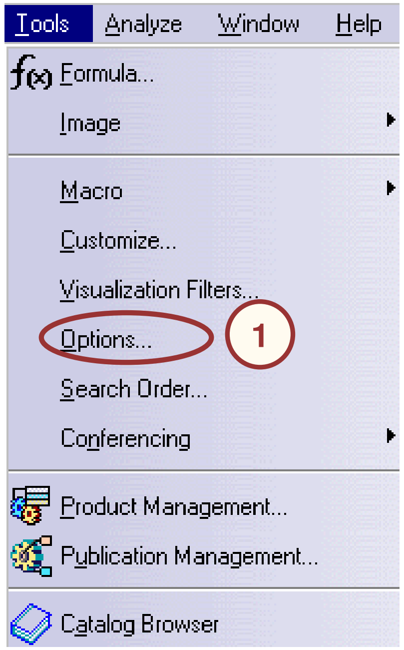
User Setting : Only Use Published Geometry for External References

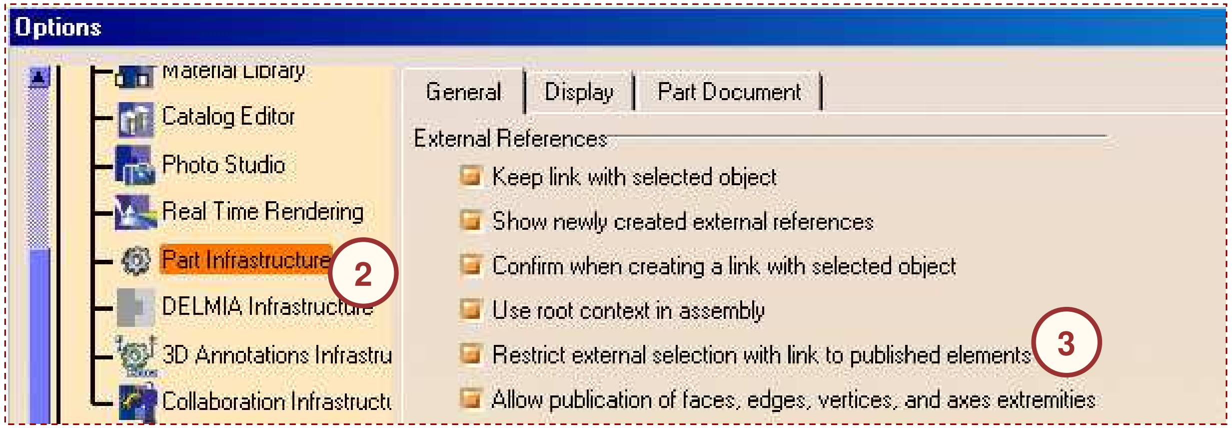
This setting prevent you from selecting other geometry than published one when creating external reference

1 Select Options... from the Tools menu

2 Select the "Part Infrastructure" branch under "Infrastructure" node

3 Activate the "Restrict external selection with link to published elements"



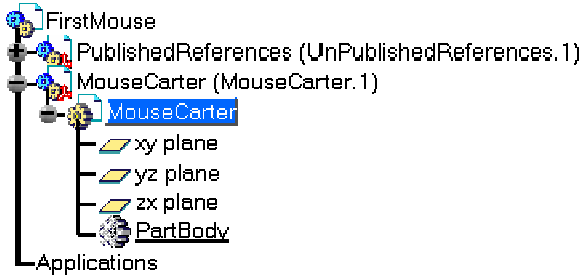


With this option activated, selection of external reference that is not published will not be possible, cursor will have this shape ⊖ when moving around non published elements.

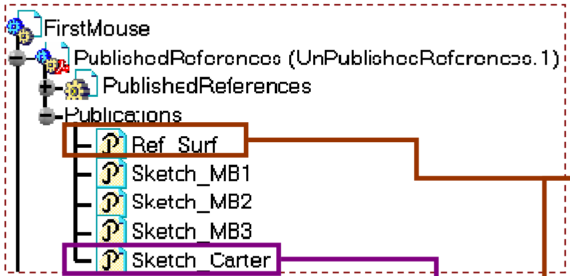
Student Notes:

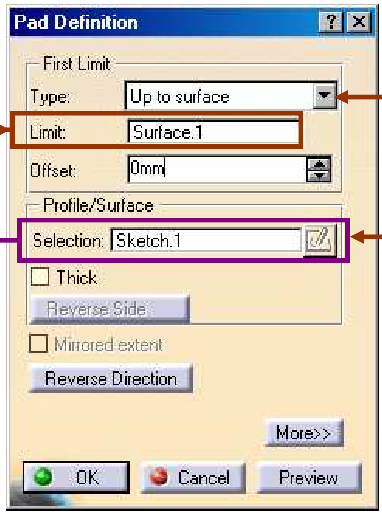
Using Published Geometry in Contextual Design (1/2)

You can select published geometry as external reference to design associative parts in context of the assembly.



1. **Activate the Part that you want to design in context**
2. **Create a Pad**





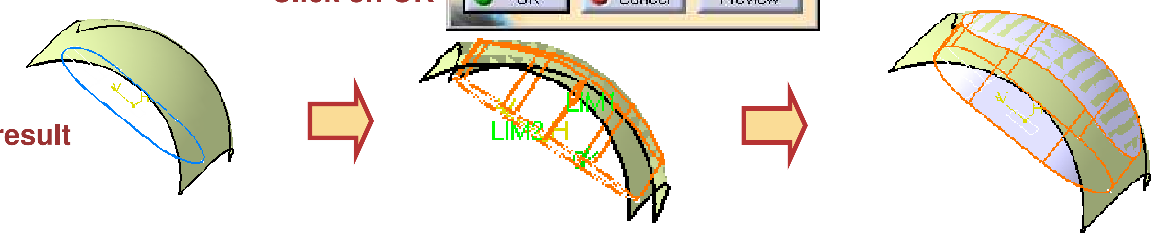
Select up to surface as type of first limit

Select as profile the Sketch_Carter publication of Published Reference

Select as limit the Ref_surf publication of Published Reference component (you can either select it in the tree or in the geometry)

Click on OK

Here is the result

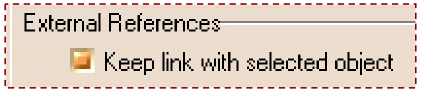


Student Notes:

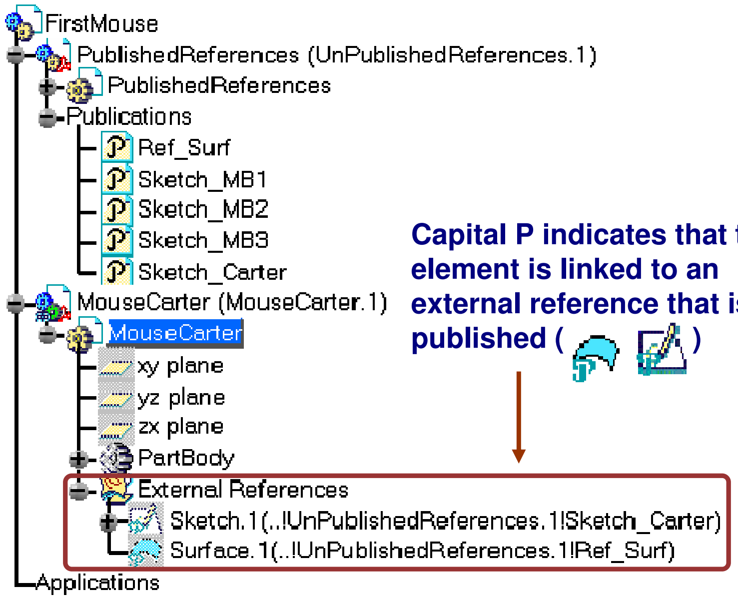
Using Published Geometry in Contextual Design (2/2)

Published geometry as any other geometry does appear under External Reference node when used to design another part in context of the assembly.

3a **If the option "keep link with selected object" was on while editing the part, then…**

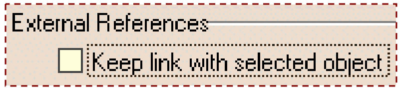


…the copies of published geometry are under External References node of the part and are associative

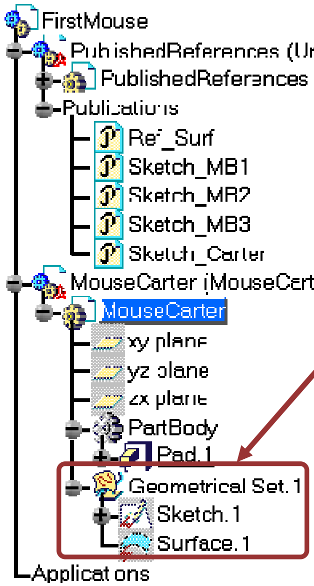


Capital P indicates that the element is linked to an external reference that is published ( )

3b **If the option "keep link with selected object" was off while editing the part, then…**



…the copies of published geometry are under an Geometrical Set of the part and are not associative



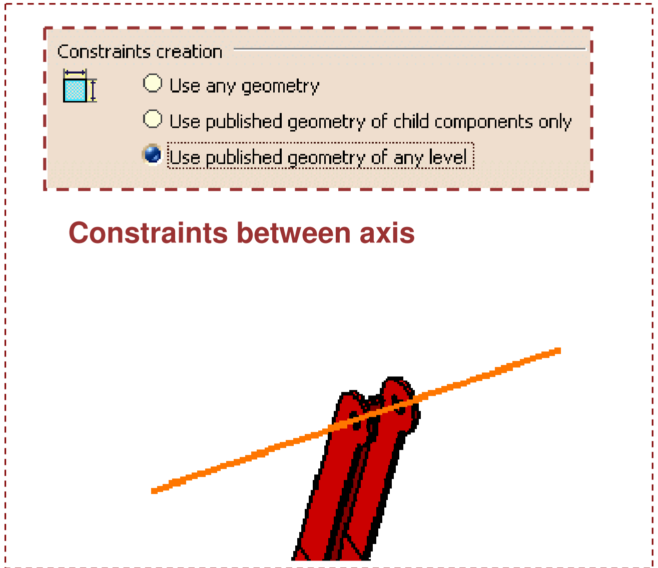
Red lightning indicates that the Wireframe & Surface element is a datum (non associative element that you can not modify) ( **)**

Student Notes:

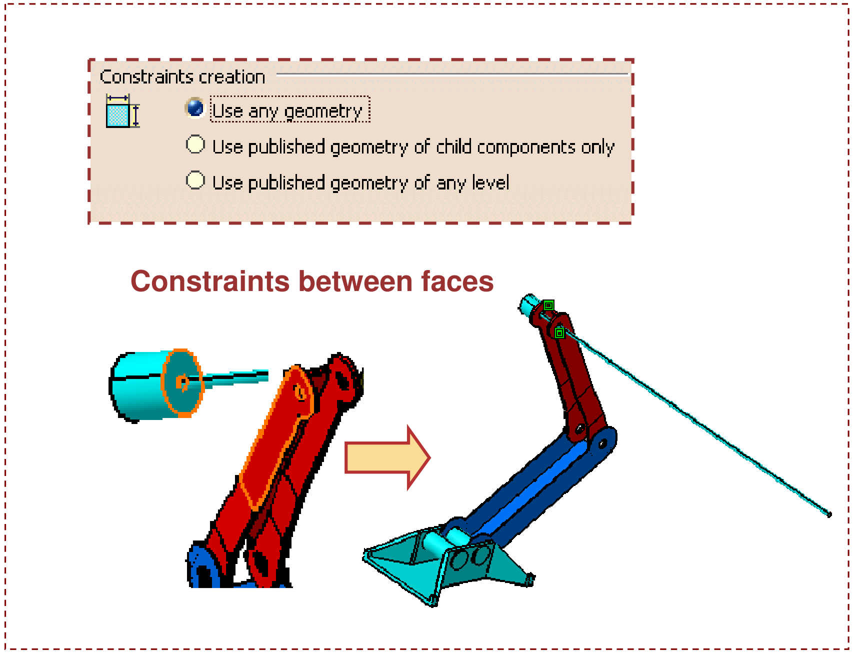
Do It Yourself (1/2)

P2

Product used: CATASMUsePubGeoInConstdoit.CATProduct



Constraints between axis

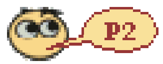


Constraints between faces

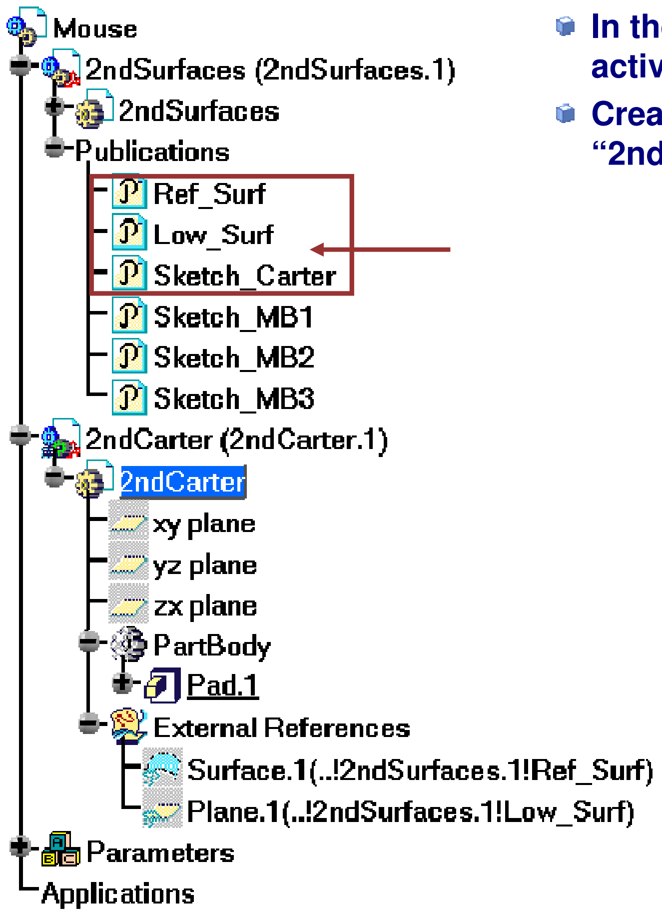
- **Activate first the option "Use published geometry of any level" then put a coincidence constraint between the two axes.**
- **Multi select elements highlighted above and click on Contact Constraint icon, you should have a message informing you it is not possible.**
- **Deactivate the "Use published geometry of any level" option and put a contact constraint between those two elements.**

Student Notes:

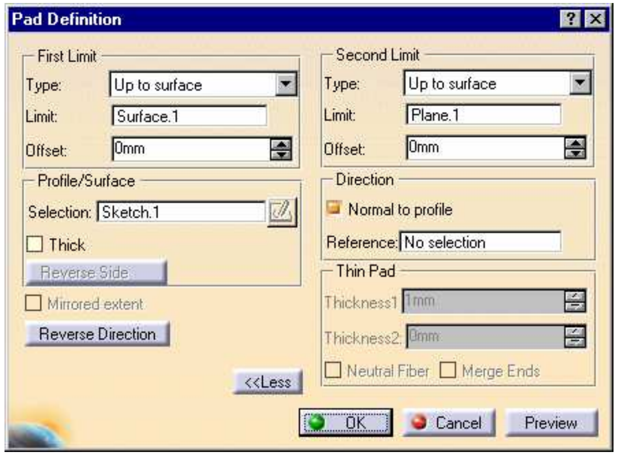
Do It Yourself (2/2)

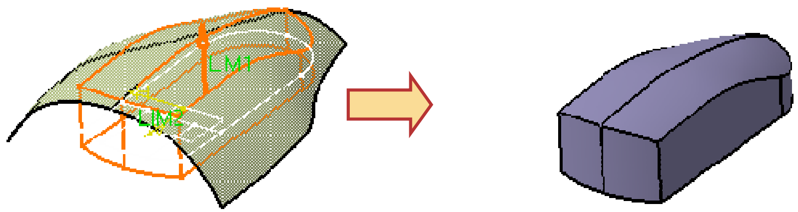


Product used: CATASMJARepPubCompWithConstDoIt.CATProduct



- In the loaded product, edit “2NDCarter” part, activate “Keep link with selected object ” option
- Create a Pad and select published elements of “2ndSurfaces”to define it as shown above.

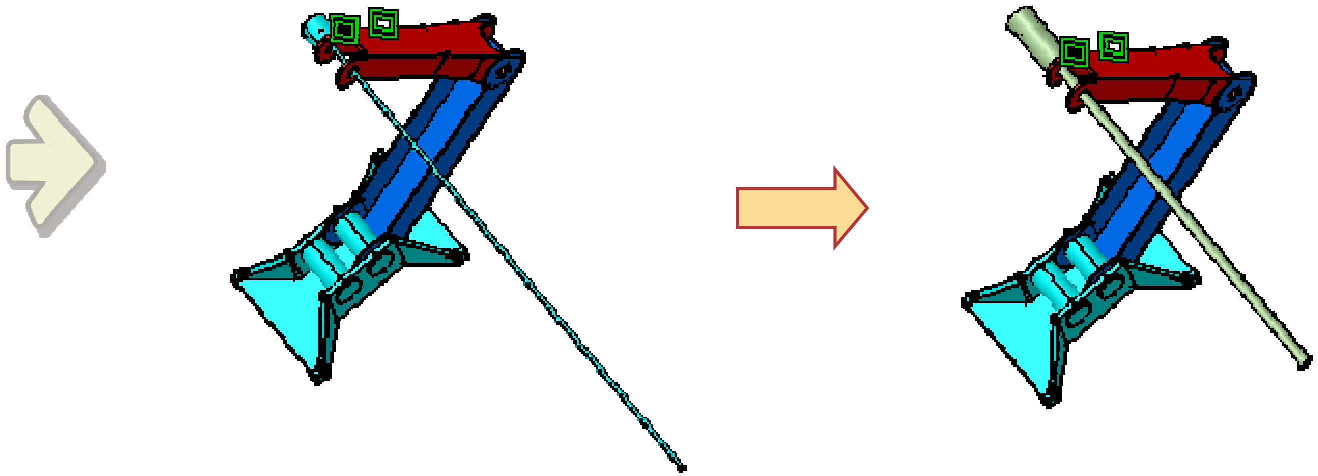




Student Notes:

Replacing Published Components

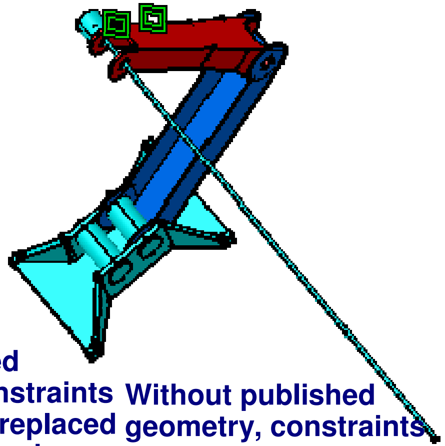
You will see the importance of Publication while replacing components.



Student Notes:

What is Replacing Published Component ?

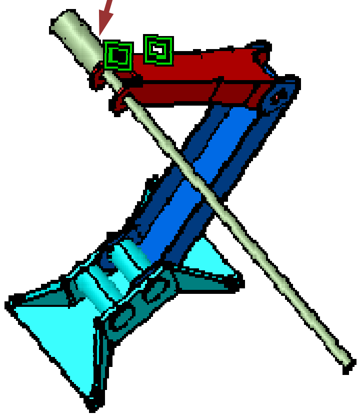
Published geometry becomes useful when you replace a component and when the replaced component is involved in a constraint or driving other contextual components.

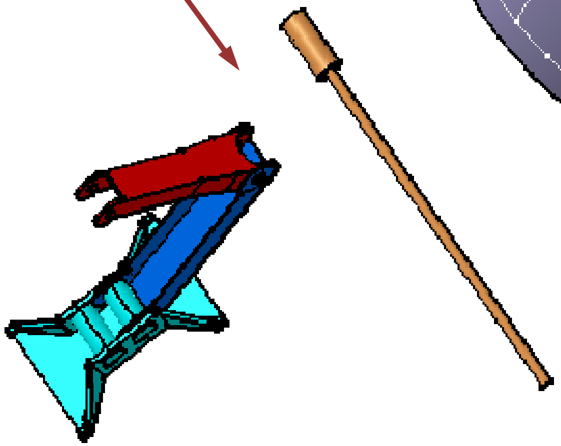


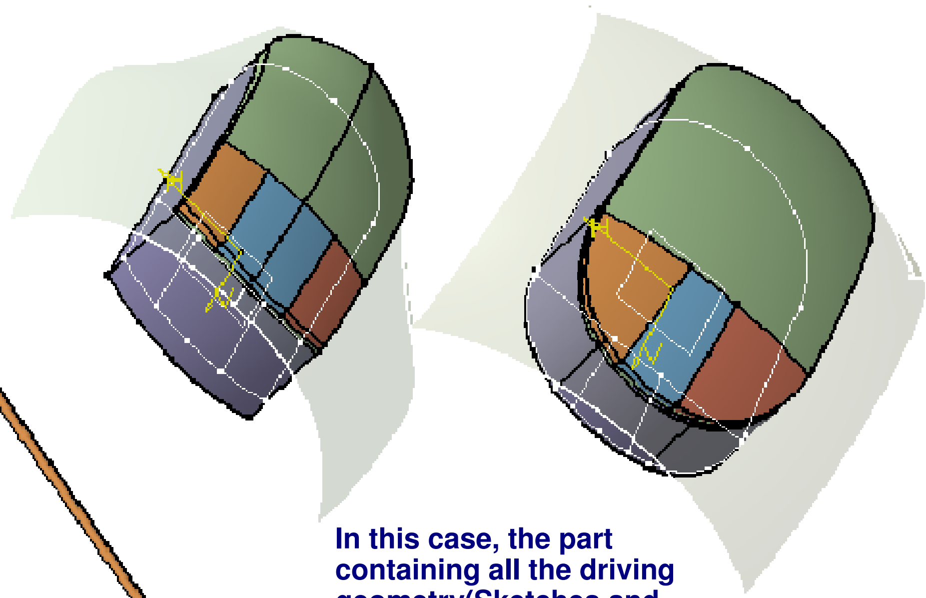
Concerning Design in Assembly context , the best way to replace a component driving geometry of other components is to use Published geometry.

With published geometry, constraints involving the replaced component can be preserved

Without published geometry, constraints involving the replaced component must be reconnected







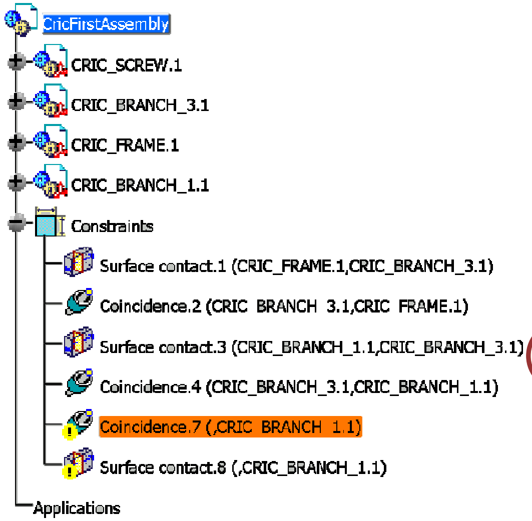
In this case, the part containing all the driving geometry(Sketches and Surfaces) has been replaced.

Student Notes:

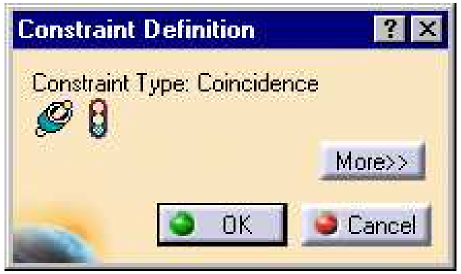
Reconnecting a Constraint (1/2)

A constraint can become unresolved after a replacement of a component or connected to a wrong geometric element. You have the possibility to redefine geometric elements involved in it.

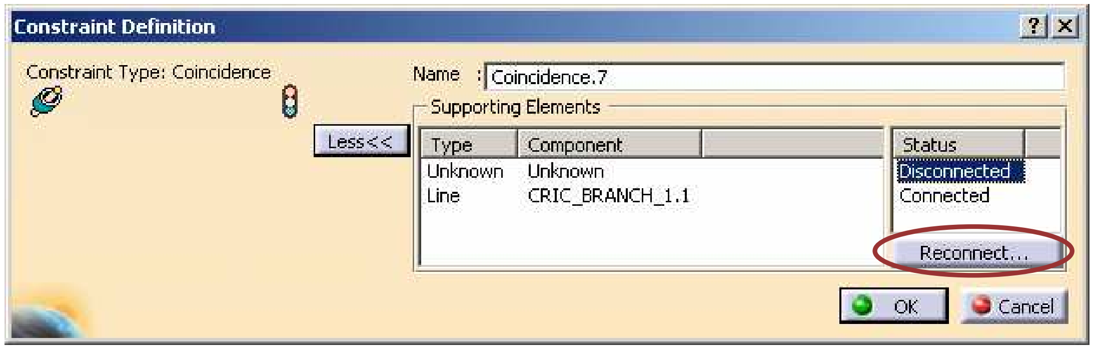
1 Edit the constraint you want to reconnect



2 Expand the dialog box



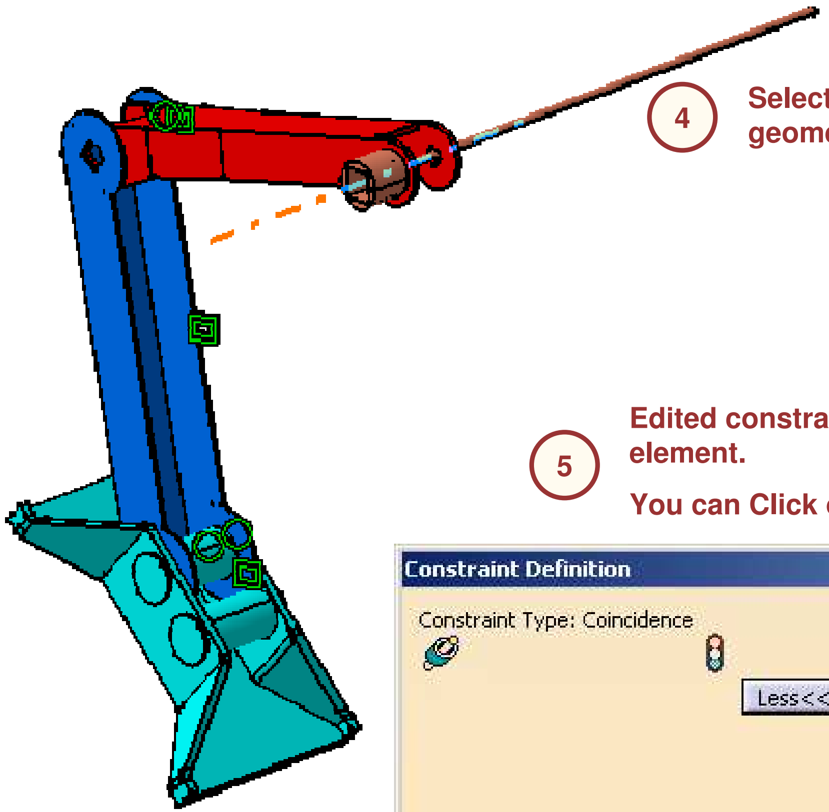
3 Select in dialog box the geometric element to reconnect and then click on Reconnect



Student Notes:

Reconnecting a Constraint (2/2)

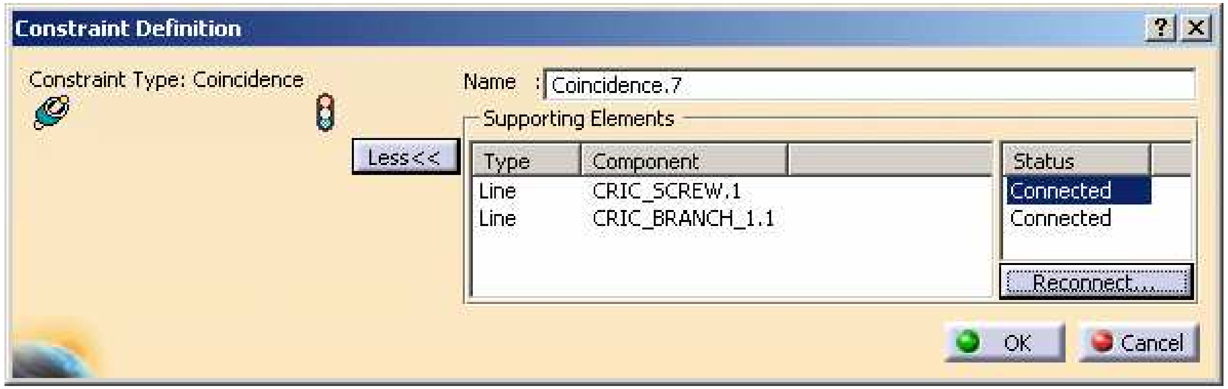
The Constraint dialog box let you have a look at geometric elements involved in it.



4 Select the new connected geometric element

5 Edited constraint is now connected to the just selected element.

You can Click on OK and Update the constraint

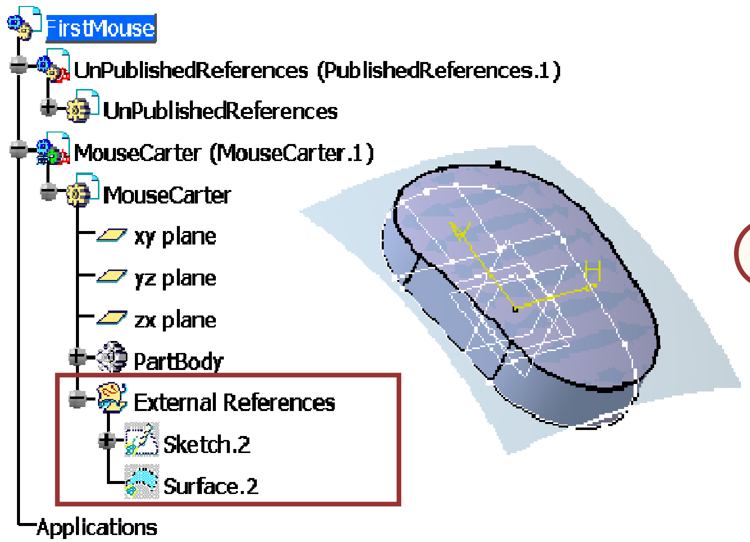


Student Notes:

Replacement Of a Non Published Driving Component (1/3)

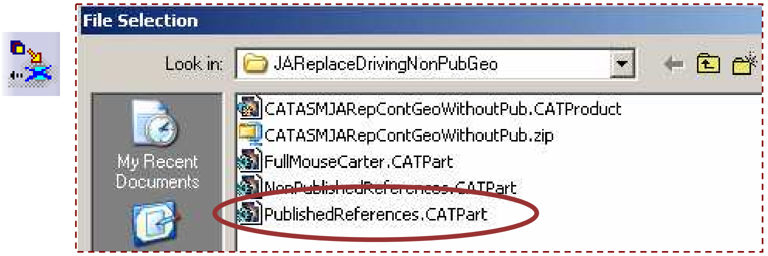
When you replace a component which contains geometry leading other contextual components of the assembly, driven components will have to be re-designed to be reconnected to the new driving geometry.

"MouseCarter" part is contextual to "FirstMouse" assembly and linked to "Unpublished References" part

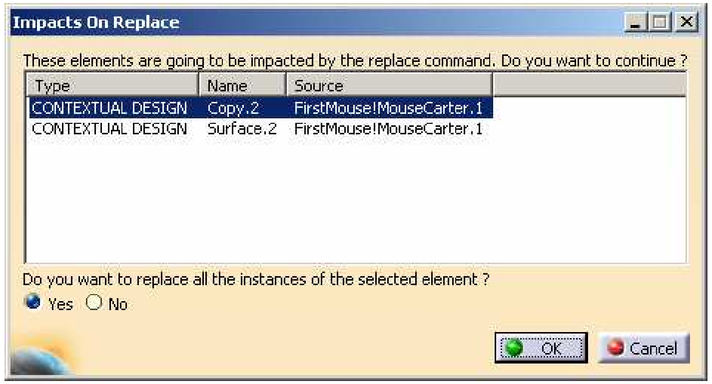


The two external references are synchronized (green light) with geometry of the driving component "Unpublished References".

1 Replace "Unpublished References" with another Part ("PublishedReferences.CATpart")

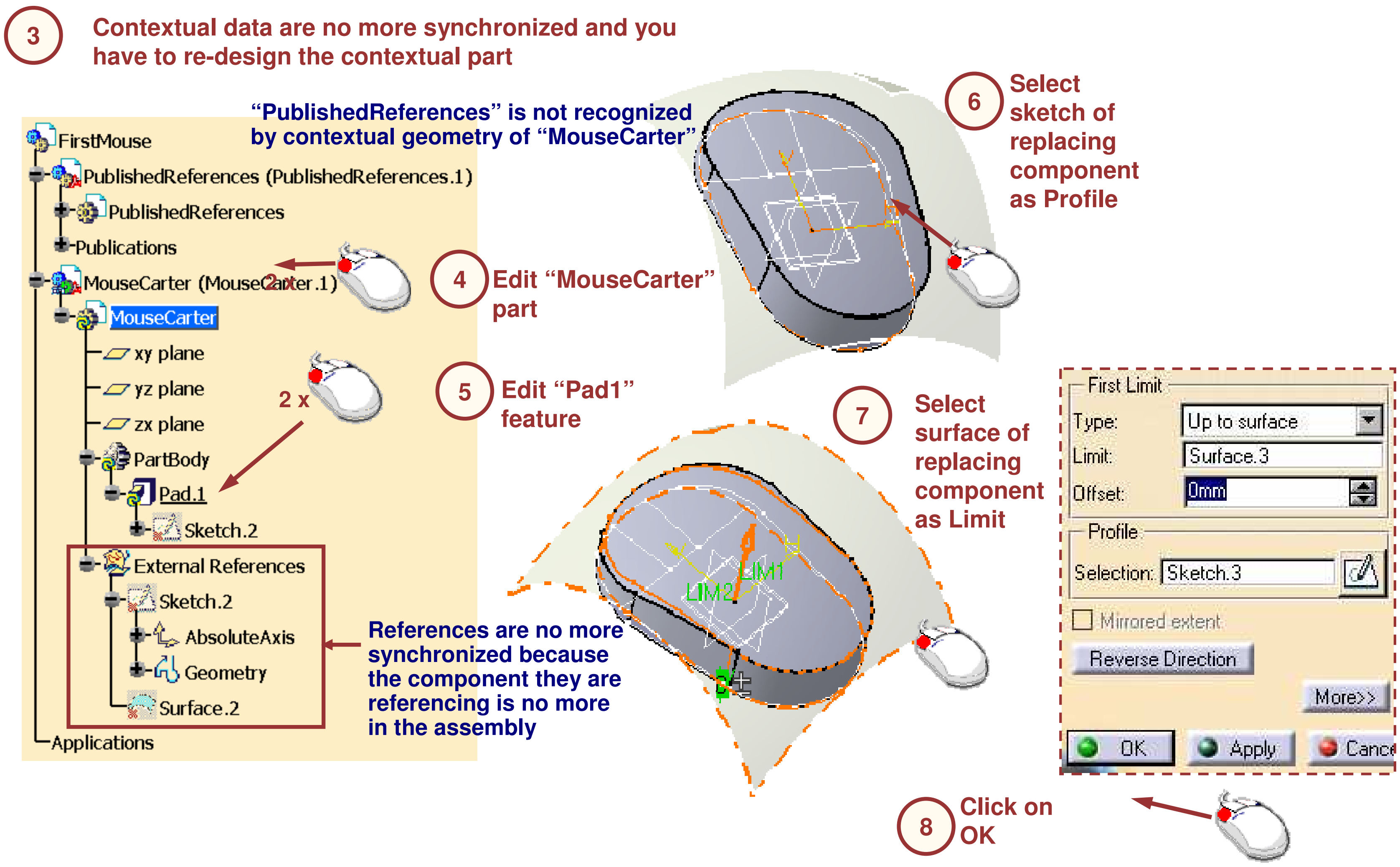


2 An 'Impact on Replace' window appears. Select the choice to replace the current or all instances and Click OK.



Student Notes:

Replacement Of a Non Published Driving Component (2/3)

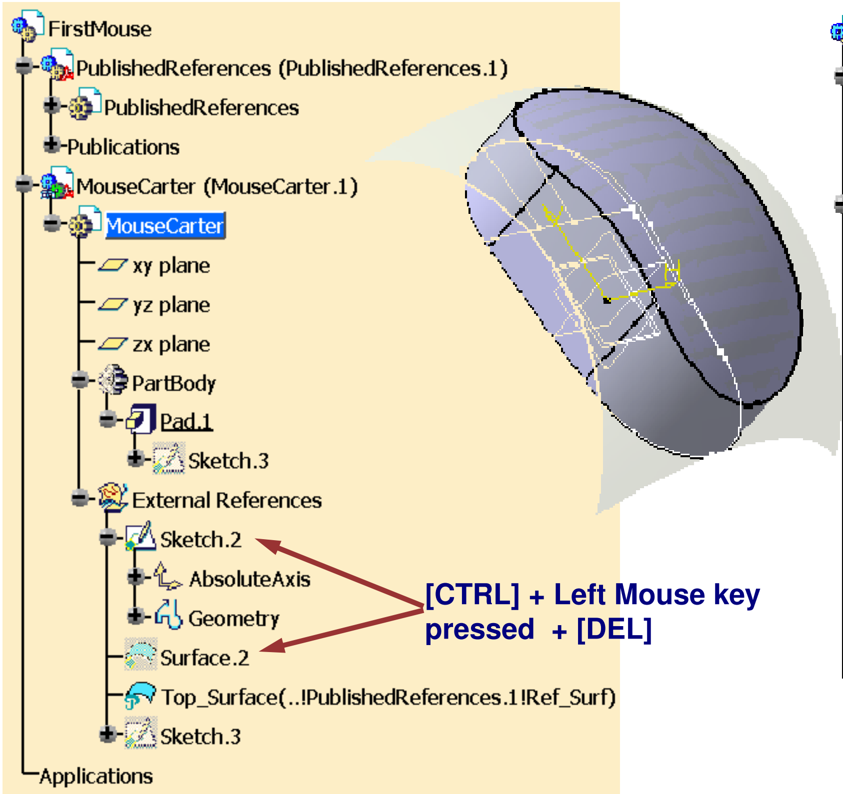


Student Notes:

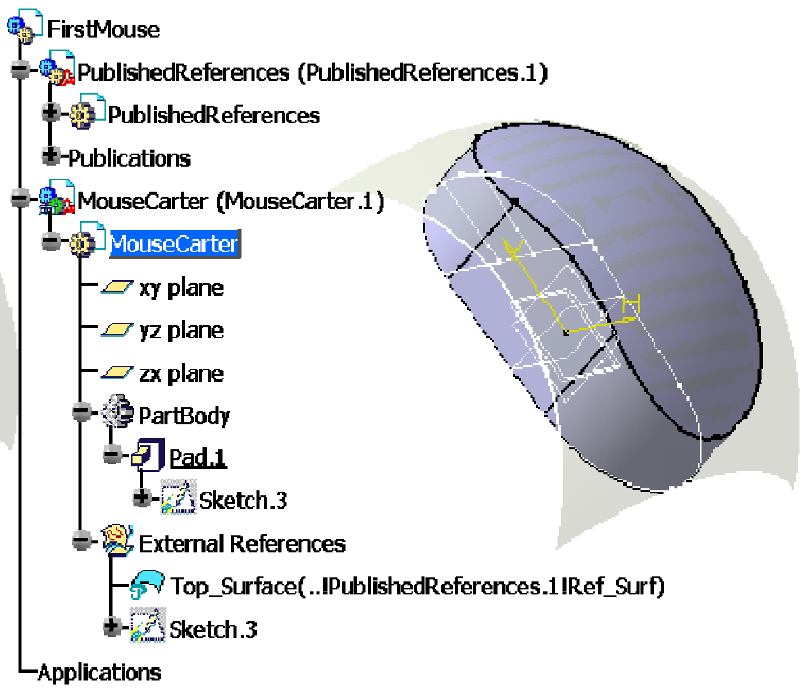
Replacement Of a Non Published Driving Component (3/3)

Re-design parts in context creates other external references, you have to delete the old ones that have become useless.

9 **Delete useless External References**



10 **Contextual part references now only geometry of the replacing component**

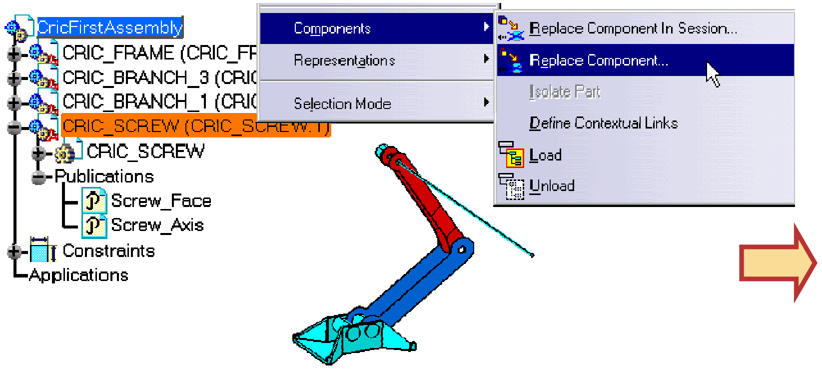


Student Notes:

Published Geometry and Assembly Constraints (1/4)

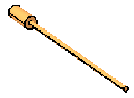
When you replace a published component that is involved in assembly constraints, it is possible thanks to published geometry to have automatic reconnection of the constraints.

1 **Select the component to be replaced and from contextual menu, select 'Replace component'**



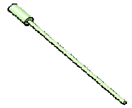
There are two cases for Replacing component: Case (2a) and Case (2b):

2a **Select the component to be replaced as 'CRIC_SCREW2.CATPart'.**



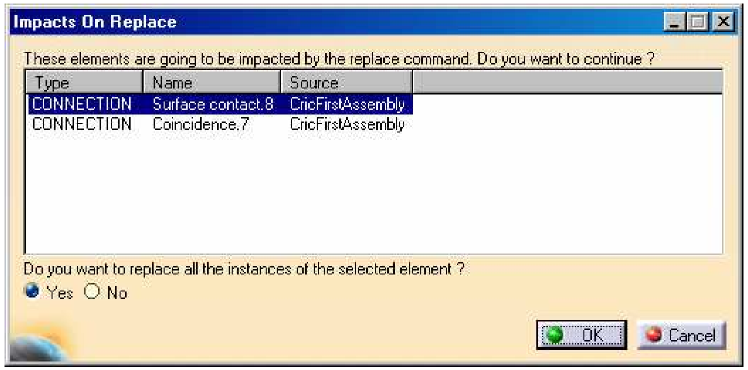
No Geometry is published in CRIC_SCREW2 component

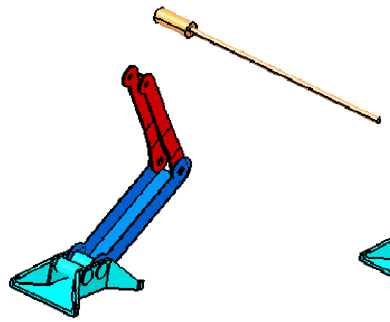
2b **Select the component to be replaced as 'CRIC_SCREW3.CATPart'.**



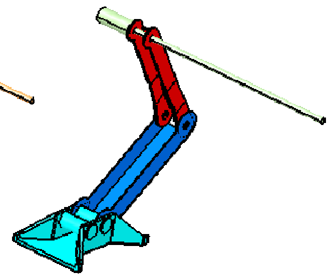
Geometry is published in CRIC_SCREW2 component

3 **The "Impacts On Replace" dialog box is displayed. Validate by clicking on OK.**





Result for Case 2a

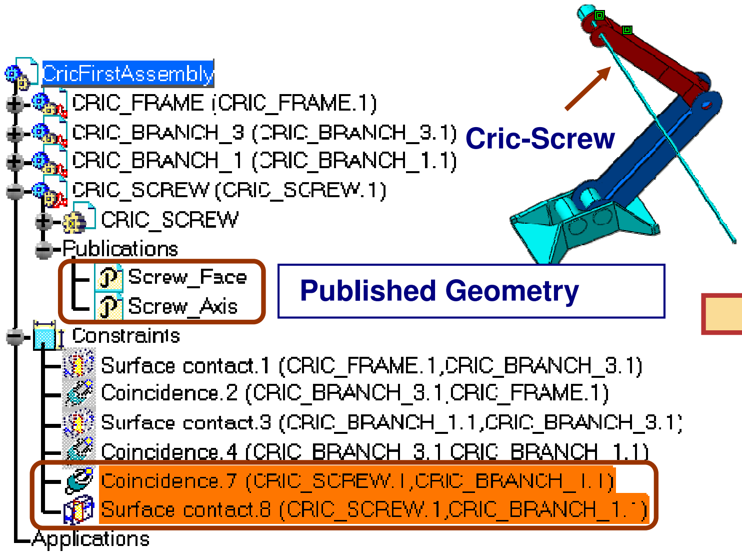


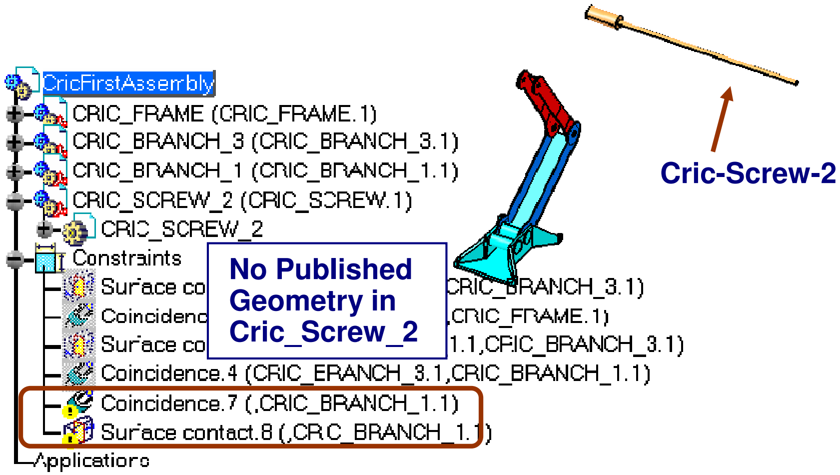
Result for Case 2b

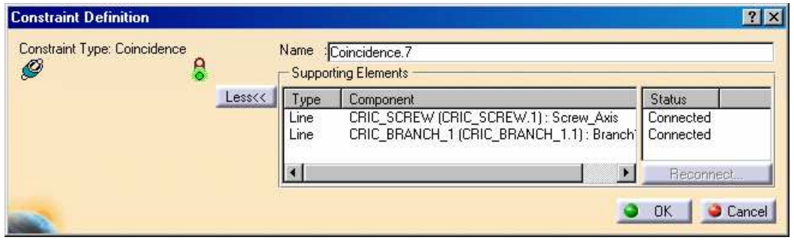
Student Notes:

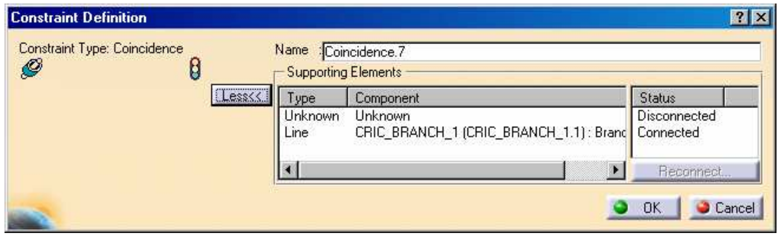
Published Geometry and Assembly Constraints (2/3)

Case 2a: Geometry involved in constraints not published









No re-connections of constraints

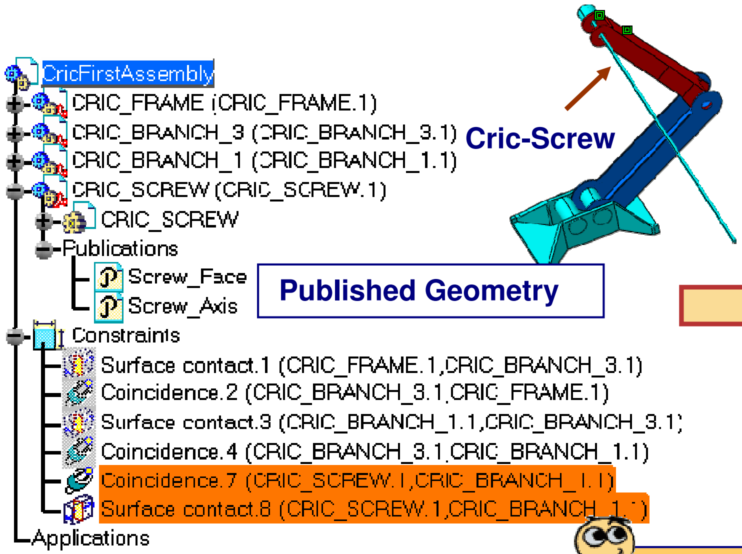
Two constraints in the assembly are connected to published elements of "CRIC_SCREW" component.

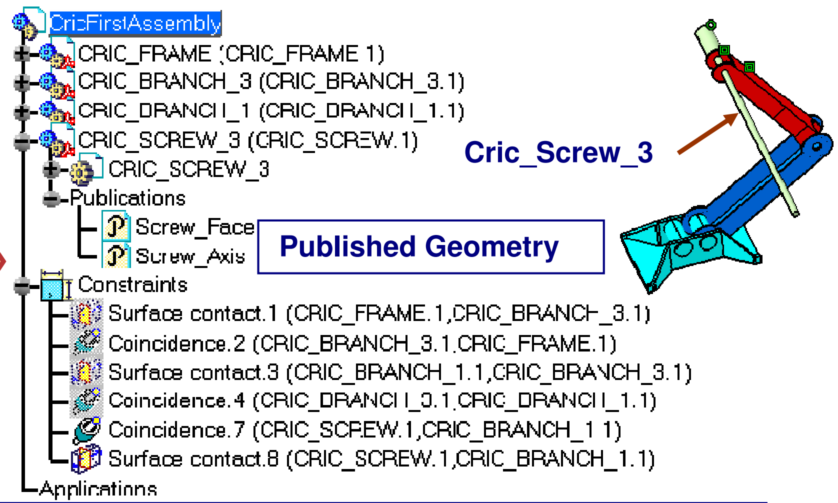
Constraints have become unresolved after component replace. You have to manually reconnect the broken constraints.

Student Notes:

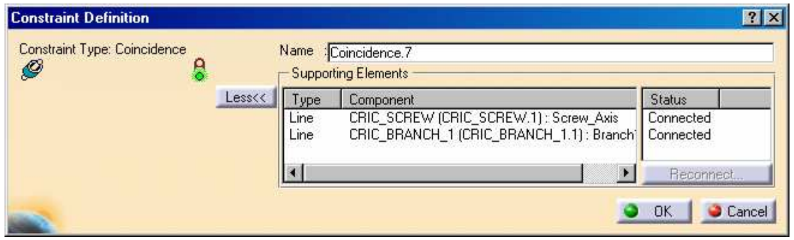
Published Geometry and Assembly Constraints (3/3)

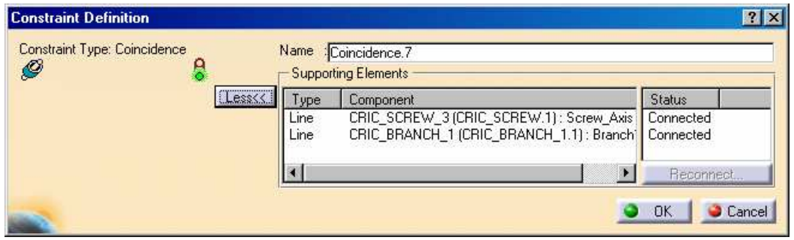
Case 2b: Geometry involved in constraints published





You have to ensure that published names are exactly matching for the component which is replacing the existing one.





Re-connections of constraints

Constraints are connected to published elements of "Cric_Screw"

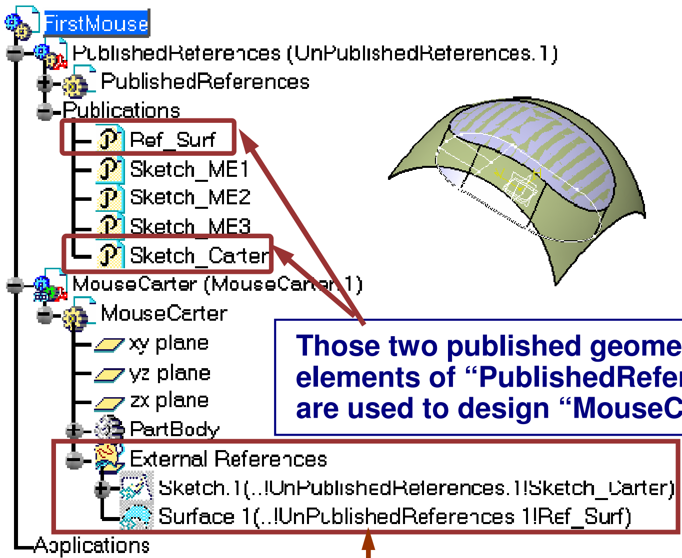
Constraints are connected to published elements of "Cric_Screw_3"

Published Geometry and Contextual Design (1/2)

When you replace a published component which contains geometry leading other contextual components of the assembly, there can be automatic reconnection to the external references depending upon the presence of Published Geometrical elements in the part being replaced.

Case 1a: Replaced part has no published geometry

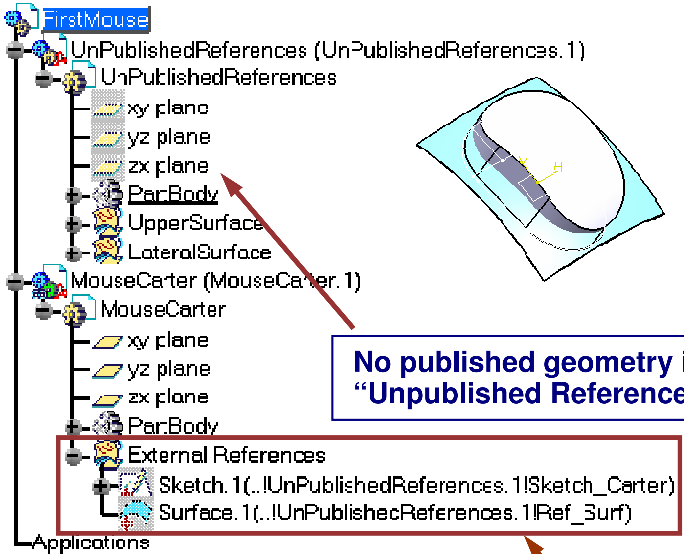
Use contextual design to design the Mouse carter



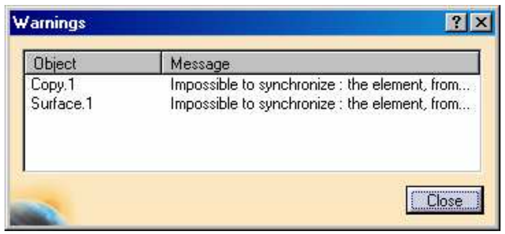
Those two published geometric elements of "PublishedReferences" are used to design "MouseCarter"

External references are synchronized with published geometry of "Published Reference"

Replace "PublishedReferences" with "UnPublishedReferences" part.



No published geometry in "Unpublished References"



External references are no more synchronized with any geometry

You have warnings about non synchronized geometry

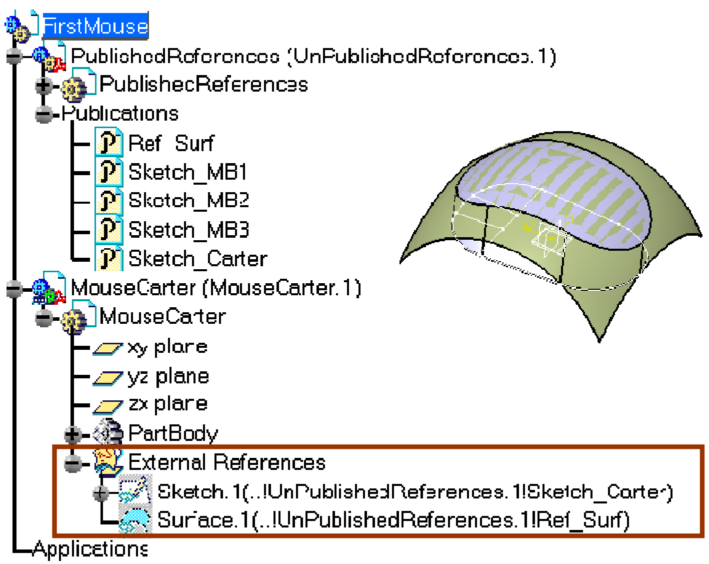
Student Notes:

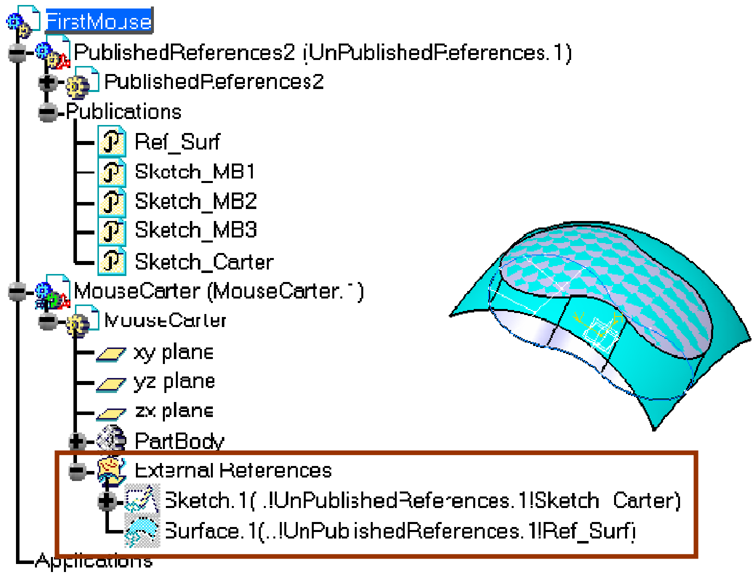
Student Notes:

Published Geometry and Contextual Design (2/2)

Case 1b: Replaced part has published geometry

"PublishedReferences" has been replaced with "PublishedReferences2" which has the same published geometry as the original PublishedReferences part.





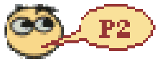
External references are synchronized with published geometry of "Published References"

Resynchronization

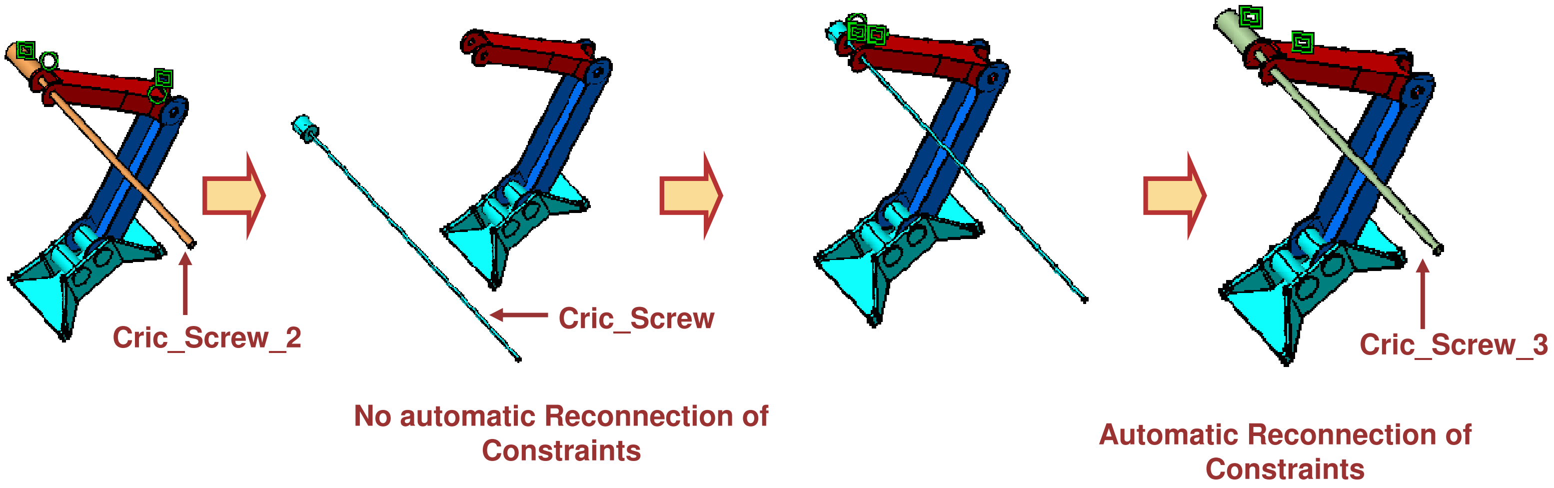
External references are synchronized with published geometry of "PublishedReferences2"

Student Notes:

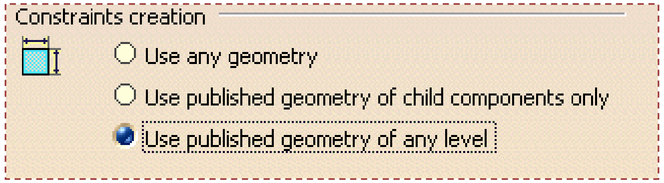
Do It Yourself (1/2)



Product Used: "CATASMJARepPubCompWithConstDoIt.CATProduct"



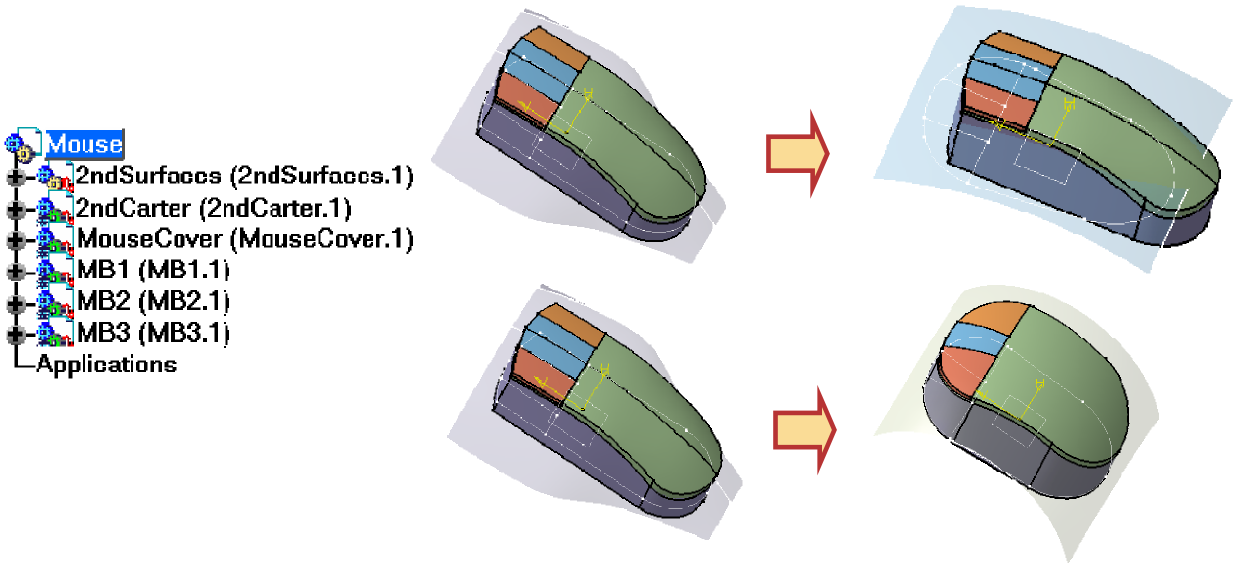
- **Replace "Cric_Screw_2" component with Cric_Screw.CATPart**
- **With setting shown activated, reconnect broken constraints to published geometry of "Cric_Screw"**
- **Replace "Cric_Screw" component with Cric_Screw3.CATPart and notice the automatic reconnection of constraints**



Student Notes:

Do It Yourself (2/2)

Product Used: “CATASMJAPubCompDesignCont.CATProduct”



External References are not synchronized

External References are synchronized

- Replace “2ndSurfaces” component with “UnpublishedReferences” CATPart
- Notice that External References of other components are no more synchronized and make as many Undo as necessary to get back “2ndSurfaces” in the assembly
- Replace “2ndSurfaces” component with PublishedReferences and notice that External References of other Components are synchronized

Student Notes:

Publication

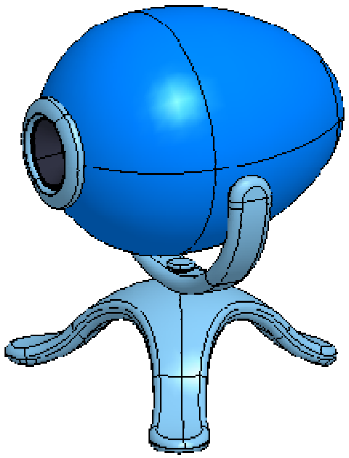
Recap Exercise: Webcam

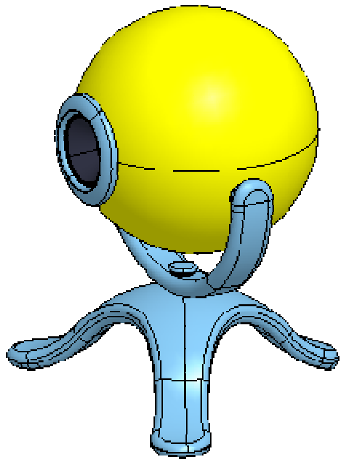
25 min

P2

In this exercise you will experiment the benefits of Publication while replacing a component. You will:

- **Position components and notice that assembly constraints become unresolved after replacement of the reference component.**
- **Design a part in context and notice that its links are broken after replacement of the reference component.**
- **Publish reference elements and notice that contextual links and positioning links are reconnected automatically after replacement of the reference part.**

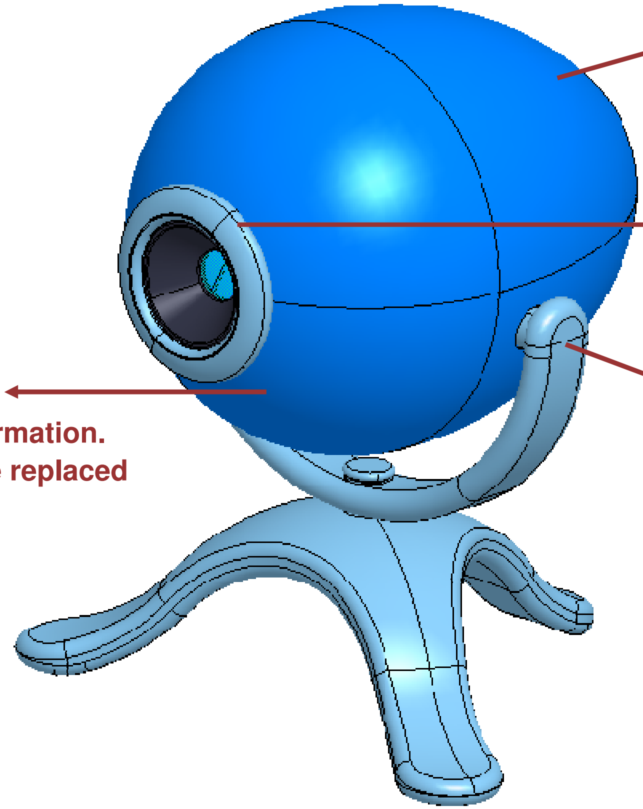




Student Notes:

Design Intent – Webcam

- In this exercise, you will study a Webcam, whose geometry is driven by the inferior part of its body.
- In case the inferior body needs to be replaced, all the elements pointing to it have to be reconnected automatically.



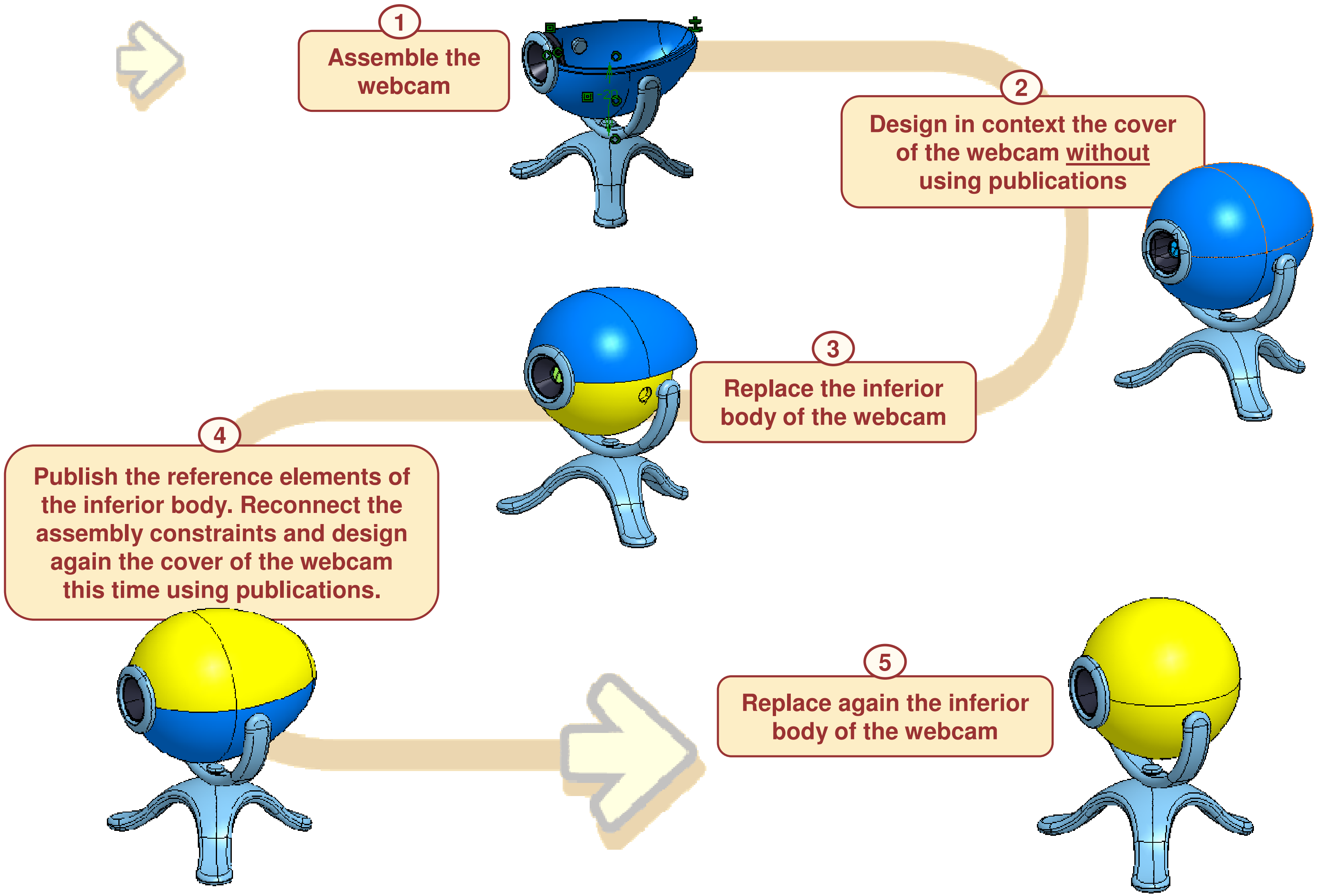
Cover or "Top_Body":
Its geometry must fit to inferior body's.

Camera:
In contact with the interior face of inferior body.

Inferior body:
Contains reference information.
The inferior body will be replaced by another one.

Support of the webcam:
Its axis must be coincident to the fixation axis of inferior body.

Design Process - Webcam

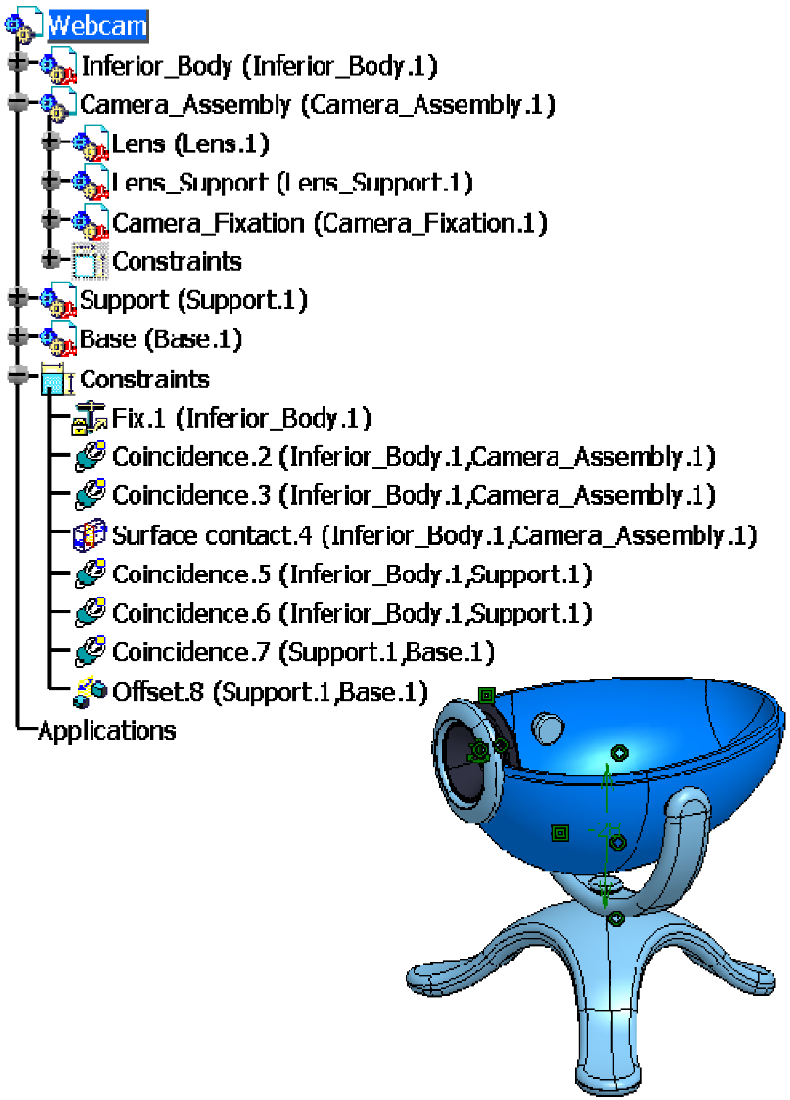


Student Notes:

Student Notes:

Step 1: Assemble the Webcam

- Create a new Product named “Webcam”.
- Insert “Inferior_Body.CATPart” and fix it immediately.
- Insert “Camera_Assembly.CATProduct” and position it with assembly constraints:
  - Coincidence between xy and zx planes of “Inferior_Body” and “Lens_Support”respectively
  - Contact between interior face of “Inferior_Body” and published face of “Lens_Support”
- Insert “Support.CATPart” and position it relatively to “Inferior_Body”:
  - Coincidence between zx planes
  - Coincidence between “Support_Axis” of “Inferior_Body” and “Webcam_Fixation_Axis” of “Support”
- Insert “Base.CATPart” and position it using the elements of “Support”:
  - Coincidence between vertical axes
  - Offset of 28mm between xy planes

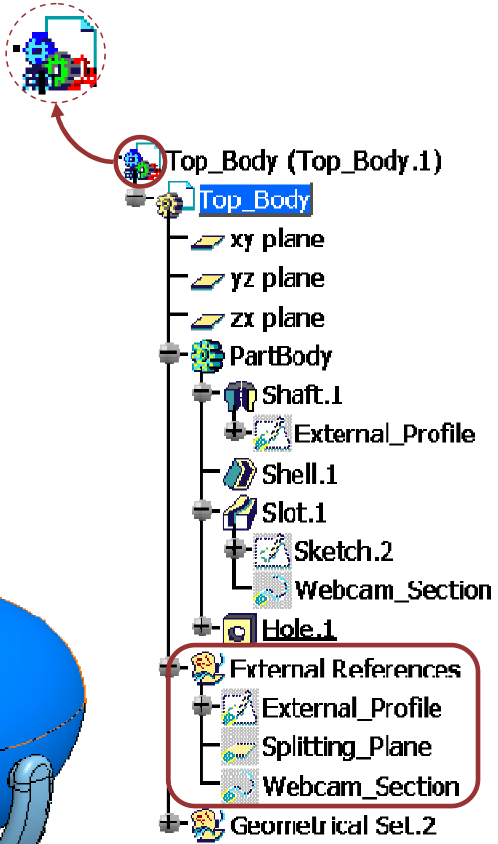


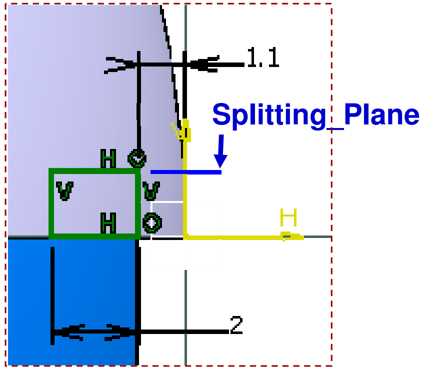
Student Notes:

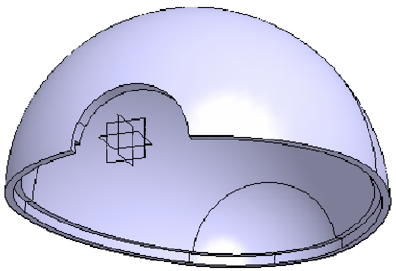
Step 2: Design in Context the Cover

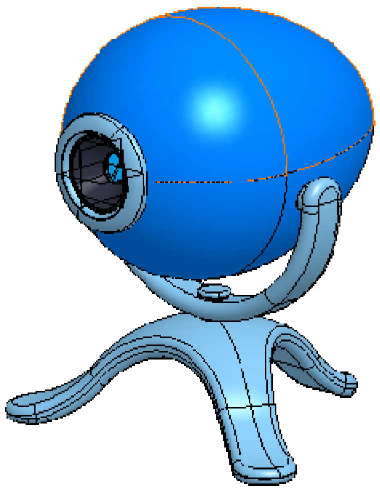
You can compare your result with “Webcam_without_Publication.CATProduct”

- Check that “Keep link with selected object” option is activated.
- Create a new part named “Top_Body”.
- Position it on “Inferior_Body” by creating coincident constraints between xy, yz and zx planes.
- Design the cover in context with “Inferior_Body”:
  - Create a 180deg Shaft reusing “External_Profile” sketch of “Inferior_Body”.
  - Create a 2mm Shell in order to remove the planar face.
  - Create the sketch as shown on zx plane and limited by “Splitting_Plane”.
  - Create a slot with previous sketch as profile and “Webcam_Section” as center curve.
  - Create a simple hole of diameter 15mm, lying on yz plane and centered on part origin.





