Electrical 3D Design & Documentation

Overview

Welcome to the *Electrical 3D Design and Documentation User's Guide!*
This guide is intended for users who need to become quickly familiar with the product.

This overview provides the following information:

Electrical 3D Design & Documentation in a Nutshell

Electrical 3D Design & Documentation  is a product which provides machinery and consumer goods industry with a dedicated toolset for designing and documenting their electrical systems.

This product offers the following main functions:

- electrical devices on parts and assemblies definition
- geometrical and electrical bundle creation
- assisted device placement according to the electrical specifications coming from external data (iXF file)
- creation and management of wire and electrical device catalogs
- wire routing
- harness flattening capabilities.

Before Reading this Guide

Before reading this guide, you should be familiar with basic Version 5 concepts such as document windows, standard and view toolbars. Therefore, we recommend that you read the *Infrastructure User's Guide* that describes generic capabilities common to all Version 5 products. It also describes the general layout of V5 and the interoperability between workbenches.

Getting the Most Out of this Guide

To get the most out of this guide, we suggest that you start reading the User Tasks section, which deals with handling all the product functions.

The Workbench Description section, which describes the Electrical 3D Design & Documentation workbench, and the Customizing section, which explains how to set up the options, will also certainly prove useful.

Navigating in the Split View mode is recommended. This mode offers a framed layout allowing direct access from the table of contents to the information.

Accessing Sample Documents

To perform the scenarios, sample documents are provided all along this documentation. For more information about this, refer to Accessing Sample Documents in the Infrastructure User's Guide.

Conventions Used in this Guide

To learn more about the conventions used in the documentation, refer to the Conventions section.

Conventions

Certain conventions are used in CATIA, ENOVIA & DELMIA documentation to help you recognize and understand important concepts and specifications.

Graphic Conventions

The three categories of graphic conventions used are as follows:

- Graphic conventions structuring the tasks
- Graphic conventions indicating the configuration required
- Graphic conventions used in the table of contents

Graphic Conventions Structuring the Tasks

Graphic conventions structuring the tasks are denoted as follows:

This icon...	Identifies...
	estimated time to accomplish a task
	a target of a task
	the prerequisites
	the start of the scenario
	a tip
	a warning
	information
	basic concepts
	methodology
	reference information
	information regarding settings, customization, etc.
	the end of a task

 functionalities that are new or enhanced with this release

 allows you to switch back to the full-window viewing mode

Graphic Conventions Indicating the Configuration Required

Graphic conventions indicating the configuration required are denoted as follows:

This icon...	Indicates functions that are...
	specific to the P1 configuration
	specific to the P2 configuration
	specific to the P3 configuration

Graphic Conventions Used in the Table of Contents

Graphic conventions used in the table of contents are denoted as follows:

This icon...	Gives access to...
	Site Map
	Split View mode
	What's New?
	Overview
	Getting Started
	Basic Tasks
	User Tasks or the Advanced Tasks
	Workbench Description
	Customizing
	Reference
	Methodology
	Glossary

Text Conventions

The following text conventions are used:

- The titles of CATIA, ENOVIA and DELMIA documents *appear in this manner* throughout the text.
- **File** -> **New** identifies the commands to be used.
- Enhancements are identified by a blue-colored background on the text.

How to Use the Mouse

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

Use this mouse button...	Whenever you read...
	• Select (menus, commands, geometry in graphics area, ...) • Click (icons, dialog box buttons, tabs, selection of a location in the document window, ...) • Double-click • Shift-click • Ctrl-click • Check (check boxes) • Drag • Drag and drop (icons onto objects, objects onto objects)
	• Drag • Move
	• Right-click (to select contextual menu)

User Tasks

The User Tasks section explains and illustrates how to create various kinds of features.
The table below lists the information you will find.

Using Electrical 3D Design & Documentation

This task explains how to set up the environment to work with CATIA - Electrical 3D Design & Documentation. CATIA - Electrical 3D Design & Documentation includes two workbenches:


Electrical 3D Design Assembly workbench to work at the level of an assembly of electrical objects.


Electrical 3D Design Part workbench to add electrical behavior at the level of a part.

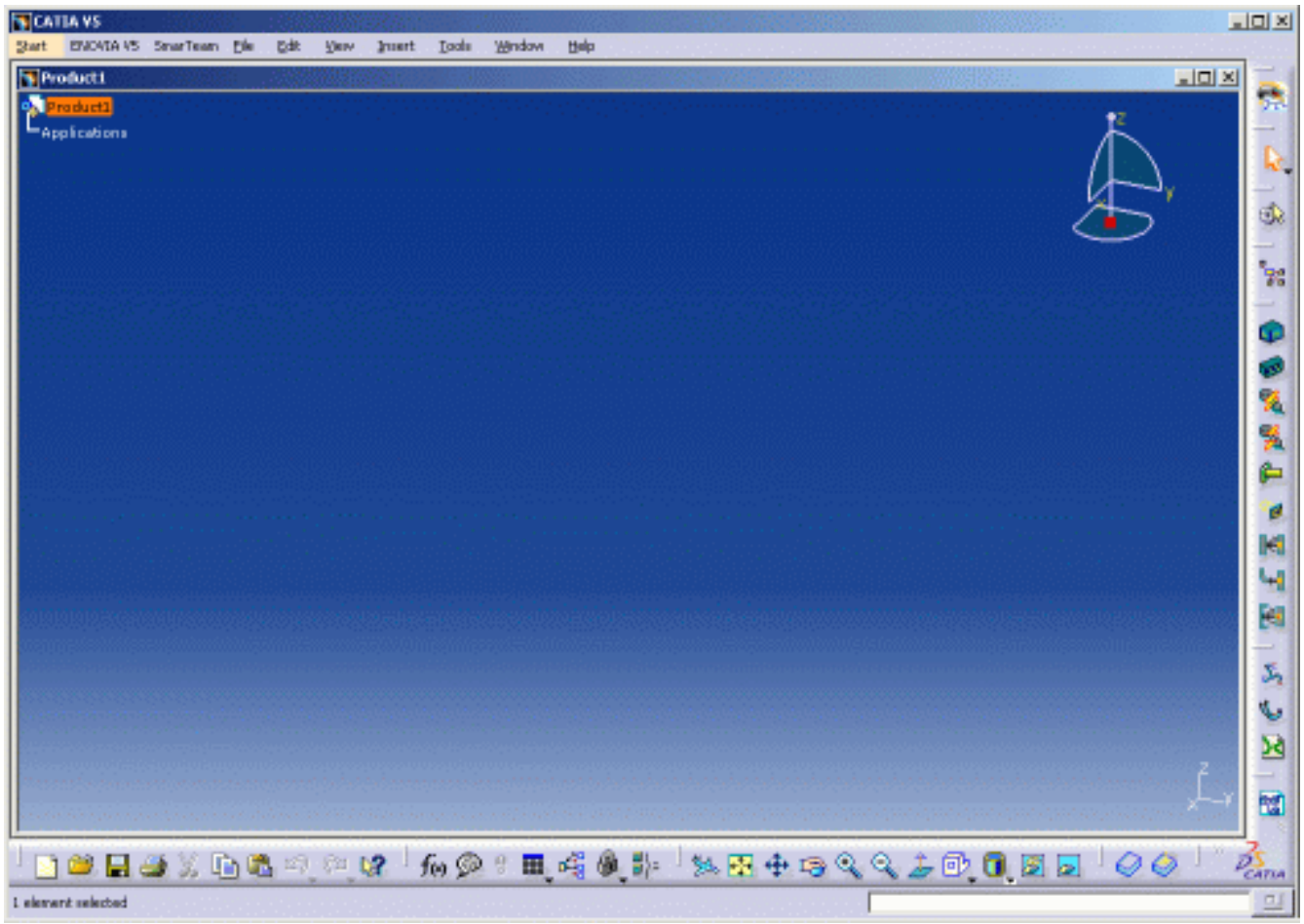
Entering the Electrical 3D Design Assembly Workbench

The Electrical 3D Design Assembly workbench allows you to create electrical assemblies in CATProduct documents.

CATIA V5 is launched. A CATProduct document is displayed.

1. Choose the **Electrical 3D Design Assembly** item from the **Start -> Equipments & Systems** menu.

   The Electrical 3D Design Assembly workbench is displayed and ready to use.



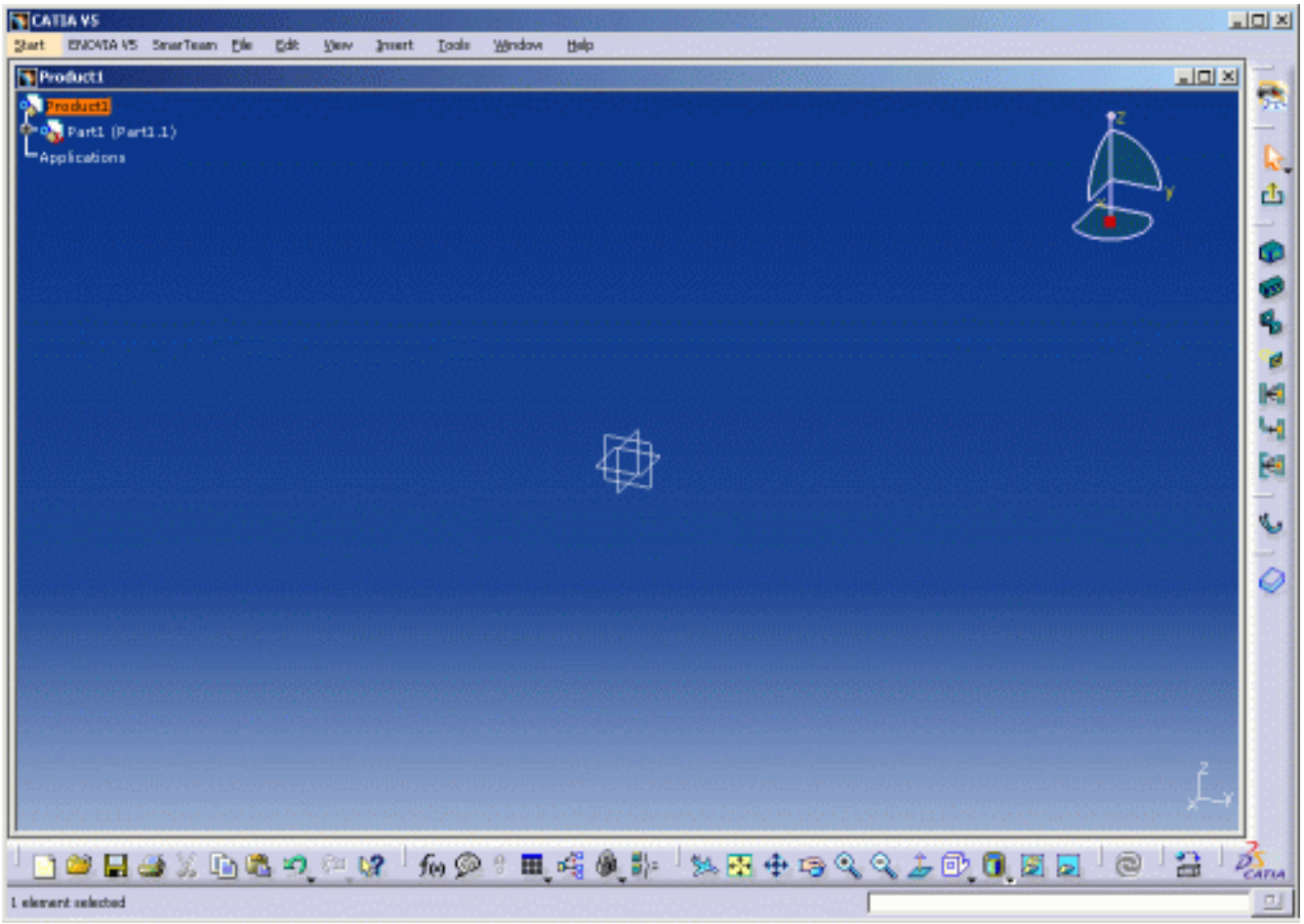
Using the Electrical 3D Design Part Workbench

The Electrical 3D Design Part workbench is used to define electrical connectors or to convert standard parts into electrical devices.

CATIA V5 is launched. A CATPart document is displayed.

1. Choose the **Electrical 3D Design Part** item from the **Start -> Equipments & Systems** menu.

The Electrical 3D Design Part workbench is displayed with a CATPart document.



Defining Electrical Devices

These functionalities are available in both the Electrical Assembly and Part workbenches.
They are used to add an electrical behavior to a product or a part.
As a result, the product itself, the instance or the reference becomes an electrical element:

- When you open a CATPart document and you add an electrical behavior to the part, it is the **reference** which will be modified. If you insert this part in an assembly, all the occurrences will be modified.
- When you open a CATProduct document containing a CATPart, if you double-click to activate the product of the part or the part, and add an electrical behavior to it, the result is similar: you have modified the **reference** and all the occurrences will have an electrical behavior. As a consequence, you will have to save the part.
- On the other hand, if the root product is activated and you select the product of the part, it is only this **instance** of the part which will become an electrical device. The reference is NOT modified.

This rule applies for all the devices.

Equipment: Click this button and select the part or product to be converted into an equipment.

Connector: Click this button and select the part or product to be converted into a connector.

Defining an Equipment

This task explains how to add an electrical behavior to a standard part.
An equipment is an electrical device with one or more associated connectors placed into cavities.

Let's see two different cases to illustrate:

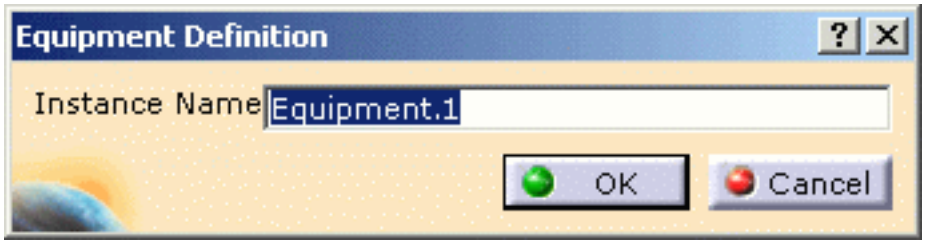
- first, you will work with an **instance** of the product, i.e. only this instance will get the electrical behavior.
  If you insert the part to another assembly, it remains a standard part.
- then, you'll work with the **reference** of the part, i.e. all the instances of the part will get the electrical behavior.

Adding the electrical behavior to an instance...

The CATProduct document contains standard parts, which do not have electrical behavior.

1. Click the **Define Equipment** button  .
   You are prompted to select a part or a product.
2. Click the equipment either in the specification tree or in the geometry.

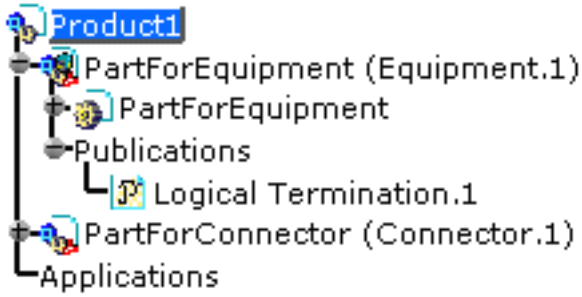
   The Define Equipment Part dialog box opens:



3. Enter/change the name in the **Instance name** field if necessary.
4. Click **OK** to validate.

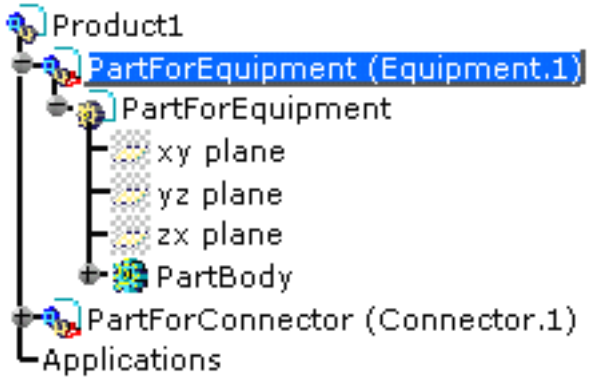
   The specification tree is updated. The part is not modified.

   The equipment is now an electrical object as the electrical behavior has been added to this instance.



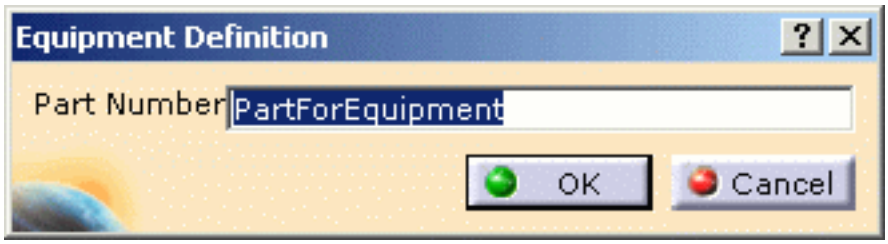
Adding the electrical behavior to the reference...

Now if you open the CATPart document or, if you double-click to activate the PartForEquipment document in the example shown below, you will modify the reference document.



1. Click the **Define Equipment** button  .
   You are prompted to select a part or a product.
2. Click the equipment (here PartForEquipment) either in the specification tree or in the geometry.

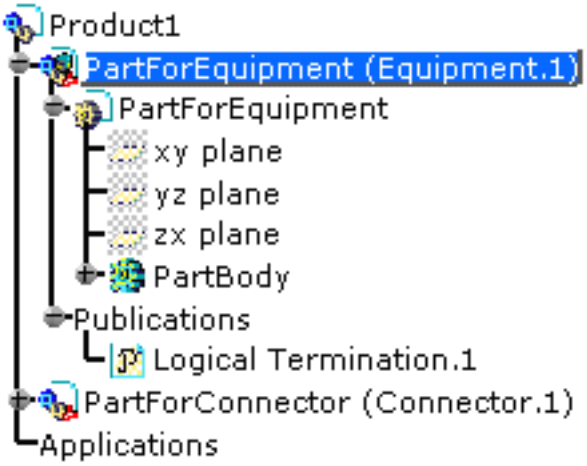
   The Equipment Part Definition dialog box opens:



   Note that it is the **Part Number**, which is displayed for edition.
3. Change the name in the **Part Number** field if necessary.
4. Click **OK** to validate.

   The specification tree is updated. The part has been modified.

   The equipment is now an electrical object as the electrical behavior has been added to this reference.



An **equipment** allows bundle connection points, cavities as well as single insert connectors connected into cavities through a cavity connection point.

Defining an Electrical Connector

This task explains how to add an electrical behavior to a standard part to turn it into an electrical connector.

