# Getting Started

If in Sketcher and Part Design you generated parts, now will learn how to finish your design by assembling parts in Assembly Design workbench.

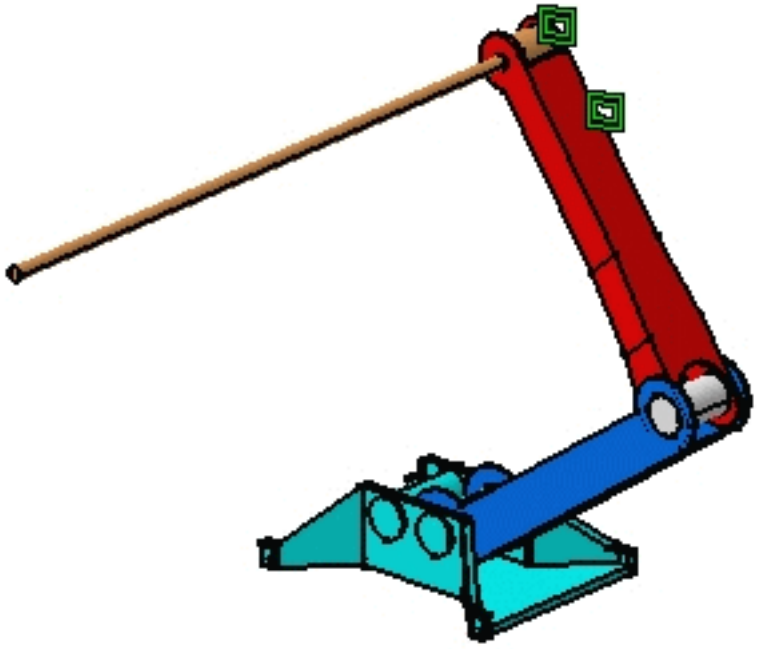
Before we discuss the detailed instructions for using the Assembly workbench, the following scenario aims at giving you a feel for what you can do with an Assembly document. You just need to follow the instructions as you progress.

The Getting Started section is composed of the following tasks:

Entering the Workbench
Fixing a Component
Inserting an Existing Component
Setting Constraints
Moving
Adding and Renaming a New Component
Designing a Part
Editing a Parameter
Replacing a Component
Analyzing Constraints
Reconnecting Constraints
Detecting Clashes
Editing a Component
Bill of Material
Exploding the Assembly

This scenario should take about 15 minutes to complete.

Eventually, the assembly will look like this:



# Entering Assembly Design Workbench and Opening a CATProduct Document

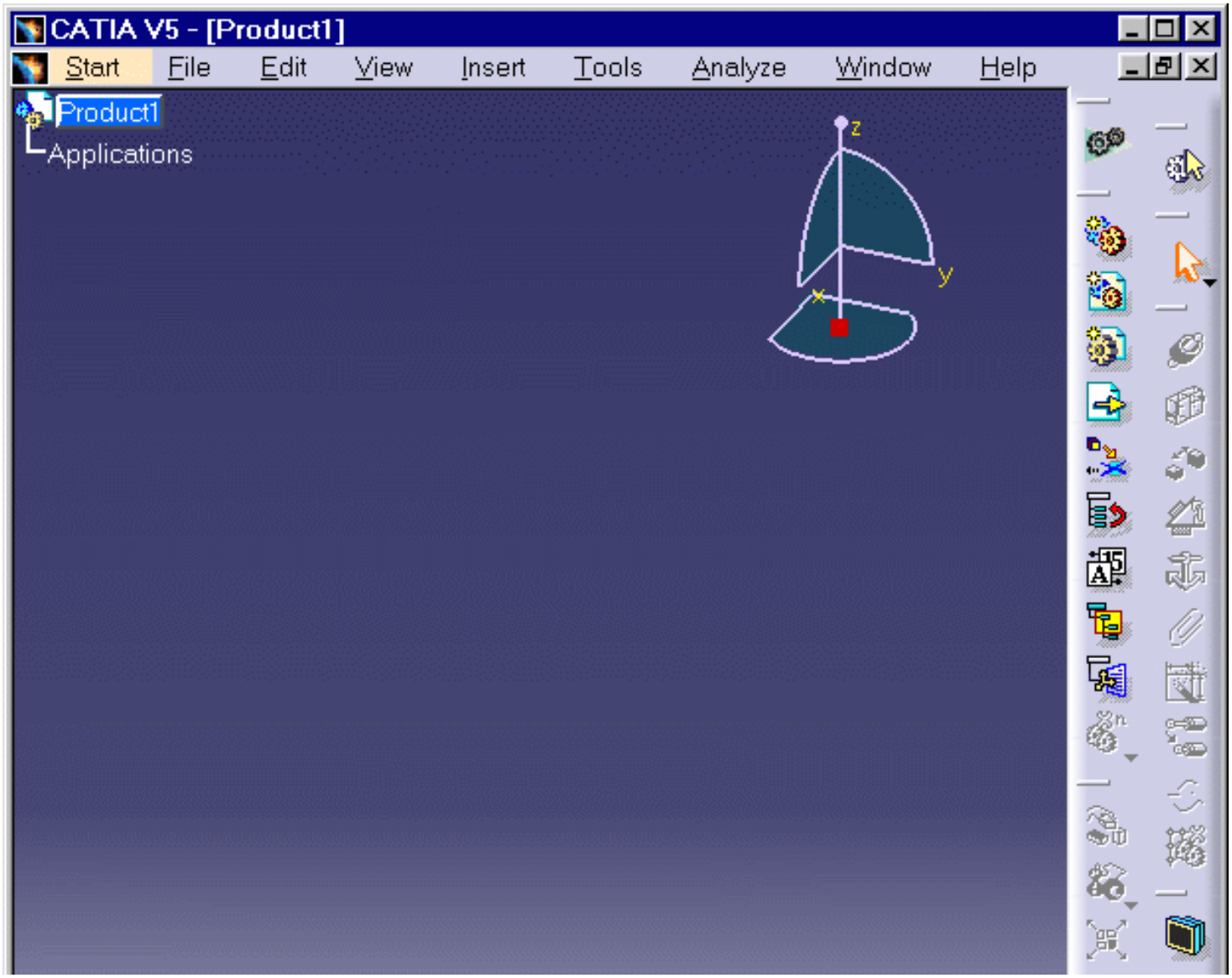
This first task shows you how to enter Assembly Design workbench and how to open an existing product.

1. Select the Start -> Mechanical Design -> Assembly Design command to launch the required workbench.

   The workbench is opened. The commands for assembling parts are available in the toolbar to the right of the application window. For information on these commands, please refer to *CATIA- Product Structure Version 5.*

   You will notice that "`Product1`" is displayed in the specification tree, indicating the building block of the assembly to be created.

To know how to use the commands available in the Standard and View toolbars located in the application window border, please refer to *CATIA- Infrastructure User's Guide Version 5*.




2. Before following the scenario, set the following options:
   - make sure the option Work with the cache system is deactivated : use the Tools -> Options command, click Infrastructure -> Product Structure to the left of the dialog box that appears and uncheck the option "Work with the cache system". Do not forget to restart CATIA after turning off the cache. For more information, refer to Working with a Cache System.
   - use the Tools -> Options command, click Infrastructure -> Product Structure to the left of the dialog box that appears, then click the Product Structure tab and uncheck the option "Manual Input". For more information, refer to Customizing Product Structure Settings.
   - use the Tools -> Options command, click Infrastructure ->Part Infrastructure to the left of the dialog box that appears, then check the option "Keep link with selected Object". For more information, refer to Customizing General Settings.

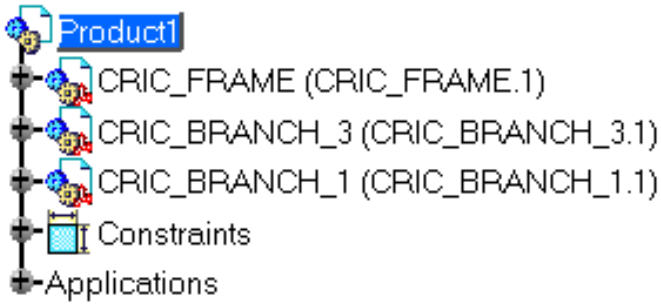
   Note also that the default mode for the Update capability is "manual". For the purposes of this scenario, set the automatic mode.

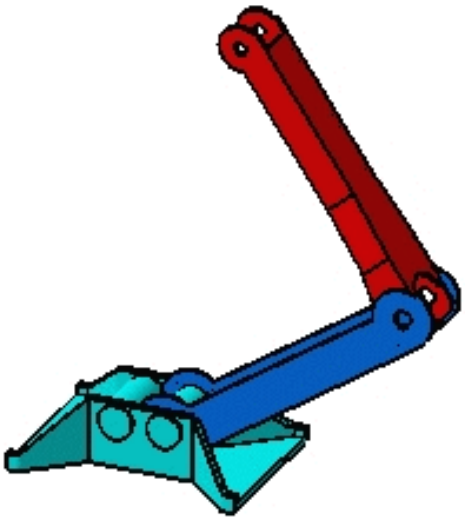
3. Open GettingStarted.CATProduct.

   You will start the scenario with an existing assembly. Product1 is composed of three parts created in the Part Design Workbench:

   1. `CRIC_FRAME` (in turquoise)
   2. `CRIC_BRANCH_3` (in blue)
   3. `CRIC_BRANCH_1` (in red)

   From now on, these parts will be referred to as 'components'.



Surface and Coincidence constraints have been defined for these parts in the Assembly workbench.

4. Select Edit -> Representations -> Design Mode. This mode lets you access technical data.

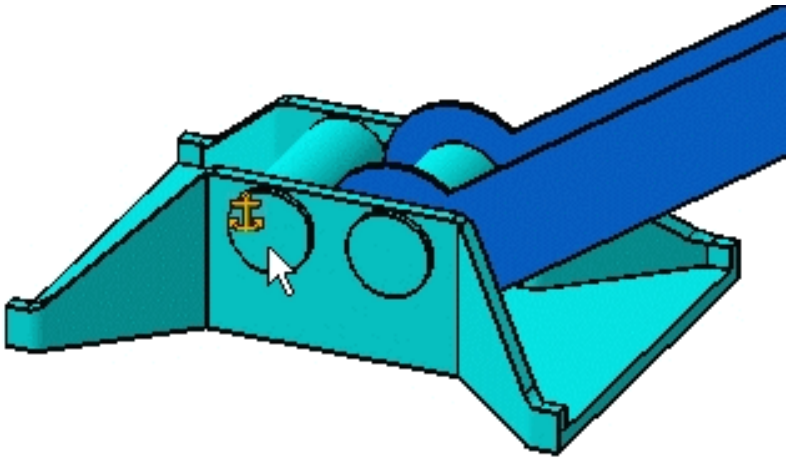
5. Click the + sign to the left of the `Constraints` text in the tree and apply the show mode on these constraints if you wish to view them in the geometry area.