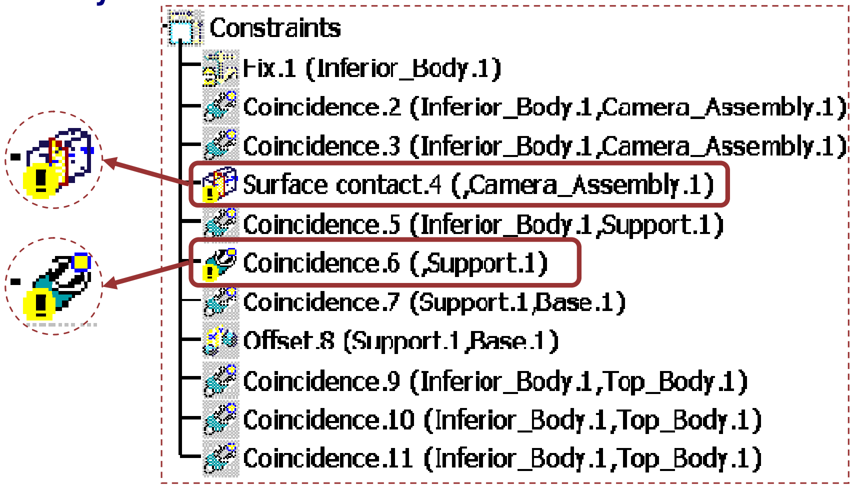


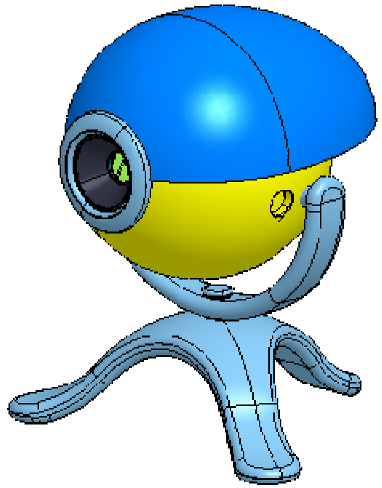


Student Notes:

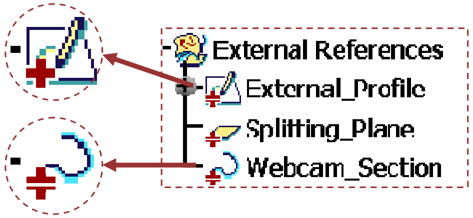
Step 3: Replacement Trial

- **Save the entire Product.**
- **Replace “Inferior_Body” by “Spherical_Body” and update the product.**
- **Notice that some assembly constraints are broken. These are the constraints that were pointing to geometry from “Inferior_Body”. The constraints that were pointing to reference planes of “Inferior_Body” have been reconnected automatically.**





- **Notice that the external references of “Top_Body” are also disconnected.**



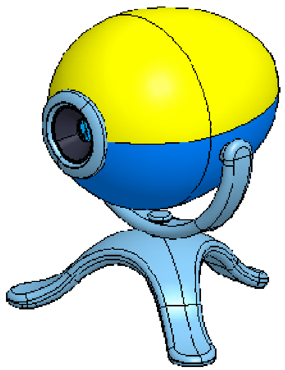
- **Close the Product without saving the modifications.**

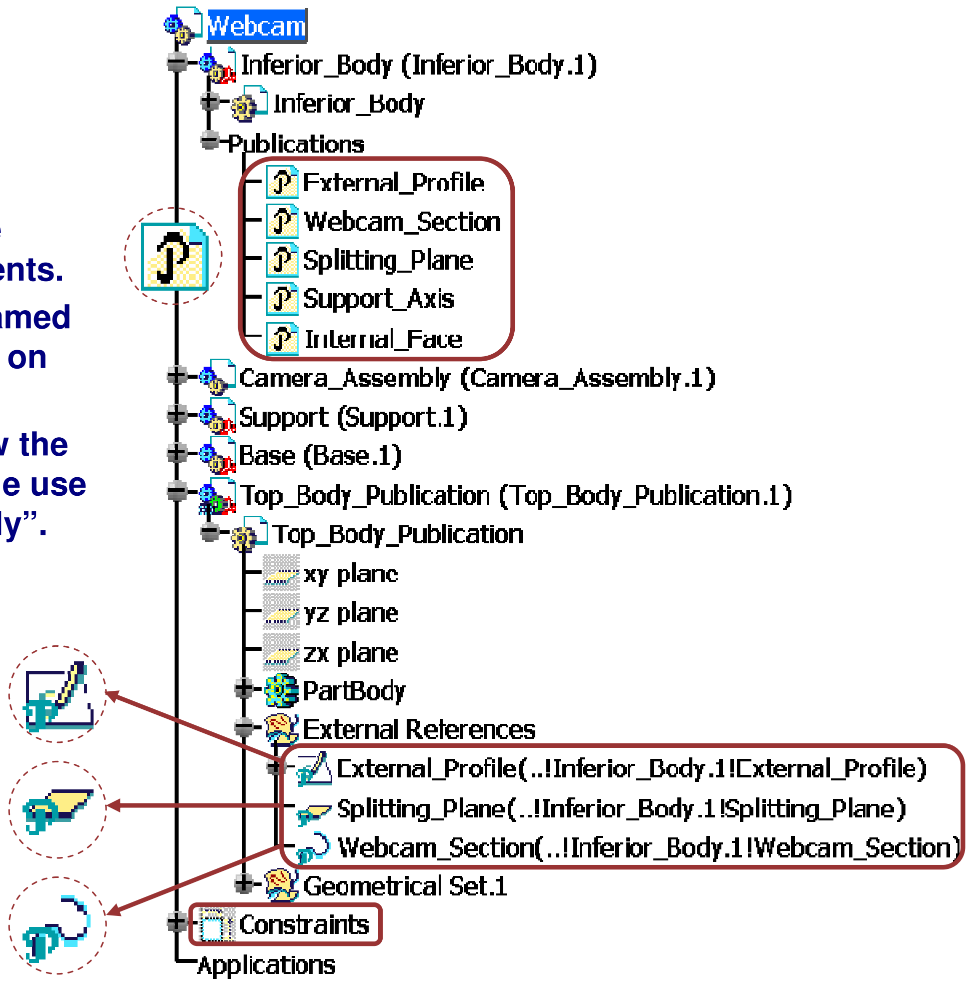
Student Notes:

Step 4: Publish and Recreate the Links

You can compare your result with “Webcam_Published.CATProduct”

- Reopen “Webcam.CATProduct”.
- Publish the reference elements of “Inferior_Body”, used either to design “Top_Body” or to position “Camera_Assembly” and “Support”.
- Recreate the constraints that need to be connected to the newly published elements.
- Delete “Top_Body”. Insert a new part named “Top_Body_Publication” and position it on “Inferior_Body” using reference planes.
- In this new part design the cover. Follow the same process than in step 2 but this time use the Published elements of “Inferior_Body”.

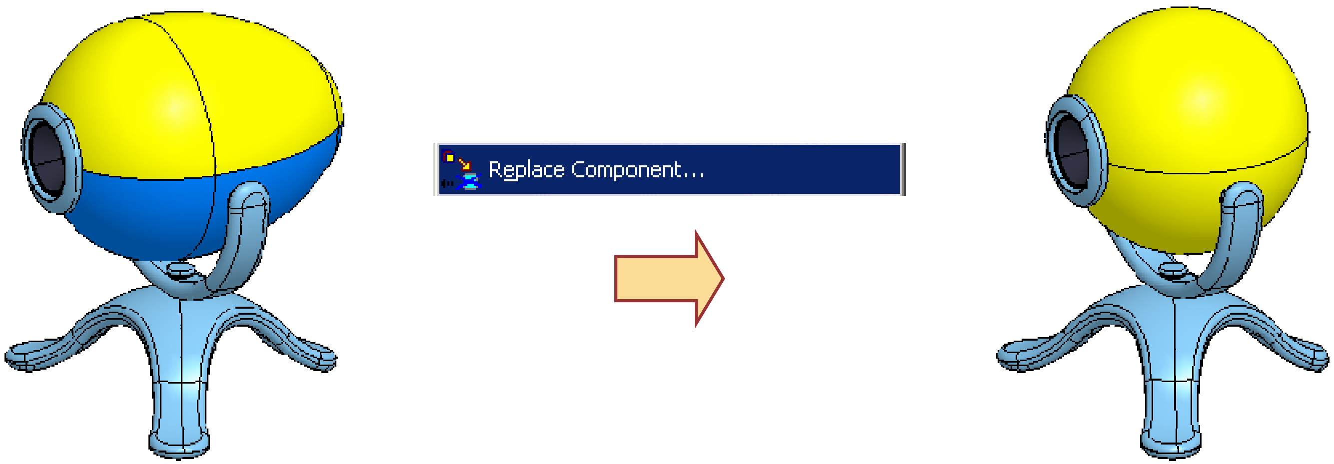




Student Notes:

Step 5: Successful Replacement

- Save the Product.
- Open “Spherical_Body” and check that it contains publications with exactly the same names as in “Inferior_Body”. If not, rename the Publications of “Inferior_Body” and save the Product again.
- Replace “Inferior_Body” by “Spherical_Body”.
- Update the Product.
- Notice that this time the geometry of the cover has adapted to the spherical shape of the new inferior body: the external references of the cover have automatically been reconnected to the published elements of the replacing part.
- Also notice that the axis of the support is coincident to the axis of the holes of the inferior body. The constraints have also been automatically reconnected thanks to Publication.



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Student Notes:

To Sum Up

In this lesson you have seen how to :

- **Publish Geometry : Publishing geometry means associating a name to it so that it will be recognized by other documents. You can publish any geometry like wireframe features(points, lines, curves), part design features (pads, pockets,holes), sketches, Generative shape design features(extruded surfaces, offsets, joins), sub elements of all geometrical elements (faces, vertices).**

- **Use Published Geometry in Contextual Design: You can design contextual parts by using external published geometrical elements like sketches, planes, points.**

- **Use Published Geometry while creating Assembly Constraints : You can use published geometrical elements while creating assembly constraints between components.**

- **Replace parts with published Geometries : When you replace components with published geometry, you can have automatic reconnection of external references of a replaced contextual part and assembly constraints.**

Student Notes:

Flexible Sub-Assembly

You will learn how to reuse several times the same sub-assembly with its components in different relative positions.

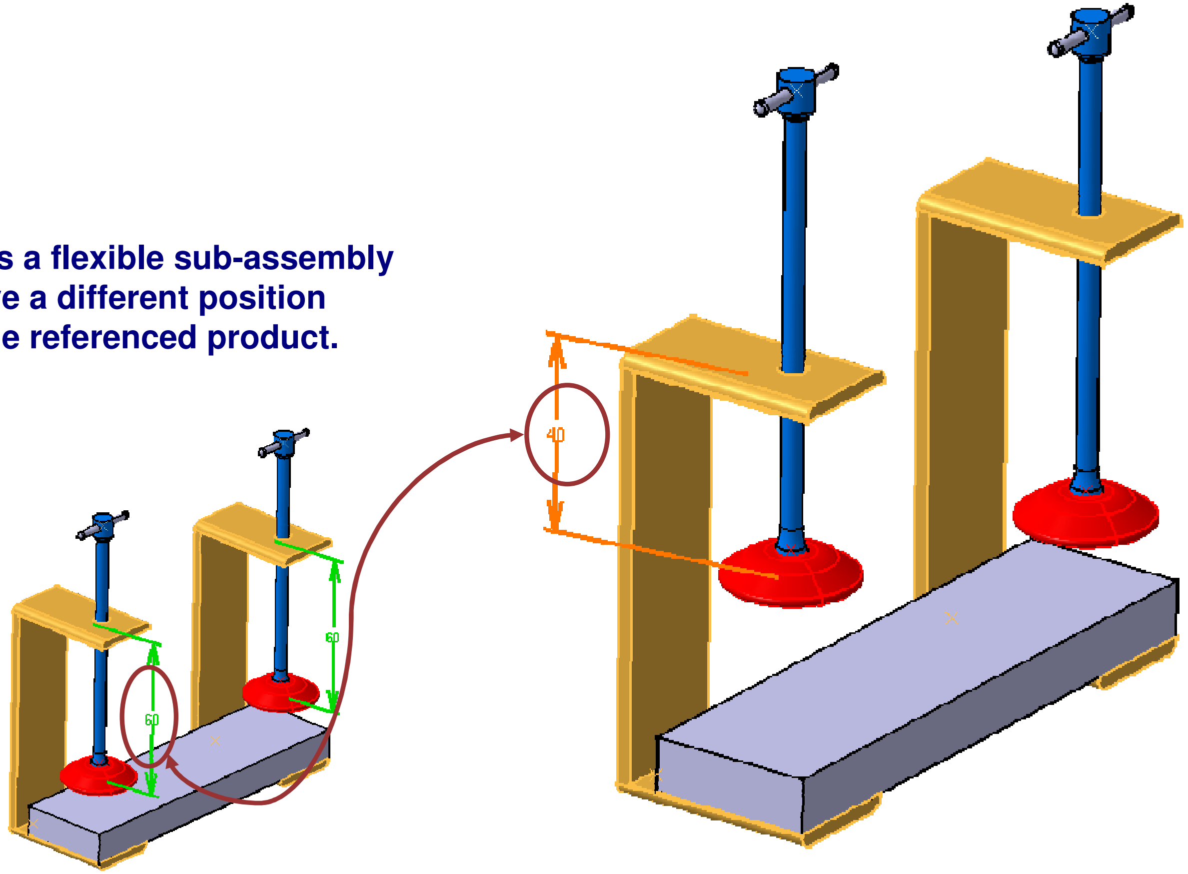
- Introduction to the Flexible Sub-assemblies
- Flexible Sub-Assemblies
- Using Flexible Sub-Assemblies
- Managing Flexible Sub-Assemblies
- Propagating Position to Reference
- Recap Exercise: Engine Assembly
- To Sum Up

Student Notes:

Introduction to the Flexible Sub-assemblies

Products or assemblies contains components as part, and also assemblies named sub-assemblies which are able to be instantiate. These instantiation may not have the same configuration from one to the other in the root product.

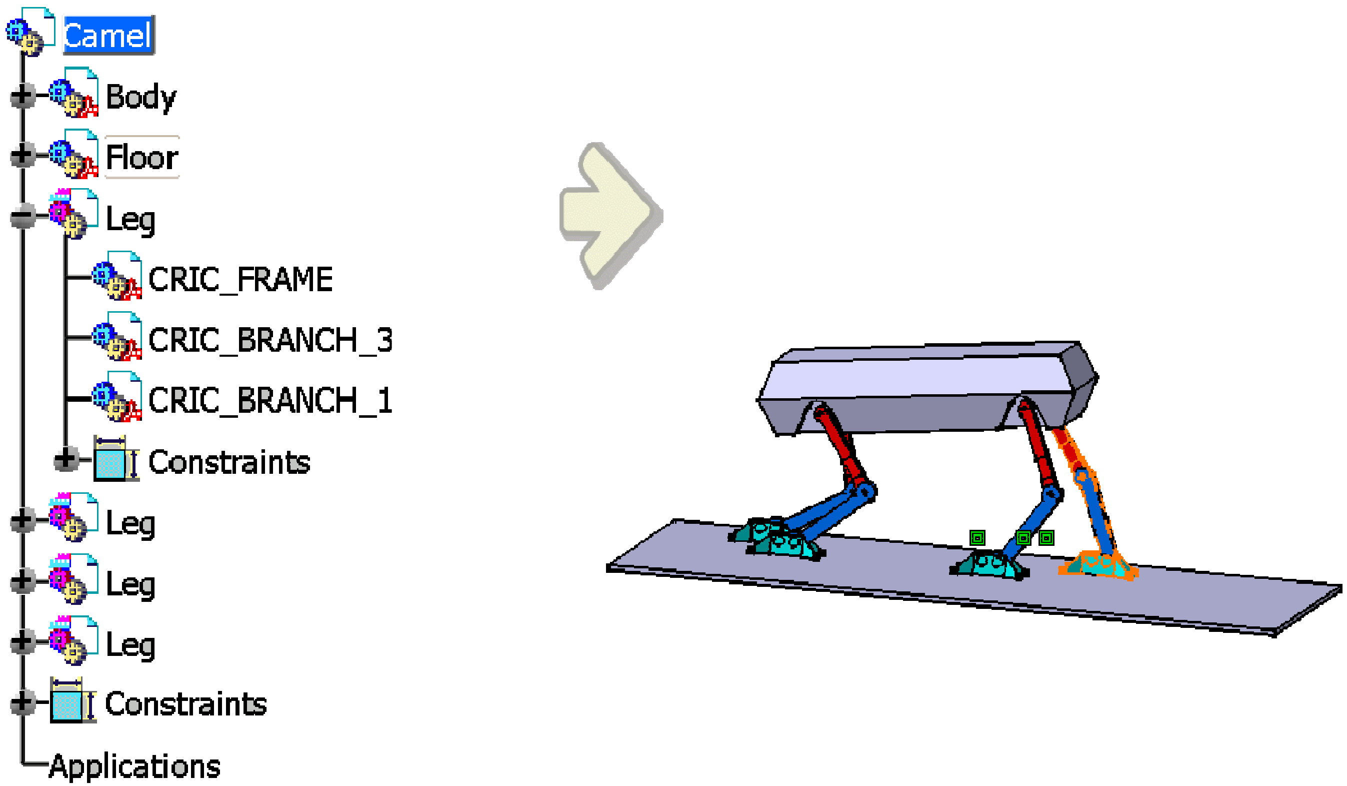
Defining a product as a flexible sub-assembly allow this one to have a different position without modifying the referenced product.



Student Notes:

Flexible Sub-Assemblies

You will learn how to make an assembly "flexible" thus allowing you to change the position on the fly of its parts without changing the stored assembly

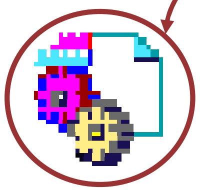


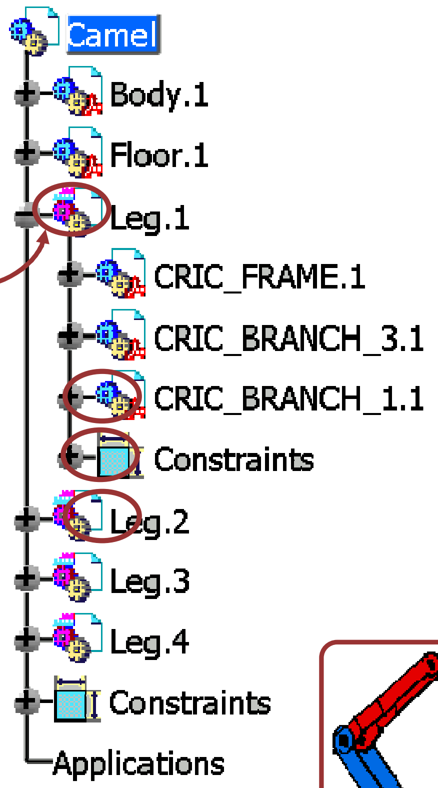
Student Notes:

What Are Flexible Sub-Assemblies?

A flexible Sub-Assembly is a Sub-assembly whose child components can be moved disregarding the fact it is not the active component. Relative positions of its child components can be different than those stored in the reference CATProduct File.

The purple little wheel to the left corner of the CATProduct icon identify a flexible sub-assembly

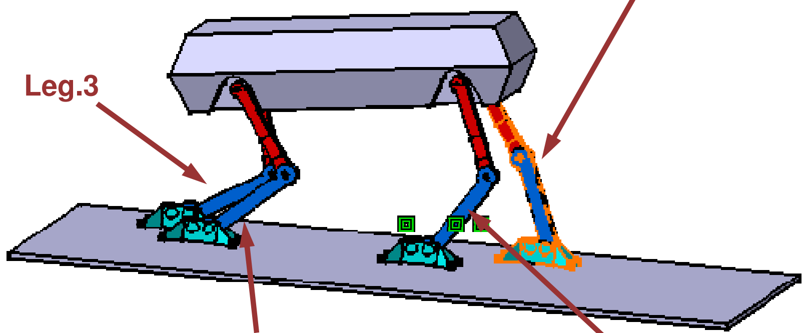




There you can see 4 instances of the Leg sub-Assembly : notice that the components of leg are not positioned the same way in each instance ; this is possible because all instances of Leg Assembly are flexible

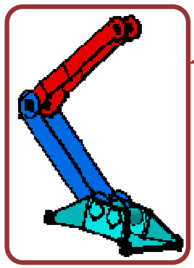
Leg.1

Leg.3



Leg.4

Leg.2



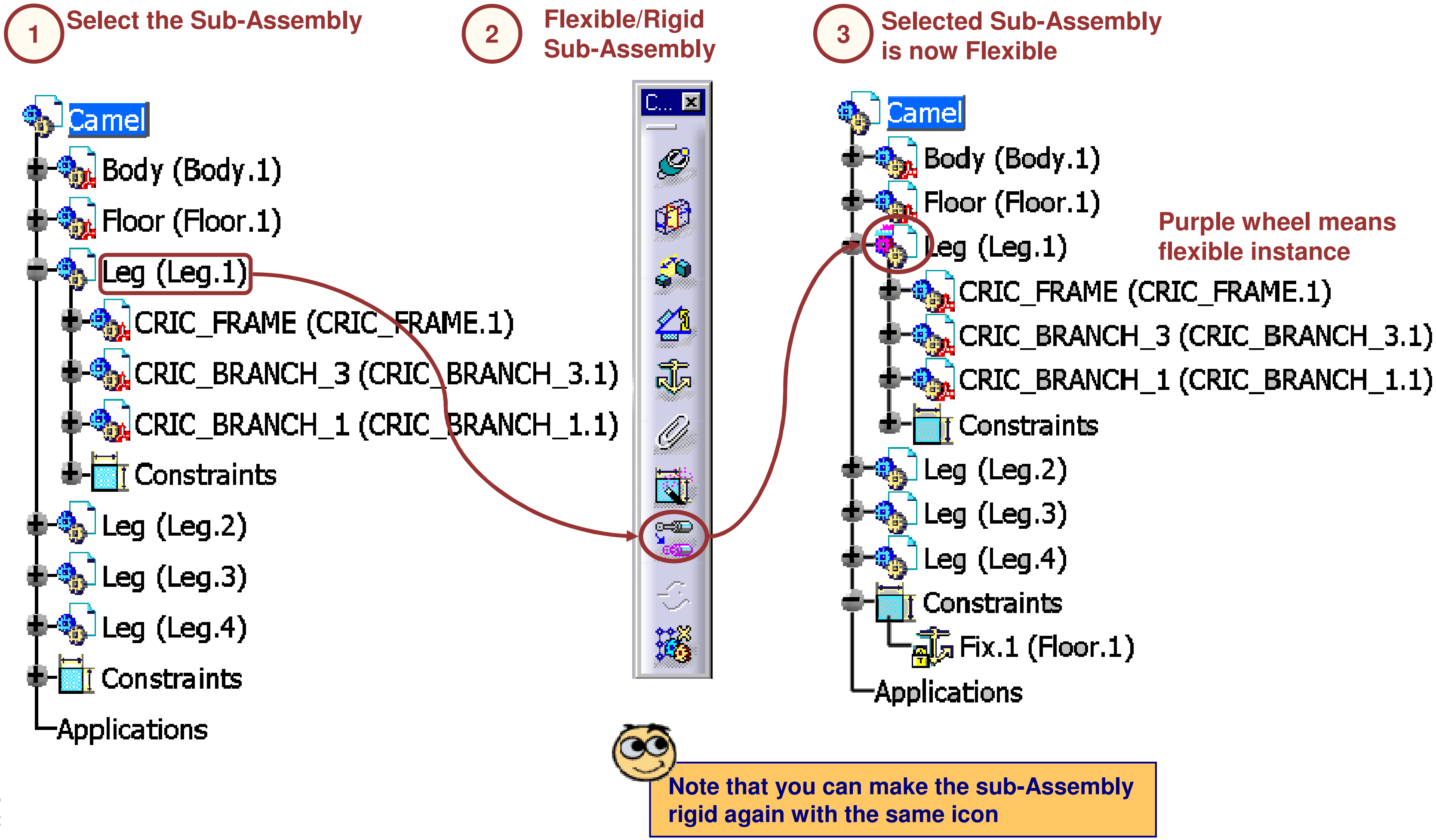
Leg

Relative positions of components in the reference of the Sub-Assembly

Student Notes:

Making a Sub-Assembly Flexible

Rigid/Flexible sub-Assembly tool is a switch : you click once to make an assembly flexible and you click on it again to make the assembly rigid

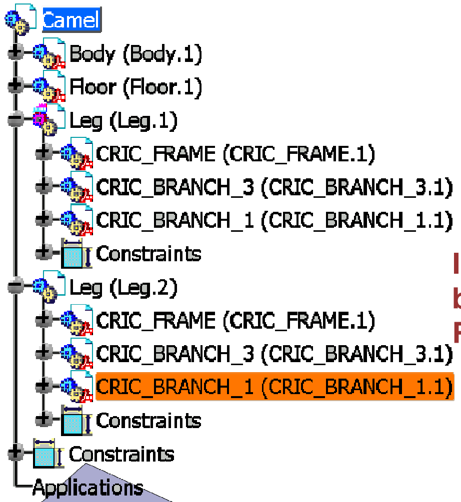


Student Notes:

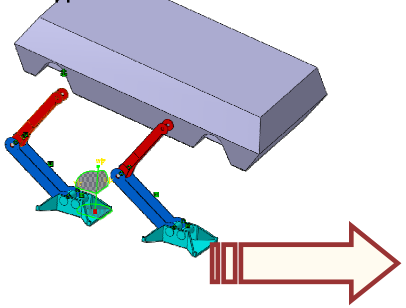
Positioning Components Of a Flexible Sub-Assembly(1/2)

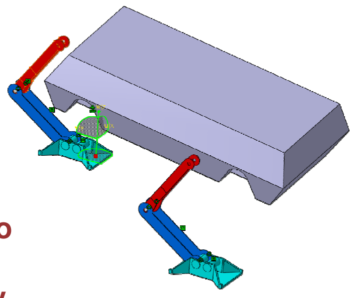
You can position components by freely moving them with the compass or by constraining them. In both cases the Rigid/Flexible state is important.

Freely moving Components

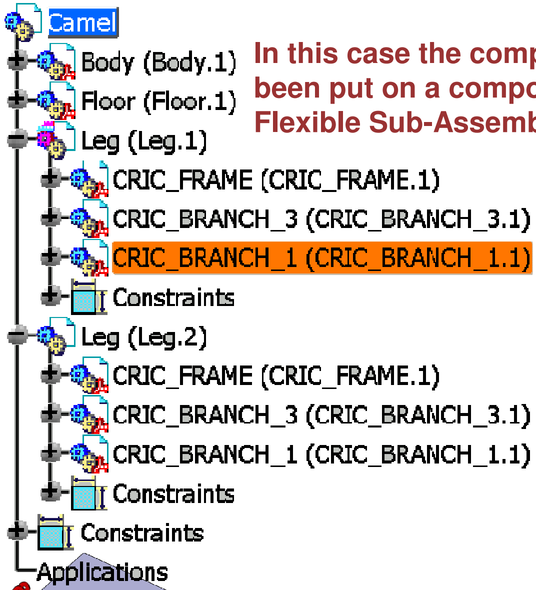


In this case the compass has been put on a component of a Rigid Sub-Assembly.

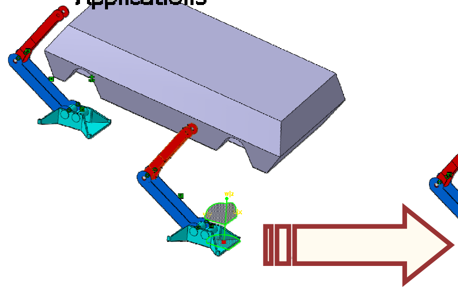


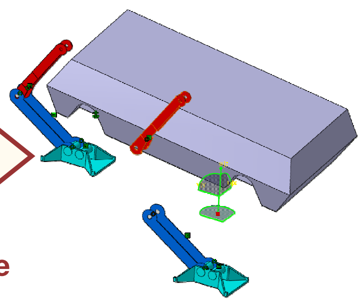


Root assembly is active, so the compass will drag the whole Rigid Sub-Assembly



In this case the compass has been put on a component of a Flexible Sub-Assembly.





Root assembly is active, but as the sub-Assembly is flexible, the compass will only move the selected component

Student Notes:

Positioning Components Of a Flexible Sub-Assembly(2/2)

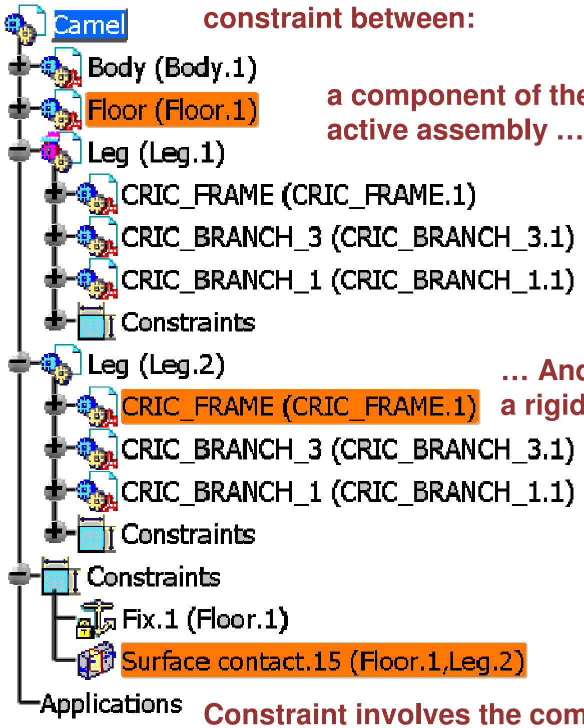
Relative Positions of components of a Flexible Sub-Assembly are stored with instance information in containing CATProduct.

Constraining Components

When you create a constraint between:

a component of the active assembly …

… And a component of a rigid sub-assembly …



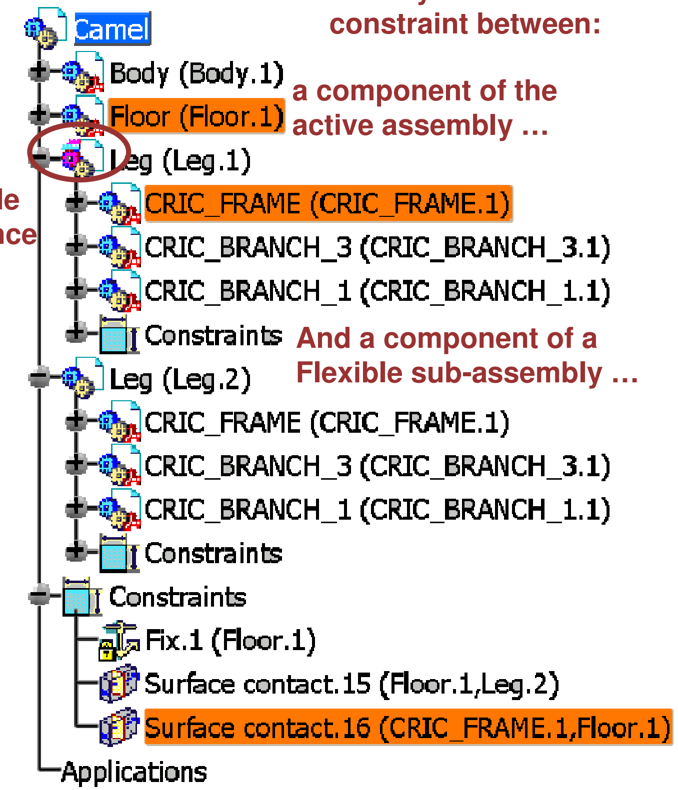
Constraint involves the component and the whole Rigid Sub-Assembly

When you create a constraint between:

a component of the active assembly …

flexible instance

And a component of a Flexible sub-assembly …



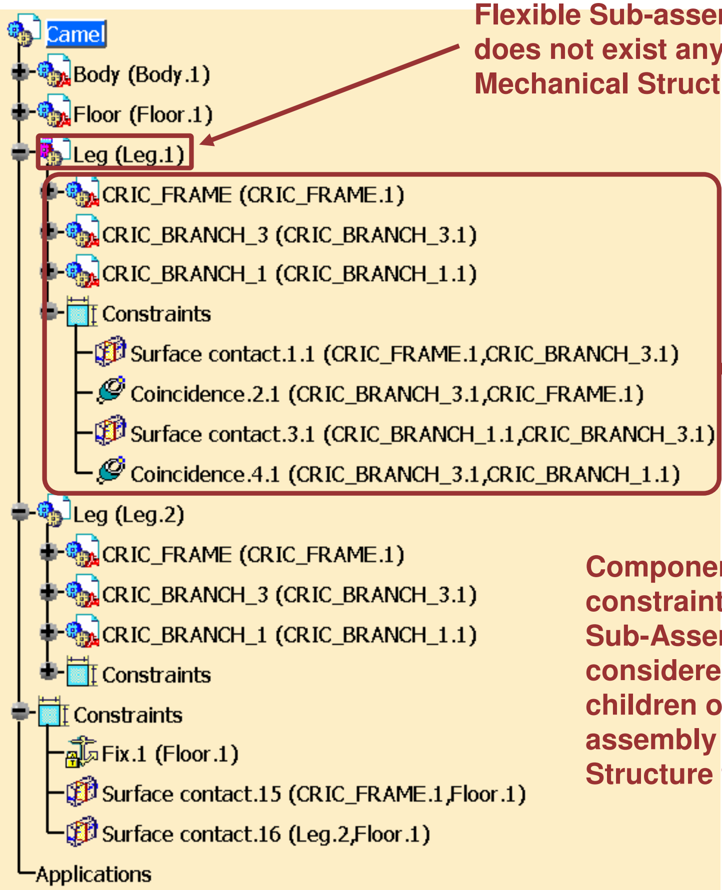
Constraint involves the component and the selected component of the Flexible sub-assembly

Student Notes:

What Is Mechanical Structure?

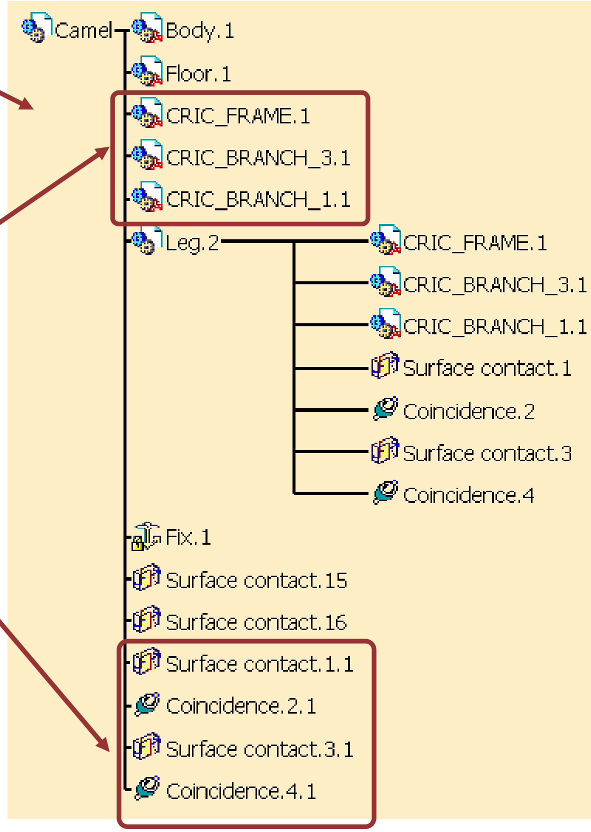
There are two types of structure when you use flexible Sub-Assemblies

Product Structure



Flexible Sub-assembly does not exist anymore in Mechanical Structure tree

Mechanical Structure



Components and constraints of Flexible Sub-Assemblies are considered as direct children of the root assembly in mechanical Structure tree

Product Structure Tree shows which assemblies and sub-assemblies Parts and constraints belong to

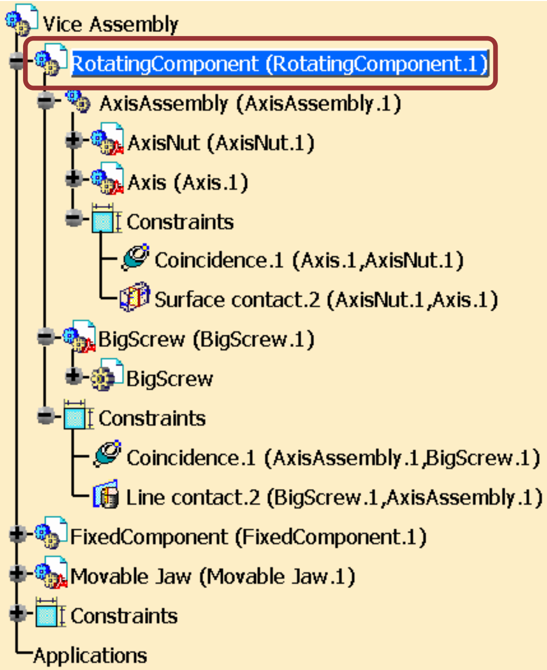
Mechanical Structure Tree shows what components you can constrain together (they are at the same level)

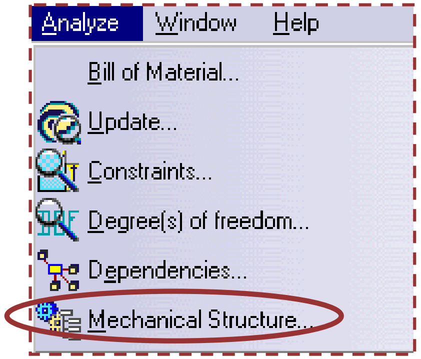
Student Notes:

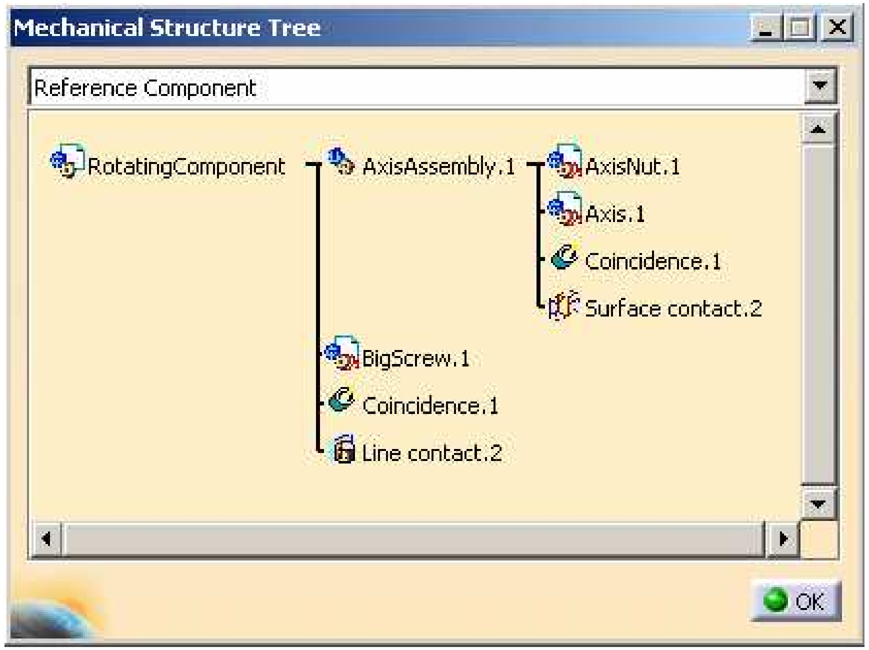
Viewing Mechanical Structure

1 Activate the Assembly or Sub-Assembly you want to analyze

2 Select Mechanical Structure from Analyze menu



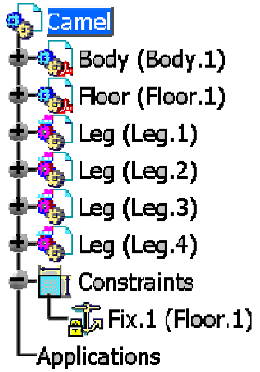


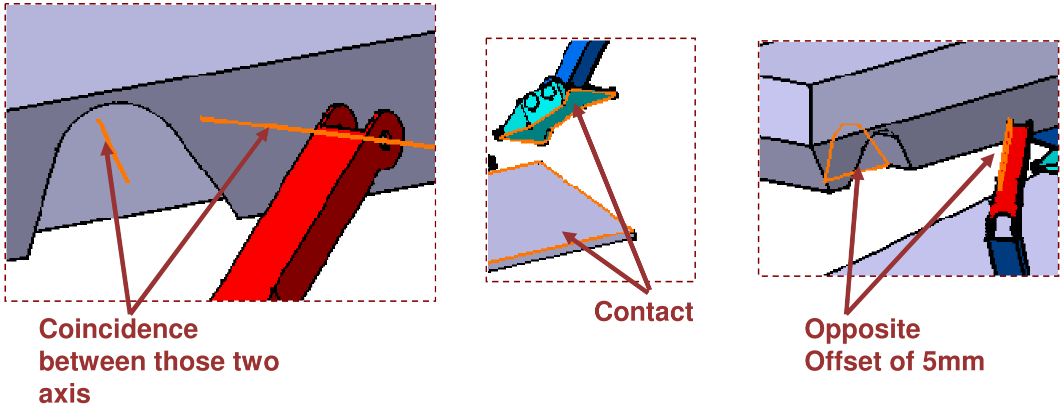


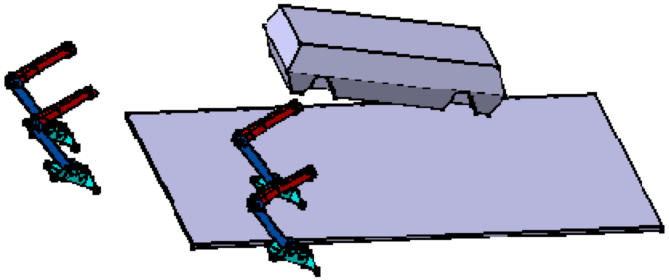
Student Notes:

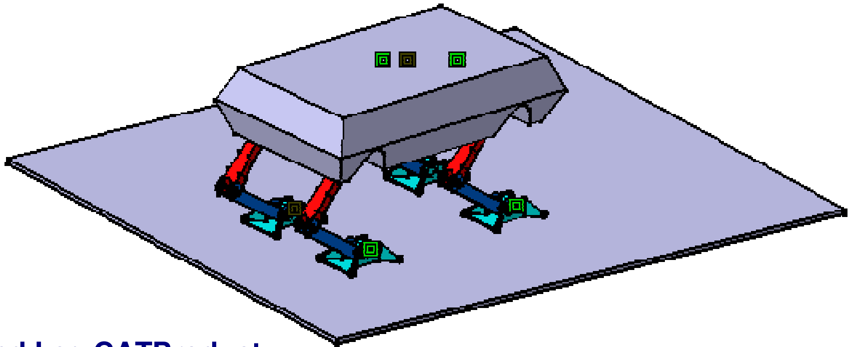
Do It Yourself

CATASMFlexibleSubAssembly.CATProduct, Leg.CATProduct





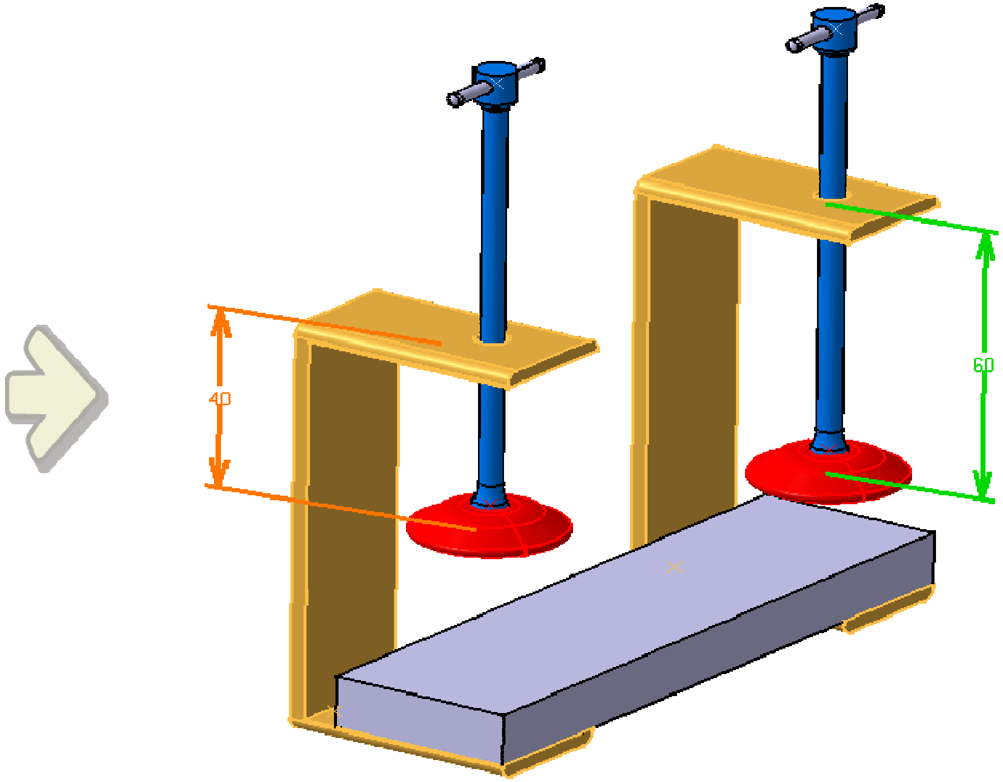




1- Load CATASMFlexibleSubAssembly.CATProduct and Leg.CATProduct
2- Insert four instances of Leg.CATProduct in your assembly
3- Constrain each Leg as shown below
4- Try to move the Body component with respect to constraints
5- Make all Leg instances flexible
6- Try to move the body again with respect to constraints

Using Flexible Sub-Assemblies

You will learn to manipulate Flexible Sub-Assemblies

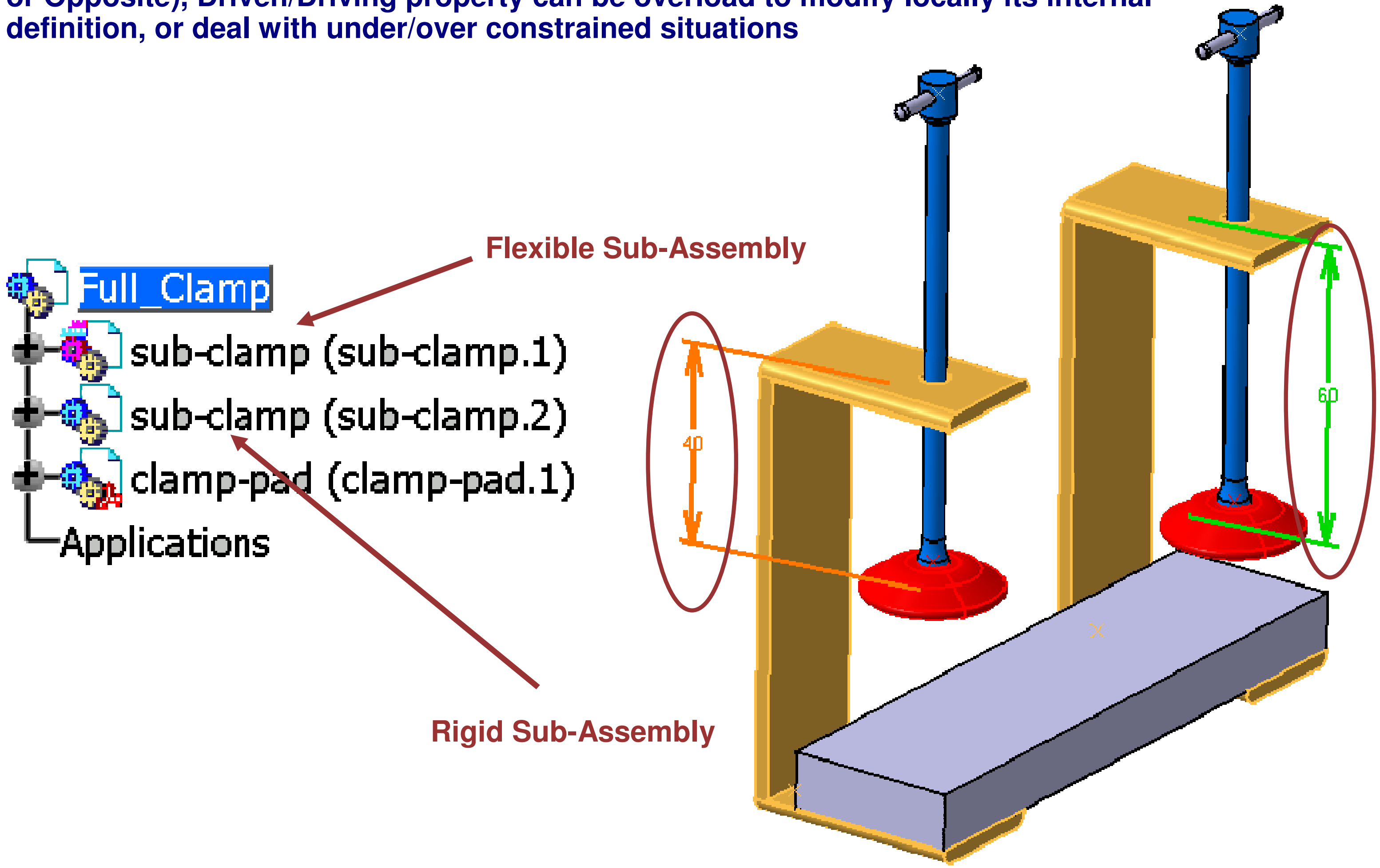


Student Notes:

Student Notes:

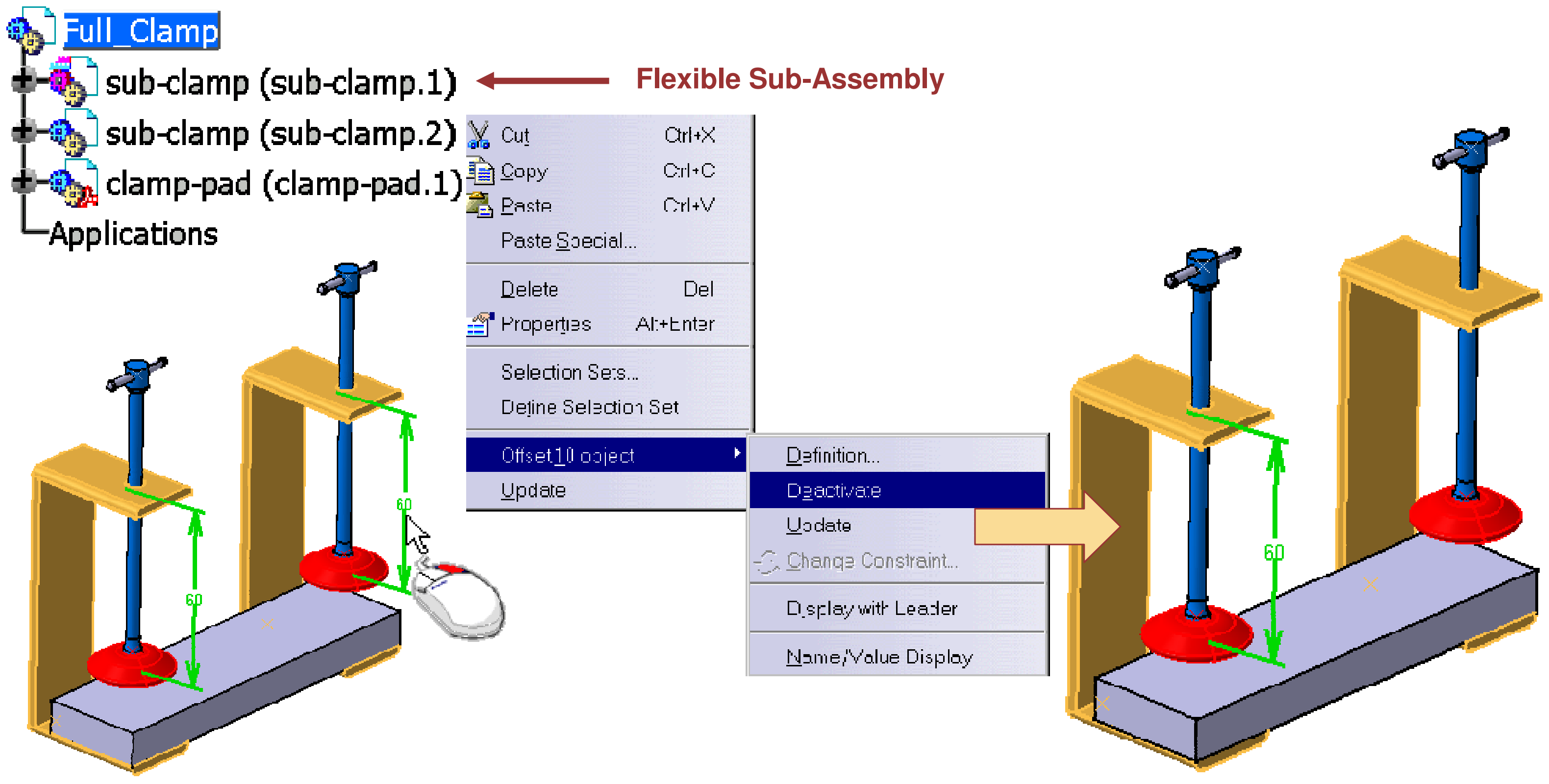
What Can You Overload with Flexible Sub-Assemblies?

Once the sub-assembly is flexible, Numerical Value, Activity status, Orientation (Same or Opposite), Driven/Driving property can be overload to modify locally its internal definition, or deal with under/over constrained situations



Student Notes:

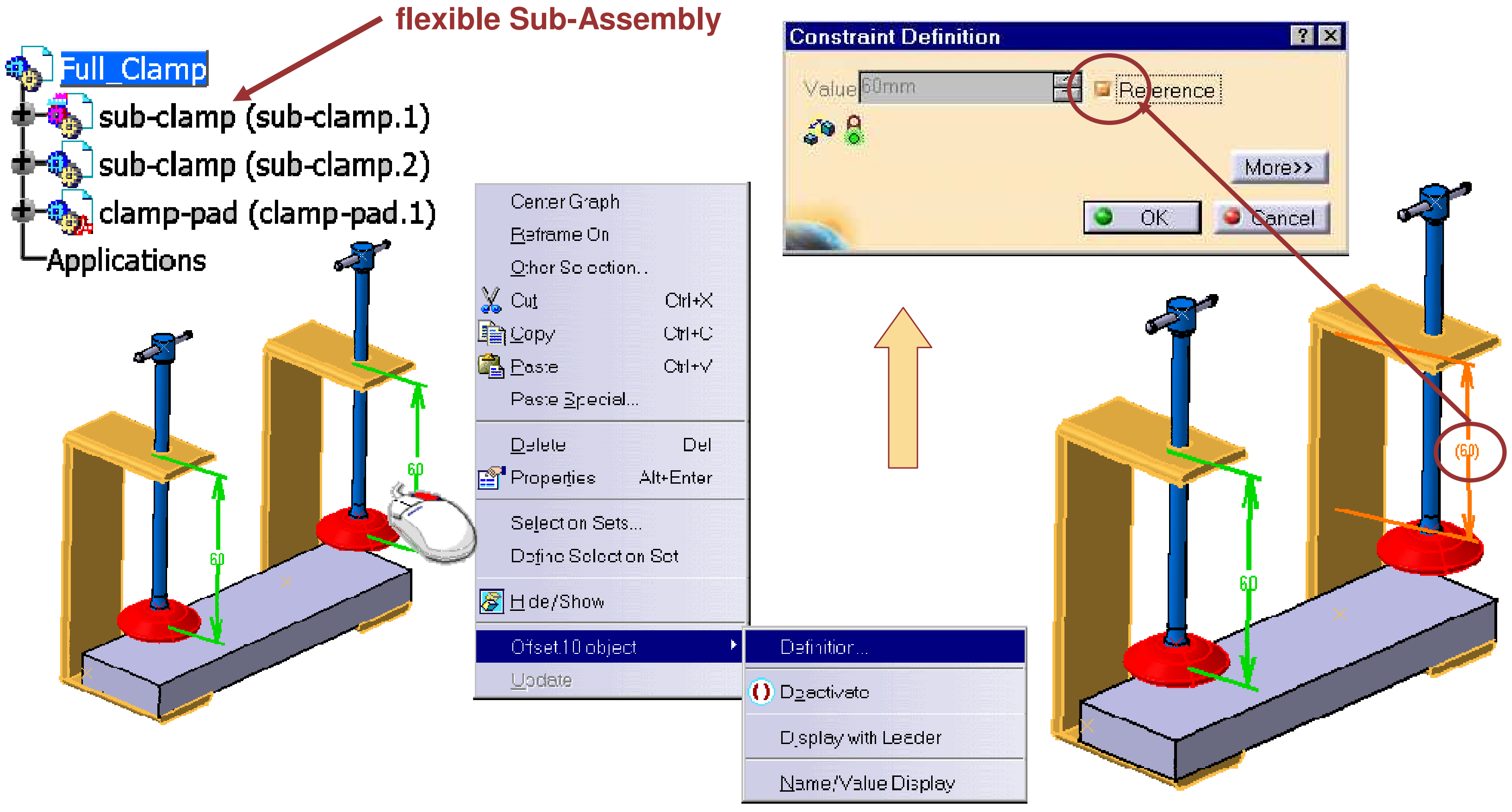
Activate / Deactivate Status



Concerning methodology using flexible sub-assemblies, you can change the Activity Status on a constraint

Student Notes:

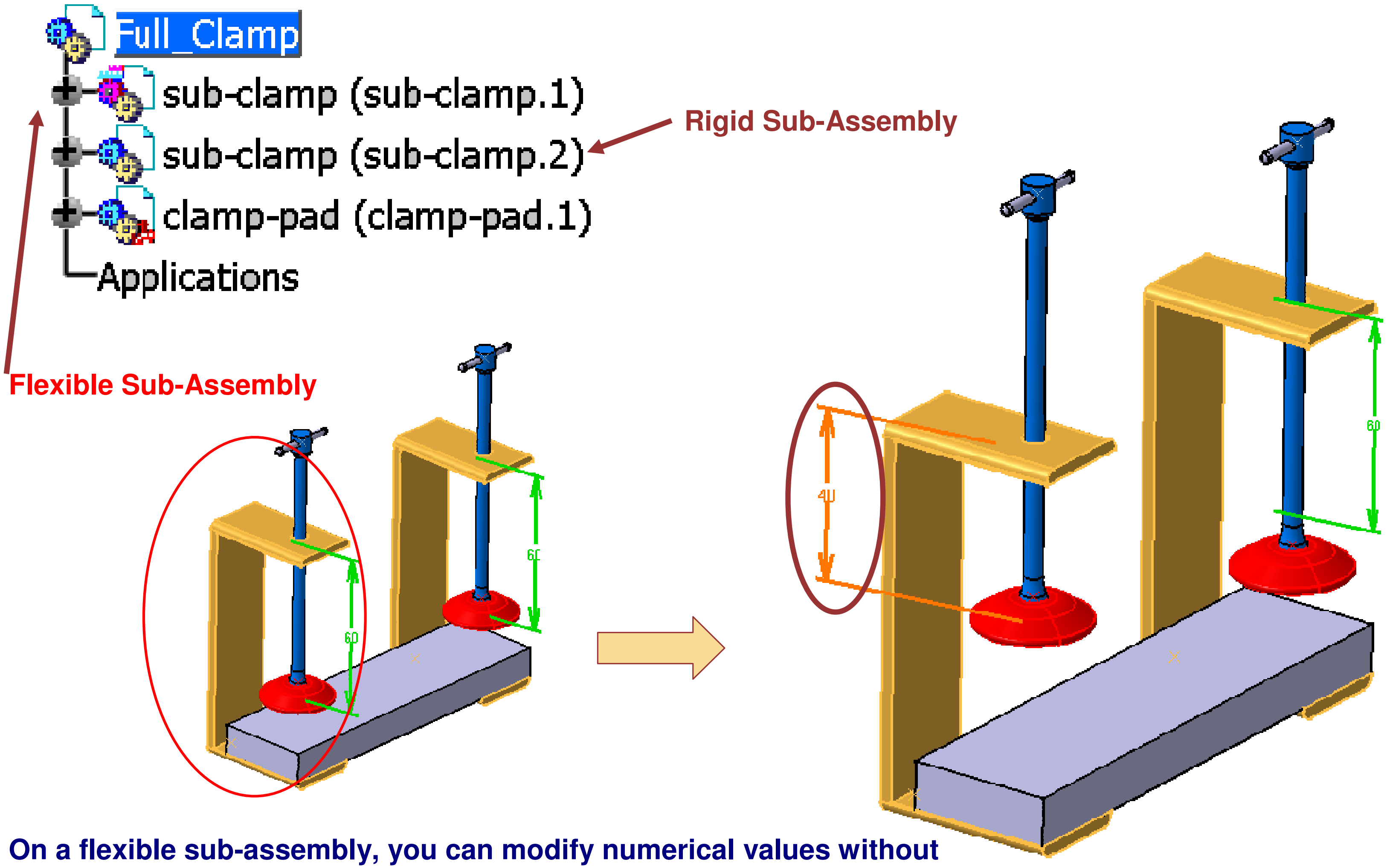
Driven / Driving Property



Concerning methodology using flexible sub-assemblies, you can toggle the driven / driving status on a contraint.

Student Notes:

Numerical Value

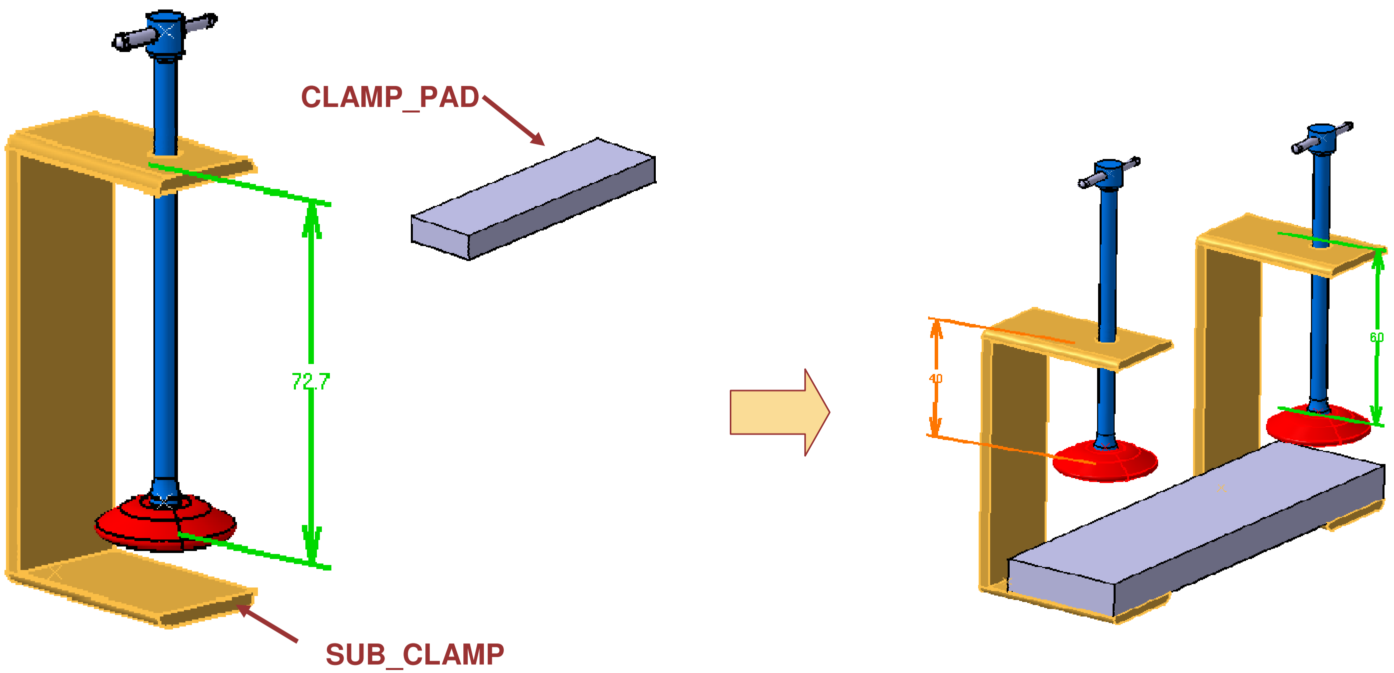


On a flexible sub-assembly, you can modify numerical values without impacting others instances

Student Notes:

Do It Yourself

Use SUB_CLAMP.CATProduct, CLAMP_PAD.CATPart

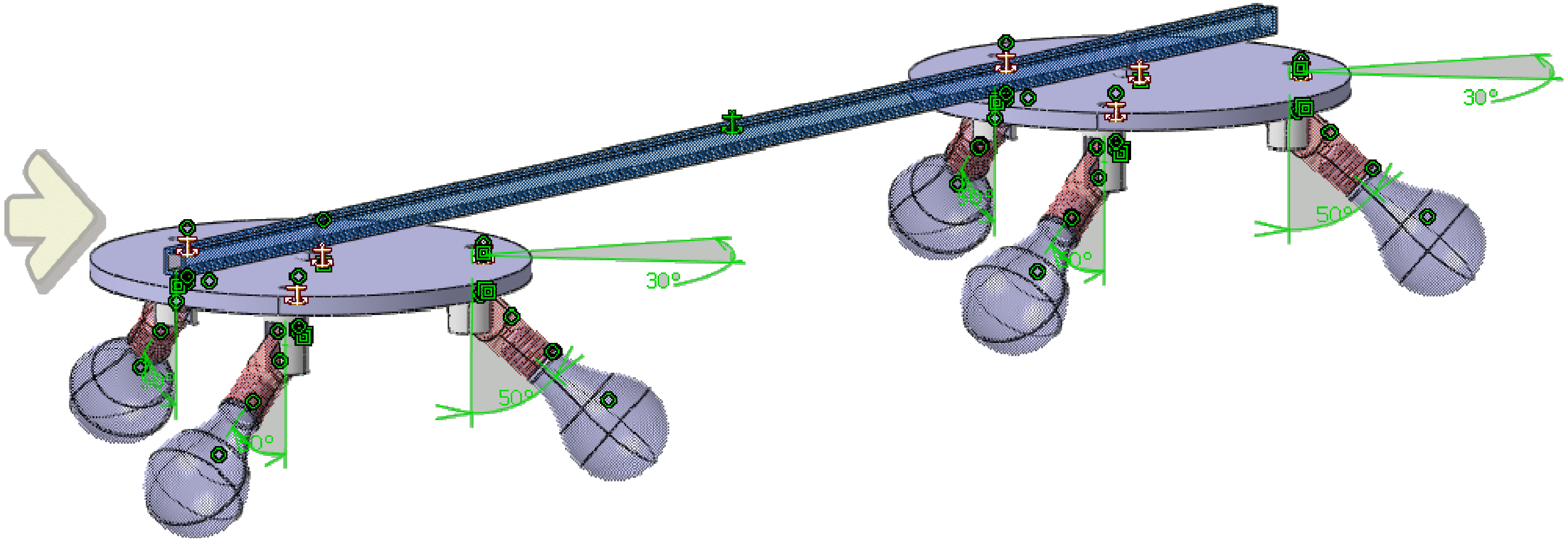


1- Create a new Assembly
2- Insert two instances of SUB_CLAMP.CATProduct and one instance of CLAMP_PAD.CATPart in your assembly
3- Constrain in design intent your assembly
4- Make one instance of SUB_CLAMP flexible
5- Try to get this following mechanical structure

Student Notes:

Managing Flexible Sub-Assemblies

You will learn how to use and see impacts of flexible / rigid command on a large assembly which contains several levels of sub-assemblies

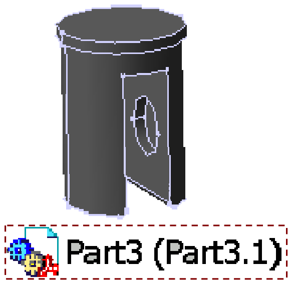


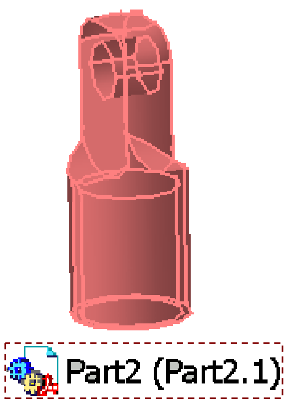
Student Notes:

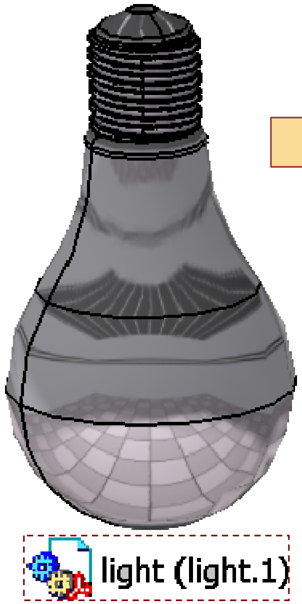
Description Of the Root Assembly (1/3)

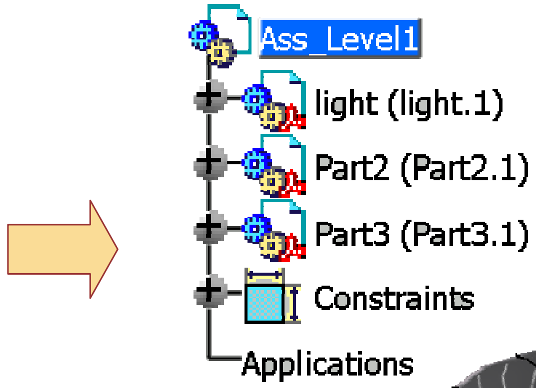
We start describing and explaining each level of the Root Assembly in order to see impacts of the Flexible/Rigid command :

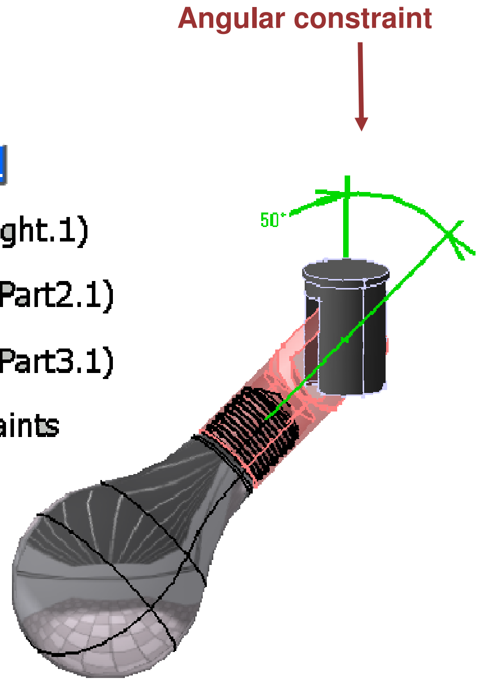
Components of the first level:











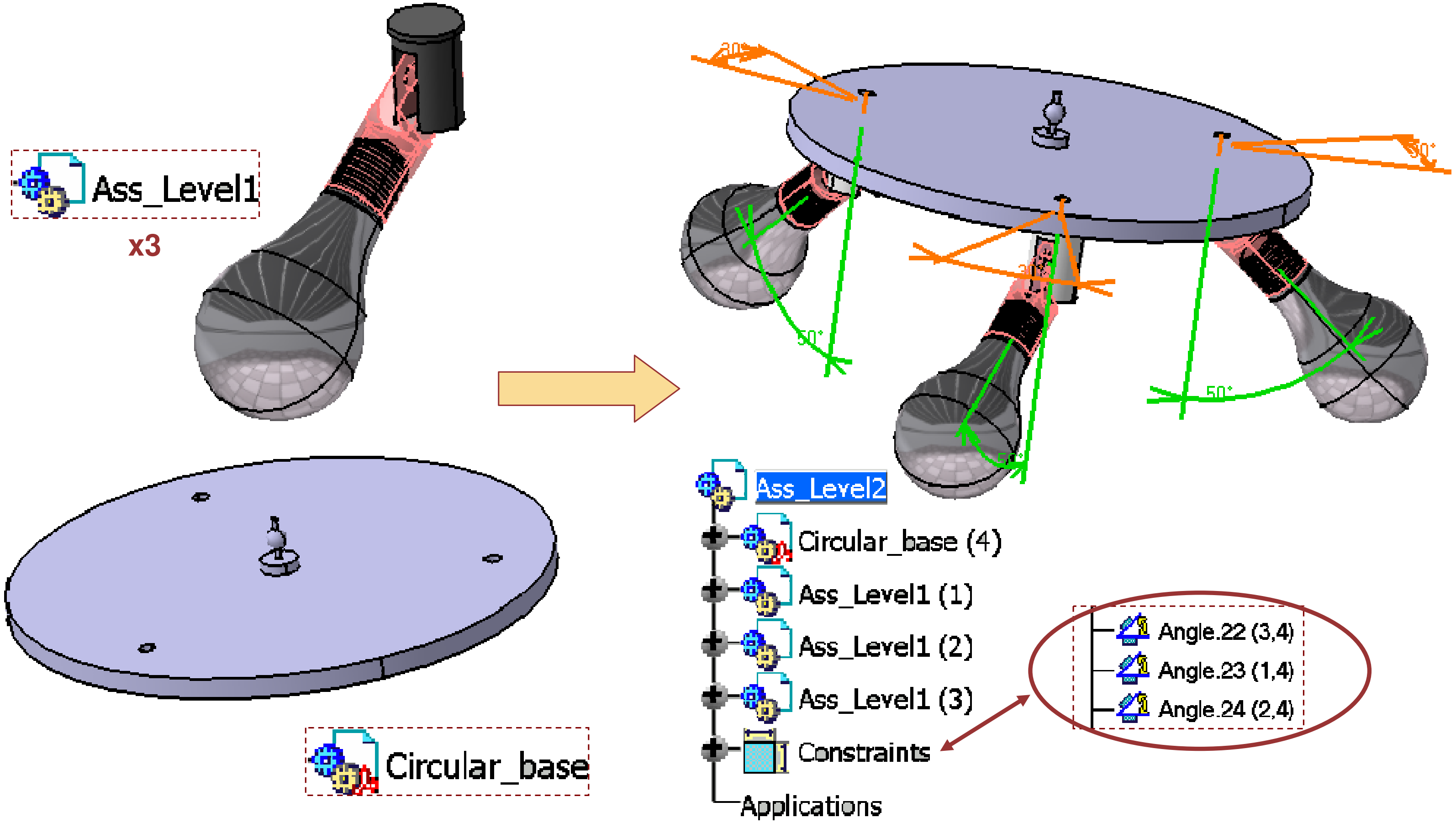
This assembly contains three parts. A driving angular constraint was created

Student Notes:

Description Of the Root Assembly (2/3)

Components of the second Level:

This assembly contains three instances of the last Product and one part. A specific angular constraint was created to position each sub-assembly to the part.

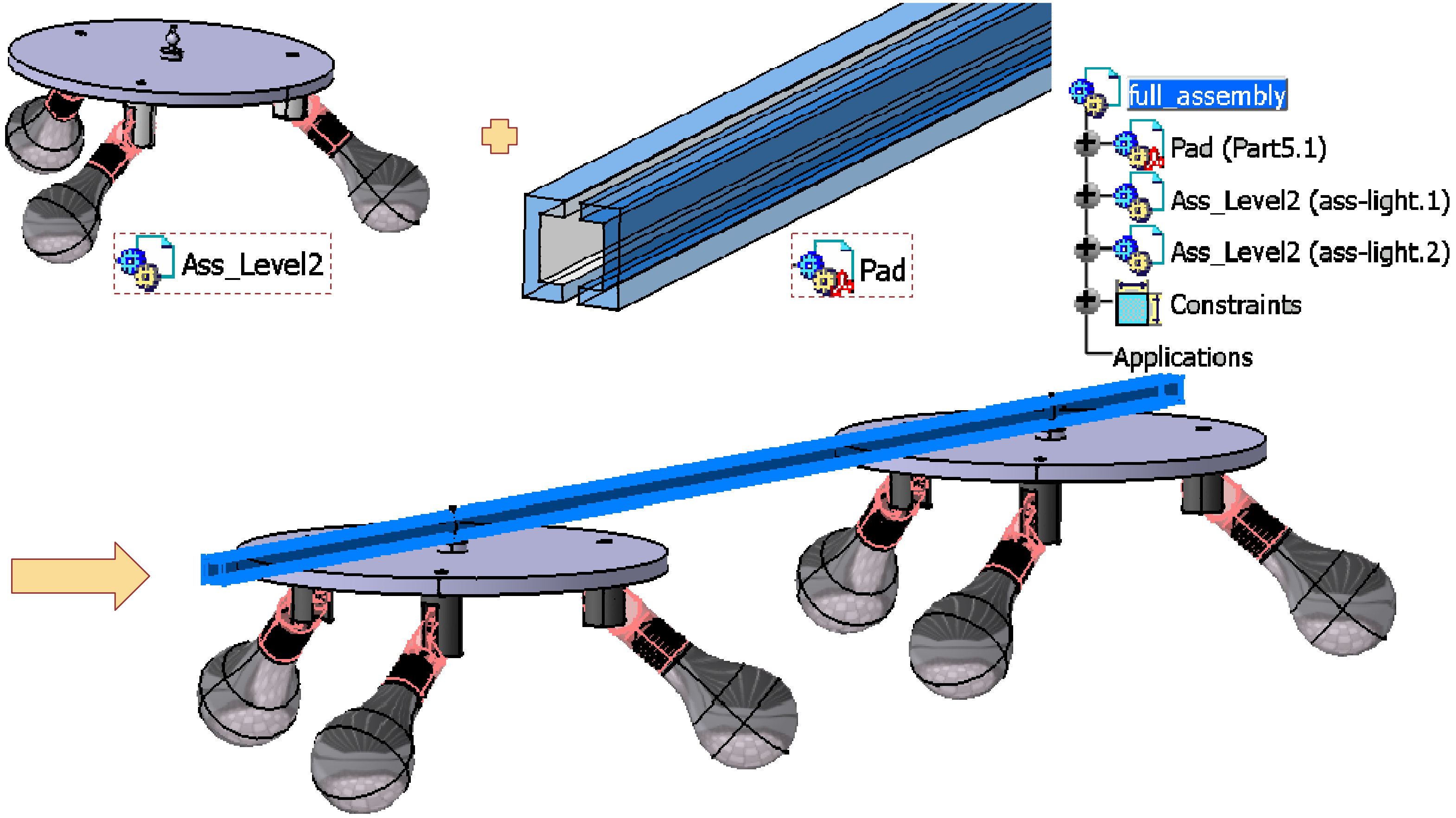


Student Notes:

Description Of the Root Assembly (3/3)

Hierarchy structure of the Root Level:

This assembly contains two instances of the level 2 .CATProduct and one part.

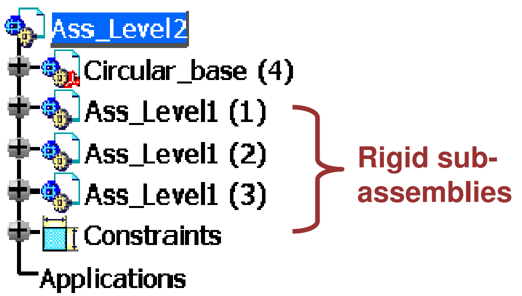


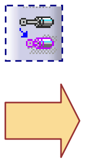
Student Notes:

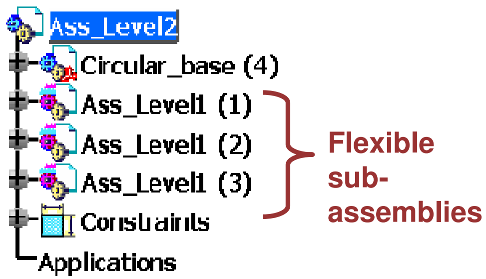
Use Flexible/Rigid Command On the Ass_Level2 Assembly

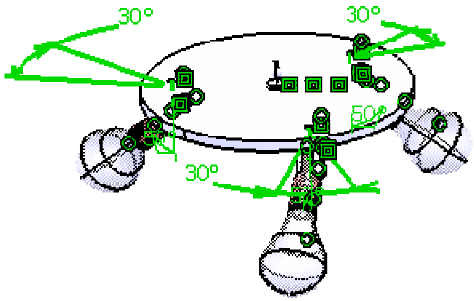
Use the Flexible / Rigid Command to overload position of child components of one product instance (Ass_Level1). You can modify value of angular constraint and change the sub-assembly configuration.

The other instances stay in the same relative position as in the reference CATProduct (Rigid Sub-Assembly)

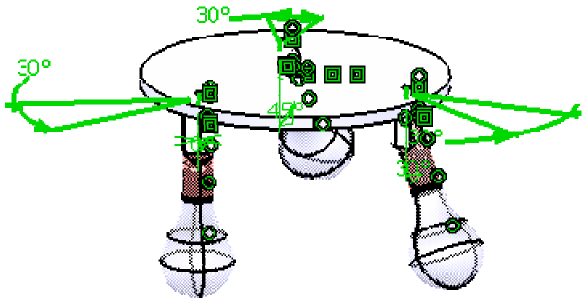








Rigid sub-assembly



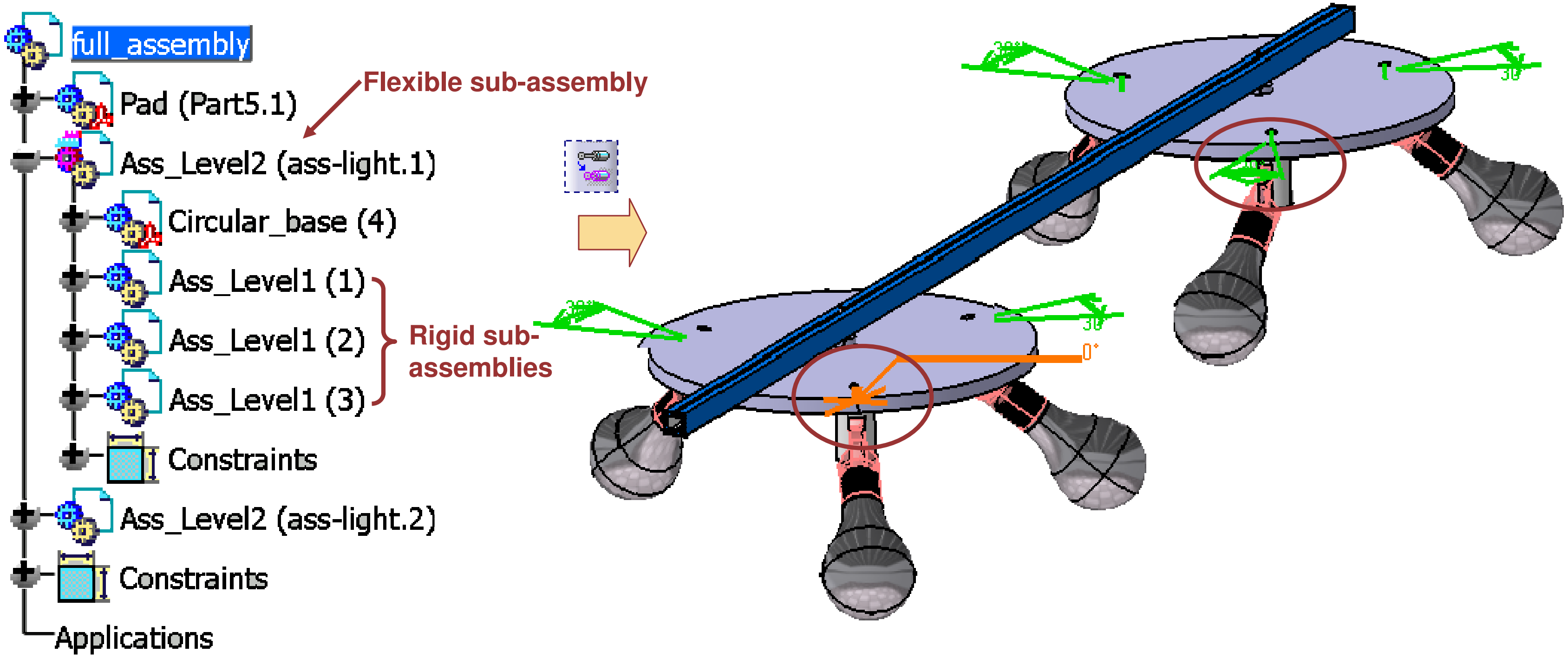
Flexible sub-assembly

Student Notes:

Use Flexible/Rigid Command On the Root Assembly

Using the Flexible/Rigid Command to overload position of child components of one product instance (Ass_Level2) does not impact mechanical structure of its child instances.