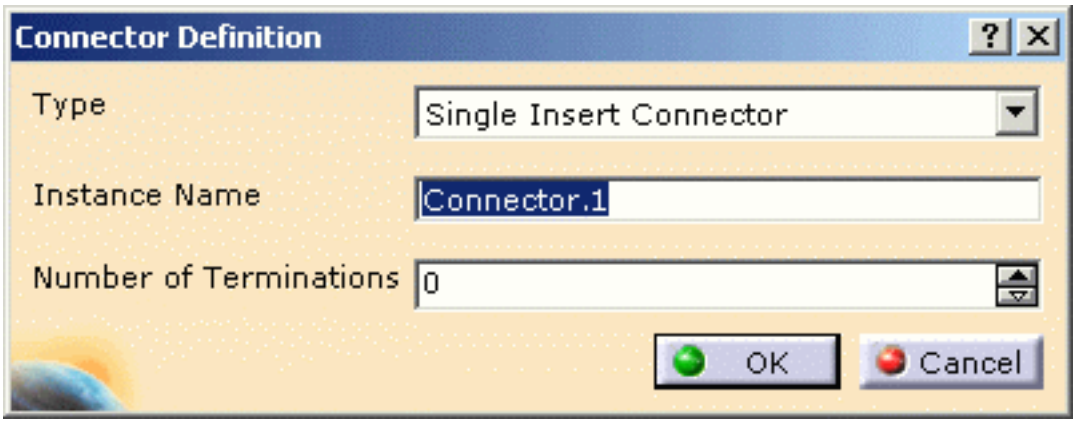
You can add an electrical behavior to an instance or a reference.

Refer to Defining an Equipment.

The CATProduct document contains standard parts, which do not have electrical behavior.

1. Click the **Define Connector** button  .
   You are prompted to select a part or a product.
2. Click the connector either in the specification tree or in the geometry.

   The Define Connector dialog box opens:

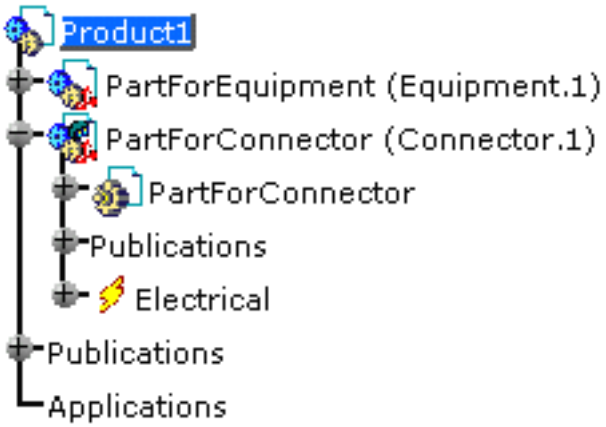


   The only type available is Single Insert Connector: male or female connector.
3. Change the name in the **Instance name** field if necessary.
4. Enter the **Number of termination** to be defined onto the connector.

   The electrical terminations are used to connect wires.
5. Click **OK** to validate.

   The specification tree is updated. The part is not modified.

   The instance of the connector is now an electrical object as the electrical behavior has been added.



A **single insert connector** allows one connector connection point, bundle connection points and terminations.

Defining Electrical Connection Points

The connection functionalities are available in both the Electrical Assembly and Part workbenches.


Cavity: Click this button and select the equipment where you want to define a cavity.


Termination: Click this button and select the device where you want to define a termination.


Bundle Connection Point: Click this button and set the placement constraints for the connection point between connectors and bundle segments.


Connector Connection Point: Click this button and set the placement constraints for the connection point between connectors.


Cavity Connection Point: Click this button and set the placement constraints for the connection point between cavities and equipments.

It is possible to delete the electrical cavities, terminations and the connection points.
This applies to:

- termination
- cavity
- cavity connection point
- connector connection point
- bundle connection point.

Note that when deleting these objects, the associated publications are also deleted.

But the publications of the geometries which constrain their placement are NOT automatically deleted, since they may have been created earlier, from another application.
According to your choice, you can delete them using the publication management available in the Assembly Design workbench (**Tools -> Publication** menu item).

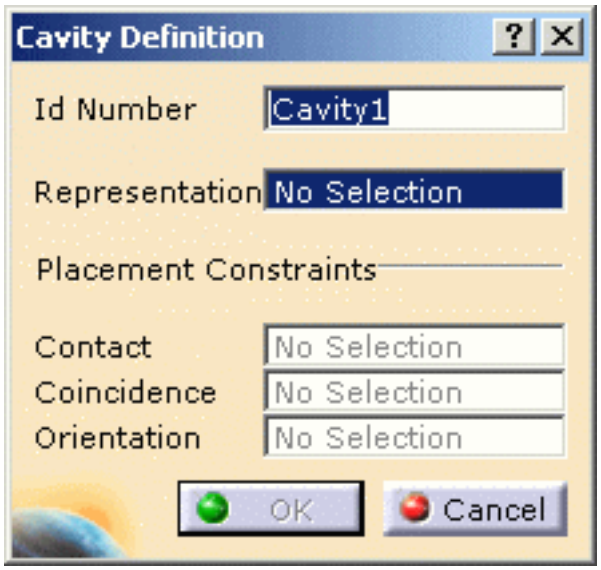
Defining a Cavity

This task explains how to define a cavity on an electrical device.
The cavity is used to specify the location of the electrical object when connecting.

Open any document containing an equipment where you want to place a cavity.

1. Click the **Define Cavity** button .

   The Cavity Definition dialog box opens:

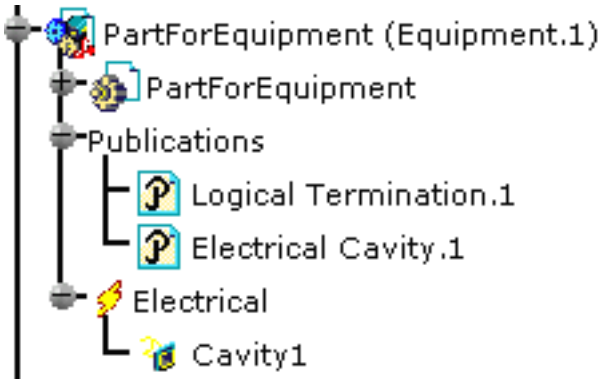


2. Select the equipment where you want the cavity to be defined.
3. Change/enter a value in the **IdNumber** field.

   Note that the IdNumber must be unique.
4. Select a **Representation**, for example a pocket, a pad, a face...
5. Optionally, place a **Contact** constraint, for example a surface or a point.
6. Optionally, place a **Coincidence** constraint, for example a surface, a line or an axis.
7. Optionally, place an **Orientation** constraint, for example a surface, a line or an axis.

   The orientation is used to constrain the rotation i.e. the third degree of liberty.
8. Click **OK** to validate.

   The specification tree is updated.



For more information about the placement constraints, refer to Using Assembly Constraints.

A **cavity** is allowed on equipments.

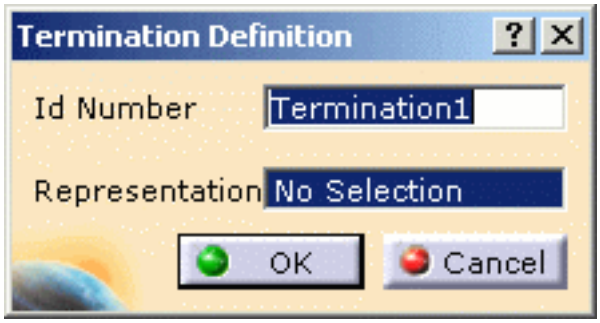
Defining a Termination

This task explains how to define a termination on a connector.
The termination is indissociable from the electrical component and corresponds to a contact crimped into a cavity.

Open any document containing a connector where you want to place a termination.

1. Click the **Define Termination** button .
2. Select the electrical device where you want the termination to be defined.

   The Termination Definition dialog box opens:

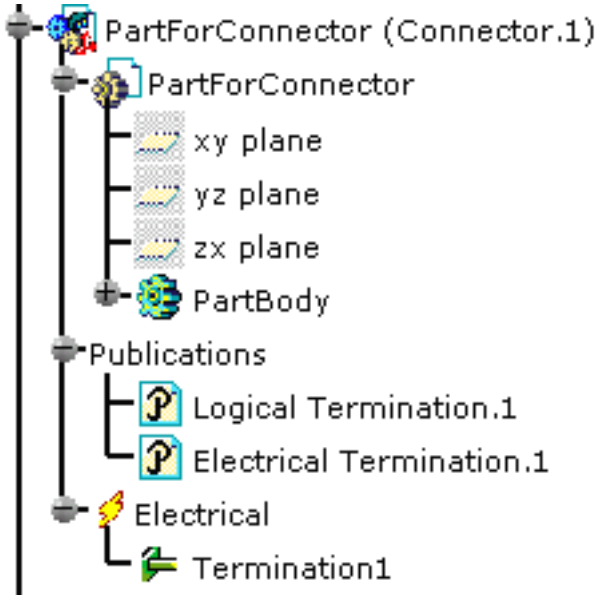


3. Change/enter a value in the **IdNumber** field.

   Note that the IdNumber must be unique.

4. Optionally select a **Representation**, for example a pocket, a pad, a face...
5. Click **OK** to validate.

   The specification tree is updated.



A **termination** is allowed on connectors.

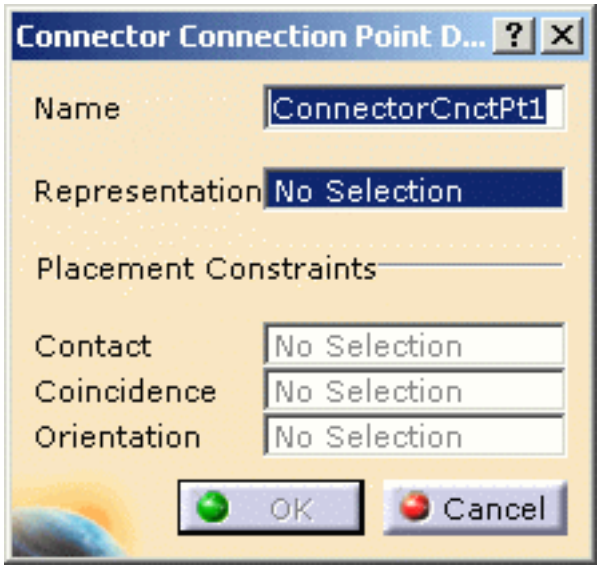
Defining a Connector Connection Point

This task explains how to define a connector connection point on a connector.
The connector connection point is used to specify how the mating connectors are placed when connecting.

Open any document containing a single insert connector where you want to place a connector connection point.

1. Click the **Define Connector Connection Point** button .
2. Select the electrical device where you want the connector connection point to be defined.

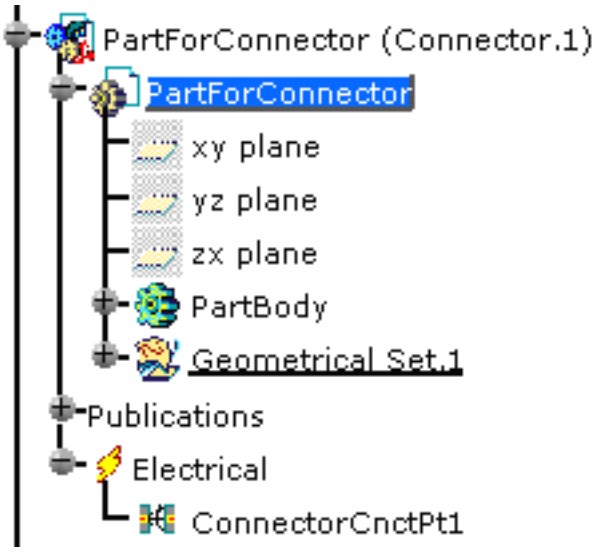
   The Connector Connection Point Definition dialog box opens:



3. Change/enter a value in the **Name** field.
4. Select a **Representation**, for example a pocket, a pad, a face...
5. Optionally, place a **Contact** constraint, for example a surface or a point.
6. Optionally, place a **Coincidence** constraint, for example a surface, a line or an axis.
7. Optionally, place an **Orientation** constraint, for example a surface, a line or an axis.

   The orientation is used to constrain the rotation i.e. the third degree of liberty.
8. Click **OK** to validate.

   The specification tree is updated.



For more information about the placement constraints, refer to Using Assembly Constraints.

A **connector connection point** is only allowed on single insert connectors.

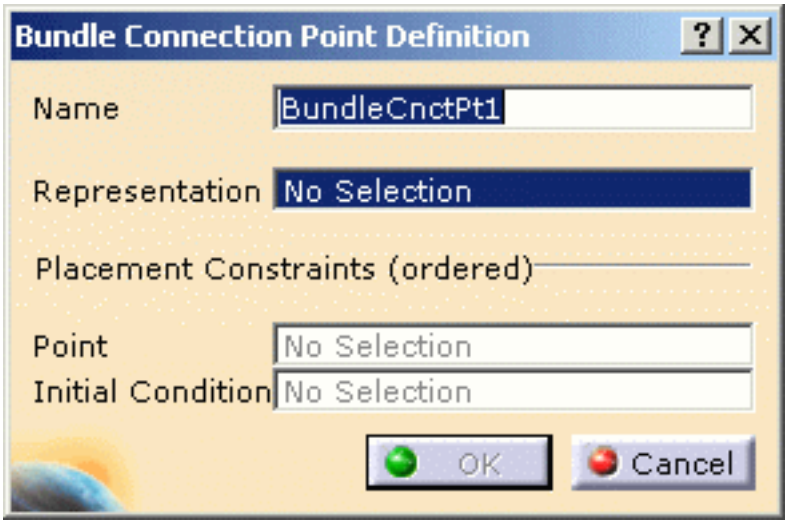
Defining a Bundle Connection Point

This task explains how to define a bundle connection point on a device.
The bundle connection point is used to specify the position and the direction of the bundle segment connected to the connector or the equipment.

Open any document containing a connector where you want to place a bundle connection point.

1. Click the **Define Bundle Connection Point** button .
2. Select the connector or equipment where you want the bundle connection point to be defined.

   The Bundle Connection Point Definition dialog box opens:

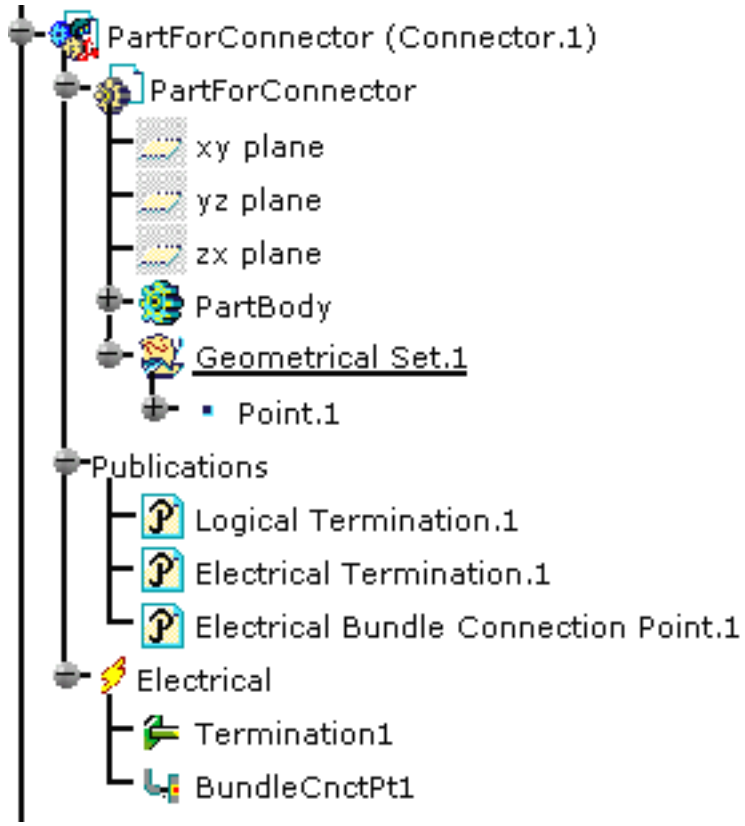


3. Change/enter a value in the **Name** field.
4. Select a **Representation**, for example a surface or a point.
5. Set a **Point** constraint: select a point.

   This point will possibly be used as bundle segment extremity.
6. Set a **Initial Condition** constraint: select a plane or an axis.

   This plane or axis will possibly be used to orientate the bundle segment.
7. Click **OK** to validate.

   The specification tree is updated.



A **bundle connection point** is allowed on equipments and connectors.

Defining a Cavity Connection Point

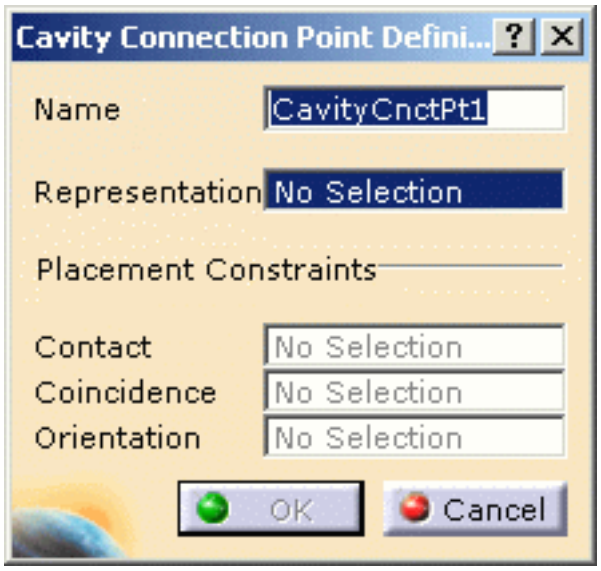
This task explains how to define a cavity connection point on an equipment.
The cavity connection point is used to specify how the connector is placed in the cavity when connecting.

Open any document containing an equipment and a connector, where you want to place a cavity connection point.

1. Click the **Define Cavity Connection Point** button .
2. Select the connector where you want the cavity connection point to be defined.

   The Cavity Connection Point Definition

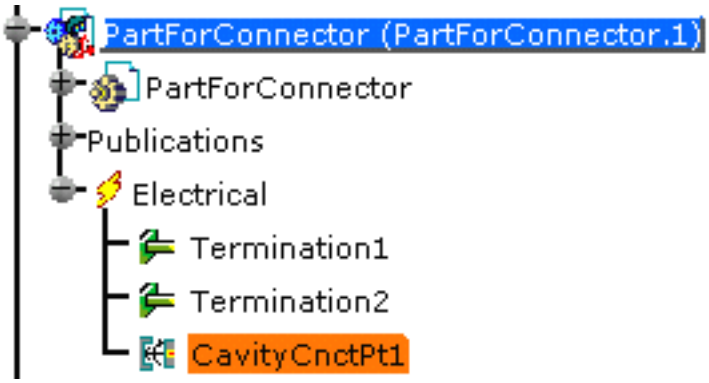
   dialog box opens:



3. Change/enter a value in the **Name** field.
4. Select a **Representation**, for example a pocket, a pad, a face.
5. Optionally, set a **Contact** constraint, for example a surface or a point.
6. Optionally, set a **Coincidence** constraint, for example a surface, a line or an axis.
7. Optionally, set an **Orientation** constraint, for example a surface, a line or an axis.

   The orientation is used to constrain the rotation i.e. the third degree of liberty.
8. Click **OK** to validate.

   The specification tree is updated.



For more information about the placement constraints, refer to Using Assembly Constraints.

A **cavity connection point** is allowed on connectors.