# Fixing a Component

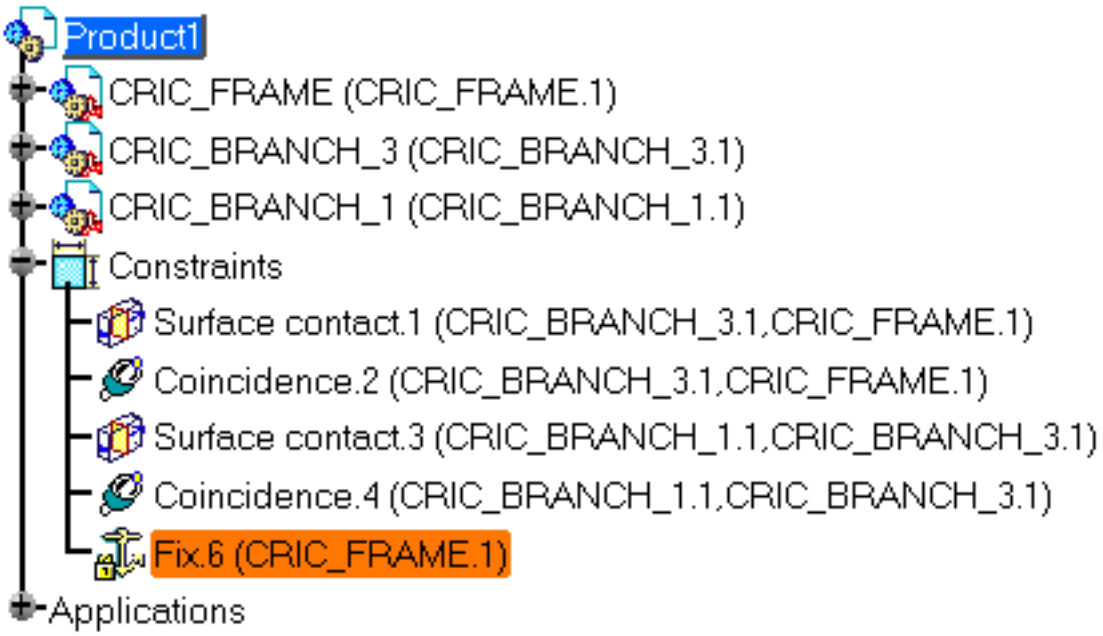
This task shows you how to set the first constraint. This operation consists in fixing the position of a component in space so as to use this component as the base of the assembly.

1. Select CRIC_FRAME in the specification tree or in the geometry area.
2. Click the `Fix Component` icon in the Constraints toolbar.

   The component `CRIC_FRAME` is immediately fixed. The application indicates this by displaying a green anchor symbol on the component.



Note also that the Constraints branch now displays the new constraint. The anchor symbol is preceded by a lock symbol, to make a distinction between "fix in space" and "fix operations". For more information, pleaser refer to Fixing a Component.


Product1
CRIC_FRAME (CRIC_FRAME.1)
CRIC_BRANCH_3 (CRIC_BRANCH_3.1)
CRIC_BRANCH_1 (CRIC_BRANCH_1.1)
Constraints
Surface contact.1 (CRIC_BRANCH_3.1,CRIC_FRAME.1)
Coincidence.2 (CRIC_BRANCH_3.1,CRIC_FRAME.1)
Surface contact.3 (CRIC_BRANCH_1.1,CRIC_BRANCH_3.1)
Coincidence.4 (CRIC_BRANCH_1.1,CRIC_BRANCH_3.1)
Fix.6 (CRIC_FRAME.1)
Applications

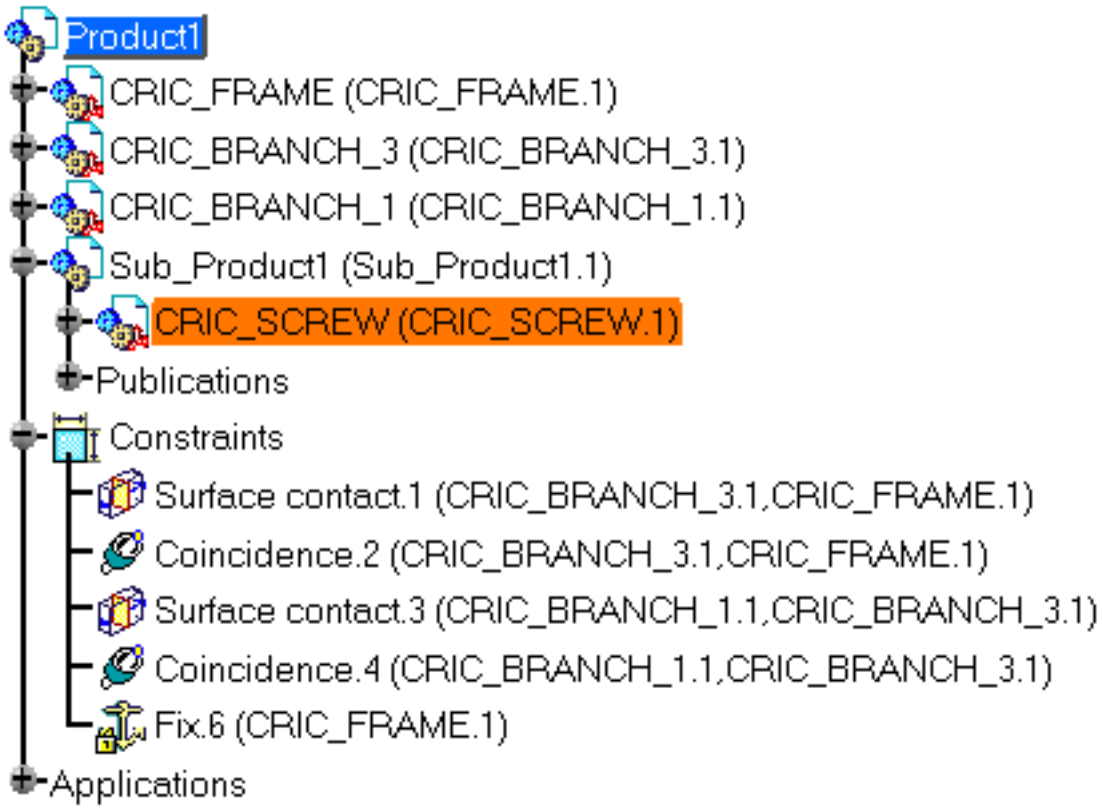
# Inserting an Existing Component

This task shows you how to insert an existing component into the assembly.

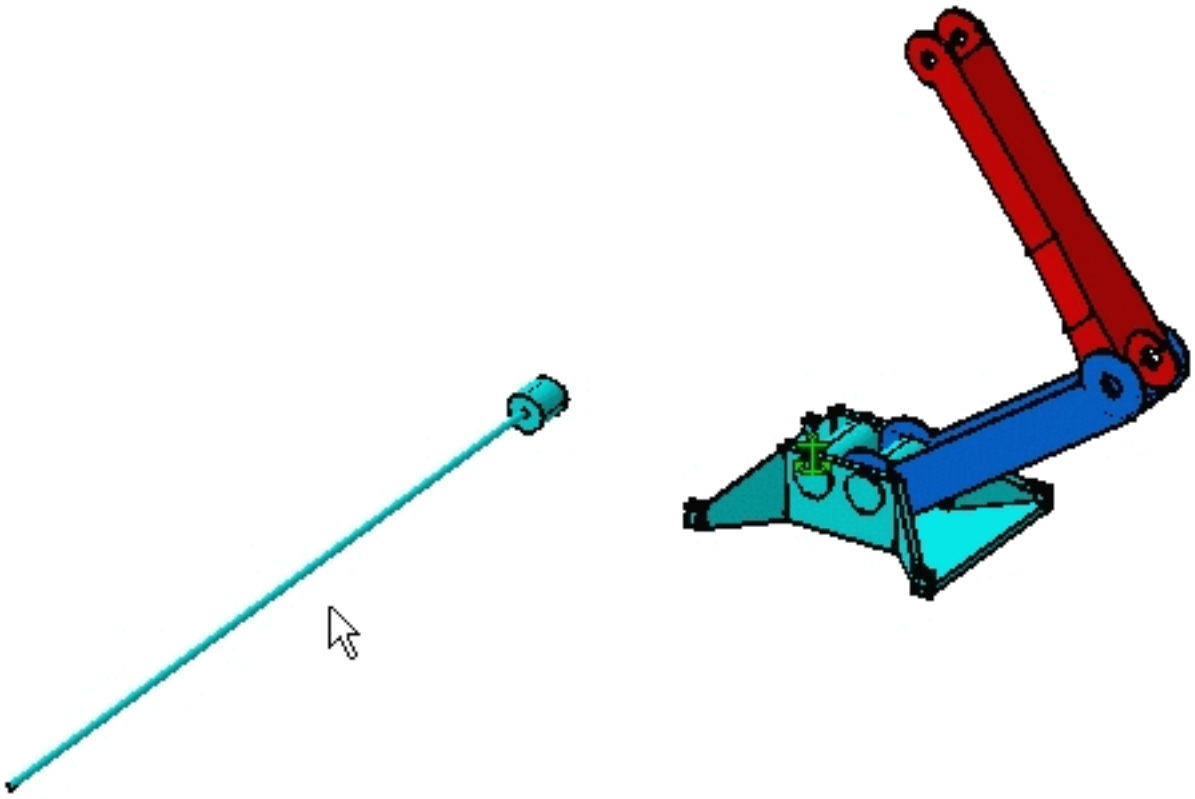
1. Select `Product1` in the specification tree.
2. Click the `Existing Component` icon  in the Product Structure Tools toolbar.

   The File Selection dialog box is displayed.
3. Navigate to `C:\Program Files\Dassault Systemes\B10doc\online\asmug\samples` directory and select `Sub_Product1.CATProduct`
4. Click Open.

   A new component is added to the specification tree. The assembly now includes four components: three parts and a sub-assembly.



This is the component you have just imported:



To know the different document types you can insert in a CATProduct document, refer to *Product Structure Version 5*. However, to know how to insert .asm documents properly, refer to Opening a .asm Document.

# Setting Constraints Between Components

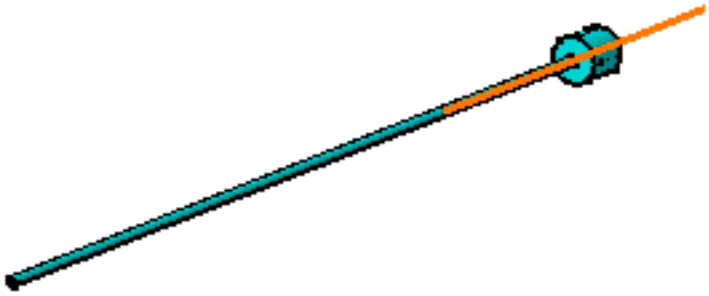
This task consists in setting a coincidence constraint, then a contact constraint between the component you have just inserted (Sub_Product1) and CRIC_BRANCH_1.