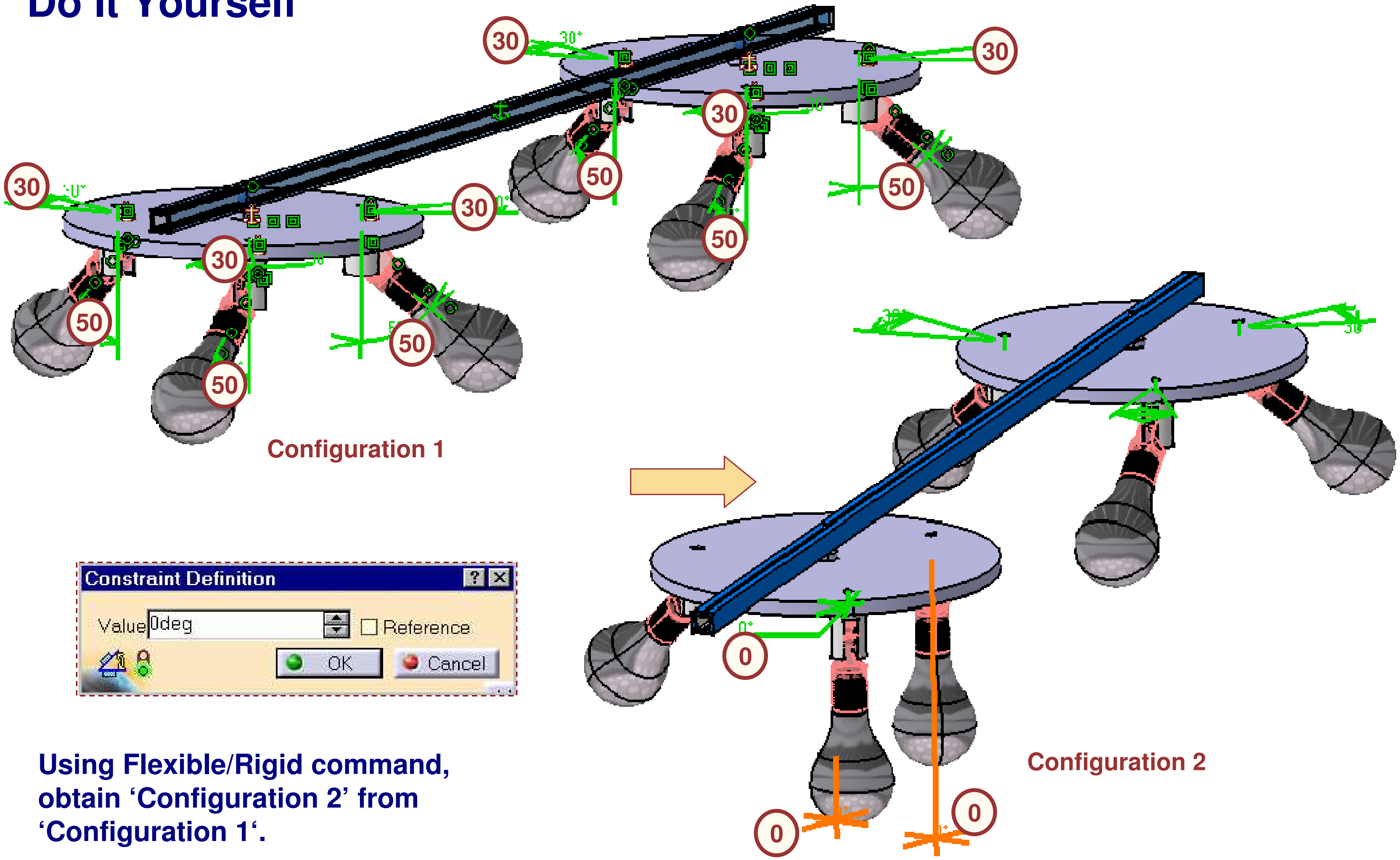
All product instances of inferior levels stay rigid (in the same relative position as in the respective reference CATProduct).



We can drive constraints of the ‘Ass_Level2 (1)’ instance without impacting mechanical structure of the ‘Ass_Level2 (2)’ instance. By default, all ‘Ass_Level1’ stay rigid.

Student Notes:

Do It Yourself



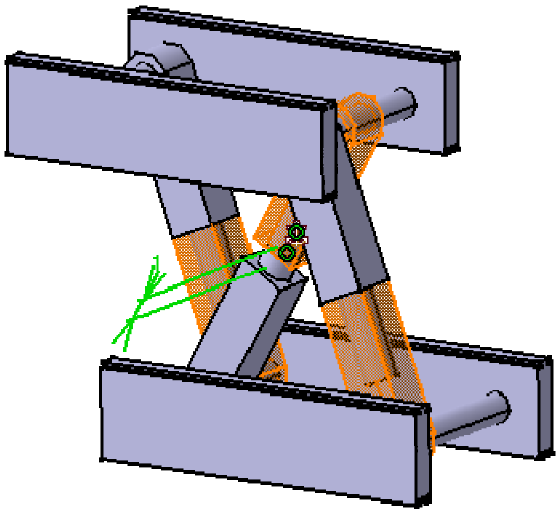
Using Flexible/Rigid command, obtain 'Configuration 2' from 'Configuration 1'.

Use "Full_Assembly_light.CATProduct"

Student Notes:

Propagating Position to Reference

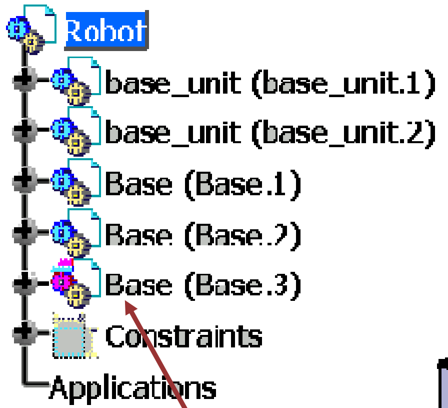
You will see what is propagating position to reference.



Student Notes:

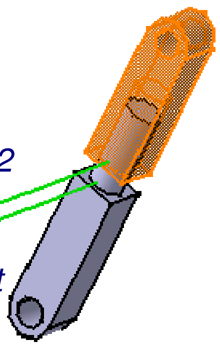
Propagating Position (1/2)

1 **Modify position of the Flexible Base Instance.**



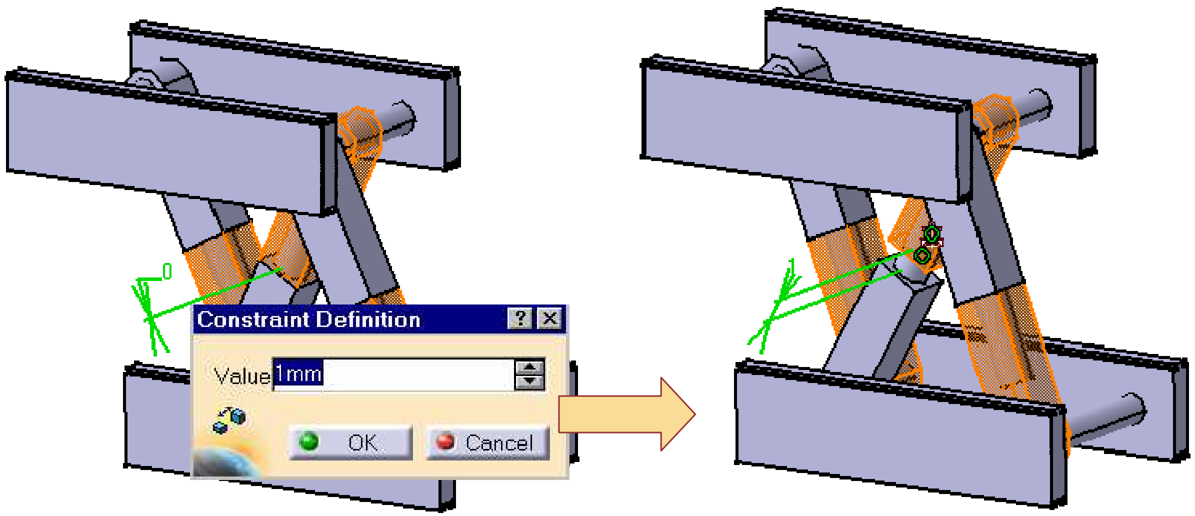
Flexible sub-assembly

This Product contains 2 sub-assemblies :
Base.CATProduct
Base_unit.CATProduct



Base (Base.3): **flexible instance**

Base (Base.1): **rigid instance**

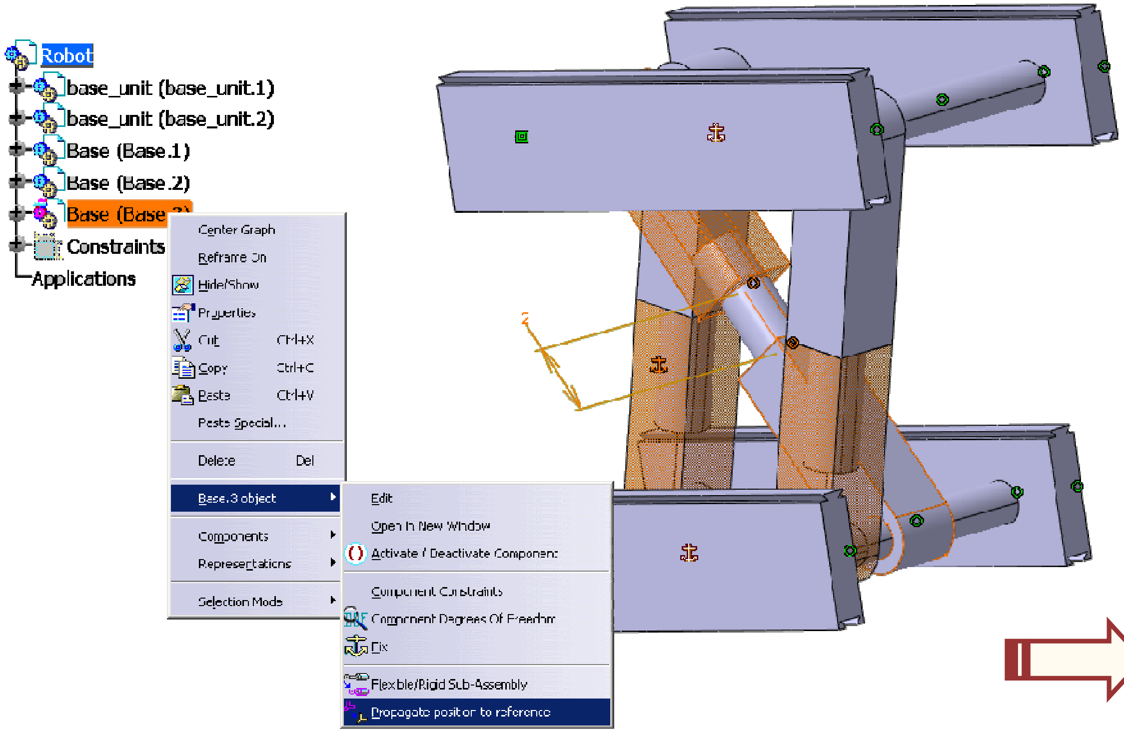


Student Notes:

Propagating Position (2/2)

2 **Apply overloaded position to reference**

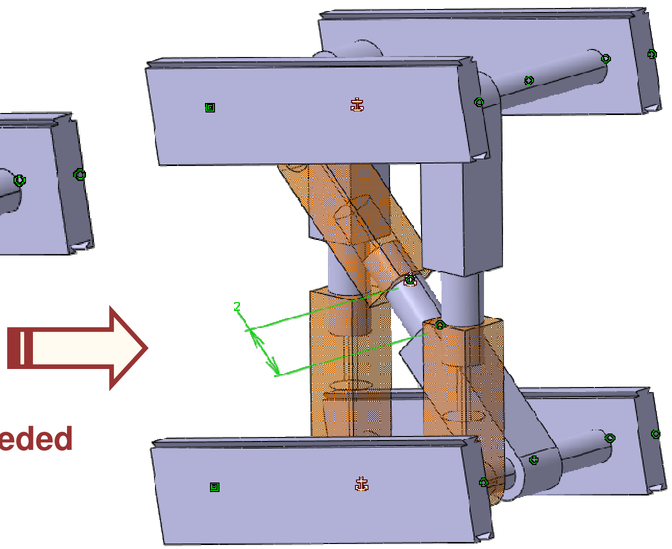
- *Select the flexible Base instance*
- *MB3+ ….object+Propagate position to reference*



Result : all rigid instances should have the same position than the flexible one.

Internal position of flexible instances are not impacted by the command.

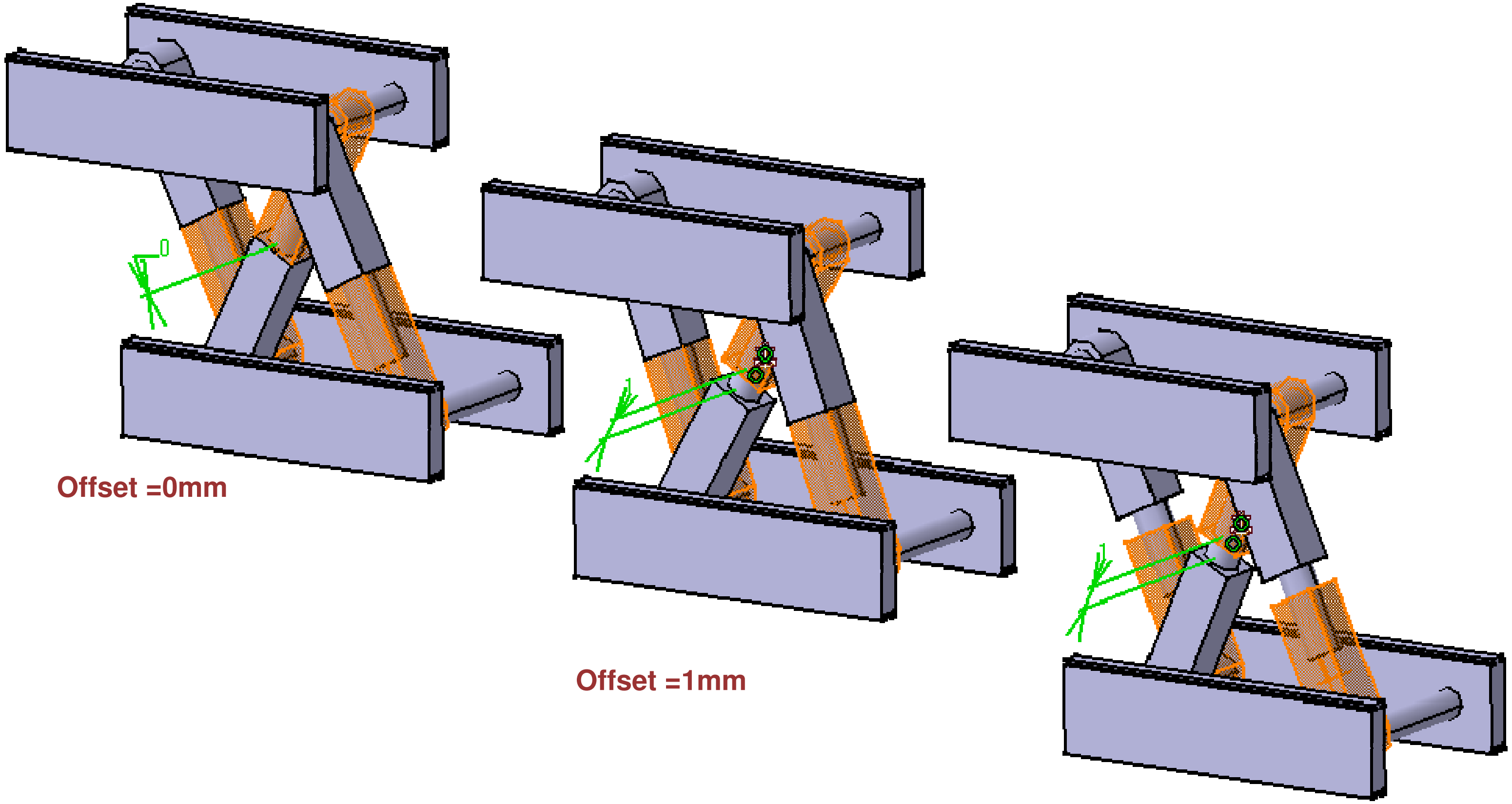
After an update if needed



Student Notes:

Do It Yourself

Use Robot.CATProduct.



- Modify Dimension to 1 mm.
- Propagate position to reference

Student Notes:

Engine Assembly

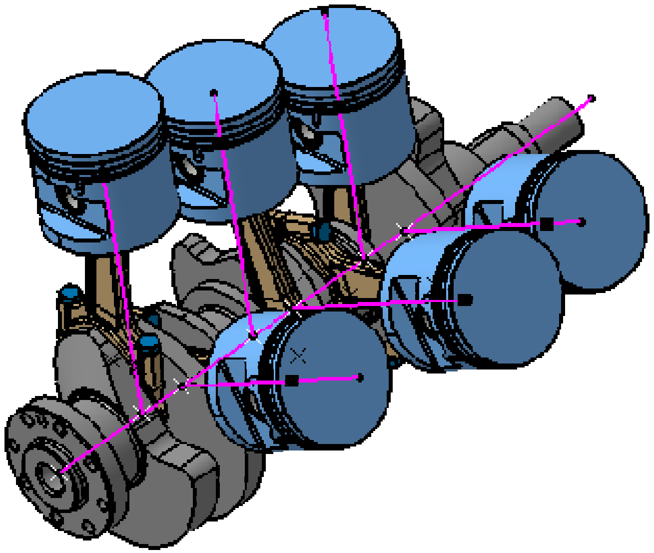
Recap Exercise : Flexible Assemblies

45 min

In this recap exercise you will create a piston assembly by inserting various components and constraining them.

You will then instantiate multiple instances of Piston assembly into Engine assembly, position them and make these sub assemblies flexible.

You will demonstrate behavior of flexible subassemblies by rotating the crank to see pistons and connected parts occupy various positions.



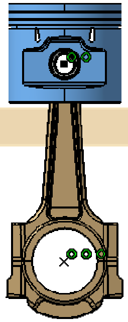
You will use following commands :

- Assembly Constraints
- Propagate position to reference
- Rigid/Flexible Assembly

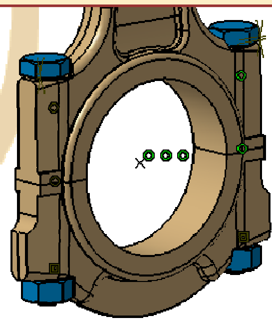
Student Notes:

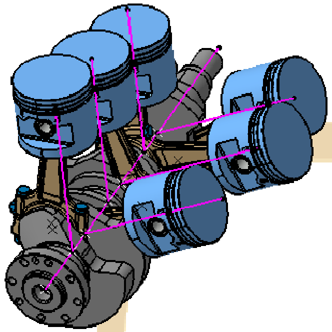
Design Process: Engine Assembly

1 Create a new Assembly and insert piston and other components and constrain them using assembly constraints.



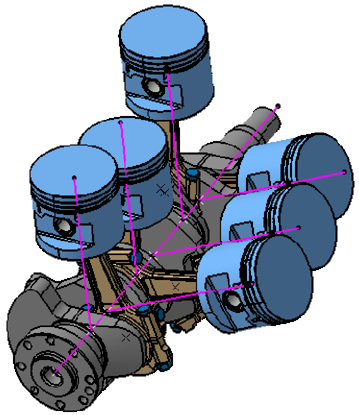
2 Insert a new CATProduct “Fixations” and insert multiple instances of Nut and Bolt.Constrain the Nut and Bolt.





3 Insert 6 instances of Piston assembly in Engine.CATProduct and apply assembly constraints. Make all instances of Piston assembly flexible

4 Modify / Add constraints, modify the position of crank. Piston positions are automatically updated.

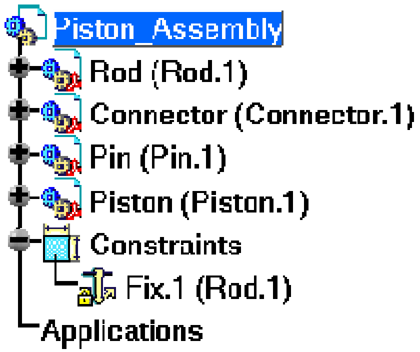


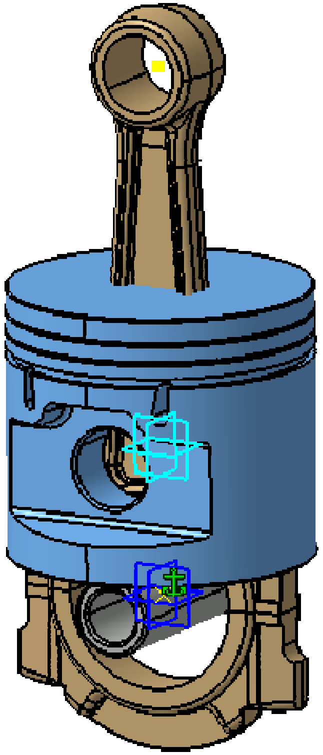
Student Notes:

Do It Yourself : Step 1 (1/2)

Parts used:Rod.CATPart, Connector.CATPart, Pin.CATPart, Piston.CATPart

- Create a new CATProduct "Piston_Assembly".
- Insert following CATParts in this product.
  - Rod.CATPart. Fix this part immediately after you insert it.
  - Connector.CATPart
  - Pin.CATPart
  - Piston.CATPart

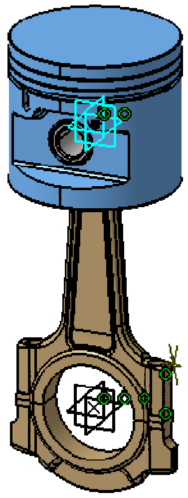


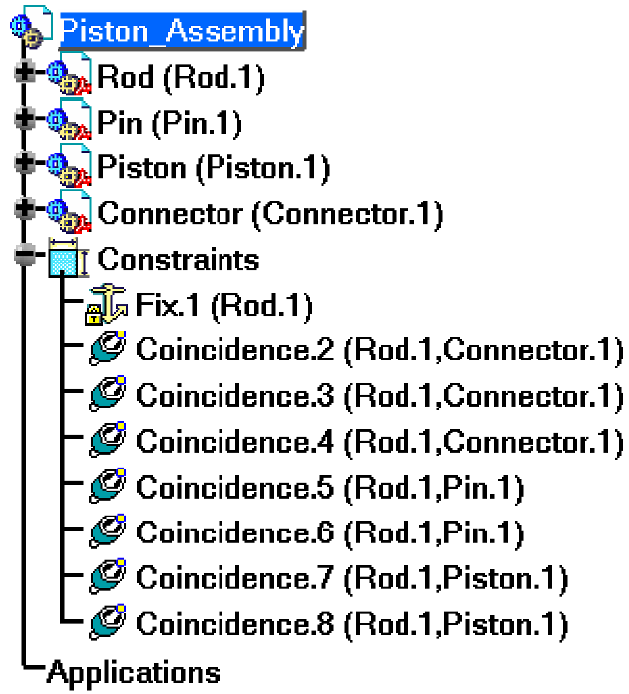


Do It Yourself : Step 1 (2/2)

In "Piston_Assembly", create following assembly constraints :

- **To position the 'Connector':**
  - **Coincidence constraint between xy, yz and zx planes of 'Rod' and 'Connector'**
- **To position the 'Pin' :**
  - **Coincidence between yz planes of 'Rod' and 'Pin'**
  - **Coincidence between "Pin_Axis" of 'Rod' and "Axis" of 'Pin'**
- **To position the 'Piston' :**
  - **Coincidence between yz planes of 'Rod' and 'Piston'**
  - **Coincidence between "Pin_Axis" of 'Rod' and "Hole_Axis" of 'Piston'**





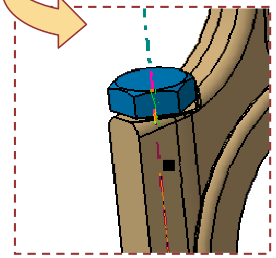
There is no degree of freedom between rod and connector. There is rotational degree of freedom between rod and pin, and between rod and piston about the 'Pin_Axis'.

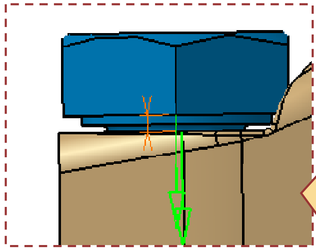
Student Notes:

Do It Yourself : Step 2 (1/4)

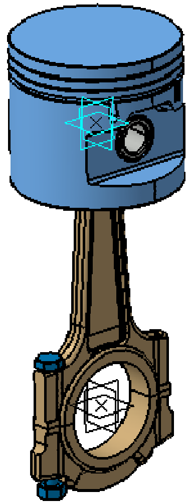
Parts used:Bolt_M8.CATPart, Nut_M8.CATPart

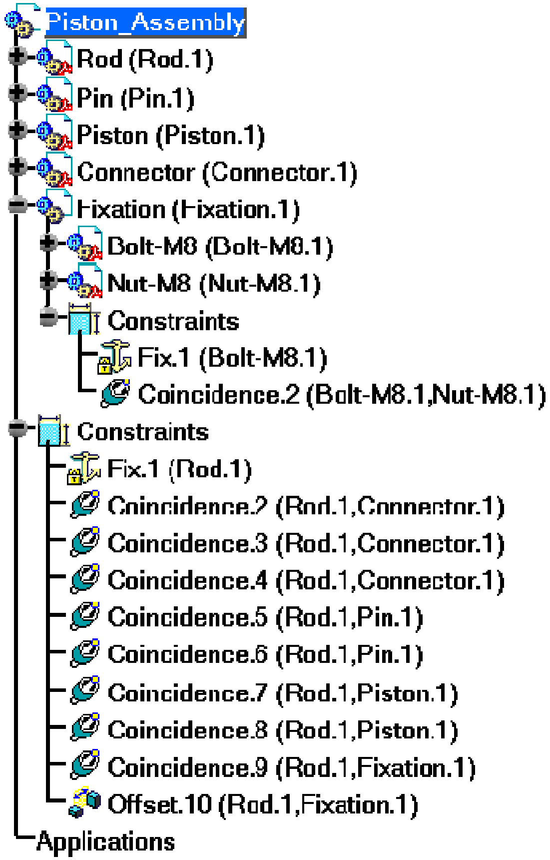
- **Create a new product named “Fixation” in ‘Piston_Assembly’.**
- **Insert ‘Bolt_M8.CATPart’ in ‘Fixation.1’ and fix it immediately.**
- **Insert ‘Nut_M8.CATPart’ in ‘Fixation.1’ and create :**
  - **Coincidence constraint between axes of nut and bolt.**
- **To position ‘Fixation.1’ Sub assembly create :**
  - **Coincidence between the published “Axis” of the ‘Bolt_M8’ and the geometrical Axis of left hole of the ‘Rod’**





  - **1mm Offset between “Mating-Face” of the ‘Bolt_M8’ and the Corresponding face of the ‘Rod’.**





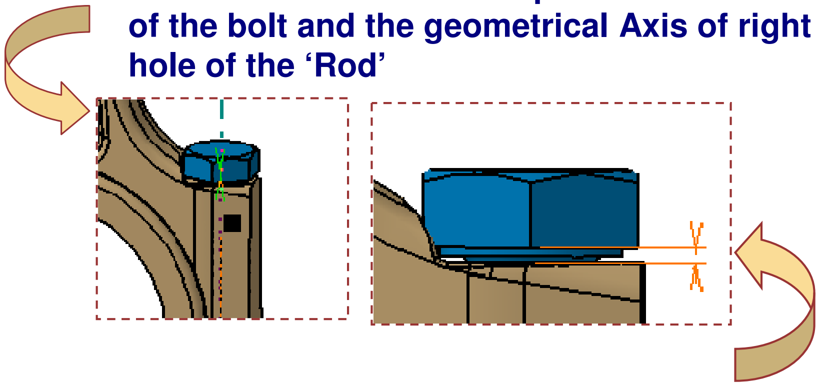
Fixation product is created to demonstrate the use of ‘Propagate Position to Reference’. In real-life the screws and bolts can be directly inserted in ‘Piston_Assembly’.

Student Notes:

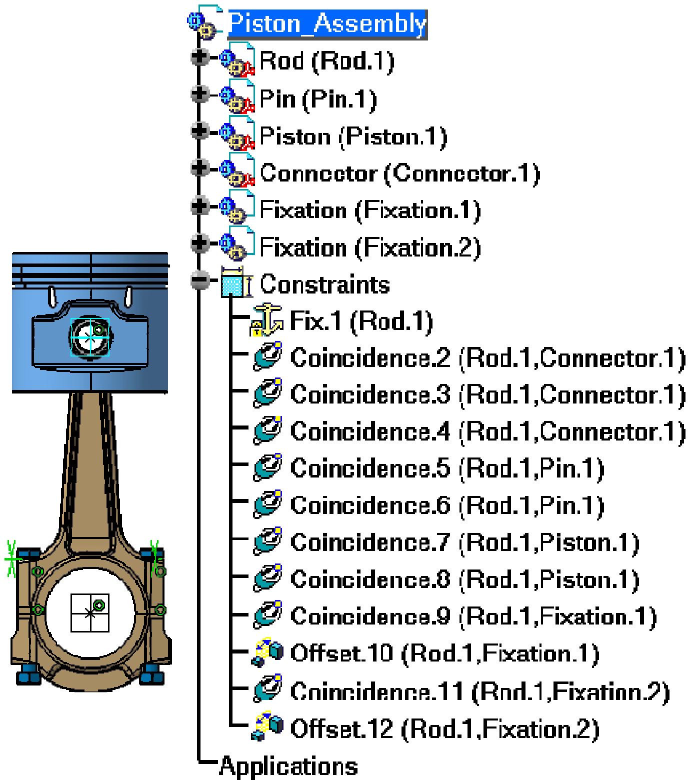
Student Notes:

Do It Yourself : Step 2 (2/4)

- Instantiate a second instance of 'Fixation.CATProduct' in 'Piston_Assembly'.
- To position 'Fixation.2' sub assembly create:
  - Coincidence between the published "Axis" of the bolt and the geometrical Axis of right hole of the 'Rod'



  - 1mm Offset between "Mating-Face" of the 'Bolt_M8' and the Corresponding face of the 'Rod.
- Save 'Piston_Assembly' using "Save Management."

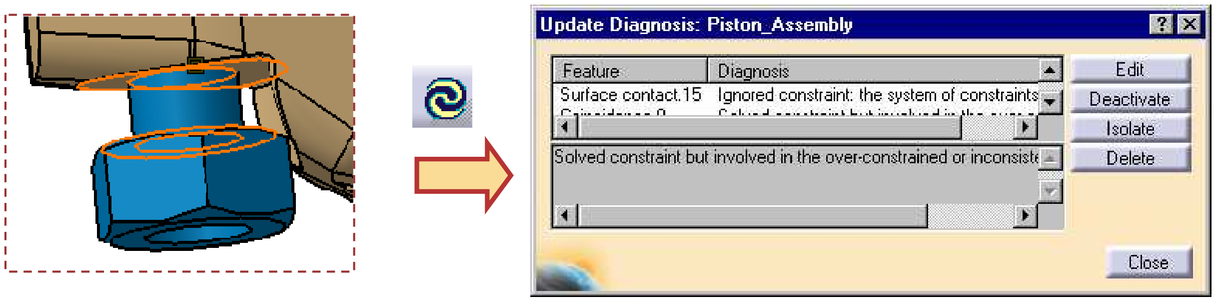


Student Notes:

Do It Yourself : Step 2 (3/4)

Positioning the nuts :

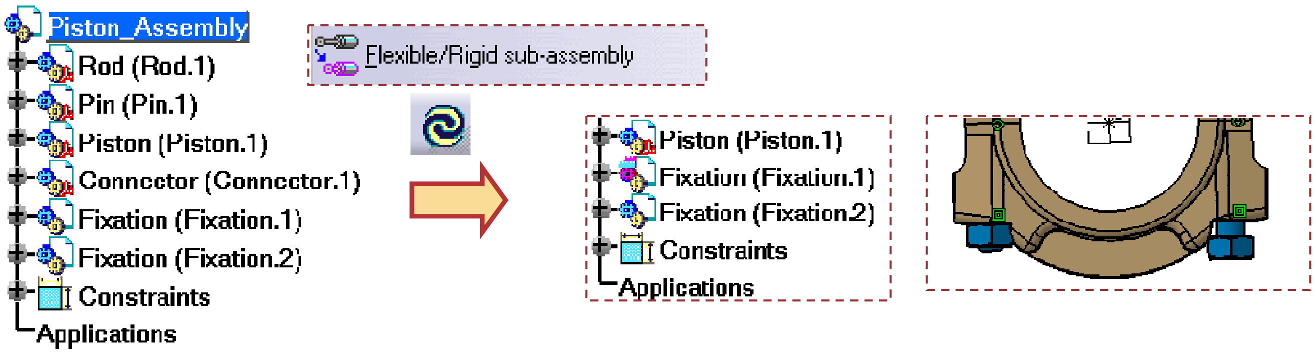
- **Try to position the 'Nut_M8.1' in 'Fixation.1' by creating:**
  - **Surface contact constraint between the mating face of the nut and corresponding face of 'Connector'.**



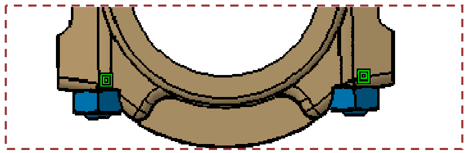
- **An Update error occurs. The system of constraints is not consistent as we have tried to define position of 'Fixation.1' by conflicting constraints (for positioning bolt as well as nut).**

Do It Yourself : Step 2 (4/4)

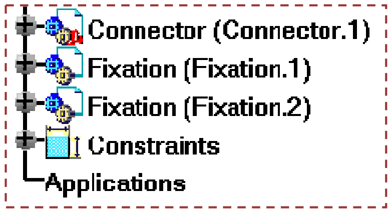
- **Make ‘Fixation.1’ assembly flexible, by using “Flexible/Rigid Sub-Assembly” command from the contextual menu.**
- **Update the constraints. Now the surface contact constraint is respected.**



- **Use the “Propagate position to reference” command in order to propagate this position to the other instance of the ‘Fixation’.**



- **Make ‘Fixation.1’ sub assembly rigid again.**



- **Save the assembly using Save Management Command.**

Student Notes:

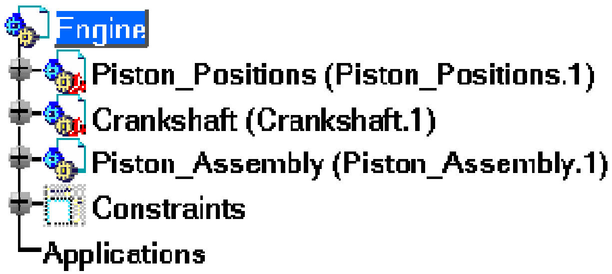
Student Notes:

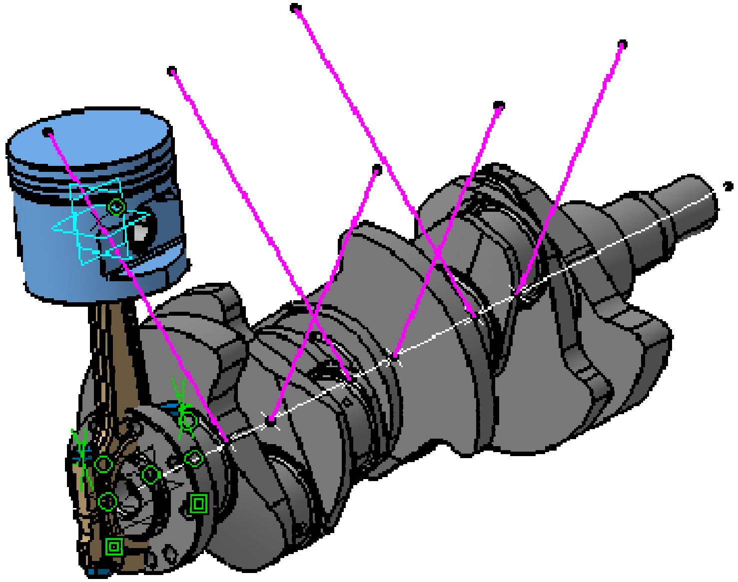
Do It Yourself : Step 3 (1/4)

Product used: Engine.CATProduct

In Step 3, you will complete Engine Assembly :

- **Load ‘Engine.CATProduct’.**
- **Insert ‘Piston_Assembly.CATProduct’ (created in Steps ‘1’ and ‘2’), in ‘Engine’ assembly.**





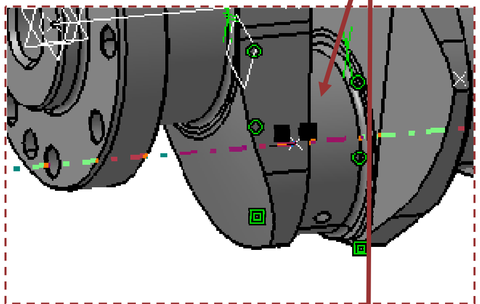
Engine Assembly

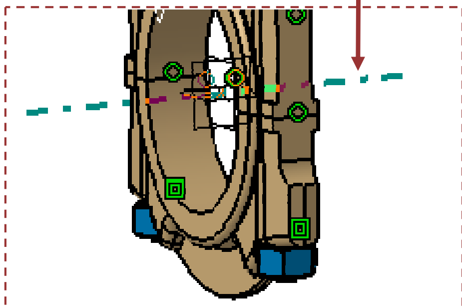
Student Notes:

Do It Yourself : Step 3 (2/4)

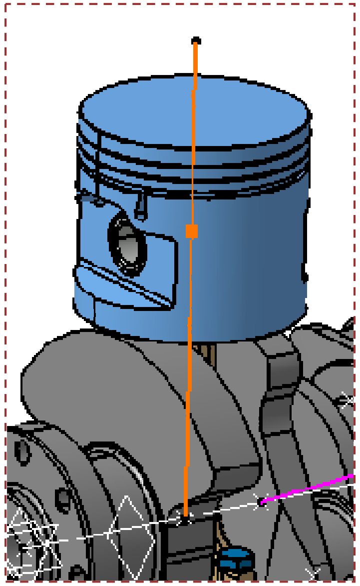
To position first instance of Piston create :

- **Coincidence constraint between the published "Axis_L1" of 'Crankshaft' and the"Crank_Axis" of the Rod.**

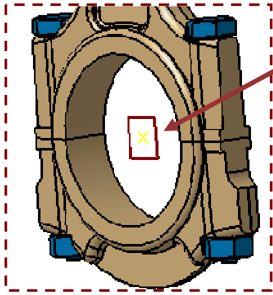


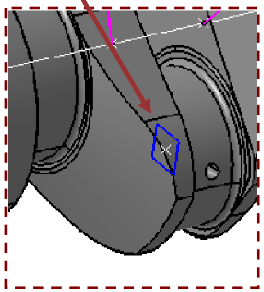


- **Coincidence between "Pin_grip_center" of the Rod and "Direction_L1" of 'Piston_Positions'.**



- **Coincidence between yz plane of the 'Rod' and "Plane L1" of 'Crankshaft'.**

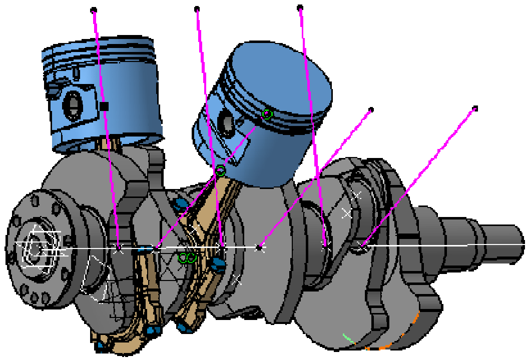


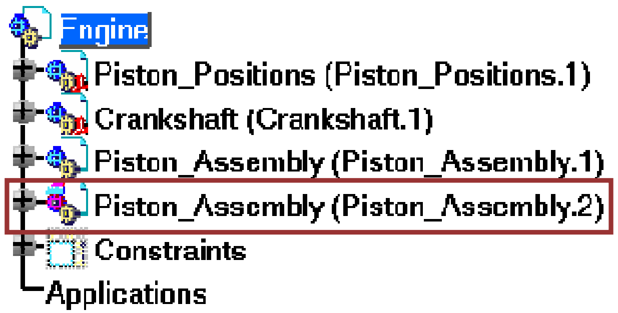


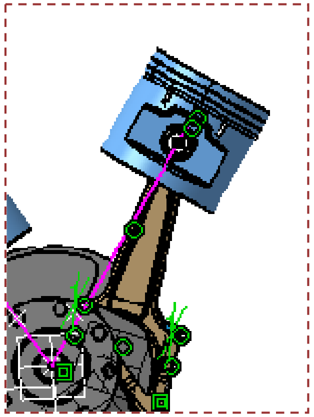
Student Notes:

Do It Yourself : Step 3 (3/4)

- Instantiate a second instance of Piston Assembly in Engine.CATProduct.
- To position 'Piston_Assembly.2' create:
  - Coincidence between the published "Axis_R1" of the 'Crankshaft' and the "Crank_Axis" of the 'Rod'.
  - Coincidence between "Pin_grip_center" of the 'Rod' and "Direction_R1" of "Piston_Positions".
  - Coincidence between yz plane of the 'Rod' and "Plane R1" of 'Crankshaft'.







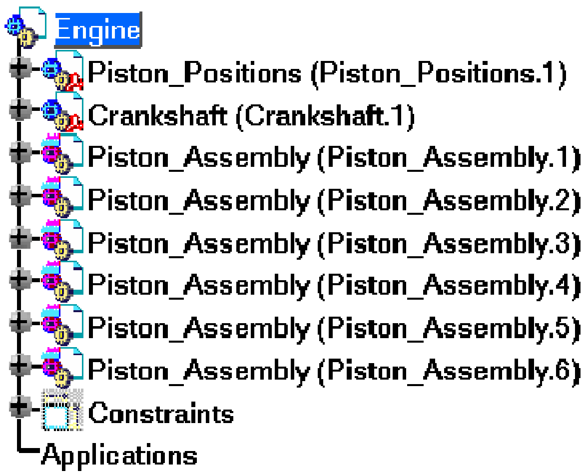
- To position second 'Piston':
  - Make 'Piston_Assembly.2' flexible.
  - Create a coincidence between the geometrical axis of 'Piston' and "Direction_R1" of 'Piston_Positions'.

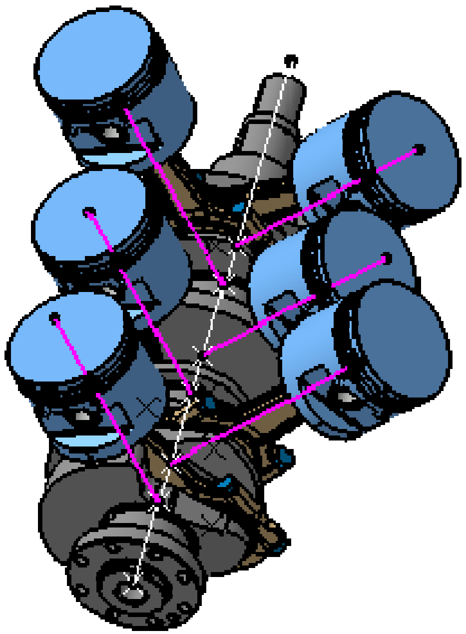
When Piston_Assembly.2 is made flexible, the constraints related to "Piston_Assembly.2" are belonging to its Rod and its Piston and NOT "Piston_Assembly.2".

Student Notes:

Do It Yourself : Step 3 (4/4)

- **Make the first Instance of ‘Piston_Assembly’ “flexible” and position its piston by creating axis coincident constraints.**
- **Similarly instantiate another four instances of Piston Assembly and position them along directions L2, R2,L3 and R3.**





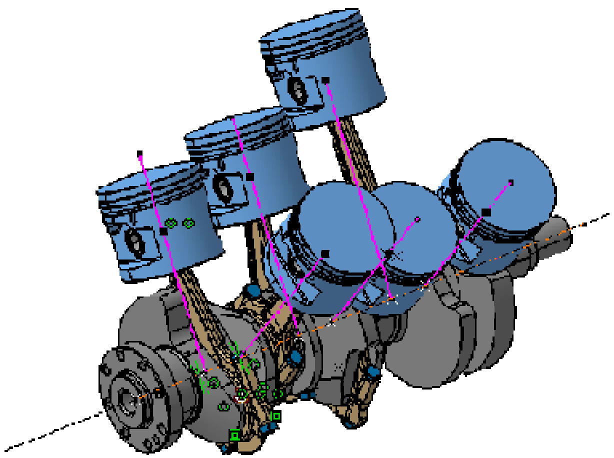
Student Notes:

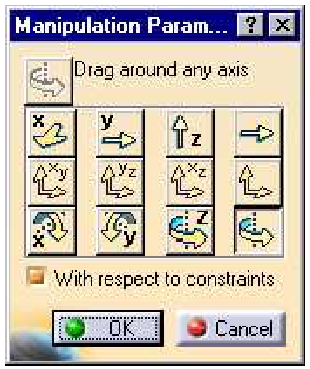
Do It Yourself : Step 4

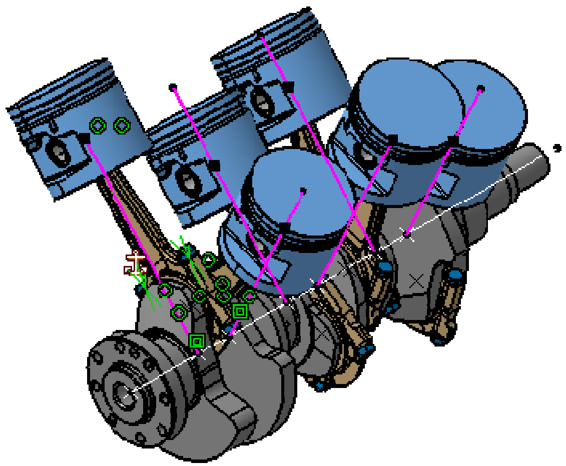
Product used: Engine_Step4.CATProduct

In this step, you will see how to manipulate the positions of ‘Crankshaft’ and see that ‘Piston’ positions are updated automatically.

- **Deactivate the Fix Constraint for ‘Rod’ in each ‘Piston_Assembly’.**
- **Launch the manipulation command and select “Around any Axis” and “With respect to constraints” options.**
- **Select the “Rotation_Axis” from skeleton as axis of rotation.**
- **Drag and rotate Crankshaft by some amount in clockwise or anti-clockwise direction.**
- **You will see that all piston positions are automatically updated with respect to the new positions of ‘Crankshaft’.**







Student Notes:

To Sum Up …

In this lesson you have seen what are flexible sub assemblies. You have also seen that :

- A flexible sub assembly is indicated by the purple colored wheel at the left corner of the icon.
- Rigid sub-assemblies are always synchronous with the original product, whatever mechanical modification you perform, but flexible sub-assemblies can be moved individually, without considering the position in the original product
- A flexible sub assembly of a product is never displayed in its mechanical structure
- You can edit the constraints defined for flexible sub-assemblies. The changes made to these constraints do not affect the constraints defined for the original product contained in the reference document.

Student Notes:

Working with Large Assemblies

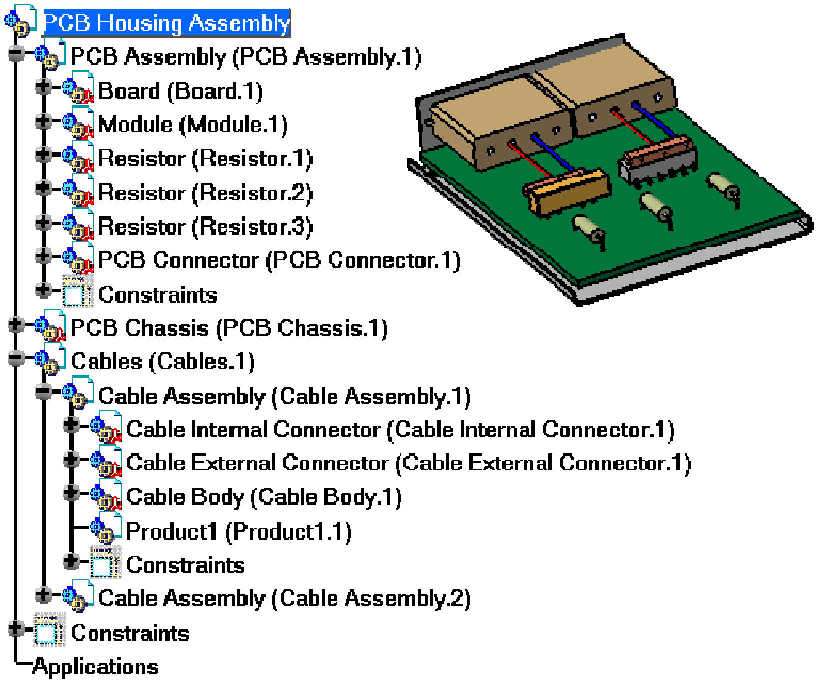
You will learn how to optimize the display performances of large assemblies.

- Introduction to Working with Large Assemblies
- Hiding Components
- Deactivating Representations
- Deactivating a Component
- Selective Load
- Using Visualization Mode
- Summary of Modes
- Recap Exercise : Washing Machine
- To Sum Up

Student Notes:

Introduction to Working with Large Assemblies

In case of complex industrial assemblies, the root assembly contains large number of components, with large number of instantiations of components, thus increasing the size of the final assembly. This decreases the performance of CATIA and it takes longer time open, zoom, pan, update and save large assemblies. It also takes more time to generate and update drafting views.



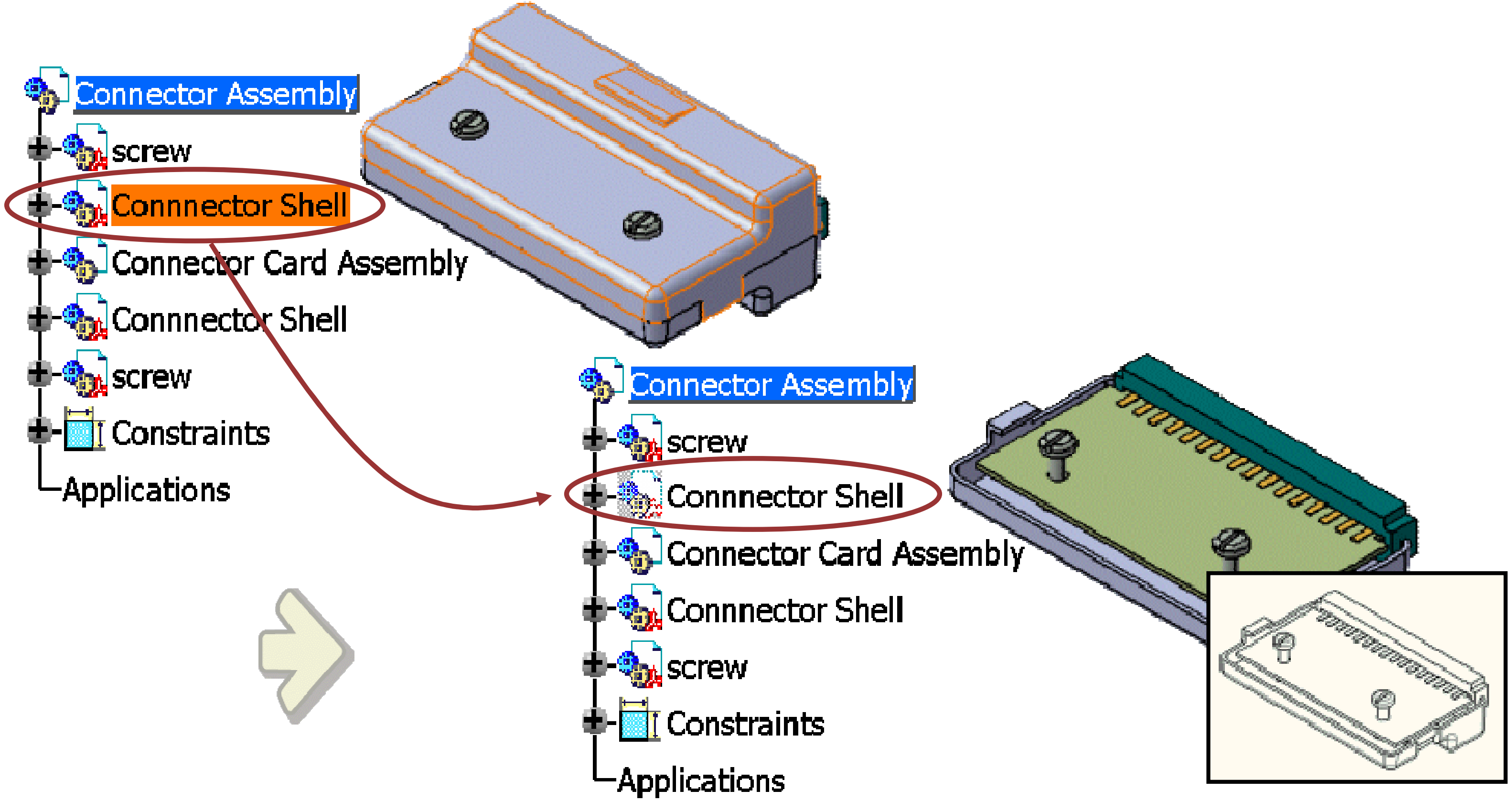
In this lesson you will see how improve visualization and CATIA performance while working with large assemblies by:

- **Hiding components which are not being edited**
- **Deactivating Representations**
- **Deactivating components**
- **Selectively loading only necessary components**
- **Using Visualization Mode**

Student Notes:

Hiding Components

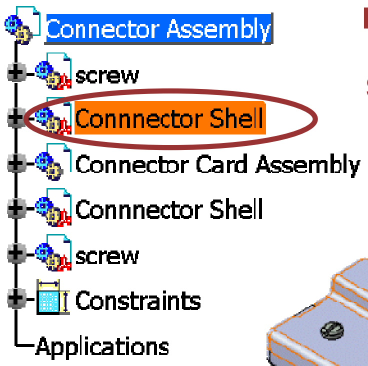
You will see how to hide components to improve performance and reduce clutter in "show space" and exclude components from drawing views.



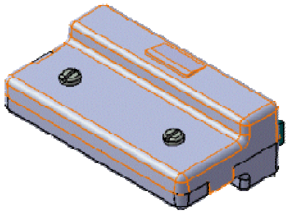
Student Notes:

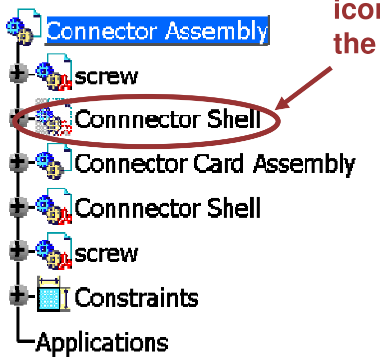
What Is Hiding Components ?

Hiding components can improve performance and reduce clutter in “show space”.
Hiding also excludes components from drawings views.

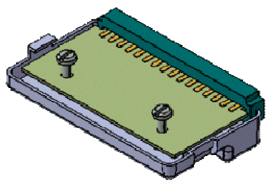


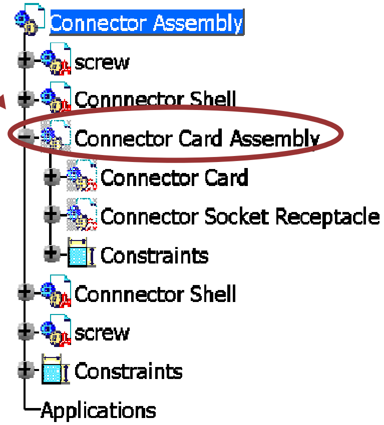
Here one instance of the Connector Shell is hidden or no-shown.





The component’s icon is dimmed in the tree.

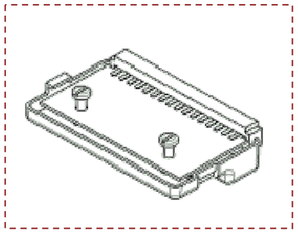




Here an assembly is hidden.

Hide/Show state is <u>stored</u> in CATProduct files so that the state is maintained when the assembly is opened.

Hidden components are not visible in “show space” or in drawing views.



Hiding components is similar to deactivation of representations, but with the added advantages of :

- **excluding components from drawing views**
- **part elements accessible to design parts and assemblies**

Student Notes:

Differences Between Show and Hide

This table compares the capabilities of Show and Hide while in Design Mode.

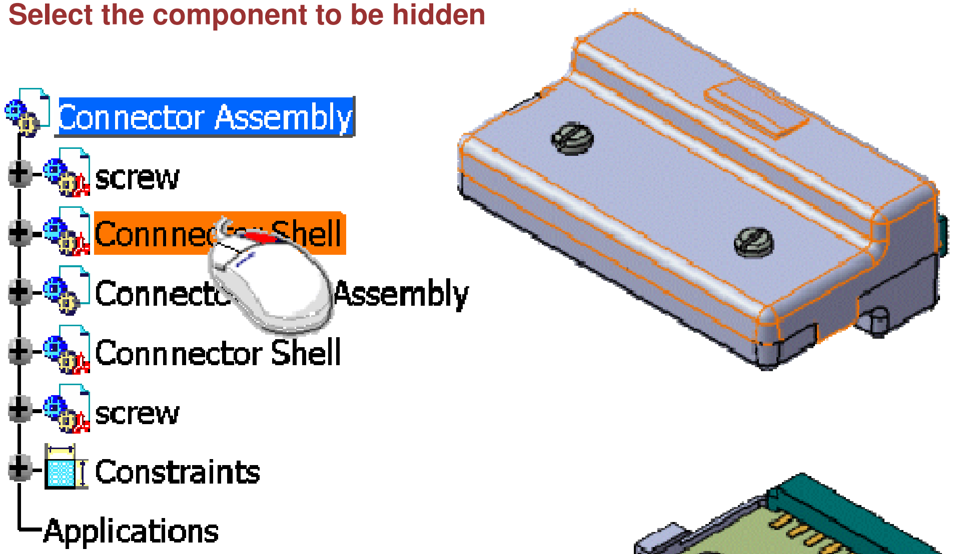
Comparison of Show & Hide (in Design Mode)		
Behavior	**Shown**	**Hidden**
Memory and Performance		
Loaded in Memory	Fully Loaded	Fully Loaded
Load and Update Performance	Normal	Normal
Display Performance	Normal	Faster, which is a benefit over being Shown
Visibility		
Visible in Show	Yes	No
Visible in No-show	Yes	Yes
Viewable in non-shaded mode	Yes	Yes
Viewable in DMU and sketcher sections	Yes	Yes
Visible in drafting	Yes	No, which is a benefit over being Deactivated
Assembly Constraints and Transformations		
Accessible for adding Assembly constraints	Yes	Yes
Assembly Constraints re-generated/updated	Yes	Yes
Accessible to define translations & rotations	Yes	Yes
Analysis		
Calculated in Clash, Clearance, Contact	Yes	No
Calculated in Mass Property analysis	Yes	Yes
Accessible for Measurements	Yes	Yes
Part Geometry		
Geometry features accessible in tree	Yes	Yes
Geometry may be edited	Yes	Yes
Geometry may be used to define sketches and features in other parts in the assembly (e.g. up-to-plane)	Yes	Yes
In-context features re-generated/updated (e.g. associativity)	Yes	Yes

Student Notes:

Hiding Components

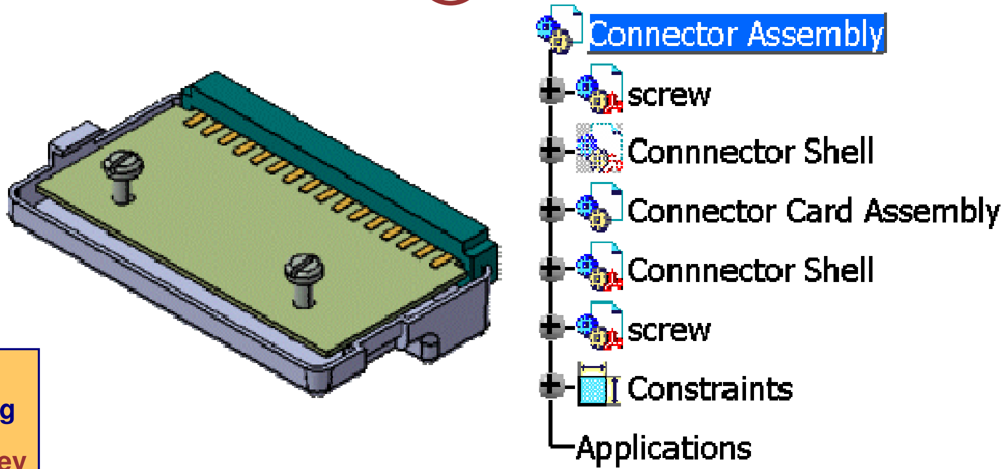
Hiding can be performed on individual components, multi-selected components, or an entire assembly.

1 Select the component to be hidden



2 Hide the component

3 The component is hidden



You can hide more than one component at a time by selecting with the mouse while holding the [CTRL] key

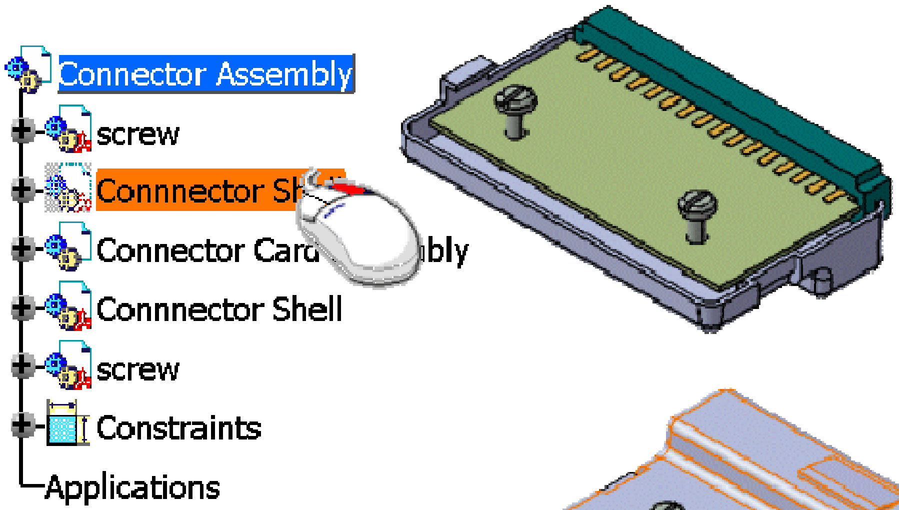
[CTRL] key

Student Notes:

Showing Components

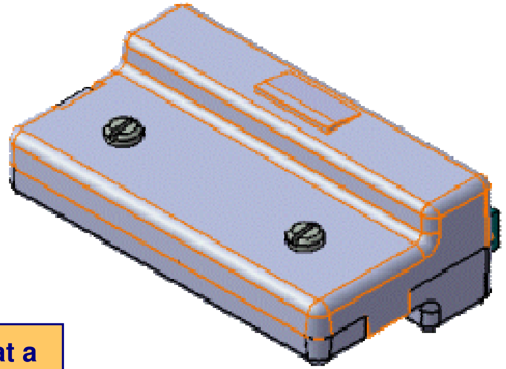
Showing a component makes it available for designing the assembly and inclusion in drawing views.

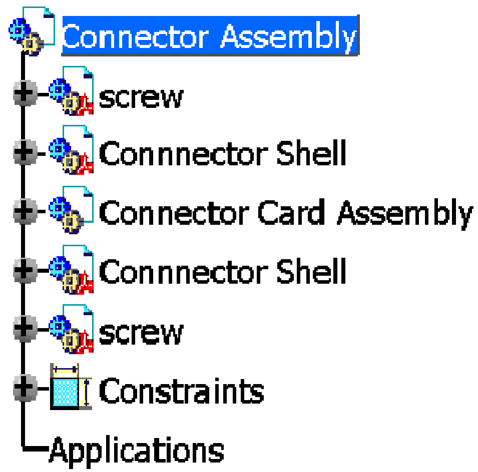
1 Select the component to be shown



2 Show the component

3 The component is shown





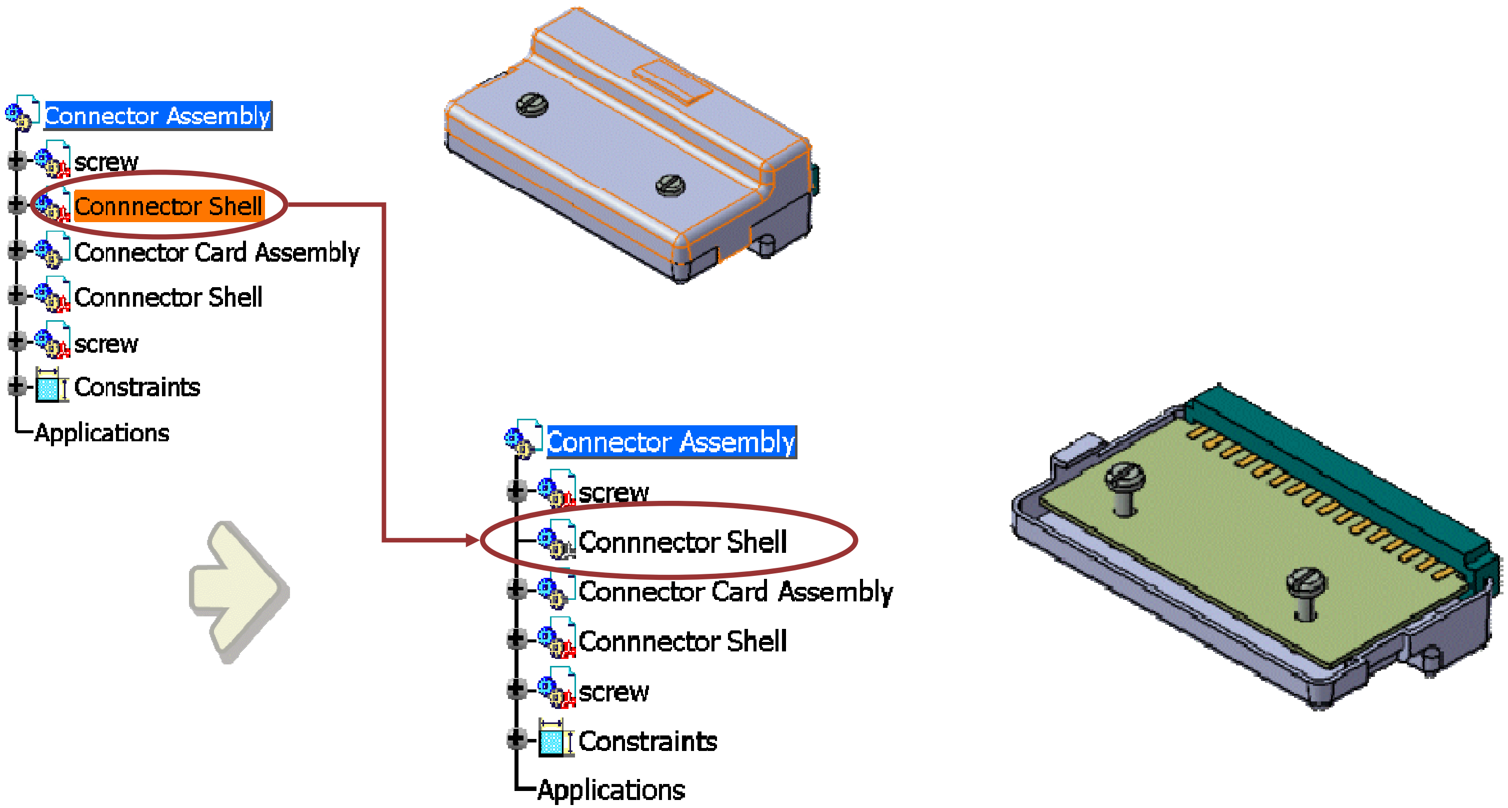
You can show more than one component at a time by selecting with the mouse while holding the [CTRL] key

[CTRL] key

Student Notes:

Deactivating Representations

You will see how to deactivate representations to improve performance, reduce clutter in "show space" and "no-show space", and exclude representations from mass property analysis.



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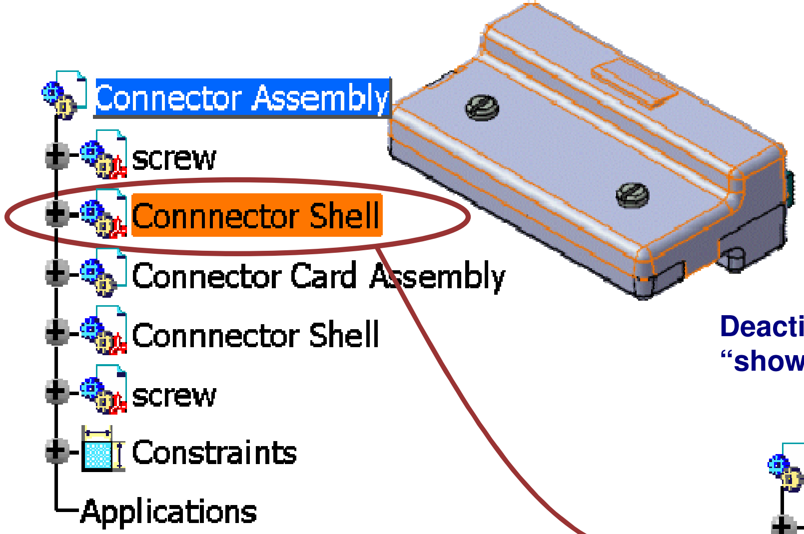
Student Notes:

What Is Deactivating Representations ?

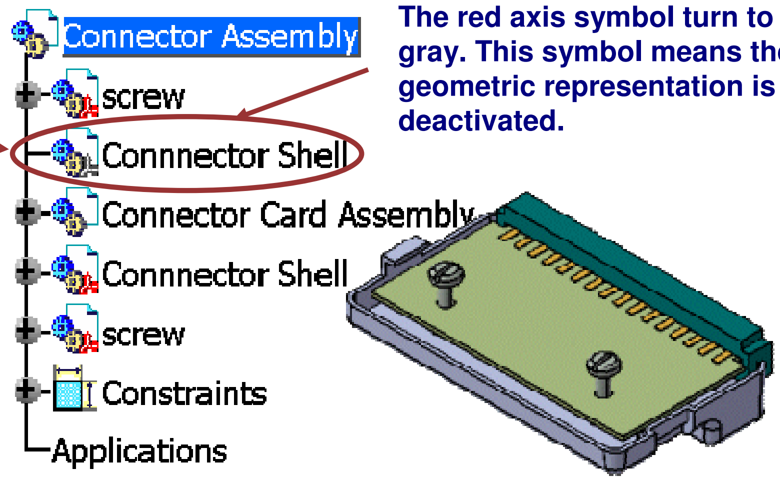
Deactivating representations can improve performance and reduce clutter in no-show space. Deactivation can also be used to exclude representations from mass property analysis.



Deactivated representations are excluded from mass property analysis.

Deactivation state can be stored in CATProduct files. The <u>default</u> geometric representation is activated when opening an assembly. If there is only one representation, it is the default.

Deactivated representations are not visible in “show space” or “no-show space”.



The red axis symbol turn to gray. This symbol means the geometric representation is deactivated.

Deactivation of representations is similar to hiding components, but with the added advantages of :

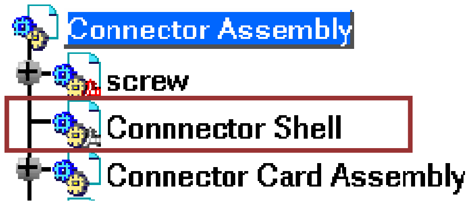
- **improving performance when opening assemblies**
- **excluding representations from mass property analysis**

Student Notes:

Why Deactivate Representations ? (1/2)

Deactivating representations provides following benefits :

- **Mask Active representations in the specification tree and in the geometry : By this means, you choose to visualize the geometric representation of CATIA elements, belonging to a CATProduct. With the Deactivate Node functionality, only the selected element is hidden. Whereas with the Deactivate Terminal Node functionality, the last node's elements of the selected node are masked.**



- **Improve Performance : Deactivating representations will not load these components in memory. This improves the performance of CATIA. CATIA takes less time to open, pan, zoom and save large assembly documents.**

- **Hide Representations from No Show Space : By deactivating representations, these components are not represented even in No Show space. Hiding representations will move the representations in No Show space, causing cluttering of No Show space. Hence deactivating representations is useful than hiding representations.**

Student Notes:

Why Deactivate Representations ? (2/2)

Deactivating representations provides following benefits :

- **You can activate or deactivate Shape representation in Tools -> Options -> Infrastructure, select the Product Visualization tab and check the box entitled Do not activate default shapes on open. The entity representation disappears, it is a profit for memory space. You can work only on the tree.**

- **Analysis of Assemblies : Deactivated representations are excluded from mass property analysis. At times you are interested to evaluate mass property of partial assemblies. You can deactivate representations which should not be considered for mass property analysis.**

Student Notes:

Differences Between Activation and Deactivation

This table compares the capabilities of Activation and Deactivation while in Design Mode.

Comparison of Activation & Deactivation Mode (in Design Mode)		
Behavior	**Activated**	**Deactivated**
Memory and Performance		
Loaded in Memory	Fully Loaded	Fully Loaded
Load and Update Performance	Normal	Normal
Display Performance	Normal	Faster, which is a benefit over being Activated
Visibility		
Visible in Show	Yes	No
Visible in No-show	Yes	No, which is a benefit over being Hidden
Viewable in non-shaded mode	Yes	No
Viewable in DMU and sketcher sections	Yes	No
Visible in drafting	Yes	Yes, even though not visible in the assembly
Assembly Constraints and Transformations		
Accessible for adding Assembly constraints	Yes	No
Assembly Constraints re-generated/updated	Yes	Yes
Accessible to define translations & rotations	Yes	No
Analysis		
Calculated in Clash, Clearance, Contact	Yes	No
Calculated in Mass Property analysis	Yes	No, which is a benefit over being Hidden
Accessible for Measurements	Yes	No
Part Geometry		
Geometry features accessible in tree	Yes	No
Geometry may be edited	Yes	No
Geometry may be used to define sketches and features in other parts in the assembly (e.g. up-to-plane)	Yes	No
In-context features re-generated/updated (e.g. associativity)	Yes	Yes, after activating and updating the associated part

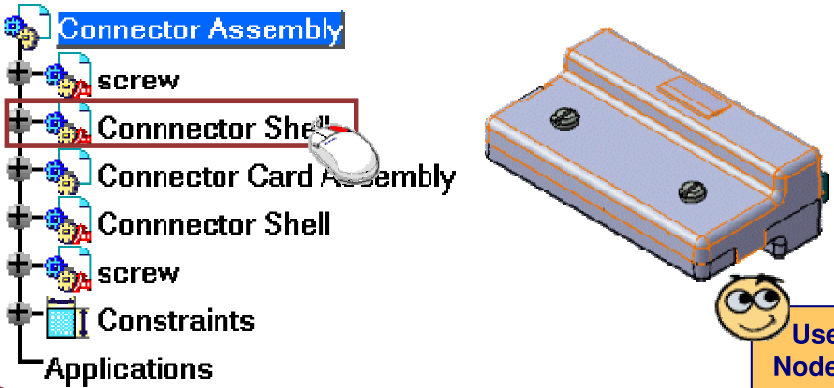
Student Notes:

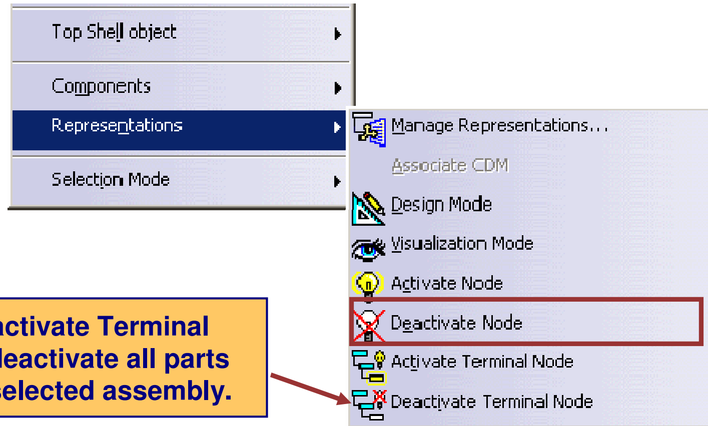
Deactivating Representations

Deactivation can be performed on individual components, multi-selected components, or all components in an assembly.

1 Right-click the component to be deactivated

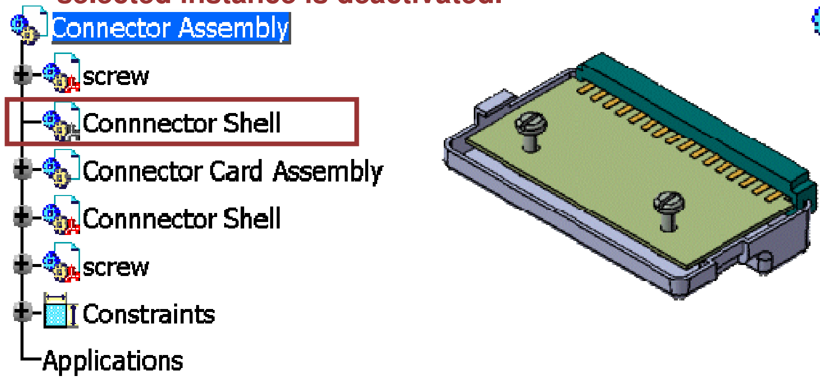
2 With the contextual menu select "Deactivate Node"

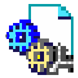




Use Deactivate Terminal Node to deactivate all parts within a selected assembly.

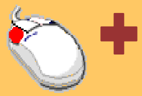
3 The geometric representation of the component is deactivated. Note that only the selected instance is deactivated.





Deactivated component is represented by this icon in the tree.

You can deactivate more than one component at a time by selecting with the mouse while holding the [CTRL] key.



[CTRL] key

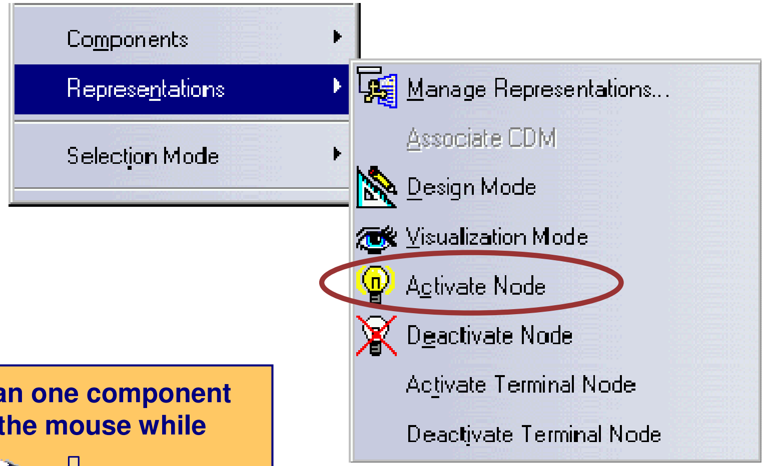
Student Notes:

Activating Representations

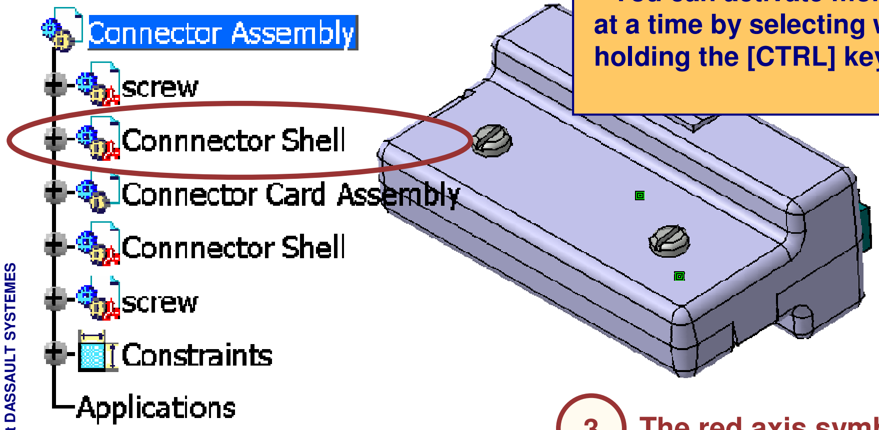
Activating a representation makes it available for designing the assembly.

1 **Right-click the component to be activated**

2 **Select Representations + Activate Node**



You can activate more than one component at a time by selecting with the mouse while holding the [CTRL] key + **[CTRL] key**



3 **The red axis symbol turn to red.**

Use Activate Terminal Node to activate all parts within a selected assembly.

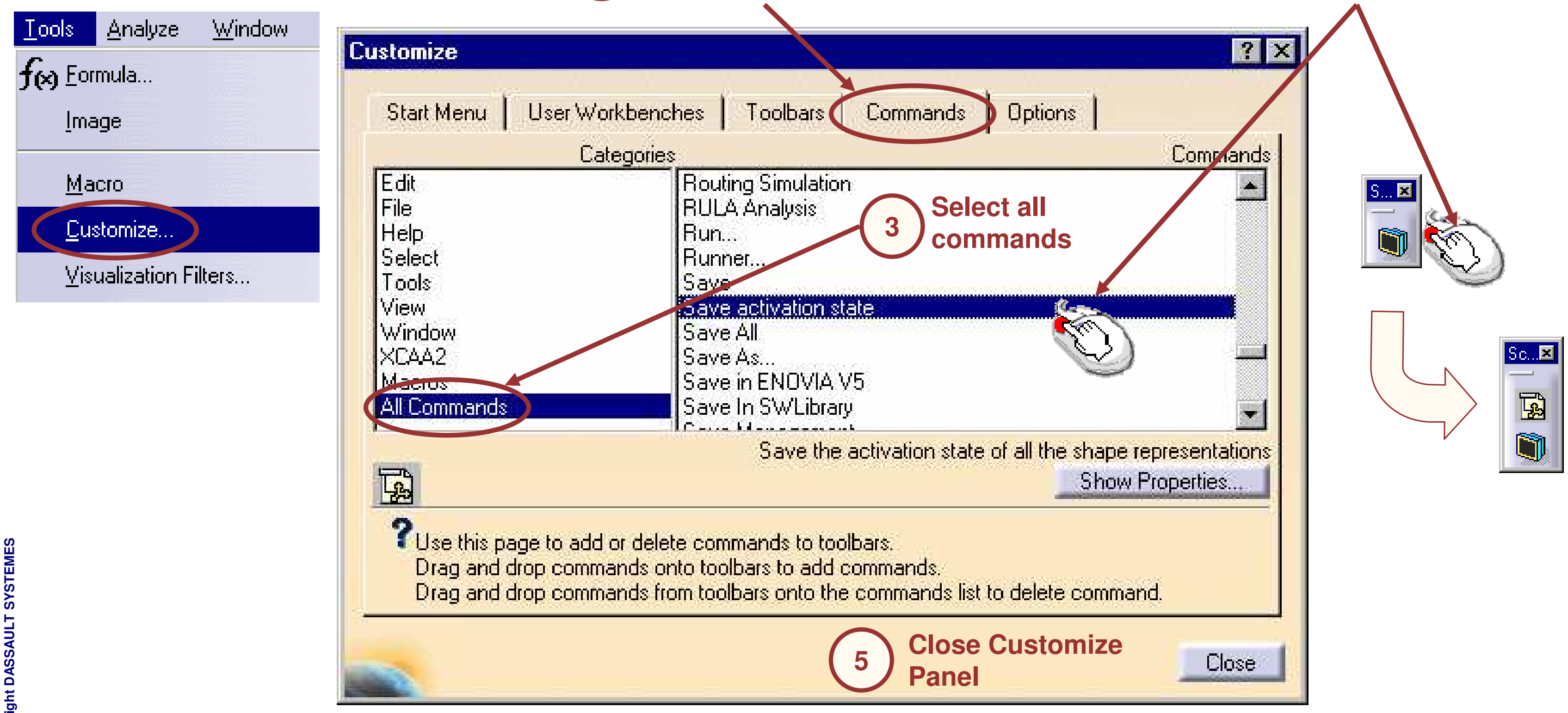
Saving Activation State (1/2)

If you want to save your assembly with representation of some components deactivated you have to store their status in the CATProduct file. A command allows you to store activation state but you need to create the access to this command.