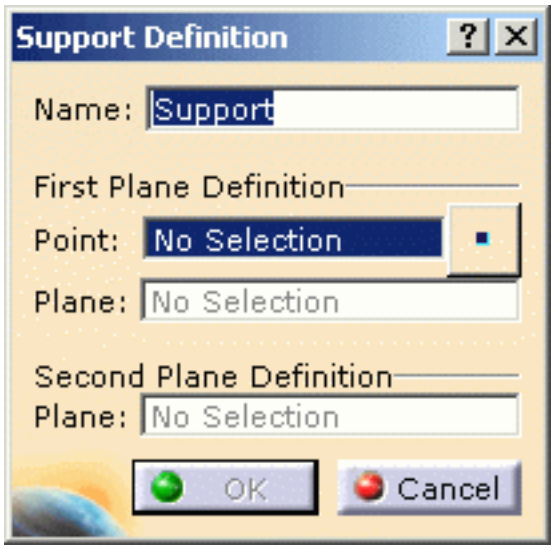
Creating Supports

This task explains how to define a support for electrical bundle segments.
This functionality is available in Electrical Part Design workbench only.
A support is a mechanical object used to hold the bundle segments in position.

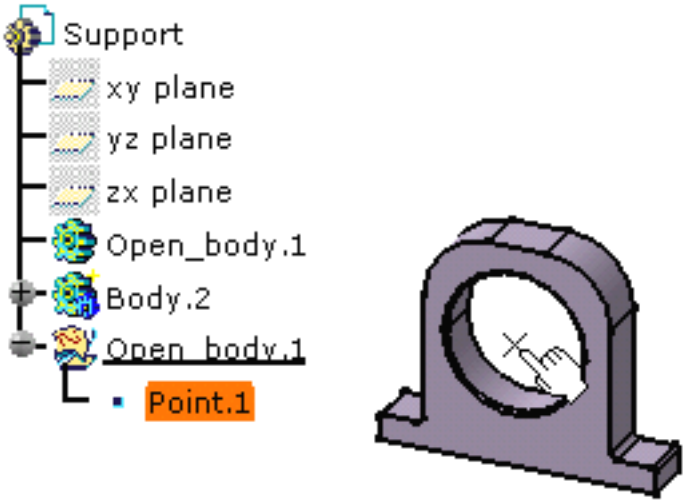
Open any document containing a standard part (with no electrical behavior).

1. Click the **Define Support** button  .
   You are prompted to select a part.
2. Click the **Support** either in the specification tree or in the geometry.

   The Support Definition dialog box opens:



3. Enter/change the name for the support.
4. Select the point through the support, to define the reference position of the bundle segment in the support.



If the point is not already defined, click the  button.
For more information, refer to the Point Definition.

You are prompted to select the first plane: the way in of the support.

5. Select the front face.

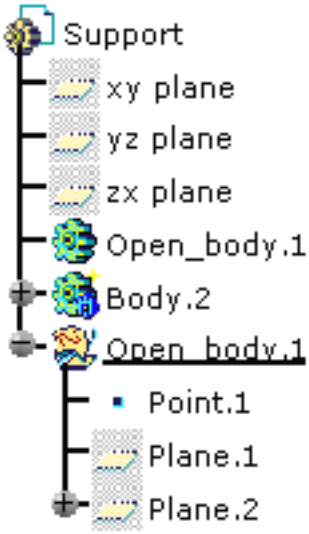
You are prompted to select a second plane: the way out of the support.

6. Select the opposite face.

To be selectable, it must be parallel to the first face.

7. Click **OK** to validate.

The specification tree is updated:
two extra planes have been defined (**Plane.1** and **Plane.2**).



Connecting/Disconnecting Electrical Devices

These functionalities are only available in the Electrical Assembly workbench.

Connecting: Click this button and select the devices you want to connect.

Disconnecting: Click this button and select the devices you want to disconnect .

Connecting Electrical Devices

This task explains how to connect the electrical devices.

When you create an electrical connection between two devices:

- an electrical link is created between the connected components. See Related Objects.
- if placement constraints have been defined on connection points, the mechanical assembly constraints are automatically created.

Since it's possible to use this command in visualization mode (with the cache activated), the scenario is described according to the two modes:

- The design mode is mandatory to perform the connections.
- In visualization mode, the model is lighter since the geometry is not loaded.
  It allows you to display large assemblies.
  When you work in visualization mode, CATIA switches to the design mode only for the elements which contribute to the connection.

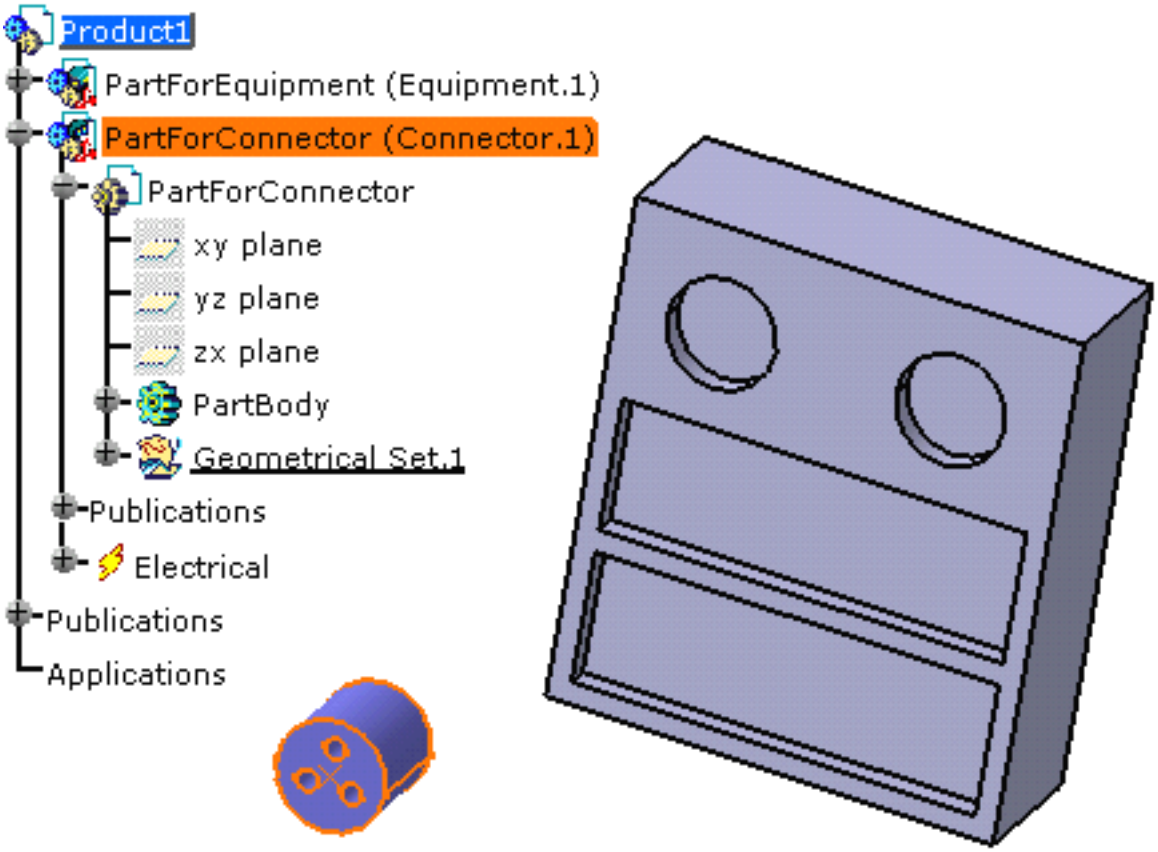
In Design Mode

If you open the document in design mode (the cache is not activated), the whole geometry is loaded.

1. Click the **Connect Electrical Devices** button  to connect two devices.
   You are prompted to select the first device.

   - it's the first selected object that moves to the second one's location
   - an electrical connection has been created as well as mechanical constraints.

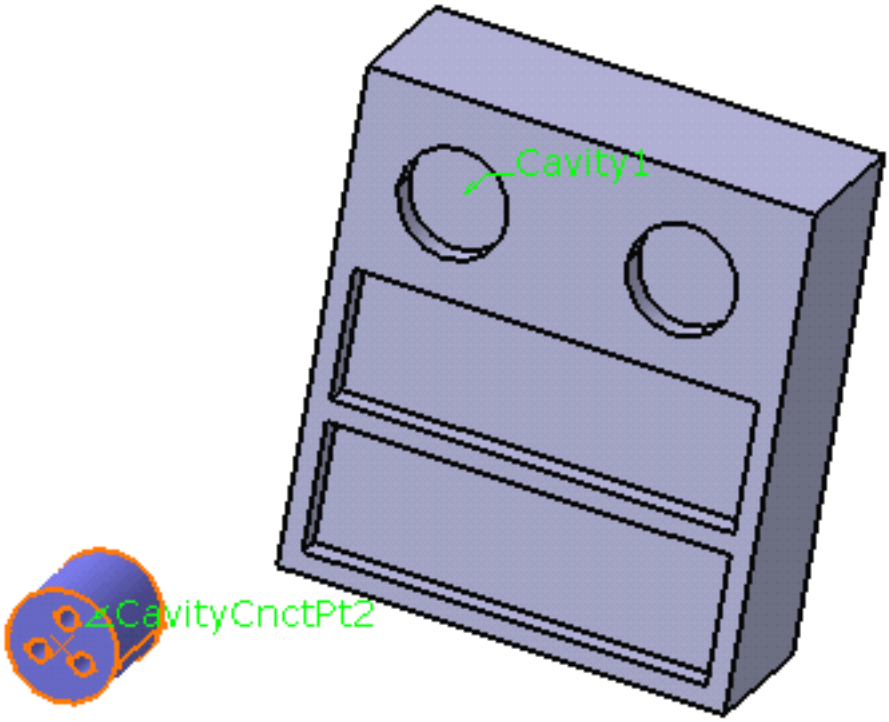
2. Select the first device either in the geometry or in the specification tree:



You are prompted to select another device.

3. Move the cursor onto the second device before selecting it, as shown below:

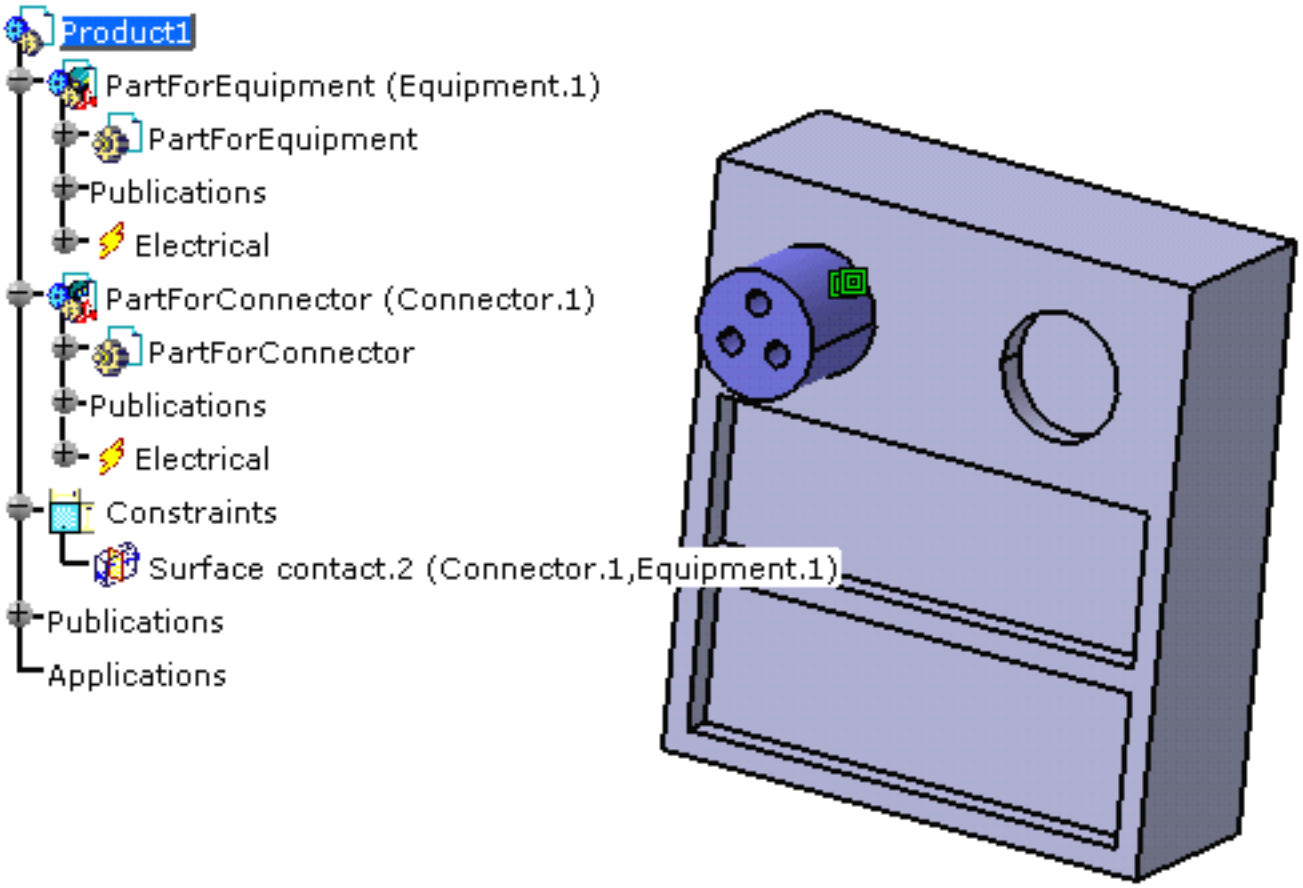
The cavity connection point of the first device and the cavities of the second one are displayed in green: this means that it is possible to connect the connector into the cavity of the equipment.



According to the selection, if a cavity is already used or if no connection point is defined on the equipment, it is displayed in red.

4. Click to select a connection point available on the equipment, **Cavity1** for example.

The devices are connected together and the mechanical constraints are added to the specification tree:



Note that you can select the second connection point using one of the three following ways to get the same result:

- the representation of the connection point in the geometry
- the label with the name of the connection point
- the connection point in the specification tree.

In Visualization Mode

If you open the document in visualization mode (the cache is activated), the geometry is not loaded.

1. Select the **Connect Electrical Devices** button  to connect two devices.

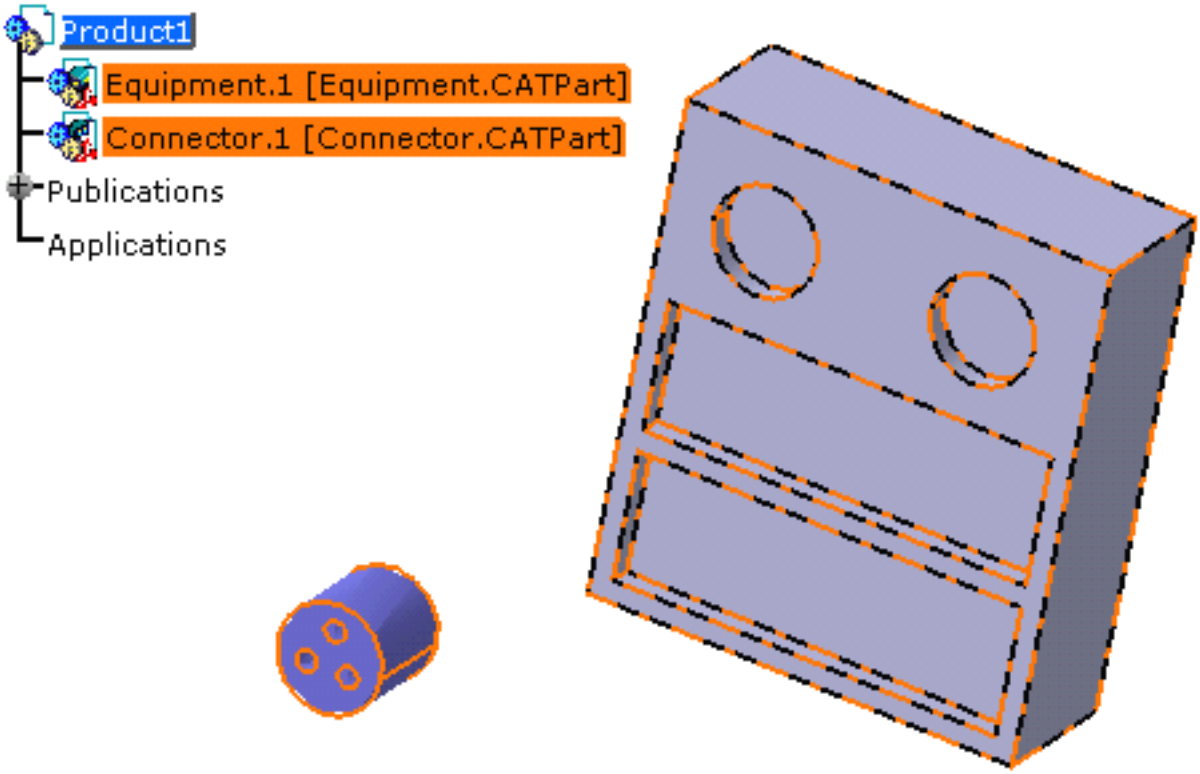
   You are prompted to select the first device.

   - it's the first selected object that moves to the second one's location
   - an electrical connection has been created together with mechanical constraints.

2. Select the first device either in the geometry or in the specification tree.

   You are prompted to select the second device.

3. Move the cursor onto the second device before selecting it, as shown below:

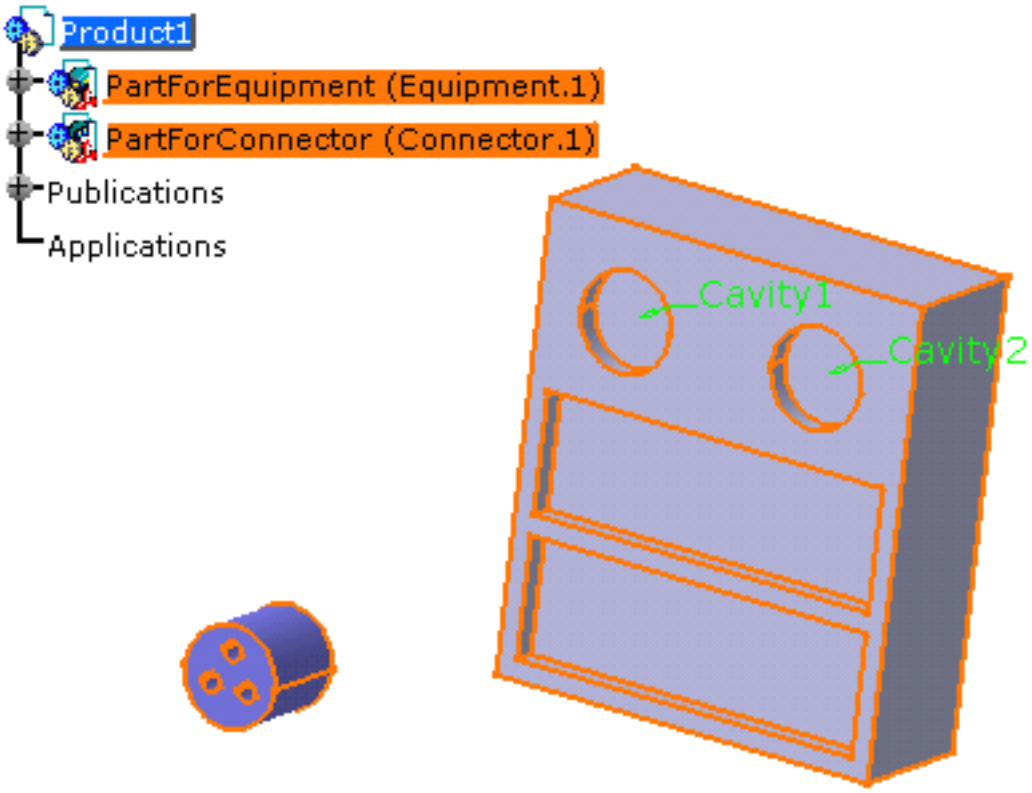


Note that:

- You cannot expand the specification tree since you are in visualization mode.
- No annotations are displayed on the device connection points.