1. Click the `Coincidence` icon  .

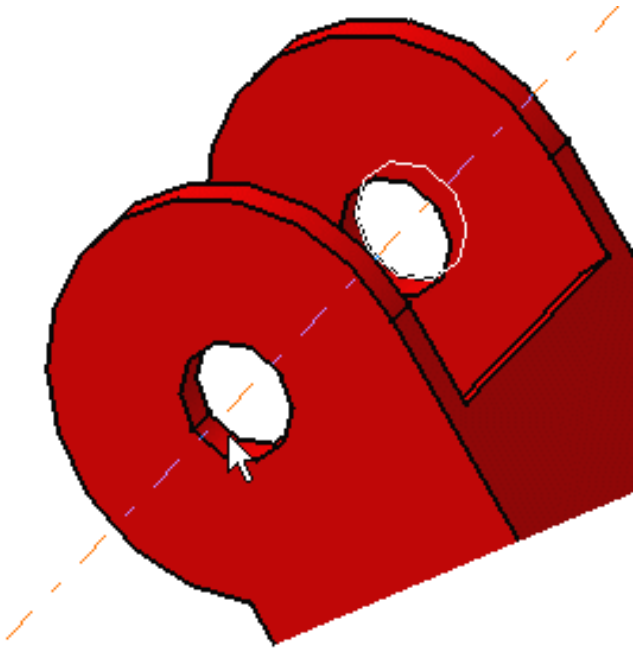
   A message window appears, providing information on the coincidence constraint command. If you do not want to see this dialog box appear any more, check Do not prompt in the future.

2. Select Axis in the geometry area.

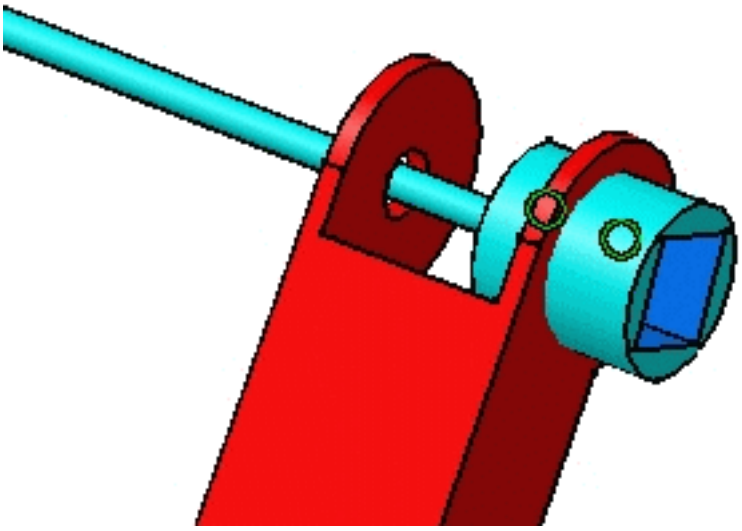
   The application detects it once selected. The axis is now highlighted in the geometry.



3. Select one of the two inner faces of CRIC_BRANCH_1 to select the associated axis.



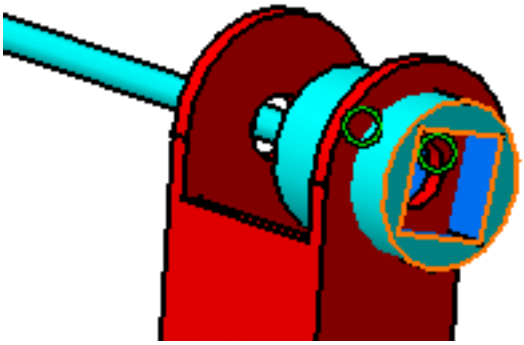
As the coincidence constraint is created, CRIC_SCREW and CRIC_BRANCH_1 are aligned:



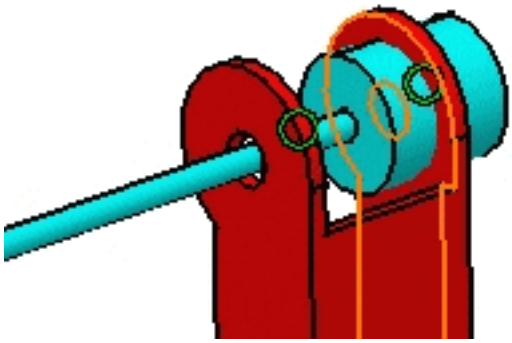
4. Now, you are going to set a contact constraint between CRIC_SCREW and a circular face of CRIC_BRANCH_1.

   To do so, click the `Contact Constraint` icon  .

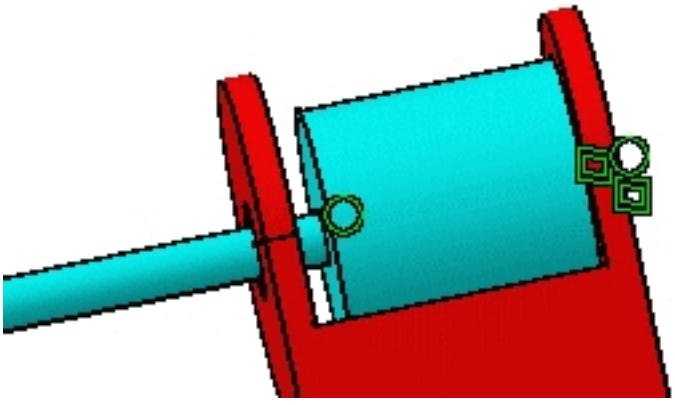
5. Select the face as shown in the geometry area.



6. Select the red circular face in the direction opposite to the published face.

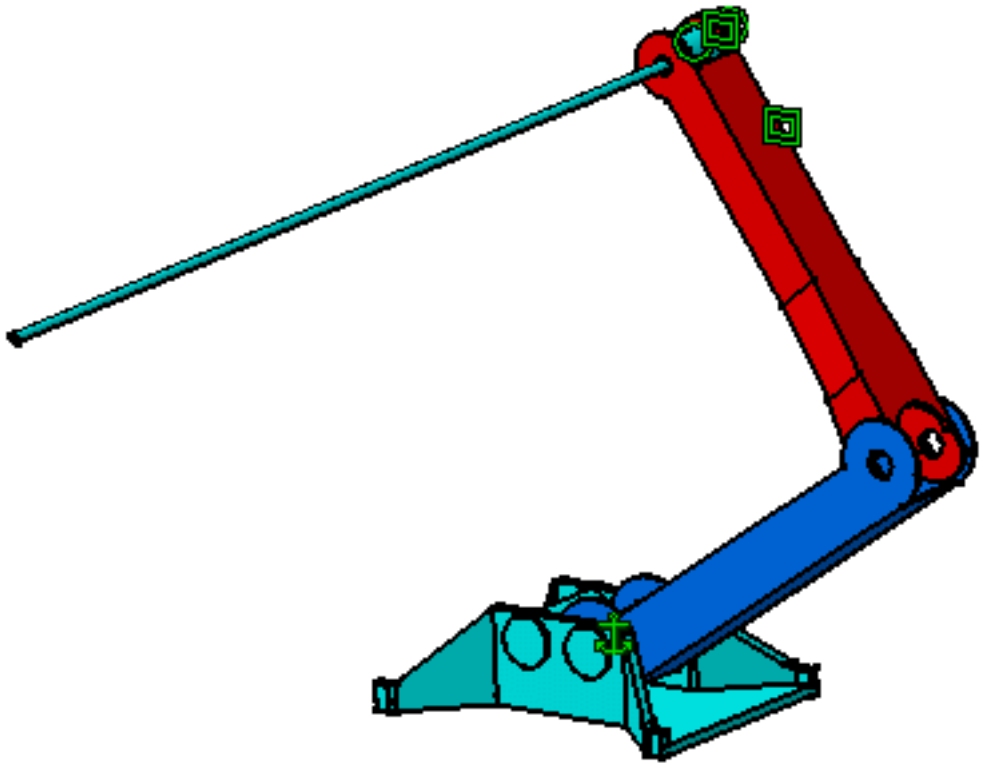


As the contact constraint is created, the turquoise cylinder is located exactly on the red face.



The created constraints are automatically updated because the automatic update mode is activated. As the color defining valid constraints is green, our constraints are green. The application allows you to customize constraint colors as explained in Customizing Constraint Appearance.

The assembly now looks like this:

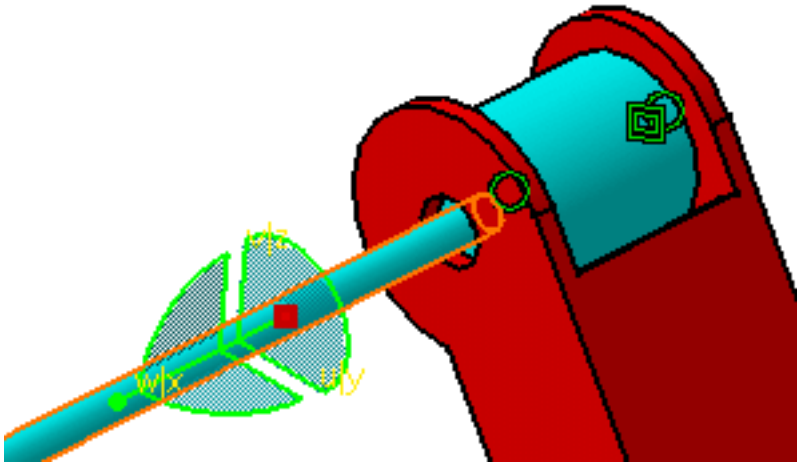


# Moving Constrained Components Using the Compass

This task consists in manipulating the assembly to check if the components react the way we want, i.e. according to the constraints we set in the previous task.

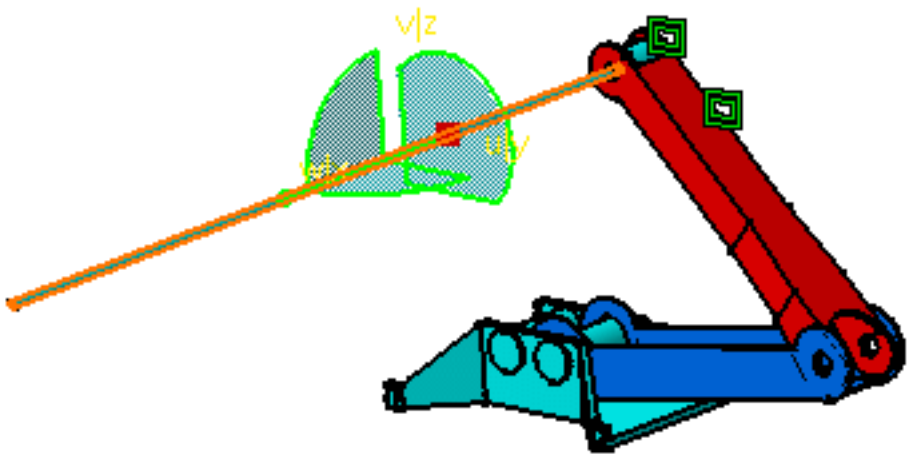
1. Select the red patch at the center of the compass and drag it onto `CRIC_SCREW`. For details about how to use the compass, please refer to *CATIA- Infrastructure User's Guide Version 5*.

   As the compass is snapped to the component, you can manipulate the component.



2. Now, if you press and hold down the Shift key, select v/z axis on the compass, then drag and drop the component up and down, you can see that three components are moving.

   This is an example of what we can get:



3. Repeat the operation as many times as you wish.

   The assembly reacts correctly. CRIC_FRAME does not move because it is fixed. The other three components can move.