1 **Select "customize …" command from Tools menu**

2 **Select Commands tab**

4 **Drag and drop "Save Activation State" command into a toolbar**

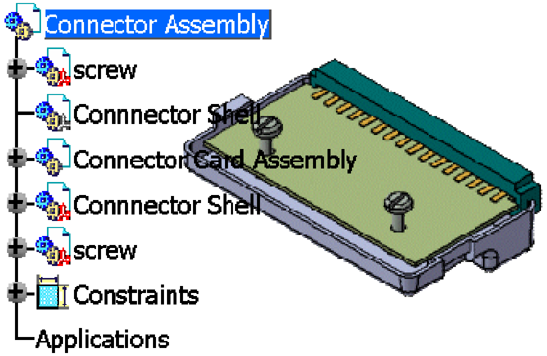


Student Notes:

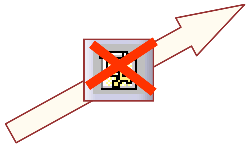
Saving Activation State (2/2)

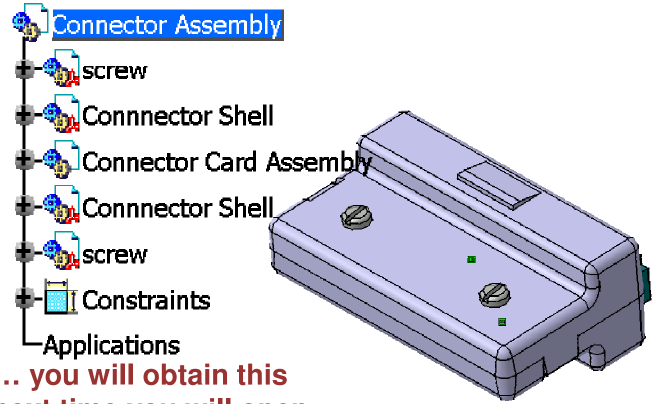
This command will allow to keep activation state of components into the CATProduct files.

This assembly has one component with deactivated shape

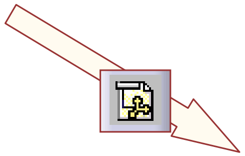


If you save it without having clicked on the icon …

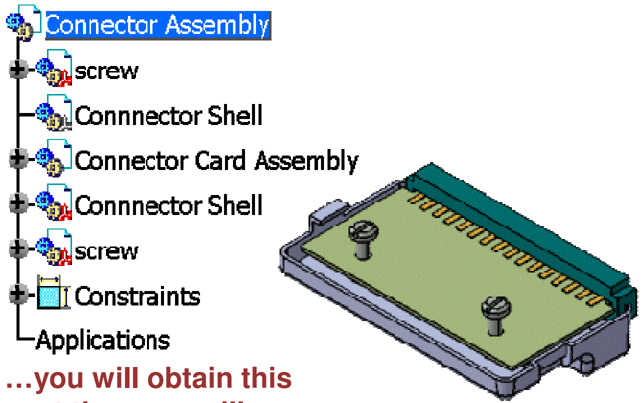




… you will obtain this next time you will open the CATProduct.



If you click this icon and save your CATProduct …



…you will obtain this next time you will open the CATProduct.

One click on the icon will make all save operations of CATProducts in the session, keep activation states of components.

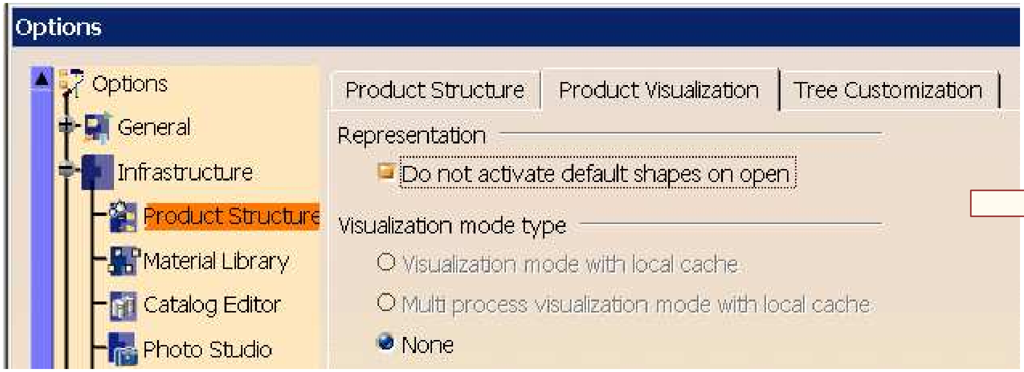
Student Notes:

Student Notes:

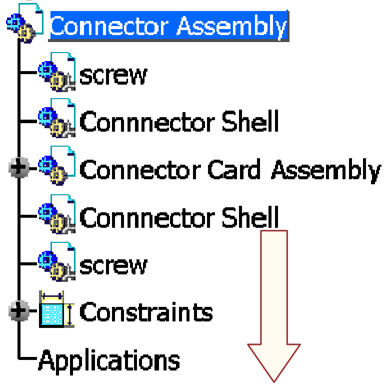
Using Deactivation when Opening an Assembly

You can improve performance by deactivating automatically representations when you open assemblies.

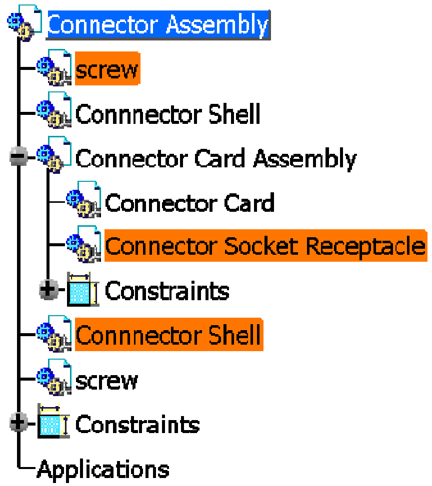
1 **Turn ON the option Do not activate default shapes on open**



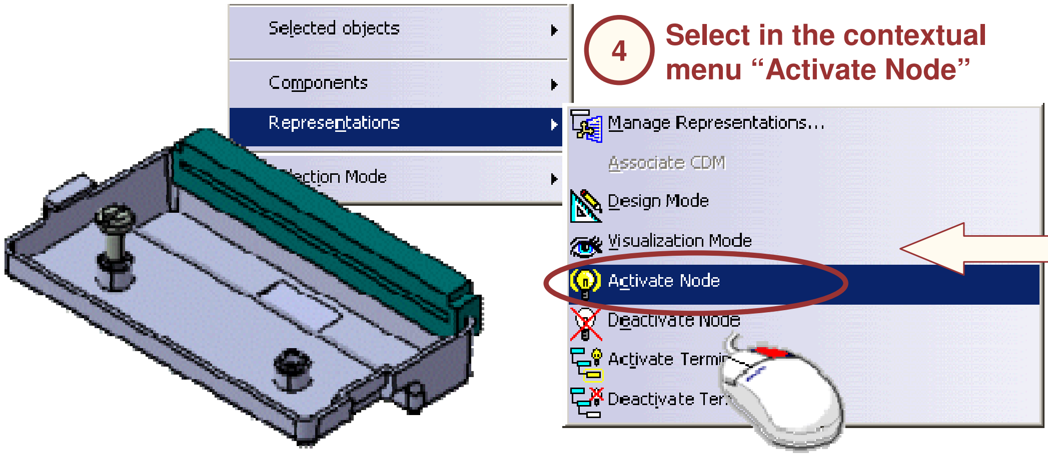
2 **Open an assembly**



3 **Multi-select the components**



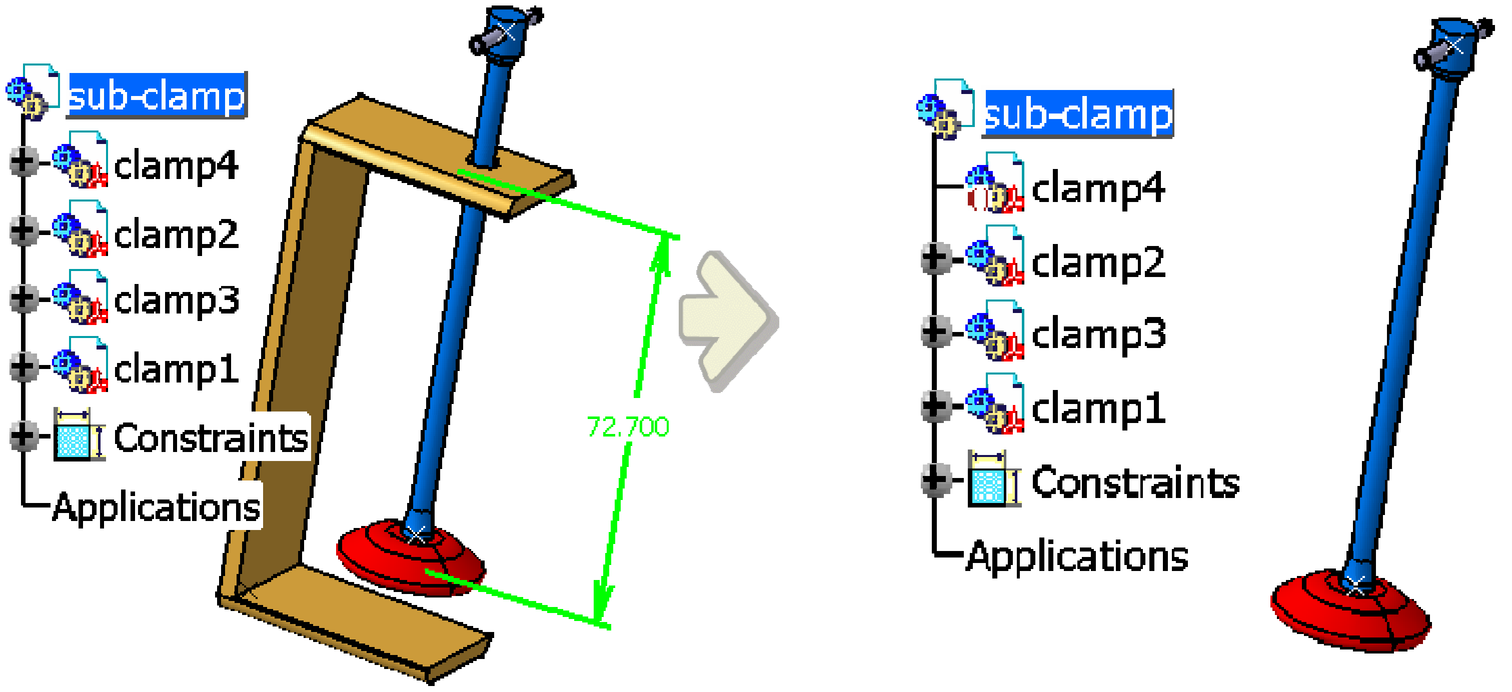
4 **Select in the contextual menu "Activate Node"**



Student Notes:

Deactivating a Component

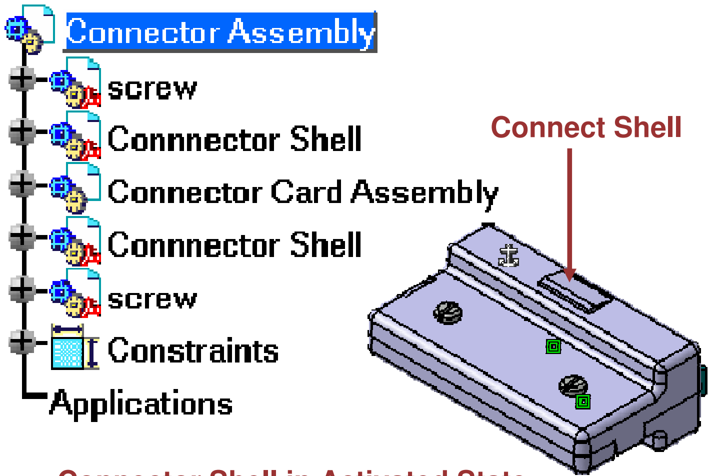
You will see how to deactivate components to improve performance, reduce clutter in “show space” and “no-show space”, and exclude it from Bill Of Material.



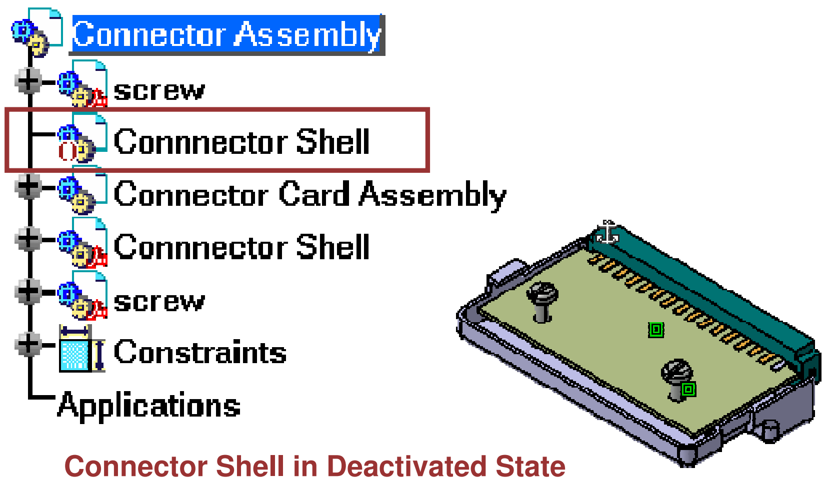
What Is Deactivating Components?

Deactivating a Component is removing its representation and instance. The operation is simultaneous in all the CATIA documents containing this element . This operation is shared by all the instances of this part. You can apply this functionality on CATProducts, CATParts and models.

In this example, component Connector Shell is shown in activated and deactivated state.

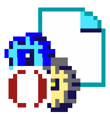


Connector Shell in Activated State



Connector Shell in Deactivated State

Deactivation state of a component is represented in specification tree by the symbol shown below :



Deactivated rcomponents are not visible in "show space" or "no-show space".

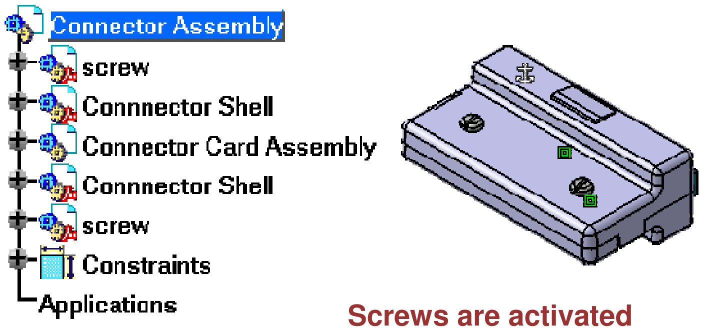
Student Notes:

Student Notes:

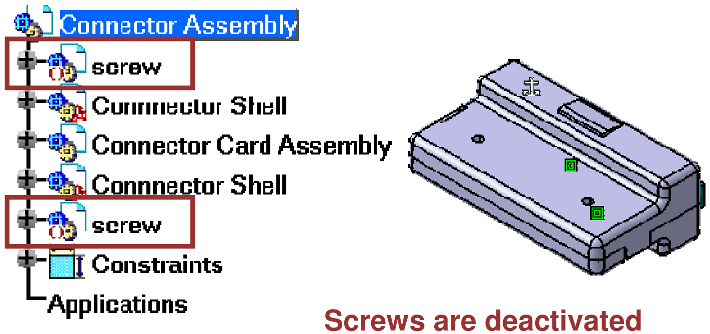
Why Deactivate Components ?

Deactivating components provides following benefits :

- **Hide Components from No Show Space : By deactivating components, these components are not represented even in No Show space, hence,causing less cluttering of No Show space.**

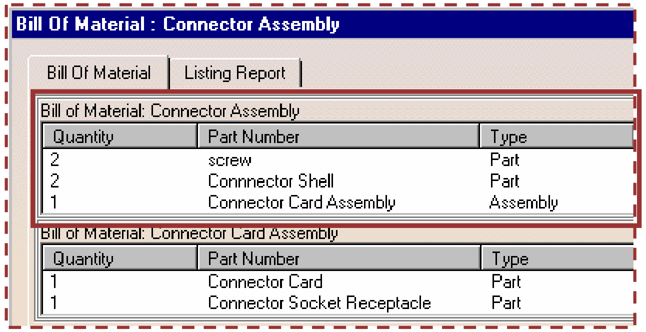


Screws are activated



Screws are deactivated

- **Analyze Bill of Material : Deactivating a component will not list this component in Bill of Material. Hence, it is useful to generate Bill of Material for various configurations of Assembly.**



Bill Of Material : Connector Assembly

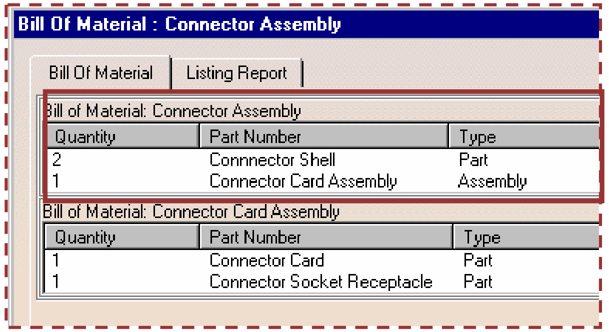
Bill Of Material | Listing Report

Bill of Material: Connector Assembly

Quantity	Part Number	Type
2	screw	Part
2	Connnector Shell	Part
1	Connector Card Assembly	Assembly

Bill of Material: Connector Card Assembly

Quantity	Part Number	Type
1	Connector Card	Part
1	Connector Socket Receptacle	Part



Bill Of Material : Connector Assembly

Bill Of Material | Listing Report

Bill of Material: Connector Assembly

Quantity	Part Number	Type
2	Connnector Shell	Part
1	Connector Card Assembly	Assembly

Bill of Material: Connector Card Assembly

Quantity	Part Number	Type
1	Connector Card	Part
1	Connector Socket Receptacle	Part

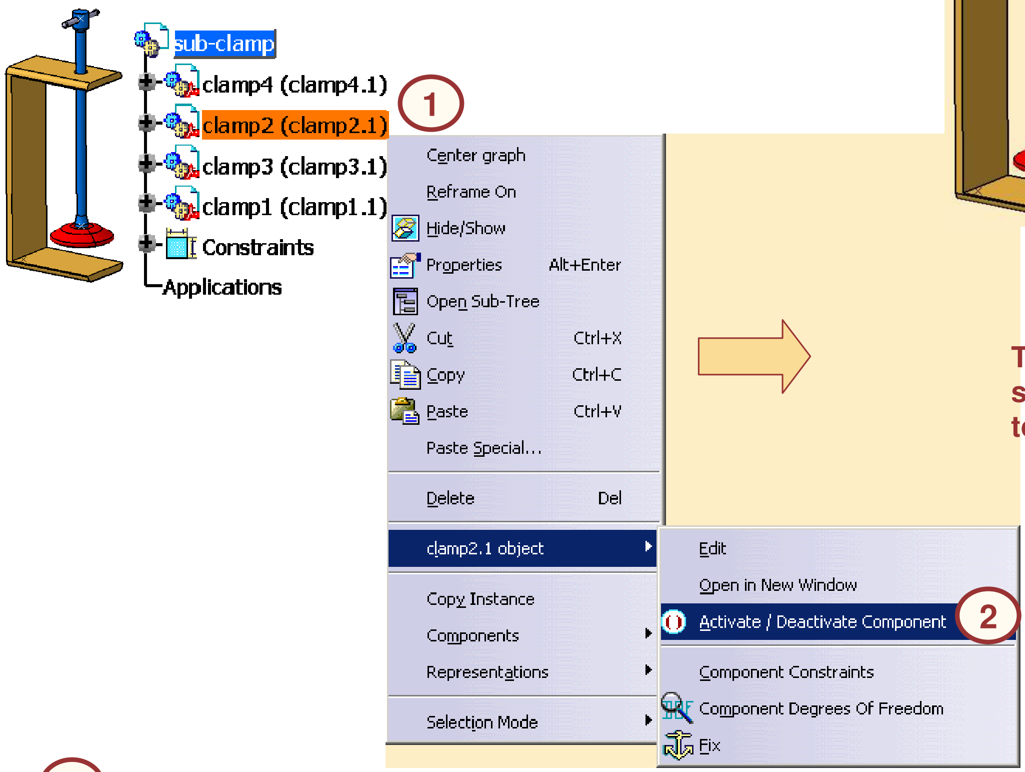
Bill Of Material with Screw Component in Activated State

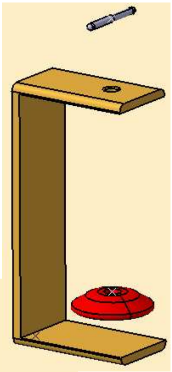
Bill Of Material with Screw Component in Deactivated State

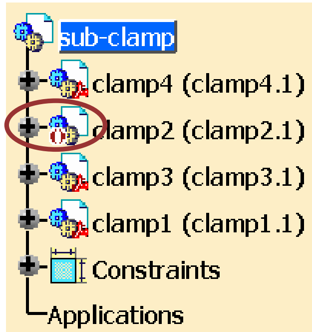
Student Notes:

Deactivating a Component (1/2)

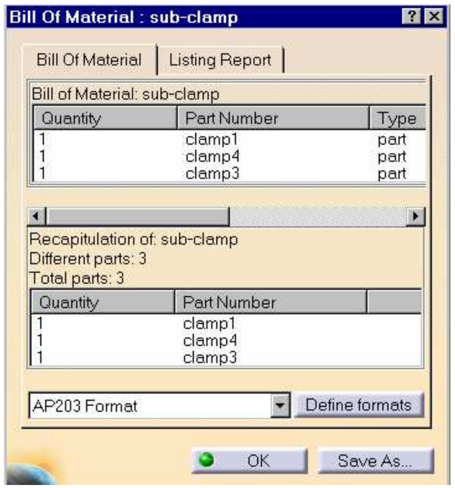
1 Select the instance and activate its contextual menu







The symbol in the specification Tree shows you that it is still possible for you to reactivate it by the reverse operation



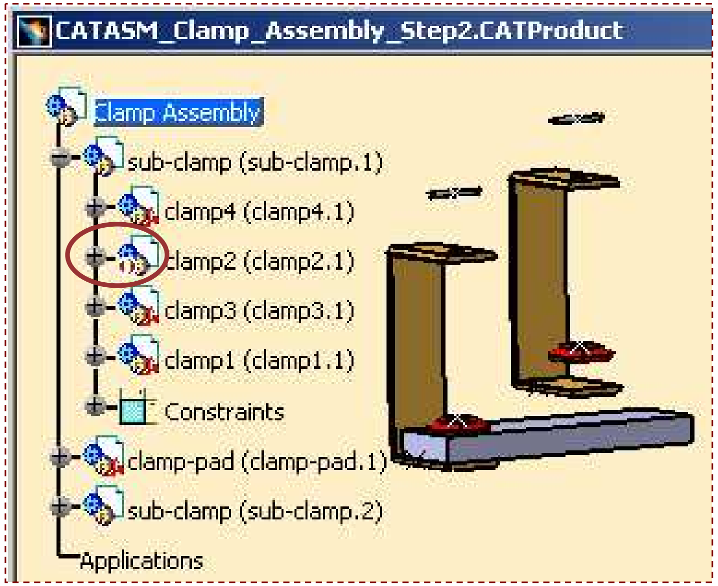
2 Select the Activate/Deactivate Component

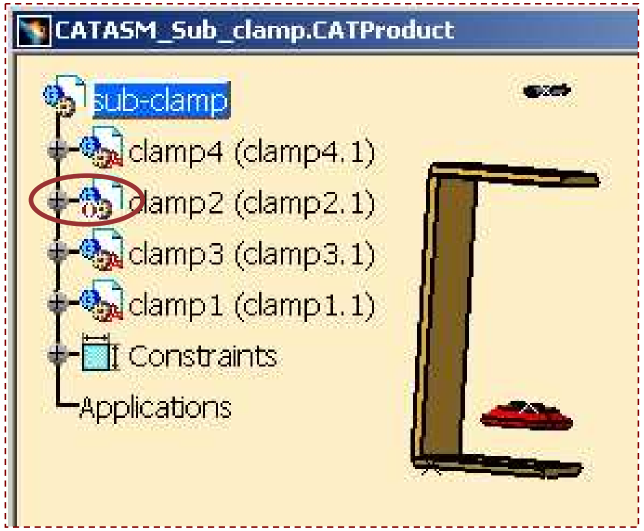
Its shape is deactivated and there are no traces of its specifications in the Bill Of Material

Student Notes:

Deactivating a Component (2/2)

As opposed to deactivate a node, deactivating a component inside a assembly means deleting its representation in all the CATIA documents in which it is being referred.



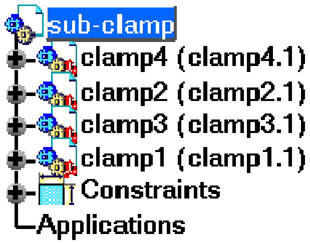


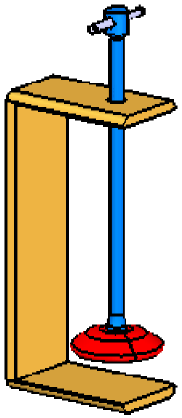
Student Notes:

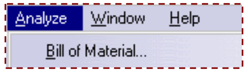
Effect of Deactivation on Bill Of Material (1/2)

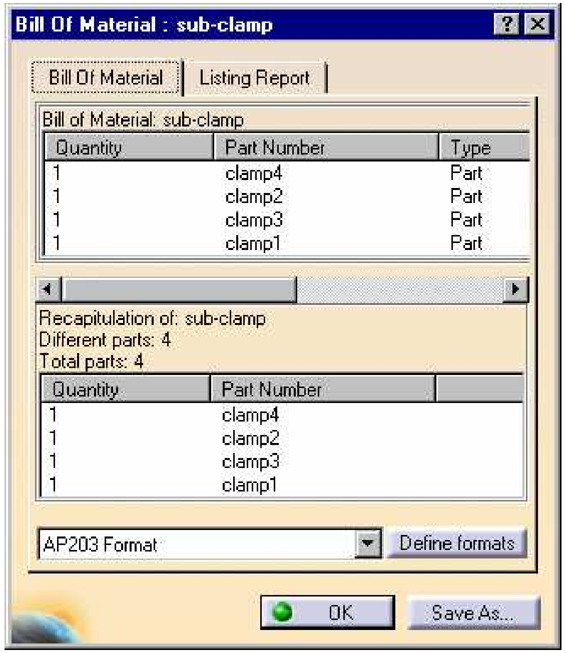
The deactivated and unloaded component is not displayed in Bill of Material

1 Access the Bill of Material of the entire assembly, when all its components are loaded and activated.







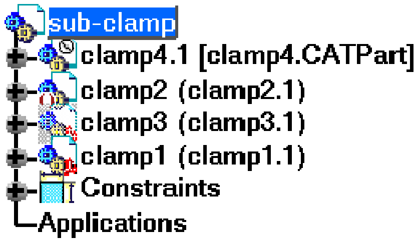


Bill Of Material of the 'Sub-clamp' assembly

2 Unload 'clamp4.1'

3 Deactivate 'clamp2.1'

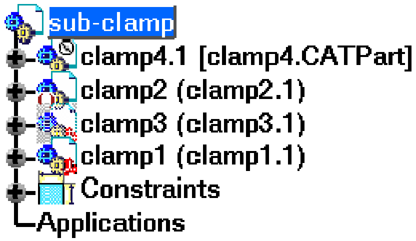
4 Hide 'clamp1.1'

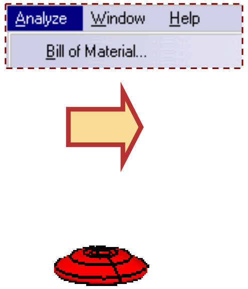


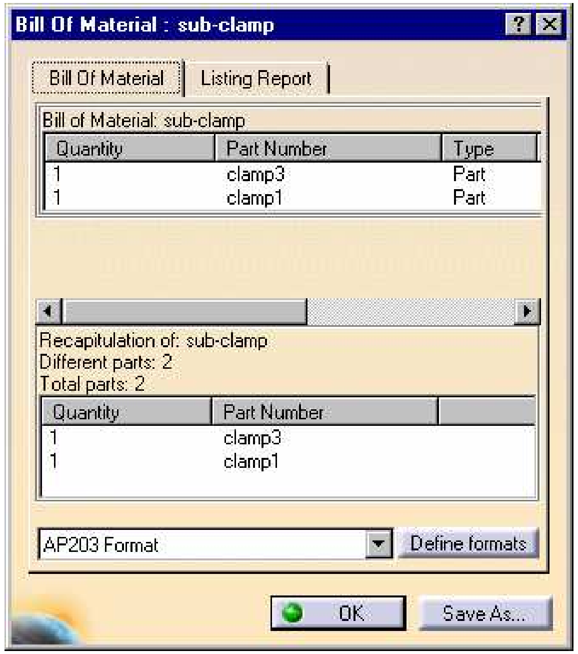
Student Notes:

Effect of Deactivation on Bill Of Material (2/2)

5 **Access the Bill of Material of the entire assembly.**





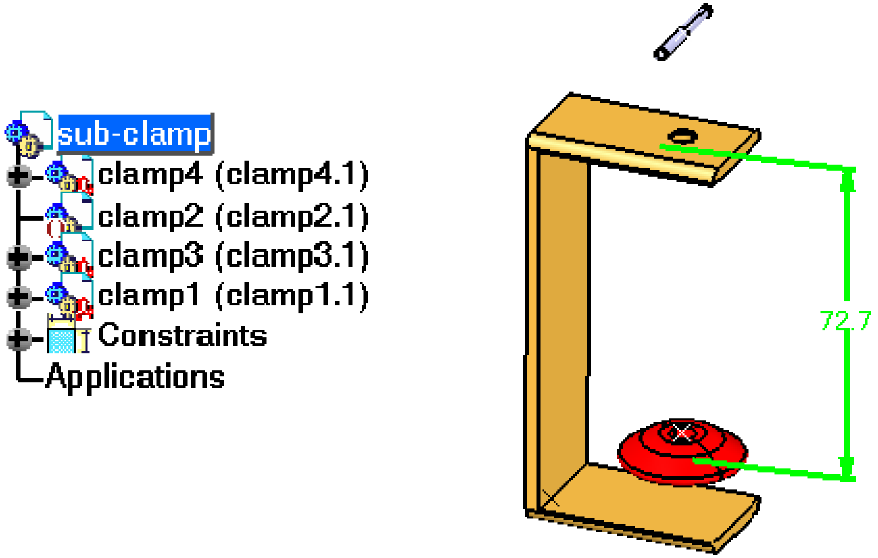


Bill Of Material of the 'Sub-clamp' assembly

Student Notes:

Do It Yourself

Product used : ‘SUB_CLAMP.CATProduct”

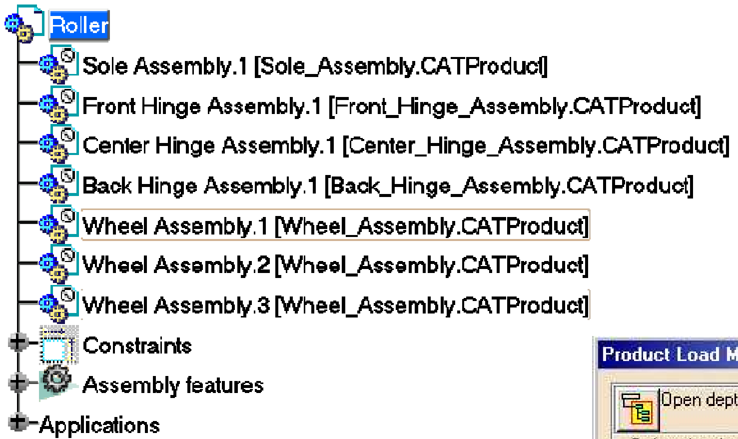


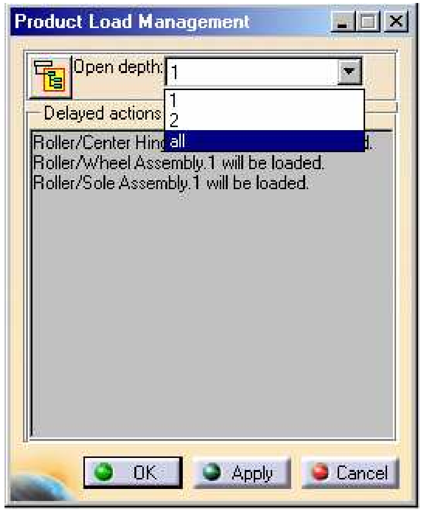
- Deactivate a Component as shown
- Analyze ‘Bill of Material’

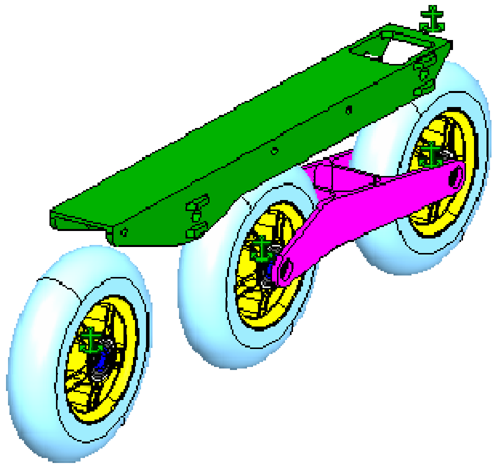
Student Notes:

Selective Load

You will see how to specify the depth of an opened Product Structure

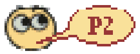






Student Notes:

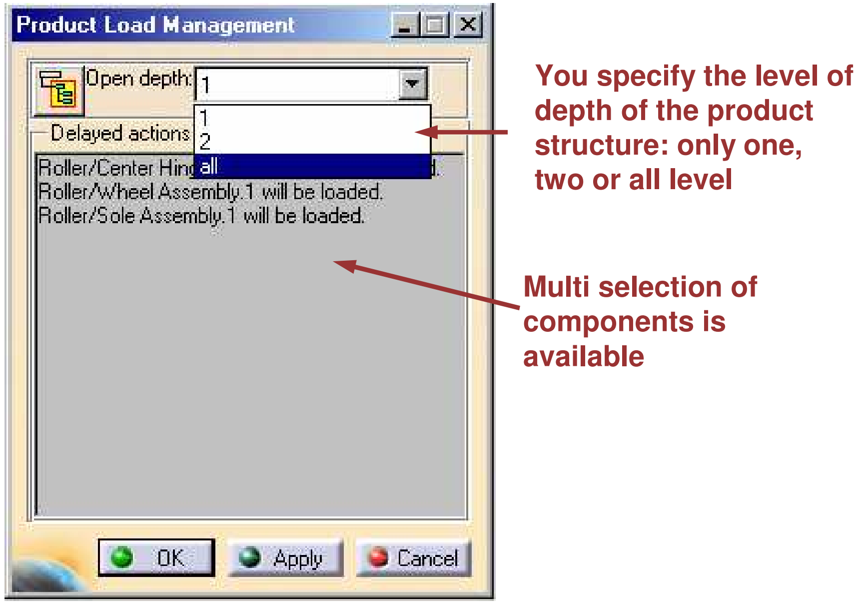
What does this tool do?



In the case of large assemblies, you may not have to load all the documents in CATIA. You need to load only those sub-assemblies and/or components which you wish to visualize and edit. 'Selective Load' tool allows you to manage progressive loading of a product by specifying the level of depth.

This tool is functional if the options:

- "Load Referenced documents" is not checked (in Tools/Options/General)
- "Work with the cache" is checked (in Tools/Options/Infrastructure/Product Structure/Cache Management)
- And optionally "Do not activate default shapes on open" is checked (in Tools/Options/Infrastructure/Product Structure/Product Visualization)



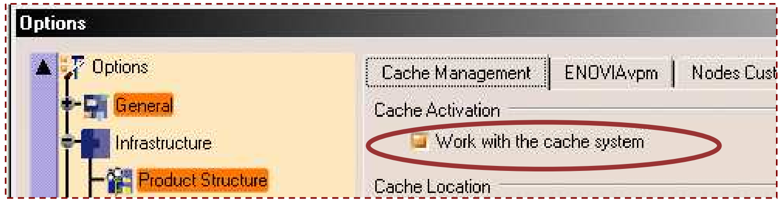
Student Notes:

Selective Load (1/2)

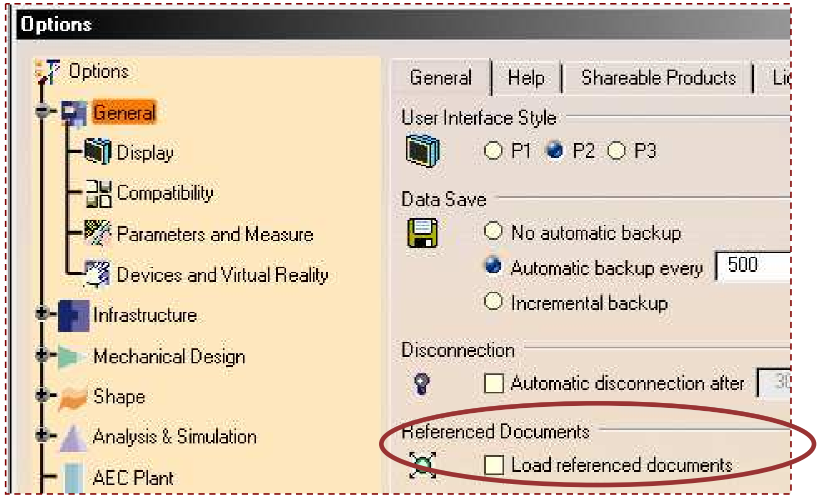
You will learn how to selectively load various components of an assembly using the 'Selective Load' tool.

1 Ensure the following CATIA Options are set.

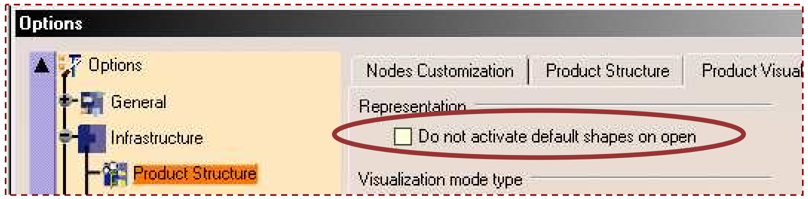
'Work with the cache system' option is activated



'Load referenced documents' option is deactivated



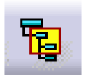
'Do not activate default shapes on open' option is deactivated. (This is an optional setting).



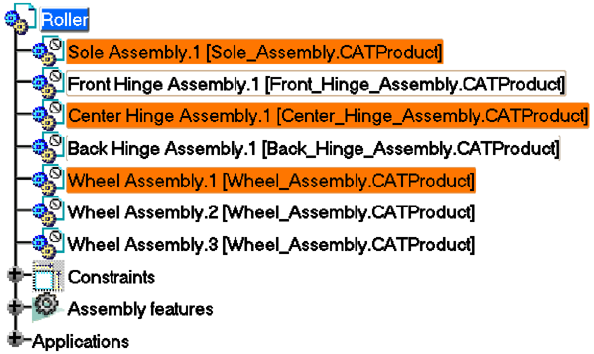
Student Notes:

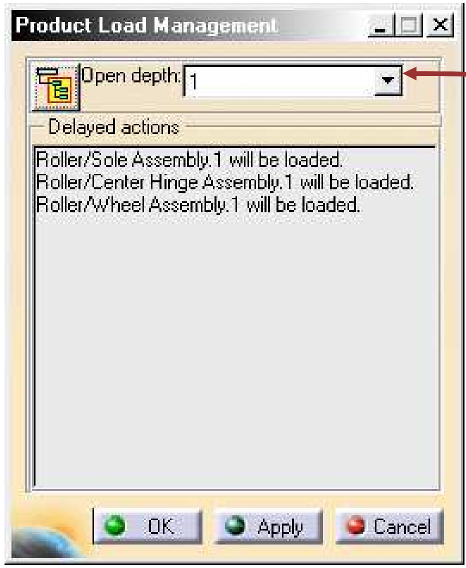
Selective Load (2/2)

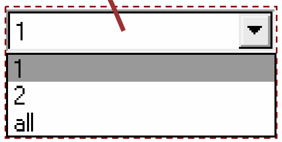
② Click on the “Selective load” tool



③ Multi select the components to load



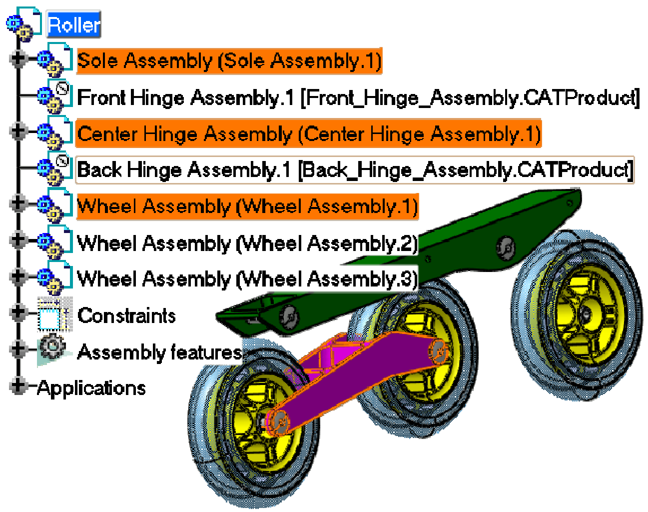




④ Click on the button to add items to the list

⑤ Specify the level of depth

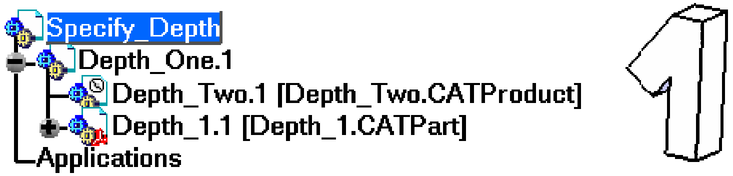
⑥ Click OK to load the selected components.

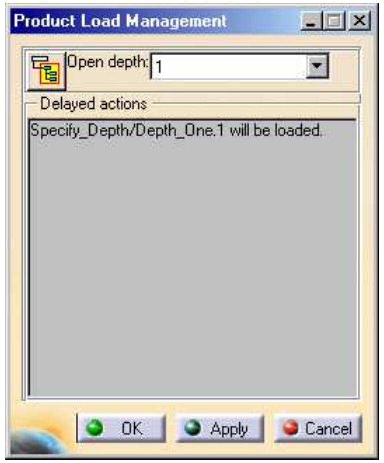


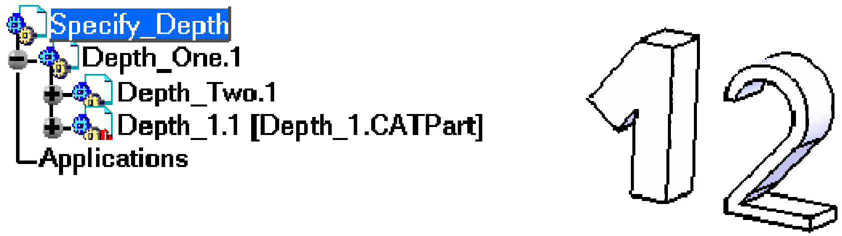
Student Notes:

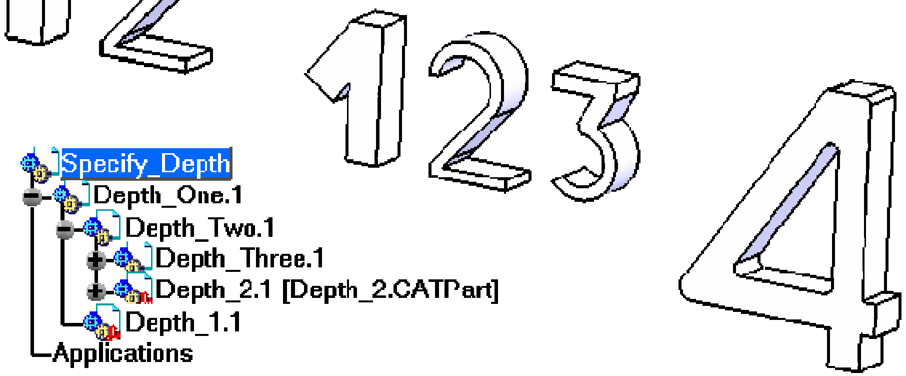
Do It Yourself

Use Specify_Depth.CATProduct.







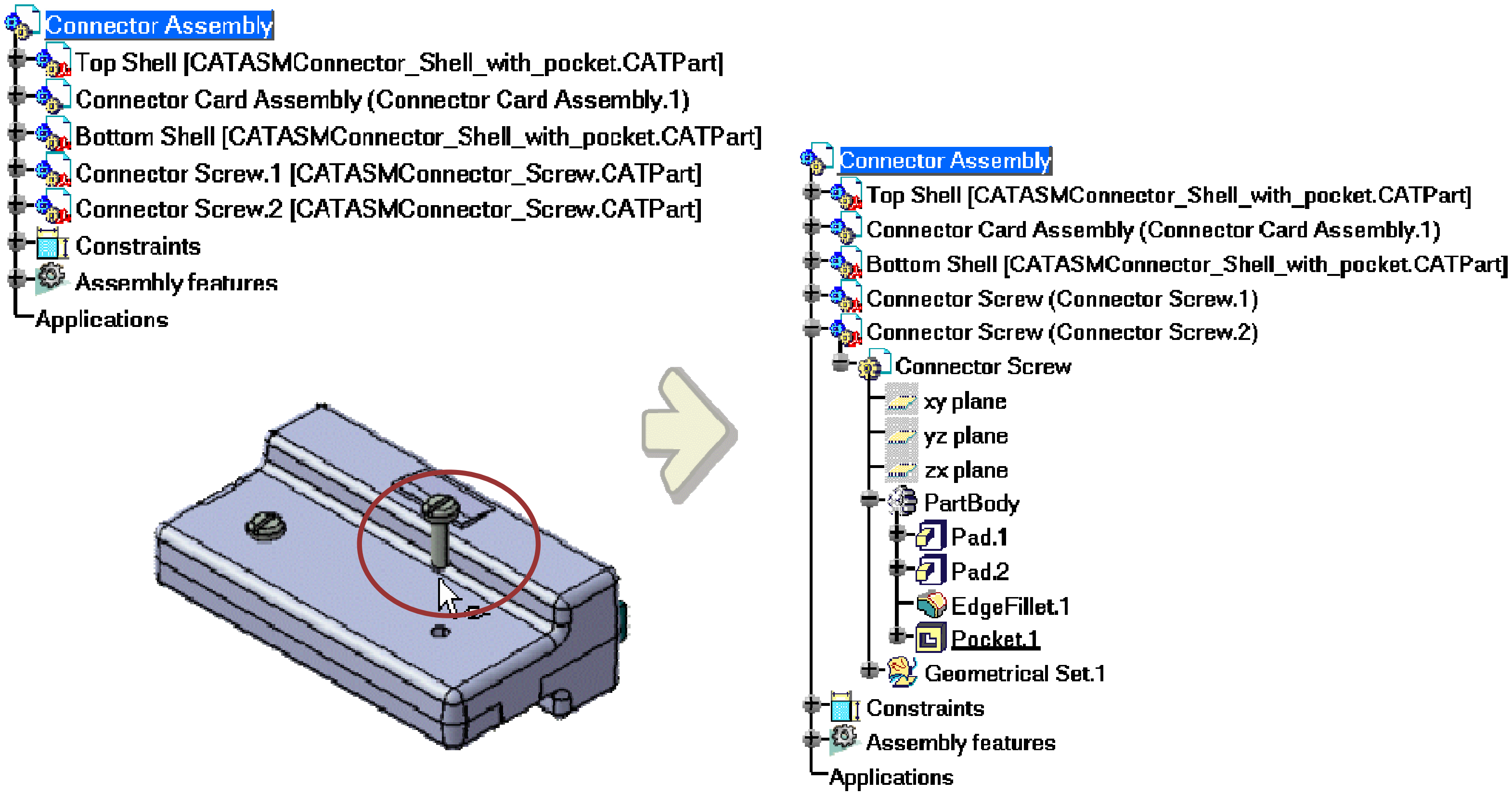


- Launch Specify_Depth.CATProduct
- Load ‘Depth_One.1’ with depth =1
- Unload ‘Depth_One.1’ component
- Load ‘Depth_One.1’ with depth =1
- Unload ‘Depth_One.1’ component
- Load ‘Depth_One.1’ with depth =all

Student Notes:

Using Visualization Mode

You will see how to use Visualization Mode to improve performance.

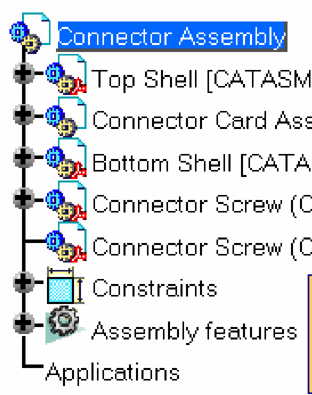


Student Notes:

What Is Called "Visualization Mode"?

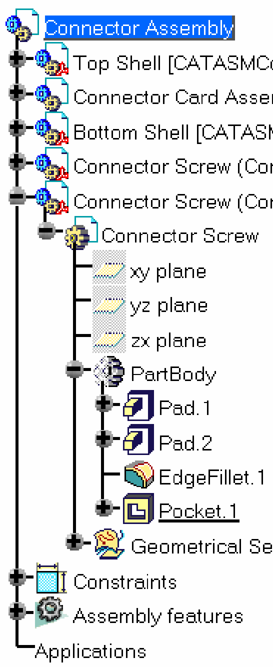
By default an assembly is loaded in design mode. Thus part's definition of all components are loaded in memory (this involve exact geometry and parameters). This step can involve more or less time according to the assembly to load. Then to improve performance you can set CATIA's option to load an assembly in visualization mode. In this mode, a representation of the geometry only is available.

In visualization mode the representation of the geometry is loaded and the corresponding file is a cgr file.



Notice that the screw branch is not expandable and therefore the PartBody is not accessible.

In design mode the exact geometry is available.



Notice that the screw branch is expandable and therefore the PartBody is accessible.

Student Notes:

Visualization Mode vs. Design Mode (1/2)

You will see the differences between the Design mode and Visualization mode:

Behavior	Design Mode	Visualization Mode
Memory and Performance		
Loaded in Memory	Fully Loaded	Partially Loaded
Update Performance	Slow	Fast as entire geometry is not loaded
Display Performance	Normal	Tessellated geometry is loaded
Manipulation Performance (Zoom, Pan, Rotate)	Slow	Fast
Visibility		
Visible in Show	Yes	Yes
Visible in No Show	Yes	Yes
Viewable in DMU and sketcher sections	Yes	Yes
Visible in drafting	Yes	Yes, partially or fully loaded

Student Notes:

Visualization Mode vs. Design Mode (2/2)

Behavior	Design Mode	Visualization Mode
Assembly Constraints and Transformations		
Unpublished Geometry accessible for adding / updating Assembly constraints	Yes	Yes, automatically switches to Design Mode
Published Geometry accessible for adding / updating Assembly constraints	Yes	Yes, partially switches to Design mode
Assembly Analysis		
Compute Clash, Clearance, Contact	Yes	Yes
Compute Mass Property	Yes	No
Compute distances	Yes	Yes, only approximate minimum distance
Part Geometry		
Geometry features accessible in tree	Yes	No
Geometry may be used to define sketches and features in other parts in the assembly (e.g. up-to-plane)	Yes	Yes, automatically switches to Design Mode
In-context features re-generated/updated (e.g. associativity)	Yes	Yes, automatically switches to Design Mode

Student Notes:

User Setting : Turning On the Cache (1/2)

Turning ON the cache system will cause CATIA to load automatically parts and models in Visualization Mode.

The cache is a read/write path located locally on your machine or anywhere on your network and is used to store cgr files. The first time a component is inserted, it is tessellated. This means that the corresponding cgr file is computed and saved in the local cache as well as displayed in the document window. The next time this component is required, the cgr file which already exists (and not the original document) is automatically loaded from the local cache.

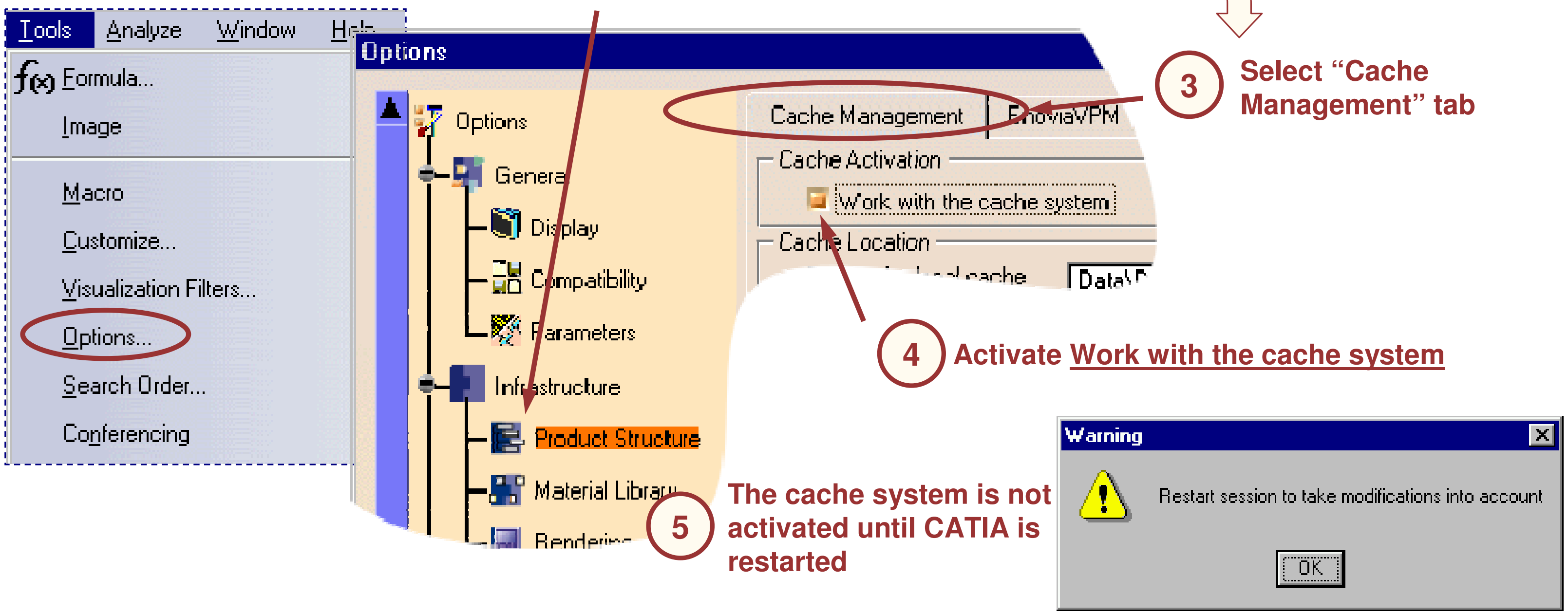
1 **Select Options... from the Tools menu**

2 **Select "Product Structure" branch under "Infrastructure" node**

3 **Select "Cache Management" tab**

4 **Activate Work with the cache system**

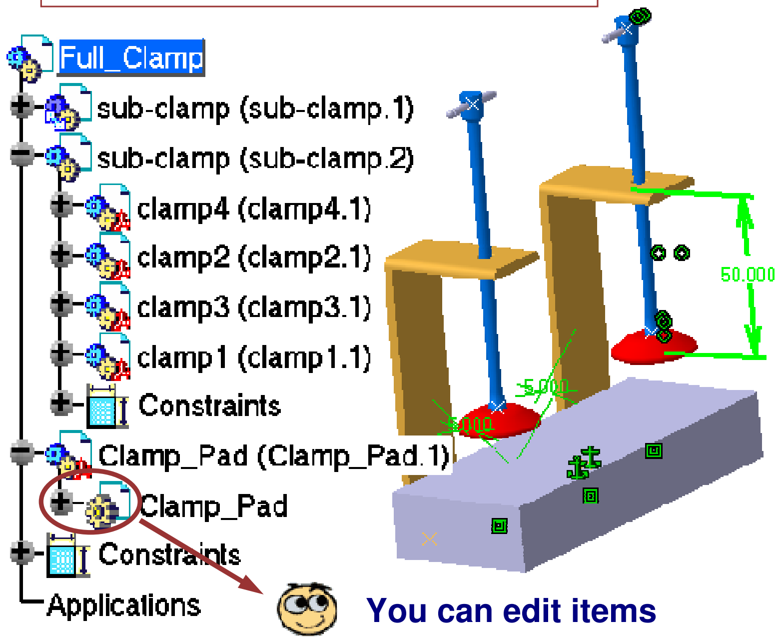
5 **The cache system is not activated until CATIA is restarted**



Student Notes:

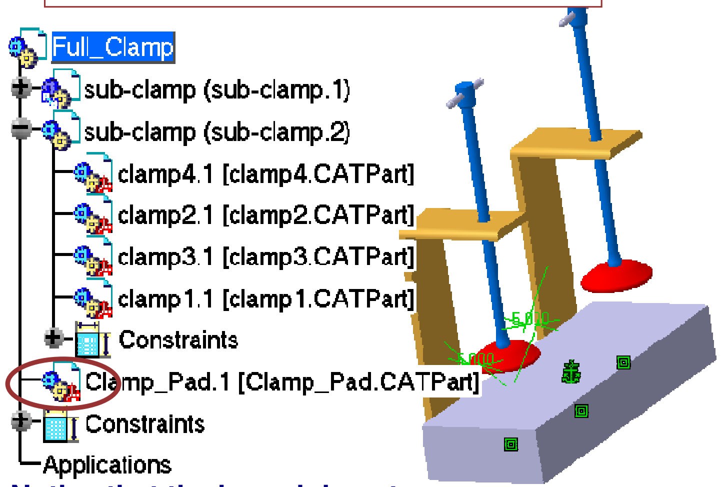
User Setting : Turning On the Cache (2/2)

without Cache System



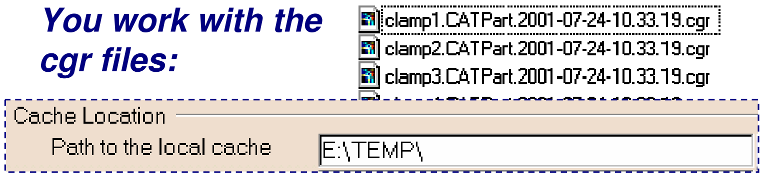
You can edit items

with Cache System



Notice that the branch is not expandable and therefore the PartBody is not accessible.

You work with the cgr files:

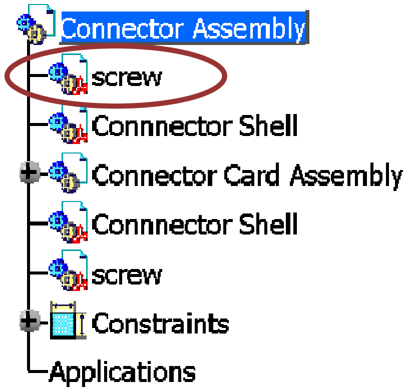


Right-clicking on a component and selecting Design Mode in the contextual menu also switches the part or model to Design Mode:

Manually Switching to Design Mode

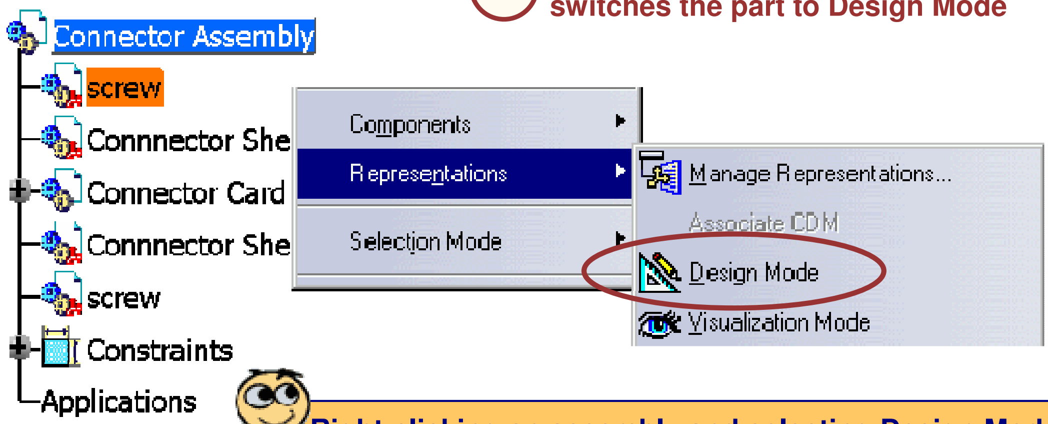
Parts and models can be switched manually to Design Mode on the fly.

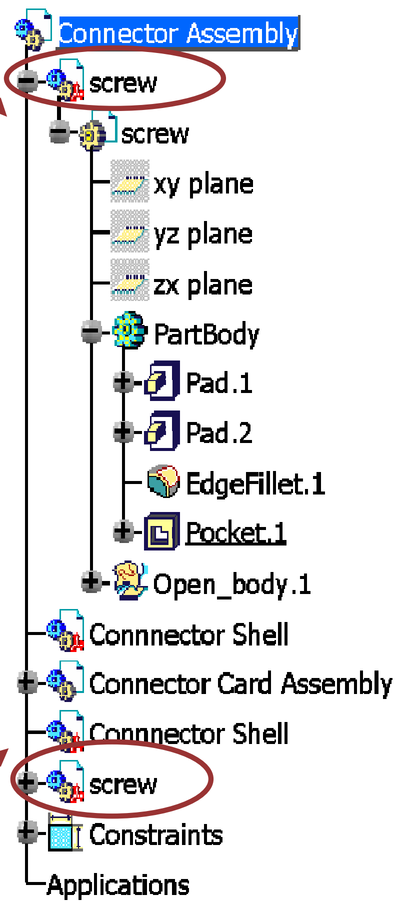
1 **When opening an assembly with the cache activated, parts are loaded in Visualization Mode**



2a **Double-clicking a part in an assembly switches it to Design Mode. Note that all instances of the part switch to Design Mode.**

2b **Right-clicking selecting Design Mode also switches the part to Design Mode**





Right-clicking an assembly and selecting Design Mode switches all parts in the assembly to Design Mode.

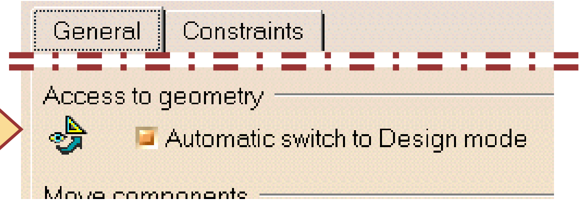
Student Notes:

Student Notes:

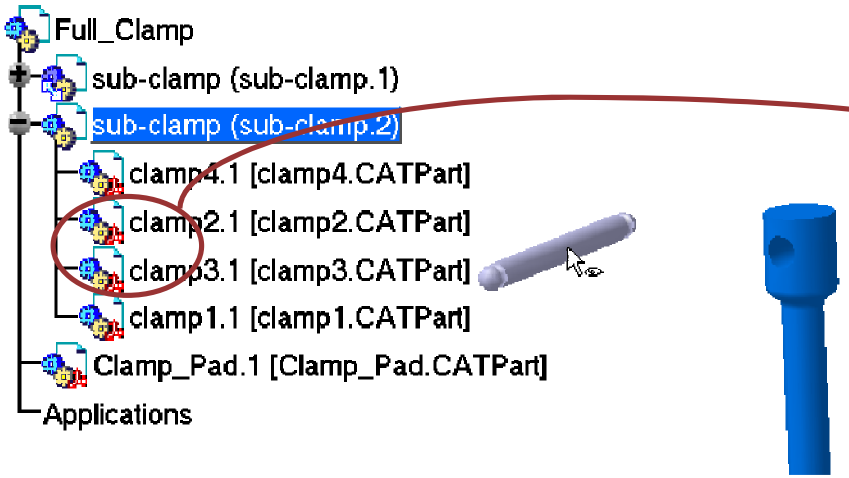
Automatic Switch to Design Mode

This setting allows you to put constraints between components on a product loaded in visualization mode.

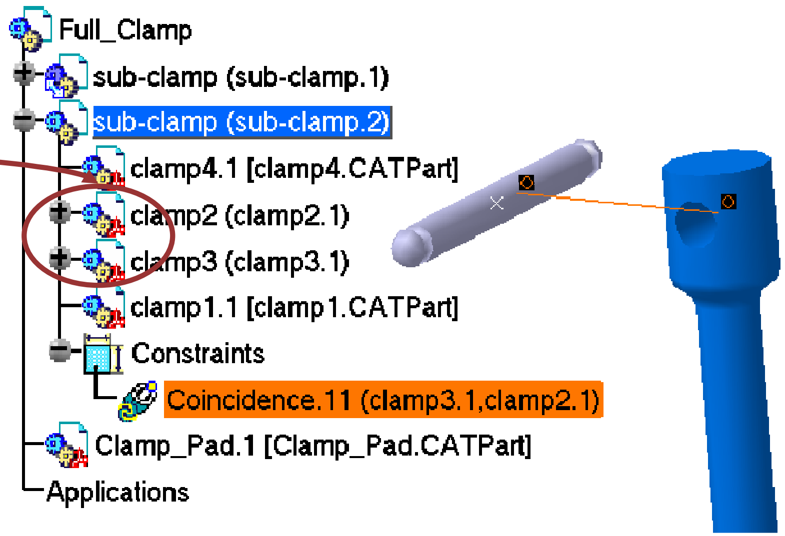
1 **Check that the "Automatic switch to Design Mode" option is activated**



2 **Select a constraint tool. Around a geometry , the cursor will have this shape . Click the geometry**



3 **The two constrained components switch to a Mode (Exact) which enables to select elements.**



Student Notes:

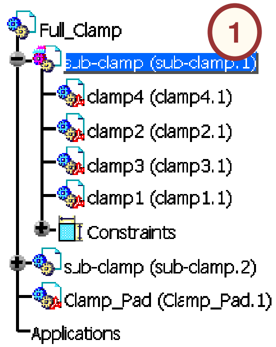
Assembly Constraints and Visualization Mode

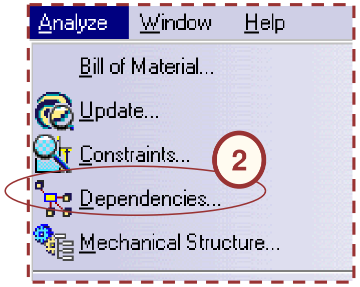
The existing Assembly Constraints are not resolved fully in visualization mode, we need to switch to design mode to resolve these assembly constraints.

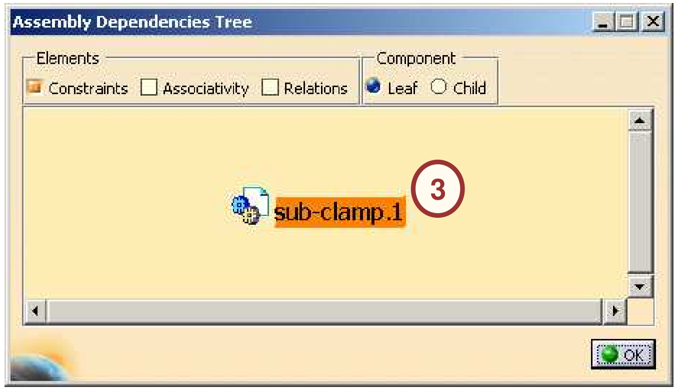
1. Activate the product you want analyze

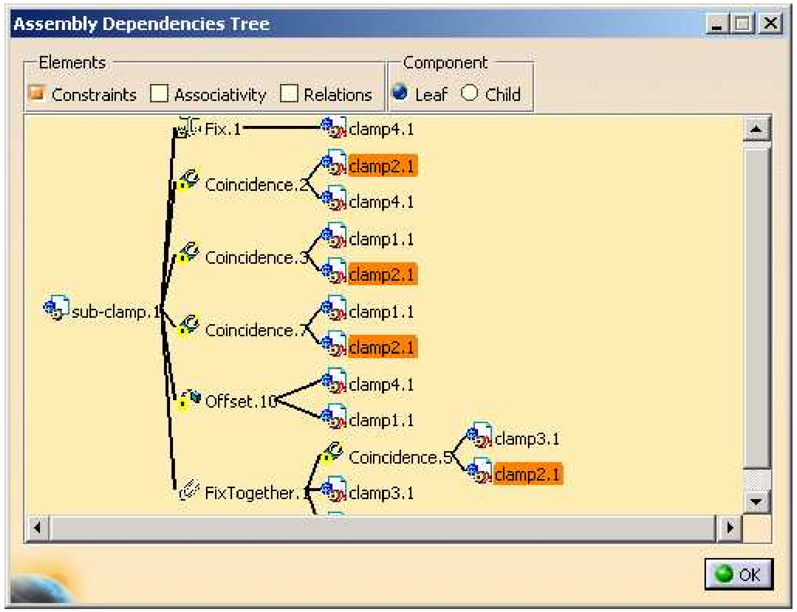
2. Select "Dependencies…" in the Analyze menu

3. Right-click the component and select Expand All to see the components in the network of constraints









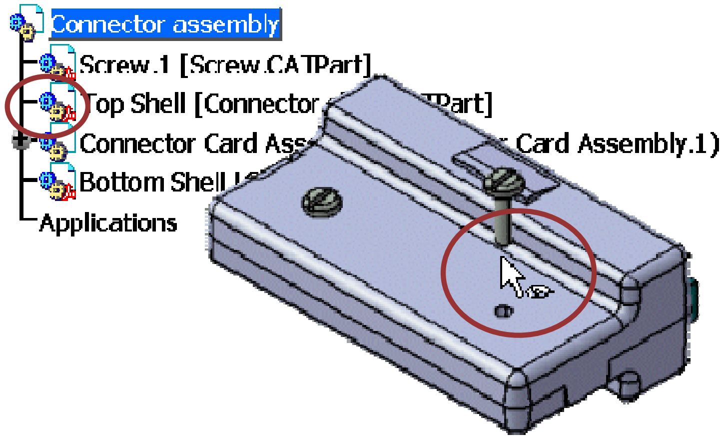
4. Switch to Design mode and again repeat Steps 2 and 3. You will find that these constraints are updated.

Student Notes:

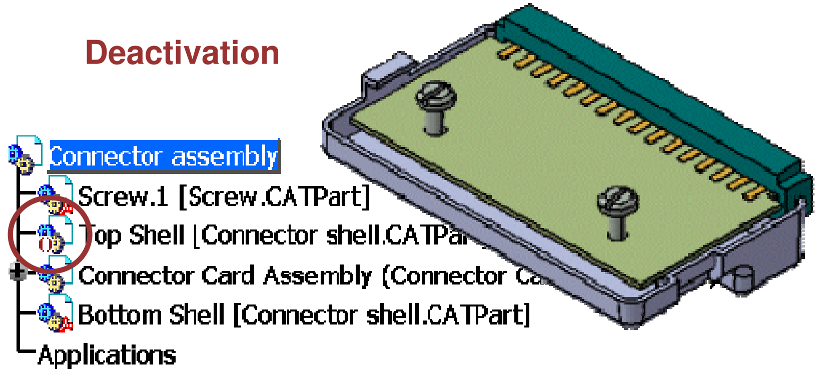
Summary Of Modes

You will see a summary of the capabilities of Visualization Mode, Hide and Deactivate.

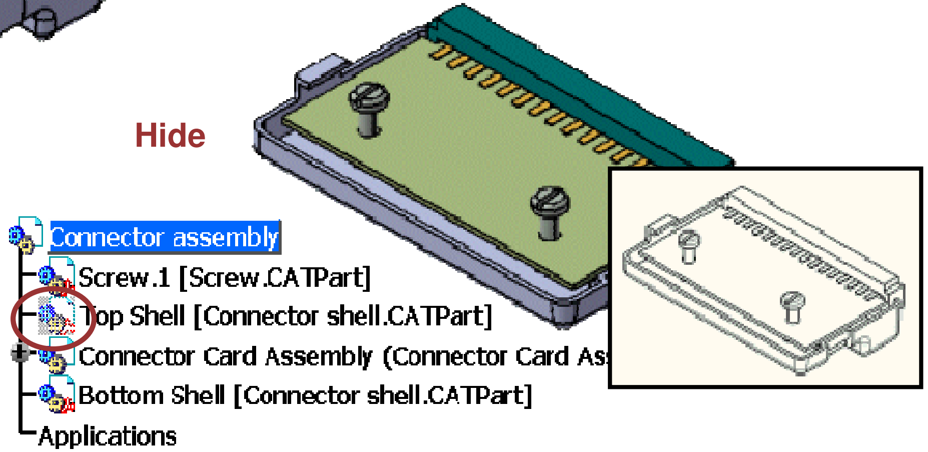
Visualization Mode



Deactivation



Hide



Student Notes:

Differences Between Modes

This table highlights some key reasons for using Visualization Mode, Deactivation, and Hide.

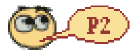
Comparison of Modes				
Behavior	Design Mode	Visualization Mode	Deactivated Node (Design Mode)	Hidden (Design Mode)
Memory and Performance				
Loaded in Memory	Fully Loaded	Partially Loaded	Fully Loaded	Fully Loaded
Load and Update Performance	Normal	**Faster**	Normal	Normal
Display Performance	Normal	Normal	**Faster**	**Faster**
Visibility				
Visible in Show	Yes	Yes	No	No
Visible in No-show	Yes	Yes	**No**	Yes
Viewable in non-shaded mode	Yes	Yes	No	Yes
Viewable in DMU and sketcher sections	Yes	Yes	No	Yes
Visible in drafting	Yes	Yes	No	**No**
Assembly Constraints and Transformations				
Accessible for adding Assembly constraints	Yes	Yes	No	Yes
Assembly Constraints re-generated/updated	Yes	Yes	Yes	Yes
Accessible to define translations & rotations	Yes	Yes	No	Yes
Analysis				
Calculated in Clash, Clearance, Contact	Yes	Yes	No	No
Calculated in Mass Property analysis	Yes	No	**No**	Yes
Accessible for Measurements	Yes	No	No	Yes
Part Geometry				
Geometry features accessible in tree	Yes	No	No	Yes
Geometry may be edited	Yes	No	No	Yes
Geometry may be used to define sketches and features in other parts in the assembly (e.g. up-to-plane)	Yes	Yes	No	Yes
In-context features re-generated/updated (e.g. associativity)	Yes	Yes	Yes	Yes

Student Notes:

Washing Machine

Recap Exercise : Large Assemblies

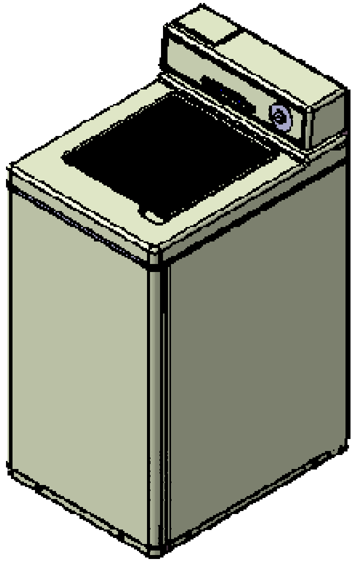
20 min



In this exercise, you will practice working on large assemblies while optimizing CATIA performance.

You will practice:

- Loading only the necessary components of an assembly
- Switching components from visualization to design mode in order to edit them
- Deactivating the representation of the parts not needed for the study

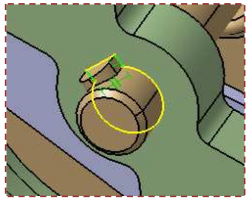


Student Notes:

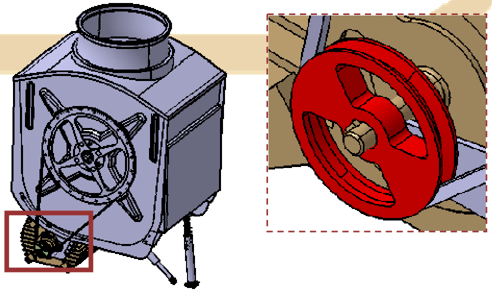
Design Process

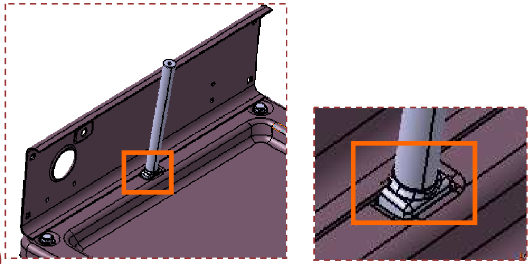
1. Environment preparation: Make necessary CATSettings for working with large assemblies.

2. Load 'Washing_Machine.CATProduct' and necessary components to design in context a missing 'key-way'.



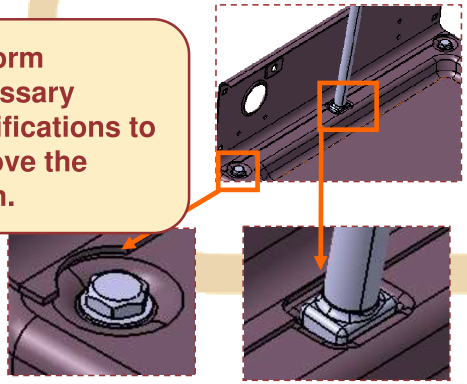
3. Update a sub-assembly without switching all its components to design mode.



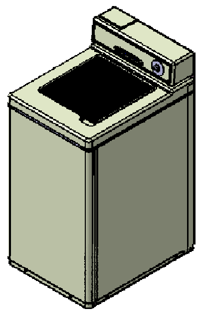


4. Visualize the rear portions of the washing machine, notice a clash, and prepare the assembly for modification.

5. Perform necessary modifications to remove the clash.



6. Visualize progressively the entire assembly.

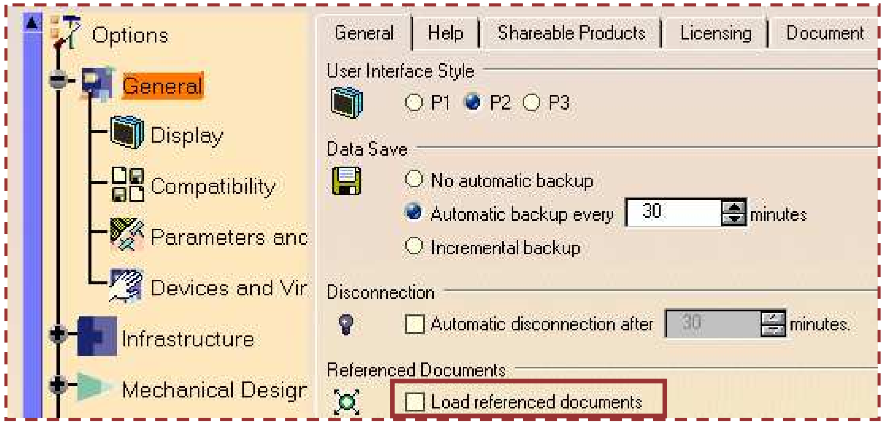


Student Notes:

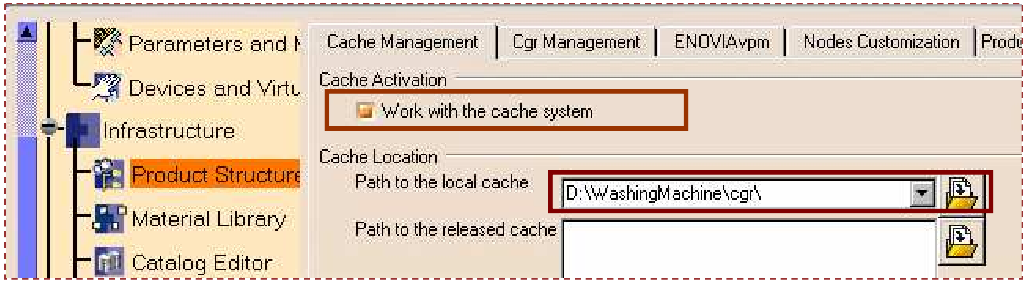
Step 1: CATSettings for Large Assemblies (1/3)

In this step, you will make the necessary CATSettings for working with large assemblies.