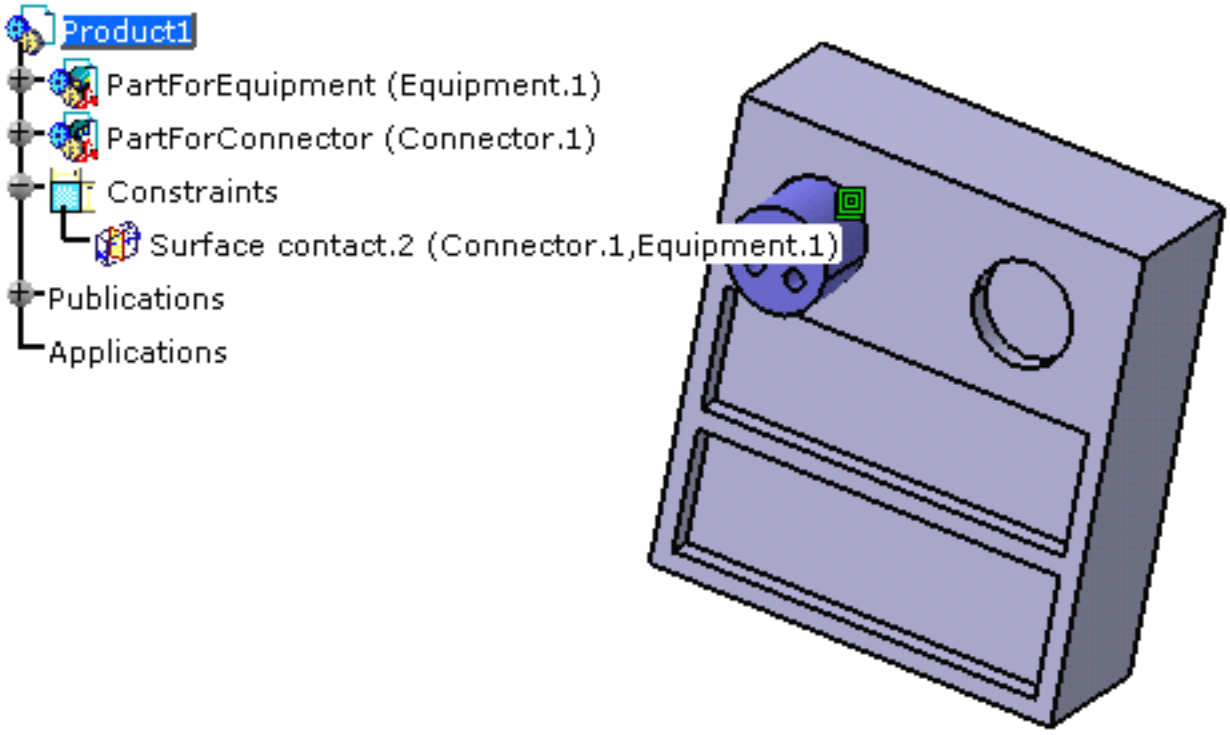
4. Click to select the equipment:

   The geometry is loaded for both devices and the annotations display:

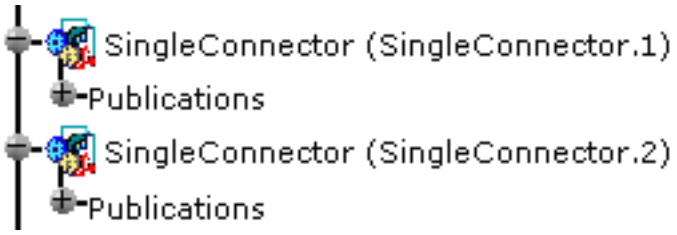


5. Click to select a connection point available on the device, for example **Cavity1**.

The devices are connected together and the mechanical constraints are added to the specification tree:



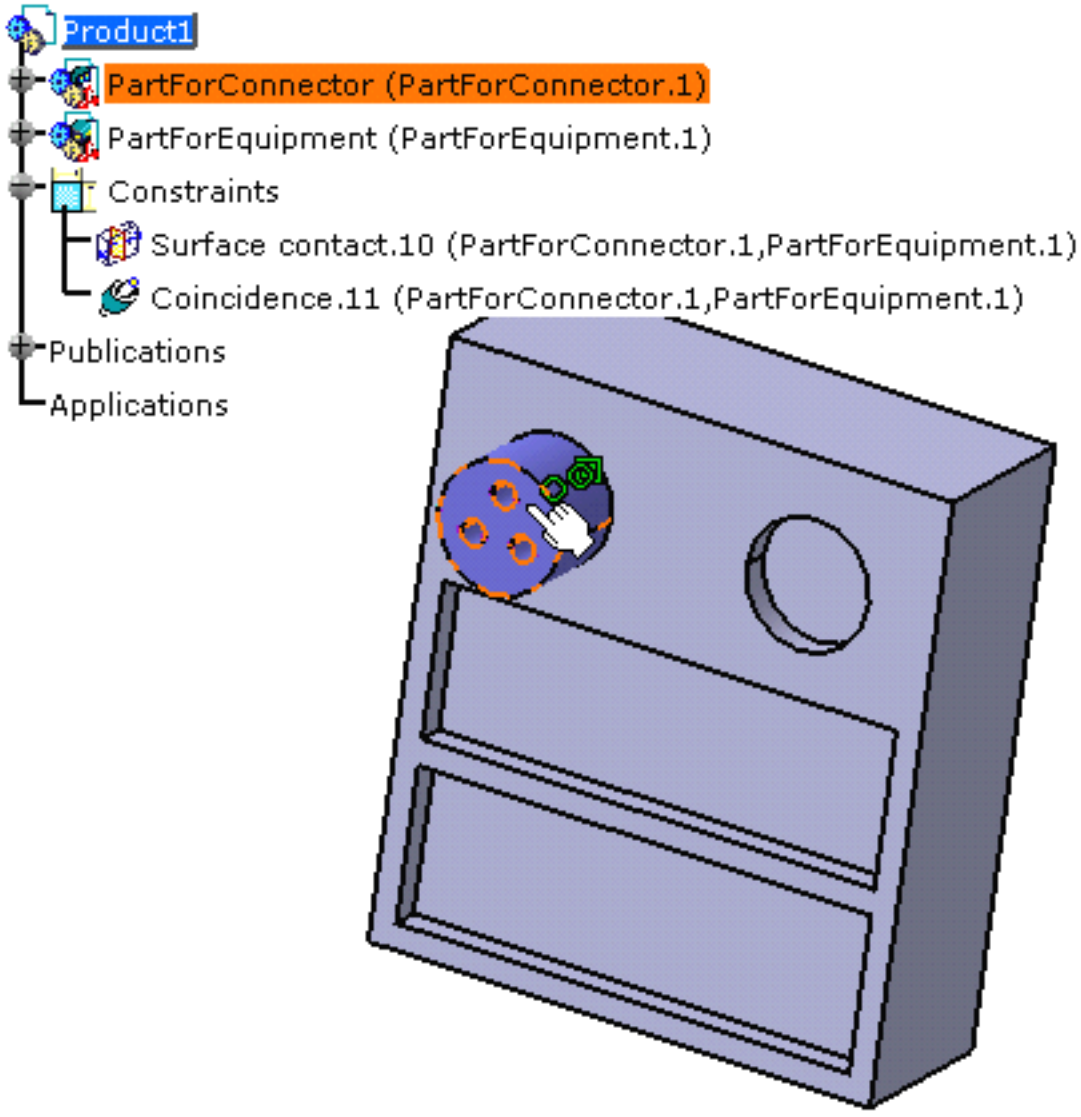
Note that the specification tree also displays the plus sign for the other devices: this is due to the update, which loads the publications. However the geometry for these components is not loaded, as shown in the picture below:



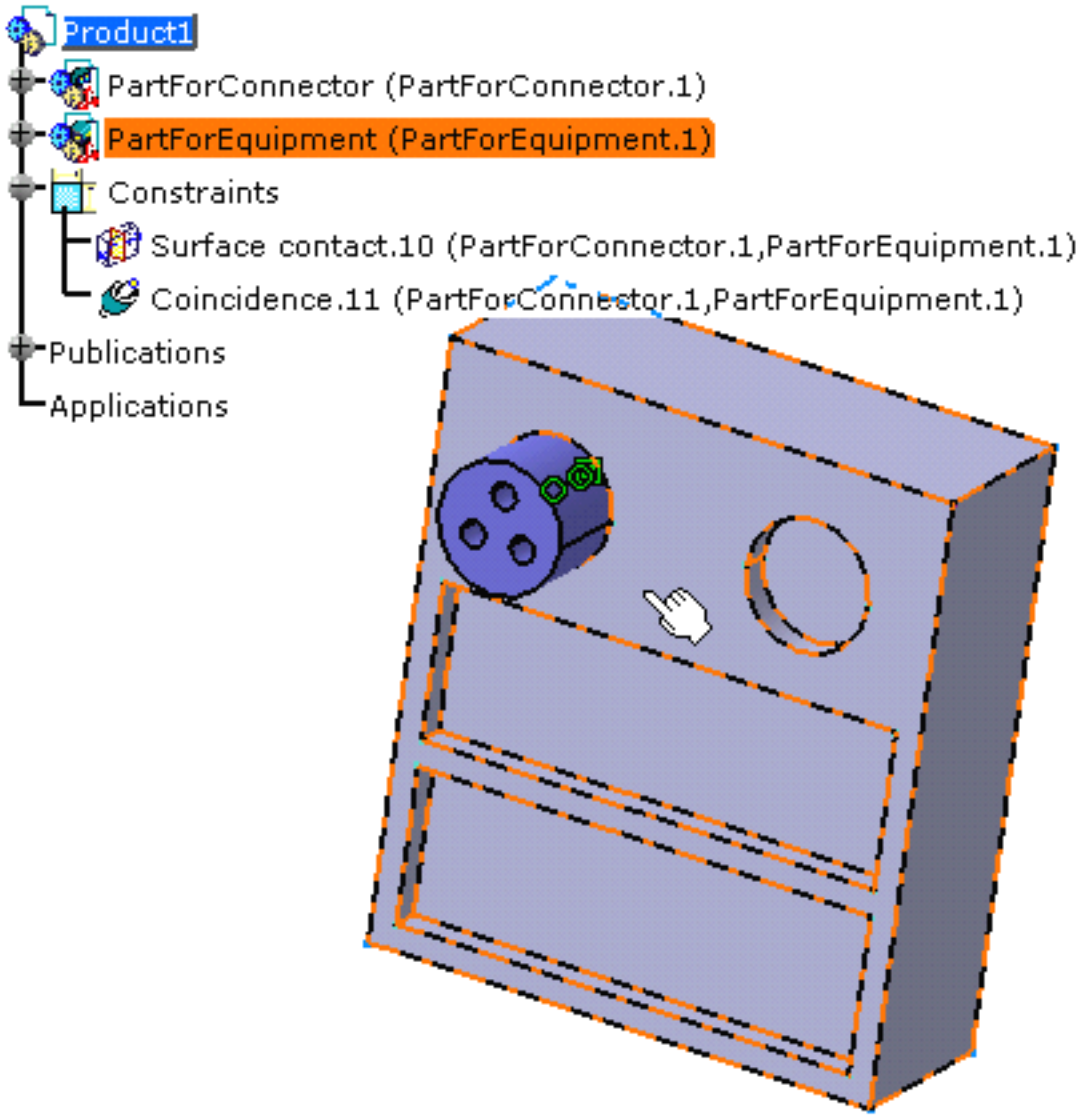
Disconnecting Electrical Devices

This task explains how to disconnect electrical devices.

1. Select the **Disconnect Electrical Devices** button  to disconnect devices.
   You are prompted to select the first device.
2. Select the first device you want to disconnect:

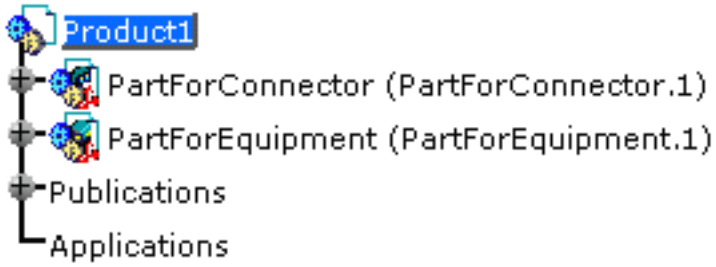


   You are prompted to select another device.
3. Select the second device that was connected to the first one:



The devices are disconnected.

- The electrical connection is deleted.
- The mechanical constraints are deleted in the geometry and the specification tree:



Note that the device position remains unchanged but as the mechanical constraints have been deleted, you can shift the connectors using the compass.

Using Catalogs

These functionalities are available in the Electrical Assembly workbench.
The first one is also available in the Electrical Part workbench.

 Connecting Devices: Drag and drop the device from the catalog at its proper placing.

 Using Smart Placement: Select the component, point to the correct position and click to place it.

Connecting Devices by Drag and Drop at Placement

This task explains how to place devices by drag and drop with automatic connection to an existing component.

Open a new product document.

1. Click the **Catalog Browser** button to open the ElecIntegration.catalog.

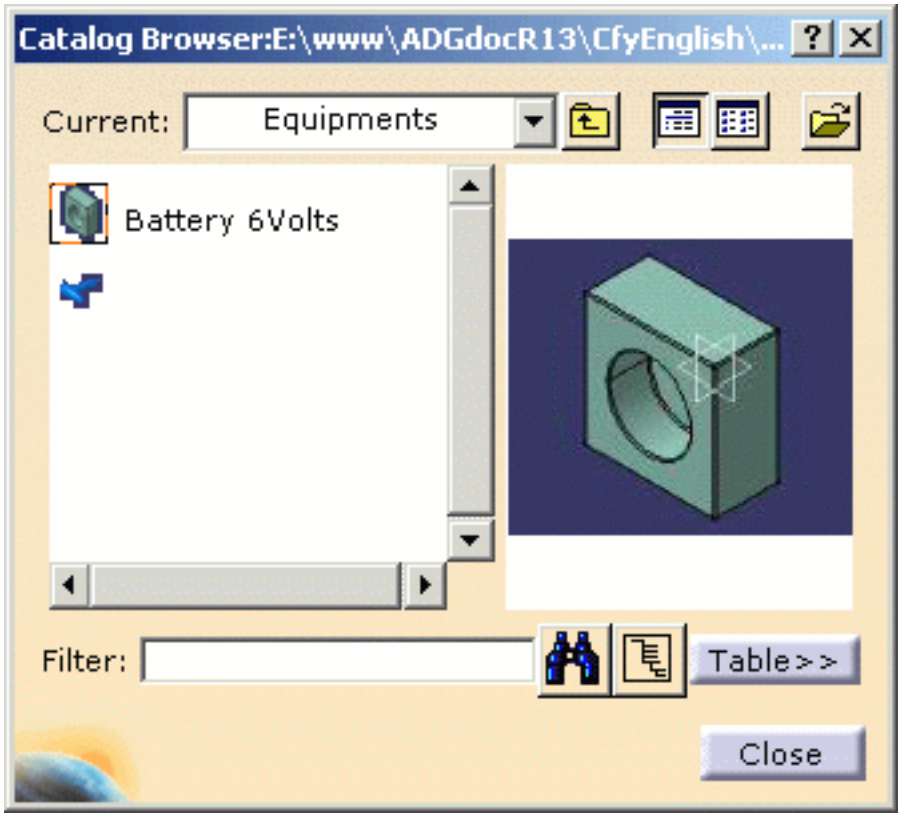
   The full path is:
   .../online/cfysa_C2/samples/ElectricalIntegration\ElecIntegration.catalog

   The dialog box opens on **Chapter.1**.

   If necessary, use the **Open** button to browse another catalog and select the ElecIntegration.catalog from the samples folder.

   To know more about the Catalog workbench, refer to Using Catalogs.

2. Double-click the **Equipments**: the folder contents displays.

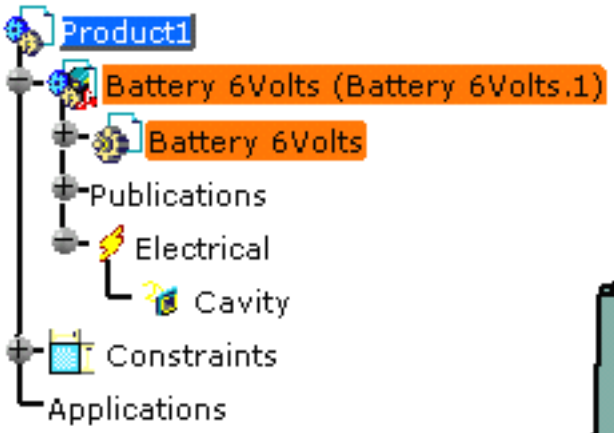


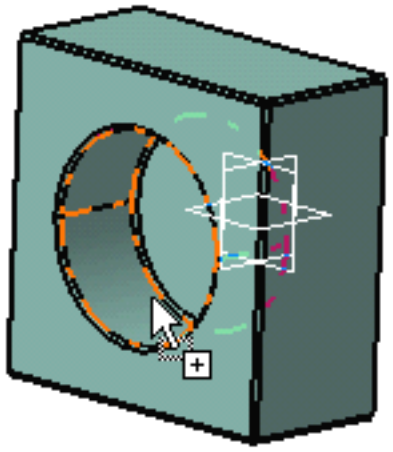
3. Select the **Battery 6volts**, drag and drop it onto the **Product1** in the specification tree.

   The equipment is imported.