4. Release the left mouse button before releasing the Shift key.

5. Drag the compass away from the selected object and drop it.

# Adding and Renaming a New Component

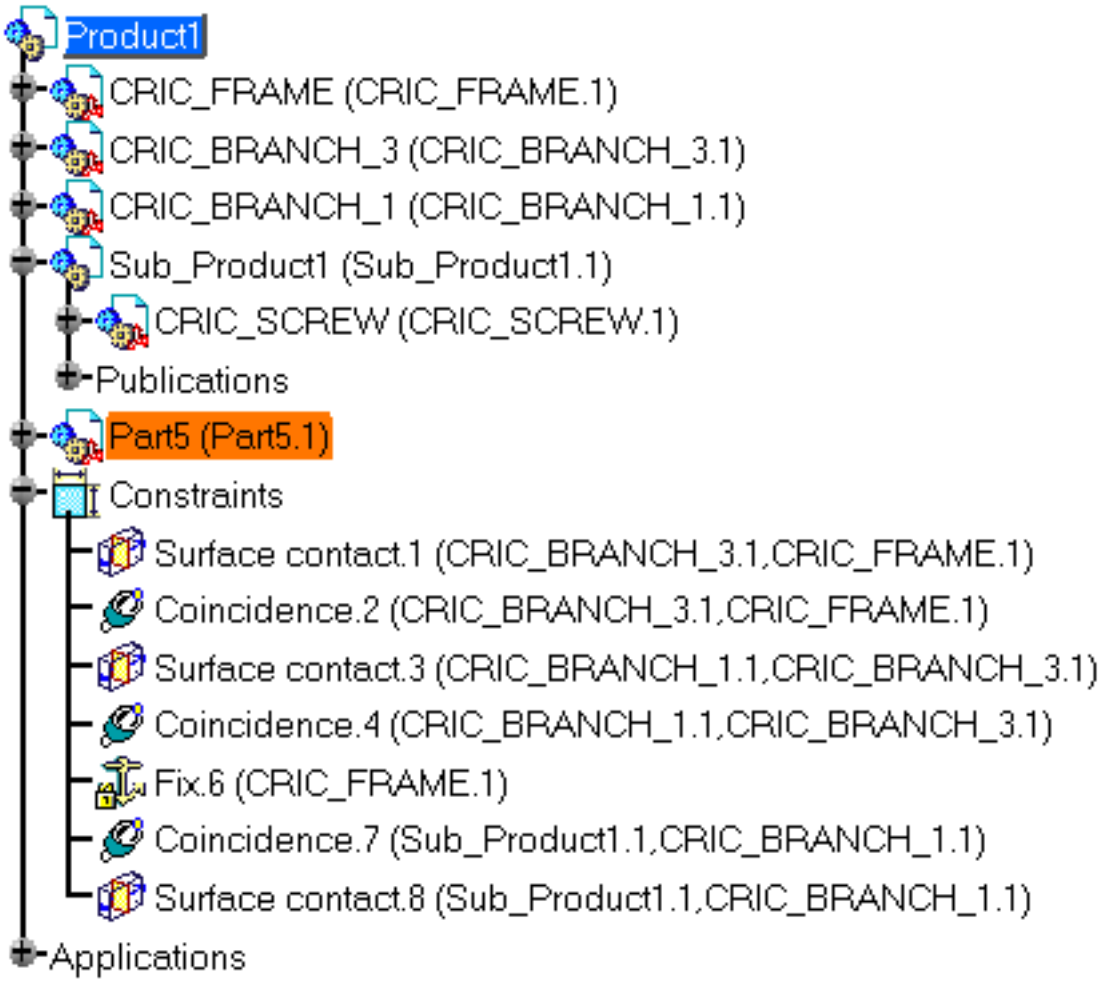
This task consists in adding a new component to the assembly. You will then rename this component. This component is a part created in the Part Design workbench.

1. Click `Product1` and select the `Part` icon in the Product Structure Tools toolbar.

   The New Part: Origin Point dialog box appears, presenting two possible options: Either you define the point of your choice to locate the new part, or you use the origin point of the assembly as the origin point to be used for the part.

2. Click No to use the origin point of the assembly.

   The new component "`Part5 (Part5.1)`" is now displayed in the specification tree:

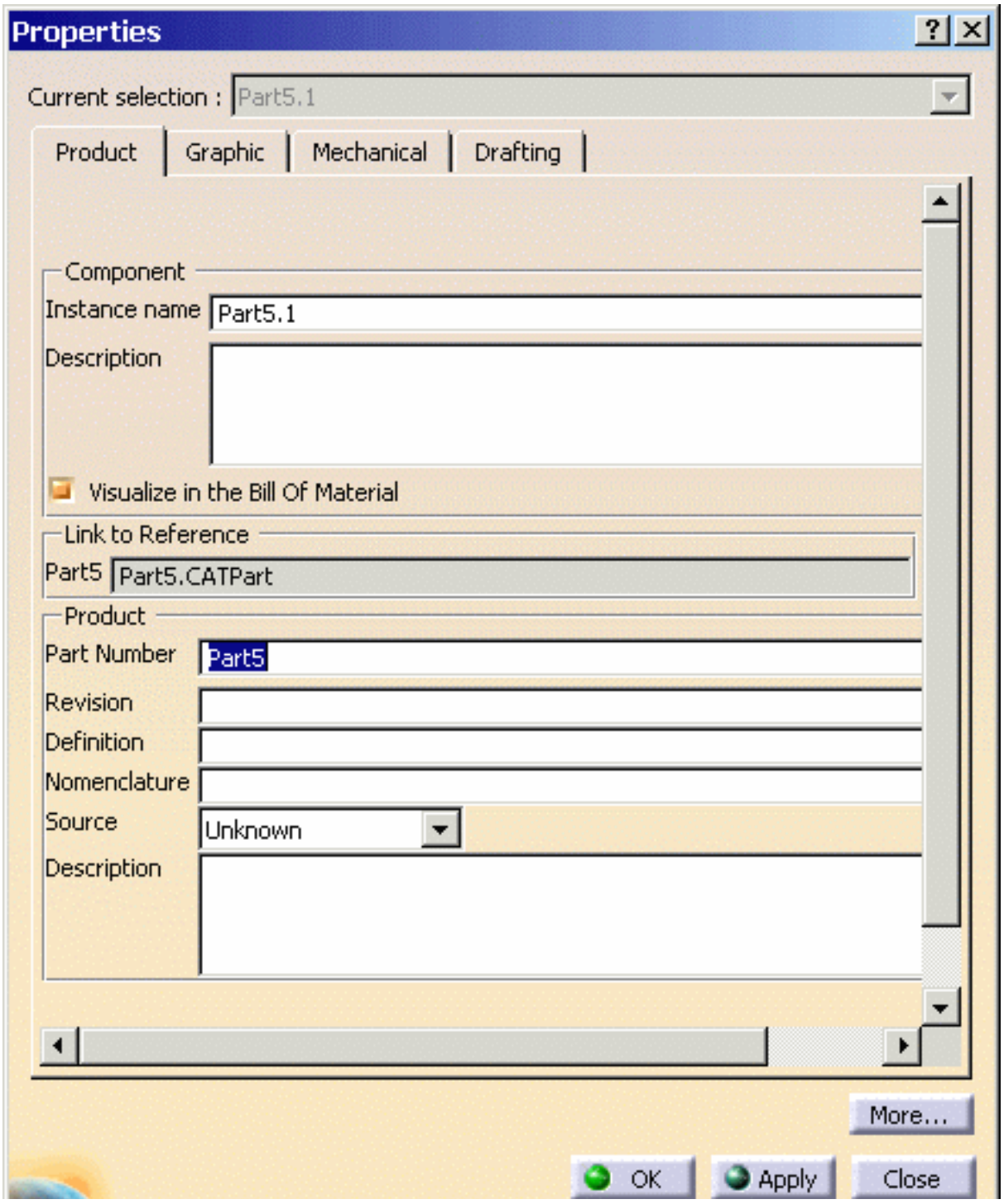


If the Manual Input option is activated (see Defining the Default Part Number), the Part Number dialog box appears before the New Part: Origin Point dialog box and lets you enter the name of your choice.

3. Right-click `Part5 (Part5.1)` and select the `Properties...` contextual command.

4. In the Properties dialog box that appears.

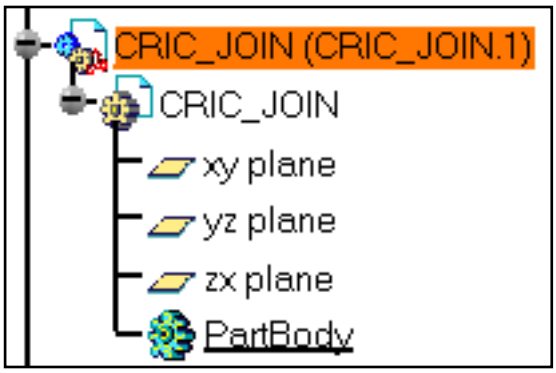
   The options available in the Product tab let you enter the information you required.




5. Enter CRIC_JOIN.1 in the Instance name field and CRIC_JOIN in the Part Number field.

6. Click OK to validate the operation.

   The new names are now displayed in the specification tree:

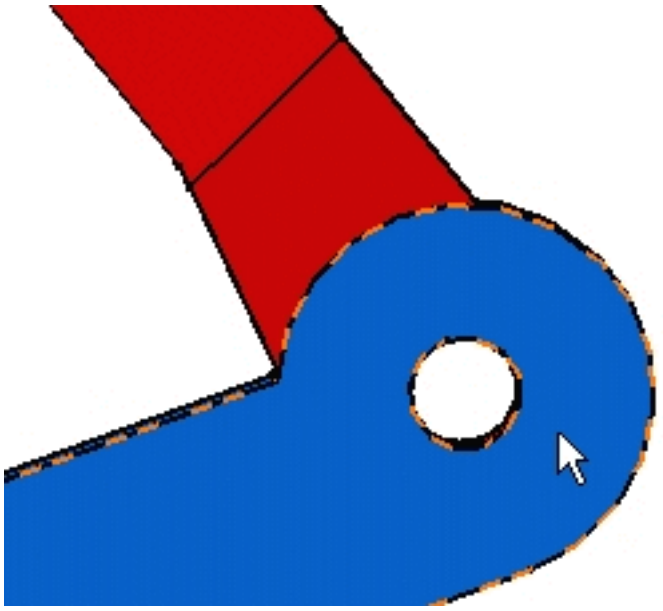



# Designing a Part in an Assembly Context

This task consists in designing the part you have just added to the assembly. It shows you how easy it is to access the tools required for designing components in an assembly context.