- **General : Uncheck "Load referenced documents" option.**



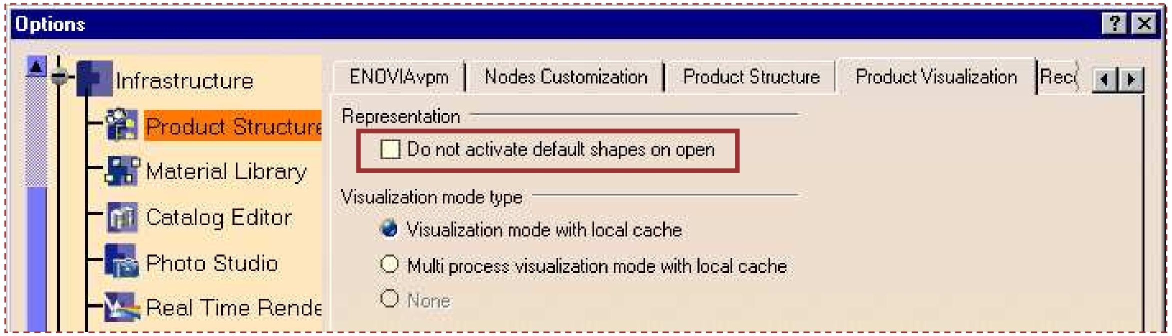
- **Cache Management : Activate "Work with the cache system", set the cache location path.**



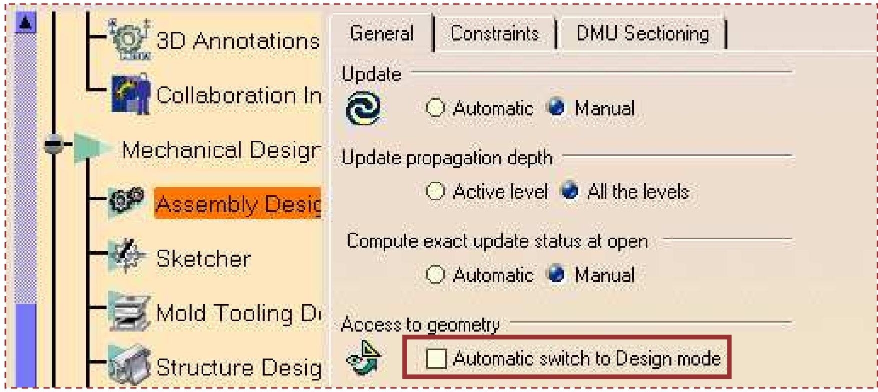
Student Notes:

Step 1: CATSettings for Large Assemblies (2/3)

- **Product Visualization : Uncheck "Do not activate default shapes on open".**



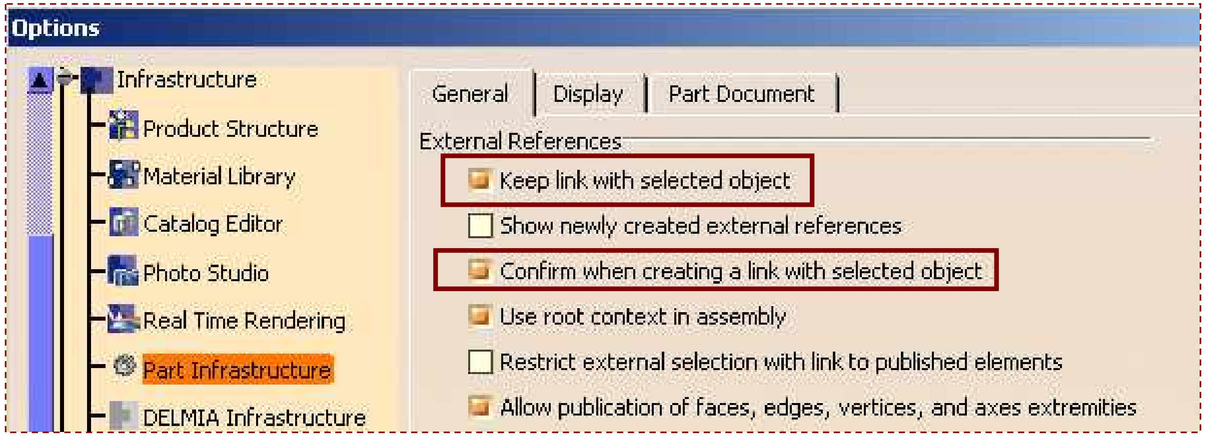
- **Assembly Design > General : Uncheck "Automatic switch to Design mode" option.**



Student Notes:

Step 1: CATSettings for Large Assemblies (3/3)

- **Part Infrastructure : Check “Keep link with selected object” and “Confirm when creating a link with selected object”.**



- **Restart CATIA to take into account Cache management settings.**

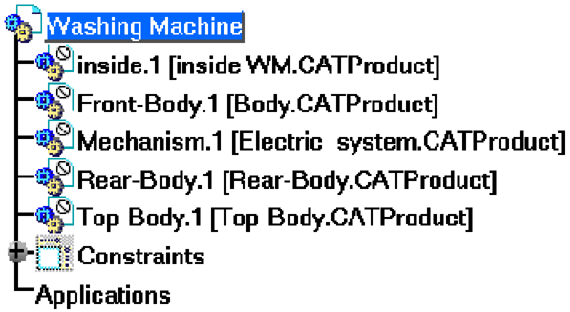
Student Notes:

Step 2: Design in Context (1/3)

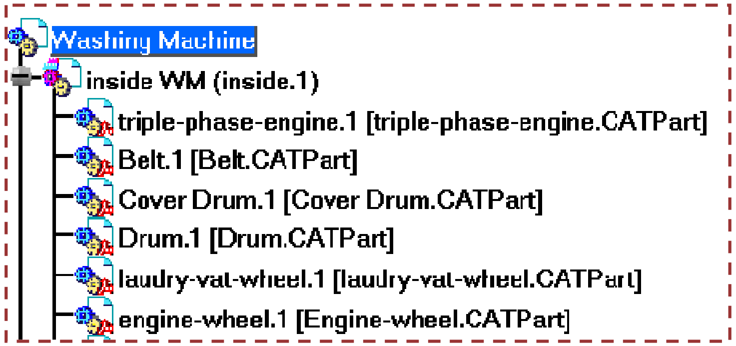
Product used: Washing_Machine.CATProduct

In this step, you will design in context a key-way slot in the "engine-wheel".

- **Open "Washing_Machine.CATProduct". Opening is very fast as no other document than the root product is loaded.**



- **You are going to work on "Inside.1" instance. Load it using contextual menu. The sub-assembly "inside.1" is loaded.**

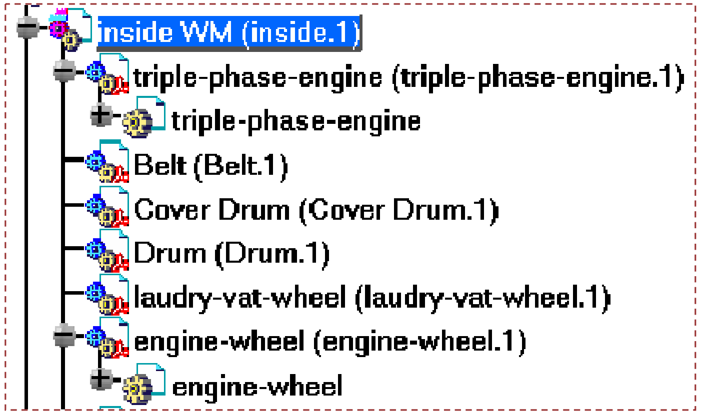


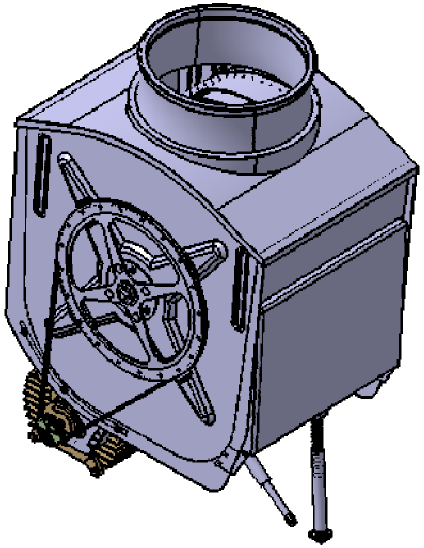
Student Notes:

Step 2: Design in Context (2/3)

- **As cache is activated, the parts are loaded in visualization mode. Notice that their representation is facetized. You can also check that cgr files have been created in the repository you indicated.**

- **Switch "engine-wheel" and "triple-phase-engine" to design mode and activate "engine-wheel".**

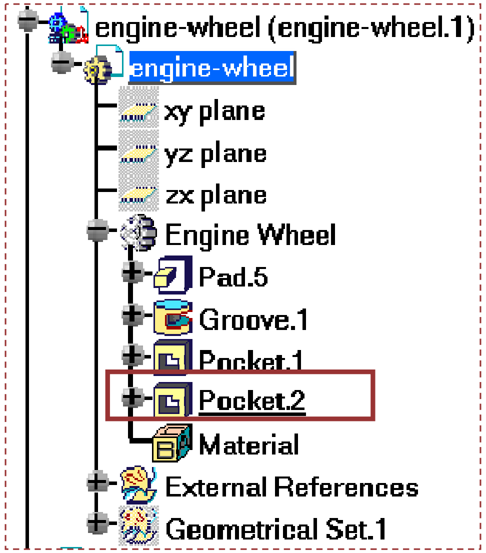


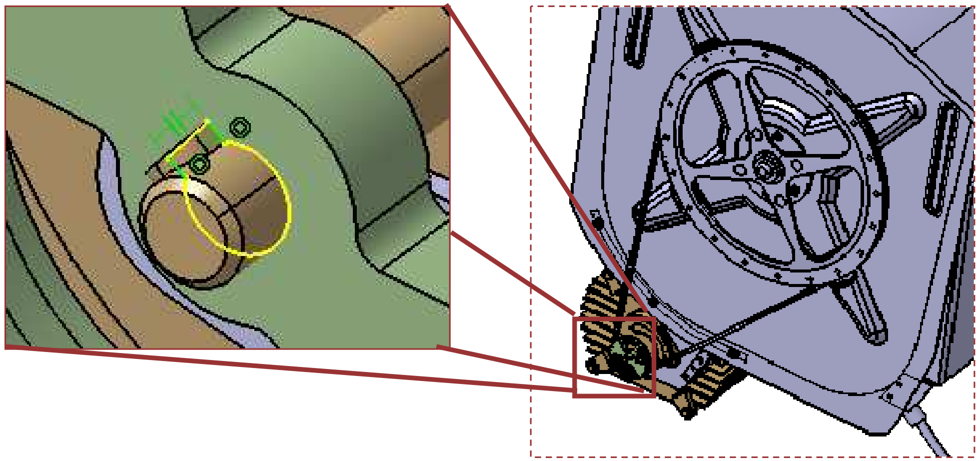


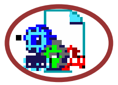
Student Notes:

Step 2: Design in Context (3/3)

- Design the keyway slot of “engine-wheel” using existing faces of “triple-phase-engine” component.







The part “engine-wheel.1” is now designed in context of “triple-phase-engine.1”

- Activate sub assembly “inside.1”.

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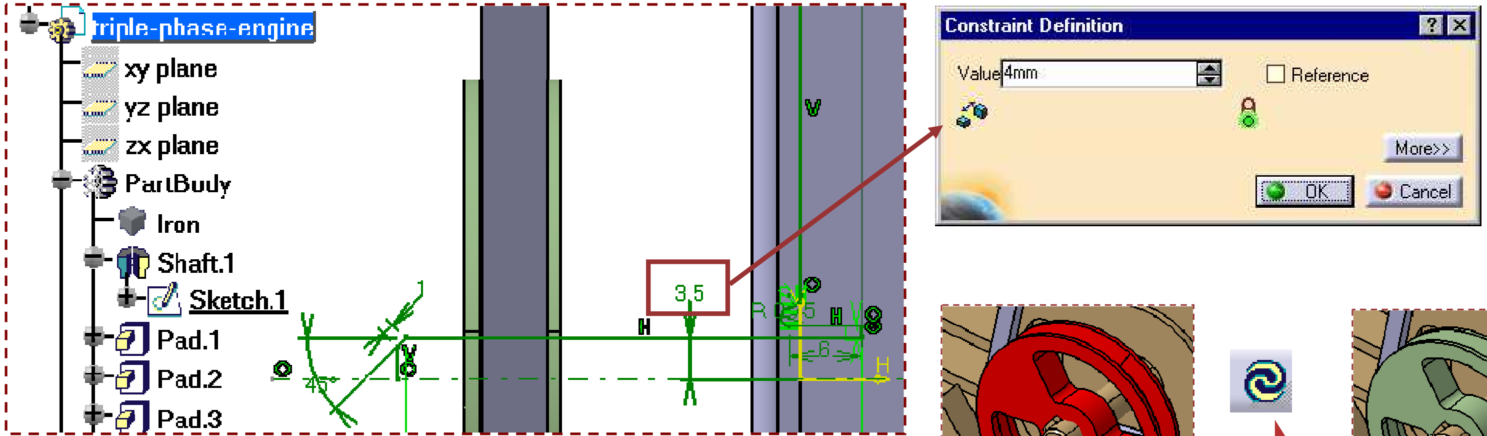
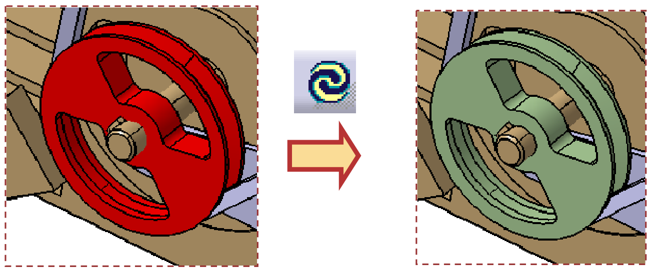
Student Notes:

Step 3: Modify Design and Update Assembly

In this step, you will modify the diameter of driving shaft, propagate changes to "engine-wheel" and update assembly.

- **Edit "triple-phase-engine.CATPart". Change the radius of shaft from 3.5 mm to 4.0mm.**
- **Update "Inside WM" assembly.**
- **Only the two parts in design mode are updated because "Automatic switch to design mode" option is deactivated. This is a gain in performance.**



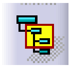


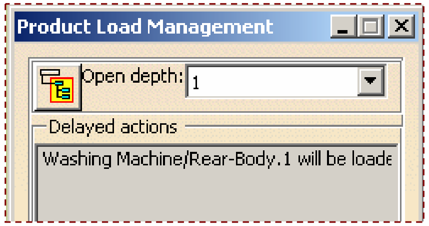
Student Notes:

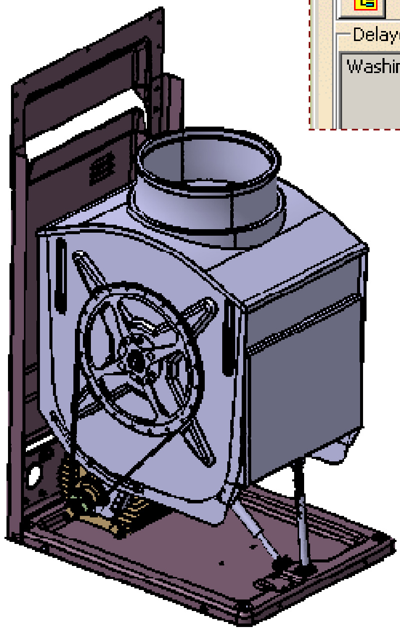
Step 4: Deactivate and Hide Components (1/2)

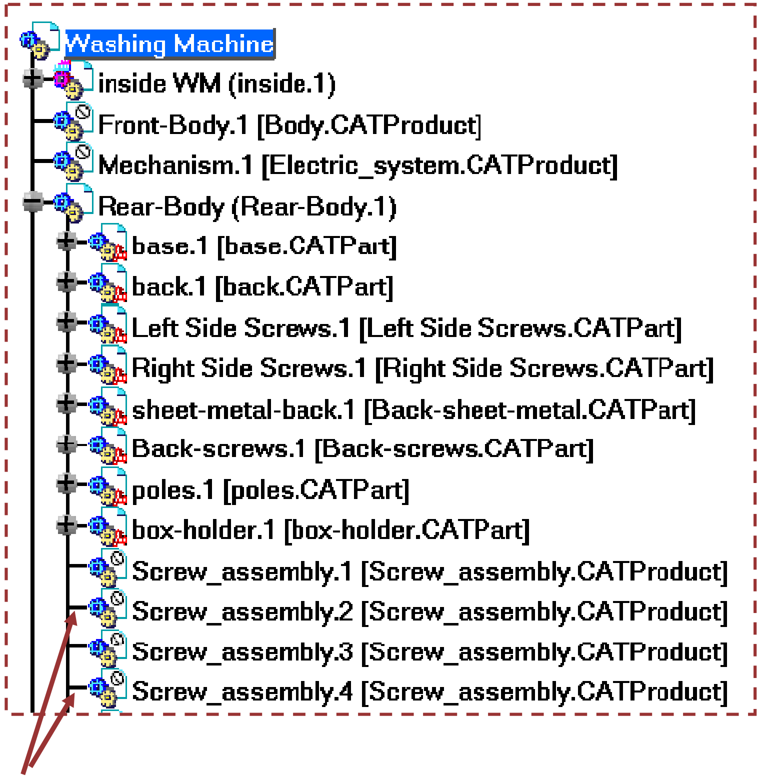
In this step, you will deactivate representations and hide components to visualize the components involved in clash clearly.

- **Activate root product and perform a "selective load" of "Rear-Body" choosing level 1 only.**







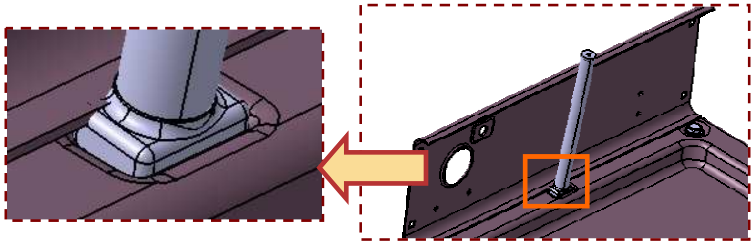


- **The "Screw-assemblies" that are not useful to our study are not loaded and this way are not affecting the performance of CATIA.**

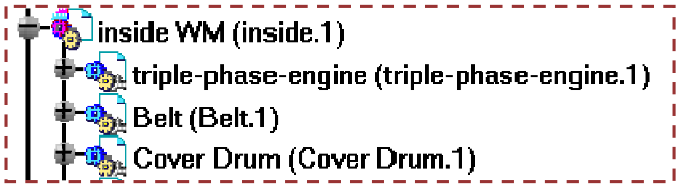
Student Notes:

Step 4: Deactivate and Hide Components (2/2)

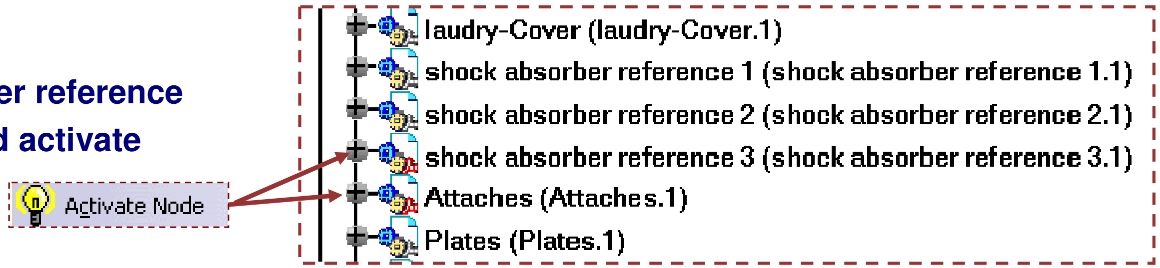
- Note that there is a clash between "Back-sheet-metal" and one of the shocks absorbers of the inside.



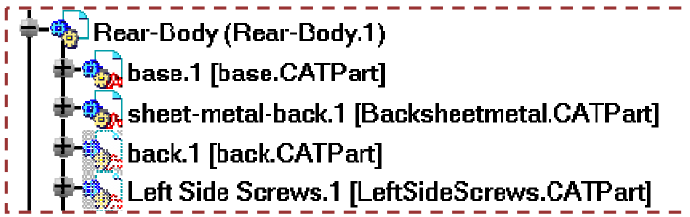
- To reduce the number of components on screen and in memory, you deactivate representations of sub assembly "Inside WM" using contextual menu "Deactivate terminal node".



- Select "shock absorber reference 3" and "Attaches" and activate these nodes.



- In "Rear-Body" hide all loaded components except "base" and "Back-sheet-metal".

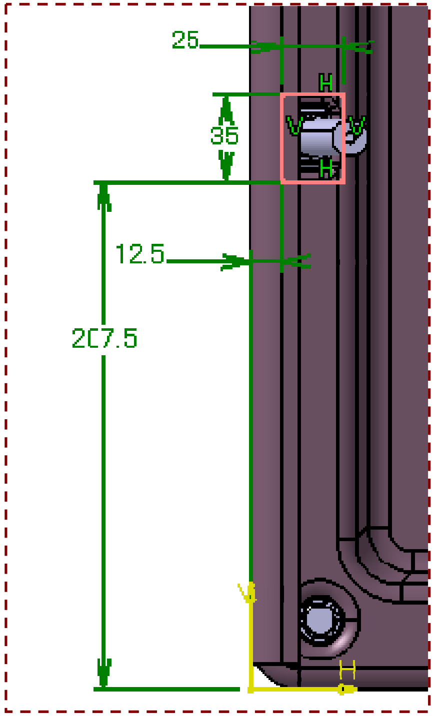


Student Notes:

Step 5: Modify Design and Update Assembly (1/2)

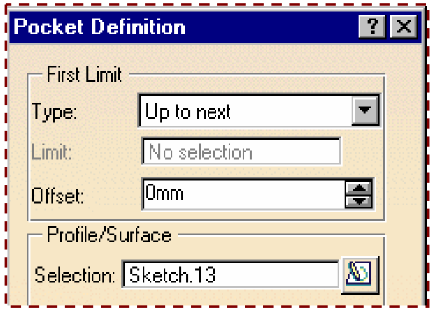
In this step, you will modify design of "Back-sheet-metal" part to remove the clash detected in previous step.

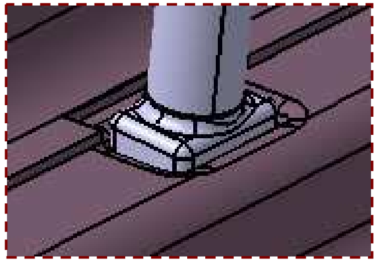
- **Double-click on "Back-sheet-metal" in order to switch the component to design mode. Activate the part "Back-sheet-metal".**
- **Create a sketch for pocket as shown, using highlighted face as sketch support.**

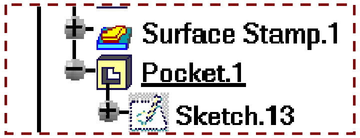


Pocket sketch

- **Create a pocket using Pocket sketch and parameters as shown.**

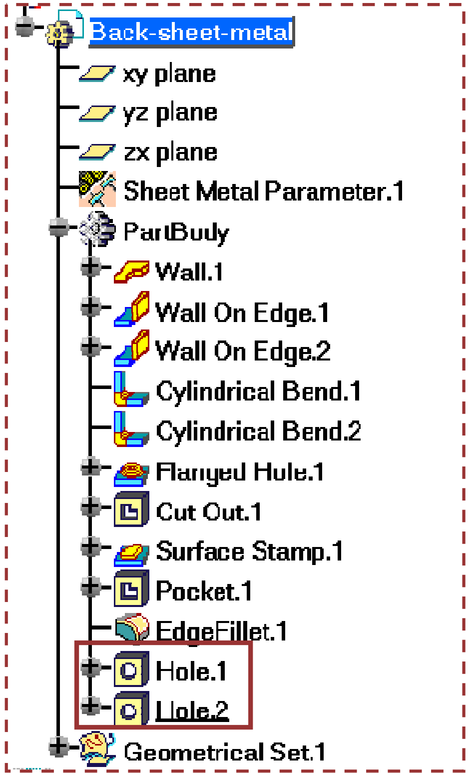




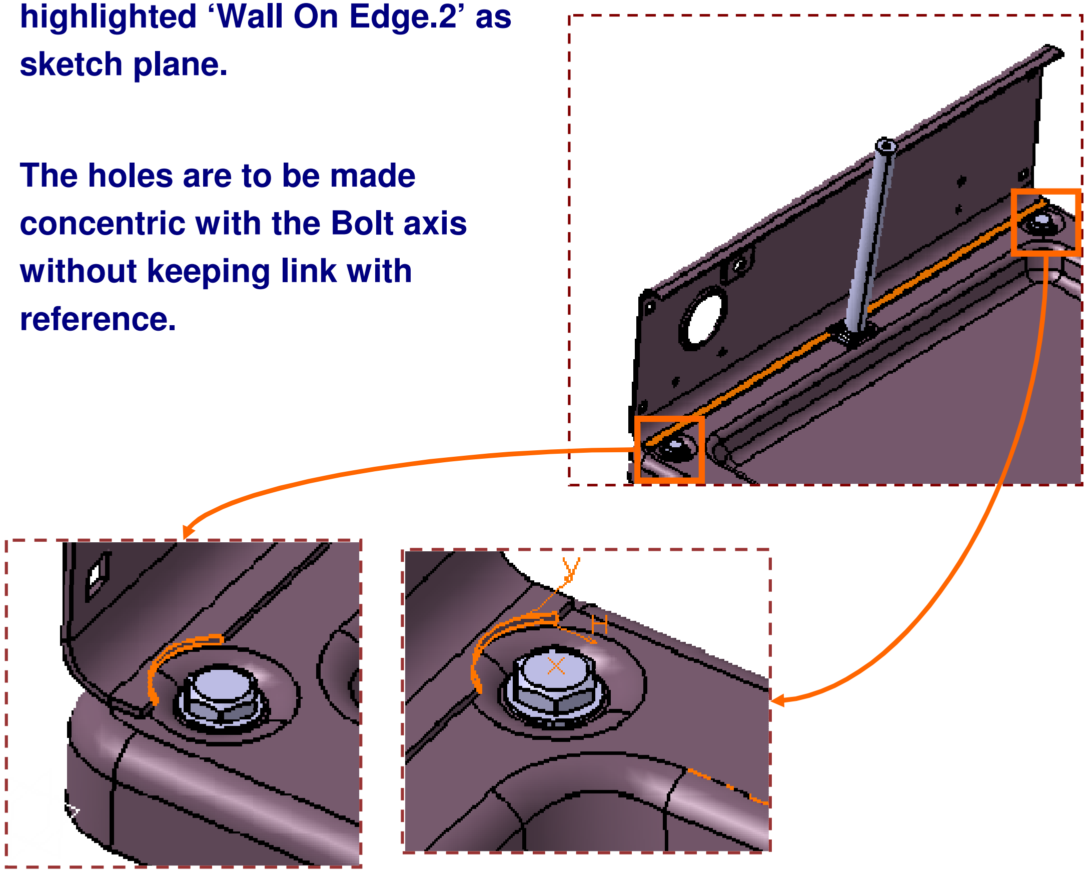


Student Notes:

Step 5: Modify Design and Update Assembly (2/2)



- Create two holes as shown, using highlighted ‘Wall On Edge.2’ as sketch plane.
- The holes are to be made concentric with the Bolt axis without keeping link with reference.



Two Holes with Diameter = 30 mm, depth=2mm

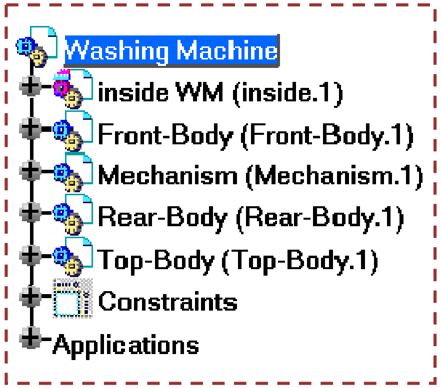
Student Notes:

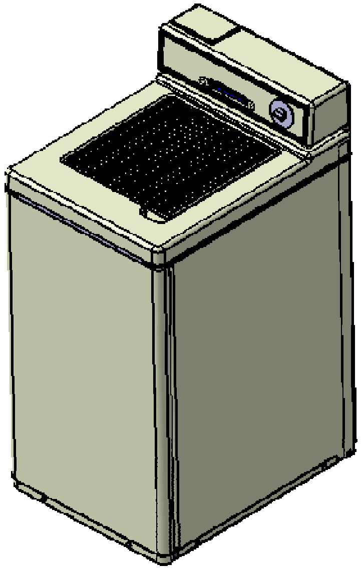
Step 6: Visualize Entire Assembly

You can compare your result with attached model “Washing_Machine_end.CATProduct”

In this step you will visualize entire assembly by showing hidden components and activating deactivated representations.

- In “Rear-Body”, show all hidden components.
- In “Inside WM”, activate terminal Node to activate all representations.
- Load in the following order:
  - Mechanism
  - Top-body
  - Front-body





Student Notes:

To Sum Up (1/2)

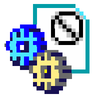
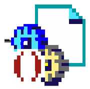
In this lesson, you have seen is how to improve performance of CATIA while working with large assemblies by :

- **Hiding components : Display performance can be improved by hiding components which are not being edited.**
- **Deactivate Representations : Deactivated representations are not loaded in the memory and this improves performance of CATIA. CATIA takes less time to open, zoom, pan, save large assemblies.**
- **Deactivate Components : Deactivated components are not represented in Bill of Material of an assembly. Hence it is possible to generate several configurations of Bill of Material of an assembly.**
- **Selective Load: This command allows you to manage progressive load of a product, you can also specify the level of depth of loaded components.**
- **Using Visualization mode : With this mode, components are partially loaded (only cgr is loaded),which improves the peformance of CATIA. To edit the component, you need to switch to design mode.**

Student Notes:

To Sum Up (2/2)

Here is a summary of viewing an assembly document in various modes.

Component Status		Visualization (Shape Representation)	Bill of Material (BOM)	Accessibility (possibility of applying constraints)	Improvement in Performance
NO SHOW (Hiding Components)		NO	YES	YES (you can apply constraints between the hidden object and the other components in the Show space)	NO
UNLOAD (Unloading Components)		NO	NO	NO	YES
Deactivating a Node		NO	YES	YES (you can apply a constraint even if the shape is deactivated)	YES
Deactivating a Terminal Node		NO	YES	YES	YES
Deactivating a Component		NO	NO	NO	NO

Student Notes:

Analyzing Assemblies to Prepare drawing

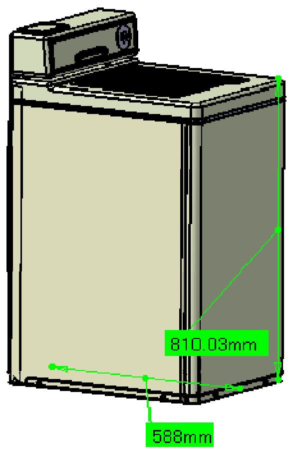
In this lesson, you will learn how to analyze assemblies and prepare assembly drawings

- Introduction to Analyzing Assemblies
- Measuring, Sectioning, Clash
- Managing Scenes
- Product Structure Numbering
- Generating Annotations
- Generating Reports
- To Sum Up

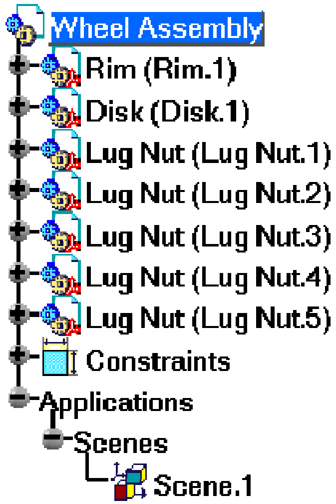
Student Notes:

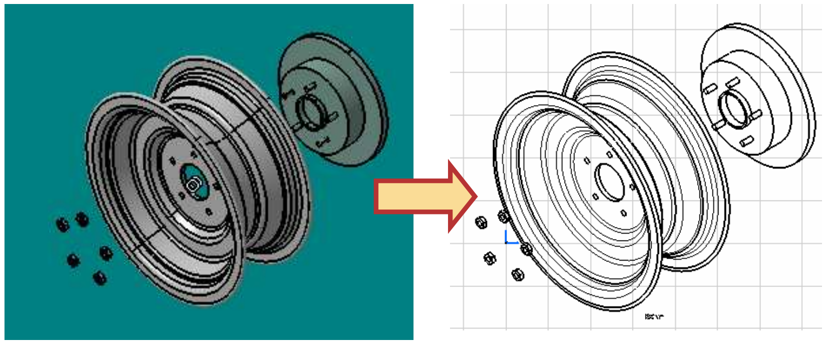
Introduction to Analyzing Assemblies (1/2)

In this lesson, you will learn how to analyze assemblies. You will see how to measure distance between components, create sections, compute clash and analyze clash.



You can create scenes to represent various state, color and position of components in an assembly. Scenes are stored in the root product file.



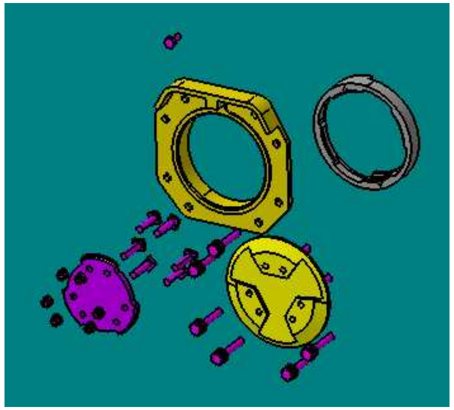


You can measure minimum distance between components.

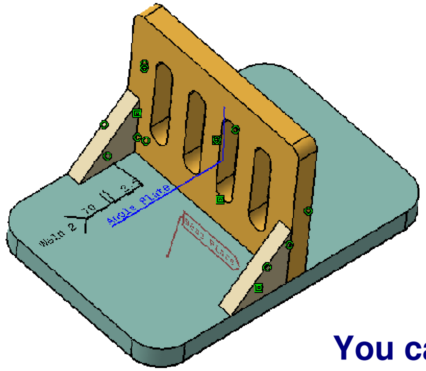
Student Notes:

Introduction to Analyzing Assemblies (2/2)

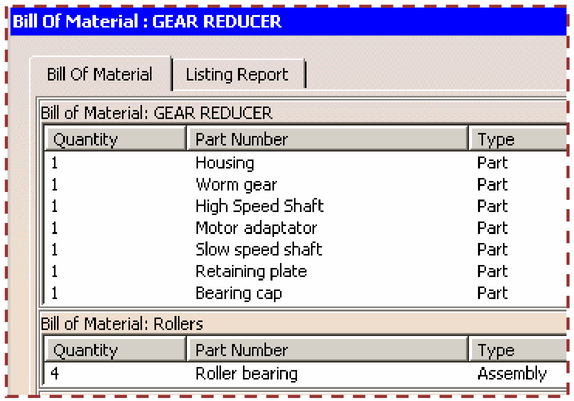
You can create exploded views in scene window.



In Product Design Workbench, you can three types of annotations : Weld Annotations, Flag with Leader and Text with Leader.



You can generate Bill of Material and Listing reports using Analyze > Bill of Material menu.


Bill Of Material : GEAR REDUCER

Bill Of Material | Listing Report

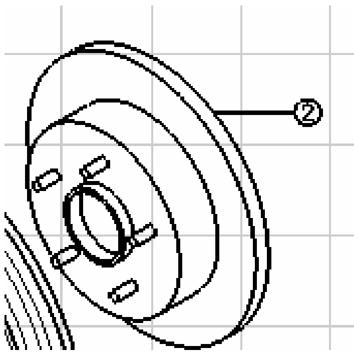
Bill of Material: GEAR REDUCER

Quantity	Part Number	Type
1	Housing	Part
1	Worm gear	Part
1	High Speed Shaft	Part
1	Motor adaptator	Part
1	Slow speed shaft	Part
1	Retaining plate	Part
1	Bearing cap	Part

Bill of Material: Rollers

Quantity	Part Number	Type
4	Roller bearing	Assembly

Using Generate Numbering command,you can associate numbers or letters to various components, which appear in the balloons in drawing generated from the CATProduct.



Student Notes:

Measuring , Sectioning , Clash

In this lesson,you will learn how to make a space analysis of your assembly using CATIA V5 Space Analysis common tools:

- Introduction to Measuring, Sectioning and Clash
- Measuring Minimum Distance
- Sectioning
- Computing Clash
- To Sum Up

Introduction to Measuring , Sectioning and Clash

Student Notes:

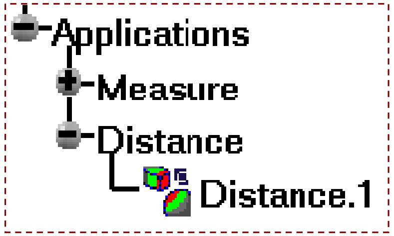
To make previsions for possible errors or to define box bounding for new part, tools in CATIA V5 are available to analyze assemblies:

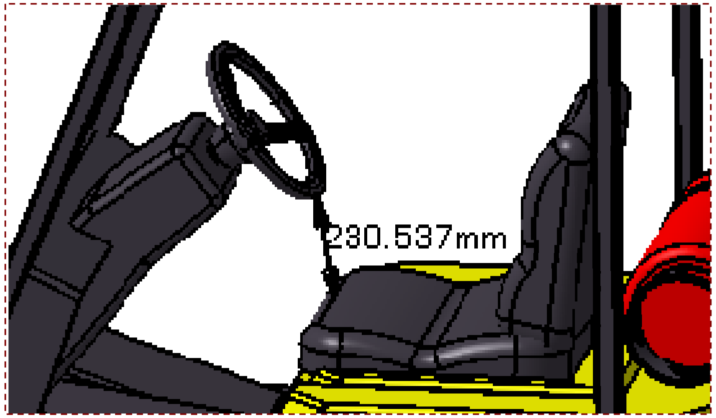
- **Analyzing Clearance**
- **Analyzing Clash**
- **Sectioning tool**
- **Measuring in the 3D view and in the section view to reuse parameter that drives a part in another part, in order to link their geometry**

Student Notes:

Measuring Minimum Distance

You will learn how to measure minimum distances and update them.





Student Notes:

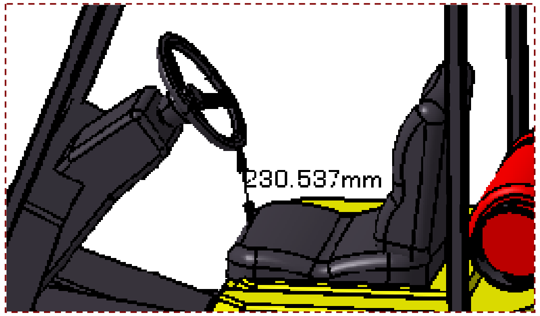
About Measuring Minimum Distances

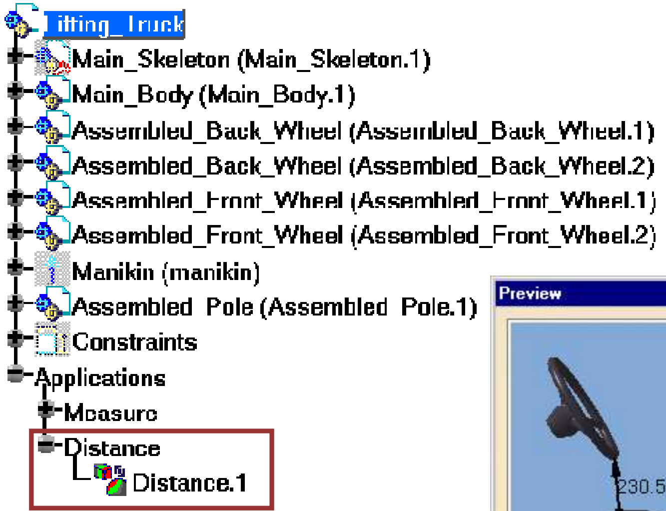
Measuring Minimum Distance

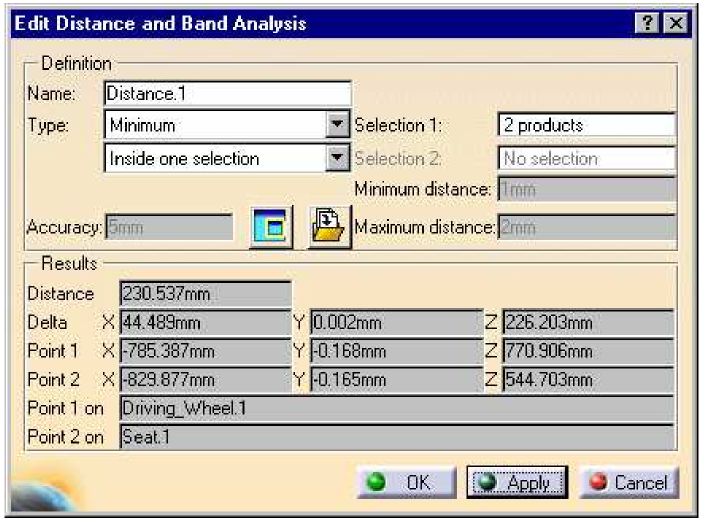
This tool will help you to measure minimum distance, at product level, in a group or between two groups, two components…

Its main characteristic is its ability to be updated. If elements have moved you can update the measure.

We will be able to make dynamic measures in Fitting or Kinematics Simulations using this function







Student Notes:

How to Fill Edit Distance Window

You will see how to give inputs to the Edit Distance and Band Analysis window.

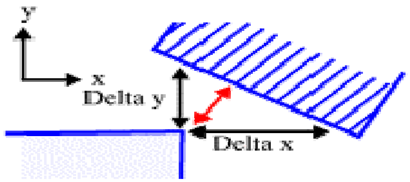
1 Name your measure

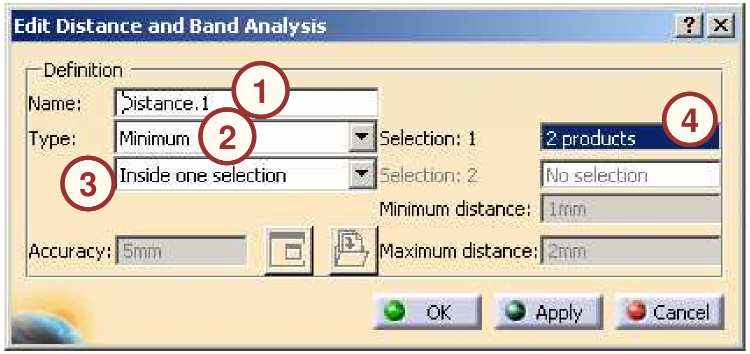
2 Select the type of measure text field

There are 5 possibilities:

- **Minimum : measure minimum distance**
- **Along X: measure minimum distance along X**
- **Along Y: measure minimum distance along Y**
- **Along Z: measure minimum distance along Z**
- **Band Analysis : compute the areas on products corresponding to a minimum distance within a user-defined range**

Measure minimum





3 Select the computation type

- **Inside one selection :Each selected components are tested against all others in the same selection**
- **Between two selections :Each component in the first selection are tested against all components in the second selection**
- **Selection against all : Each component in the defined selection against all others in the document.**

4 Select the products according to computation type

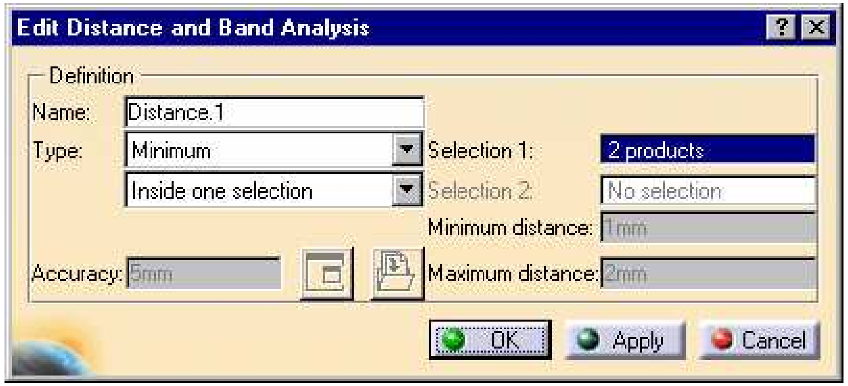
Student Notes:

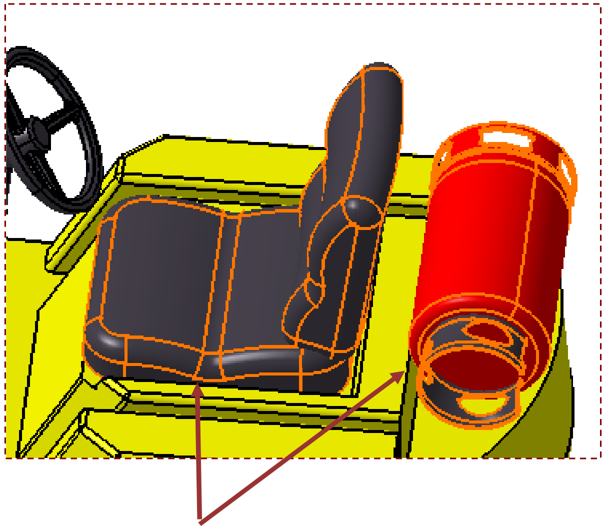
Measuring Minimum Distance (1/2)

You will find out minimum distance between two components.

1 Click on Distance icon 
The Edit Distance Window appears

2 Fill the Edit Distance and Band Analysis window, then select the highlighted components as 'Selection 1'





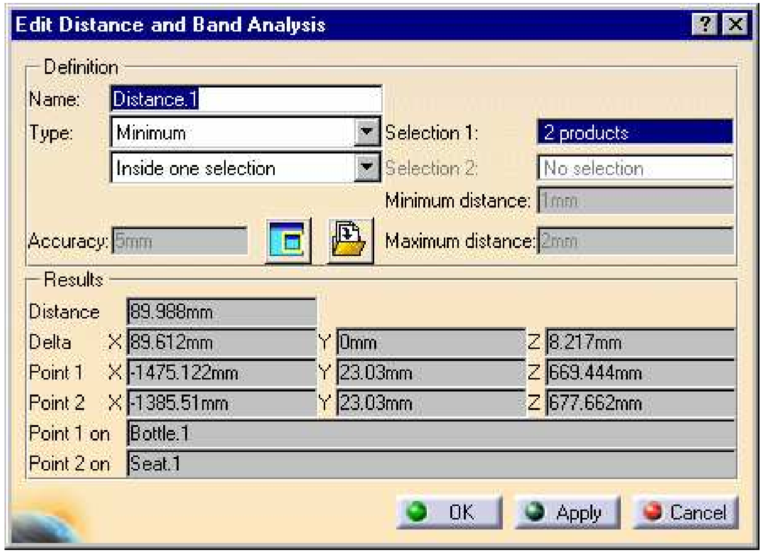
Select highlighted components

Student Notes:

Measuring Minimum Distance (2/2)

3 **Click apply to compute the distance**

The Preview window displays selected products and the minimum distance



4 **Click OK to confirm**

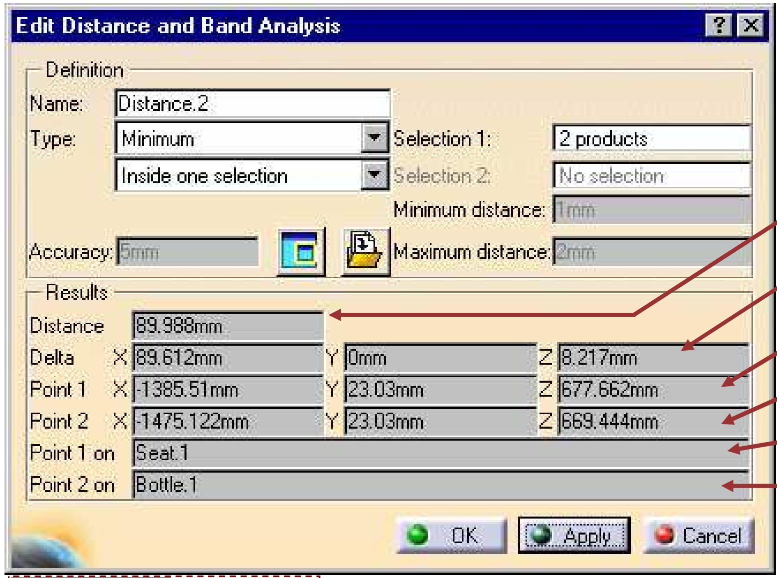
If you do not check the automatic option click on to view the results in a separate viewer.

Student Notes:

Reading Minimum Distance Results

You will read minimum distance results created.

1 Click on the 'Distance and Band analysis' icon then select the seat and the gas cylinder and apply. After this Computation, you get:



Minimum distance value

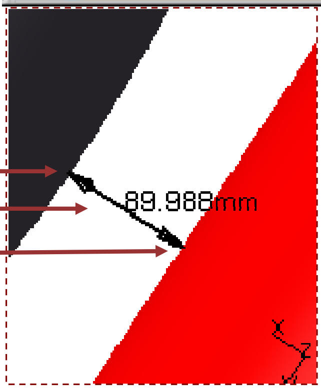
Vector values from point 1 to 2

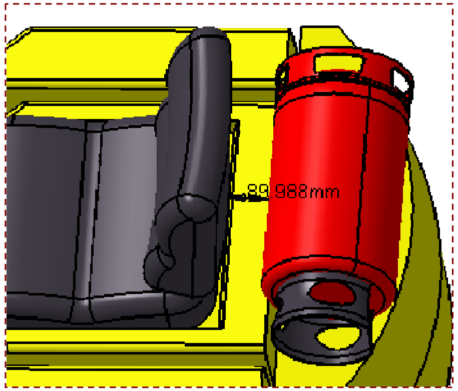
Point 1 coordinates

Point 2 coordinates

Element on which is placed Point 1

Element on which is placed Point 2





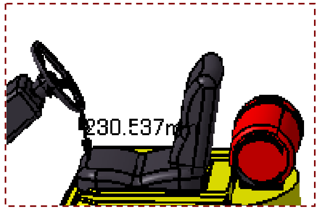
Student Notes:

Updating Measure

You will update measure by changing selection

1. In the tree, Double Click the measure to update
   The preview window and the Edit Distance window appear

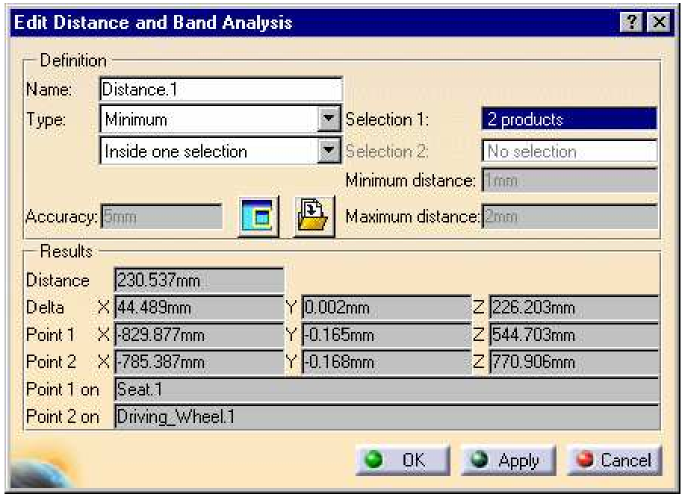
2. **Optional:** Modify values or selection

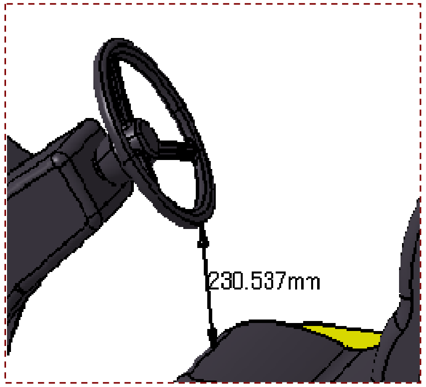


3. Click on Apply to re-compute

4. Click OK to confirm

Measure before update





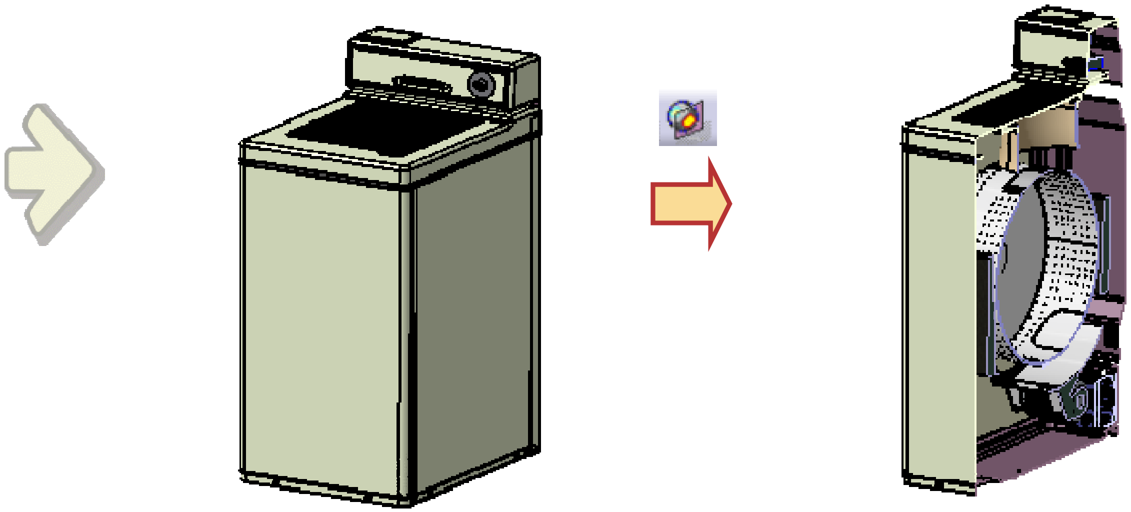
Measure after update

Click on [icon] to view the results in a separate viewer Object if it's not automatic

Student Notes:

Sectioning

You will become familiar with Sectioning tools in Assembly Design



Student Notes:

Sectioning

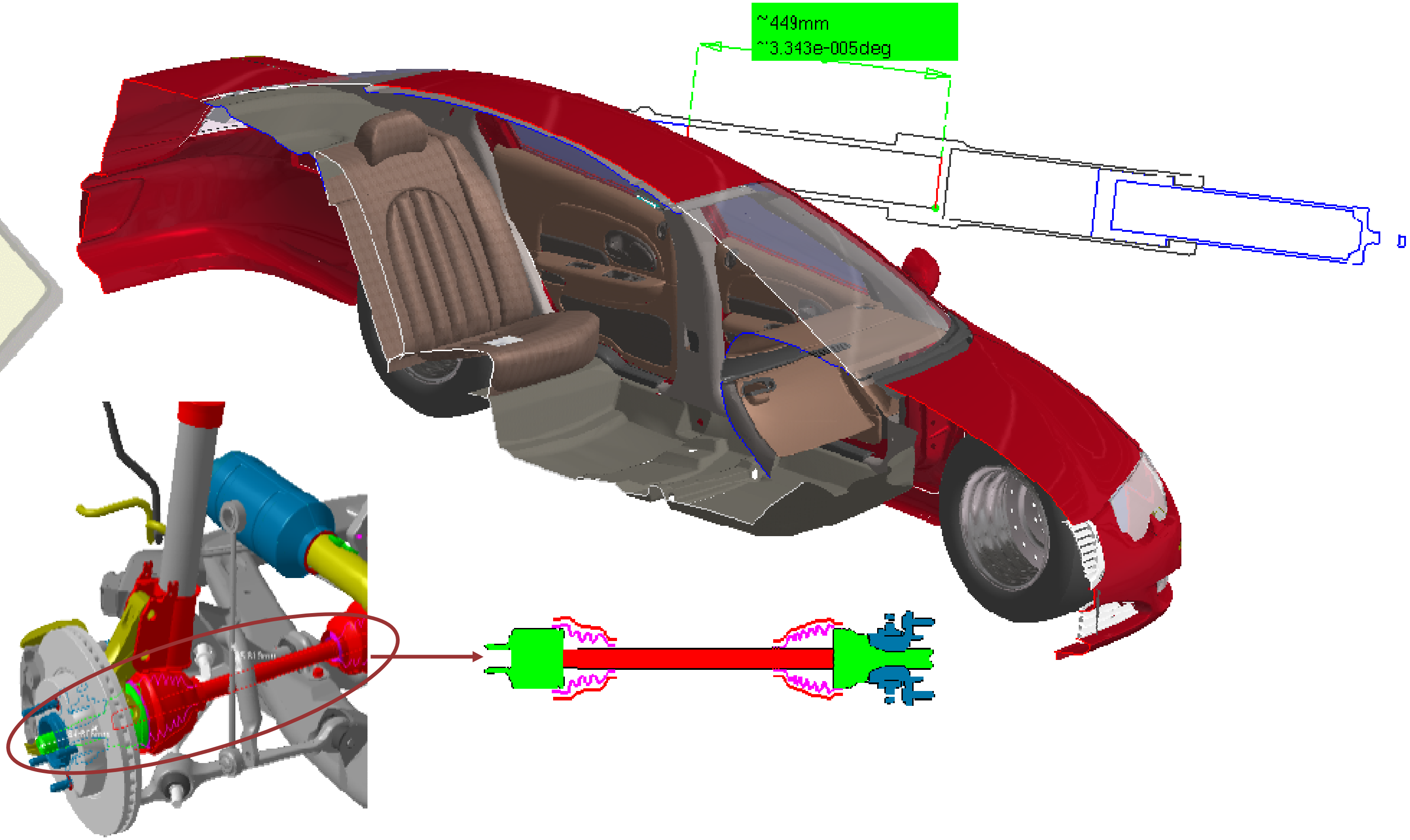
You will see how to create sections, position section plane and use section viewer to see assembly sections

- Introduction to Sectioning
- Creating Sections
- Positioning Main Section Plane
- Using the Section Viewer

Student Notes:

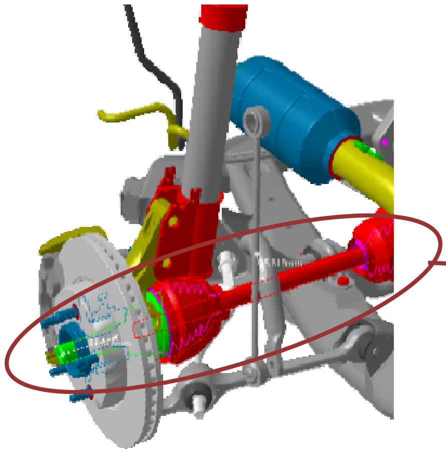
Introduction to Sectioning

You will learn how to create sections and observe internal details of components and assemblies.



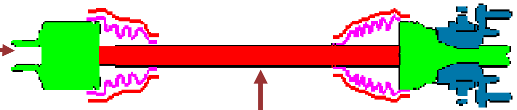
Student Notes:

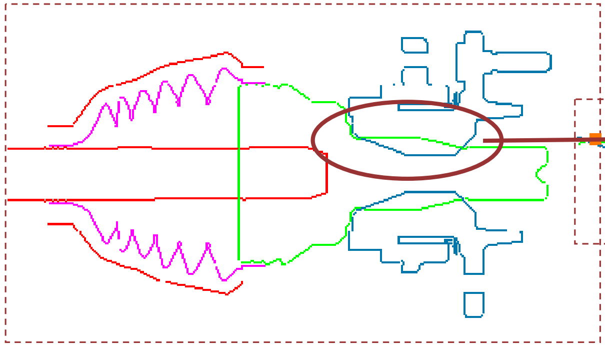
About Sections

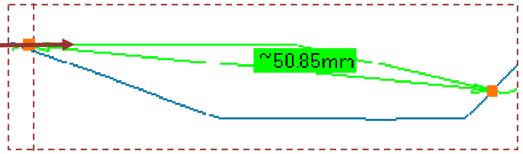


Identify conflicts

You can create Sections in the Clash Window



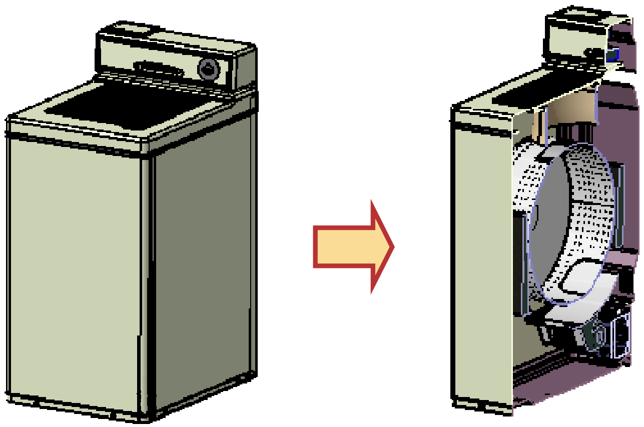




Check Distances

Line segments represent the intersection of the plane with all surfaces and volumes in the selection. By default, line segments are the same color as the products sectioned.

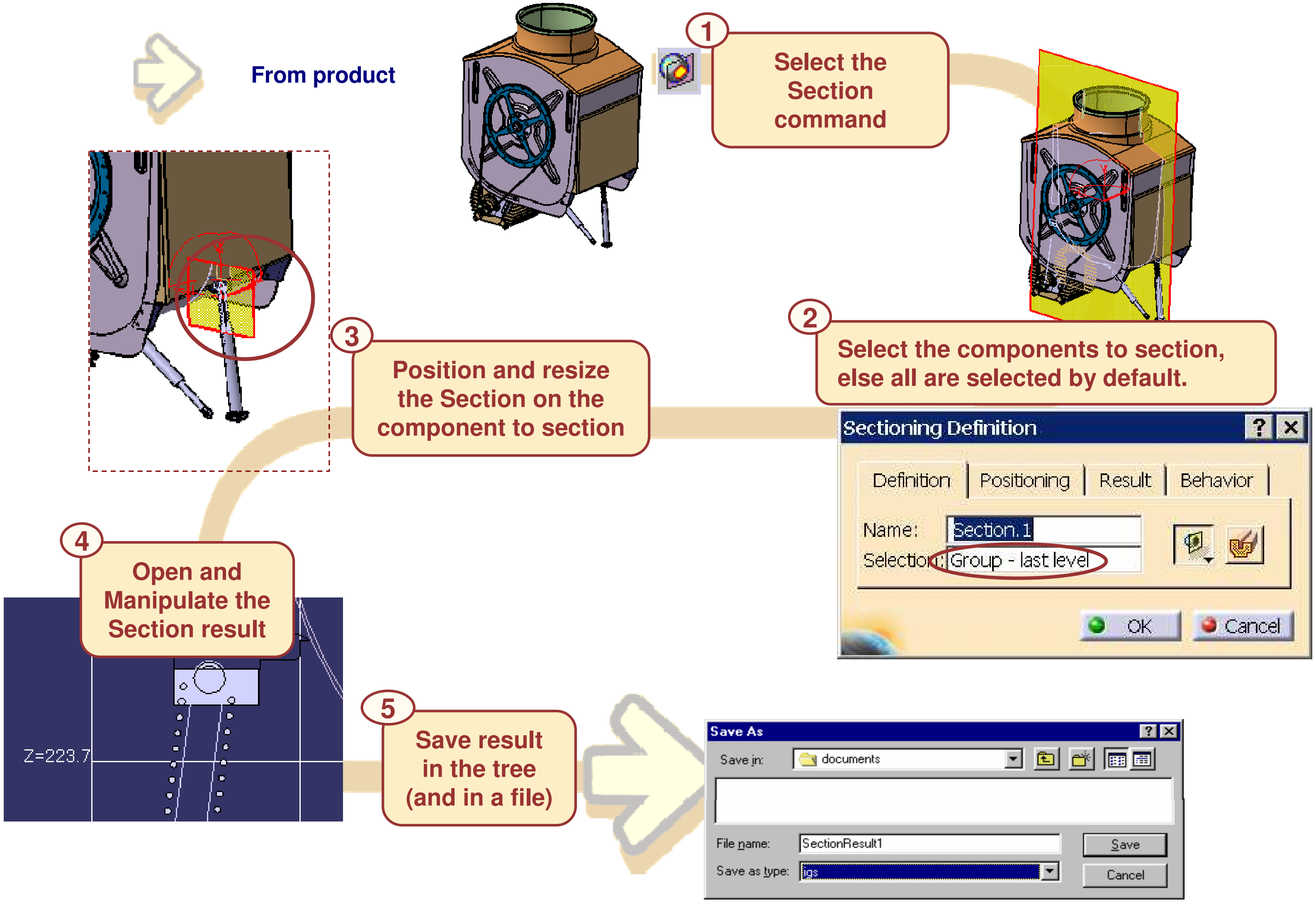
Points represent the intersection of the plane with any wire frame elements in the selection.



See what is inside Components using 3D Cuts
Check if your components are empty or not.

Student Notes:

General Process: Sectioning

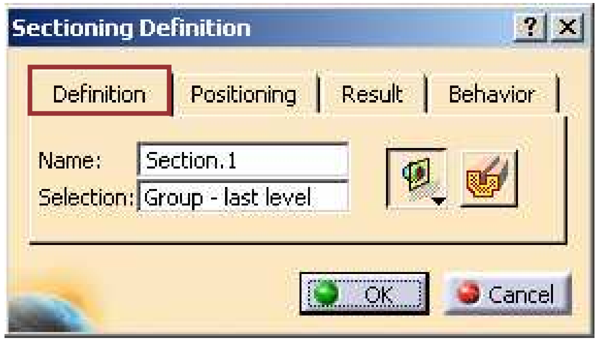


Student Notes:

How to Use Section Tools? (1/4)

You will see how to define a section view using various section tools.

1. Click on Sectioning Icon 
2. By default Definition tab is activated
3. Select Section mode

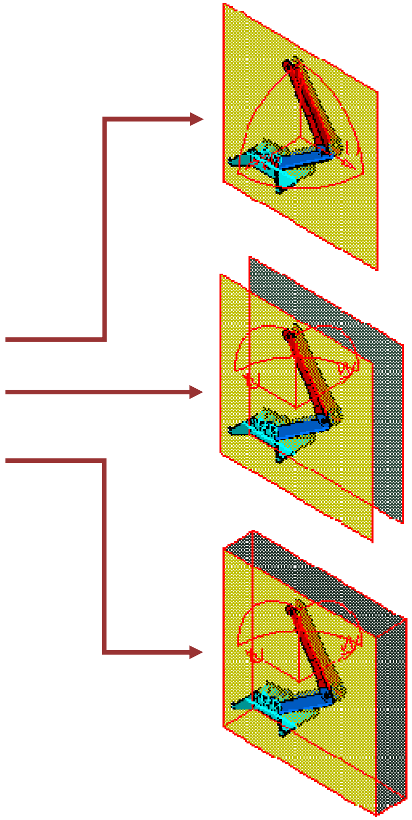


Section Mode:

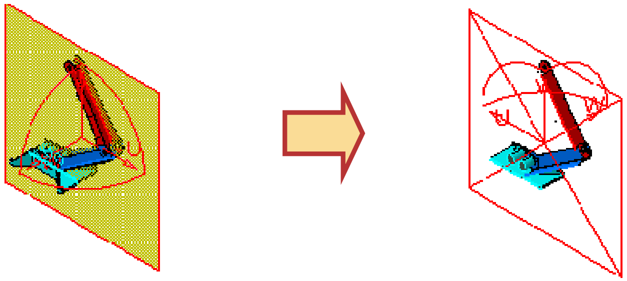
section plane

section slice

section box



4. Select Volume Cut 

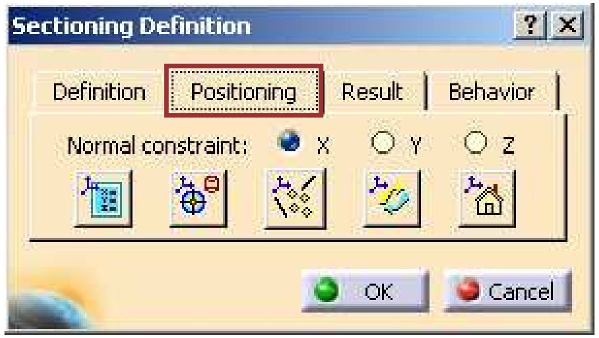


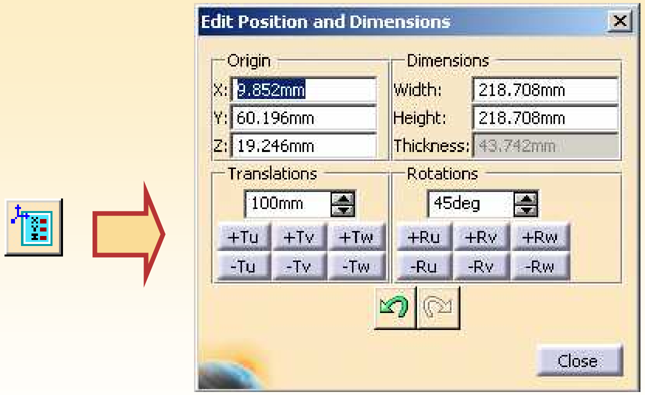
Before applying Volume Cut **After applying Volume Cut**

Student Notes:

How to Use Section Tools? (2/4)

5 Click on Positioning Tab and position the section plane using various positioning tools

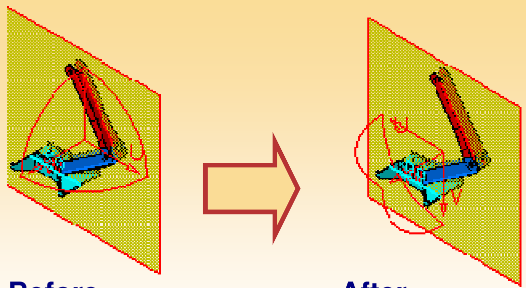




Edit Plane Window is used for Precise positioning of section plane

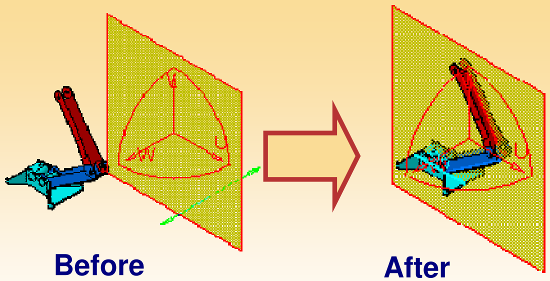
 Place your Section plane on a Geometric Target

 Allows you to position the section plane according to 2 or 3 selections



Before After

 Invert Normal direction



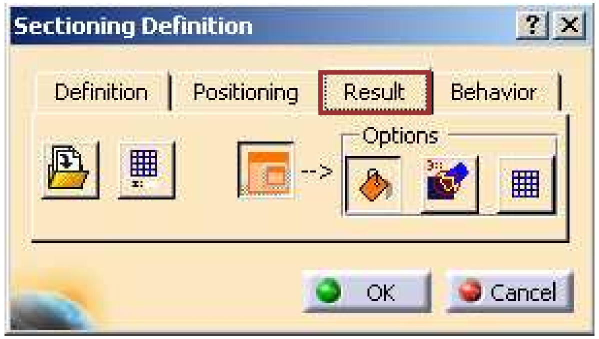
Before After

 Cancel Move

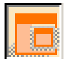
Student Notes:

How to Use Section Tools? (3/4)

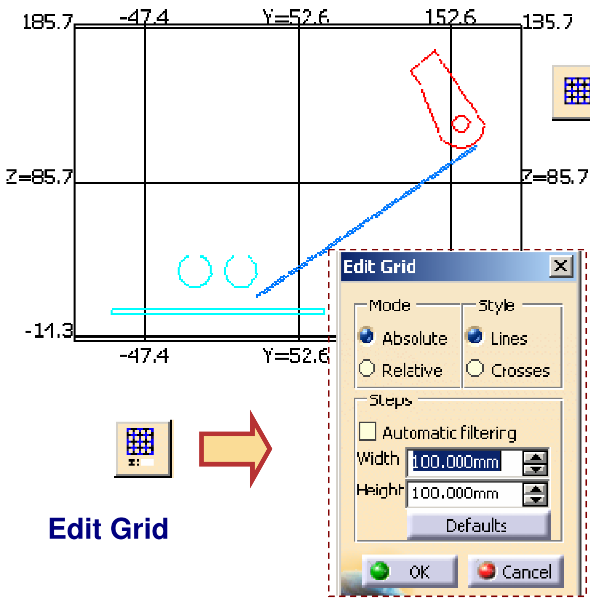
6 Click on Result Tab and export the result with various options using the tools in the Result tab



 Export Section View

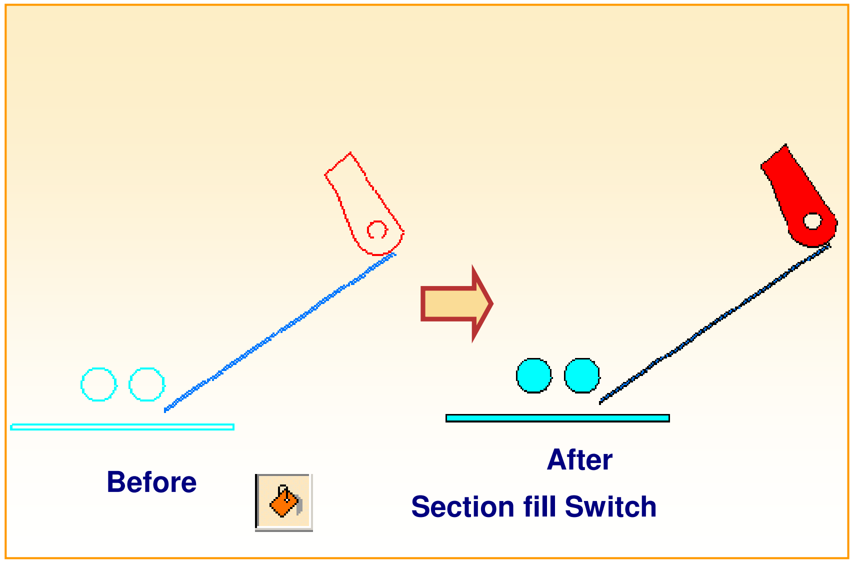
 Result window

 Clash detection in the Section Viewer



Grid Switch

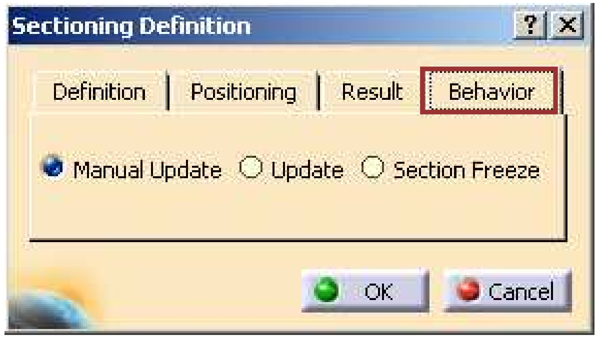
Edit Grid



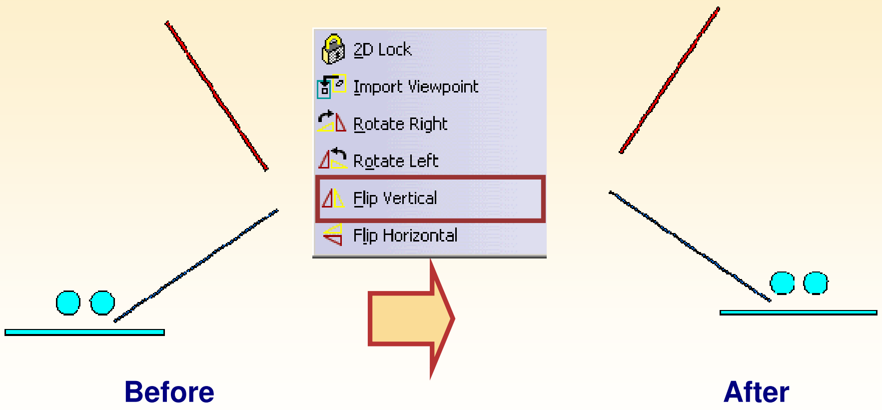
Student Notes:

How to Use Section Tools? (4/4)

7 Click on Behavior Tab and set the behavior of update



8 Click OK to validate

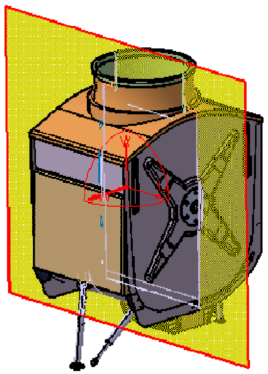


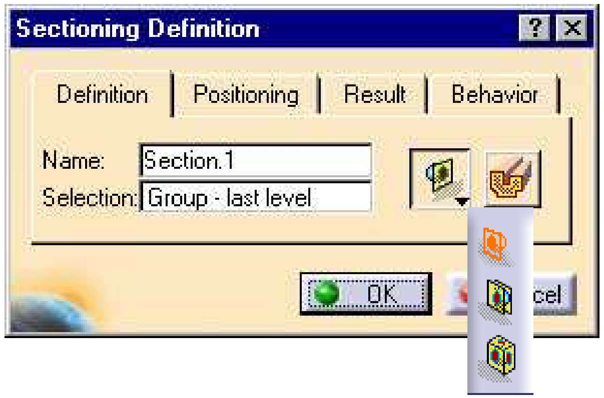
Contextual Menu available in the Section Results Window

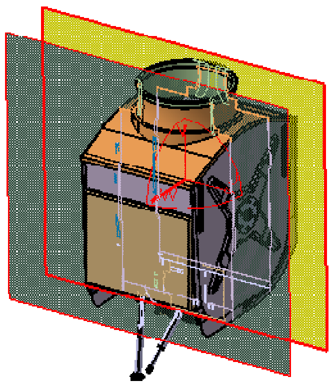
Student Notes:

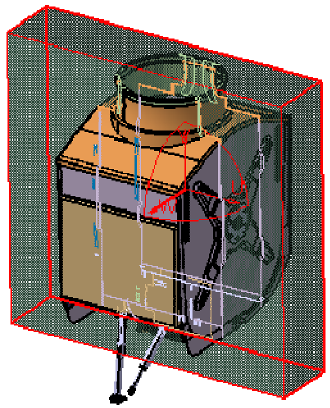
Creating Sections

You will learn the different ways to Section, and how to activate six cutting planes simultaneously







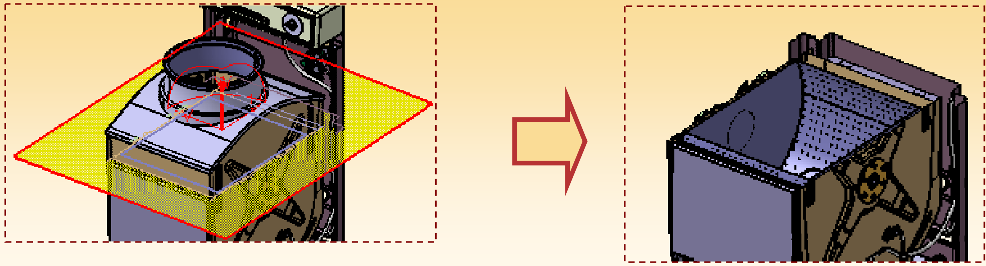


Student Notes:

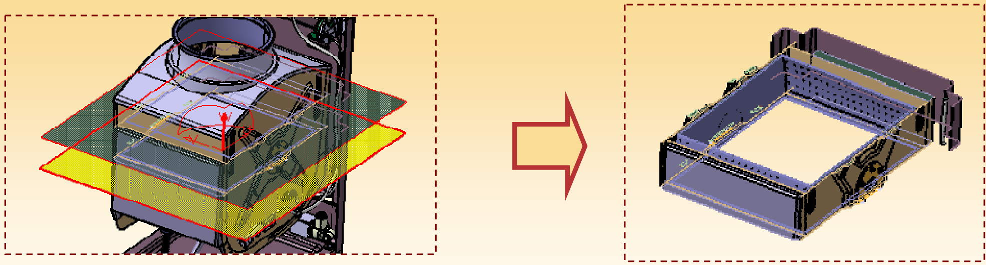
About Section Creation

There are various kind of sections from which you can get section cuts.

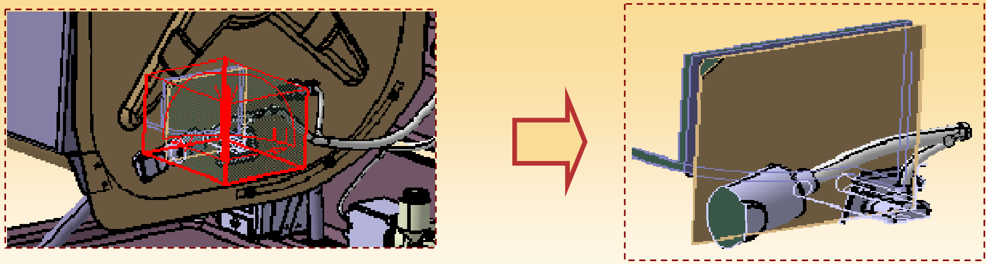
- **Section Plane**
- **Section Slice**
- **Section Box**



Section Plane



Section Slice



Section Box

Student Notes:

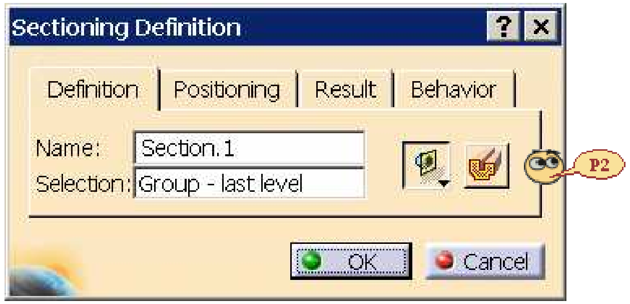
How to Create a Section

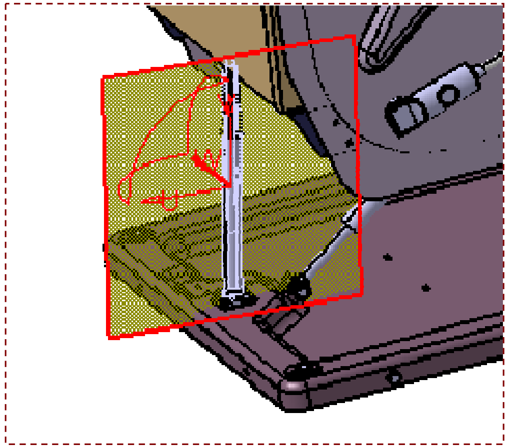
You will learn how to create a section

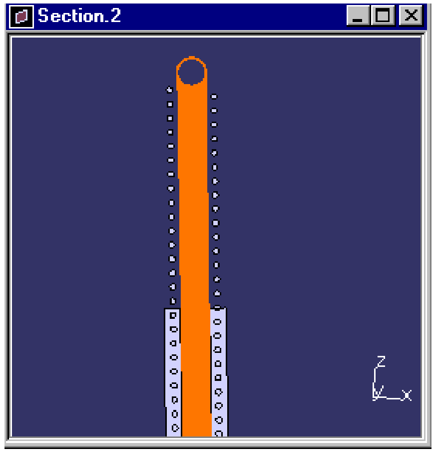
1. Click on the Sectioning icon. 

