



Product Structure Symbols

-  Product1 A product
-  Product2 A component or a sub-product
-  Flexible_Product The purple little wheel to the left corner of the CATProduct icon and the light bar  identify a flexible sub-assembly.
-  Part.1 Instance of a part. This symbol  means that there is a geometrical representation of the part and that it is activated. (No context)
-  Part_with_deactivated_Representation This symbol means the geometric representation is deactivated.
-  Component_with_deactivated_Representation The representation of this component is deactivated.

Product Structure Symbols – Contextual parts (1)

Three types of contextual parts exists:



Definition
_Instance

This is a **contextual part** and more precisely a **Definition Instance**.

The green gear and the blue chain signify the "original" instance of a contextual part in a CATProduct (it is driven by another part; built with another part's data).



Instance of the
Definition
_Instance

This contextual part is the **Instance of the Definition Instance** represented by the white gear and the green arrow. The geometry of this instance is connected with the Definition Instance (contextual link). You can edit this contextual part. (**Active sub context; Links can be created and updated/synchronized; new links to other context can't created without modifying context information**)



Other Instance
of the
Contextual Part

The brown gear and the red flash signify the second or **subsequent instance of a part that is contextual**. There is a distinction between the "original" and subsequent instances of contextual parts because the geometrical definition of contextual parts (Skilllets) is dependant upon neighboring components (support) in the Assembly. The geometry of this contextual part depends upon another instance. This symbol can appear when you import this part into another CATProduct. (**Not active sub context; Links can not be created and updated/synchronized**)

Product Structure Symbols – Contextual parts (2)



Reference of a Part



A deactivated component. The shape representation is deactivated; its geometry is not visible. This functionality can occur simultaneously on several CATIA documents containing this component, especially when this component is the instance of a reference. This operation is equivalent to the Delete operation because the reference of the component no longer exists within the Bill Of Material.



A deactivated Product.



The geometry of the component disappears. The product is downloaded, its references are missing but the user is able to find them back.

Referenced Geometry Symbols

-  Solid.1 [Geometry copied](#) from a document different from the CATPart document in which it is pasted.
-  Solid.1 Initial geometry has undertaken modifications in the original CATPart document: [solid to be synchronized](#).
-  Solid.1 Initial geometry has been deleted in the original CATPart document or the original CATPart document [has not been found](#)
-  Solid.1 Pointed document [found but not loaded](#) (use the Load contextual command or the Edit -> Links command)
-  Solid.1 External link deactivated so that geometry cannot be synchronized during the update of the part (even if the option "Synchronize all external references for update" is on).
-  Solid.1 Geometry pasted (using the As Result with Link option) [within the same CATPart document](#) from which it has been copied
-  Point.4(...!Part3.1!PublishedPoint) [Point referenced](#) in the CATPart document is [a published element](#).
-  Sketch.1(...!Part5.1!Sketch.1) Sketch referenced in the CATPart document is a published element. The published point has undertaken modifications so that a [synchronization is required](#).

Miscellaneous Symbols (1)

-  Body1. A model with geometrical representation.
-  PartBody A part body. You have access to the part's features, the way it is organized (Pad, Pocket, Intersection, etc.).
-  Open_body.1 External references branch of the part : external geometry (a face, a point or a line) is copied/imported from driving parts to contextual parts that are being driven (Design in context). You can customize External References in Tools -> Options -> Mechanical Design -> Part Design, select the General tab and check the box Keep links with selected object.
-  xy plane xy plane, yz plane or zx plane. You can click the desired reference plane either in the geometry area or in the specification tree.
-  Sketch.1 A sketch.
-  AbsoluteAxis Absolute Axis: contains information about Origin, HDirection and VDirection.
-  Origin Origin.
-  HDirection HDirection or VDirection.
-  Geometry Geometry (Point, Line,...): Wireframe and Surfaces features.

Miscellaneous Symbols (2)

-  Constraints Constraints: Parallelism, Perpendicularity, etc.
-  Point.1 Formula defined for Point.1.
-  Face Publication : a CATPart or CATProduct element is published that is to say its geometrical data is exposed.
-  Hole.1 Assembly hole.
-  Product A product in NO SHOW. By clicking in the CATProduct's contextual menu or by using the Hide/Show icon , you can put the product in the SHOW or NO SHOW area, the product's geometry is hidden. As a consequence, the documents under it, for instance the CATPart is in the NO SHOW space as well.
-  Part A part in NO SHOW.
-  Sketch The Sketcher symbol is by default in NO SHOW. By this means the geometry lighter. But you can reactivate the Sketcher representation by clicking in the contextual menu or by using the Hide/Show icon . By double-clicking on this symbol you can return into the Sketcher workbench.

Symbols reflecting an incident in the geometry

-  Part2
Part to be updated
-  Product5
No visualization of the product or the part. The product's reference cannot be found. The geometry of the component disappears.
-  PartBody
A broken link. The access to this product is impossible because the link with the root document has been lost.
-  Shaft.1
A broken Shaft.
-  Pocket.1
The pocket's representation is deactivated.
-  Plane.1
Isolated plane (can no longer be edited)
-  Offset.1
A broken constraint. The access to this product and the information about its constraints cannot be retrieved.
-  Parallelism.1
A deactivated constraint (a parallelism constraint).
-  Perpendicularity.1
A constraint to be updated (a perpendicularity constraint).

All The Best
-From-
Vidyaputra
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AND SMART.....**

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