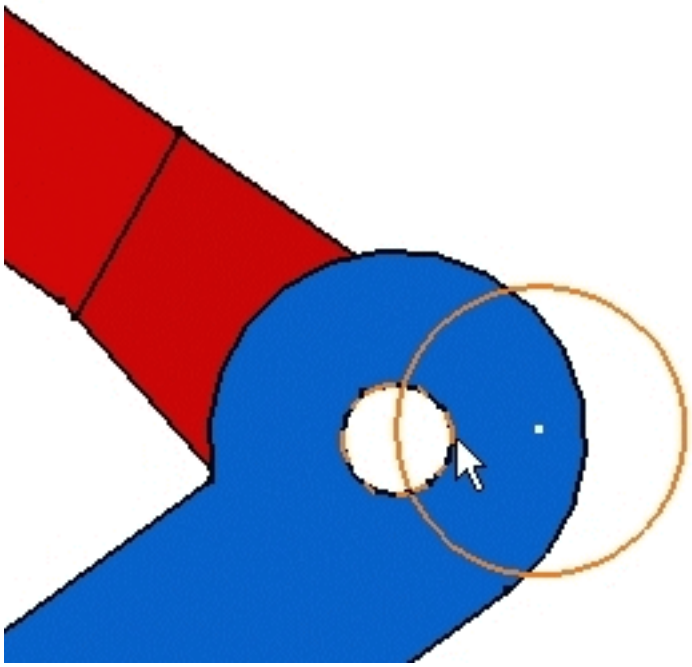
1. Double-click `CRIC_JOIN` in the specification tree to access the Part Design workbench.

2. Select the blue face as shown and click the `Sketcher` icon to access the Sketcher workbench.

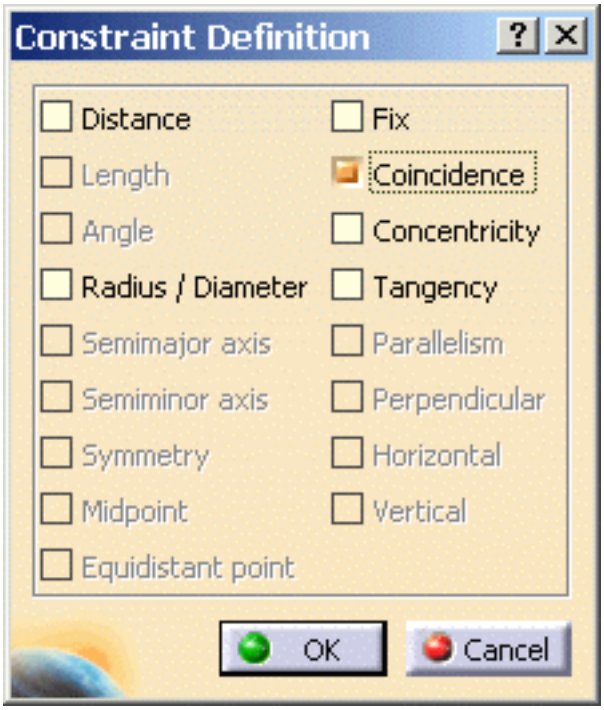


3. Now that you are in the Sketcher, click the `Normal View` icon in the View toolbar and sketch a circle on the face using the `Circle` command .

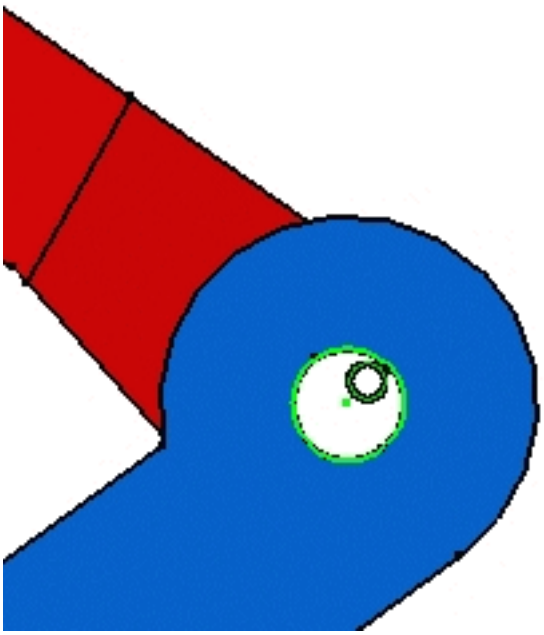
   Do not bother about positioning the circle.



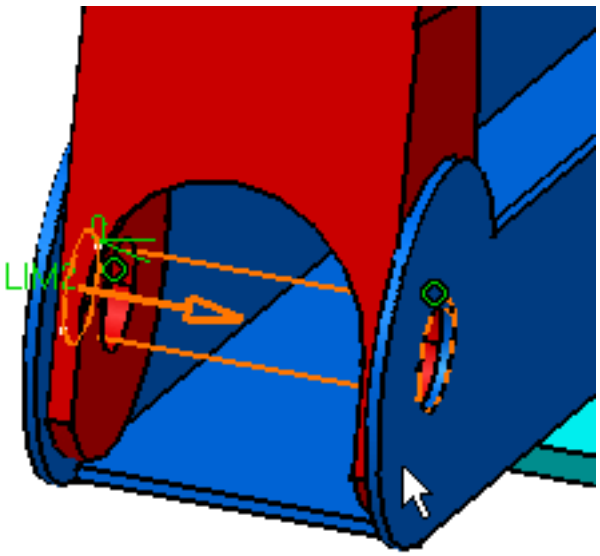
4. Now to obtain the same radius value as the one used for CRIC_JOIN circular edge and to make sure that this circular edge and the circle share the same axis, use the `Constraints Defined in Dialog Box` command  to create a coincidence constraint (select the circle -if not already done- and the circular edge, then click the `Constraint Defined in Dialog Box` command and check "Coincidence").




After validating the operation, the circle is coincident with the circular edge.
You must obtain this:

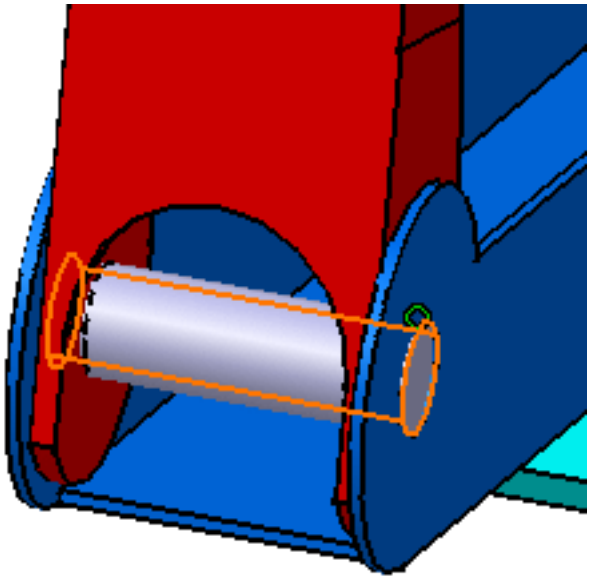


**5.** Exit the Sketcher and use the Pad command  with the "Up to Plane" option to extrude the sketched circle. Select the blue face as shown to specify the limit of the pad.




After validating the operation, you should obtain this cylinder:

The part is designed.



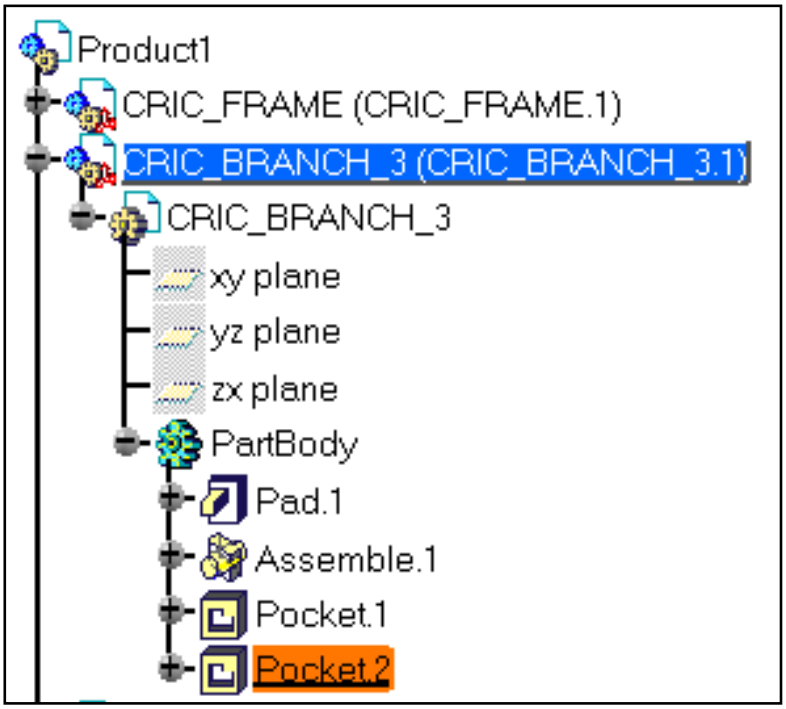
For information about Part Design and designing in context, refer to *CATIA- Part Design User's Guide Version 5* and Designing in Assembly Context respectively.

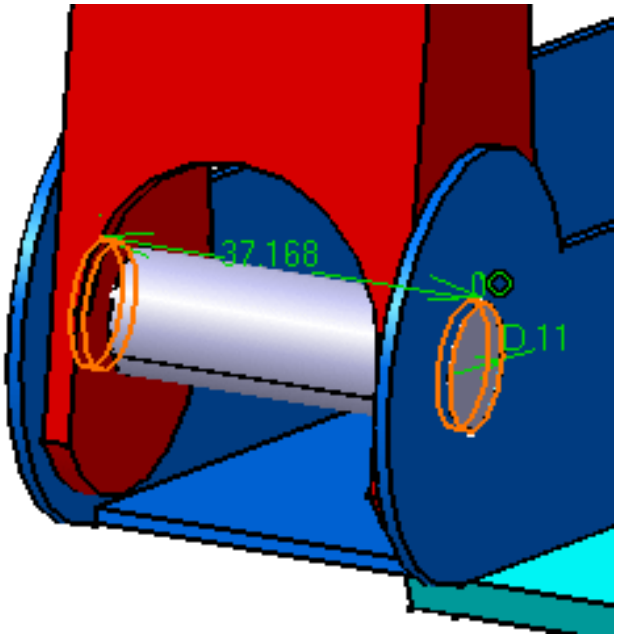
# Editing a Parameter

In this task, you are going to edit the diameter of the pocket belonging to CRIC_BRANCH_3. You will see how this edition affects the part you created in the previous task.

1. Double-click CRIC_BRANCH_3 to access the Part Design workbench.

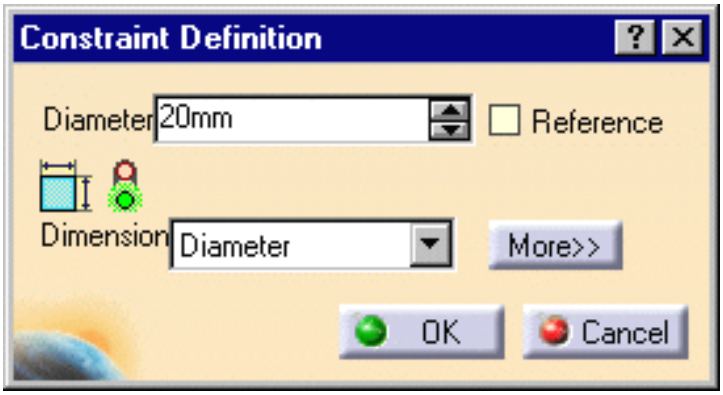
2. Select Pocket.2 and use the Pocket.2 object -> Edit Parameters contextual command to display the associated parameters.