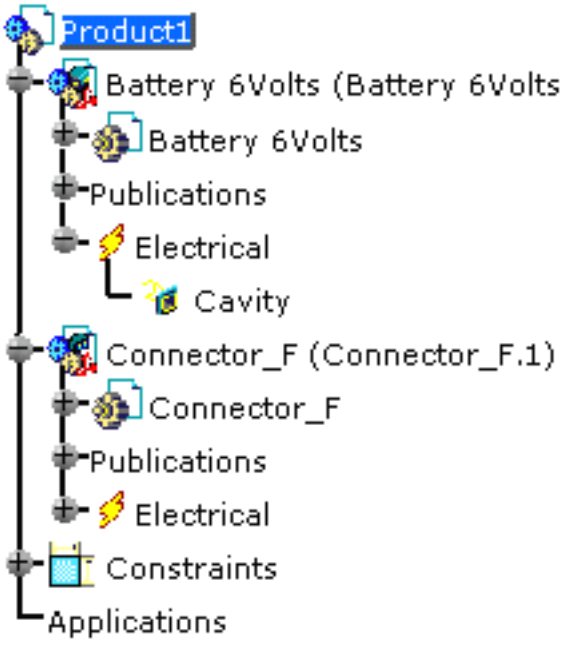
4. Select **Connectors** as Current folder, then

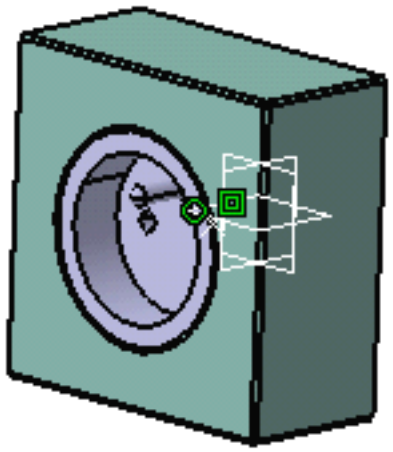
5. Select the **Connector_F1**, drag and drop it onto the cavity.





The connector is properly located into the cavity: the constraints are created.
The result looks like this:





The component is instantiated under the active product.

6. **Close** the Catalog Browser.

Note that when you insert a device into a product, you can take advantage of CATIA - Assembly capabilities to constrain the device within the digital mock-up, profiting therefore by the associativity. For more information, refer to Using Assembly Constraints.

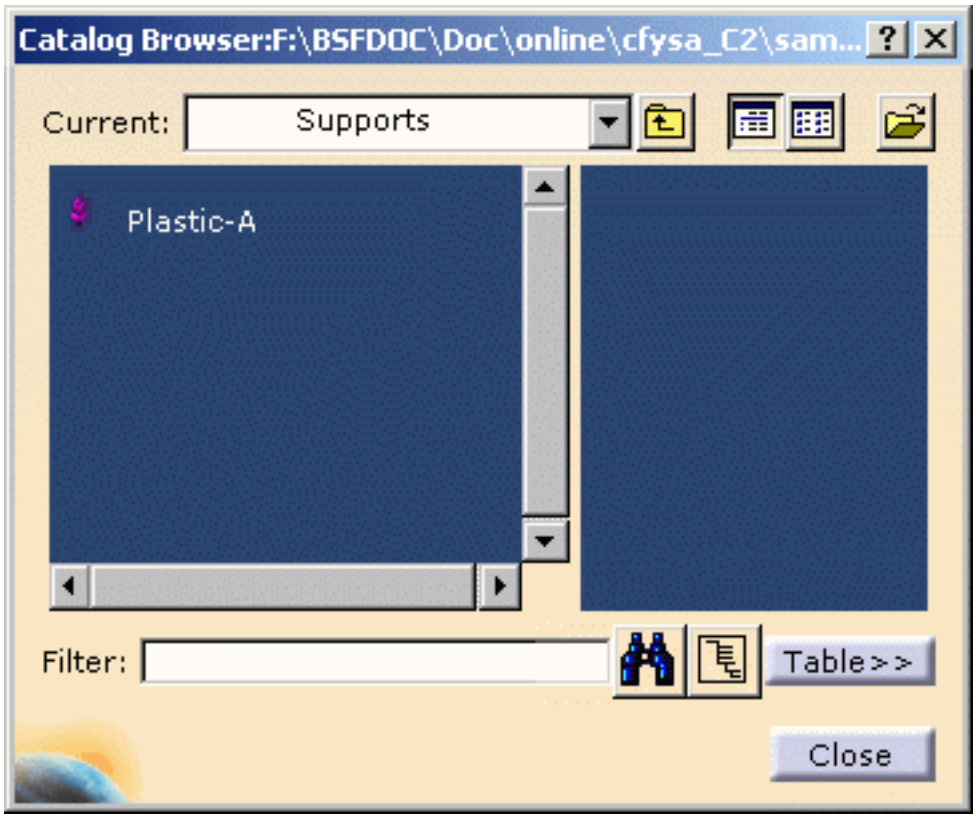
Using Smart Placement from Catalog

This task explains how to place a component from a catalog using a compass.

Make sure you have set up the automatic compass option.

1. Click the **Smart Place** button .

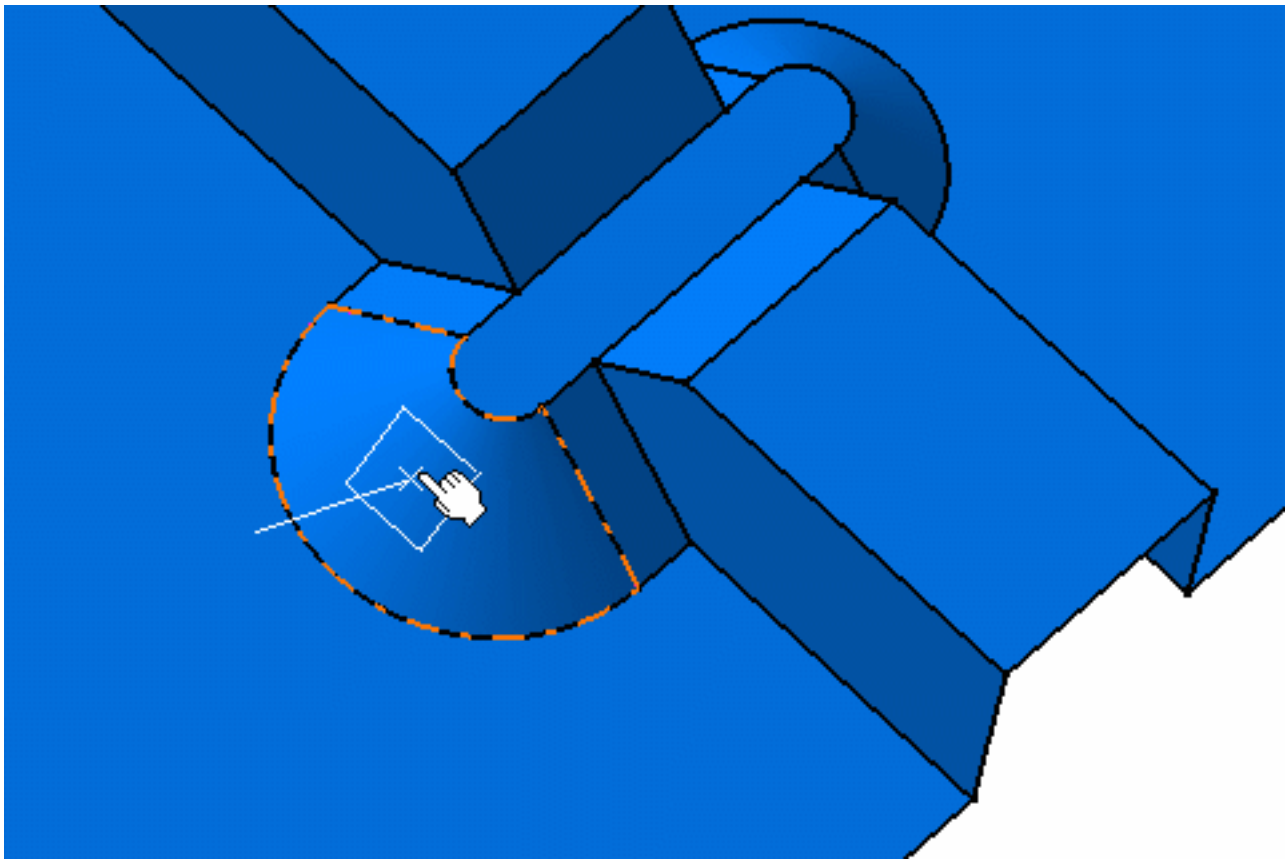
   The Catalog Browser displays.



   If necessary, navigate to select the catalog of interest using the **Browse another catalog** button .

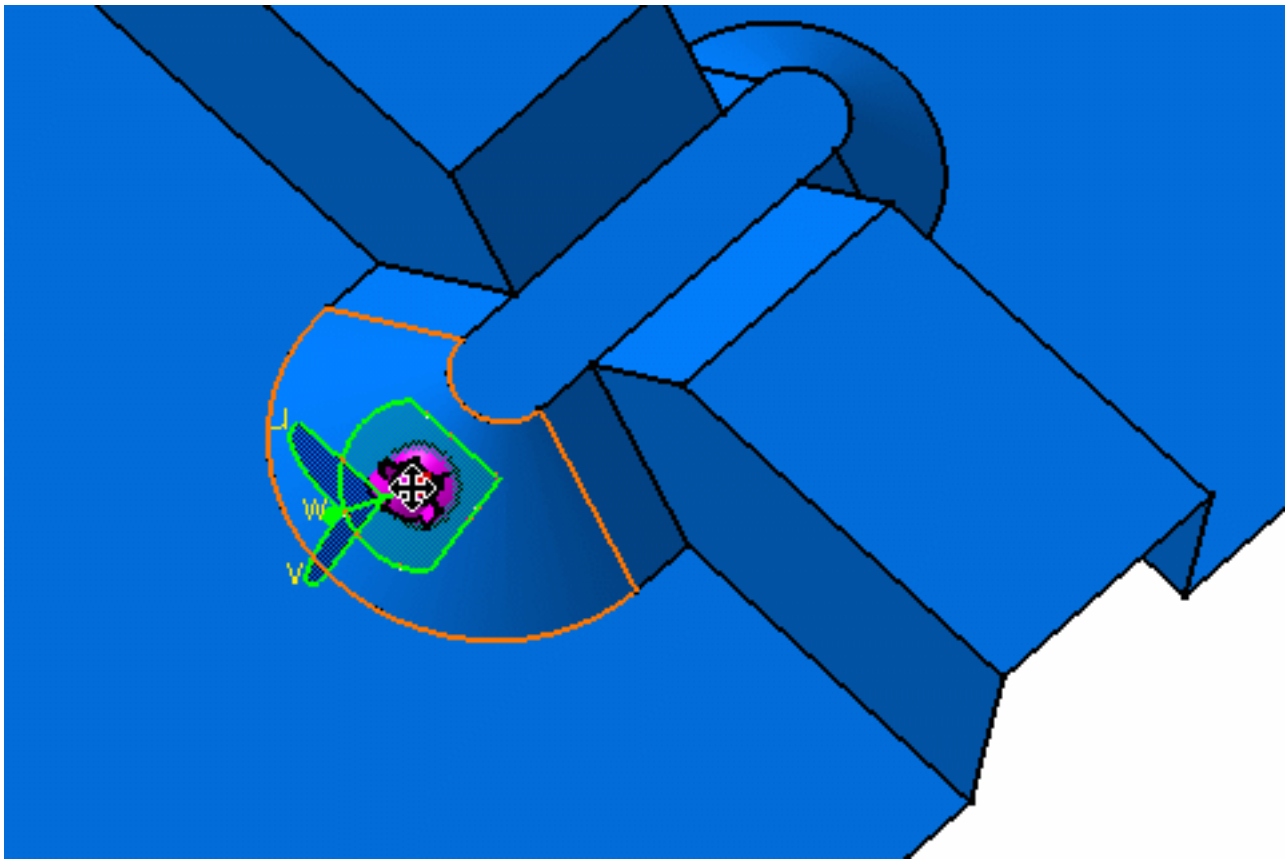
2. Select the part you want to place.

   In the graphic area, a manipulator displays allowing you to select:
   - the placement point
   - the privileged plane
   - the direction. By pressing the **Shift** key, you can invert the direction.

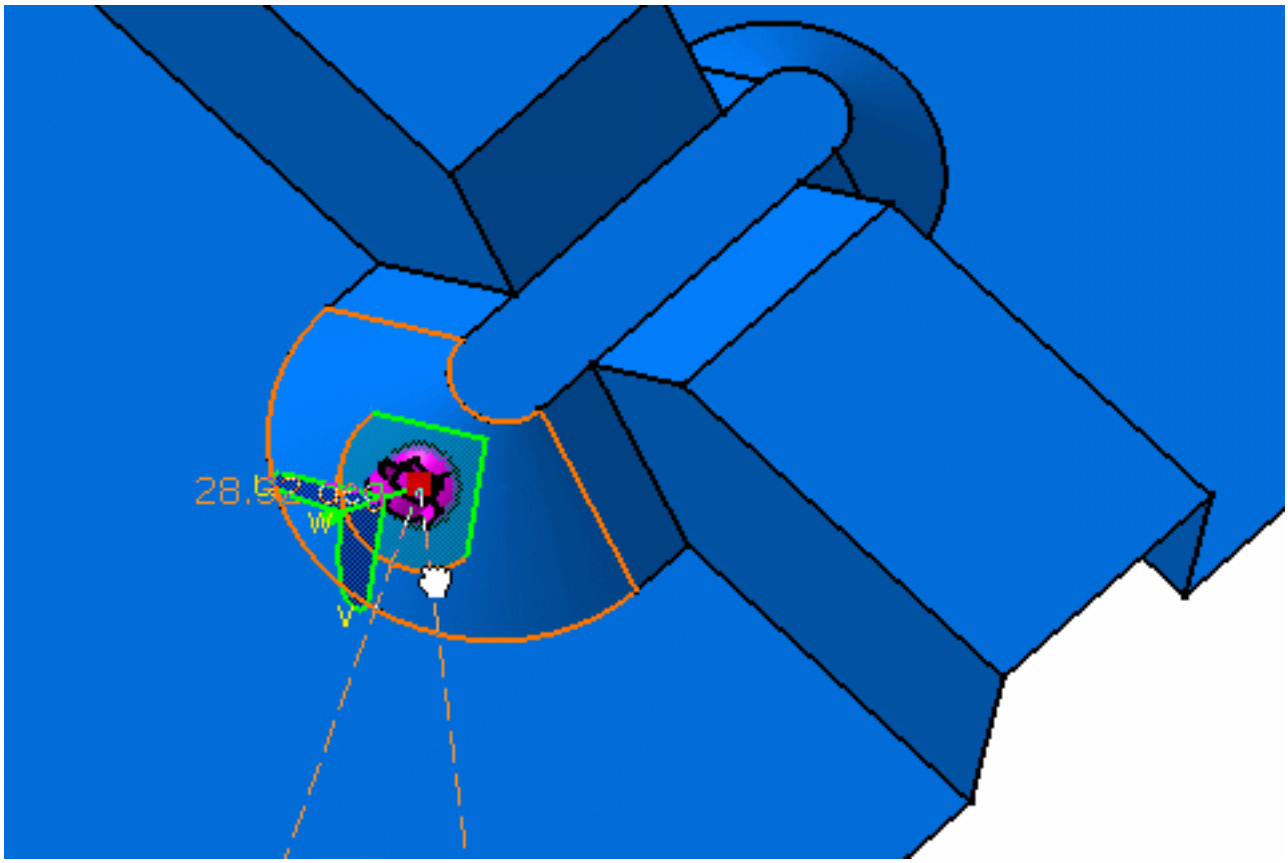


3. Point to the chosen position and click to place the support.

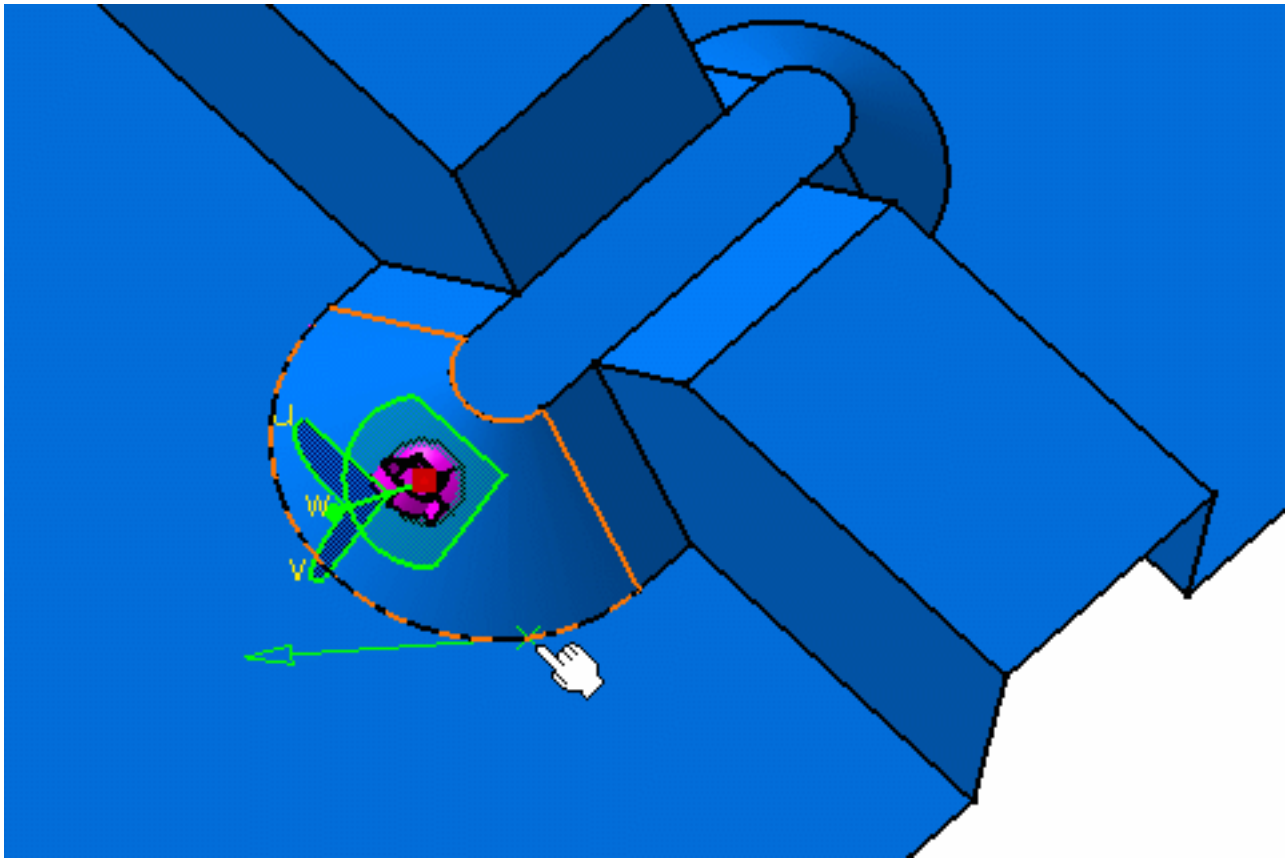
The support is placed on the plane selected, with a compass automatically snapped to the part.