2. Select the kind of Section you want to perform

3. Position the section plane







Student Notes:

Creating a Section Plane

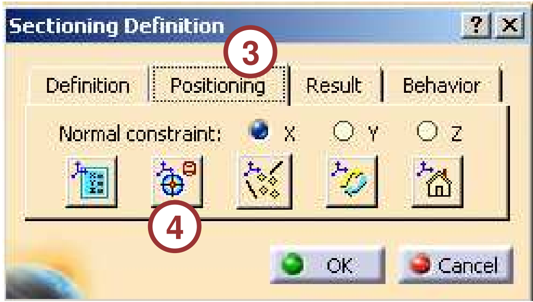
Create dynamically a section using a cutting plane

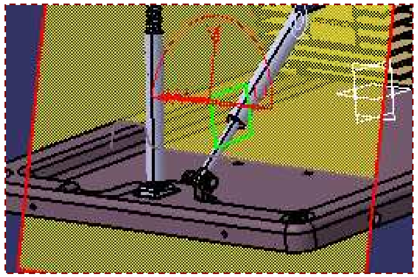
1 Choose the Products to section
If no selection made, all components will be sectioned
You can select or de-select products dynamically, during the section action, to take them into account or not. The plane created will cut only selected products

2 Click on 

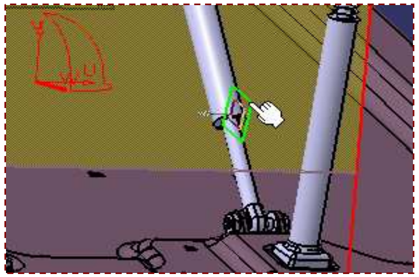
3 Select the “Positioning” tab and manipulate your section plane.

4 Select Geometrical Target

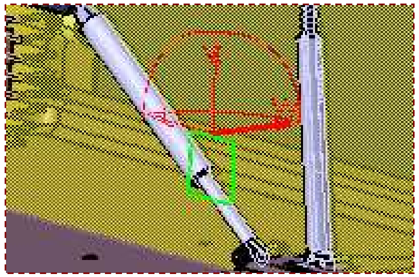




Initial Plane



Changing Plane



Different location and orientation

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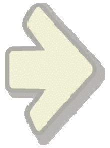
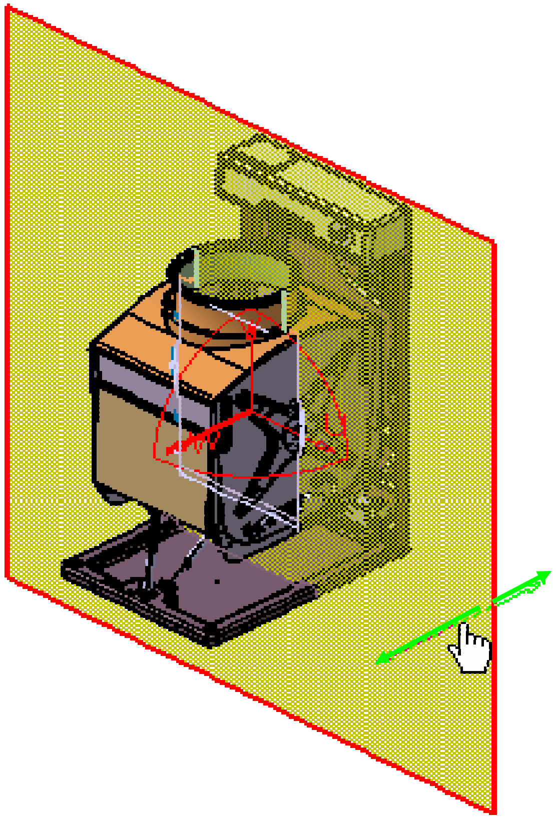
Student Notes:

Positioning Main Section Plane

You will learn how to position plane using edge as support and how to dimension main section plane



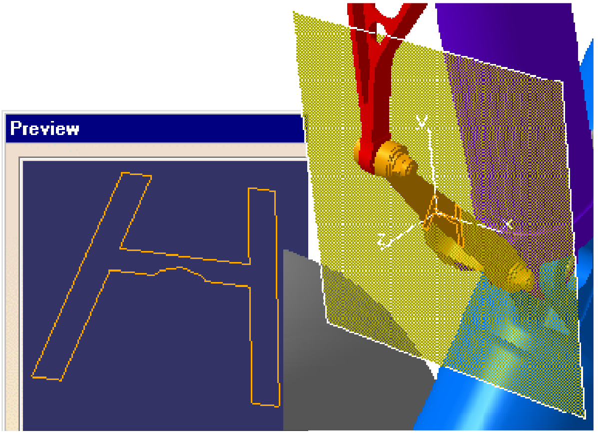


Student Notes:

About Section Manipulation

By default, the section plane is :

- **centered on the surrounding box center of the pre selected elements,**
- **oriented by the XY plane**
- **square shaped**
- **dimensioned according to the longest dimension between center of inertia and the furthest element**



Most of the time the section plane is not at the right position or size. You will have to:

- **re – center it**
- **translate and rotate it**
- **re dimension it**

Student Notes:

How to Manipulate a Section

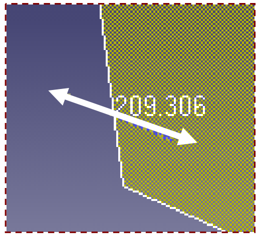
You can manipulate the section plane using the following options

Re dimension the plane using

- Direct Manipulation
- Edit Position and Dimension command

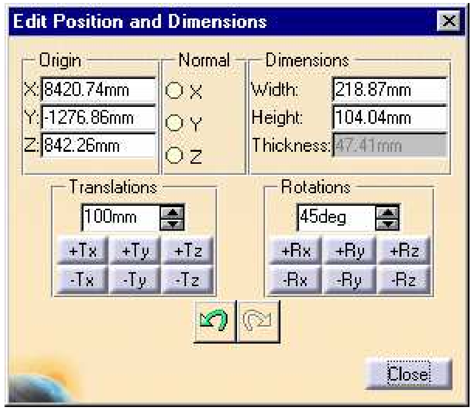
Move the plane using

- Direct Manipulation
- Geometric Target
- Edit Position and Dimension command

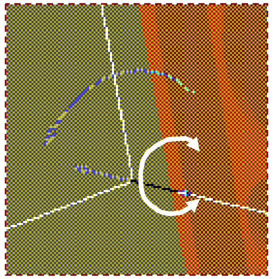


Re-dimensioning

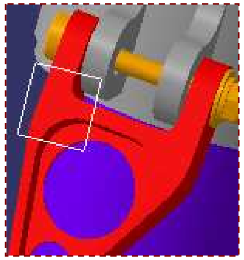
Direct Manipulation



Edit Position and Dimension



Rotating



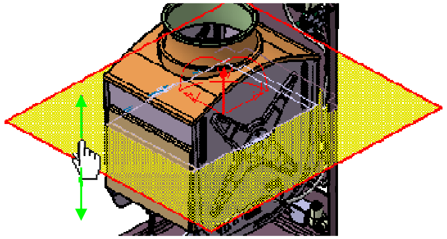
Using Geometric Target

Student Notes:

Manipulating a Section Plane

Re-dimension, move and rotate section planes

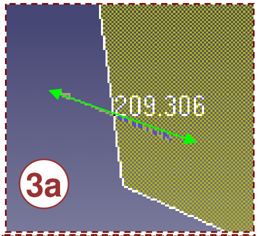
The appearance will change and arrows appear to help you moving your section plane



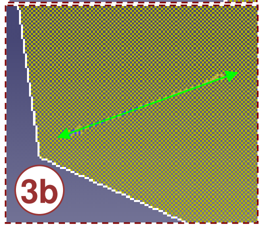
1 Edit the desired section plane by double clicking it

2 Move the cursor over the section plane, section plane edge or local axis system

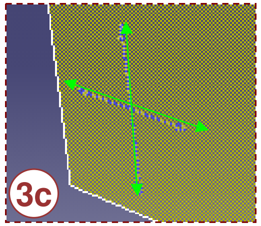
3a Re-dimension the section plane clicking and dragging plane edges



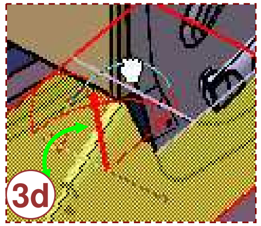
3b Move the section plane along its normal vector clicking over the plane and drag



3c Translate the plane by pressing MB1, then MB2 and dragging



3d Rotate the section plane clicking over the desired plane axis system and dragging

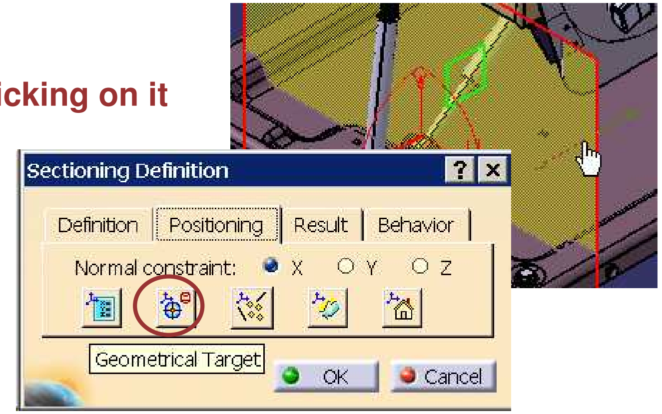


Use the Reset Position icon to restore the plane to its original position

Student Notes:

Positioning Plane with respect to a Geometrical Target

1 Edit the desired section plane by double clicking on it



2 Select the "Positioning" tab and click on Geometrical Target icon

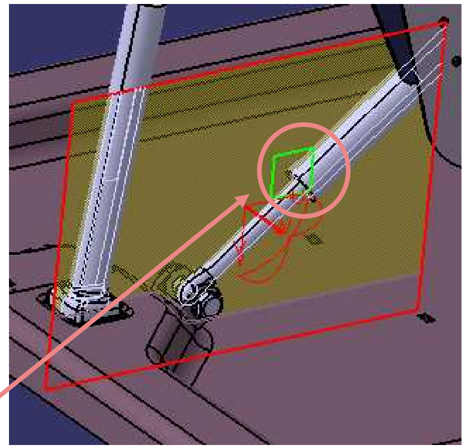
3 Point to the target of interest :

Simply Click an Edge or an Axis to position the section plane normal to the desired edge

Simply Click a Surface to position the section plane on the tangent to a surface

Simply Click a Plane to position the section plane coplanar to this plane

Simply Click on a Cylindrical Surface to position the Section Plane normal to the Axis



The Plane is a representation of the section plane, to assist you in positioning it

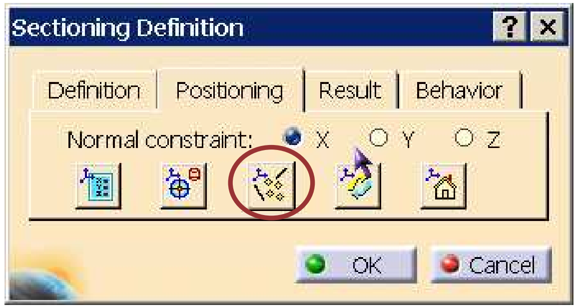
Reset Position icon :
Restore the plane to its original position

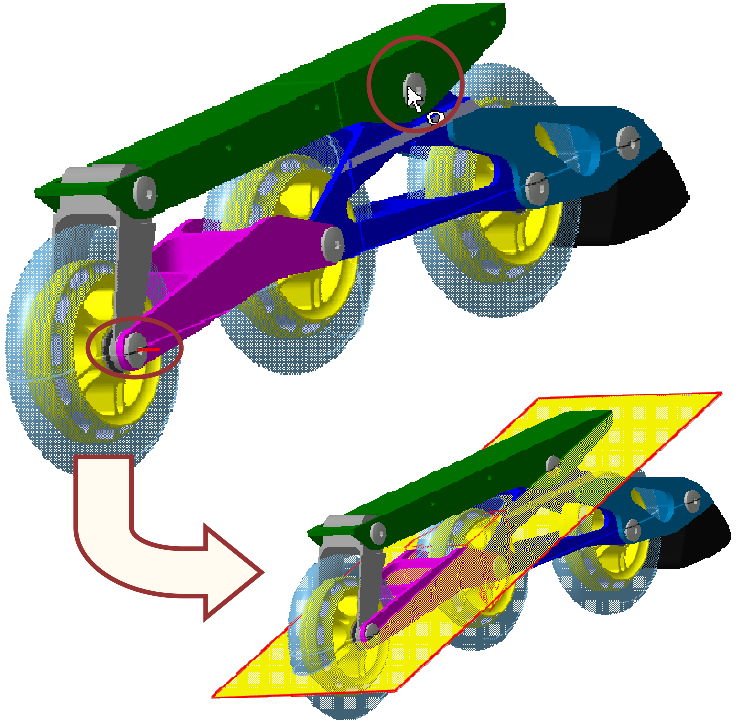
Student Notes:

Positioning a Section Plane by Points and/or Lines

Thanks to this tool, you can quickly position a section plane through specific entities. You just need to select some lines or points and the section plane will automatically cross the selected entities

1





2

Make your selection of points and/or lines in the geometry. To define a plane you have to select either 1 line and 1 point or 2 lines or 3 points

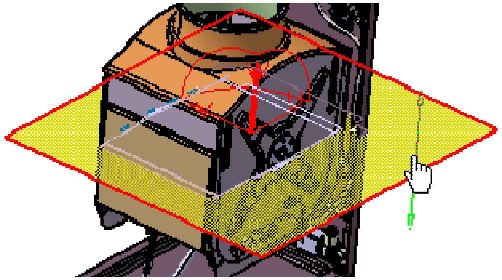
The cursor changes to assist you make your selection. It identifies the type of item (point, line, cylinder, cone, etc.) beneath it.

Student Notes:

Positioning Plane Using the Edit Position Command (1/2)

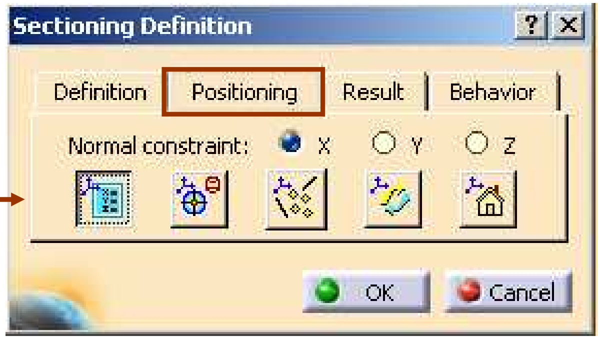
You will learn how to position the section plane more precisely.

1 Edit the desired section plane by double clicking it



2 Select the “Positioning” tab and

3 Click on “Edit Position”



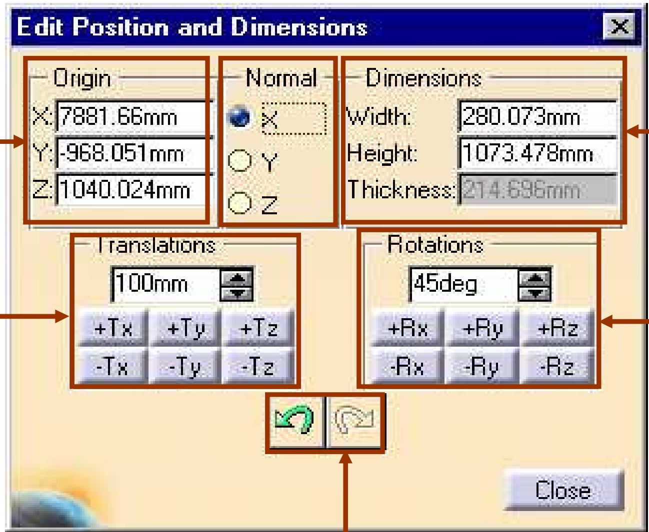
Positioning Plane Using the Edit Position Command (2/2)

(4) Fill "Edit Position" Dialog Box

Change the current position :
Click X, Y or Z to position the normal vector (z-axis) of the plane perpendicular to the selected absolute axis system

Change the plane / slice / box dimensions
Enter the width of the main section plane
Enter the height of the main section plane
Enter thickness of the box or the slice (distance between parallel cutting planes)

Change the center of the plane coordinates :
Values in Origin X, Y and Z boxes (absolute system coordinates)



Move the section plane :
Enter translation step, then click +Tx, -Ty, +Tz … to move the plane along the selected axis by the defined step (local plane axis system)

Rotate the section plane :
Same as for the translation use, to rotate the plane around the selected axis by the defined step

Undo / Redo section plane move

(5) Click close to exit and save last plane position

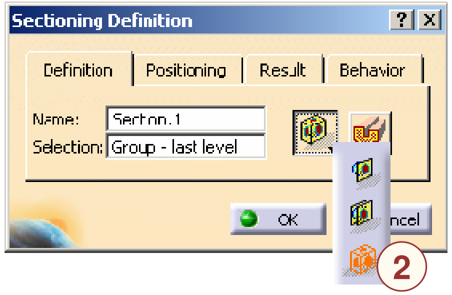
Student Notes:

Student Notes:

Using Smart Target to Snap Section Box

You will use Smart Target for Snapping section box

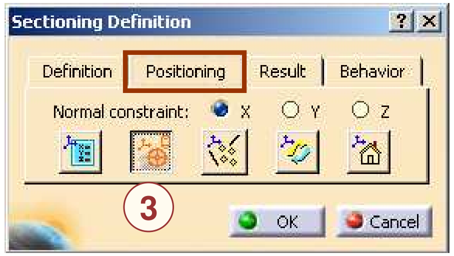
1 Click on Sectioning icon 



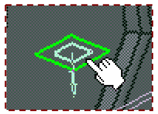
2 Select “Section box”

3 Select the “Positioning” tab and

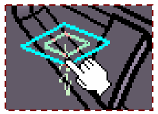
click on “Geometrical target” icon



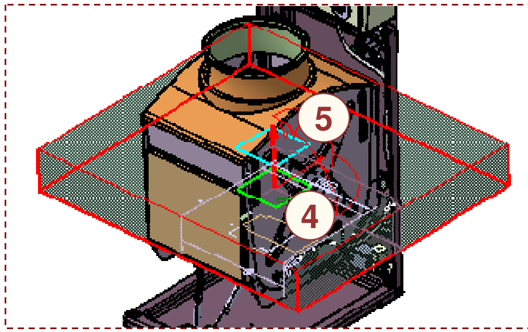
4 To snap the <u>first side</u> of the section box click on the plane desired of the geometry: this symbol appears



5 Do the same thing to snap the <u>second side</u>



Selected planes can be parallel or perpendicular

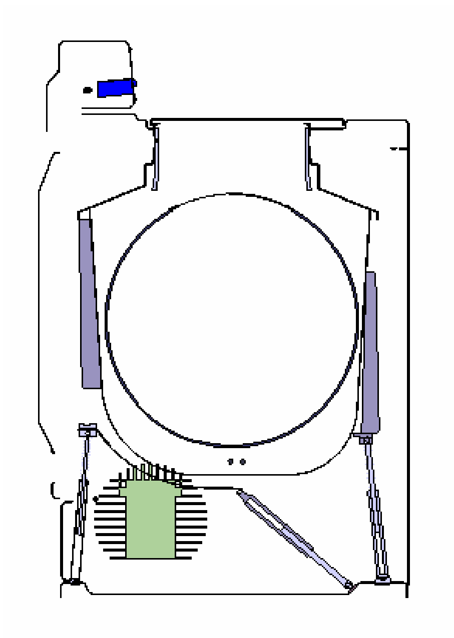


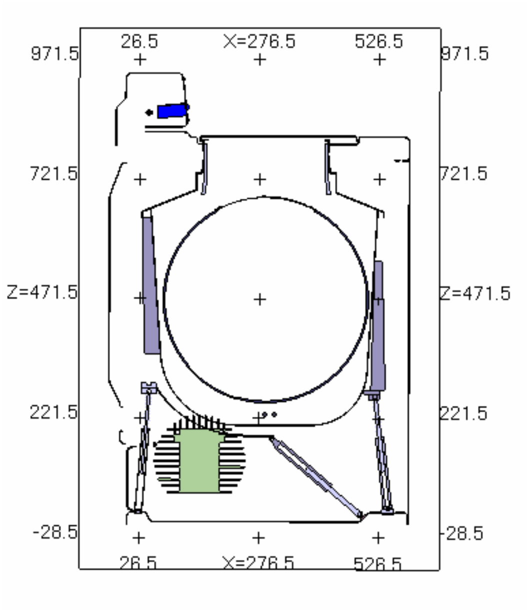
6 Click OK in the Sectioning Definition dialog box when done

Student Notes:

Using the Section Viewer

You will study the section viewer and various tools to customize the section viewer.

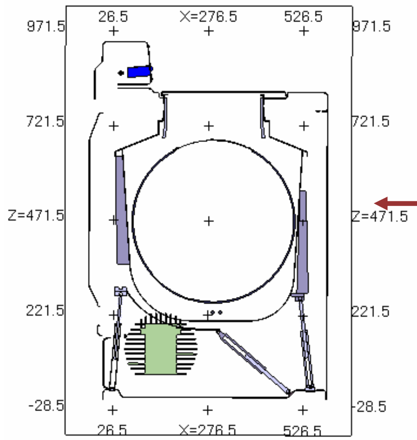




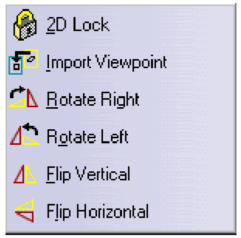
About Section Viewer

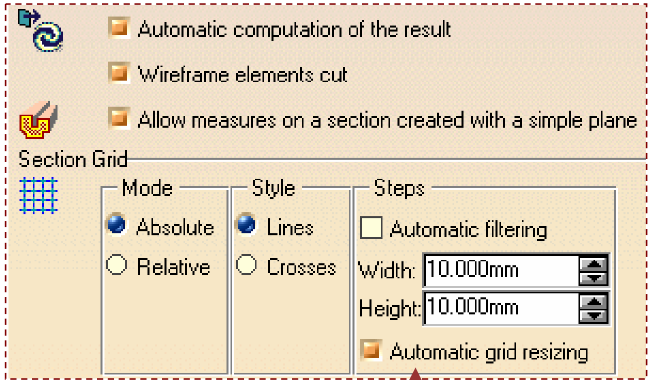
To display the result of the section, in a particular interface special tools are available to:

- **Fill Section**
- **Display Grid** P2
- **Modify Grid options**
- **Rotate section**
- **Lock 2D**

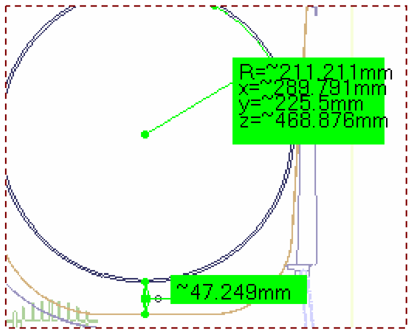


Viewer contextual menu





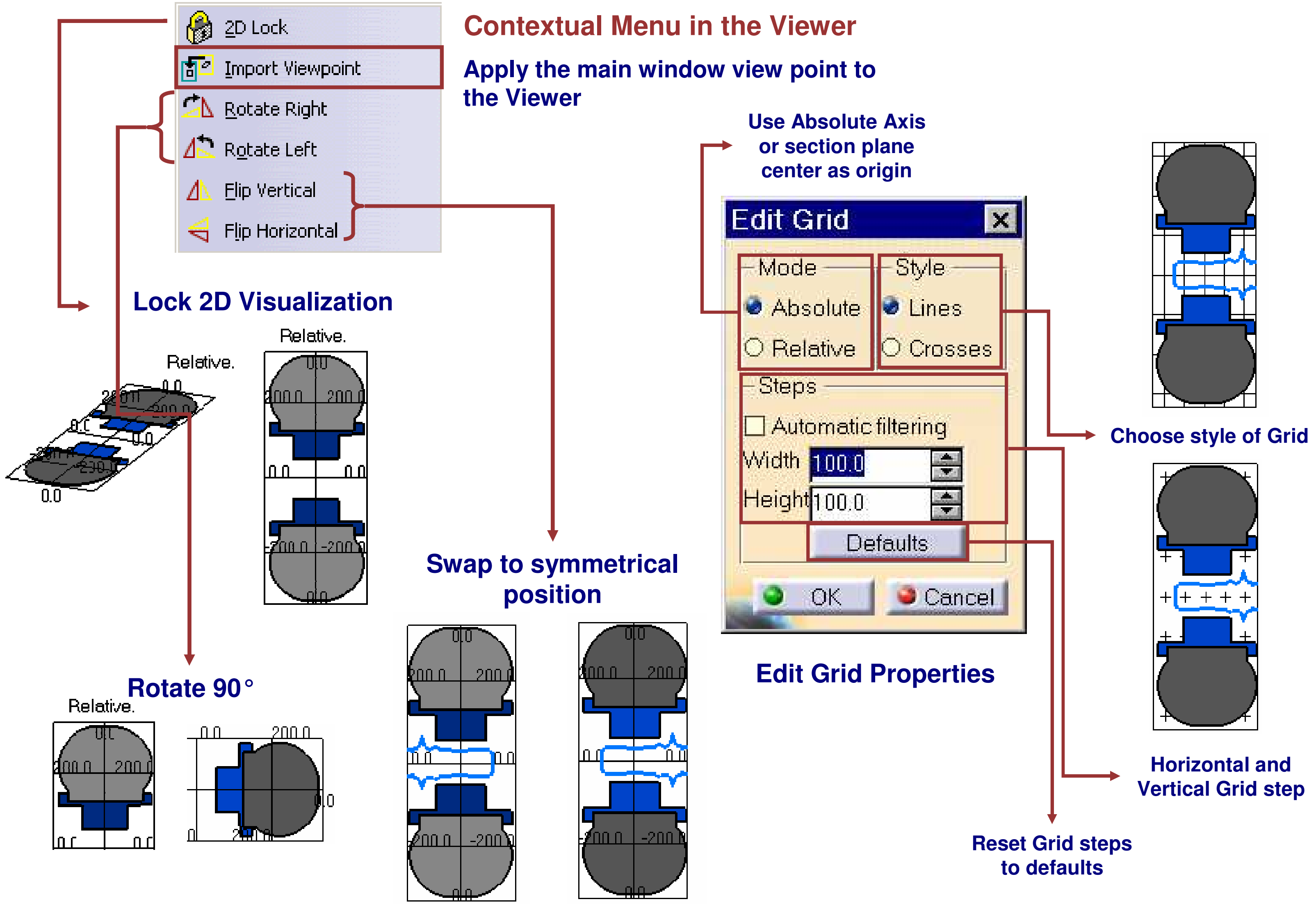
You can also add measures and annotations in the section view by enabling this option.



The grid can be automatically re-sized to section results when moving the section plane. De-activating this option means that the grid has the same dimensions as the section plane.

Student Notes:

How to use Viewer Tools?

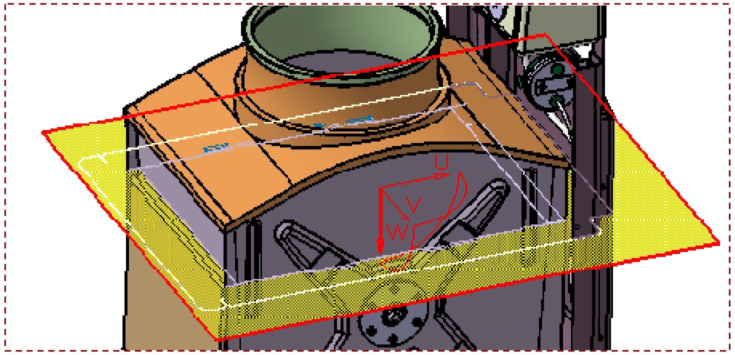


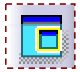
Student Notes:

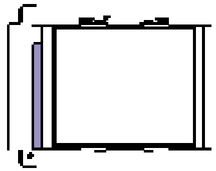
Viewing Section

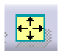
View the generated section in a separate 3D viewer

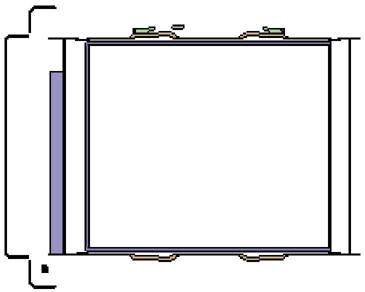
1 Edit the desired section plane



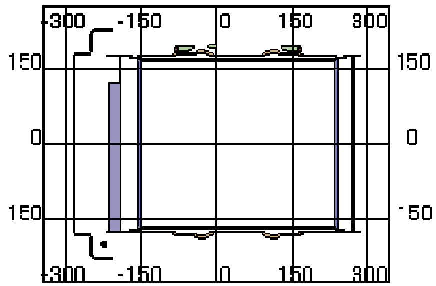
2 Click on Results Window icon 



3 Click on Fit All In icon 
to reframe the results window



4 Manipulate the Section



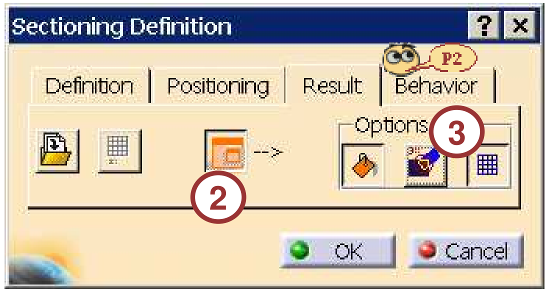
Student Notes:

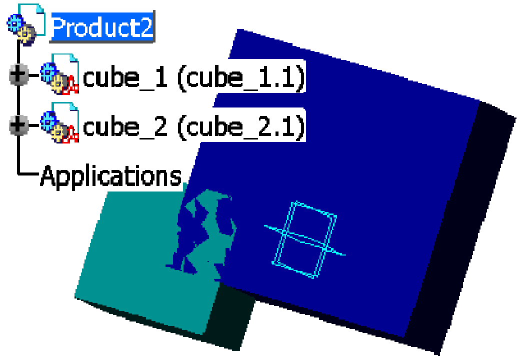
Checking Clash in Result Window

Detect collisions between 2D sections

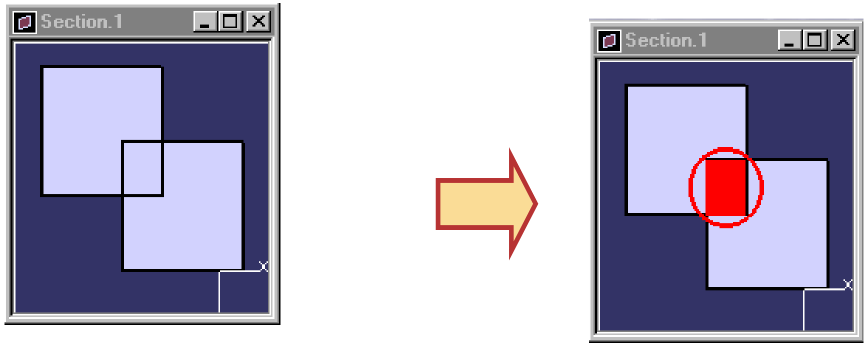
1 Create a Section

2 Open the Section Result window





3 Click on the Clash Detection icon



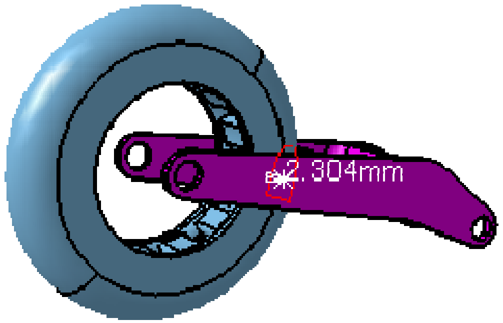
The section result must be filled to detect clash.

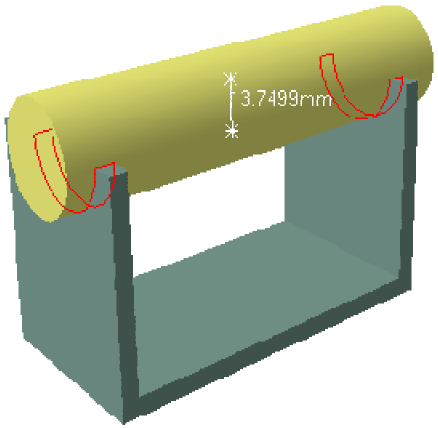
Clashes detected are highlighted

Student Notes:

Computing Clash

You will learn various ways to compute clashes in a digital mock-up.





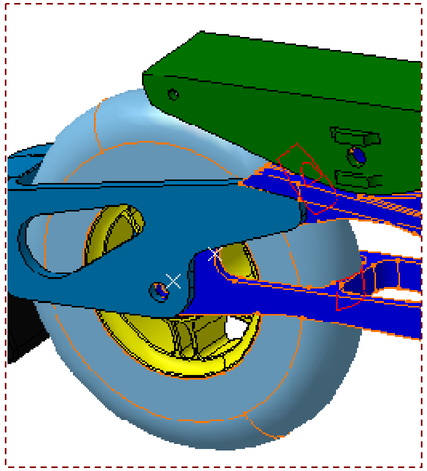
Student Notes:

What is Computing Clash (1/2)

The various ways to compute clash are shown below:

Part to Part Clash : It is the simplest tool to quickly detect clash between two components.

- **This command can be accessed from Analyze Menu.**
- **This command identifies the clash / clearance / contact between two selected parts.**
- **The result is given by the traffic lights:**
  - **Red : Clash is detected between selected components.**
  - **Yellow: Contact is detected between selected components**
  - **Green: No interference between selected components**



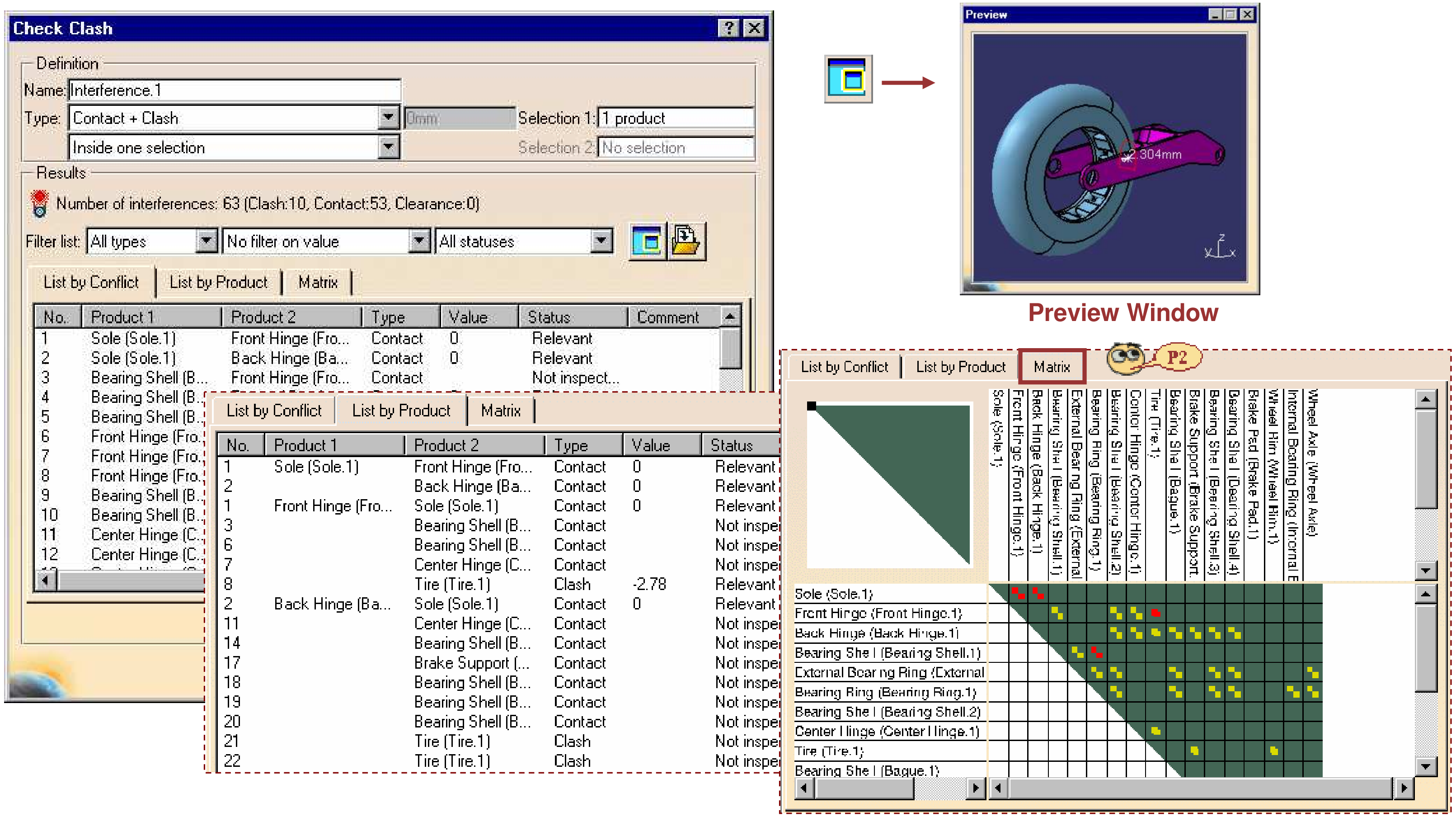
Clash computation using 'Part to Part Clash'

Student Notes:

What is Computing Clash (2/2)

Check Clash: It is powerful tool for clash detection and analysis.

- **This command can invoked by clicking on 'Clash' icon.** 
- **Check Clash window displays clash results for selected component(s).**

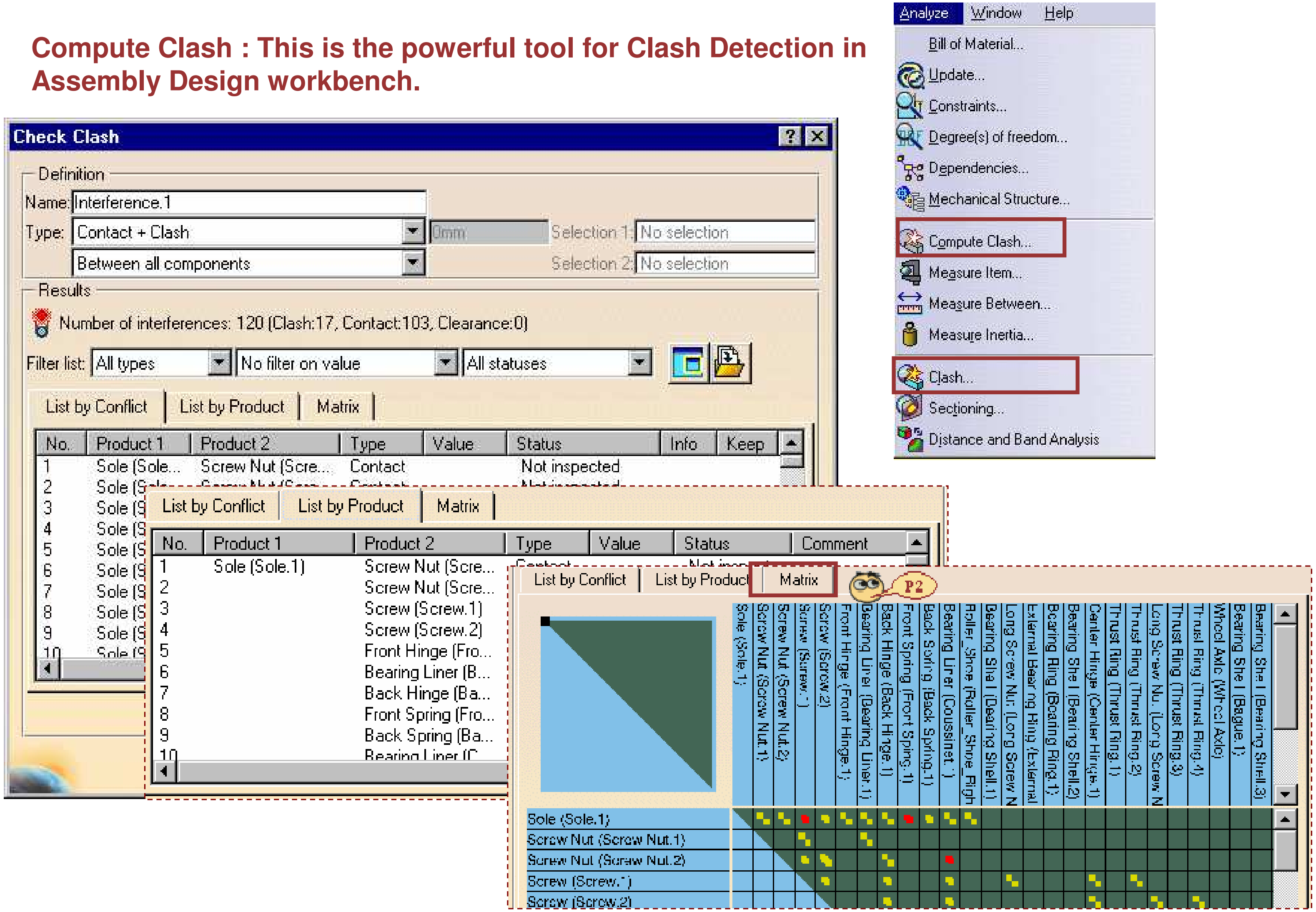


Preview Window

Student Notes:

How to Compute Clash?

Compute Clash : This is the powerful tool for Clash Detection in Assembly Design workbench.

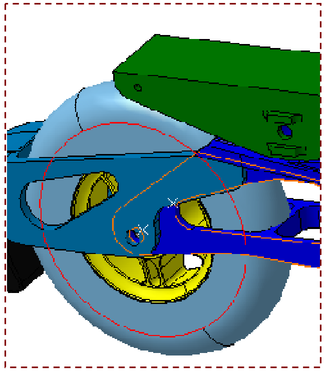


Student Notes:

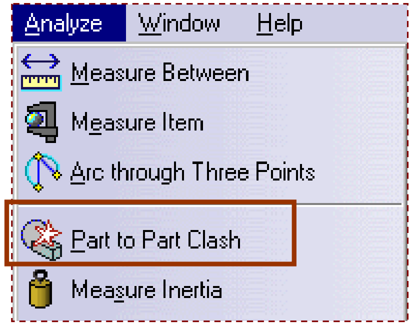
Analyze Part to Part Clashes

You can check for clashes and clearances between two parts in an assembly.

1 Select the two parts using <CTRL> key.



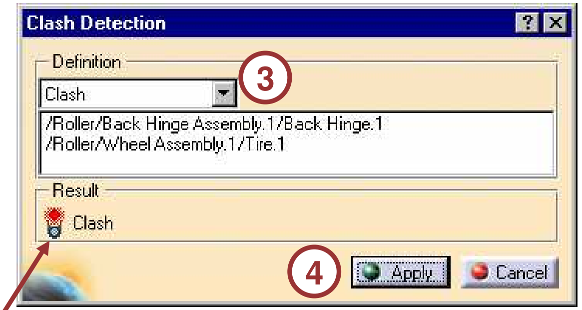
2 Activate ‘Part to Part Clash’.

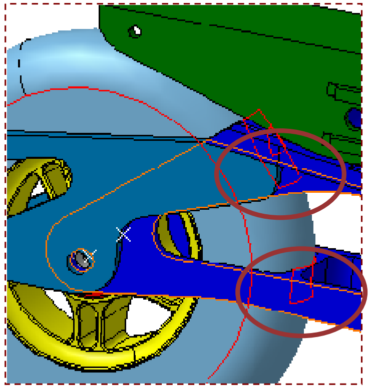


3 Choose the kind of Analyze to perform:

- Clash (Parts occupying same space or in contact)
- Clearance (Parts occupying same space zone, in contact, or separated by less than the defined clearance distance - enter the value of the clearance)

4 Click on ‘Apply’.



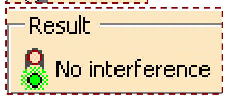


Clash detected between selected parts is highlighted in the mock up

5 Check results

Check light :

- Red for Clash
- Yellow for Contact
- Green for Clearance



Student Notes:

How to Fill 'Check Clash' Window?

Before Computing the Clash, you will have to fill the 'Check Clash' window.

1 Give a name to your measure

2 Choose Computation type

- Clash: components occupy same space zone
- Contact: components in contact
- Clearance: components separated by less than the defined clearance distance
- Authorized penetration: components in intersection by more than the defined value

3 Choose Selection type.

- Between all components : each component against all other products
- Inside one selection : within any one selection, tests each component of the selection against all other components in the same selection
- Between two selections : each component in the first selection against all components in the second selection
- Selection against all : each component in the defined selection against all other components in the document

4 Select the components (parts, products, groups,etc.) according to computation type

Student Notes:

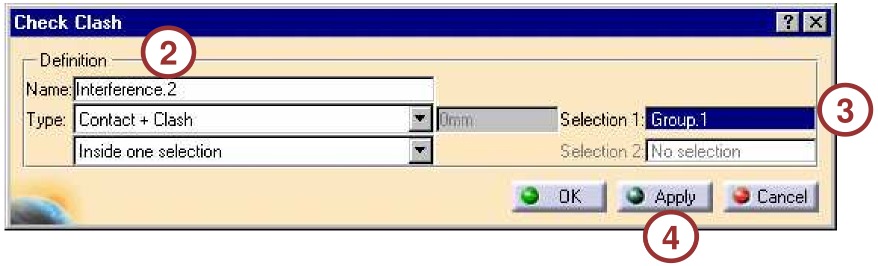
Computing Interferences

You will compute clash for an assembly

1 Click on Clash icon 

2 Fill the Check Clash Window

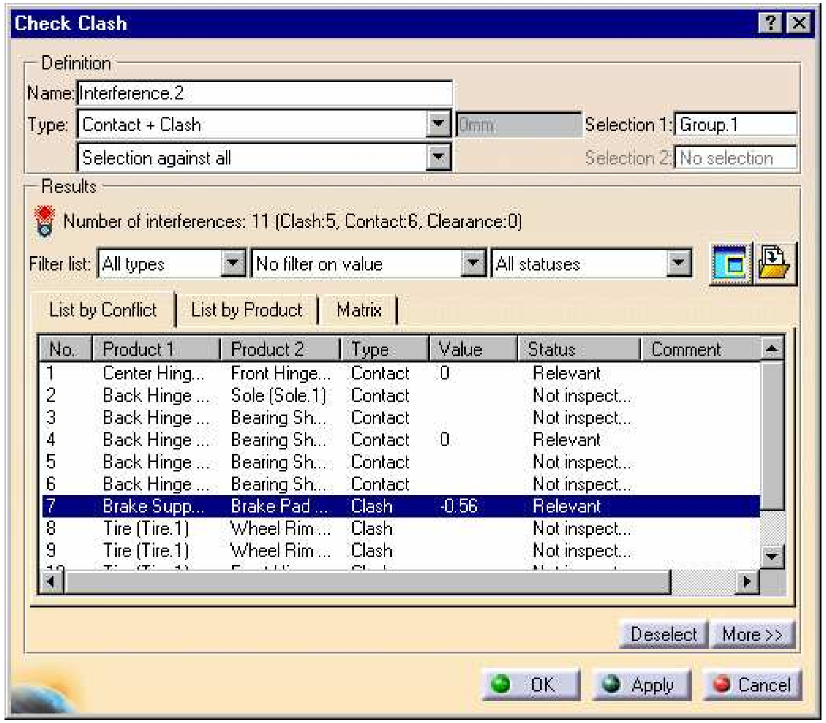
- Name of the computation
- Type of computation : Contact/ Clash/ Clearance/ Penetration
- Computation Type: Between all Components / Selection against all / Inside one Selection /Between two selections



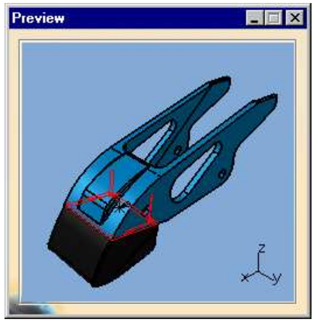
3 Select the components from the tree or 3D, according to options chosen in 'Step 2'.

4 Click apply to calculate the Clash

The Results are displayed in the 'Check Clash' window.



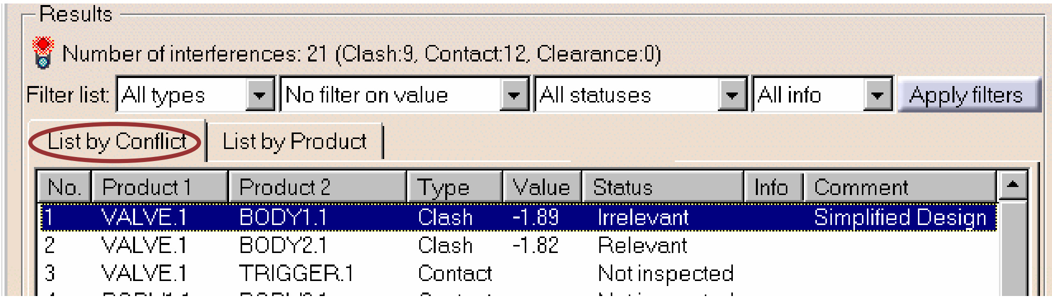
A Preview window displays the selected Clash



Student Notes:

Analyzing Interferences Basic List By Conflict

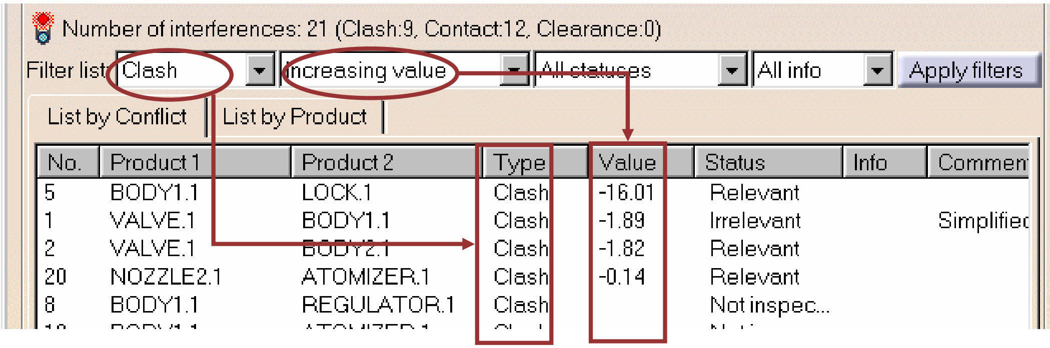
1 **Compute Clash**
Then Click on the results to be checked
- **The Clearance/ Clash value is displayed**
- **The Status changed from 'not inspected' to 'Relevant'**



2 **Click on the Status value to change it into Relevant or Irrelevant, If necessary**

3 **Click on the Comment of the selected result to add/modify the Comment**

4 **Filter Interferences to simplify result display, eventually**
Type: Clash/Contact/Clearance
Value: Increasing/Decreasing
Status: Irrelevant/Relevant/Not Inspected
Compute status : New / Old / Modified



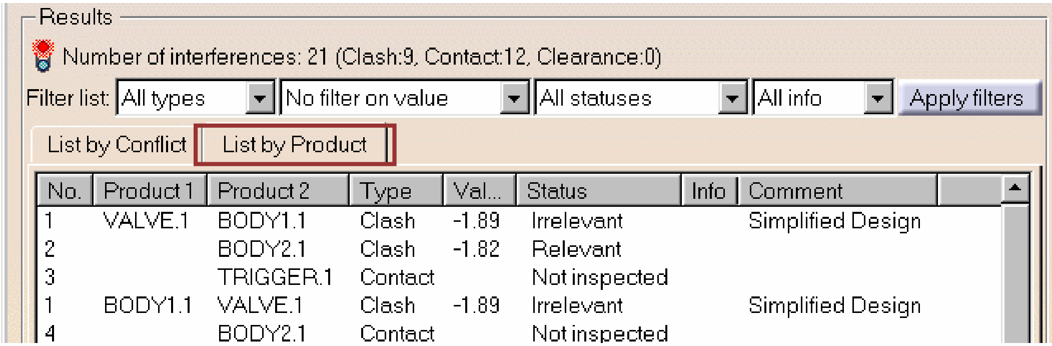
5 **Then Click Apply Filters**

6 **Click OK to Exit**

Analyzing Interferences Basic List By Product

1 **Compute Clash**
Then Click on the results to be checked
- **The Clearance/ Clash value is displayed**
- **The Status changed from ‘not inspected’ to ‘Relevant’**

2 **Click on « List by product » Tab to Display Clashes Product by Product**



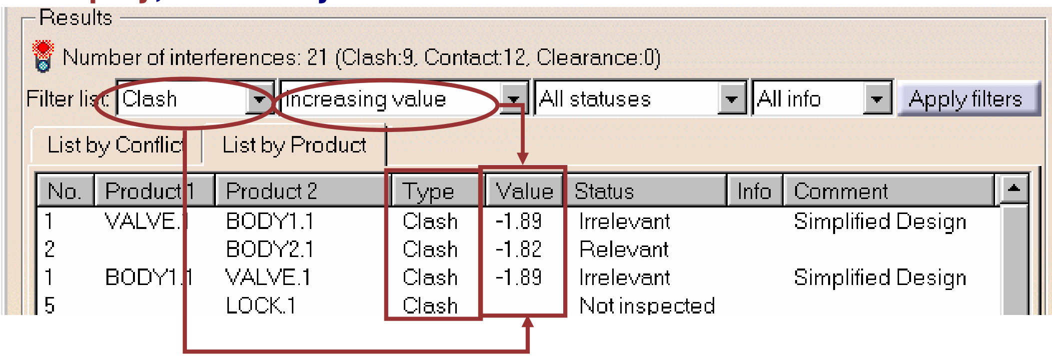
3 **Click on the Status value to change it into Relevant or Irrelevant, If necessary**

4 **Click on the Comment of the selected result to add/modify the Comment**

5 **Filter Interferences to simplify result display, eventually**
Type: Clash/Contact/Clearance
Value: Increasing/Decreasing
Status: Irrelevant/Relevant/Not Inspected
Compute status : New / Old / Modified

6 **Then Click Apply Filters**

7 **Click on OK to Exit**



Student Notes:

Student Notes:

To Sum Up ...

This concludes the sub lesson “Measuring , Sectioning and Clash”. You should be able to demonstrate:

- Measuring a minimum distance
- Creating sections
  - Simple section
  - Multiple sections
  - Positioning section planes
- Computing clashes
  - In the assembly
  - Between parts

Student Notes:

Managing Scenes

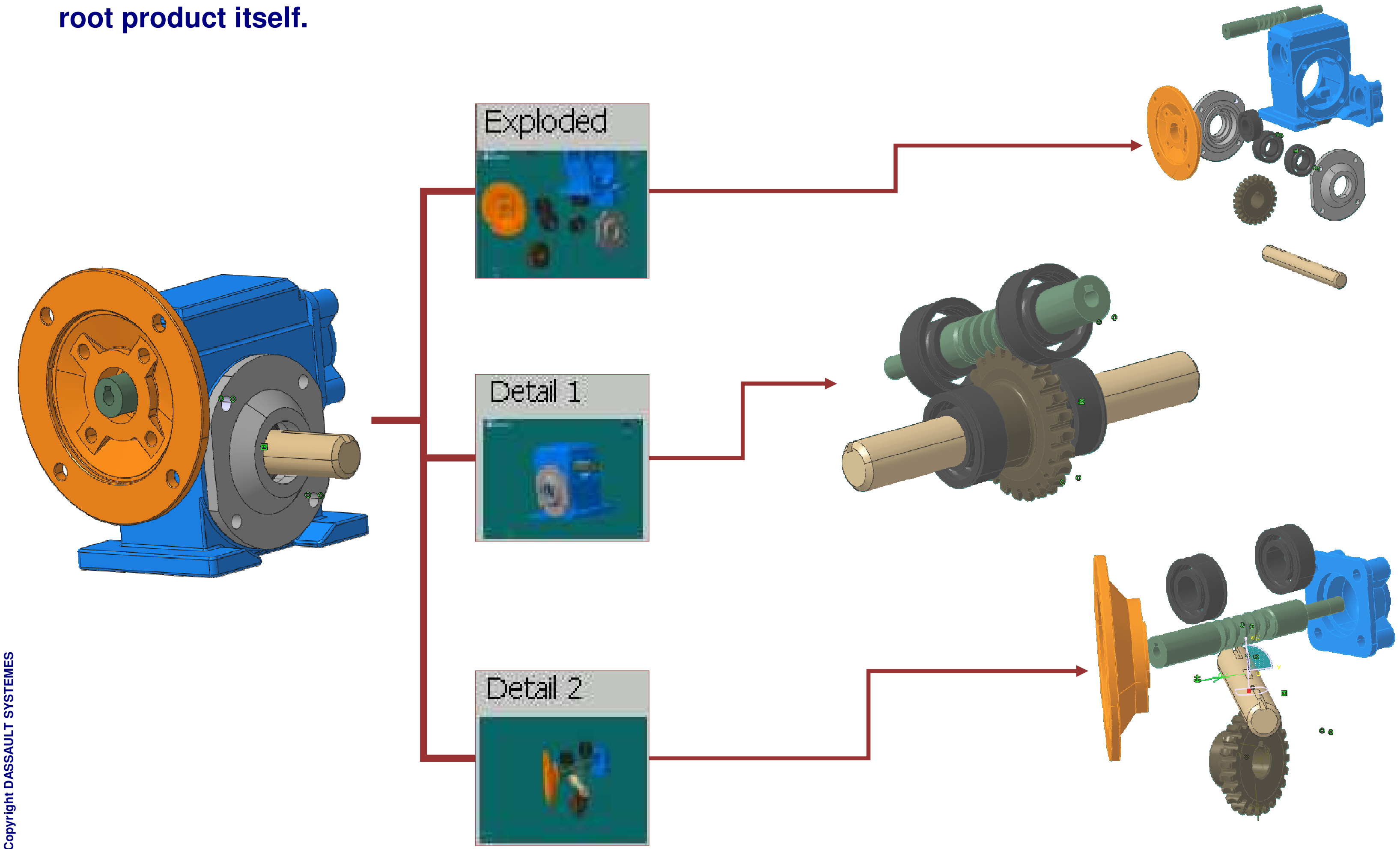
In this lesson you will learn how to create, edit and manage Enhanced Scenes in Assembly document.

- Introduction to Lesson
- Introduction to Enhanced Scenes
- Creating Enhanced Scenes
- Editing Enhanced Scenes
- Managing Components in Enhanced Scenes
- Creating Explode in Enhanced Scenes
- Creating Drafting Views based on Scenes
- Creating and Using Scenes: Recap Exercise
- To Sum Up

Student Notes:

Introduction to Lesson

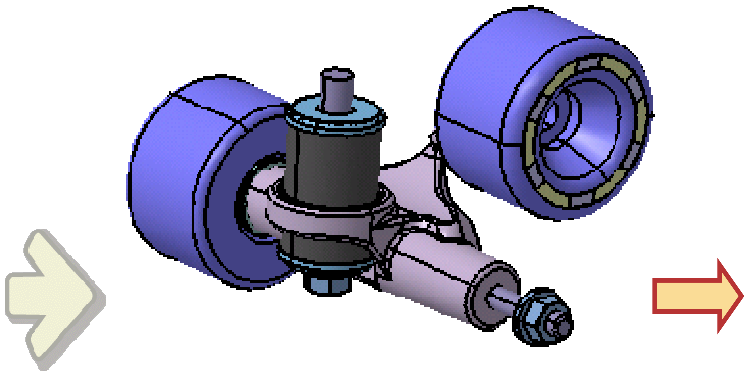
A way to different point of view of a CATProduct is the use of “Scenes”.
This tool permit to have multiple “screenshot” of an assembly without modifying the root product itself.

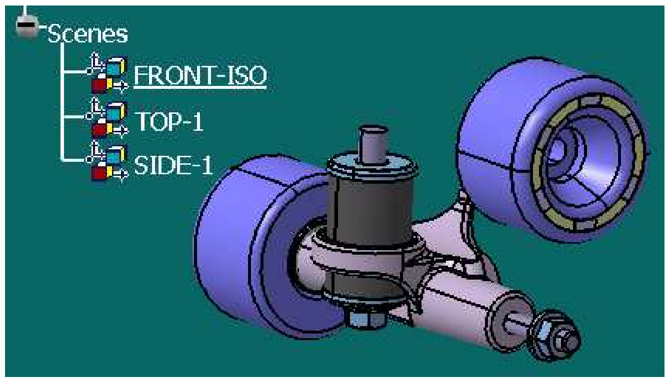


Student Notes:

Introduction to Enhanced Scenes

You will be introduced to 'Enhanced Scenes' and the 'Enhanced Scenes Workbench'





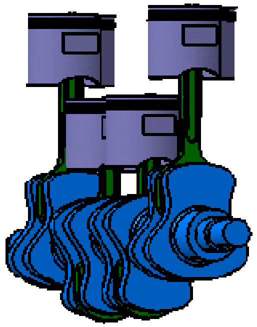
Student Notes:

What are Enhanced Scenes

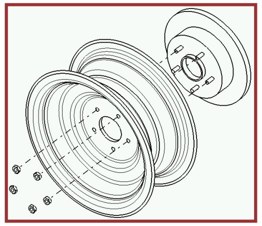
Scenes enable capturing and restoring the state of components in an assembly in a saved viewpoint

Scenes can control:

- Hidden state of components
- Color of components
- Position of components
- Deactivation of representations



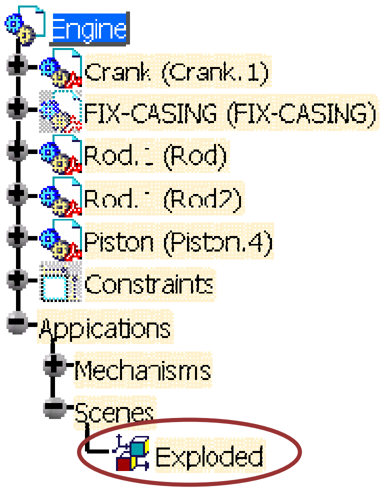
Scenes can be used to create assembly drawings.



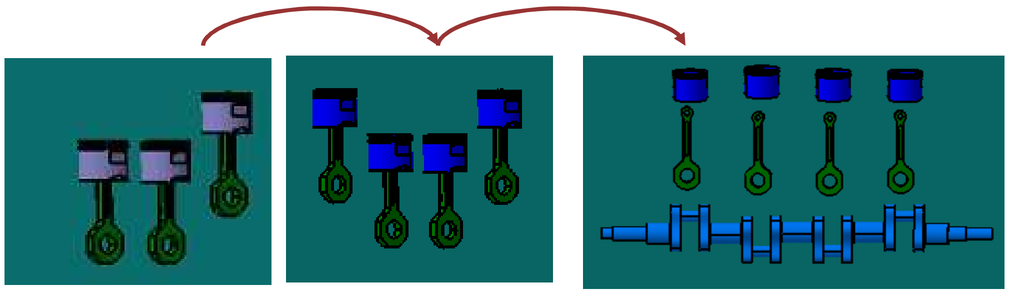
Use Scenes to try new positions and then apply positions to the main DMU window product

In Scenes, you can create reports. Check the DMU Basic course

Scenes are stored in an assembly's CATProduct file



Scenes can be used to set the working state by hiding, coloring, and positioning components.

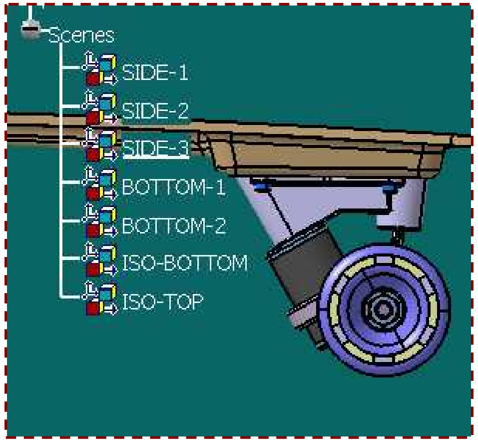


Scenes can be used to see the evolution of an assembly

Student Notes:

Enhanced Scenes Workbench Presentation

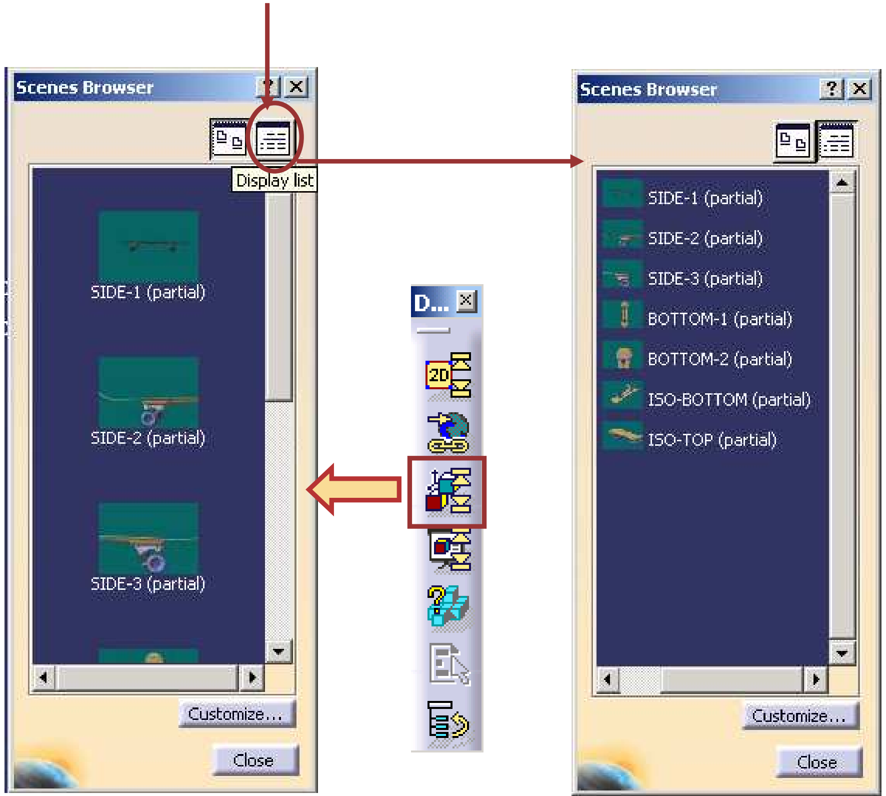
Once you have enter the scene workbench, specific tools are available



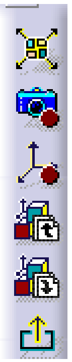
Scene browser is available in DMU/scene workbench.

The List Mode can display more number of 'Scenes' at a time in the 'Scenes Browser'

List Mode



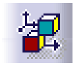
Scenes tools:



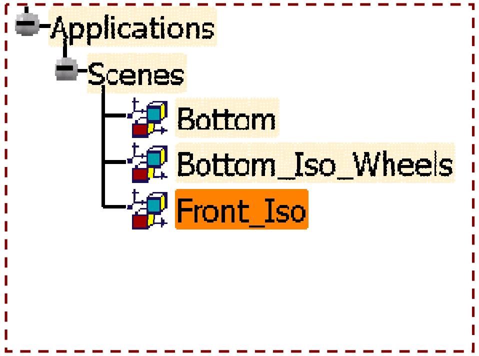
- **Explode Product**
- **Save current viewpoint**
- **Over Load position**
- **Apply scène on the assembly**
- **Apply assembly on scene**
- **Exit the workbench**

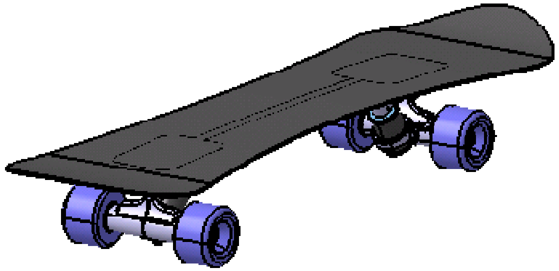
Student Notes:

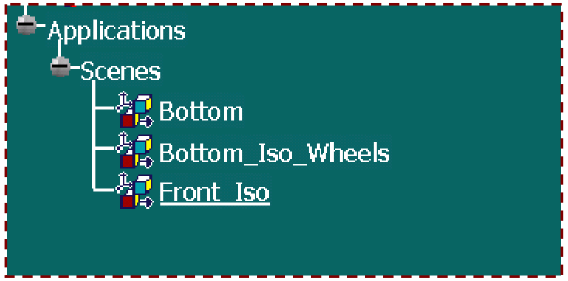
Creating Enhanced Scenes

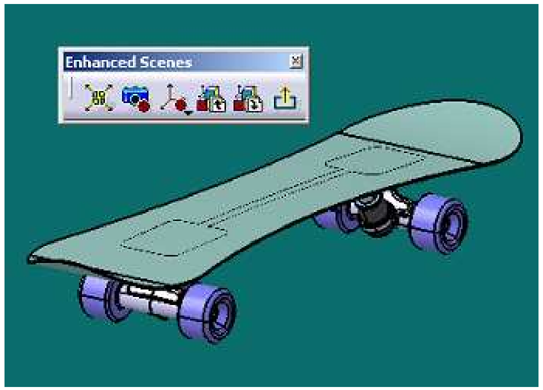


You will learn how to create Enhanced scenes

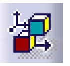






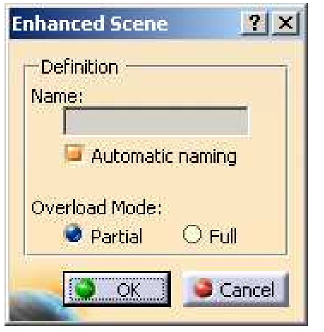


<u>*Student Notes:*</u>

What kind of Scene? 

When you define a scene, you have to choose between 2 kinds of overload mode

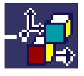
"Overload" is the capability to "spatially restrain" a part in a scene.



Overload Mode FULL: 

If the scene is " fully Overloaded" it means all the parts will keep their positions whatever the

modifications made in the assembly. In other words, The scene will not be updated.

Overload Mode PARTIAL: 

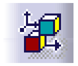
-By default none of attributes are overloaded (it means the scene will be updated if some parts are moved in the assembly).

Why Partial?

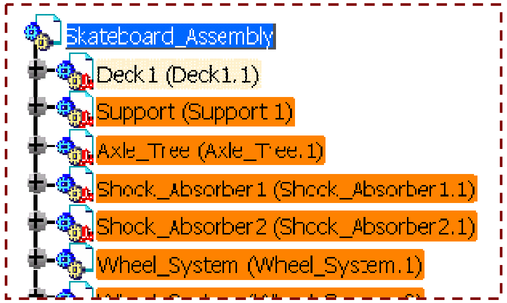
Because it is possible to overload a part in a "partial" scene. If you move a part in a scene context, Catia will consider that you have overloaded the part. Then, if you move this same part in the assembly context, it will not be updated in the scene

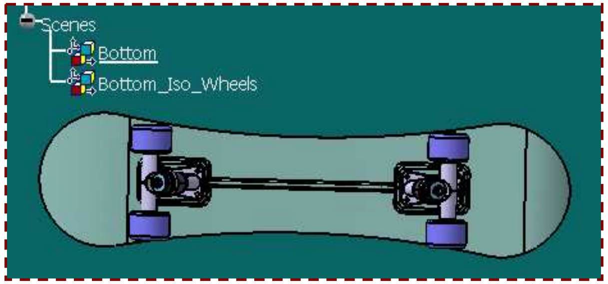
Student Notes:

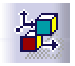
About Enhanced Scene Creation

You have only one way to create a first Scene - using the Enhanced Scene icon 

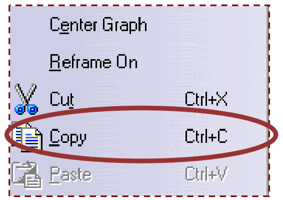
- You can create an Enhanced Scene with all components
- You can create an Enhanced Scene with selected components only





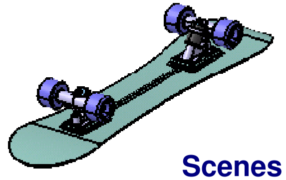
You can create the second scene 

- Using the Enhanced Scene icon OR
- Using the previous Scene as a reference



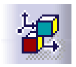
Student Notes:

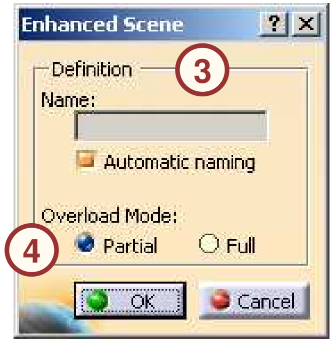
Creating an Enhanced Scene



1 Find the perfect viewpoint for your Scenes

2 Click on the Enhanced Scene icon





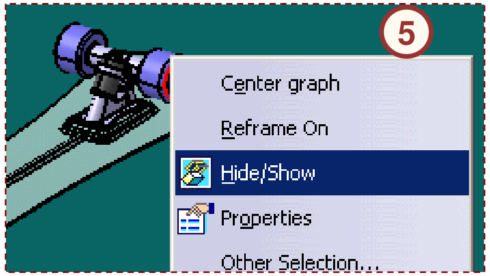
Scenes capture the orientation of the session so be sure the orientation is correct before pressing the Enhanced Scene icon.

3 Specify a Scene name

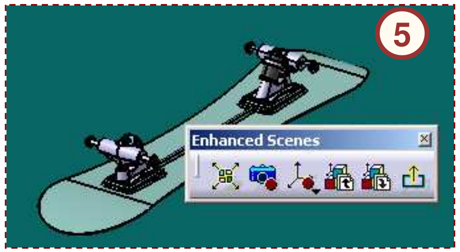
4 Choose between partial/Full overload mode

5 Manage your components

6 Exit the Enhanced Scene



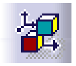
Move Components thanks to the compass, hide some or change color



On creation, all the Components are copying the state (Color, Position, Activation…) of the Main Assembly.

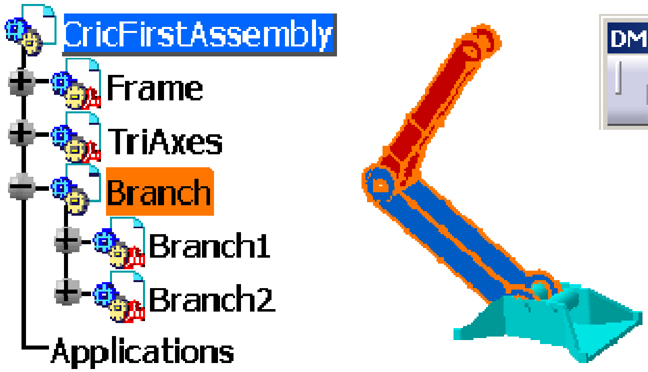
The Main Assembly drives those states unless you modified it in the scene.

Student Notes:

Creating an Enhanced Scene with a Subset of the Assembly 

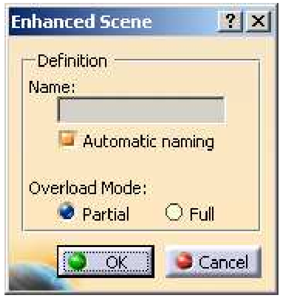
The components in a Scene can be limited to a component (and its children if the component is a sub-assembly).

1 Select a component

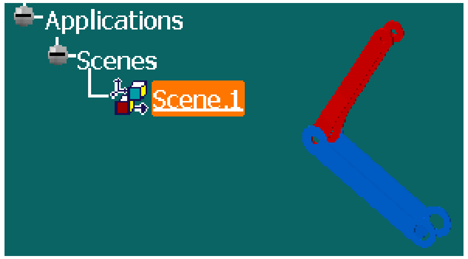


2 Select Enhanced Scene

3 Specify a Scene name/overload mode



4 Scene includes only the selected component

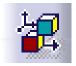


When you enter into scene CATIA background turns Green

5 Exit the Enhanced Scene

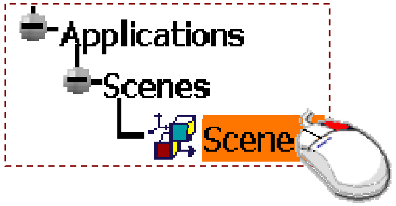
Scene captures the current color, position, and hide state of components in the assembly at the time of creating Scene.

Student Notes:

Creating a New Scene from an Existing Scene 

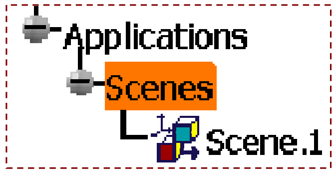
Copy / paste is used to create a Scene from an existing Scene.

1 Right-click the Scene to be copied



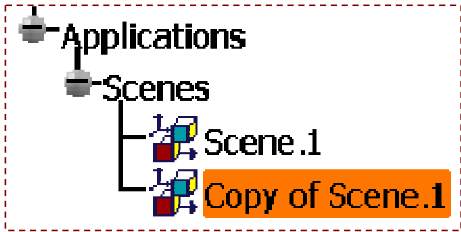
2 From Contextual menu select Copy

3 Right-click the Scenes branch

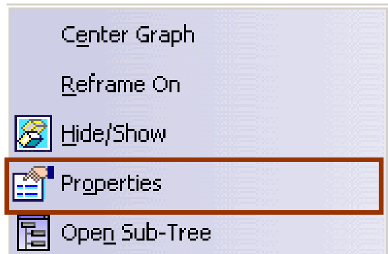


4 From Contextual menu select Paste

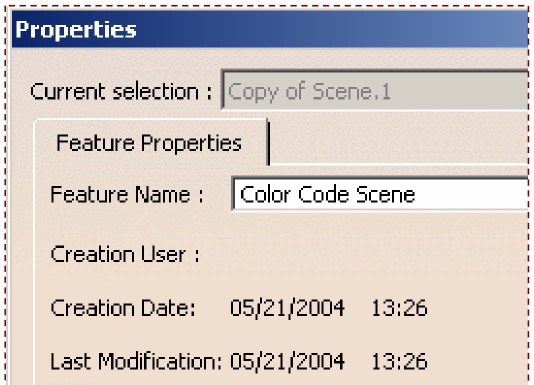
5 Right-click the new Scene

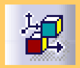


6 From Contextual menu select Properties



7 Specify a new name

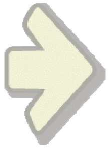


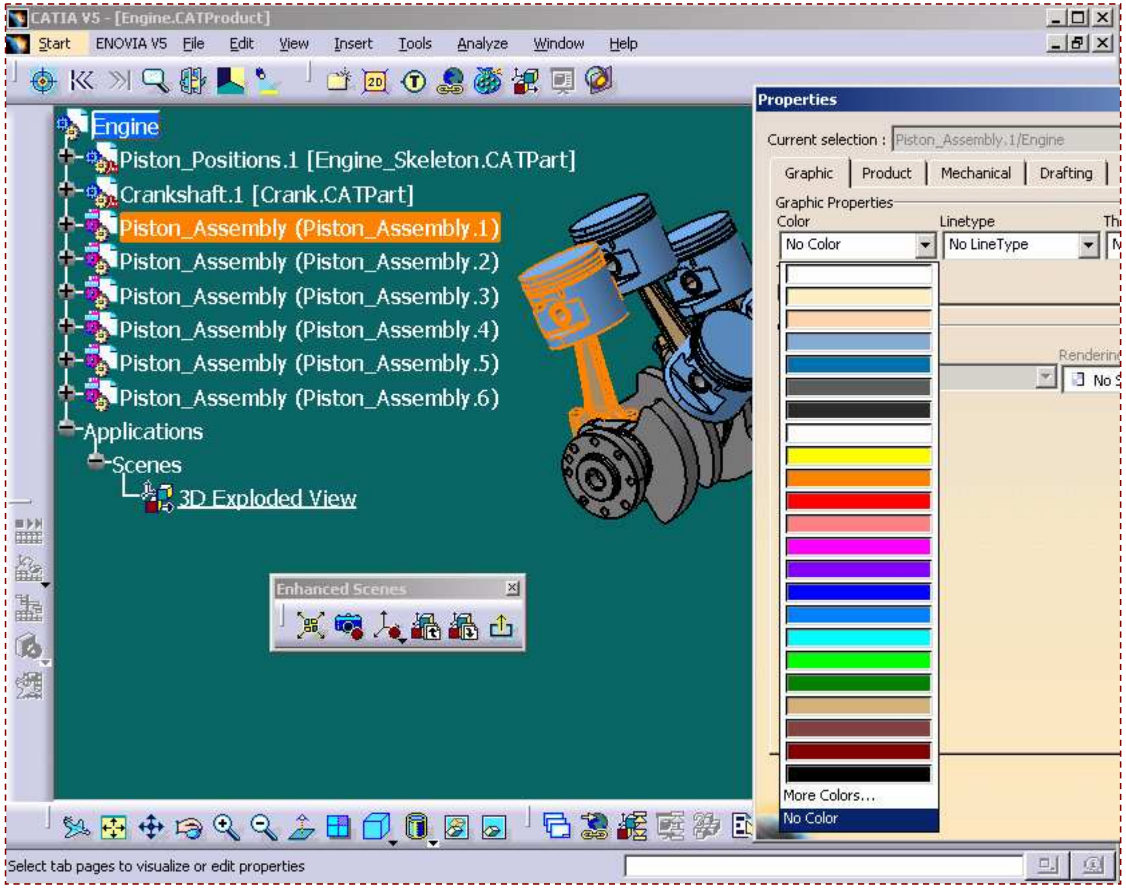
Another way to copy a Scene is to select a Scene in the tree and press the Create Scene icon. 