3. Double-click D11 in the geometry area to display the Constraint Edition dialog box.

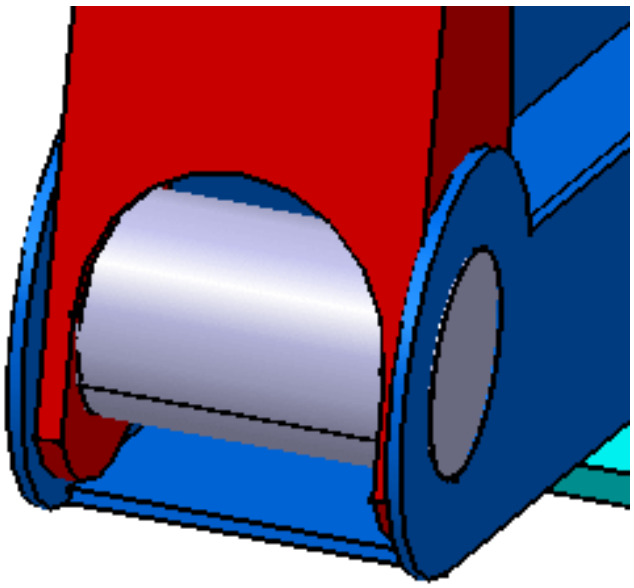
4. Enter 20 as the new diameter value and click OK to generate the new pocket.



5. Update Product1 by double-clicking on `Product1` in the specification tree.

   The pocket is modified accordingly. The coincidence previously set between the two parts is maintained.

   This result is made possible thanks to the option Keep link with selected Object you set at the very beginning of the scenario.

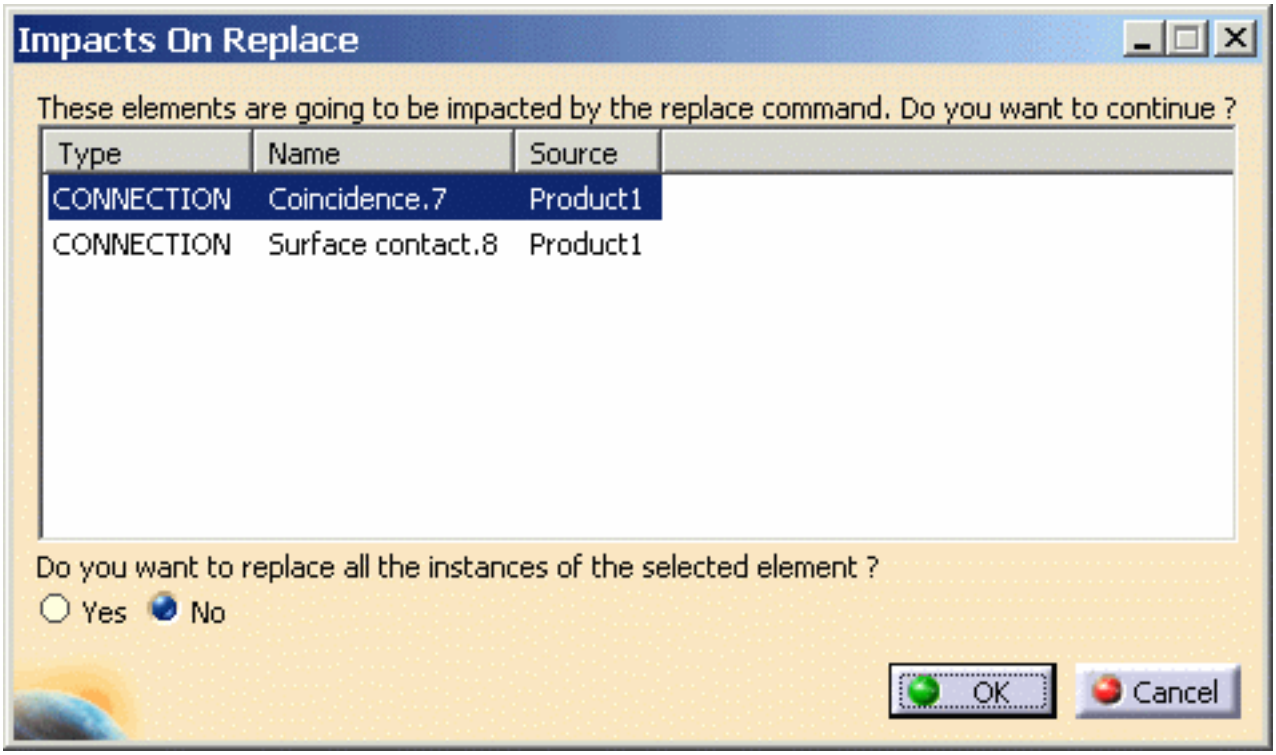


# Replacing a Component

This task shows you how to replace Sub_Product1.CATProduct by another component.

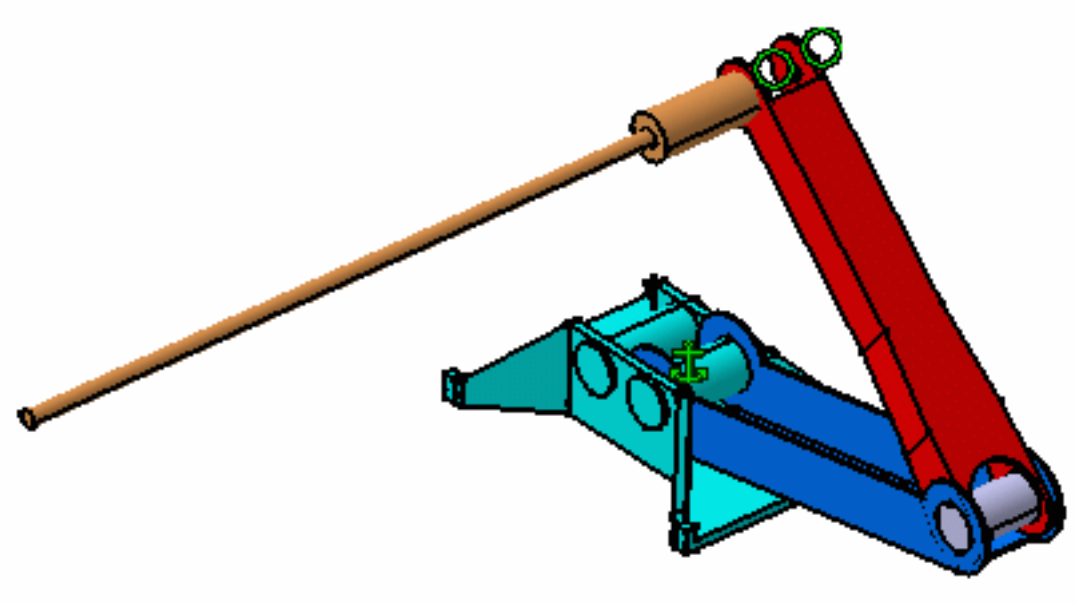
1. Select Sub_Product1.CATProduct in the specification tree.
2. Click the `Replace Component` icon  in the Product Structure Tools toolbar.
3. In the dialog box that appears, select `Sub_Product2.CATProduct` as the replacement component and click Open.

   The Impacts on Replace dialog box is displayed:




4. Check Yes to replace all instances of the selected element and click OK to confirm.

   Sub_Product1.CATProduct is no longer visible. This is Sub_Product2.CATProduct:



If necessary update the document.

Note that the coincidence constraint is maintained. This is due to the publication of the axis used in the constraint definition. As the axis is a published element, the application can reconnect the constraint.

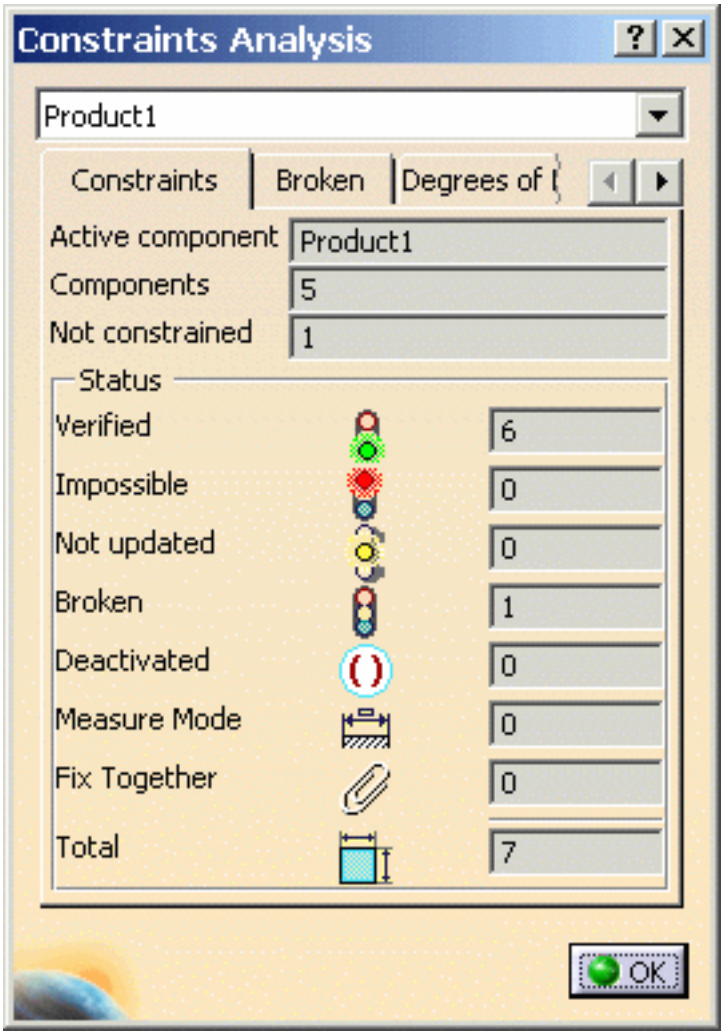
Conversely, the contact constraint is broken. You will know how to reconnect it later.

# Analyzing Assembly Constraints

This task shows you how to analyze the status of all assembly constraints defined for Product1.

1. Select the Analyze -> Constraints... command.

   The Constraints Analysis dialog box that appears displays all the information you need. The Constraints tab contains a detailed status of the assembly: the number of non-constrained components and the status of the defined constraints.




2. Click the Broken tab to see the list of broken constraints. We have only one broken constraint, a contact constraint.

3. Click on the name of the constraint.

   The constraint is highlighted in the specification tree.