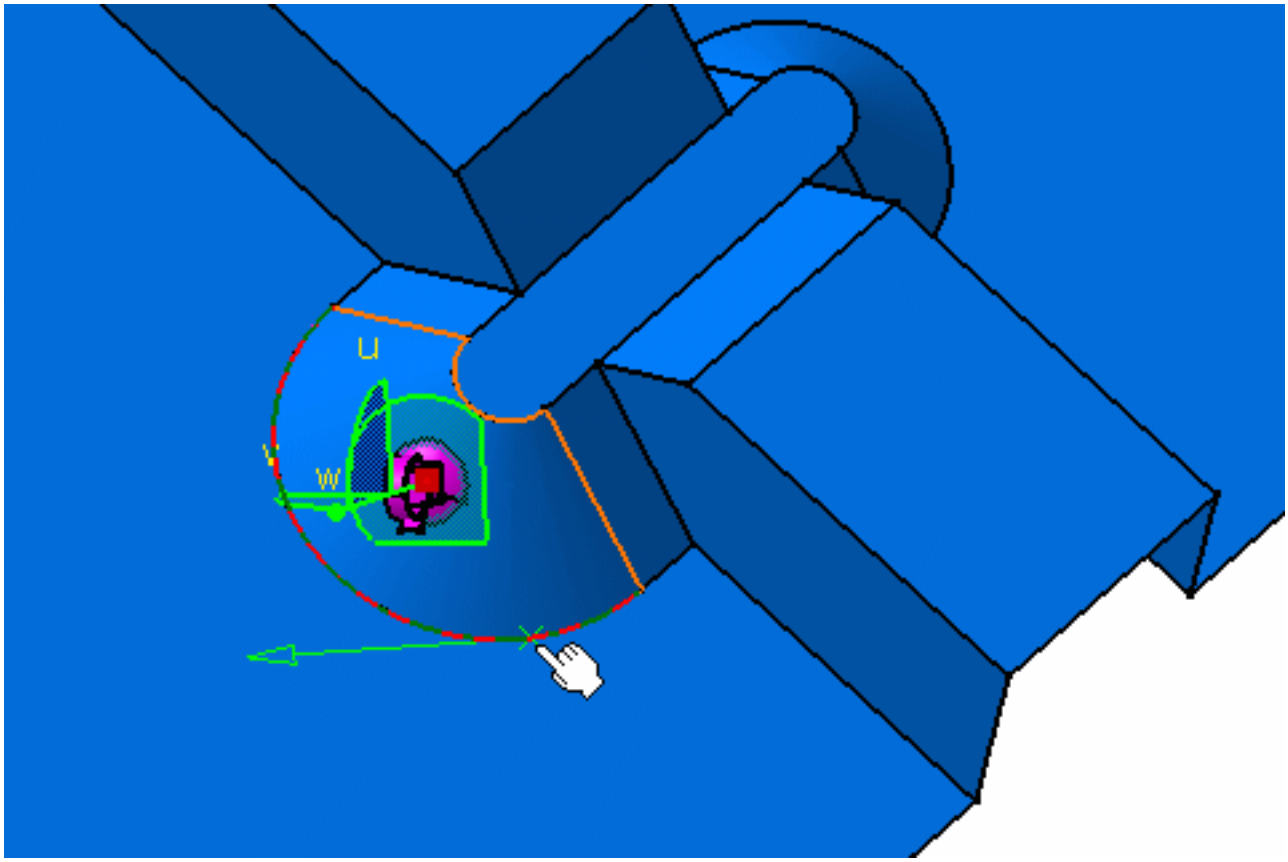
4. Use this compass to orientate the support.



You can also select an edge in the geometry:
A green arrow appears showing the orientation. By pressing the **Shift** key, you can invert the arrow direction.



Click to select the chosen orientation.
The support is orientated according to the edge you have selected: the rotation applied to the object aligns the compass V axis to the manipulator's green arrow.



5. Point to the next position if you want to place the support again.

   Otherwise, click a different part from the catalog and repeat these steps.

6. Click **Close** when you are done.

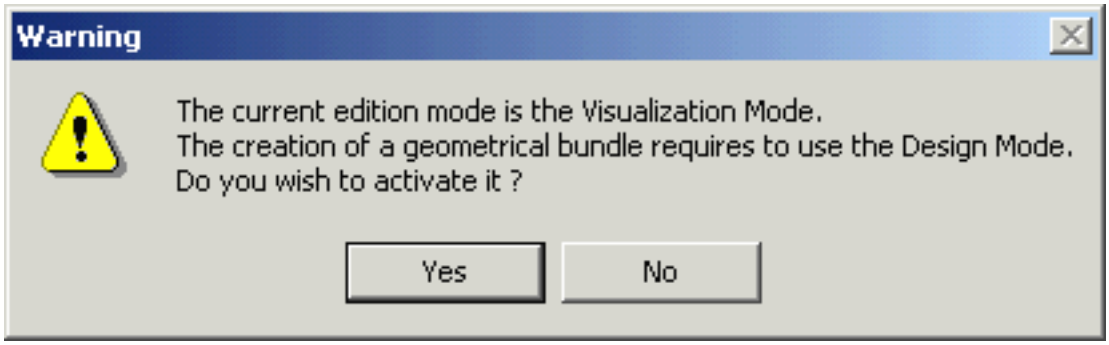
- The part is stored in the catalog with:
  - an origin point
  - a x, y and z direction.

- The origin of the part (the support in this example) is placed at the point selected with the compass.
- The x, y plane of the part is parallel to the plane selected using the compass.
- The z direction of the part is collinear to the z direction selected using the compass.

Creating a Geometrical Bundle

This task shows how to create a geometrical bundle.
A geometrical bundle is the representation of an assembly of wires grouped together with a common covering and connected to electrical connectors.

Make sure the design mode is activated otherwise a warning is displayed:

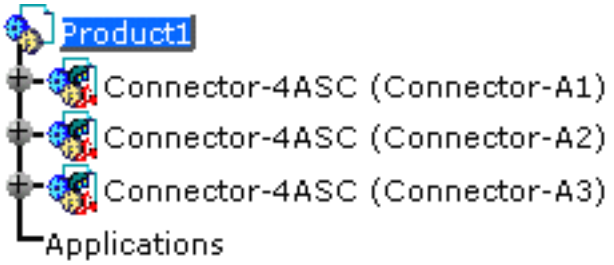


Click **Yes**.

It corresponds to the following settings in the **Tools -> Options...** menu item:

- Select the **Infrastructure -> Product Structure -> Cache Management** tab.
- In the Cache Activation, the **Work with cache system** option is not checked.
- As a consequence, in the **Product Visualization** tab, the **Visualization Mode Type** is set to **None**.

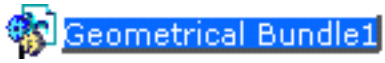
Open a document containing devices to be connected within a geometrical bundle.



1. Click the **Geometrical Bundle** button  .
   You are prompted to select the product you want to become the geometrical bundle.
2. Select the product of interest: **Product1**

   The geometrical bundle is created, with electrical capabilities.

   In the specification tree, the name has been modified as well as the icon.

Only the following can be selected to become a geometrical bundle:

- a product which is not already electrified
- a product which doesn't result from the **New Part** command
- a product which doesn't result from the **New Component** command (inline product).

Creating a Bundle Segment Document

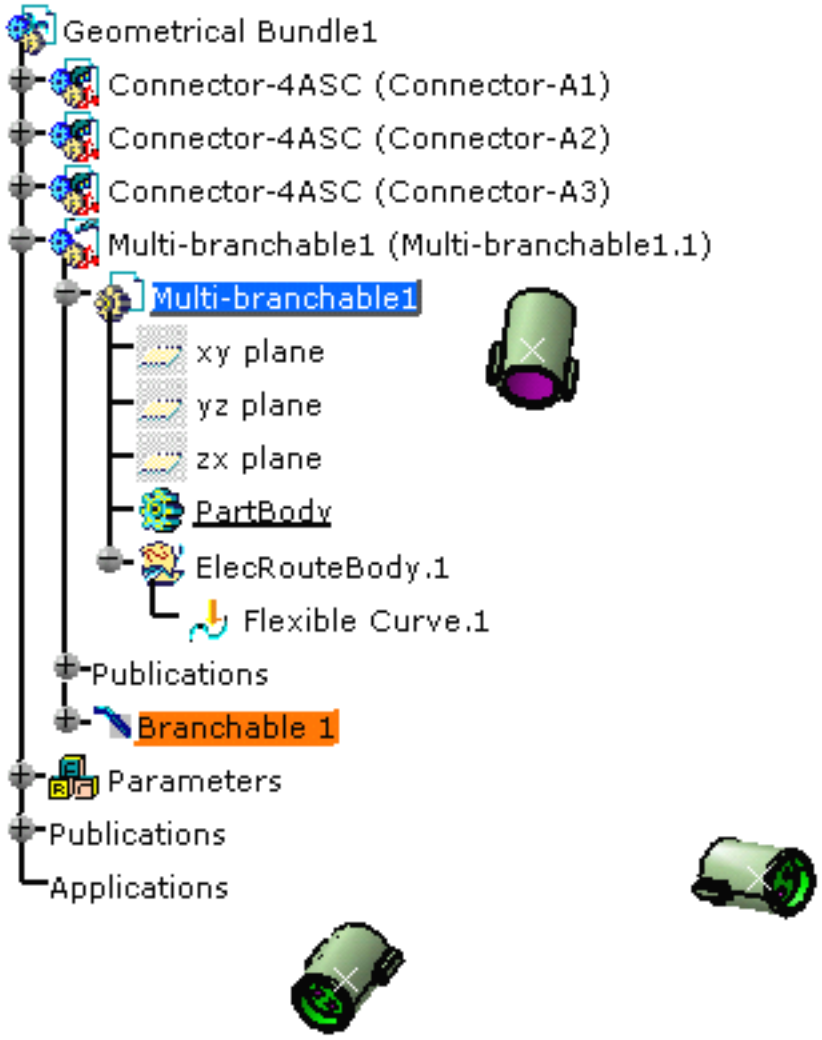
This task explains how to create the document in which the bundle segments take place. The bundle segment belongs to a part document with electrical properties.

To create and route a bundle segment, you have to:

- define the bundle segment parameters
- possibly create points, lines or planes if needed
- then define the bundle segment route.

The document now contains a geometrical bundle.

1. Double-click to activate the desired product: **Geometrical Bundle1**
2. Click the **Multi-Branchable Bundle Segment** button .



The bundle segment document is created with the **Multi-branchable1** product including:

- the **Multi-branchable1** part that becomes active
- the **Flexible Curve.1** belonging to the part, which at that time, does not have any geometrical representation.

The Electrical 3D Design Assembly workbench switches to the Electrical 3D Design Part workbench.

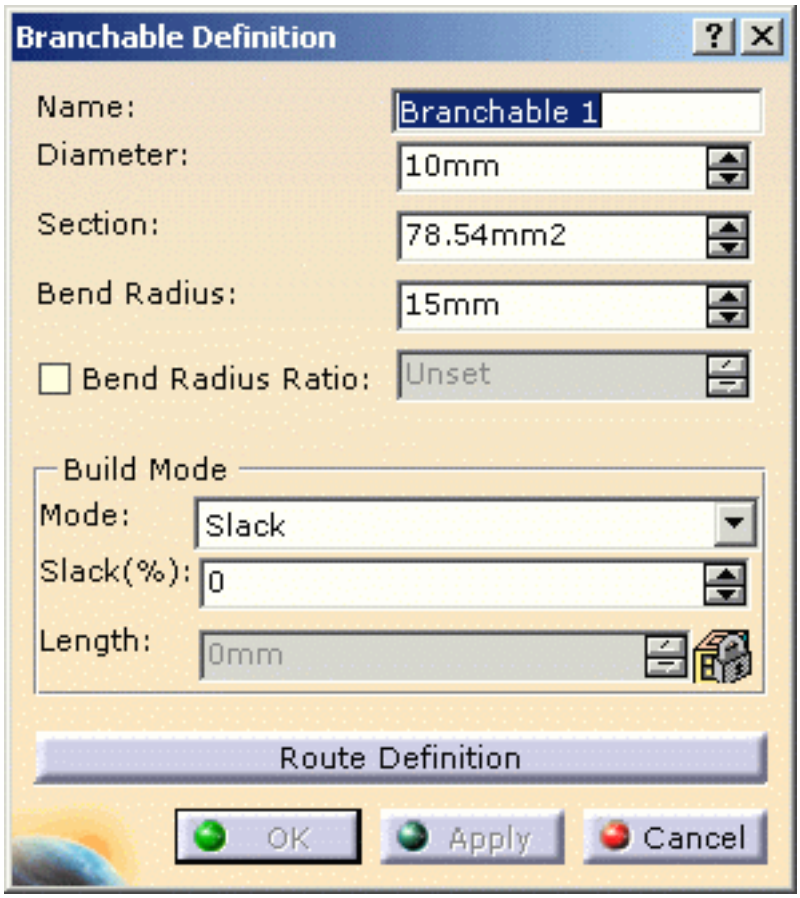
Defining the Segment Parameters

This task shows you how to define the bundle segment parameters.

The bundle segment to be defined is activated in the specification tree.
You have switched to the Electrical 3D Design Part workbench.

1. Click the **Branchable Bundle Segment Definition** button .

   The dialog box opens:



2. Enter a value in the **Diameter** field. The **Section** is automatically computed.

   As an alternative, you can enter the **Section**, the **Diameter** will be computed.

   The bend radius must be at least equal to the **Diameter** value to insure the correct bundle segment route computation. A message warns you if it is not the case.

3. Enter a value for the **Bend Radius**.

   The Bend Radius is the minimum bend radius allowed for the bundle segment.

   As an alternative, you can select the **Bend Radius Ratio** option and set the ratio: the **Bend Radius** is automatically computed.

4. Select the **Mode**:

The different options are:

- **Slack**:
  the bundle segment length is increased by the percentage indicated in the **Slack(%)** field. The **Length** field is disabled.
- **Length**:
  the bundle segment length is indicated in the **Length** field. The **Slack(%)** field is disabled.
- **Bend**:
  the bundle segment length corresponds to the minimum distance between the points defining its route. The **Slack(%)** and **Length** fields are disabled.

At this stage, the bundle segment parameters are defined.

You now need to route the bundle segment to be able to complete the definition: through this operation, you will create the geometrical representation of the **Flexible Curve**.

Note that **OK** and **Apply** are deactivated.

5. See the next task which explains how to route the bundle segment according to the geometrical constraints.

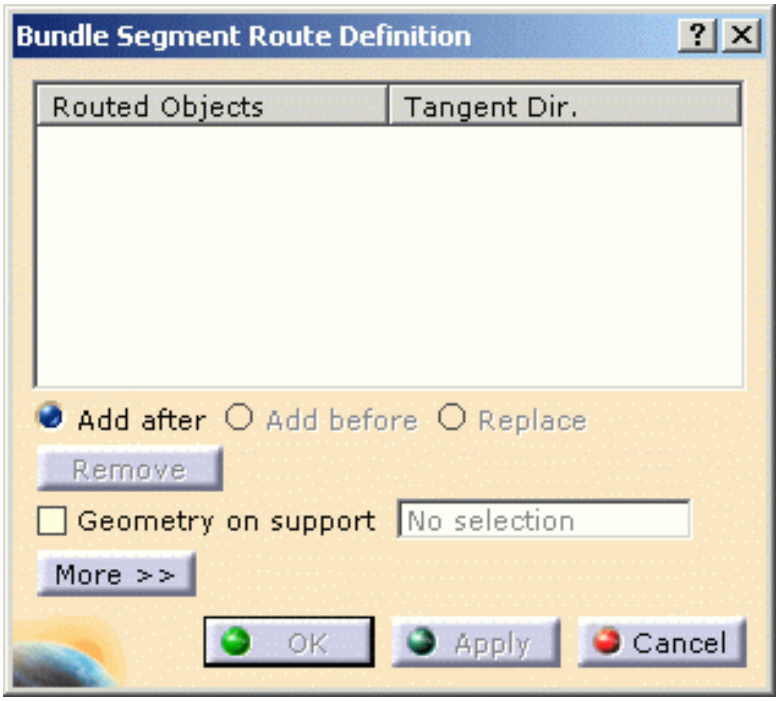
Defining the Segment Route Constraints

This section explains how to define the bundle segment route creating the **Flexible Curve** representation.
See also Getting Information from the Specification Tree Icons

The Bundle Segment parameters have been defined in the previous task.

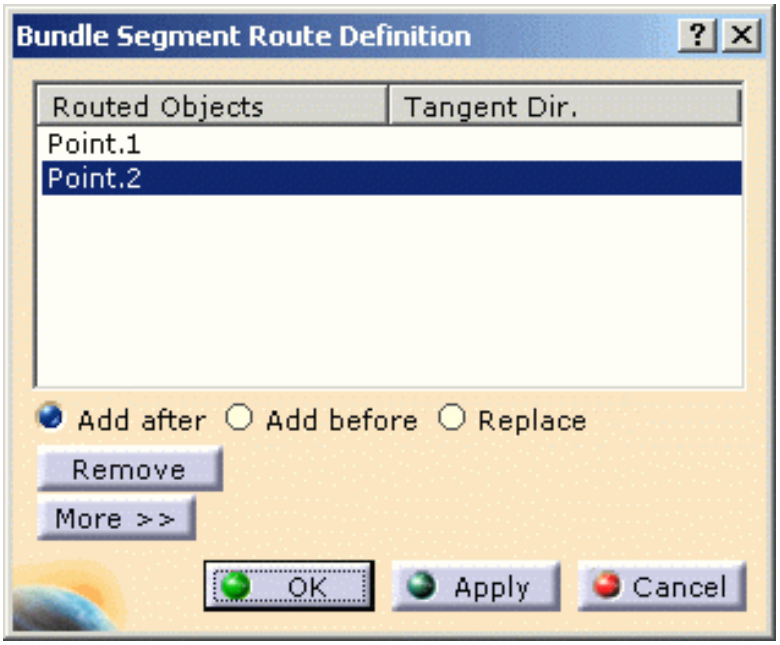
1. Click the **Route Definition** button.

   The Bundle Segment Route Definition dialog box opens:



2. Click successively the connectors and/or supports:

   CATIA finds the closest bundle connection point or section on supports, according to the selection point.