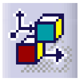
Student Notes:

Editing Enhanced Scenes

You will learn how to edit a scene and modify components' properties

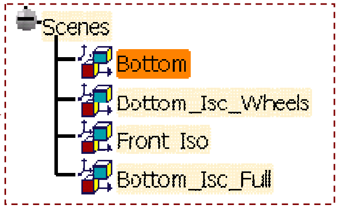


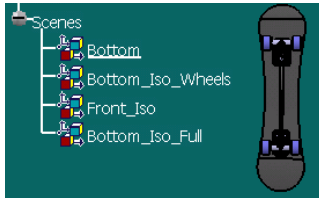


About Editing Enhanced Scenes 

General Scene Management

- **Calling a Scene: Double click on the Scene to open**

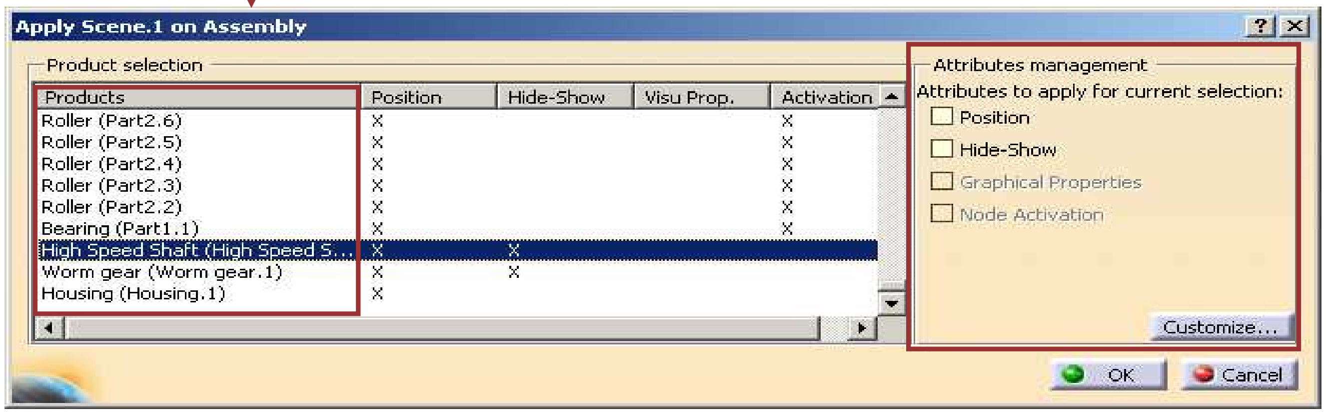




- **Deleting Scenes: Open Contextual menu and delete the Scene**

- **Replacing the Scene Viewpoint : Modify the Viewpoint and click on the this icon** 

- **Apply Selected Attributes of the Scene on the Assembly**



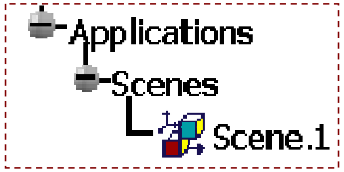
Student Notes:

Student Notes:

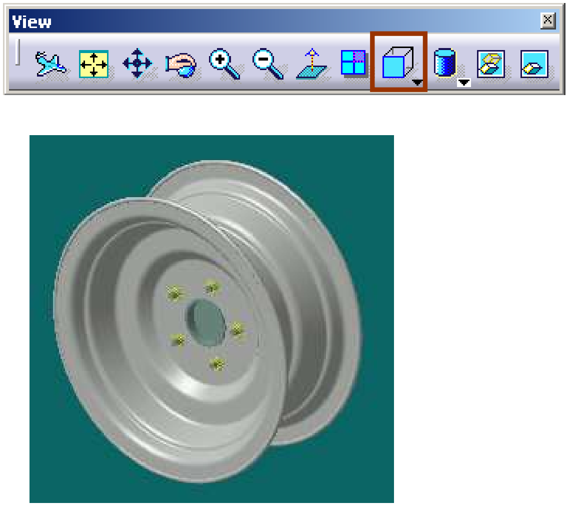
Modifying the Enhanced Scene’s Viewpoint

When You create a Scene the viewpoint you used at the moment of the scene creation, is the one you open when accessing the scene. You will learn here how to modify it

1 Double click on Scene



2 Modify the current viewpoint (for example, using standard views)



When you enter the scene you get the previously saved viewpoint

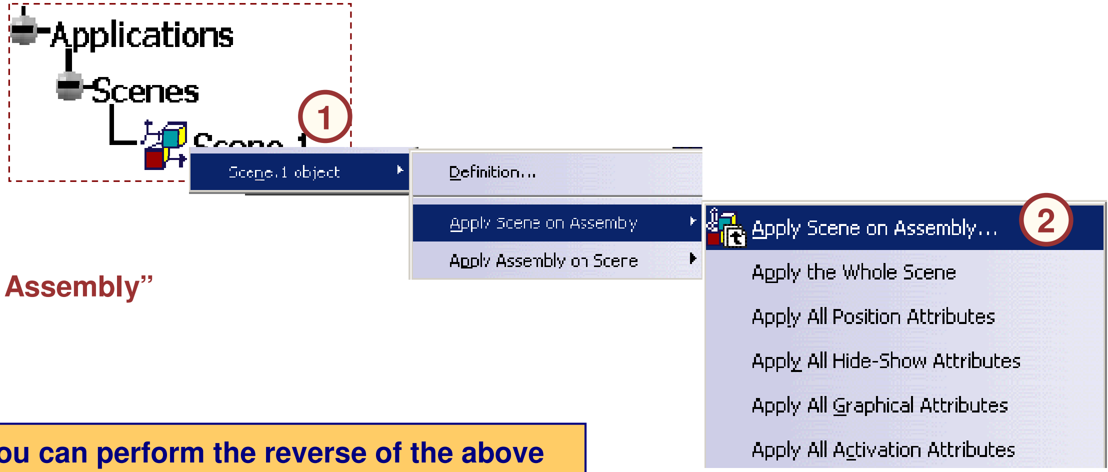
3 Click on ‘Save Viewpoint’ icon

When you will access this scene this will be the viewpoint

Applying a Scene on the Assembly

This function is very useful when you have stored different states of an assembly and want to re-apply them on the assembly

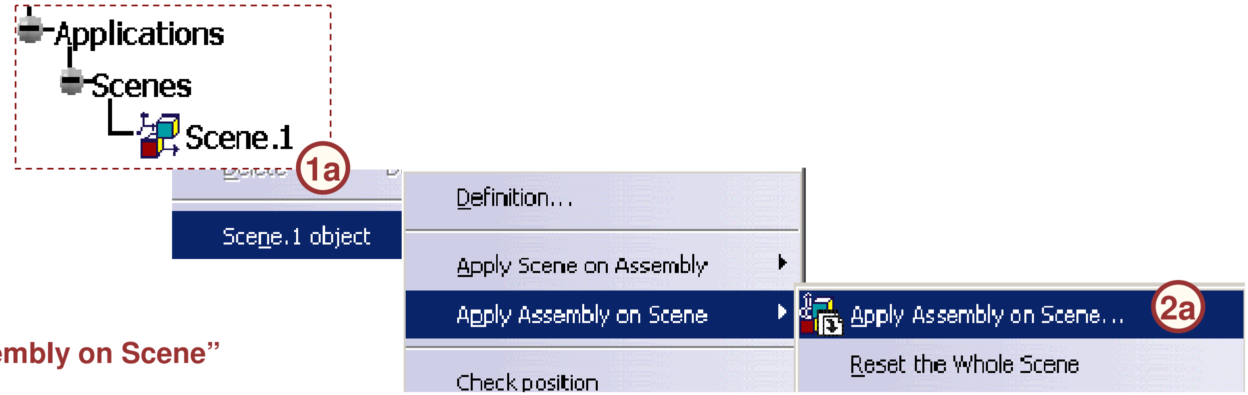
1 Right-click the Scene you want to apply on the assembly



2 Click on "Apply Scene on Assembly"

In a similar way, you can perform the reverse of the above operation i.e apply an Assembly on Scene

1a Right-click the Scene you want to apply assembly on it



2a Click on "Apply Assembly on Scene"

Student Notes:

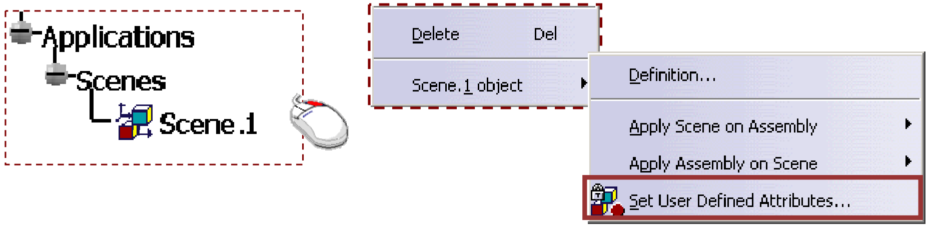
Student Notes:

Applying User defined attributes

This function is very useful when you want to apply predetermined settings

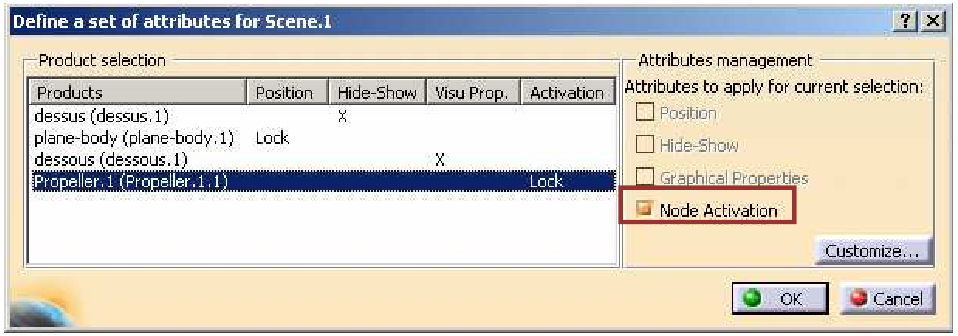
1 Right-click on Scene whose attributes you want to apply on assembly

2 Select “Set User Defined Attributes”



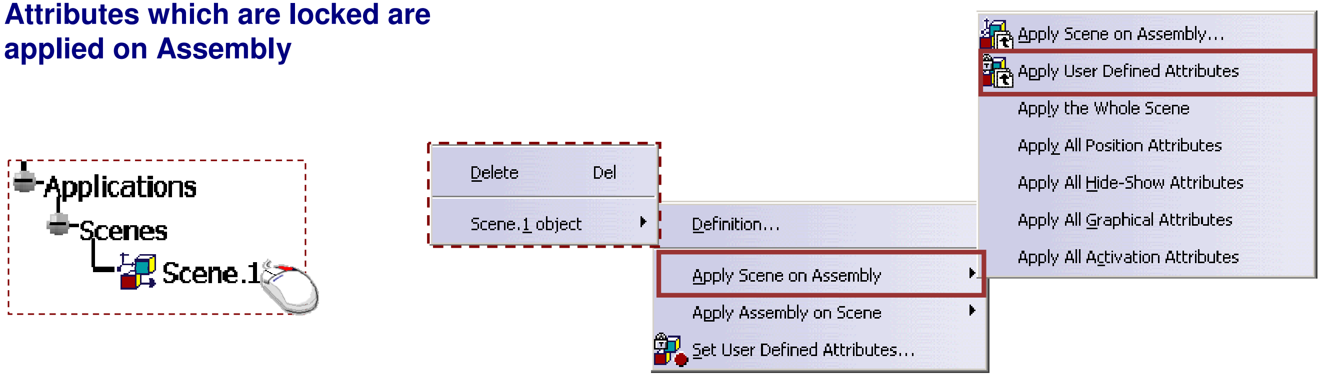
3 Activate attributes which you want to apply

Status will change to Lock



4 Right-click on Scene, go to Apply Scene on Assembly, Select Apply User Defined Attributes

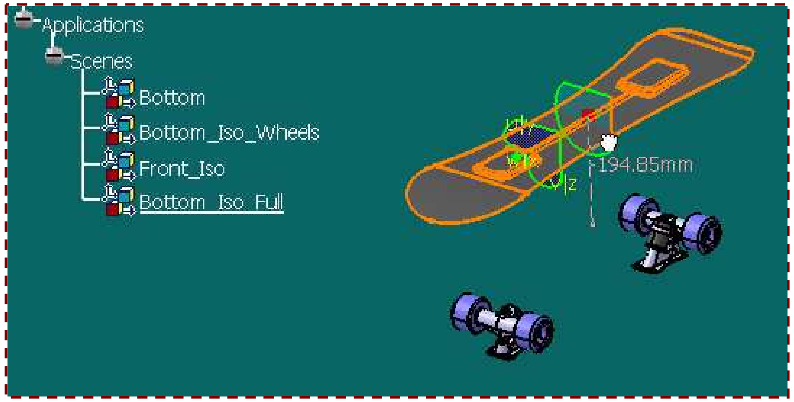
Attributes which are locked are applied on Assembly

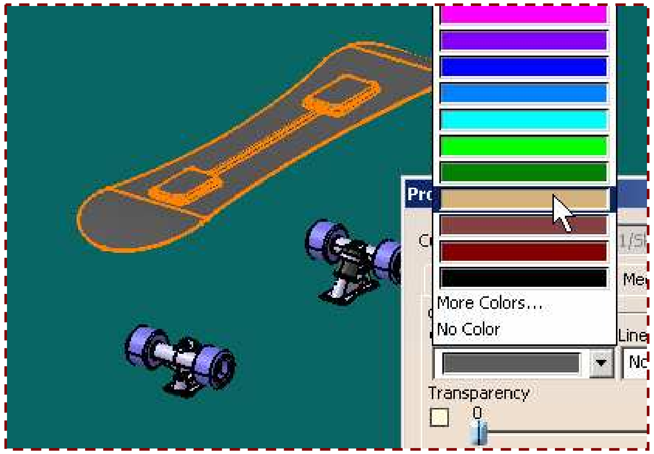


Student Notes:

Managing Components in Enhanced Scenes

You will learn how to manage components and there properties in a Scene





Student Notes:

About Components Management in Enhanced Scenes

You will see how to use the Basic tools available in scenes

Show / No Show Components

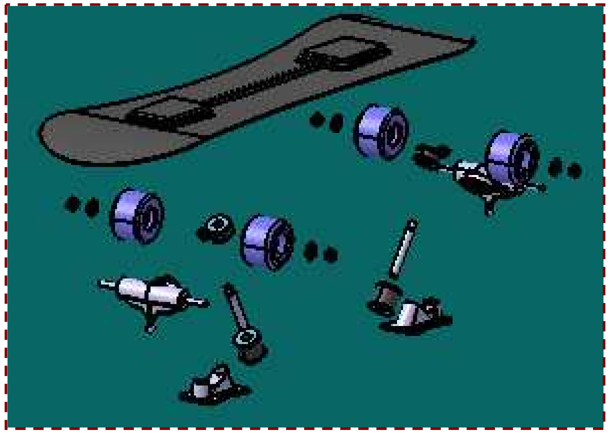
By default the scene is the copy of the Main window, or the selected scene. You may want to swap some objects to show or no show

Modify Color Properties

You can modify the color, without modifying it in the main window

Move Components

This is the main interest of the scenes. You can try new positions, there and then apply them to the main assembly and check the result, by updating the different measures



De-active Shape

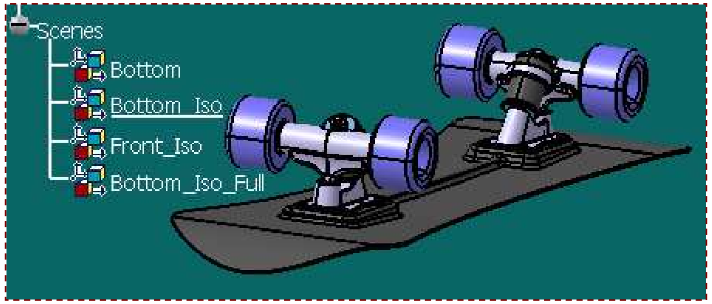
You can de-active shapes, without modifying their state in the main window

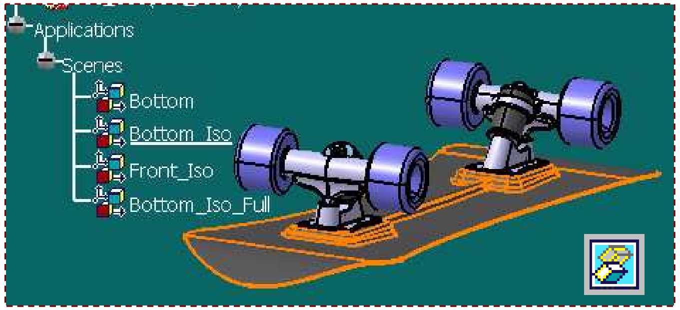
Student Notes:

Hiding Components in Enhanced Scene

Hide state of components can be set in a Scene. This does not effect the assembly.

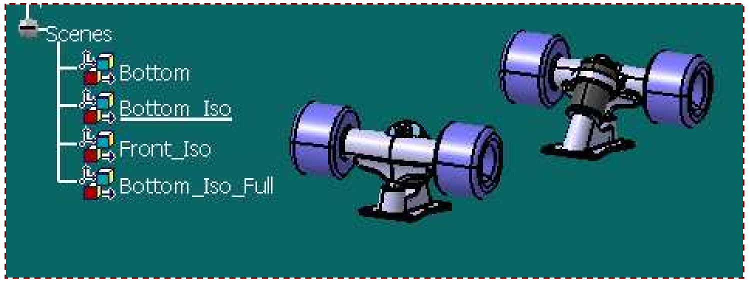
1 Access the Enhanced Scene





2 Select Components to No-Show

3 Click on Show/No-Show button

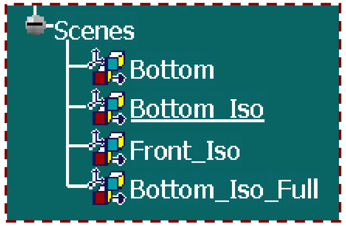


Student Notes:

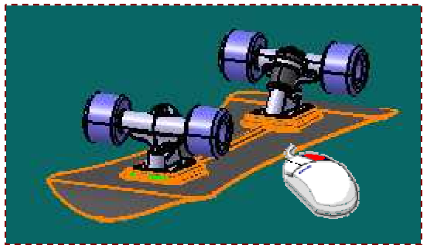
Modifying Color Properties in Enhanced Scenes

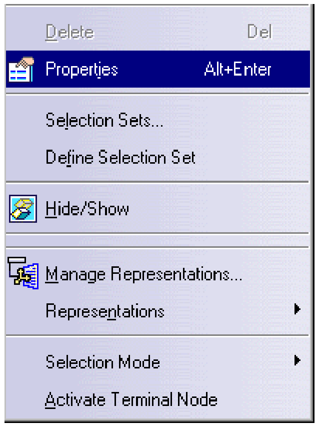
The color of components can be set in a Scene. This does not effect the assembly.

1 Double click on Enhanced Scene



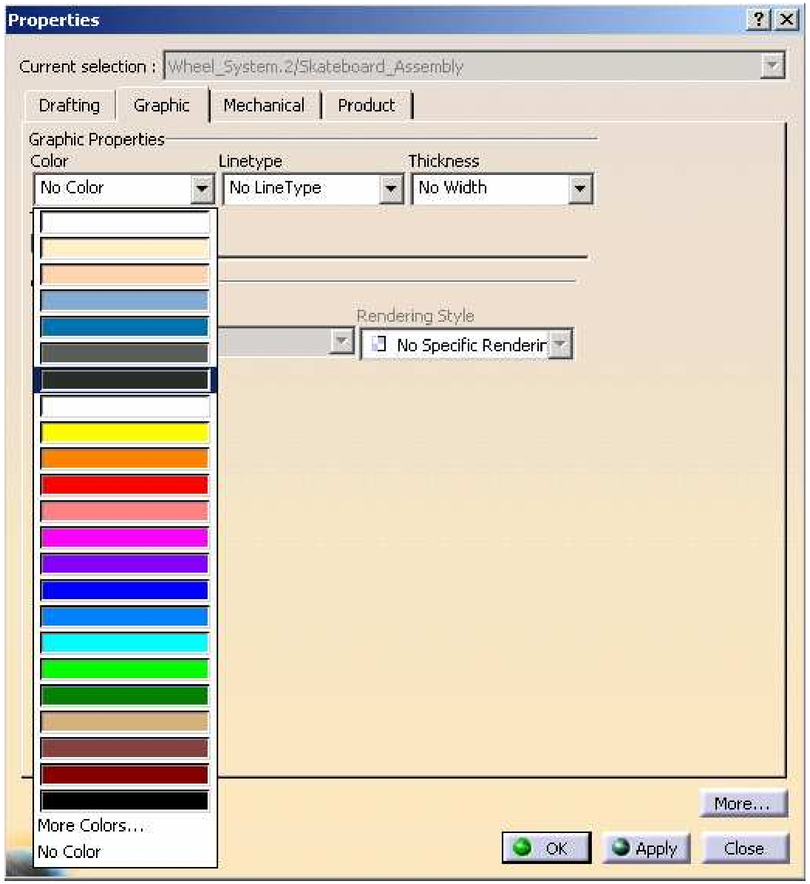
2 Select Components to modify color





3 Open the Contextual Menu and Select Properties

4 Choose a Color in Graphic Tab



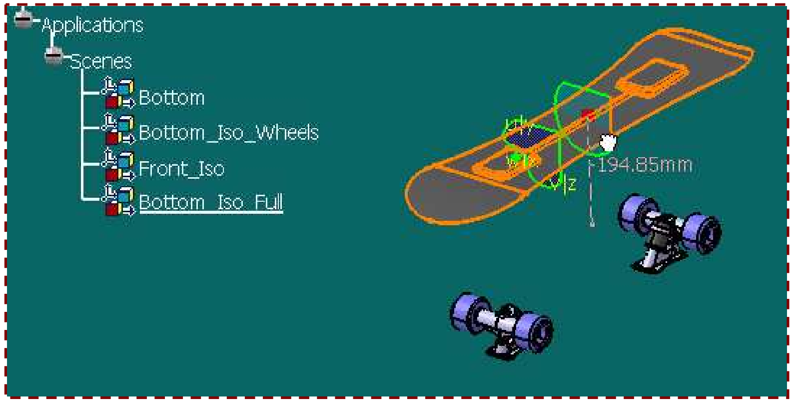
5 Click on OK to Confirm

Student Notes:

Moving Components with Compass in Enhanced Scene

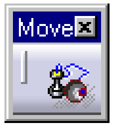
The position of components can be modified. This does not effect the assembly.

1. Drag and Drop the Compass on the object to be moved
2. Move the component by the compass
3. Move Back the Compass to its original position after use



The elements can be moved in the same way than in the main window.

You can also use the Snap function to position the components in the Scene

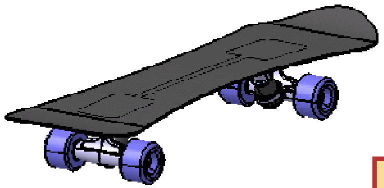


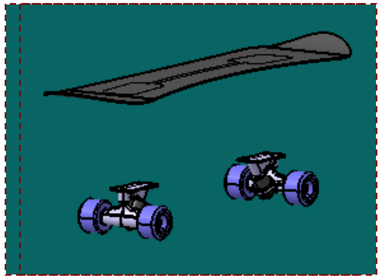
Resetting Components Position in Enhanced Scenes

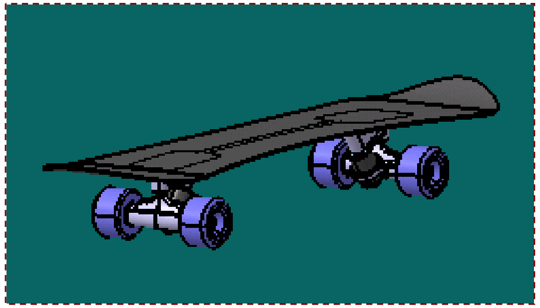
The elements can be reset, using the main window as reference

1. **Select the part to be reset**
2. **Click on the Reset Position icon**

2







State of the components outside the scene

State of the components in the Scene

State of the components in the after resetting position in Scene

Clicking on the Reset Position icon without selecting any component will reset all the components.

You can reset more than one component at a time by selecting with the mouse while holding the [CTRL] key

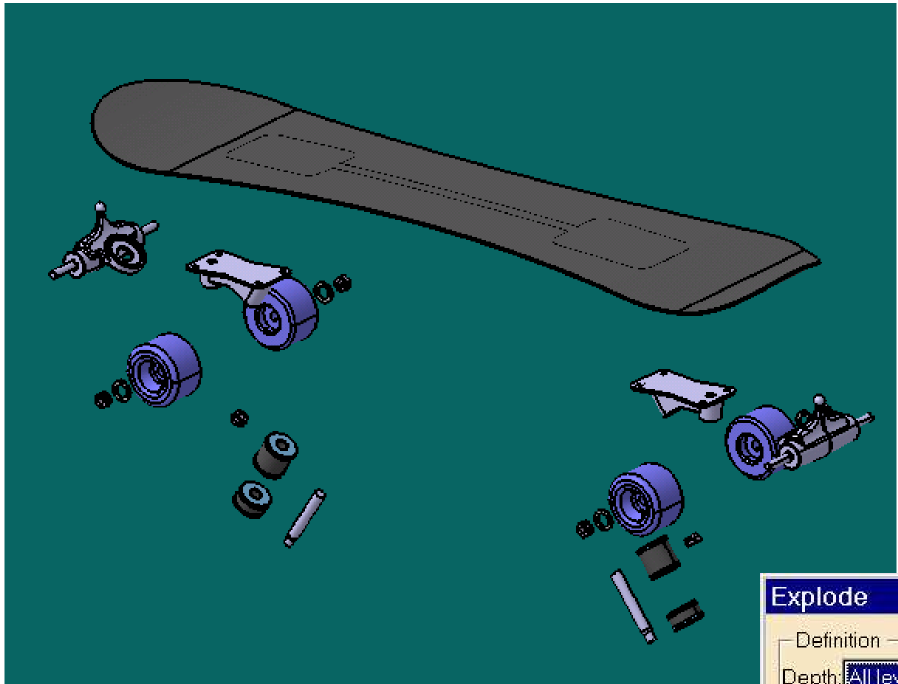
[CTRL] key

Student Notes:

Student Notes:

Creating Explode in Enhanced Scenes

You will learn how to create explode view of assembly in the Scene



Student Notes:

About the Explode Dialog Box

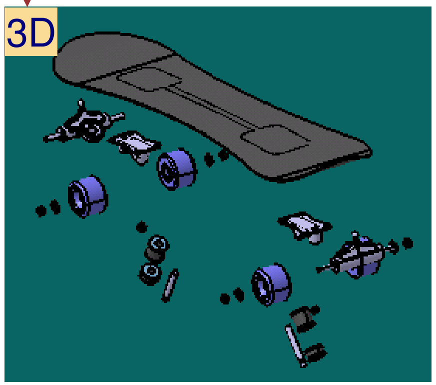
Select in Depth :
•'One Level' only the first level components of the product(s) will be exploded
•'All Level' all the components of the product(s) will be exploded

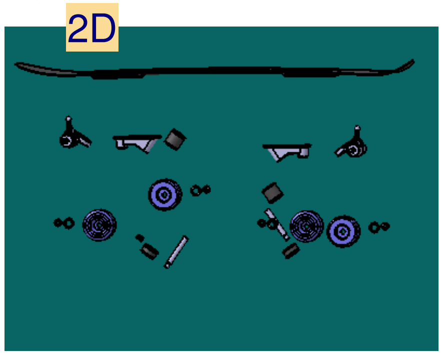
Selected Products to be exploded

Select type of Explode:

- **'3D' Product(s) are exploded in the space**
- **2D Products are exploded and placed in the same plane, parallel to the screen**
- **'Constrained' Products are exploded according to assembly constraints.**

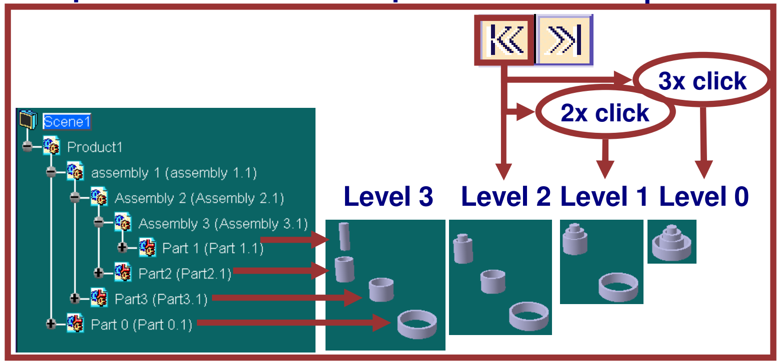
Select fixed Product





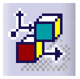
Cursor representing level of Depth in exploded Products. Example :

To navigate through levels of Explode



Student Notes:

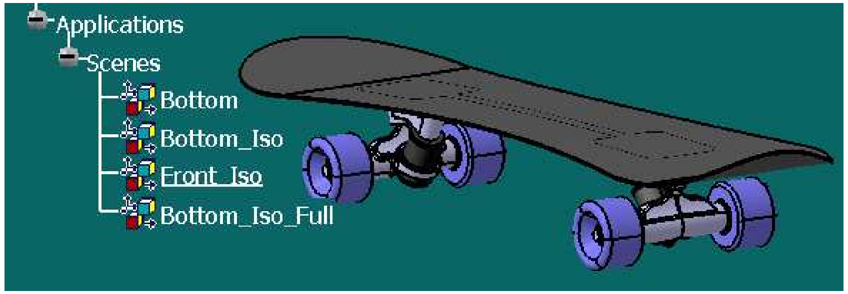
Exploding in an Enhanced Scene



1 Activate the desired Scene

2 Click on 

3 Fill the Explode Dialog Box

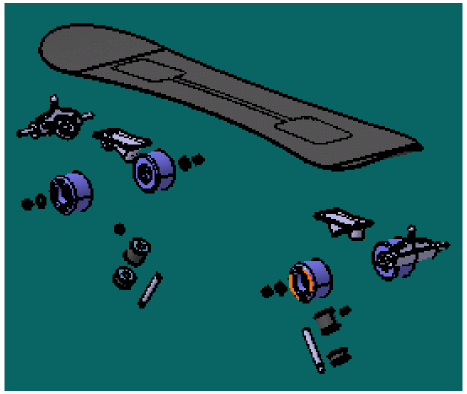


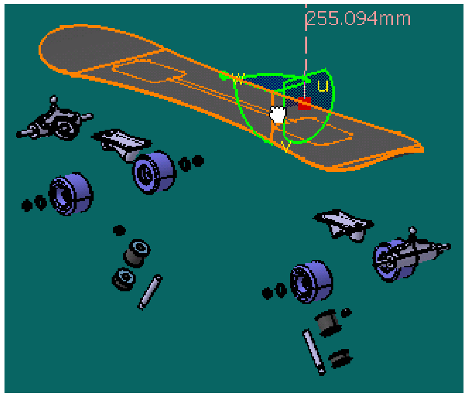
Use the Compass to move components in scenes

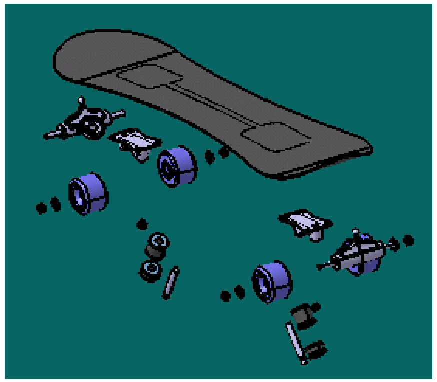
6 Click on OK to Confirm

5 Move the Components

4 Click on "Apply"



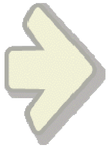


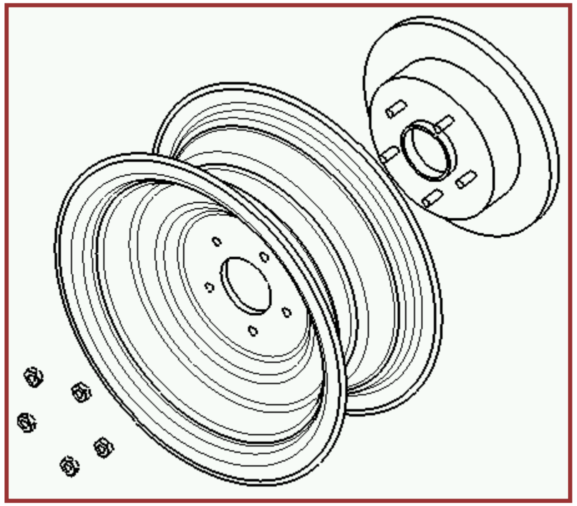


Student Notes:

Creating Drafting Views Based On Scenes

You will see how to create a drafting view based on a scene





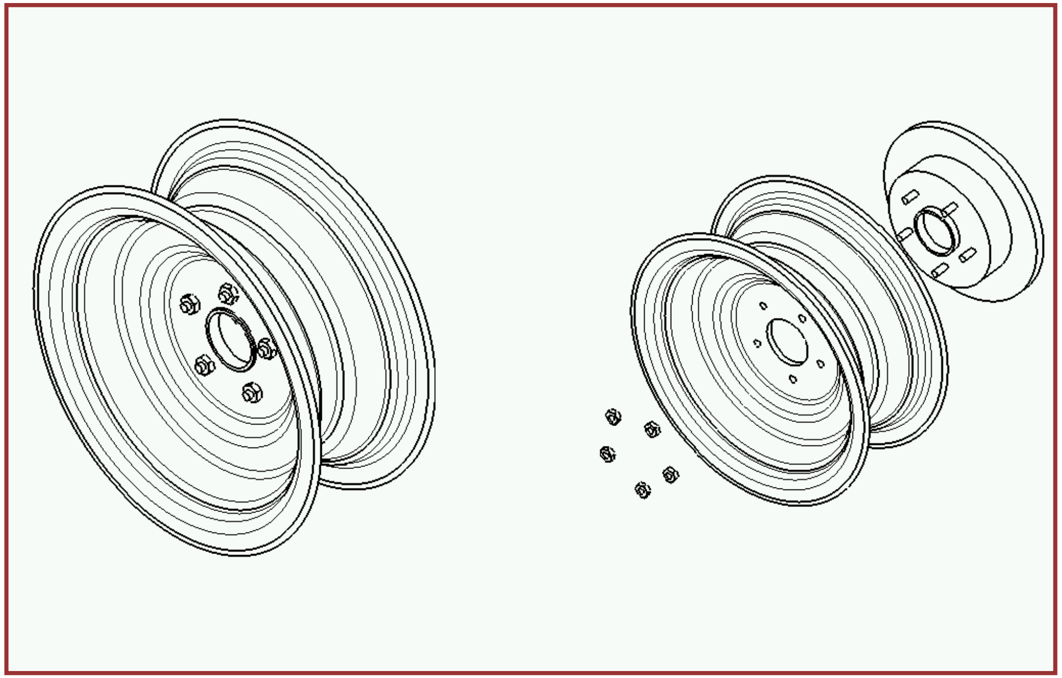
Student Notes:

What are Drafting Views Based on Scenes ?

Scenes make it possible to have drafting views where components are in different states than the assembly. Scenes also avoid the need to reconstruct manually views as components are added, deleted, replaced, and moved in an assembly.

Without Scenes it would be difficult to create a drafting sheet that shows the assembly in two different states (as shown below).

Scenes also avoid the need to reconstruct views (such as the exploded view shown below) when components are added, deleted, replaced, and moved in the assembly.

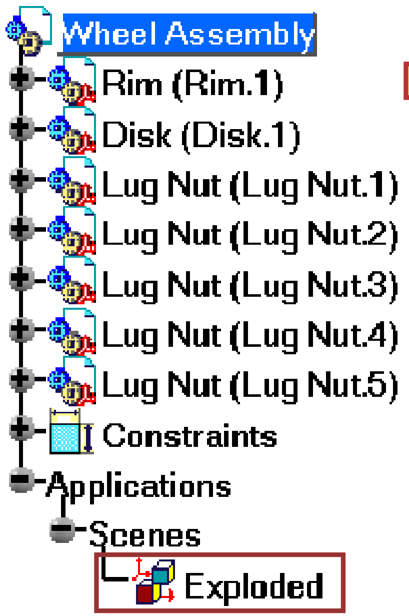


Student Notes:

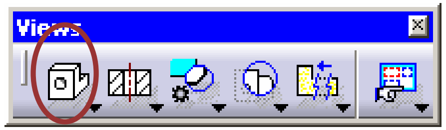
Creating a Drafting View Based On a Scene

Creating a drafting view based on a scene follows nearly the same steps as normally done when creating views.

1 Double-click the scene



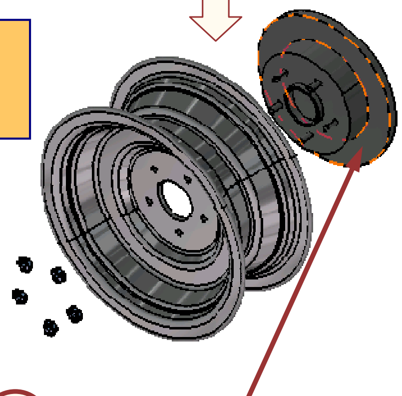
2 Switch to the Drafting Workbench and select a isometric view creation icon.



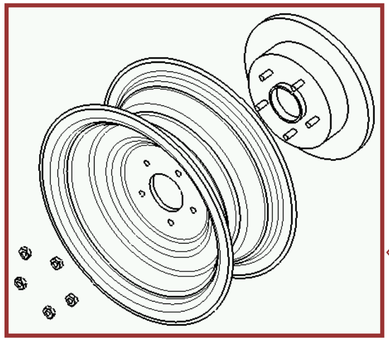
You can expand the tree and select a different node. Only selected node and it's components will appear in the view.

Do not use to hide components in a scene the Hide/Show tool ; use instead the Activate/Deactivate tool.

3 Select the entire assembly



4 Select a reference plane



5 Click the drafting sheet to accept the view (as normally done when creating a view)

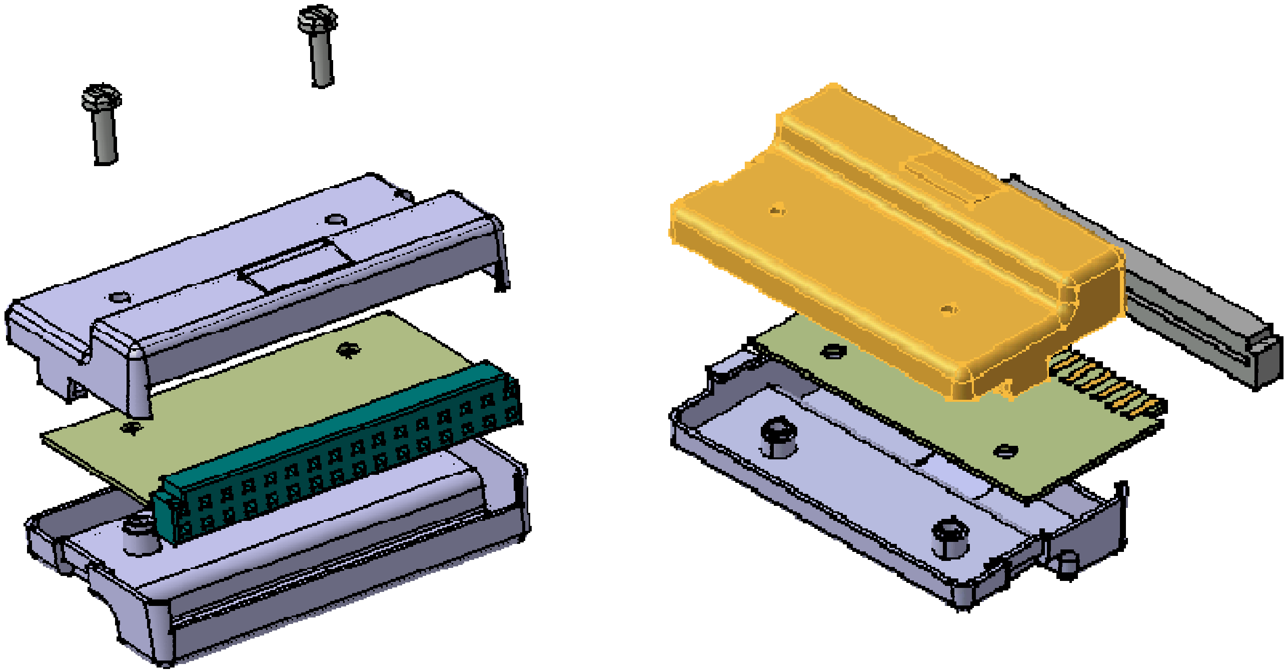
Student Notes:

Managing Scenes: Recap Exercise

Connector Housing

15 min

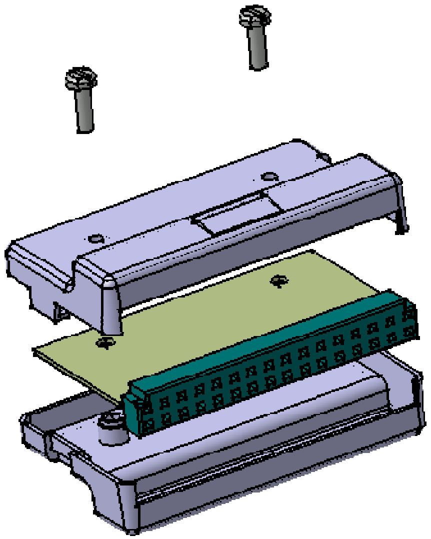
In this exercise you will create scenes that capture various states of the assembly and you will apply them to the assembly



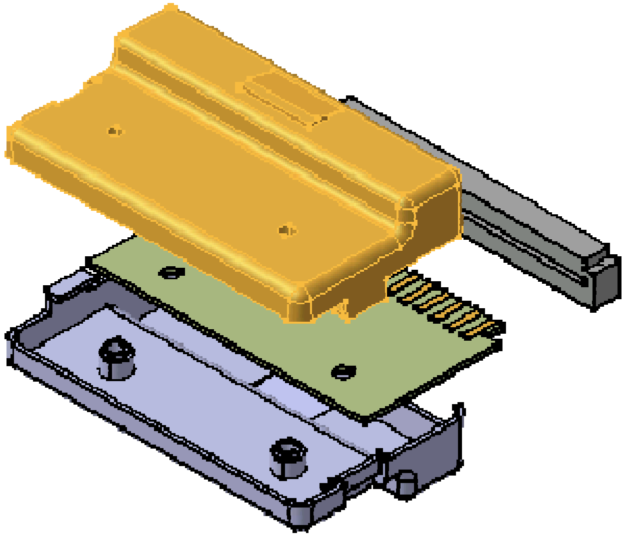
Student Notes:

Do It Yourself

Product used: CATASMConnector_Assembly_6.CATProduct



All Components Exploded



Working State

1- Create a scene named "*Assembled*" in which the components are in the same state as the assembly.
2- Create a scene named "*All Components Exploded*" where the components are positioned as above.
3- Create a scene named "*Working State*" where:
  a) The screws are hidden
  b) The connector socket receptacle is moved away from the connector card
  c) The top shell is gold color
4- Apply the scene "*All Components Exploded*" to the assembly and get back to the assembled state
5- Apply colors and hide states of the scene "*Working State*" to the Assembly then get back to the assembled state

Student Notes:

To Sum Up ...

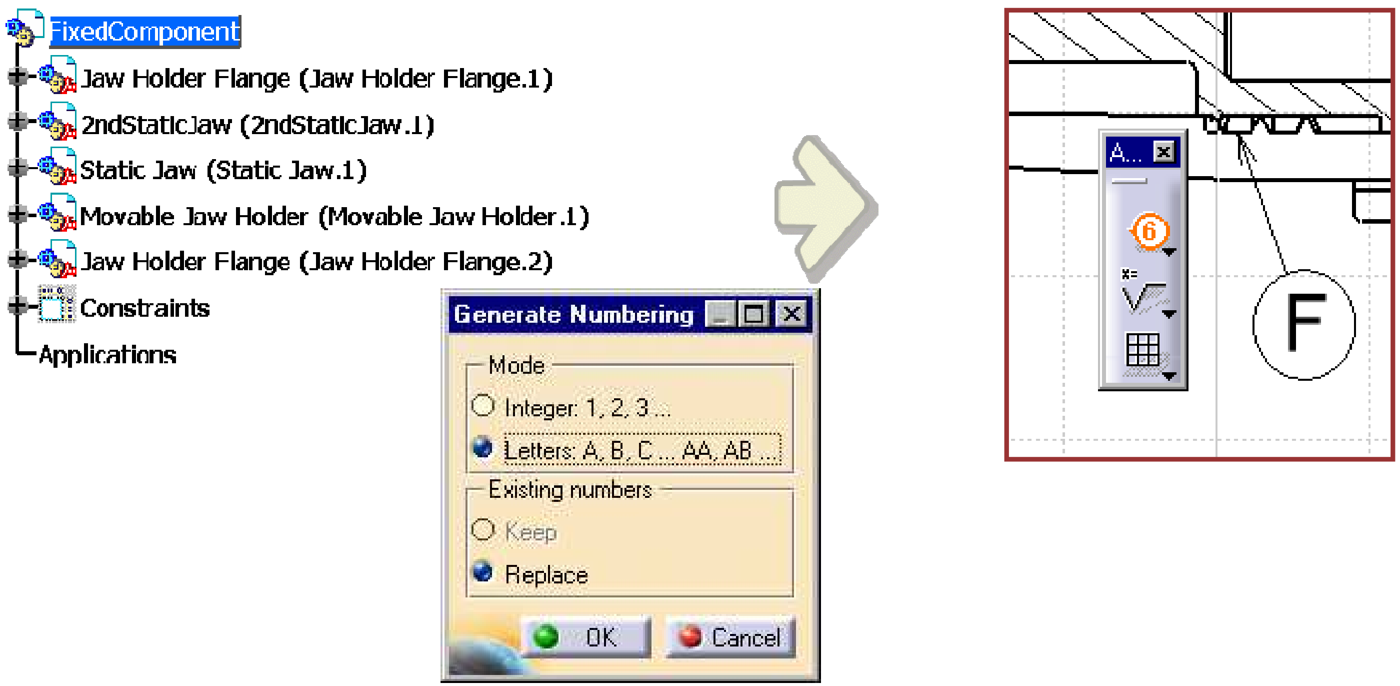
This concludes the sub lesson on managing scenes, you should be able to demonstrate:

- **Creating and editing enhanced scenes**
- **Managing components in enhanced scenes**
- **Creating exploded 3D view in a scene**
- **Creating drafting views based on scene**

Student Notes:

Product Structure Numbering

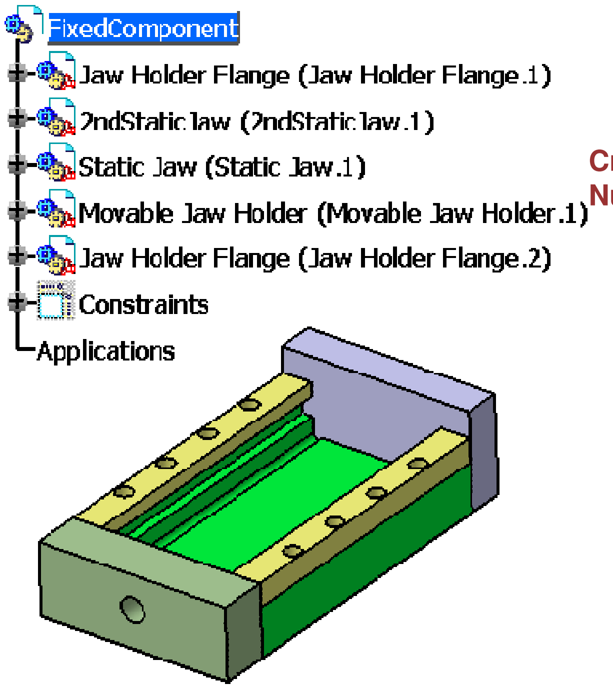
You will learn how to associate numbers usable for drafting to the different parts of your assembly



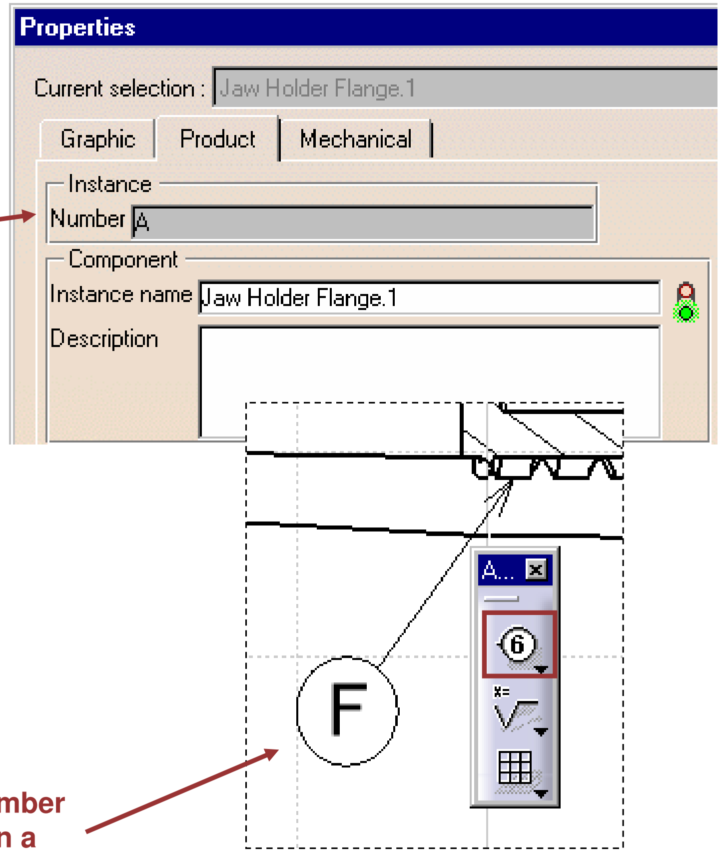
Student Notes:

What Is Product Structure Numbering?

The Generate Numbering command allows you to associate numbers or letters to the parts which make up your product. Those numbers will also appear when creating balloons on Drawings generated from the assembly



Created Number



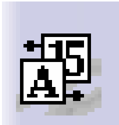
Created Number appearing in a balloon

Student Notes:

Product Structure Numbering (1/2)

The Generate Numbering command allows you to associate numbers or letters to the parts which make up your product.

1 Select the Generate Numbering icon

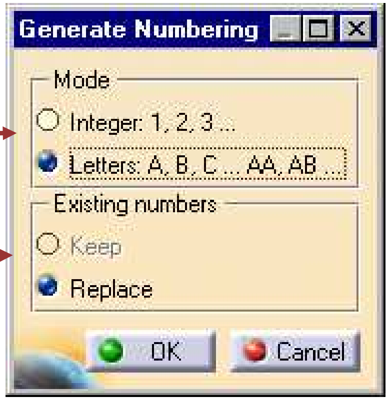


2 Select the root of the product

3 Select numbering mode

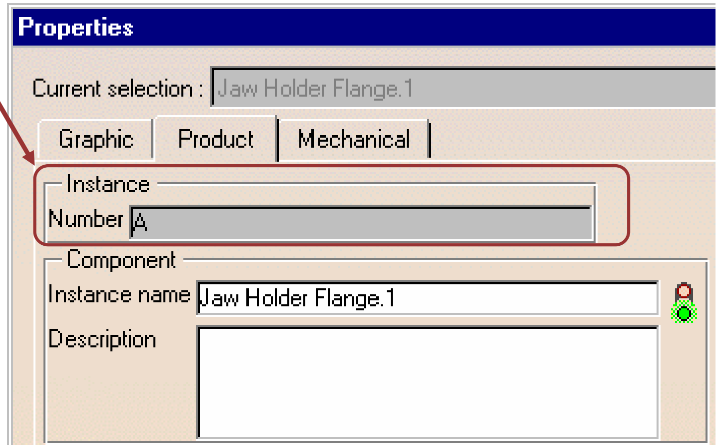
4 Select what you do with already existing numbers

5 Click on Ok

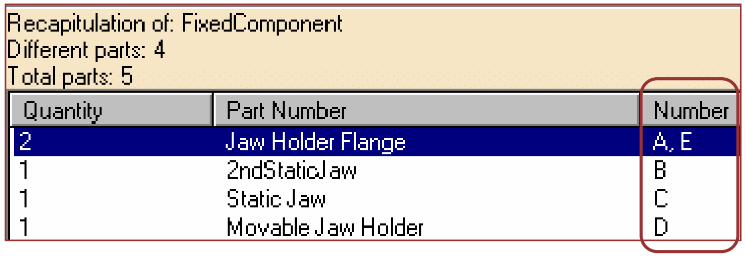


6a The generated numbers appear in the properties dialog box (Product tab)

Created Number



6b The generated numbers also appear in the bill of Material



Recapitulation of: FixedComponent
Different parts: 4
Total parts: 5

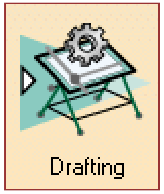
Quantity	Part Number	Number
2	Jaw Holder Flange	A, E
1	2ndStaticJaw	B
1	Static Jaw	C
1	Movable Jaw Holder	D

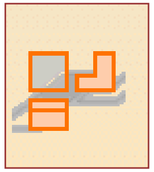
Student Notes:

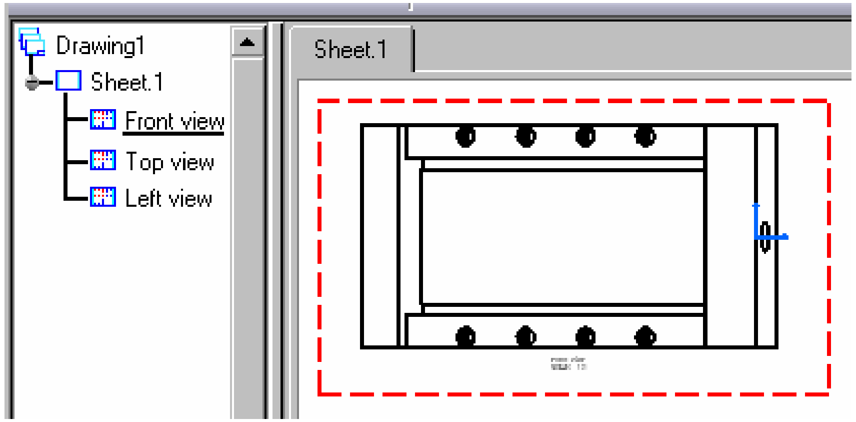
Product Structure Numbering (2/2)

On a drawing generated from the product , when creating balloons on parts you will have by default the letter or number generated for the part in 3d.

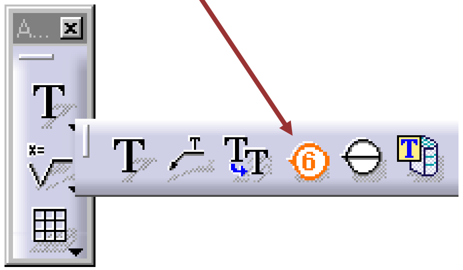
7 Generate a Drawing from the 3d



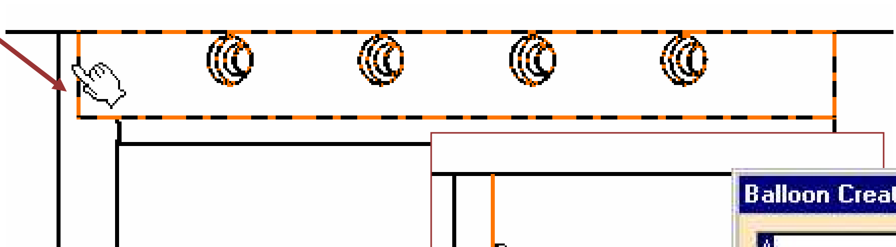




8 Select balloons icon In Drafting Workbench

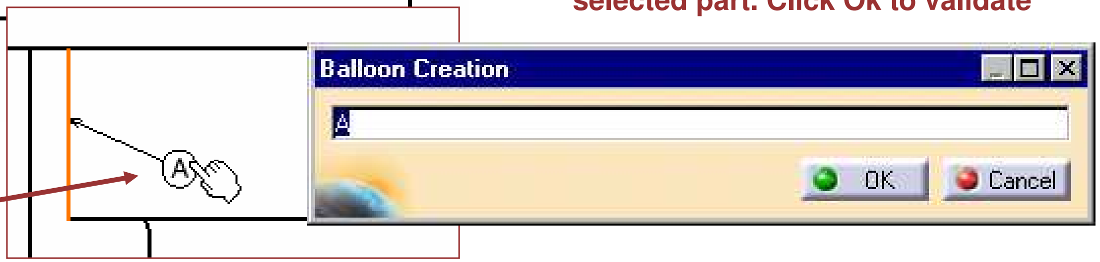


9 When selecting an edge for ballooning, you will get the whole part.Click it



11 You can change it but the default value inside the balloon is the number of the selected part. Click Ok to validate

10 click on the sheet to place the balloon



Student Notes:

Generating Annotations

You will see how to create several types of annotations in CATProduct files

- Introduction to Generating Annotations
- Weld Feature Annotations
- Text Annotations
- Flag Note
- Manipulating Annotations
- To Sum Up

Student Notes:

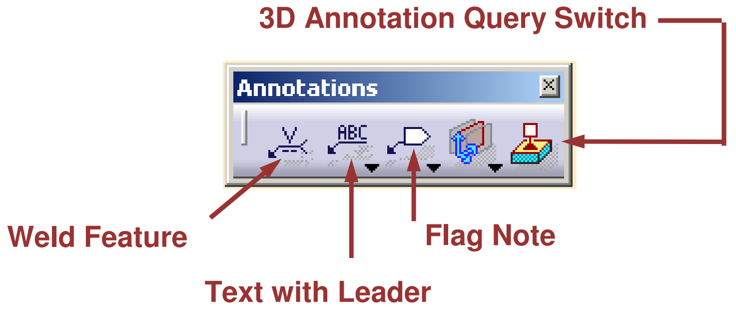
Introduction to Generating Annotations

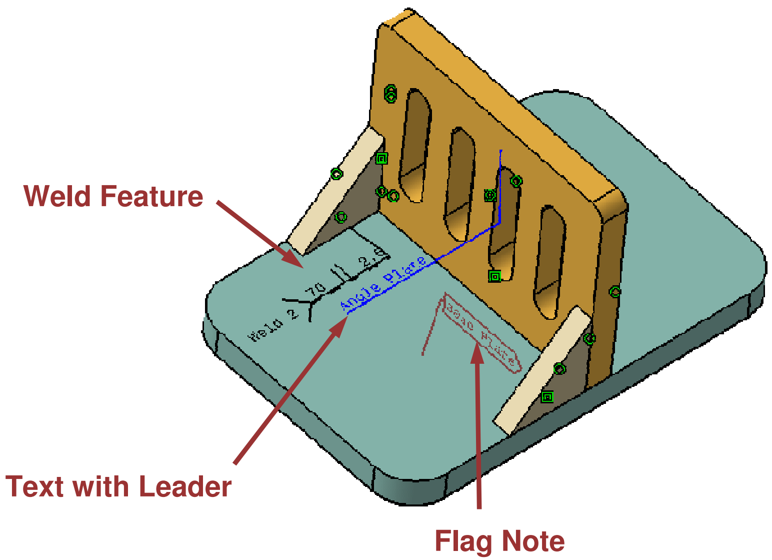
Annotations are added to the assembly documents to denote additional information about a part or product. This additional information can be a brief description of the part, material used for the part, mating component, usage of the part, finish requirements, hardness requirements and much more.

In Assembly Design Workbench, you can add following three types of annotations :

- **Weld Feature : A Weld Feature allows you to add weld symbols and notations.**
- **Text with Leader : A Text with Leader allows you to add a brief description of the part.**
- **Flag Note with Leader : A flag note allows you to add links to your document and then use them to jump to a variety of locations, for example to a marketing presentation, a text document or a HTML page on the intranet.**

  You can add links to models, products and parts as well as to any constituent elements.

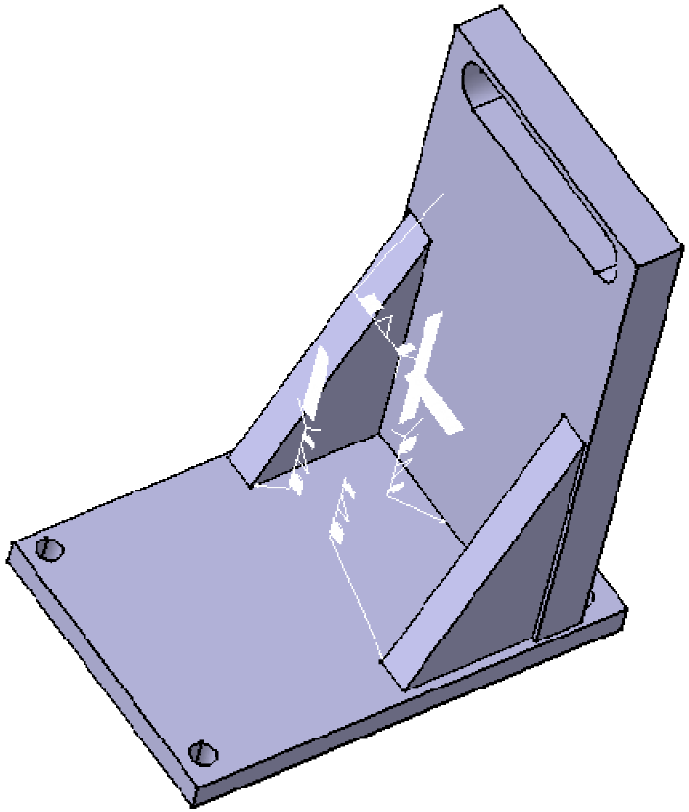




Student Notes:

Weld Feature Annotations

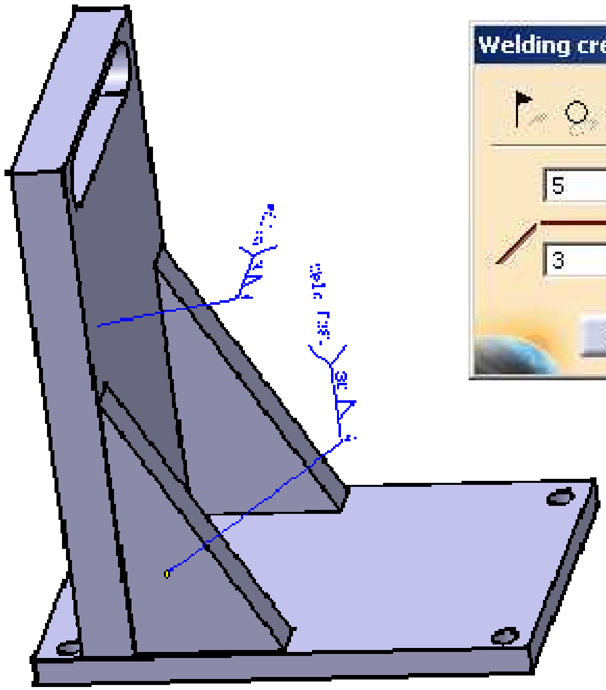
You will learn how to add annotations concerning welding between components inside FD&T 3Dviews

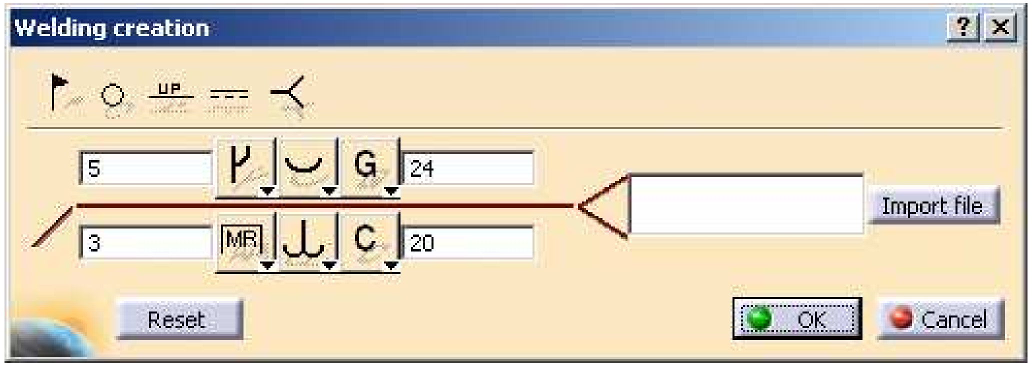


Student Notes:

What Are Weld Feature Annotations ?

Weld Feature annotations show specifications for welding between several components.
CATIA add them inside FD&T (Functional Dimensioning and Tolerancing) 3D views.



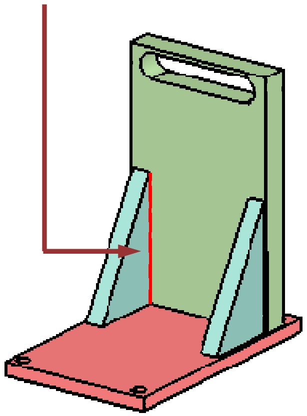


Student Notes:

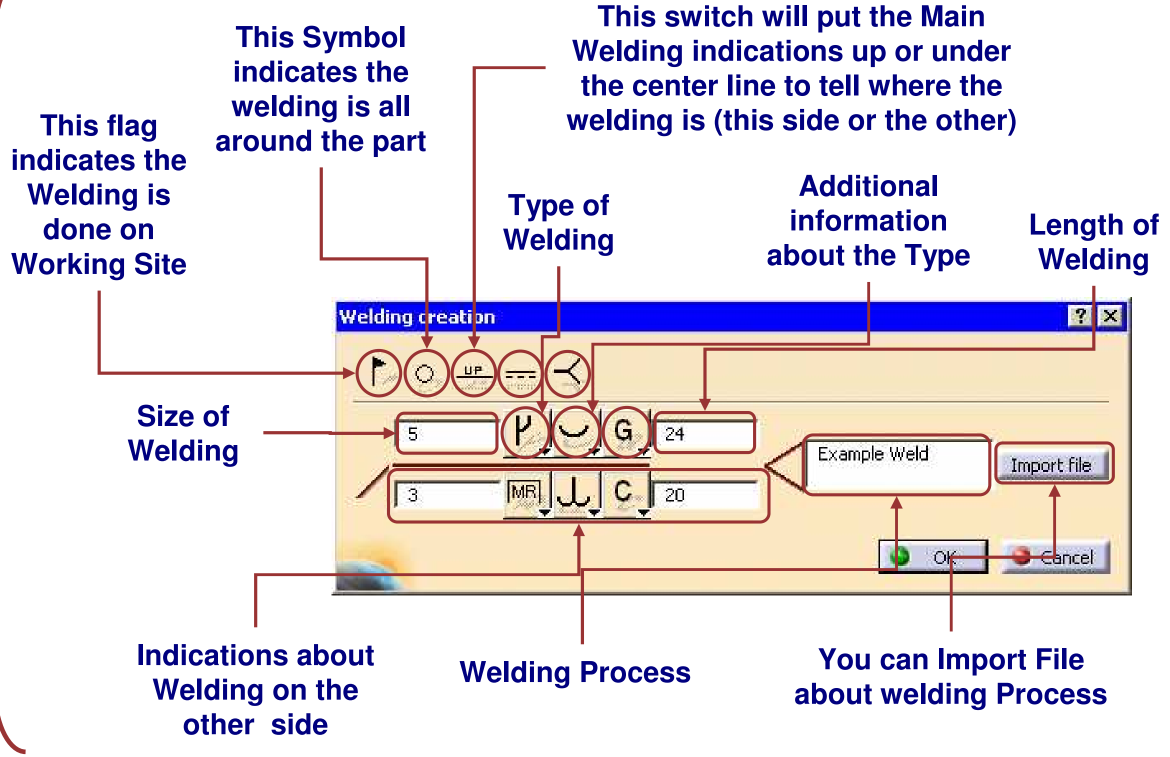
Creating a Weld Feature (1/2)

You will see how to add a Weld feature annotation to annotate the parts which are to be welded together.

1. Click the Weld Feature icon
2. Select the edge of the component as shown



3. Specify the various details in the Welding Creation dialog box.
4. When finished with specifications click on OK

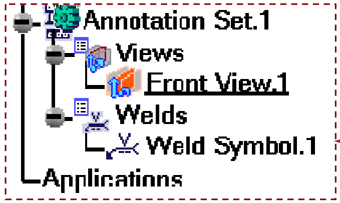


Student Notes:

Creating a Weld Feature (2/2)

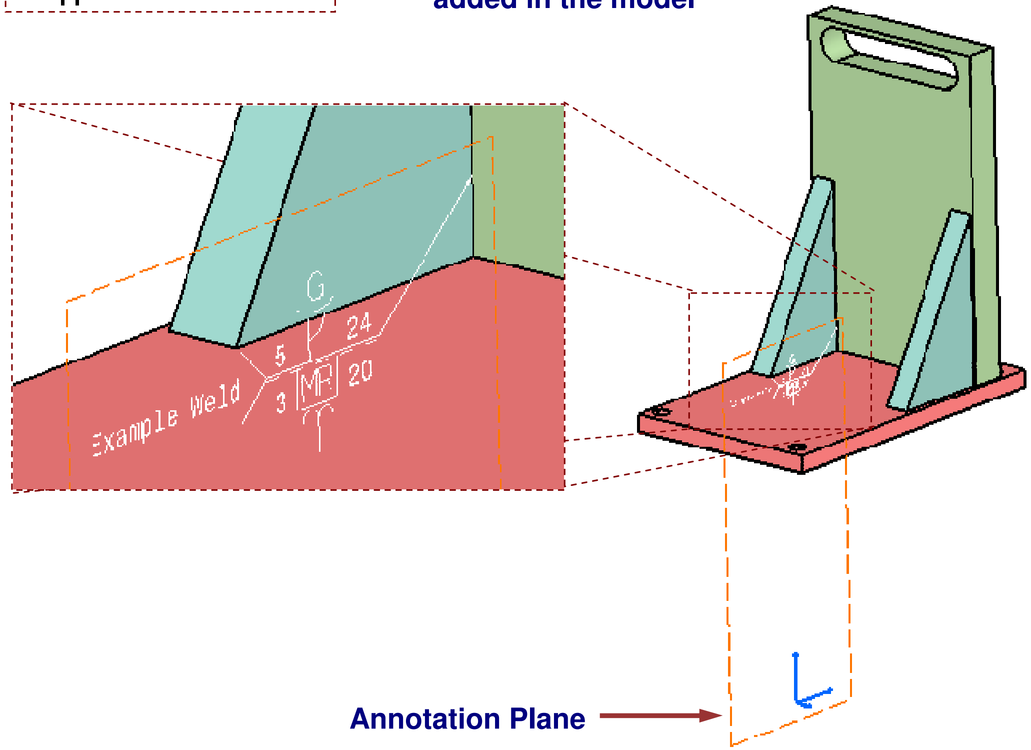
Addition of Weld Feature annotation induces creation of a support plane for the annotation.

Annotation Set is added in the specification tree. It contains two nodes – Views and Welds



Weld Node contains various Weld Feature annotations added in the model

Weld Feature annotation is displayed in the model

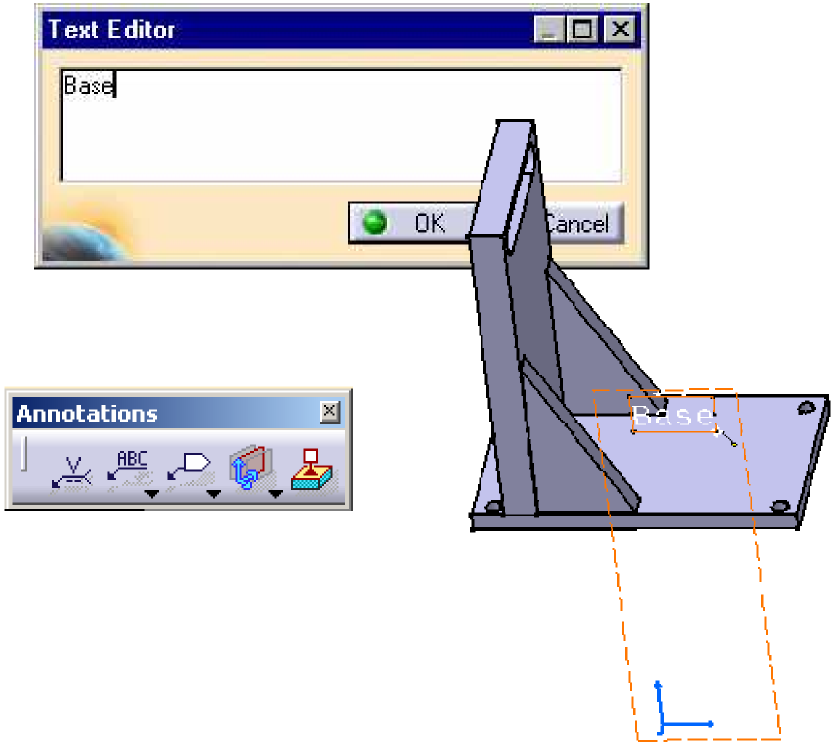


Annotation Plane

Student Notes:

Text Annotations

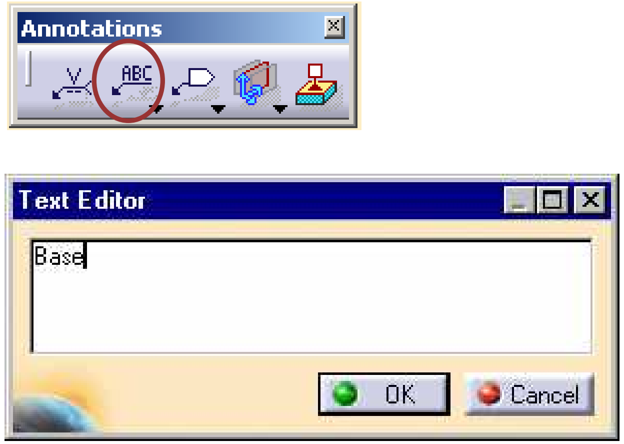
You will see how to add text annotations on your assembly

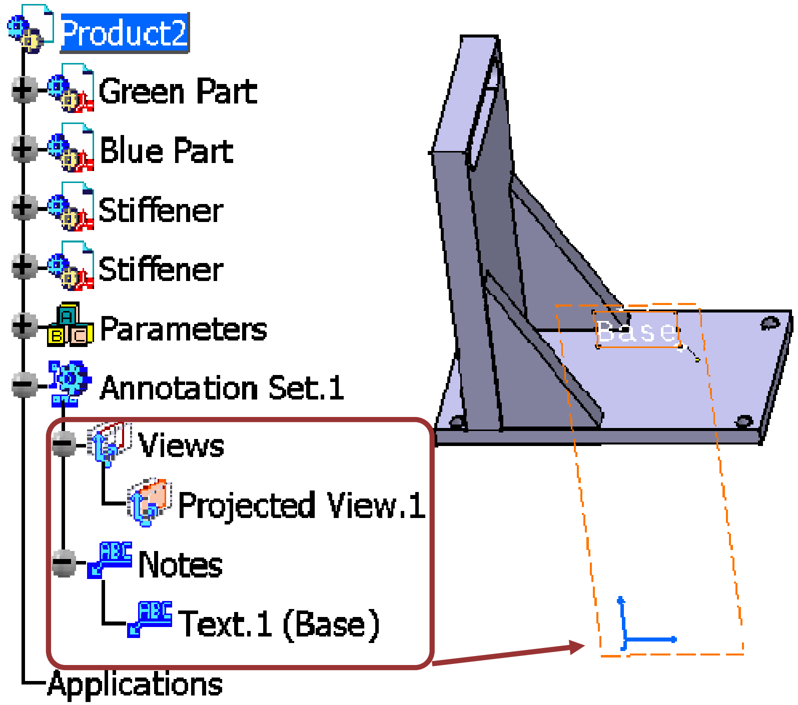


Student Notes:

What Are Text Annotations ?

Text annotations are text that you can see in the 3D view. You can edit and modify them when you want. Text annotations are associated with a geometric element of a component in the assembly.





The Text Annotation icon and the associate dialog box

Associated Features

Student Notes:

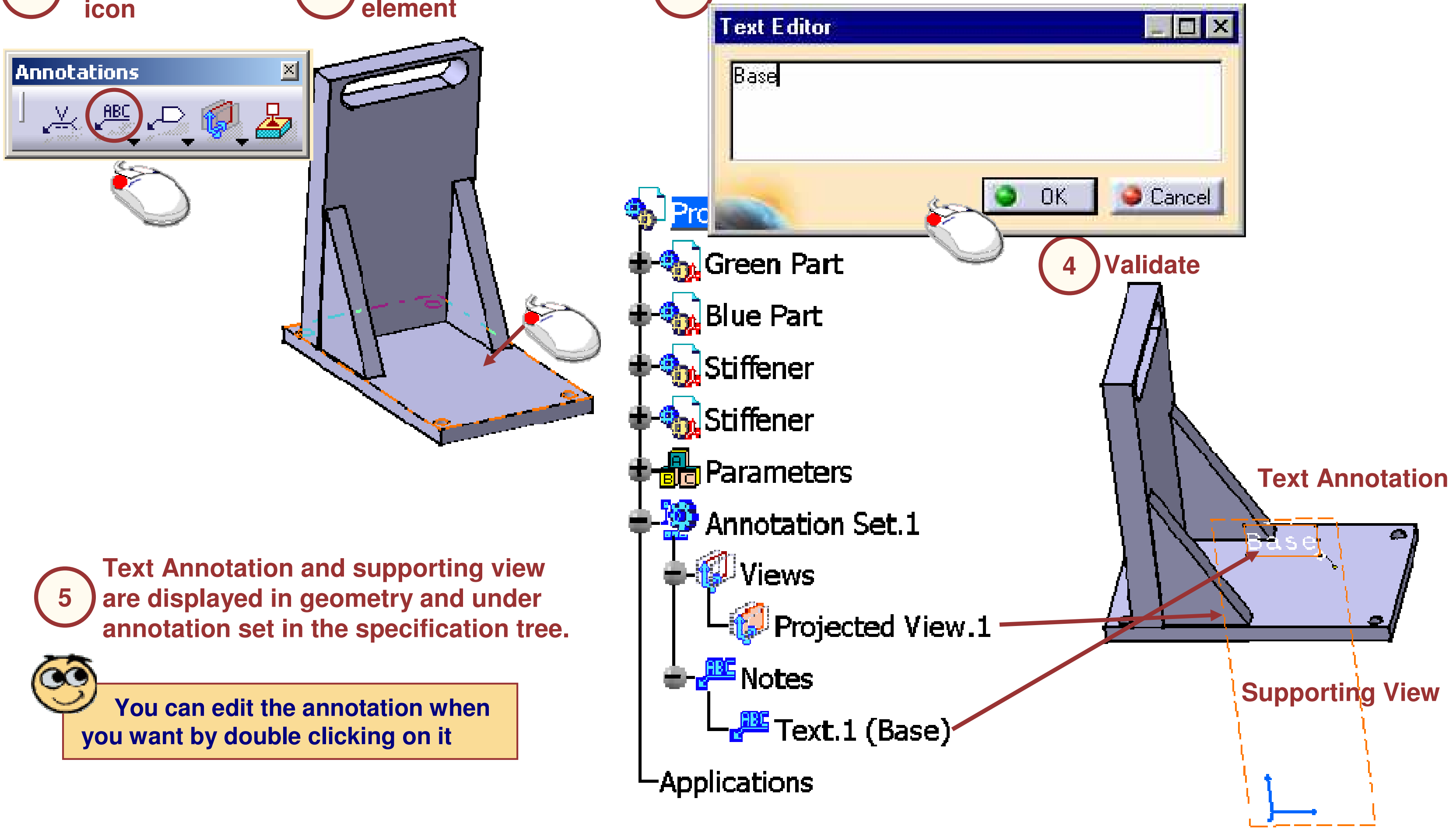
Creating Text Annotations

When you create a text annotation, you implicitly use a FD&T view that will be created if not already existing

1 Click on Text icon

2 Select a geometric element

3 Key Text



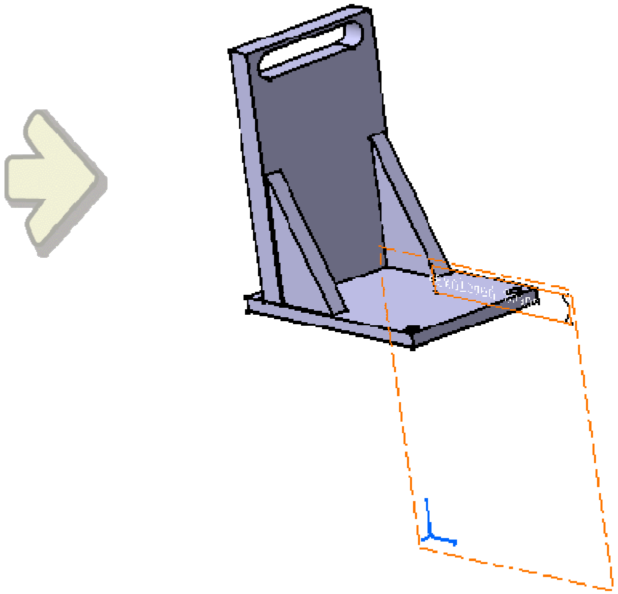
5 Text Annotation and supporting view are displayed in geometry and under annotation set in the specification tree.