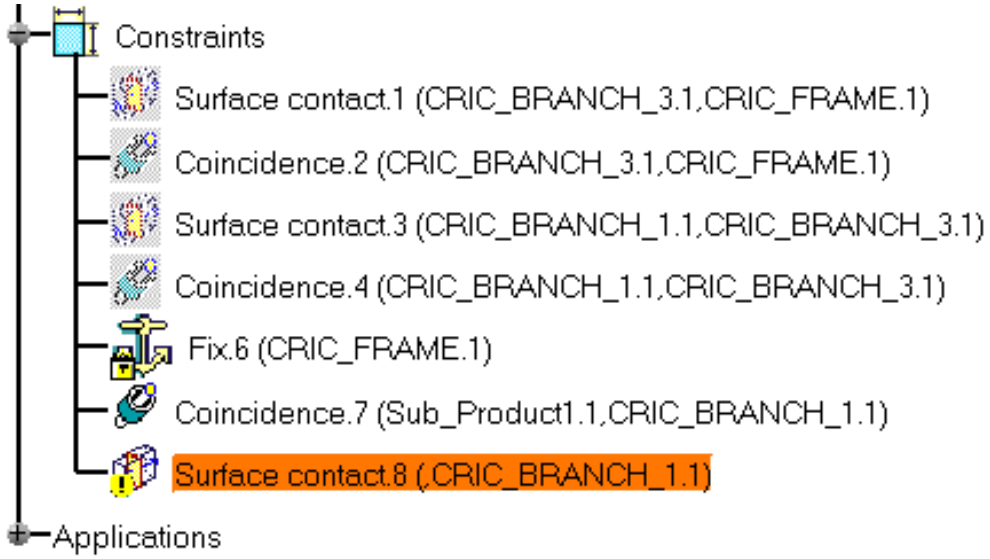
4. Click on OK to close the dialog box.

Reconnecting this contact constraint is our next task.

# Reconnecting a Broken Constraint

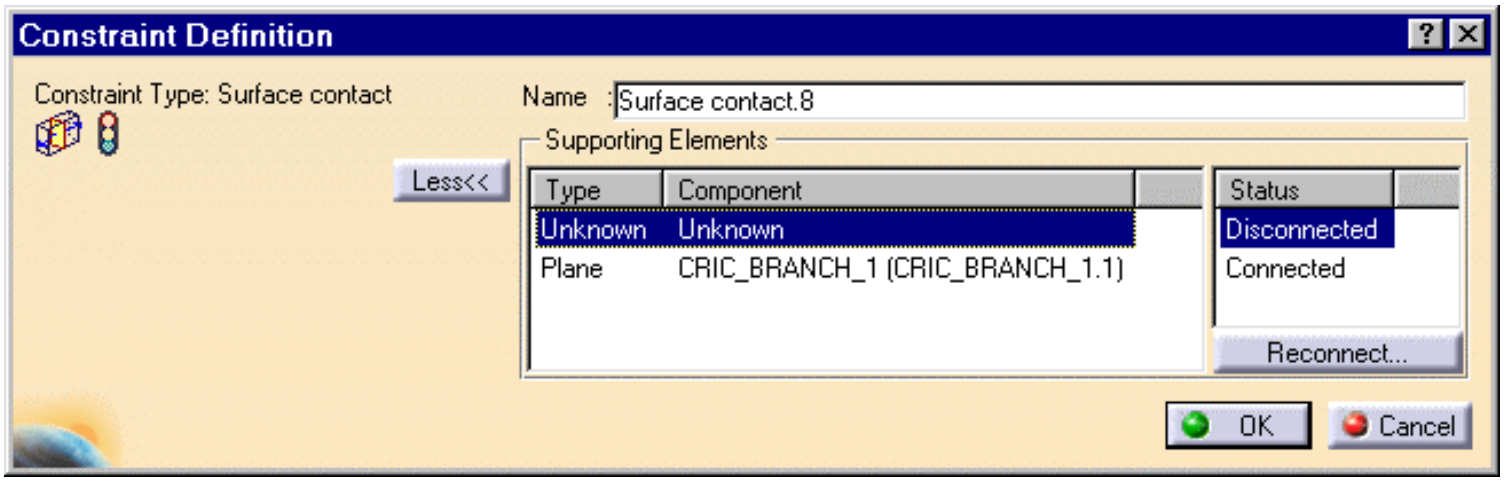
In this task, you will learn how to reconnect the broken constraint detected by the application.

1. Double-click the broken constraint in the specification tree. Note that this broken constraint is indicated by a yellow warning symbol.




2. In the Constraint Definition dialog box that appears, click More to access additional information.

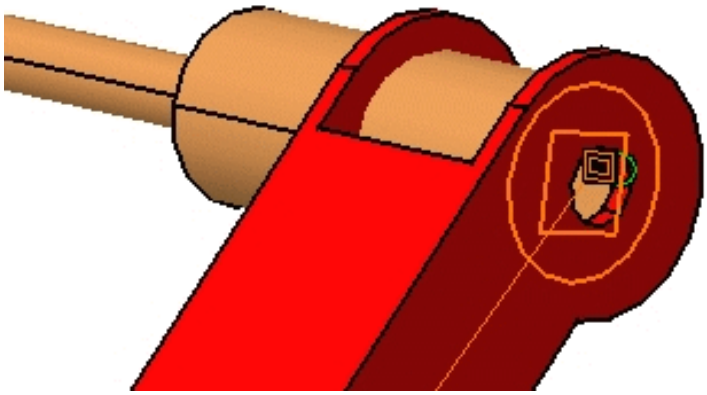
3. Click Disconnected in the Status frame, then Reconnect...




4. You are then prompted to select a component to rebuild the constraint. Select the same faces as the ones used for setting the first contact constraint. If you need some help, refer to Setting Constraints Between Components.

5. Click OK to validate the operation and update the document.

   The constraint is reconnected:



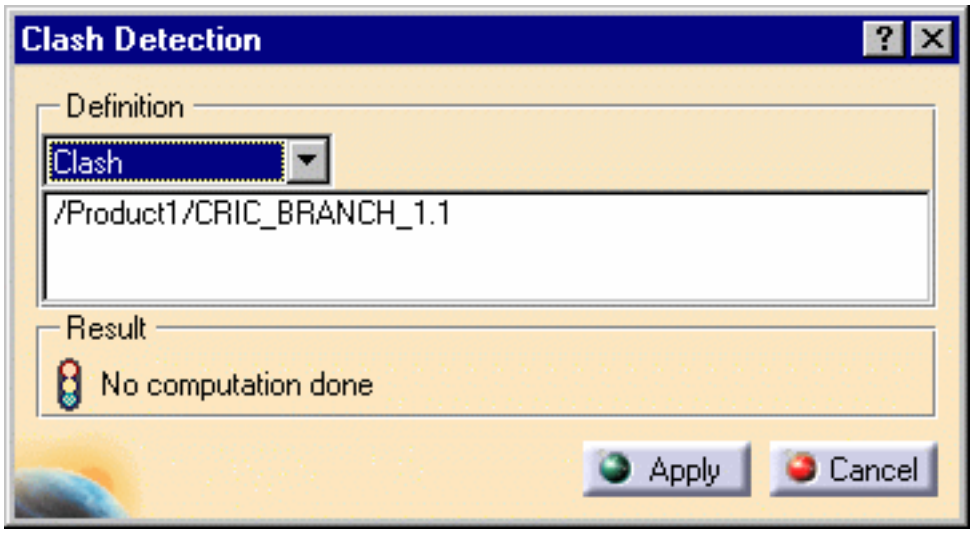
# Detecting Clashes

In this task, you will learn how to detect possible clashes between two components.

1. Select `CRIC_BRANCH_1.1` in the specification tree.

   Select the `Analyze -> Compute Clash...` command.

2. The Clash Detection dialog box appears. It displays the first component selected for computing possible clashes.

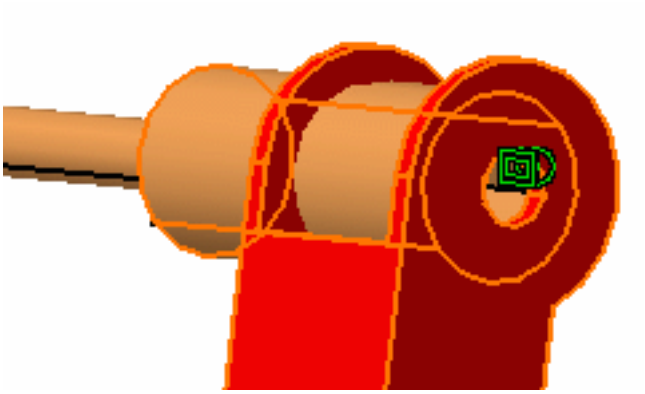



3. As you need another component, select `SUB_PRODUCT2` using the Ctrl key.

   This component also appears in the dialog box.

4. Click Apply to compute clashes.

   The application detects a clash between the brown cylinder and the red face. This is indicated by two red circles in the geometry, as the arrow shows in the figure below:



The result of the computation also appears in the dialog box.

5. Click Cancel to close the dialog box.

Well, now that you know that your assembly needs to be modified to work properly, let's edit the cylinder.

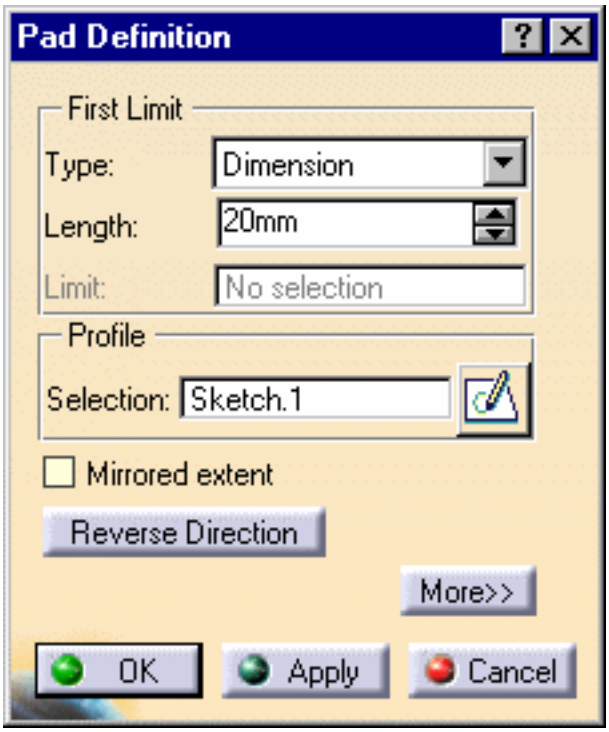
# Editing a Component

This task shows you how to edit the component causing the problem.

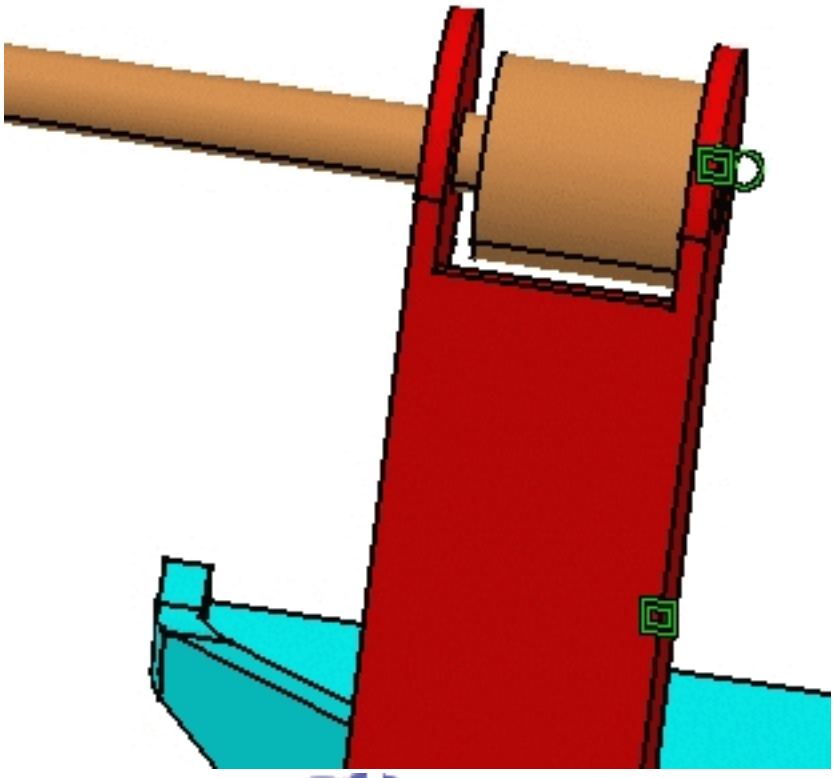
1. Double-click the brown cylinder to access the Part Design workbench.