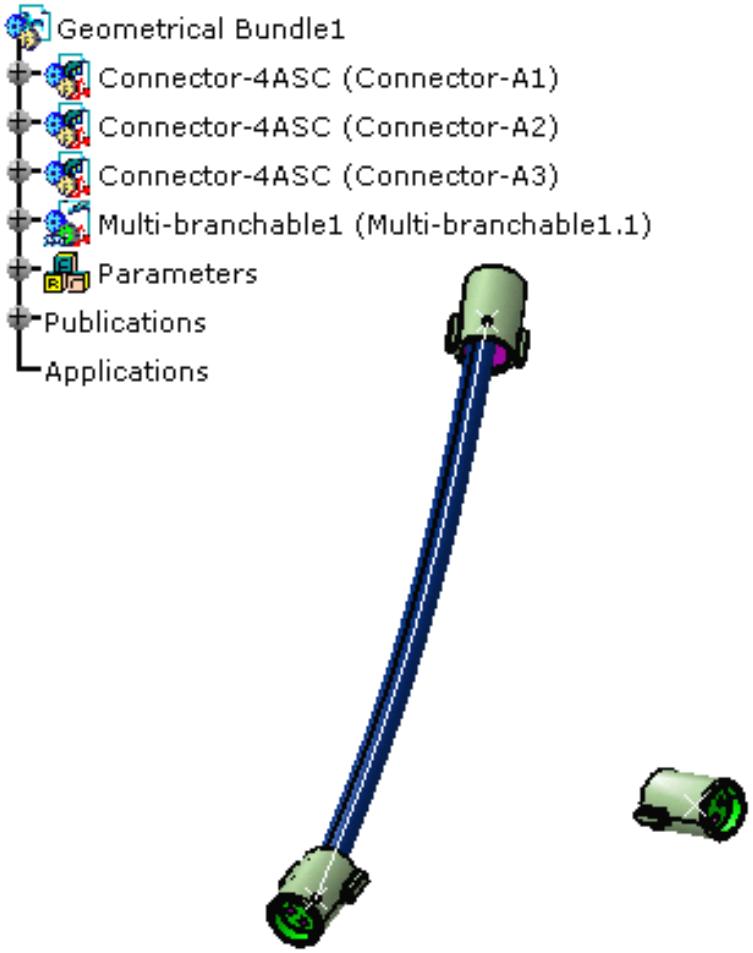
The **Flexible Curve** spline is displayed:

3. Click **OK** to validate.

   The Bundle Segment Route Definition dialog box closes and the Bundle Segment Definition is displayed afresh.
   Note that **OK** and **Apply** are now activated.

4. Click **OK** to validate the bundle segment definition.

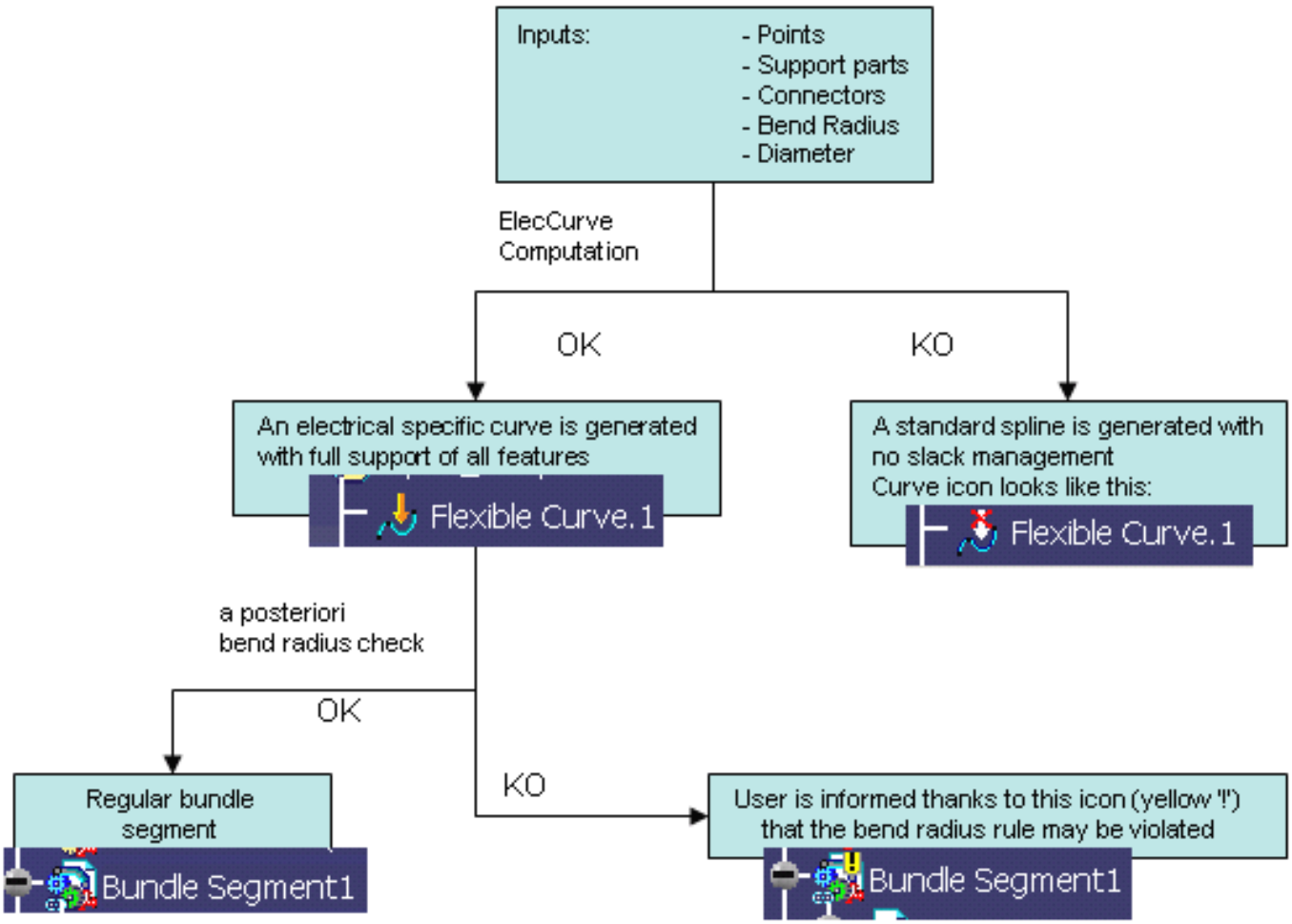
   The bundle segment is created



You can take advantage of the Related Objects viewer to focus on an object and see how it was constructed via its related objects. The related objects command identifies the parent, any children or connected objects and the relationship between objects.

Getting Information from the Specification Tree Icons

Each time you click the **Apply** or **OK** button during the definition phase, or if you update the bundle segment after any parameter modification, the following algorithm is launched to compute the best possible shape. Depending on the result, the specification tree is updated according to the following chart:



Note that working in **Cache** mode, the bundle segment does not display the icon.

Creating Points

This task shows the various methods for creating points:

- by coordinates
- on a curve
- on a plane
- on a surface
- at a circle/sphere center
- tangent point on a curve
- between

Open the Points3D1.CATPart document.

1. Click the **Point** icon .

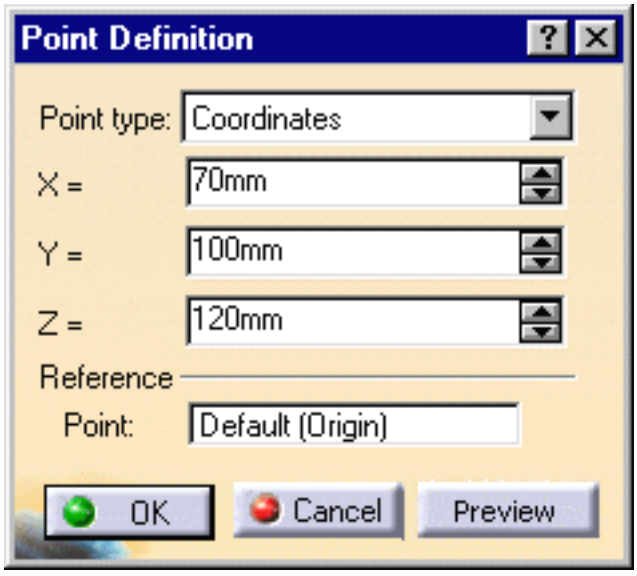
The Point Definition dialog box appears.

2. Use the combo to choose the desired point type.

Coordinates

- Enter the X, Y, Z coordinates in the current axis-system.
- Optionally, select a reference point.

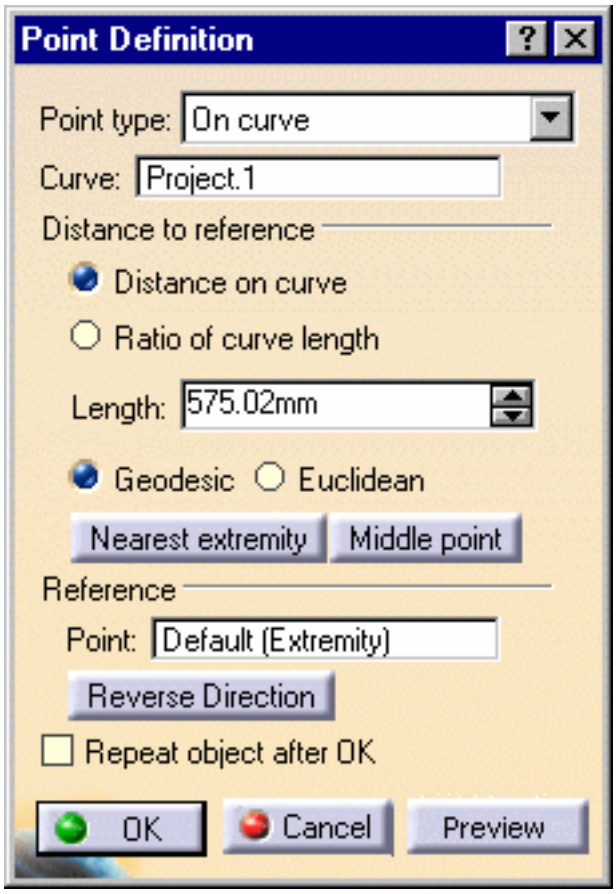
The corresponding point is displayed.



When creating a point within a user-defined axis-system, note that the **Coordinates in absolute axis-system** check button is added to the dialog box, allowing you to be define, or simply find out, the point's coordinates within the document's default axis-system.
If you create a point using the coordinates method and an axis system is already defined and set as current, the point's coordinates are defined according to current the axis system. As a consequence, the point's coordinates are not displayed in the specification tree.

The axis system must be different from the absolute axis.

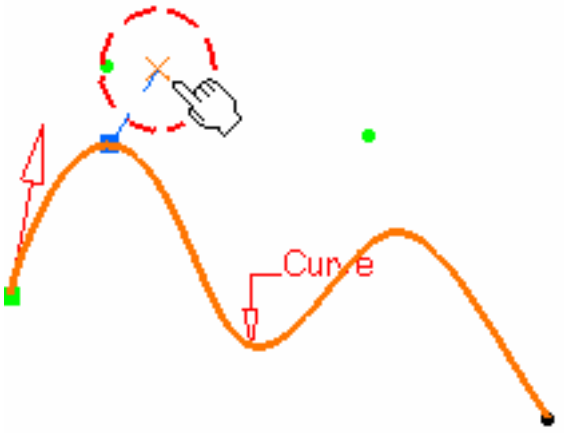
On curve



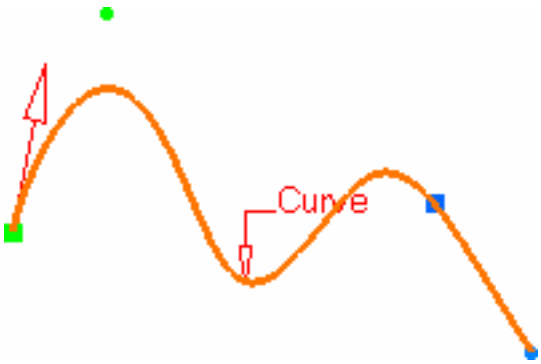
- Select a curve
- Optionally, select a reference point.

  If this point is not on the curve, it is projected onto the curve.
  If no point is selected, the curve's extremity is used as reference.

- Select an option point to determine whether the new point is to be created:
  - at a given distance along the curve from the reference point
  - a given ratio between the reference point and the curve's extremity.



- Enter the distance or ratio value.
  If a distance is specified, it can be:
  - a geodesic distance: the distance is measured along the curve
  - an Euclidean distance: the distance is measured in relation to the reference point (absolute value).



The corresponding point is displayed.

If the reference point is located at the curve's extremity, even if a ratio value is defined, the created point is always located at the end point of the curve.

You can also:

- click the **Nearest extremity** button to display the point at the nearest extremity of the curve.
- click the **Middle Point** button to display the mid-point of the curve.

Be careful that the arrow is orientated towards the inside of the curve (providing the curve is not closed) when using the **Middle Point** option.

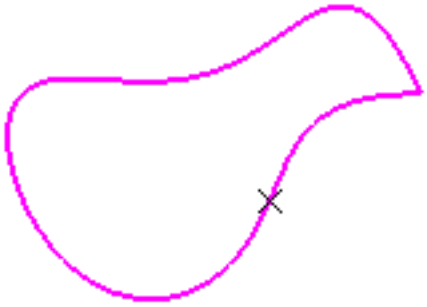
- use the **Reverse Direction** button to display:
  - the point on the other side of the reference point (if a point was selected originally)
  - the point from the other extremity (if no point was selected originally).

- click the **Repeat object after OK** if you wish to create equidistant points on the curve, using the currently created point as the reference, as described in Creating Multiple Points in the Wireframe and Surface User's Guide.

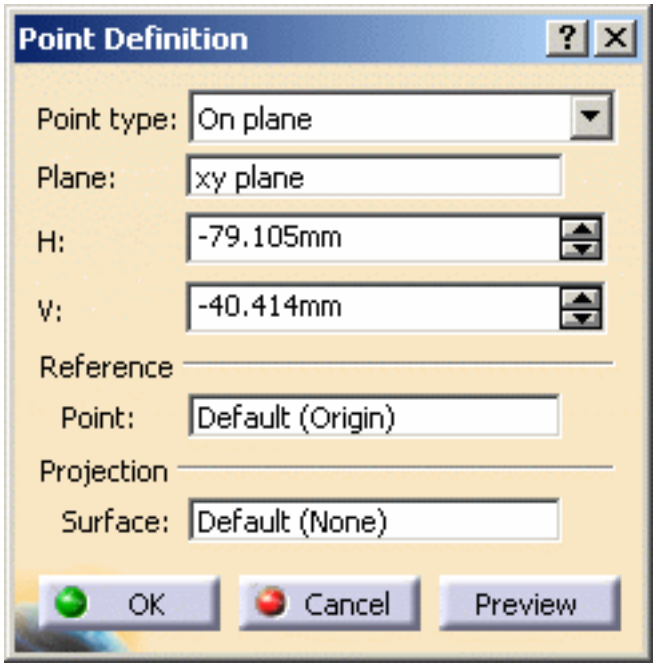
You will also be able to create planes normal to the curve at these points, by checking the **Create normal planes also** button, and to create all instances in a new geometrical set by checking the **Create in a new geometrical set** button.
If the button is not checked the instances are created in the current geometrical set .

- If the curve is infinite and no reference point is explicitly given, by default, the reference point is the projection of the model's origin
- If the curve is a closed curve, either the system detects a vertex on the curve that can be used as a reference point, or it creates an extremum point, and highlights it (you can then select another one if you wish) or the system prompts you to manually select a reference point.

Extremum points created on a closed curve are now aggregated under their parent command and put in no show in the specification tree.



On plane



- Select a plane.
- Optionally, select a point to define a reference for computing coordinates in the plane.

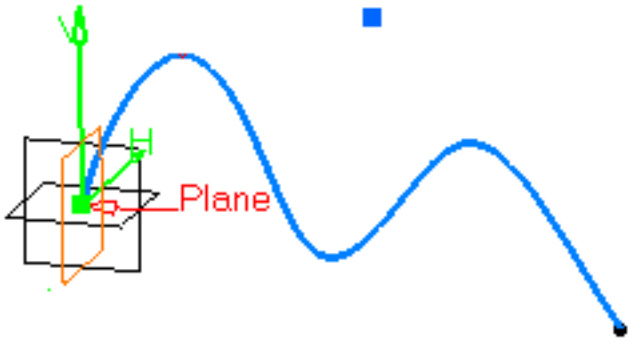
  If no point is selected, the projection of the model's origin on the plane is taken as reference.

- Optionally, select a surface on which the point is projected normally to the plane.

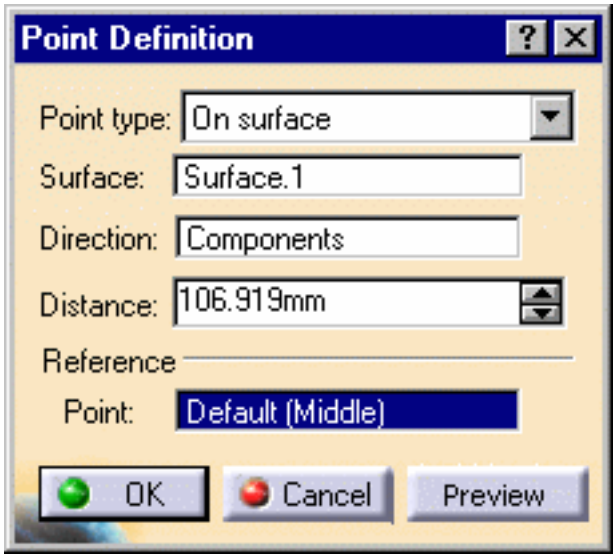
  If no surface is selected, the behavior is the same.

  Furthermore, the reference direction (H and V vectors) is computed as follows:
  With N the normal to the selected plane (reference plane), H results from the vectorial product of Z and N (H = Z^N).
  If the norm of H is strictly positive then V results from the vectorial product of N and H (V = N^H).
  Otherwise, V = N^X and H = V^N.

  Would the plane move, during an update for example, the reference direction would then be projected on the plane.



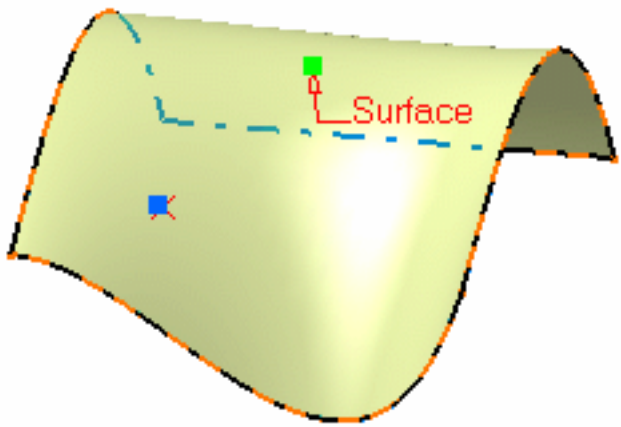
- Click in the plane to display a point.



On surface

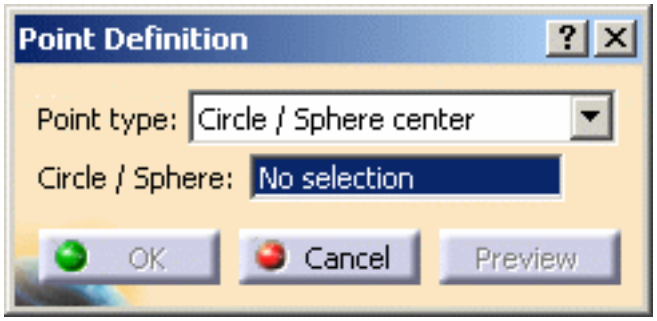
- Select the surface where the point is to be created.