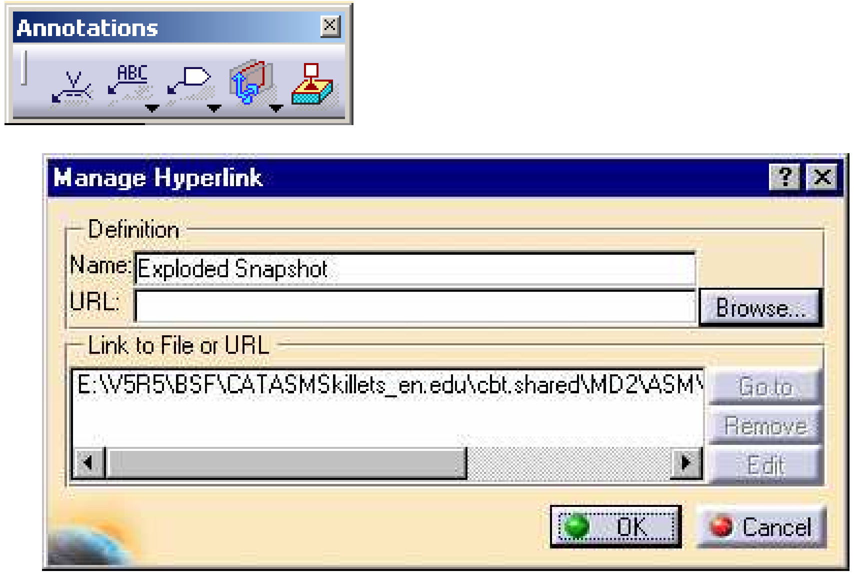
You can edit the annotation when you want by double clicking on it

Student Notes:

Flag Note

You will see how to add flag notes on geometric elements of your assembly that will call other electronic files (hyperlink)





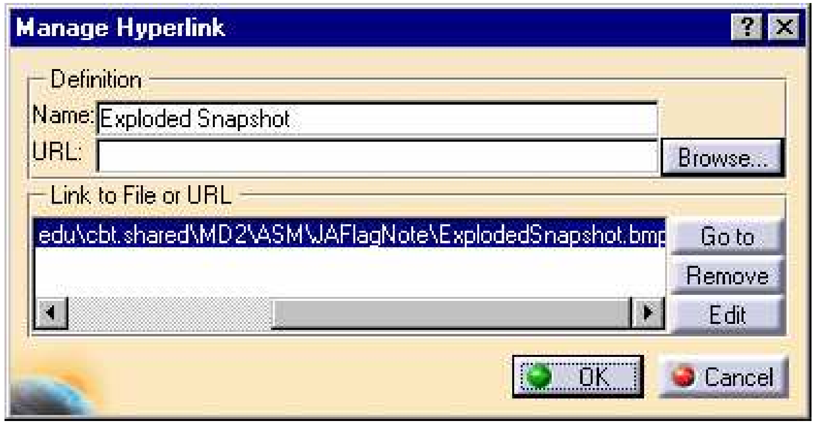
Student Notes:

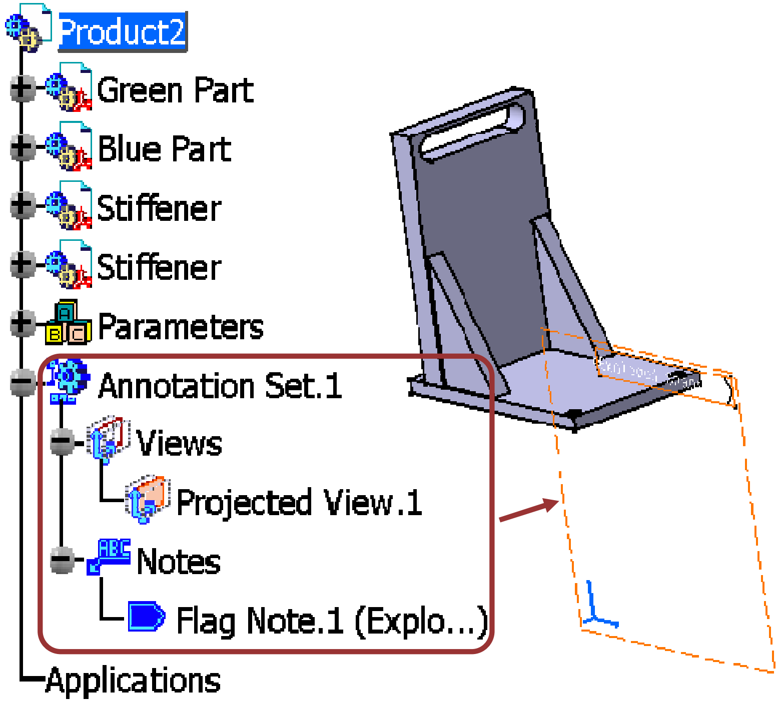
What Are Flag Notes ?

You can add hyperlinks to your document and then use them to jump to a variety of locations, for example to a marketing presentation, a Microsoft Excel spreadsheet or a HTML page on the intranet.

You can add hyperlinks to models, products and parts as well as to any constituent elements.

Flag note tool





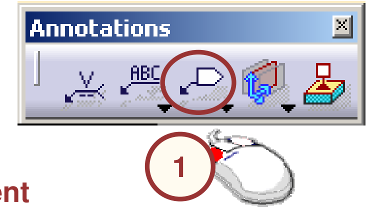
Dialog Box

Associated Features

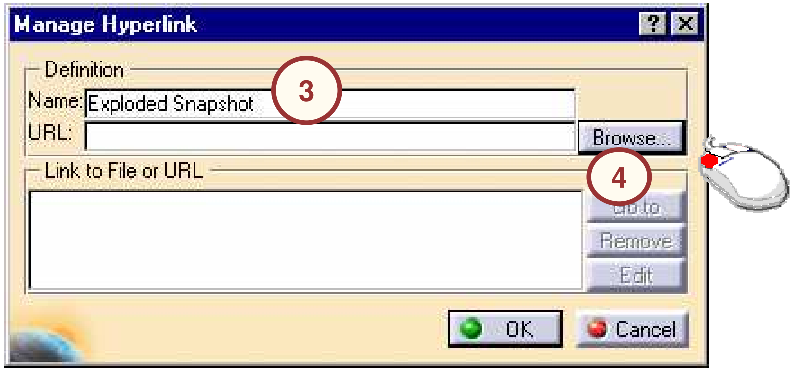
Student Notes:

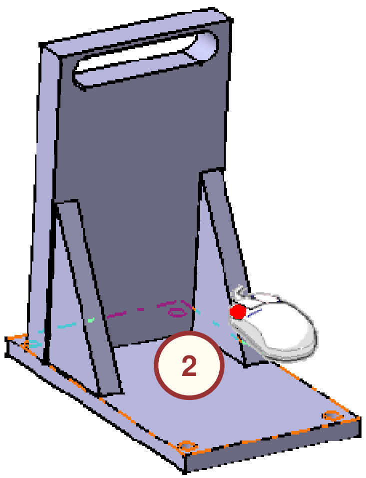
Creating Flag Note (1/2)

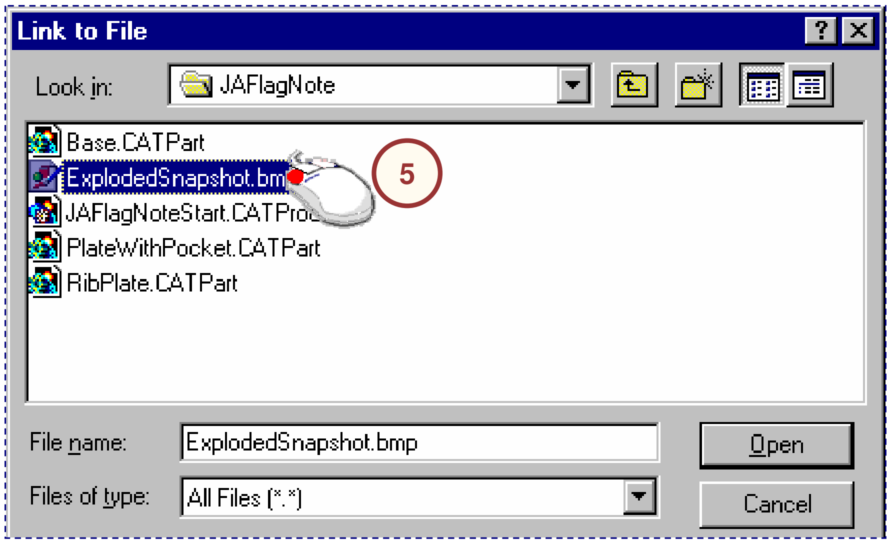
When you create a flag note, you implicitly use a FD&T view that will be created if not already existing.



1. Click on Flag note icon.
2. Select a geometric element
3. Key text that will appear on flag
4. Click on Browse to select a file to link.
5. Double click on the file you want to attach.







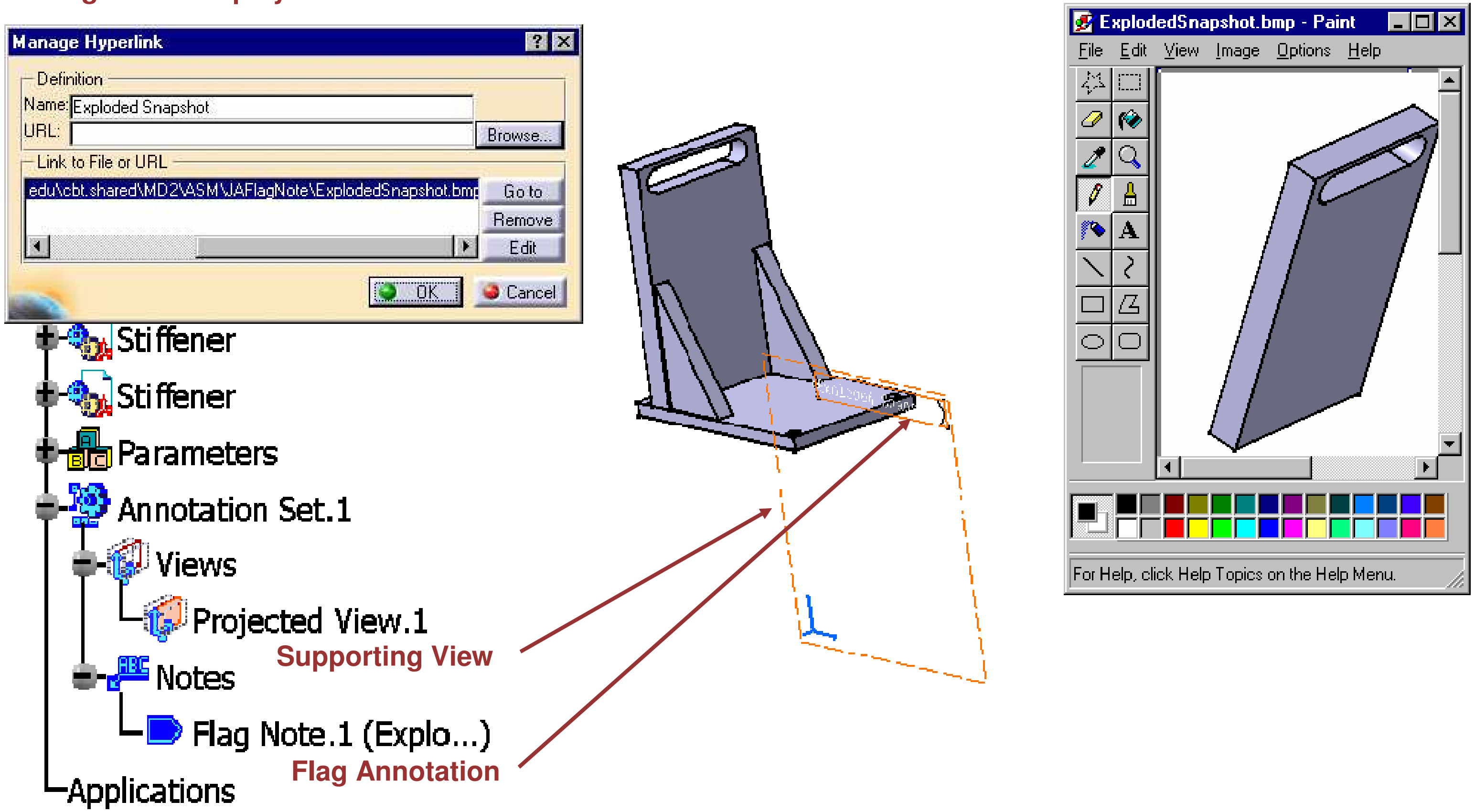
Student Notes:

Creating Flag Note (2/2)

When consulting the linked file, you implicitly launch the application that can read it.

6 When you double click on the flag note (in the specification tree or in the 3D view), the dialog box is displayed.

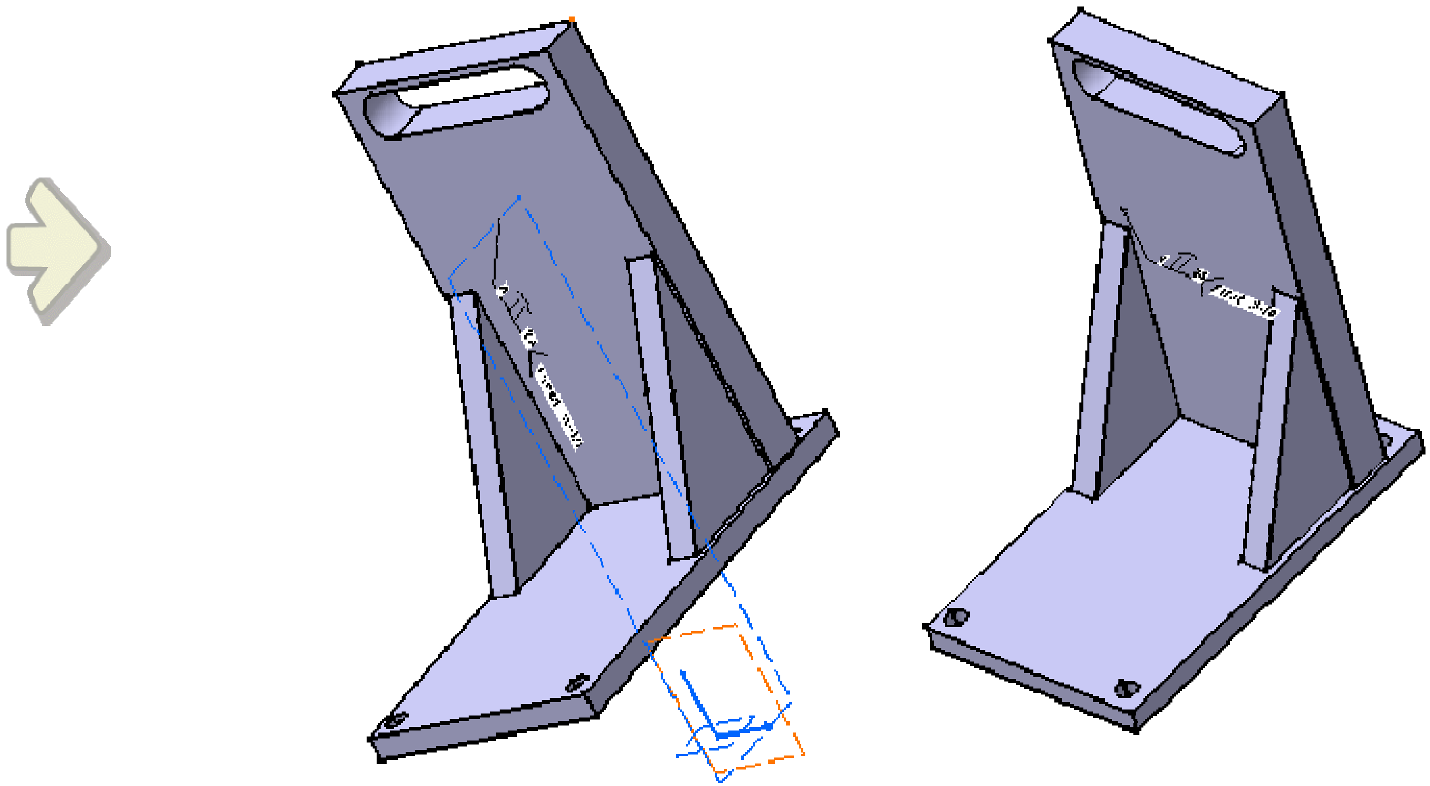
7 Thus you can edit the linked file by a double click or by the "Go to" button



Student Notes:

Manipulating Annotations

You will see how to change annotation's supporting view and how to change leader shapes. You will even see how to project annotation views on drafting.

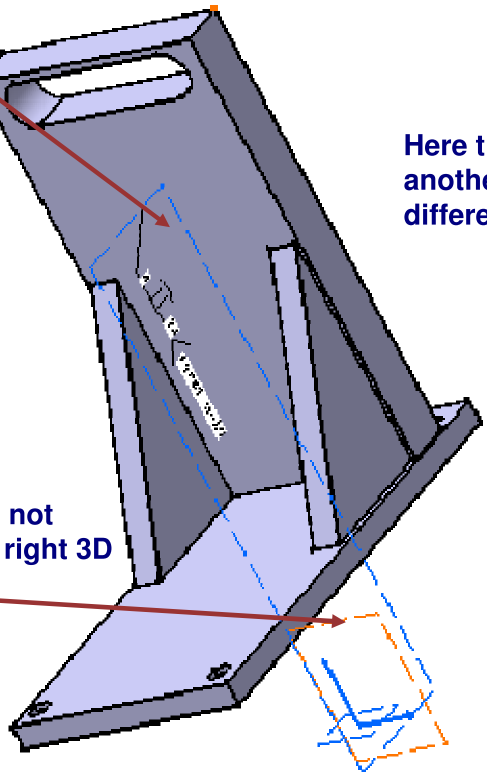


Student Notes:

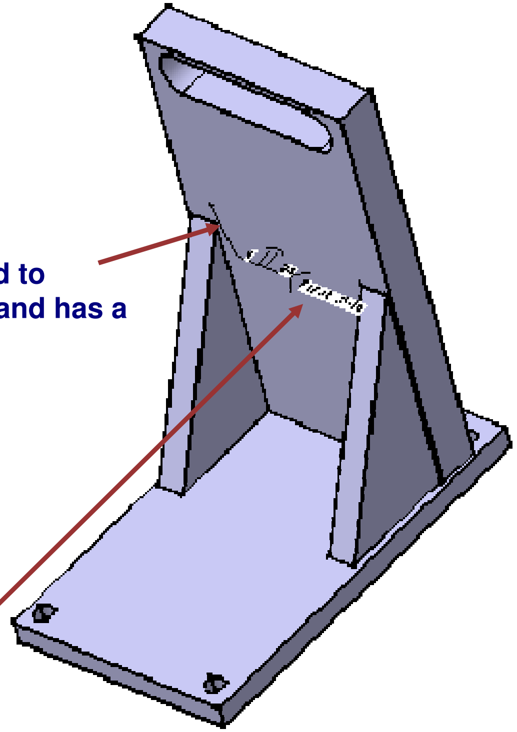
What is Manipulating Annotations ?

You can change the FD&T 3D view support of an annotation, and you can add or remove leader of annotation. You can even modify their symbol shape .

We would like to replace this leader by another one snapping the right geometric element



Here the leader is connected to another geometric element and has a different symbol shape



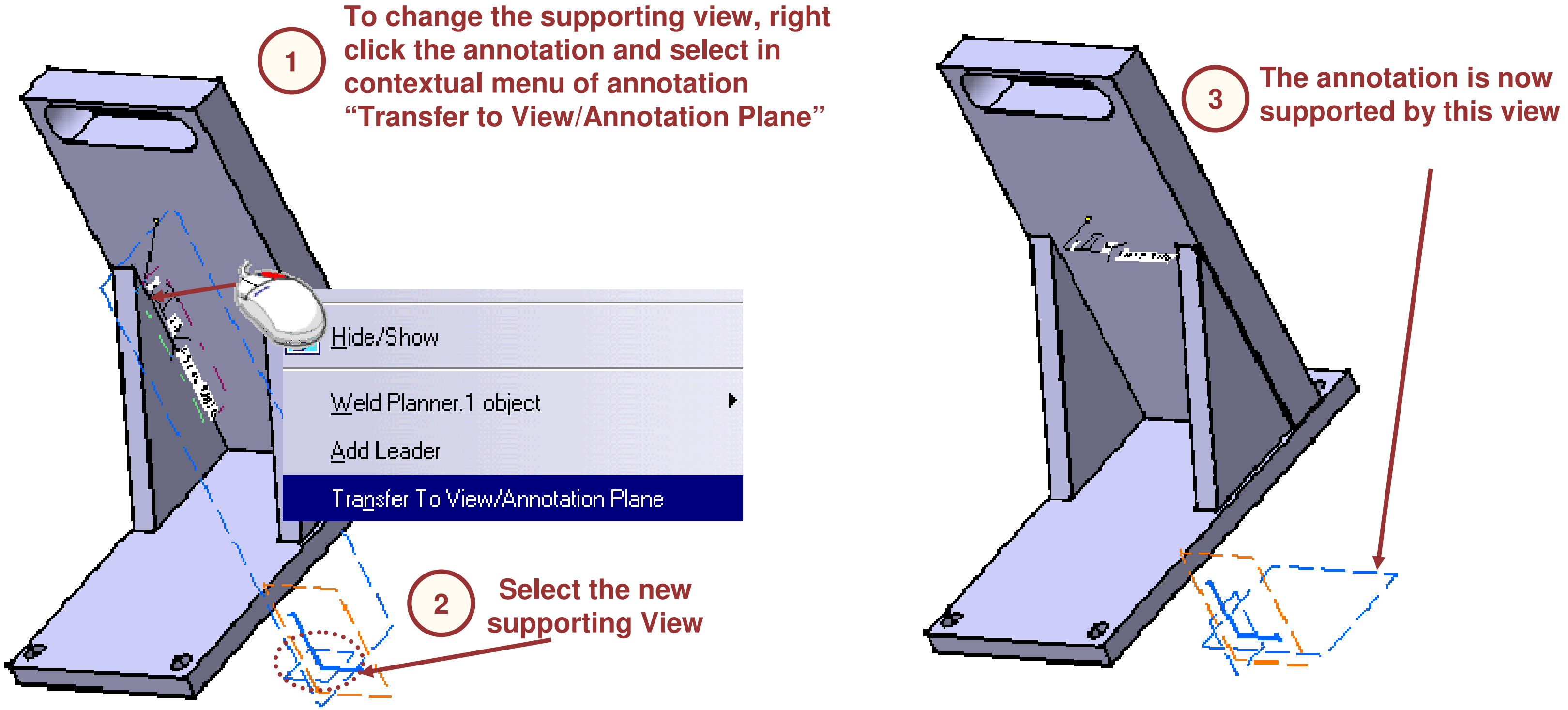
The annotation is not supported by the right 3D view

Here the annotation is differently oriented because supported by another FD&T 3D View.

Student Notes:

Changing Annotation Supporting View

CATIA choose implicitly a supporting view when creating a Weld Planner but you may not be satisfied with this choice, so you can change it.



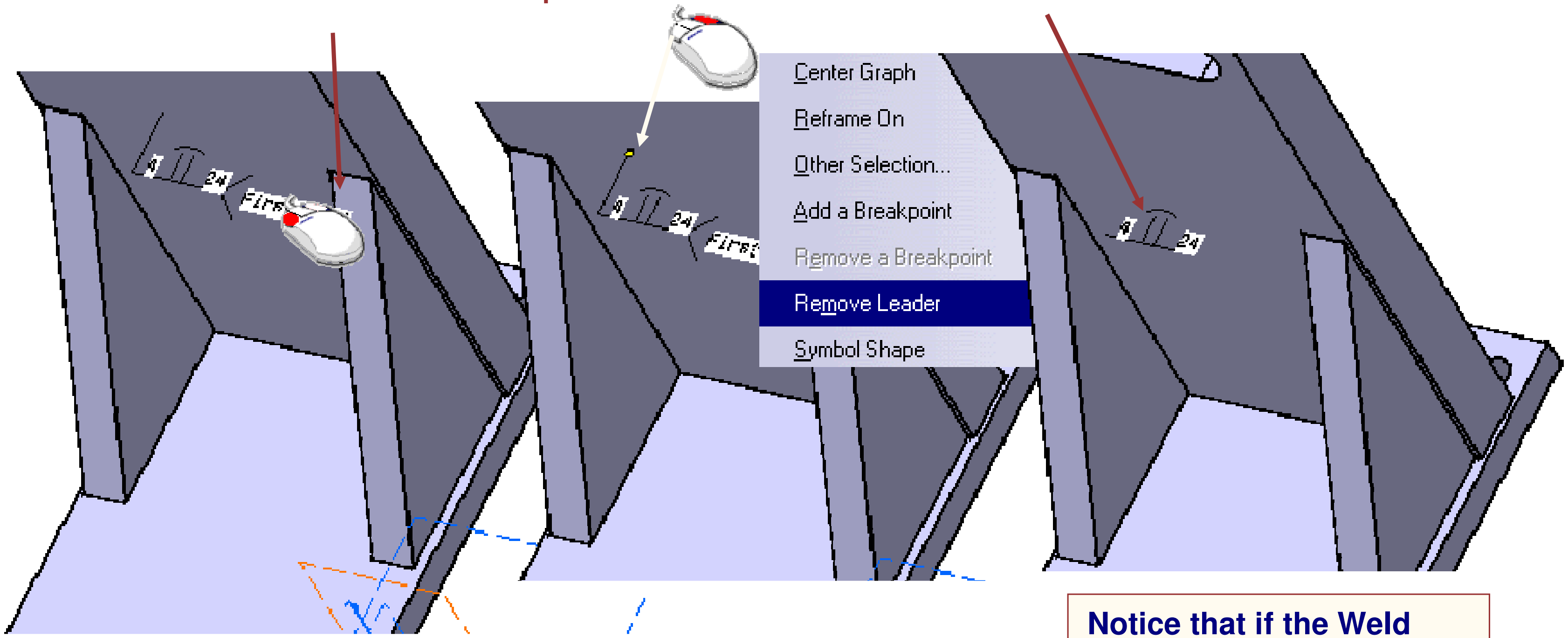
Student Notes:

Manipulating Annotation Leaders (1/3)

CATIA puts automatically a leader on your weld planner feature but you may not be satisfied with the anchor point.

To delete a leader...

1. Select the annotation
2. Select "Remove Leader" in contextual menu of the yellow point
3. The leader has disappeared



Notice that if the Weld Planner feature has not any leader, the Weld process won't appear either

Student Notes:

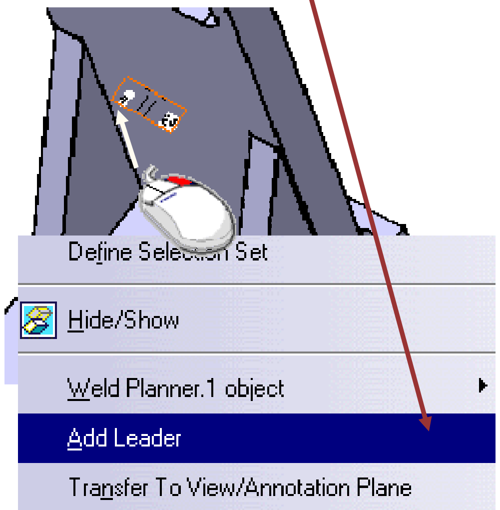
Manipulating Annotation Leaders (2/3)

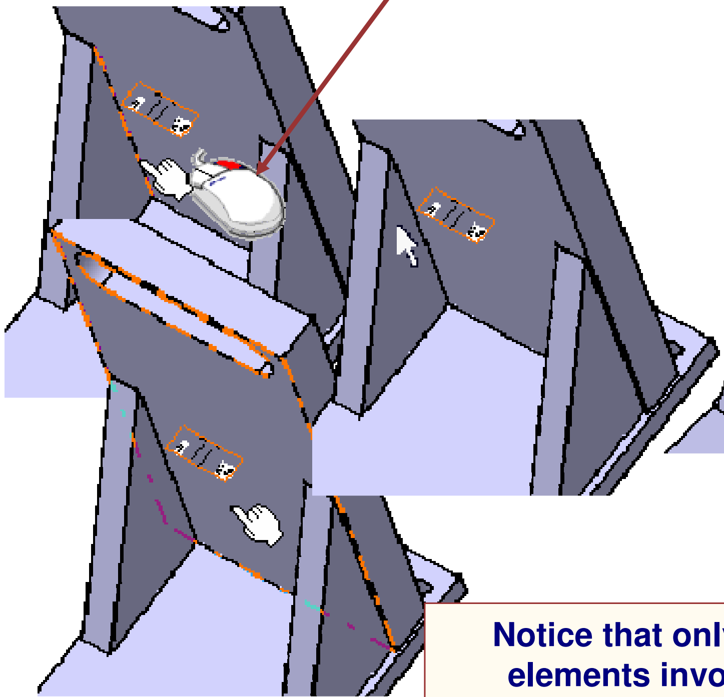
To add a leader...

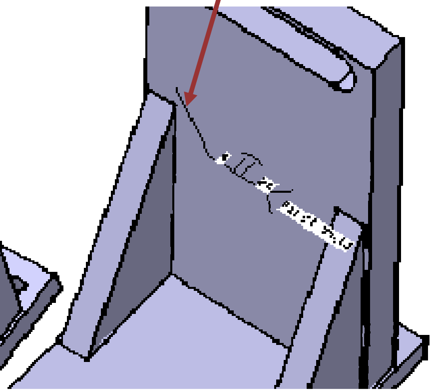
1 Select "Add Leader" in contextual menu of the Weld Planner Feature

2 Click the anchor point leader in the geometry

3 The new leader has appeared







Notice that only geometric elements involved in the annotation are selectable

Student Notes:

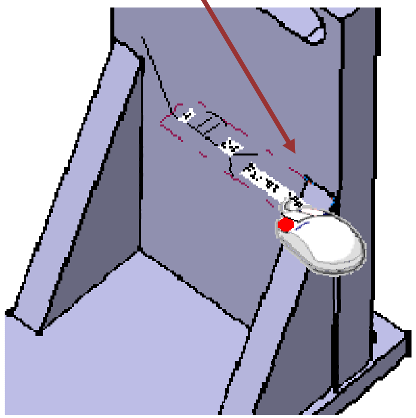
Manipulating Annotation Leaders (3/3)

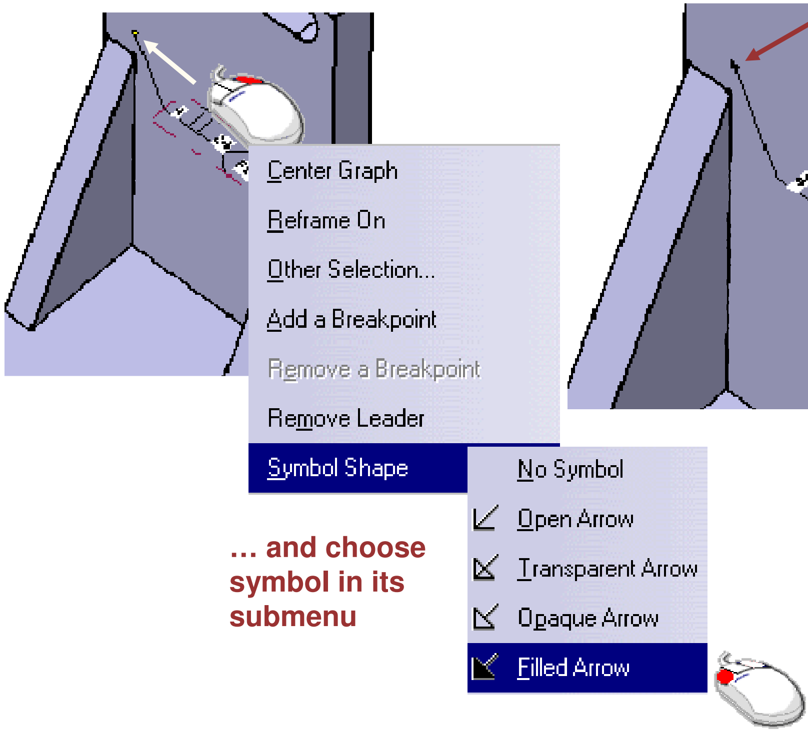
To define Symbol Shape of a leader...

1 Select the weld planner feature

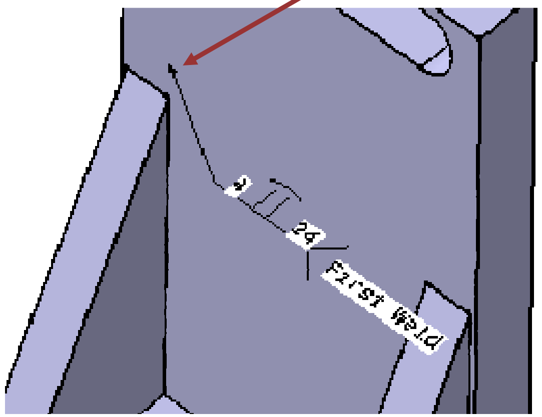
2 Select “Symbol Shape” in contextual menu of the yellow anchor …

3 The leader has a new shape





… and choose symbol in its submenu

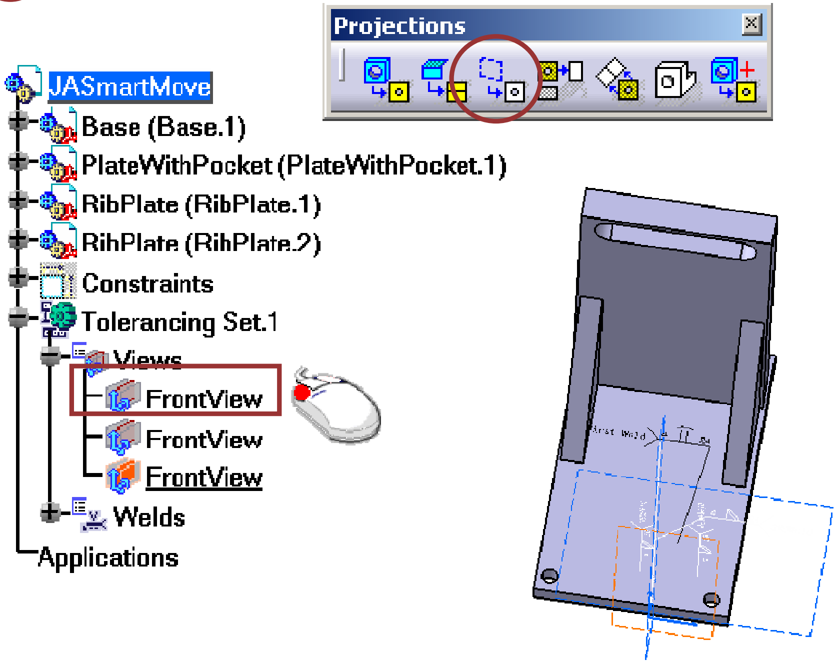


Student Notes:

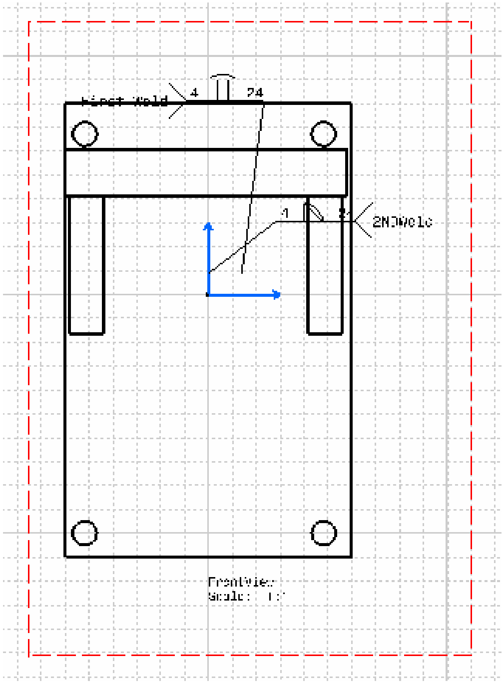
Projecting Annotation Views On a Drafting

You can dispose of FD&T 3D views in CATDrawing files

1 In the Drafting Workbench, select “View From 3D”



3 The View is now on the Drafting and displays annotations



2 Select one of the 3D views in the specification tree or in 3D window

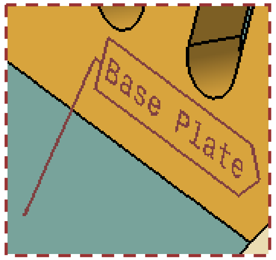
Take care of having same Standards (ISO,ANSI) in 3D and 2D views otherwise you won’t be able to add a view from 3D (default Standard in 3D Views is ANSI).

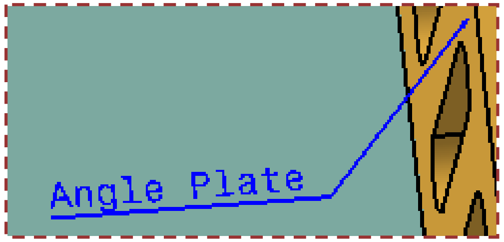
Student Notes:

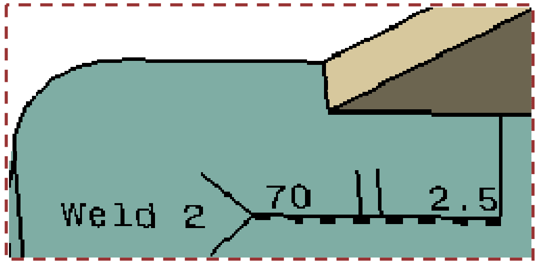
To Sum up …

In this lesson you have learned how to generate and manipulate annotations You have seen how to add the three types of annotations to an assembly :

- Weld Annotations : to add weld symbols and notations
- Text with Leader : to additional information like name, property.
- Flag Note with Leader : to add links to jump to other documents, models, parts







Student Notes:

Generating Reports

You will see how to generate Bill of Material and Assembly Listing Reports.

- Introduction to Generating Reports
- Bill of Material Reports
- Assembly Listing Reports
- Generating Reports: Recap Exercise
- To Sum Up

Student Notes:

Introduction to Generating Reports

To have a more information about an assembly or product it will be useful to generate some reports.

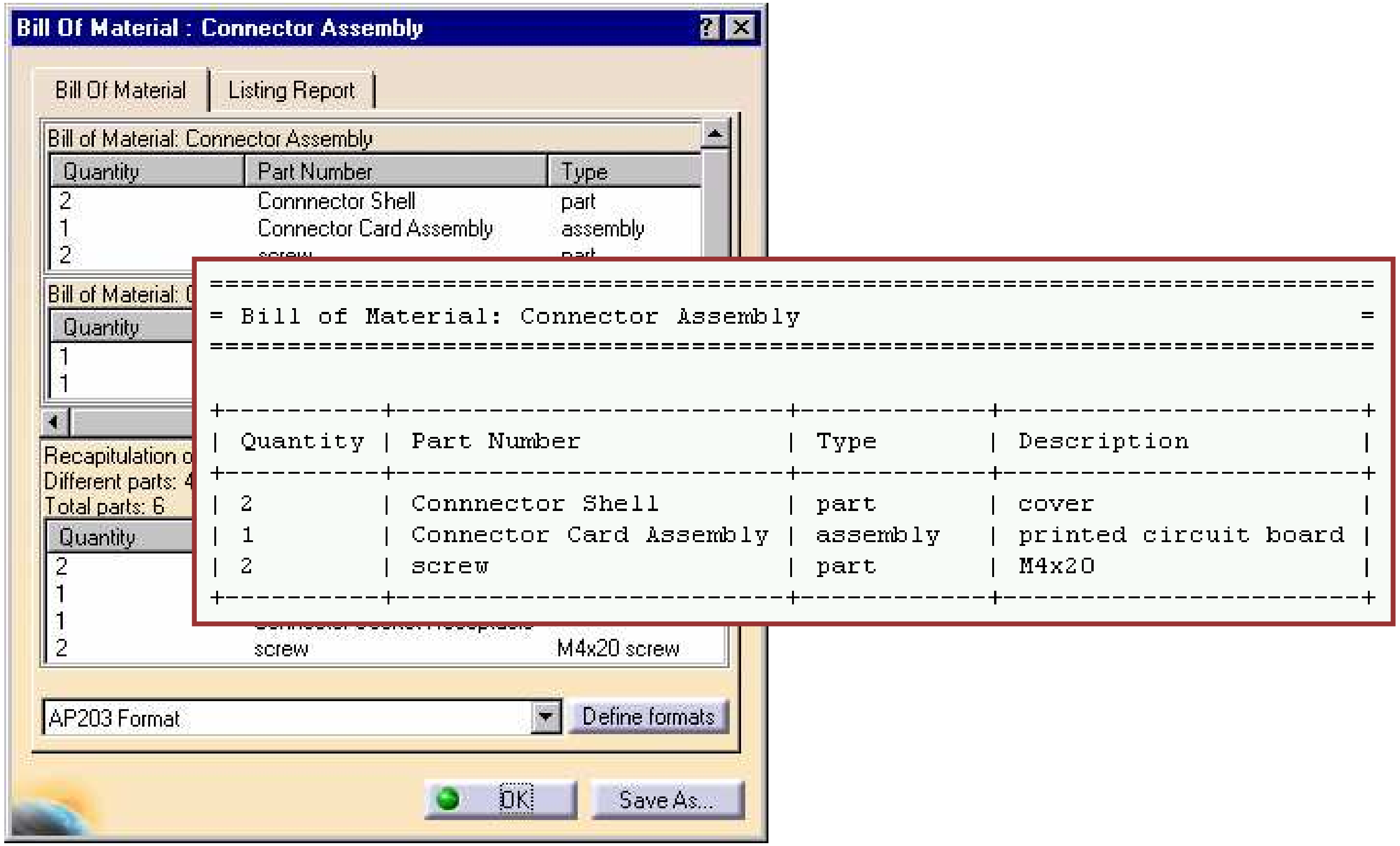
Bill Of Material can be created through CATIA V5; A way to manage the product structure is managing the BOM (Bill of Material).

An Assembly Listing Report can also be created to "list" the components belonging to the CATProduct.

Student Notes:

Bill Of Material Reports

You will see how to generate Bill of Material Reports



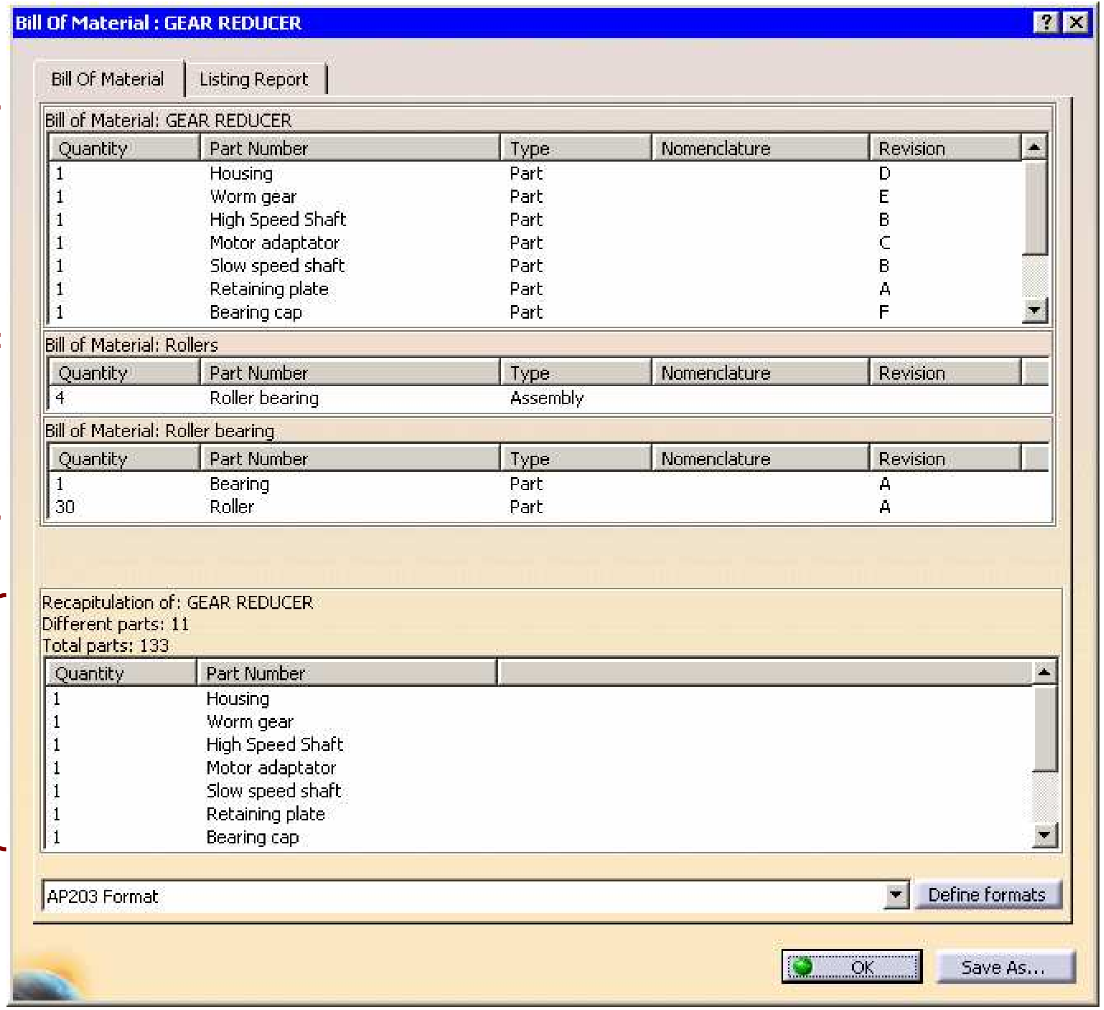
What Are Bill Of Material Reports?

Bill of Material Reports list all the components of an assembly. Components are listed by part number and the quantity of each part number is computed.

BOM with the components of the active assembly

One BOM for each sub-assembly

Recap of leaf components from the active assembly and all sub-assemblies



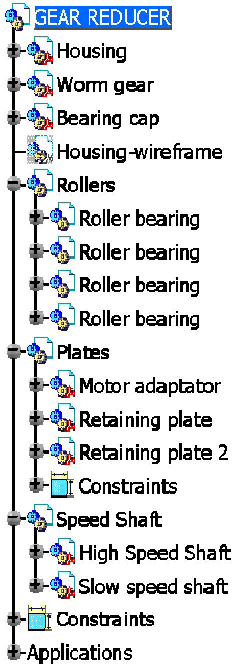
Student Notes:

Student Notes:

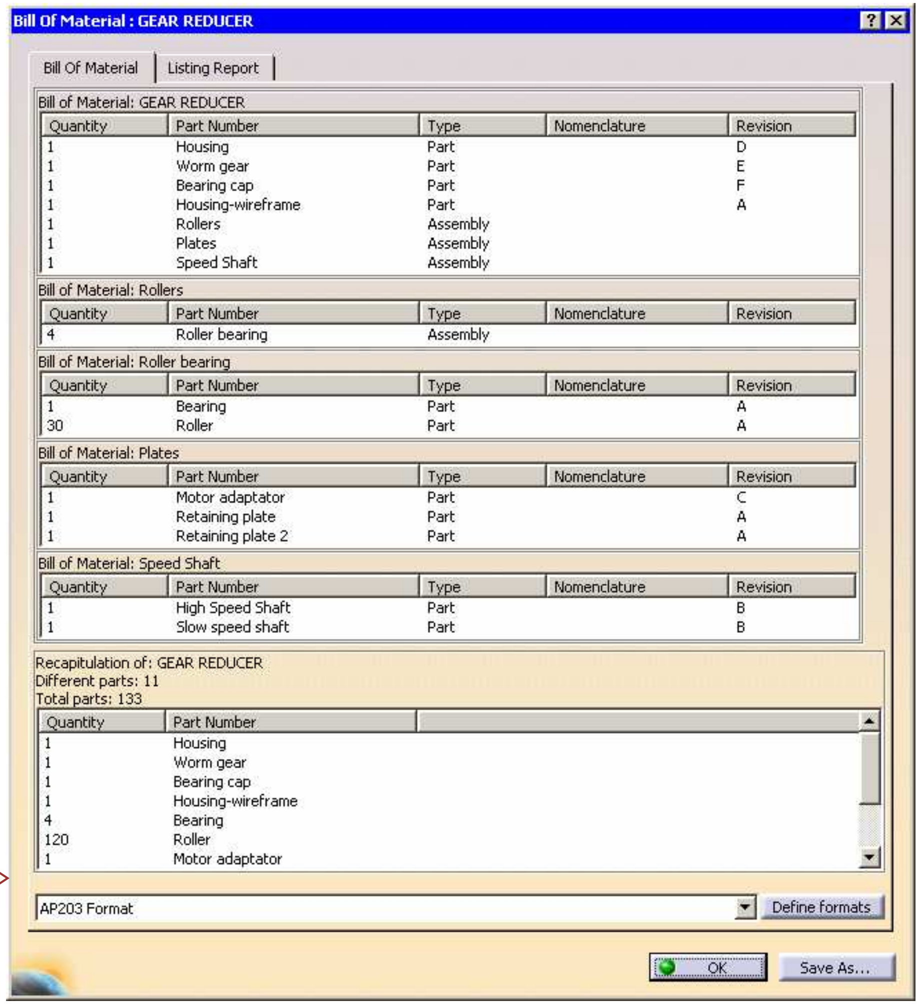
Generating Bill Of Material Reports

Bill of Material Reports can be interactively generated and viewed.

1 Activate the assembly or sub-assembly to generate the BOM



2 Analyze + Bill of Material



Bill of Material: GEAR REDUCER

Quantity	Part Number	Type	Nomenclature	Revision
1	Housing	Part		D
1	Worm gear	Part		E
1	Bearing cap	Part		F
1	Housing-wireframe	Part		A
1	Rollers	Assembly		
1	Plates	Assembly		
1	Speed Shaft	Assembly		

Bill of Material: Rollers

Quantity	Part Number	Type	Nomenclature	Revision
4	Roller bearing	Assembly		

Bill of Material: Roller bearing

Quantity	Part Number	Type	Nomenclature	Revision
1	Bearing	Part		A
30	Roller	Part		A

Bill of Material: Plates

Quantity	Part Number	Type	Nomenclature	Revision
1	Motor adaptator	Part		C
1	Retaining plate	Part		A
1	Retaining plate 2	Part		A

Bill of Material: Speed Shaft

Quantity	Part Number	Type	Nomenclature	Revision
1	High Speed Shaft	Part		B
1	Slow speed shaft	Part		B

Recapitulation of: GEAR REDUCER
Different parts: 11
Total parts: 133

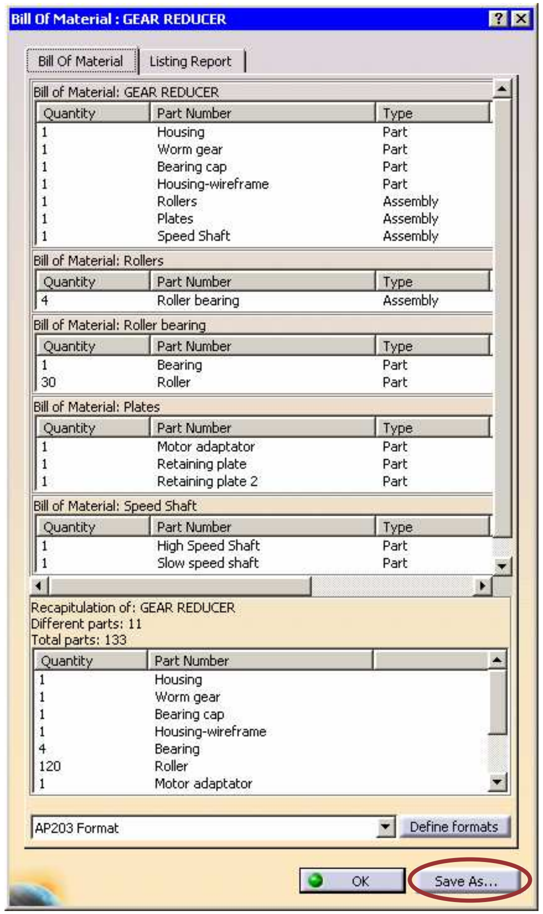
Quantity	Part Number
1	Housing
1	Worm gear
1	Bearing cap
1	Housing-wireframe
4	Bearing
120	Roller
1	Motor adaptator

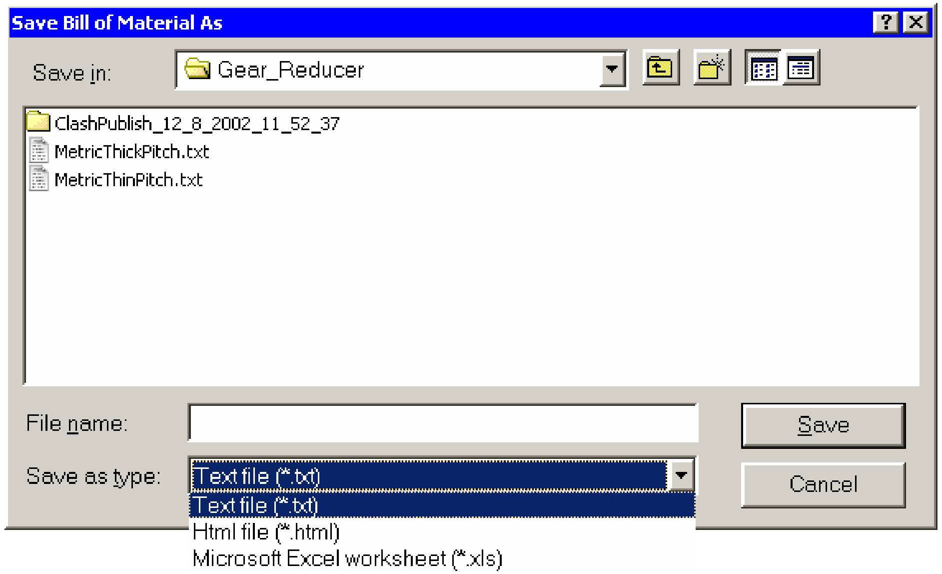
Student Notes:

Saving Bill Of Material Reports

You can export the bill of material reports to several formats that can be viewed outside CATIA.

1 To export the BOM click Save As ...





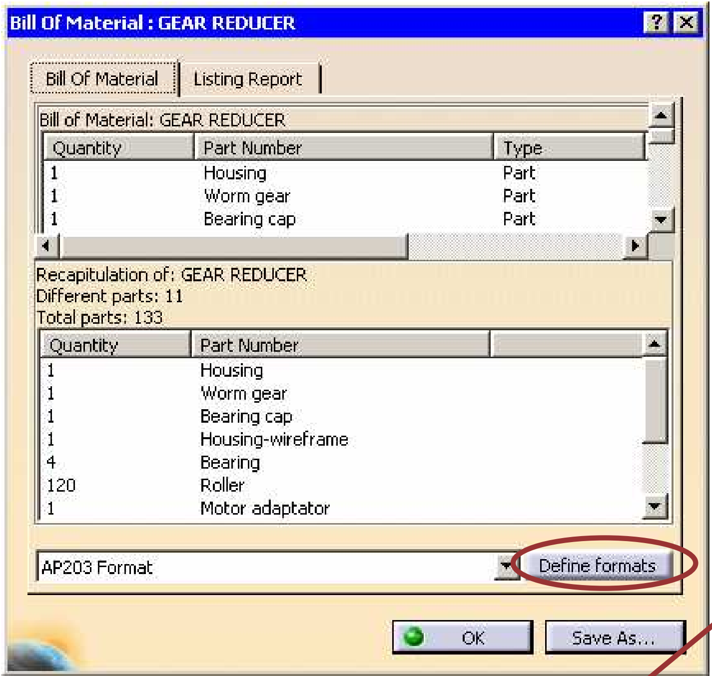
2 Select a format and specify a file name

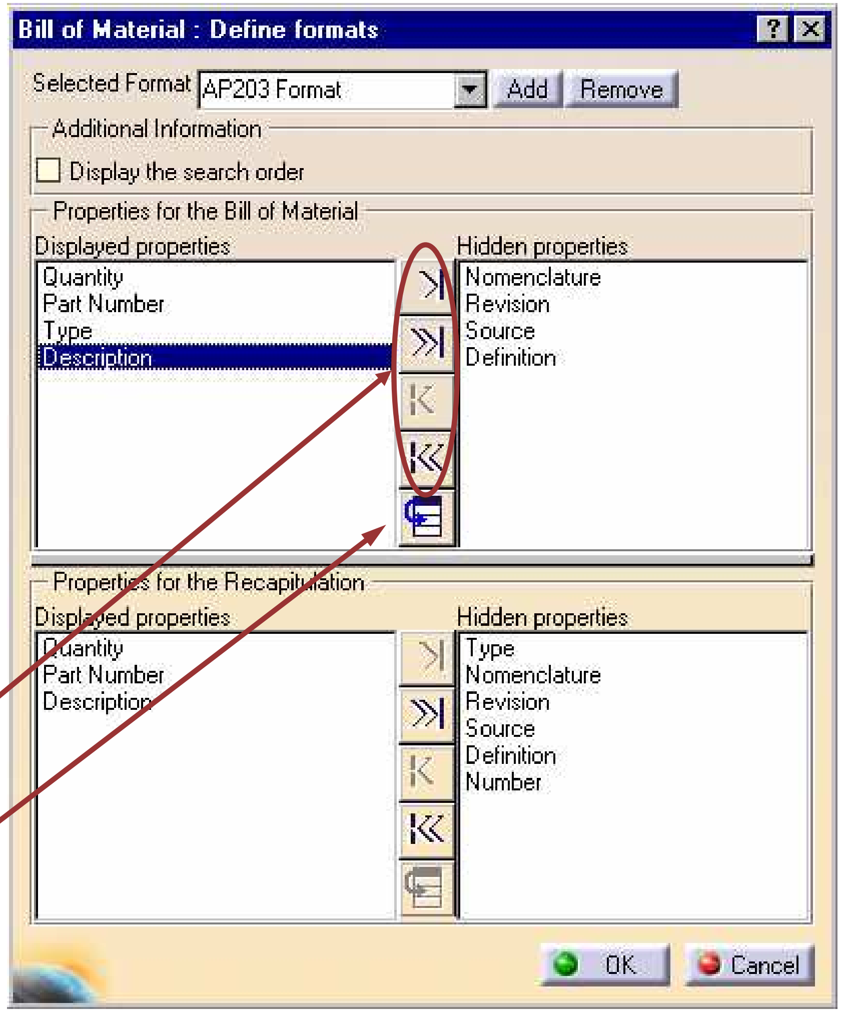
Student Notes:

Customizing Bill Of Material Reports

You can customize the report of the BOM to display the properties of your choice and their order in the list.

1 Click on Define formats





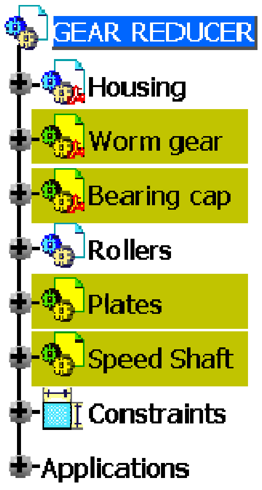
2 Specify properties to be displayed

3 Arrange order of properties

Student Notes:

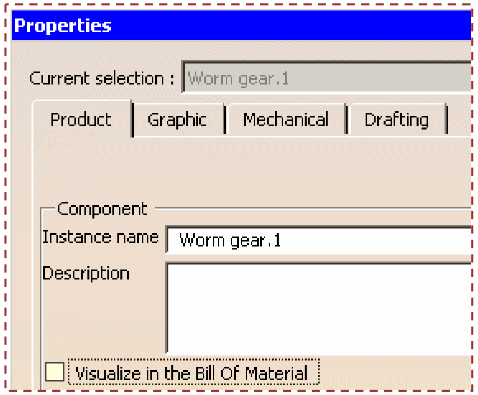
Removing a Component From the BOM

If you do not want to list in the BOM some products or components, you have to modify their properties.

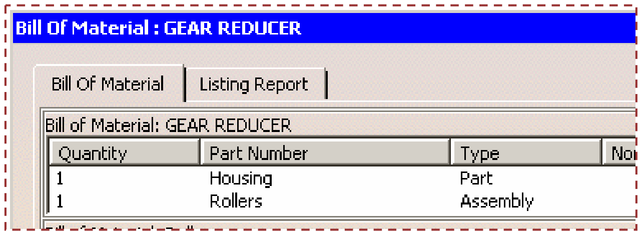


This assembly contains 3 products and 3 parts listed in the BOM

Using the contextual menu select properties and deactivate "Visualize in the Bill Of Material".



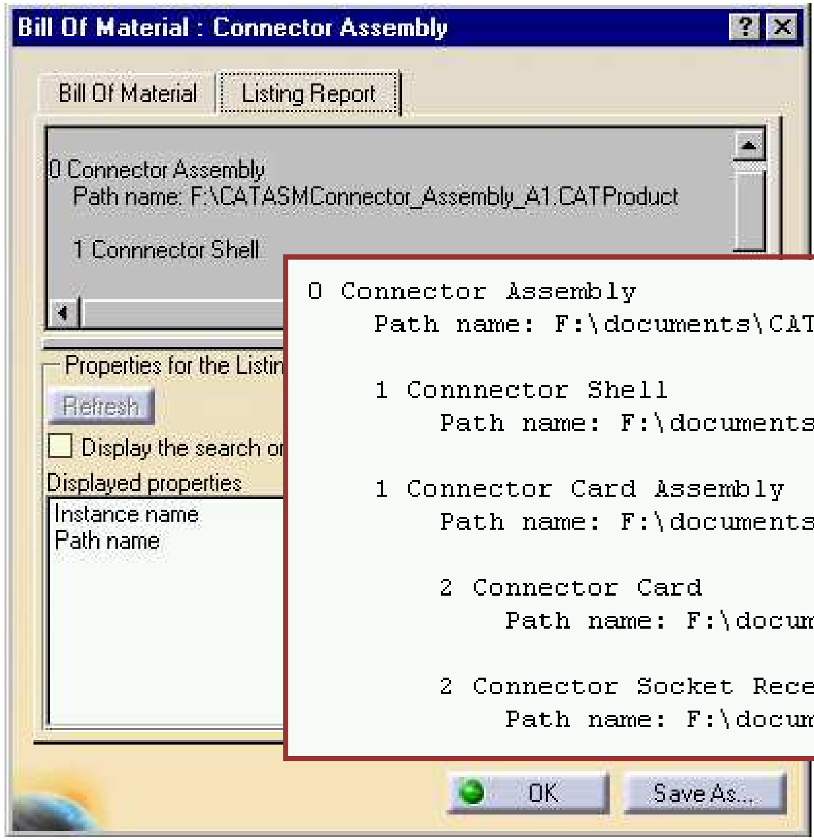
The products and parts paint in yellow have their properties modified. Only the part "Housing" and the product "Rollers" are listed.



Student Notes:

Assembly Listing Reports

You will see how to generate Assembly Listing Reports



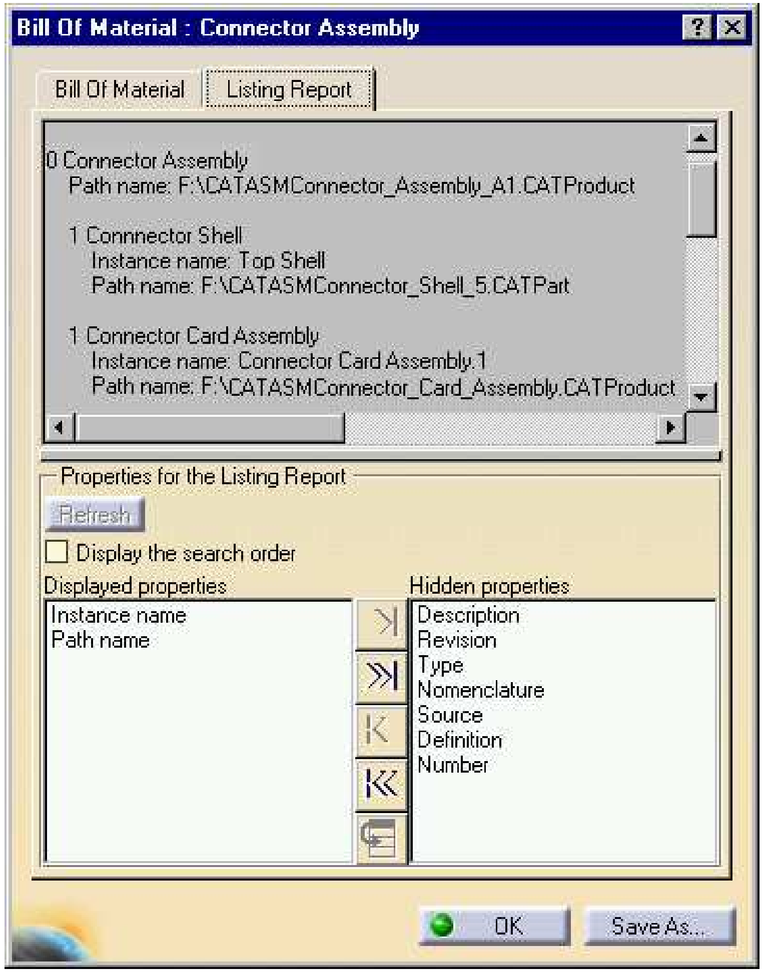

```
0 Connector Assembly
    Path name: F:\documents\CATASMConnector_Assembly_A1.CATProduct

    1 Connnector Shell
        Path name: F:\documents\CATASMConnector_Shell_5.CATPart

    1 Connector Card Assembly
        Path name: F:\documents\CATASMConnector_Card_Assembly.CATProduct

        2 Connector Card
            Path name: F:\documents\CATASMConnector_Card.CATPart

        2 Connector Socket Receptacle
            Path name: F:\documents\CATASMConnector_Socket_Receptacle.CATPart
```

Student Notes:

What Are Assembly Listing Reports?

Assembly Listing Reports list all the components of assemblies. Components are listed in a hierarchical or tree format.

Hierarchy showing every component from the active assembly and all sub-assemblies

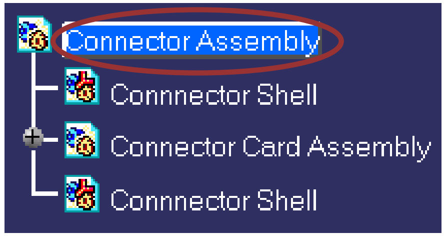


Student Notes:

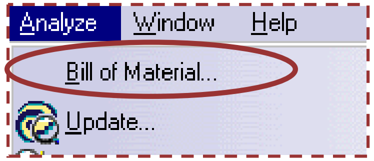
Generating Assembly Listing Reports

You can generate an assembly listing reports that can be interactively generated and viewed.

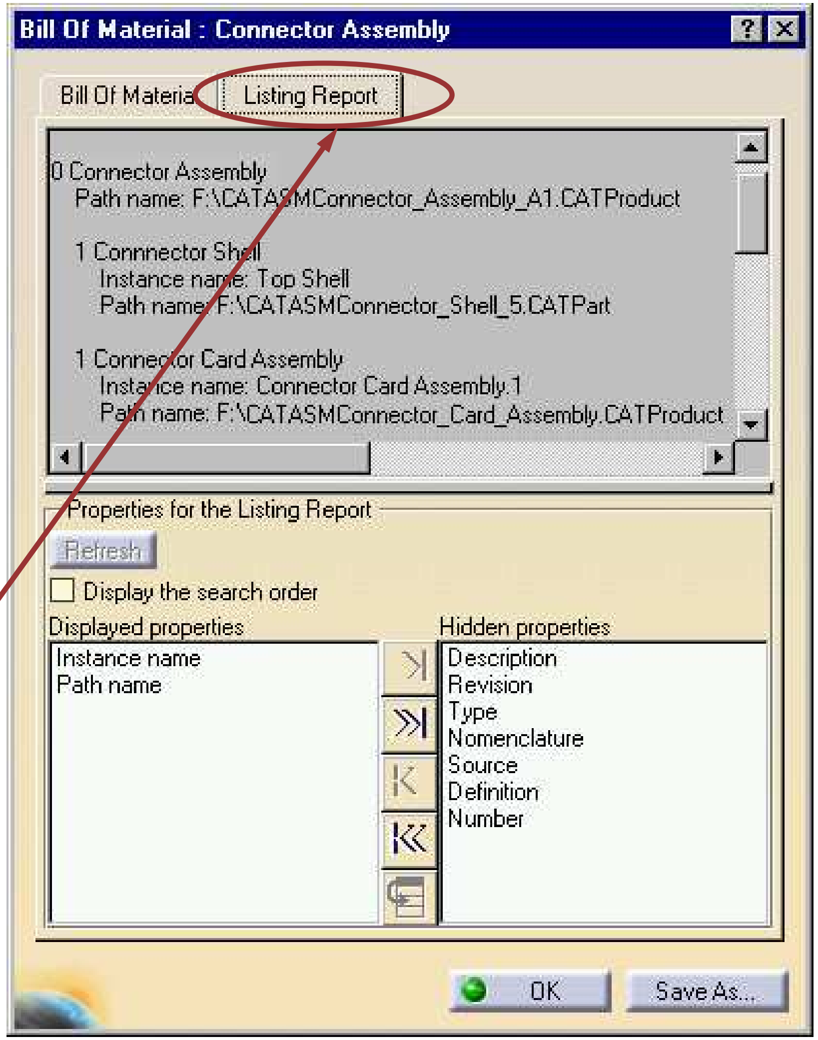
1 **Activate the assembly for which the BOM is to be generated**



2 **Select "Bill of Material" in the Analyze menu**



3 **Select the Listing Report tab and then customize your report for your needs**

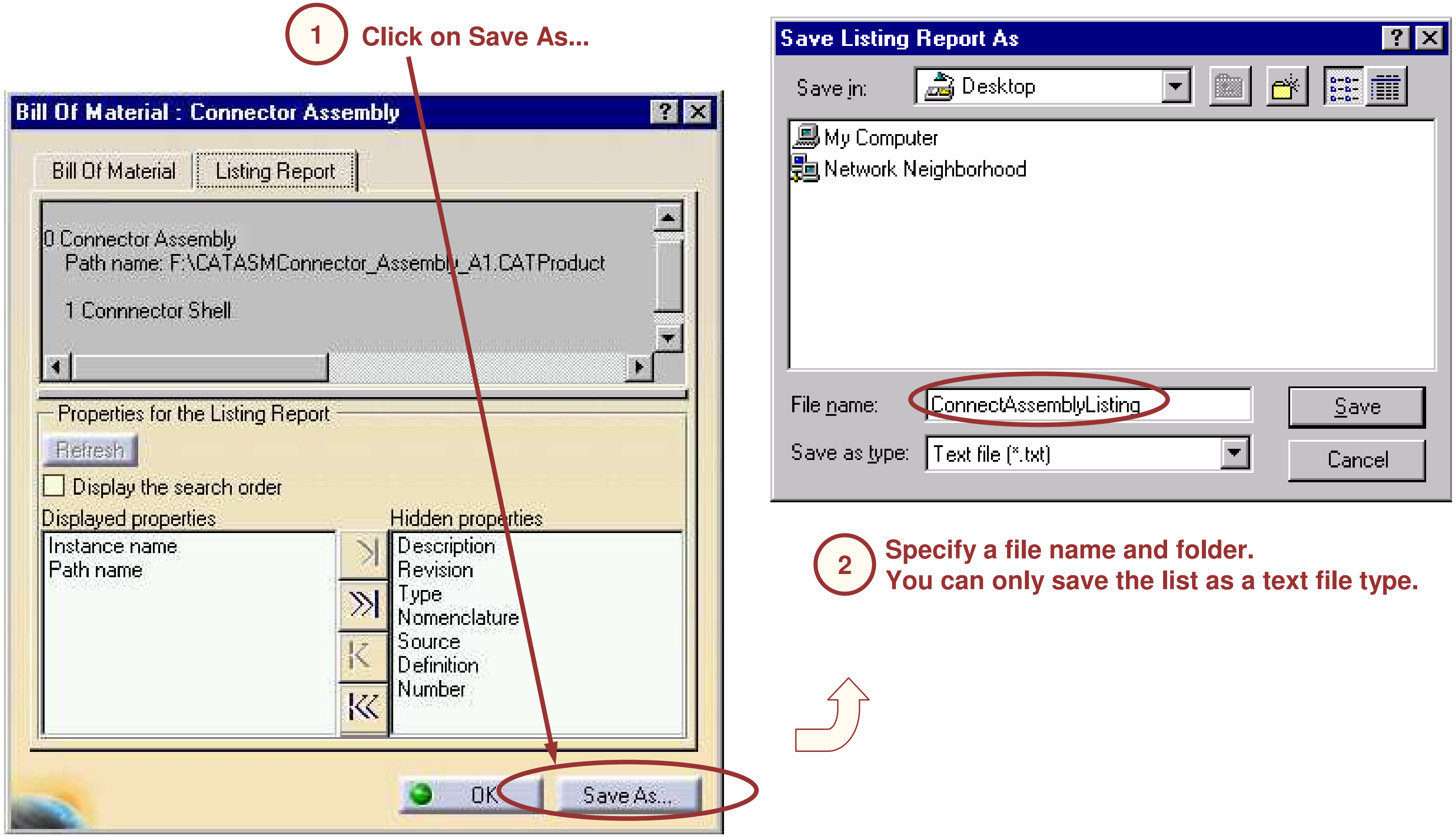


Student Notes:

Saving Assembly Listing Reports

You can export the assembly listing reports to a text file.

1 Click on Save As...



2 Specify a file name and folder.
You can only save the list as a text file type.

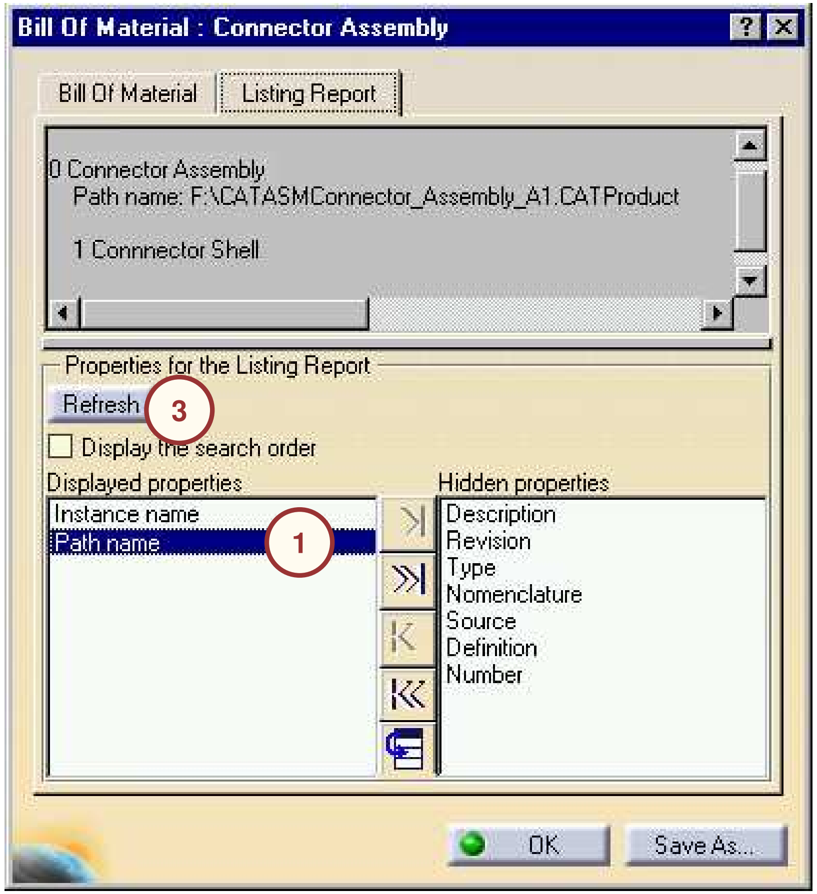
Customizing Assembly Listing Reports

You can customize the Assembly listing report to display the properties of your choice and their order in the list.

1 Specify properties to be displayed

2 Arrange order of properties

3 Click the Refresh button



Student Notes:

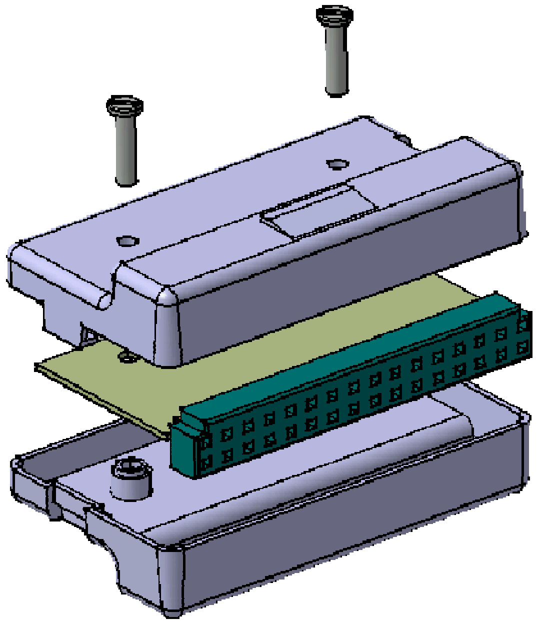
Student Notes:

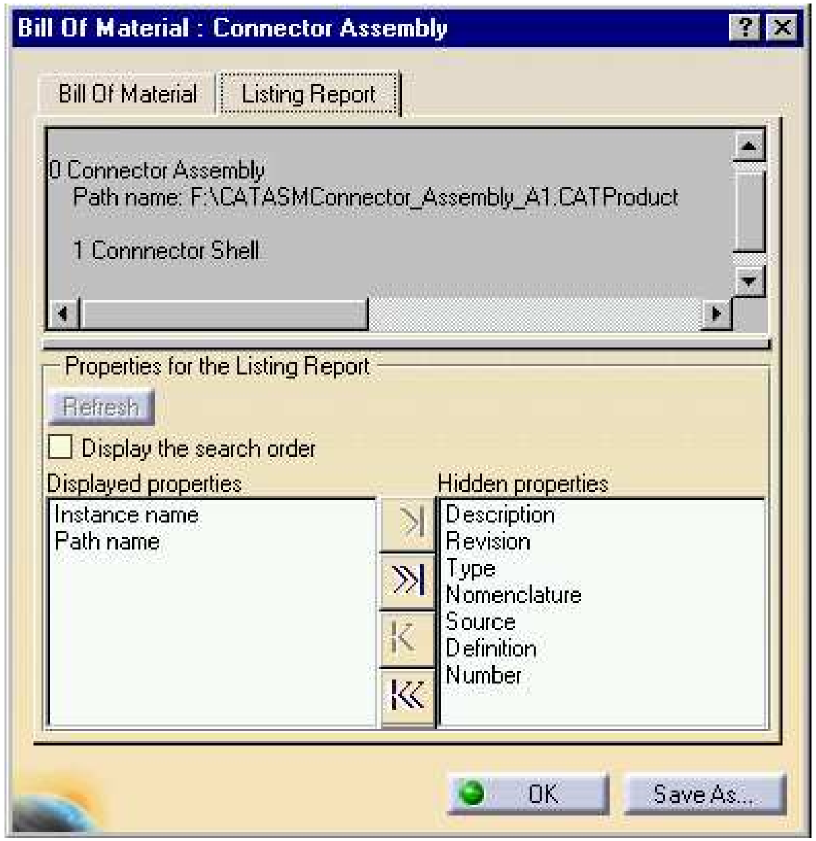
Generating Reports: Recap Exercise

Connector Assembly

15 min

In this exercise you will generate two bill of material and an assembly reports concerning the Connector Assembly

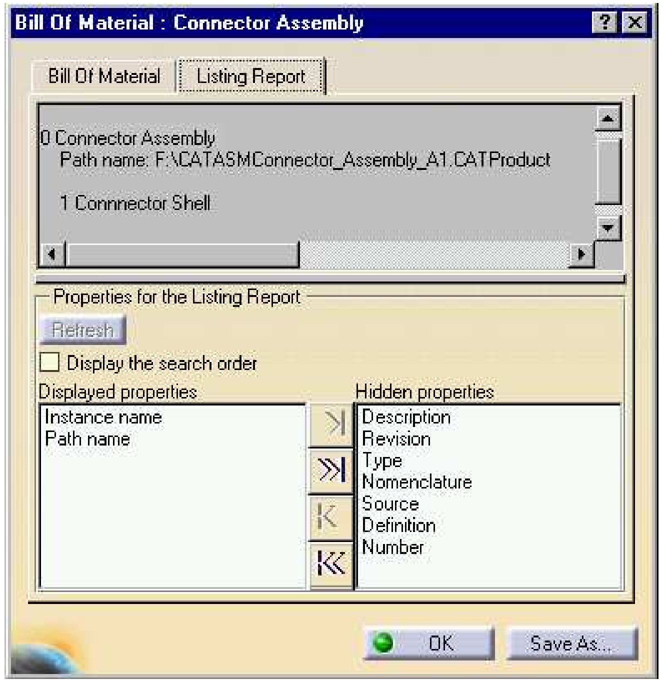


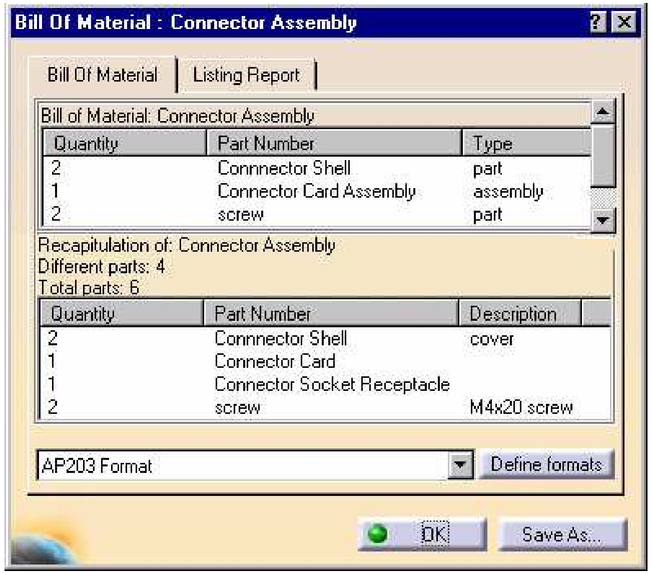


Student Notes:

Do It Yourself

Load document "CATASMConnector_Assembly_A1.CATProduct".





1- Generate a Bill of Material report for the top-most assembly (Connect Assembly).
2- Generate a Bill of Material report for the sub-assembly named Connector Card Assembly.
3- Generate a Listing Report for the top-most assembly. Customize the report to include Instance Name and Path Name.

Student Notes:

To Sum Up

This concludes the lesson on Generating Reports, you should be able to demonstrate:

- **Generating and Managing the Bill of Material**
- **Generating the Assembly Listing Reports**

To Sum Up

In this lesson, you have seen is how to analyze assemblies, generate reports and assembly drawings. You can perform following analysis on assemblies :

- Measure Minimum Distance between components.
- Create and manage component in scenes: Scenes are used to create new positions and apply positions to the main product and create drawings. You can also modify color properties of components, Show / No Show components, move them and deactivate shapes using scenes. You can also create exploded views in scene.
- Compute clearances and clashes in between selected components. Computing clash results in detailed report indicating exactly what features are involved in the clash in order to make corrections.
- Generate Assembly drawings: You can generate assembly drawings from the product and create annotations in the drawings. You can create weld annotations, text with leader and flag with leader. Flags are used to add links to the document and jump to a various locations (URL, Power point presentation, Excel file, Word document).
- Generate Bill of Material and Listing Reports: Bill of Material report lists all the components in the assembly with part number and quantities. Listing report lists all the components of an assembly in the hierarchial form like a tree.

Student Notes:

Course Summary

In the Product Design Expert Course you have seen how CATIA V5 helps you to :

- **Manage Links between various documents of a product.**
- **Design assemblies in a collaborative workspace by using**
  - **Contextual Design**
  - **Publications**
- **Optimize the performance for large assemblies by:**
  - **Managing representations and components of a product**
  - **Using Visualization Mode**
- **Analyze assemblies by:**
  - **Creating scenes and managing components in scene**
  - **Creating sections**
  - **Using assembly measure**
- **Add information in assembly drawings by:**
  - **Using Product Structure Numbering and generating annotations**
  - **Generating Bill of Material and Listing reports**