2. Double-click the cylinder again to edit it. The Pad definition dialog box is displayed.

3. Enter 20mm to reduce the pad length and click OK.




4. The cylinder is updated and now looks like this:

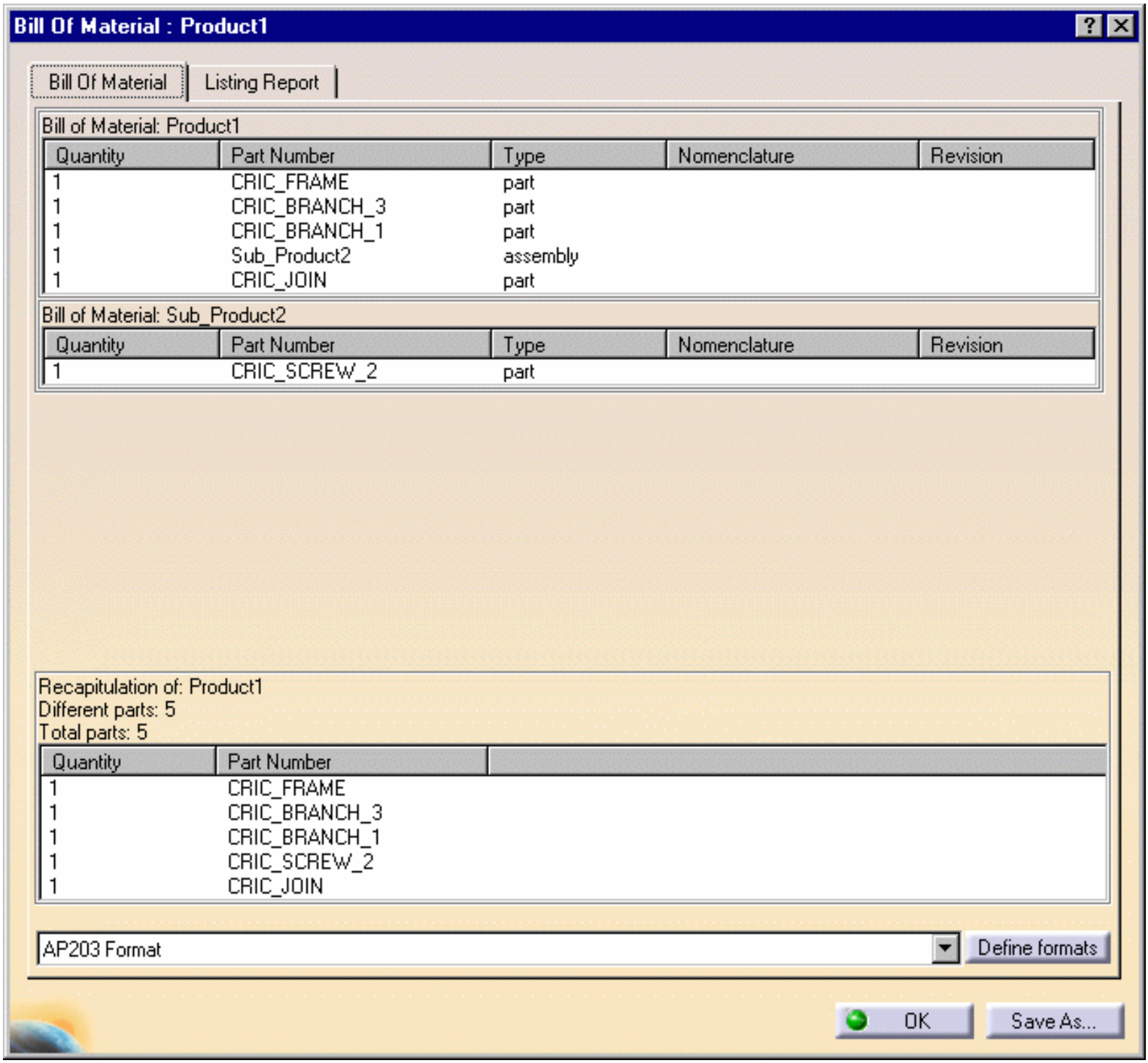


# Displaying the Bill of Material

This task shows you how to access all the information available about the structure of the assembly.

1. Return to Assembly Design workbench and select the `Analyze -> Bill of Material...` command.

   The Bill of Material is displayed.



It is composed of these sections:

- Bill of Material: lists all parts and sub-products one after the other
- Recapitulation: displays the total number of parts used in the product
- Define formats: customizes the display of the bill of material

The Listing Report tab displays the tree of the product using indents

**2.** If you wish, you can save this document using the html format or the txt format. Just click the Save As... button, then give a name and the appropriate extension to your file.

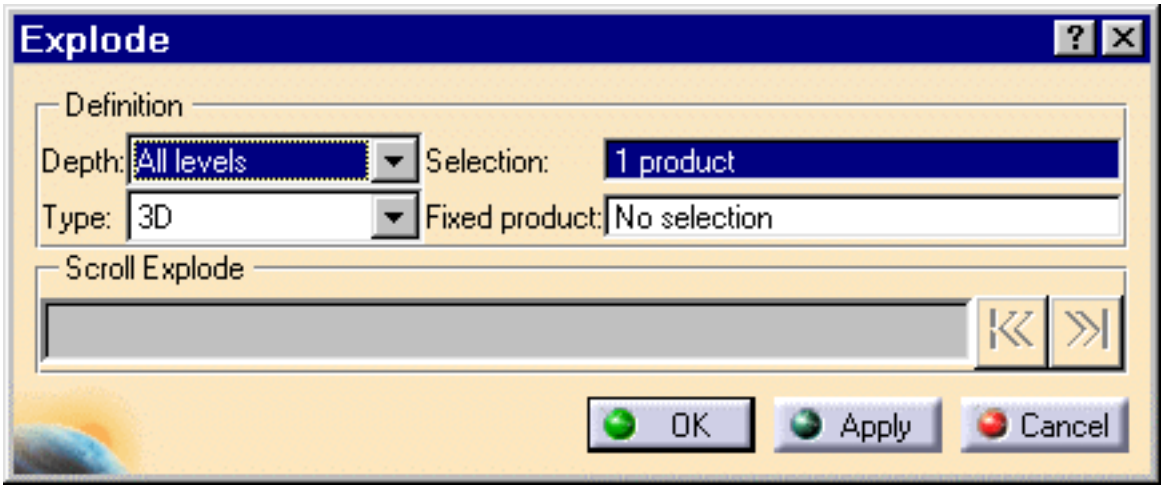
For more information about the bill of material, refer to Displaying the Bill of Material.

# Exploding the Assembly

This last task illustrates the use of the Explode capability. Exploding the view of an assembly means separating the components of this assembly to see their relationships.

1. Make sure `Product 1` is selected.
2. Click the `Explode` icon  in the Move toolbar.

   The Explode dialog box is displayed.

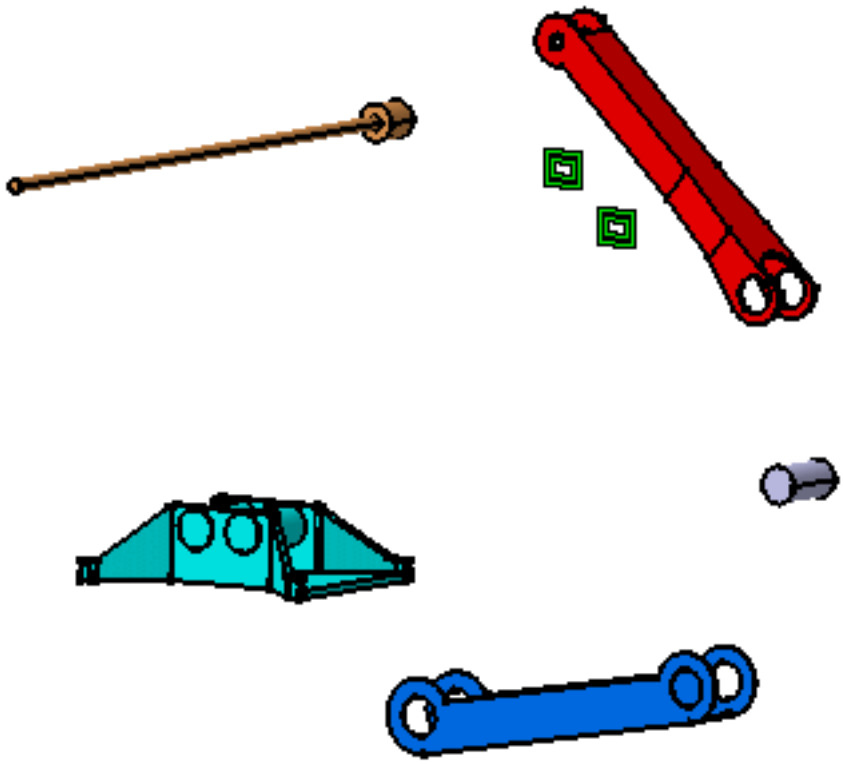



   Product 1 is the assembly to be exploded. The Depth parameter lets you choose between a total (All levels) or partial (First level) exploded view.

3. Set All levels if not already set.
4. Set 3D to define the explode type.
5. Click Apply to perform the operation.

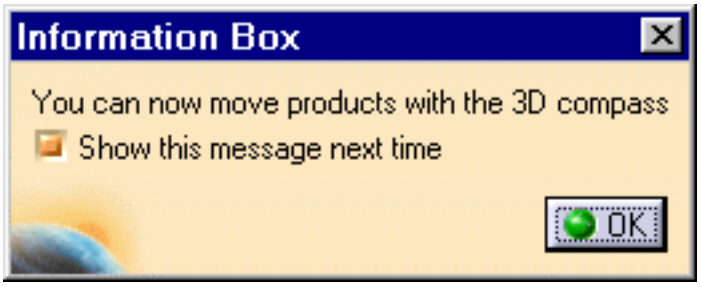
   The Scroll Explode field gradually displays the progress of the operation. The application assigns directions and distance.

   Once complete, the assembly looks like this:



The usefulness of this operation lies in the ability of viewing all components separately.

Note that you can move products within the exploded view using the 3D compass.




6. Click OK to validate the operation and then click Yes at the prompt or click Cancel to restore the original view.

Well, you have done all the tasks of the Getting Started section. Why not consult the rest of the documentation?