- Optionally, select a reference point. By default, the surface's middle point is taken as reference.
- You can select an element to take its orientation as reference direction or a plane to take its normal as reference direction.
  You can also use the contextual menu to specify the X, Y, Z components of the reference direction.
- Enter a distance along the reference direction to display a point.

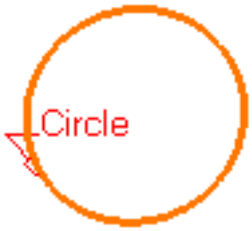


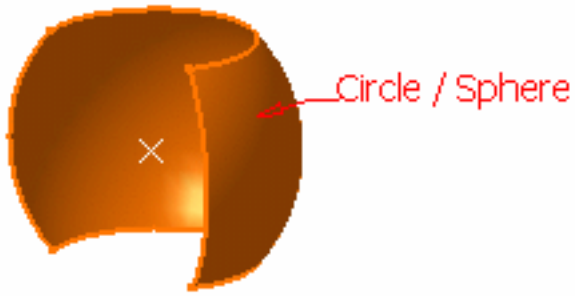
Circle/Sphere center

- Select a circle, circular arc, or ellipse, or
- Select a sphere or a portion of sphere.



A point is displayed at the center of the selected element.

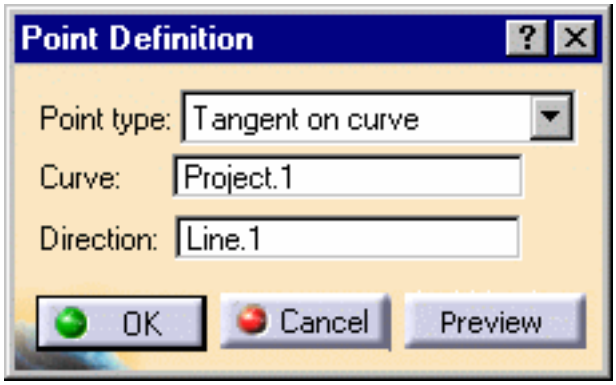




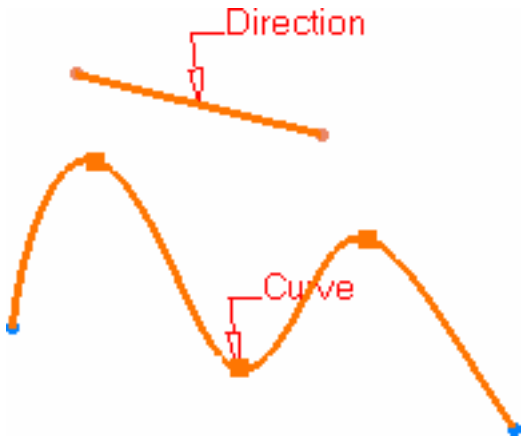
Tangent on curve

- Select a planar curve and a direction line.

A point is displayed at each tangent.



The Multi-Result Management dialog box is displayed because several points are generated.

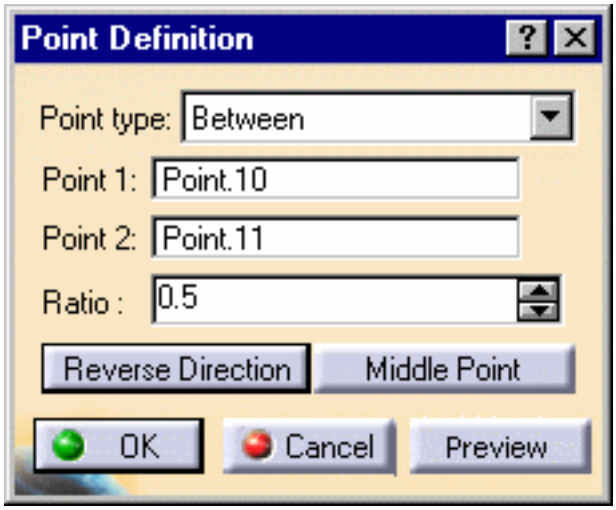


- Click **YES**: you can then select a reference element, to which only the closest point is created.
- Click **NO**: all the points are created.

For further information, refer to the Managing Multi-Result Operations chapter.

Between

- Select any two points.



- Enter the ratio, that is the percentage of the distance from the first selected point, at which the new point is to be.
  You can also click **Middle Point** button to create a point at the exact midpoint (ratio = 0.5).

Be careful that the arrow is orientated towards the inside of the curve (providing the curve is not closed) when using the **Middle Point** option.

- Use the **Reverse direction** button to measure the ratio from the second selected point.

If the ratio value is greater than 1, the point is located on the virtual line beyond the selected points.

3. Click OK to create the point.

The point (identified as Point.xxx) is added to the specification tree.

- Parameters can be edited in the 3D geometry. For more information, refer to the Editing Parameters chapter.
- You can isolate a point in order to cut the links it has with the geometry used to create it. To do so, use the **Isolate** contextual menu. For more information, refer to the Isolating Features chapter.

Creating Lines

This task shows the various methods for creating lines:

- point to point
- point and direction
- angle or normal to curve
- tangent to curve
- normal to surface
- bisecting

It also shows you how to create a line up to an element, define the length type and automatically reselect the second point.

Open the Lines1.CATPart document.

1. Click the **Line** icon  .

   The Line Definition dialog box is displayed.

2. Use the drop-down list to choose the desired line type.

A line type will be proposed automatically in some cases depending on your first element selection.

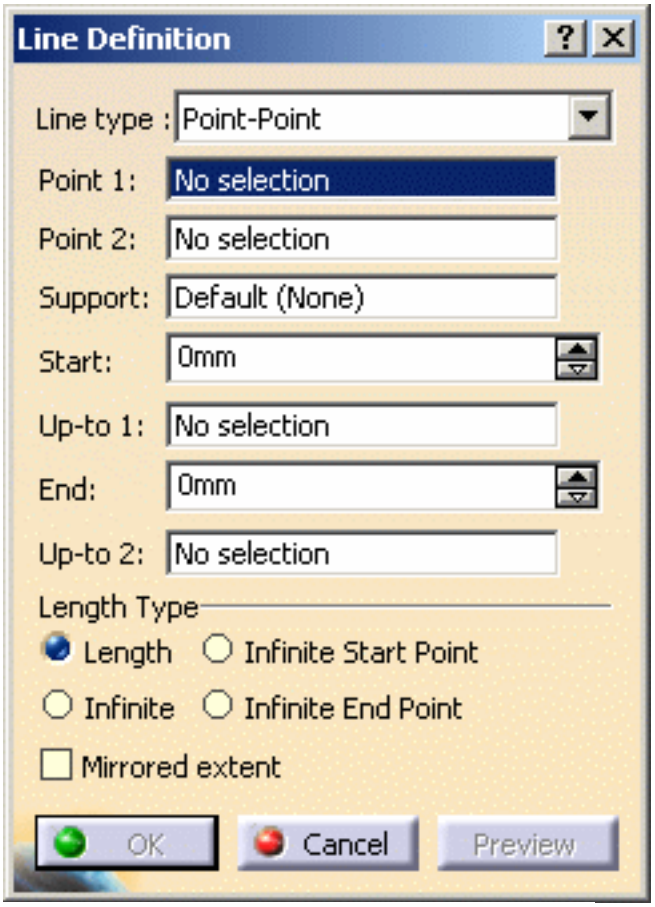
Defining the line type

Point - Point

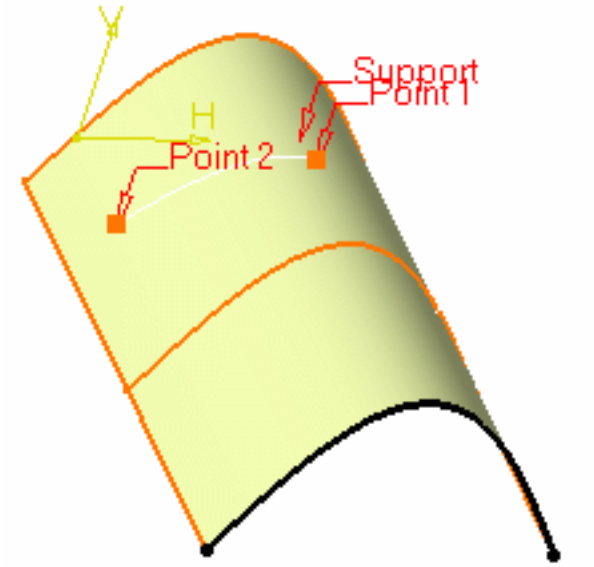
This command is only available with the Generative Shape Design 2 product.

- Select two points.

A line is displayed between the two points.
Proposed **Start** and **End** points of the new line are shown.

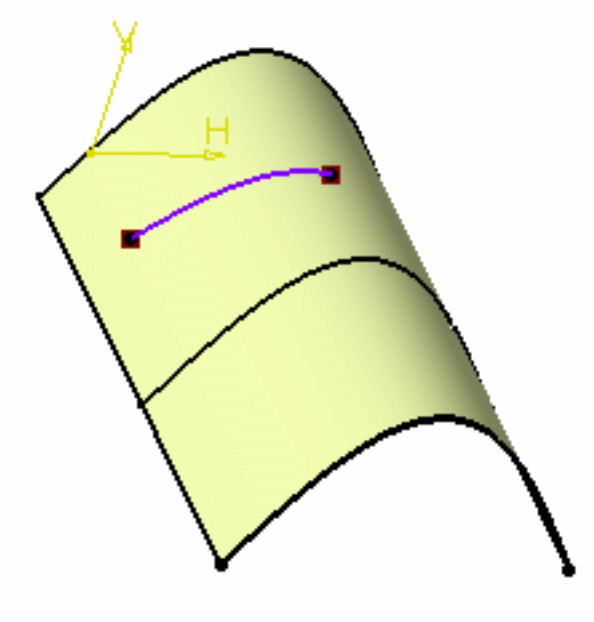


- If needed, select a support surface.
  In this case a geodesic line is created, i.e. going from one point to the other according to the shortest distance along the surface geometry (blue line in the illustration below).
  If no surface is selected, the line is created between the two points based on the shortest distance.



If you select two points on closed surface (a cylinder for example), the result may be unstable. Therefore, it is advised to split the surface and only keep the part on which the geodesic line will lie.

The geodesic line is not available with the Wireframe and Surface workbench.

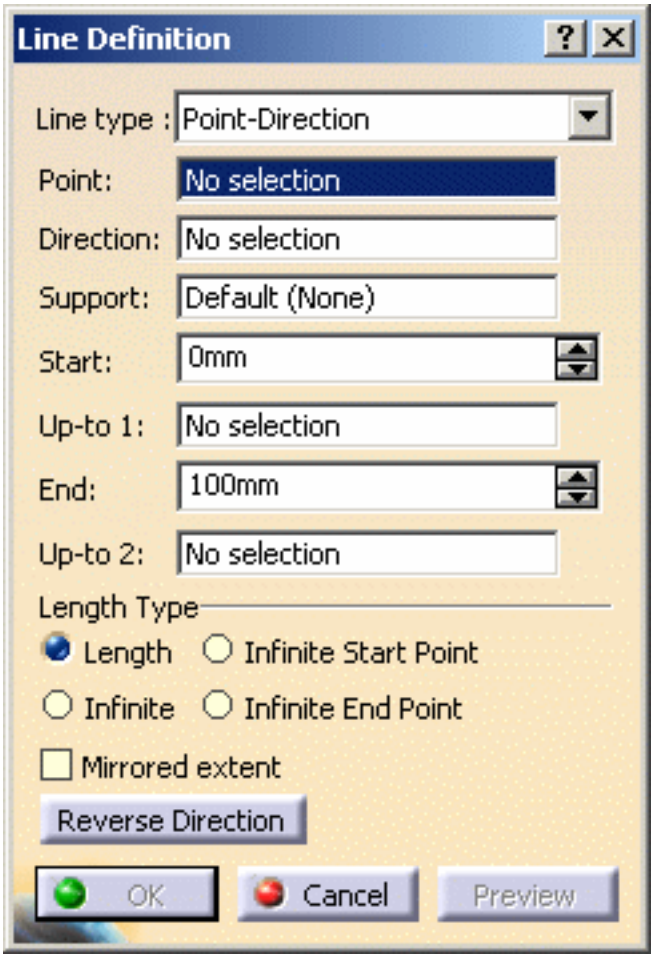


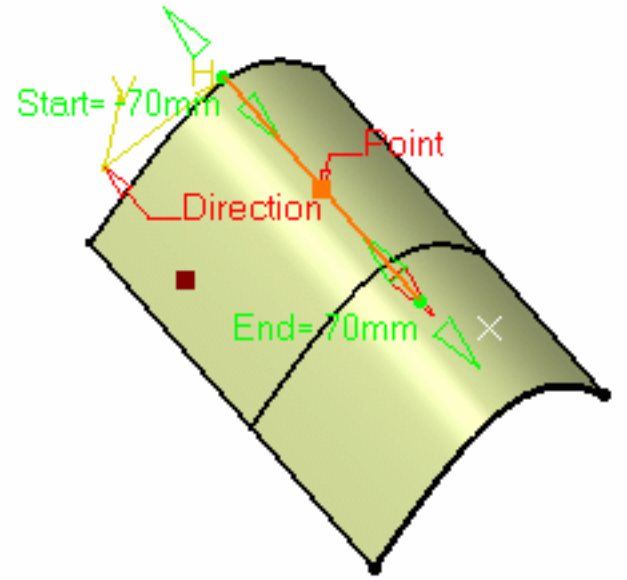
- Specify the **Start** and **End** points of the new line, that is the line endpoint location in relation to the points initially selected. These **Start** and **End** points are necessarily beyond the selected points, meaning the line cannot be shorter than the distance between the initial points.
- Check the **Mirrored extent** option to create a line symmetrically in relation to the selected **Start** and **End** points.

The projections of the 3D point(s) must already exist on the selected support.

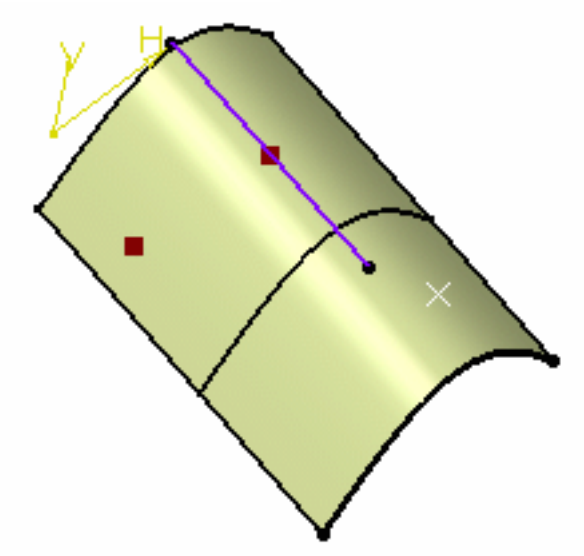
Point - Direction

- Select a reference **Point** and a **Direction** line.
  A vector parallel to the direction line is displayed at the reference point.
  Proposed **Start** and **End** points of the new line are shown.





- Specify the **Start** and **End** points of the new line.
  The corresponding line is displayed.



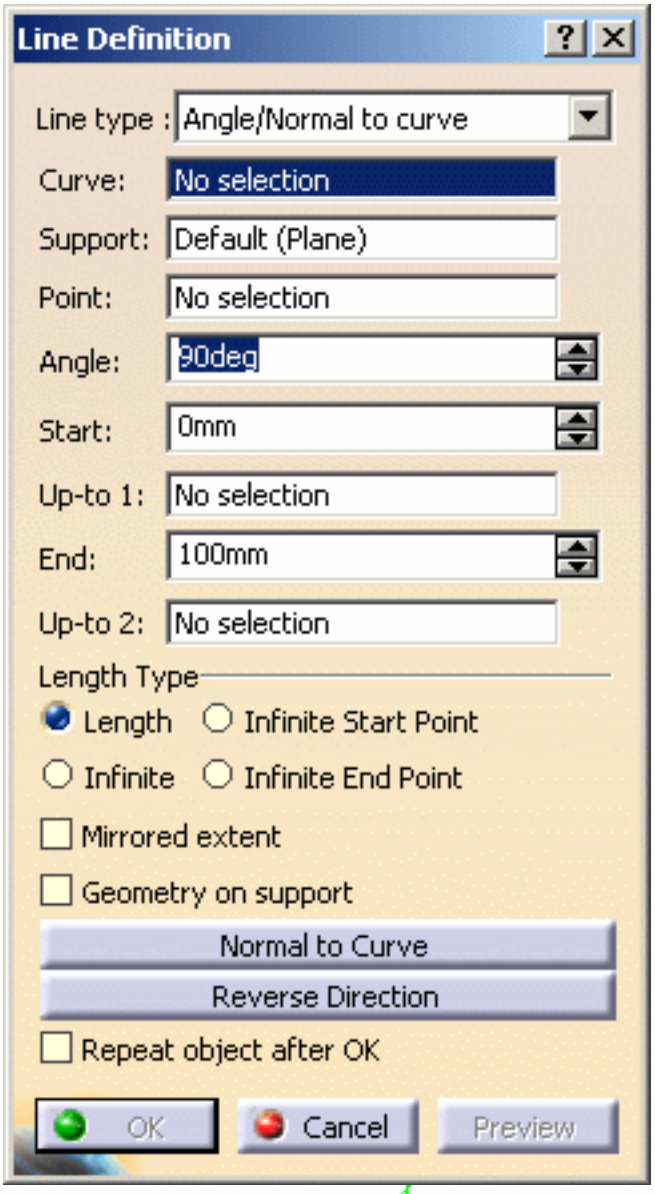
The projections of the 3D point(s) must already exist on the selected support.

Angle or Normal to curve

- Select a reference **Curve** and a **Support** surface containing that curve.

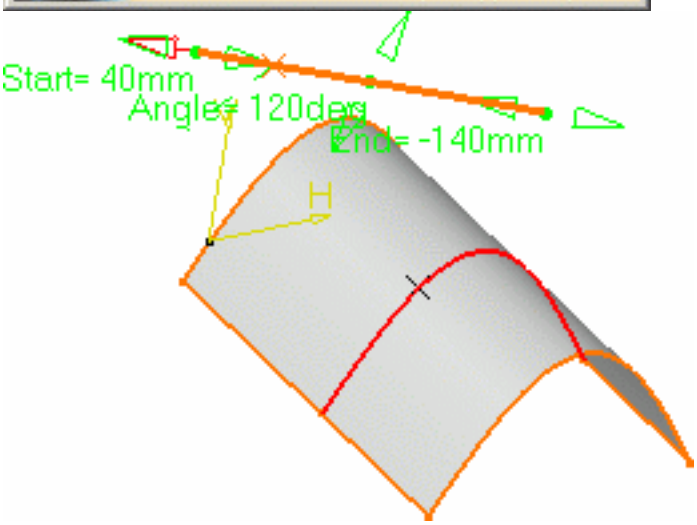
- If the selected curve is planar, then the **Support** is set to Default (Plane).

- If an explicit **Support** has been defined, a contextual menu is available to clear the selection.



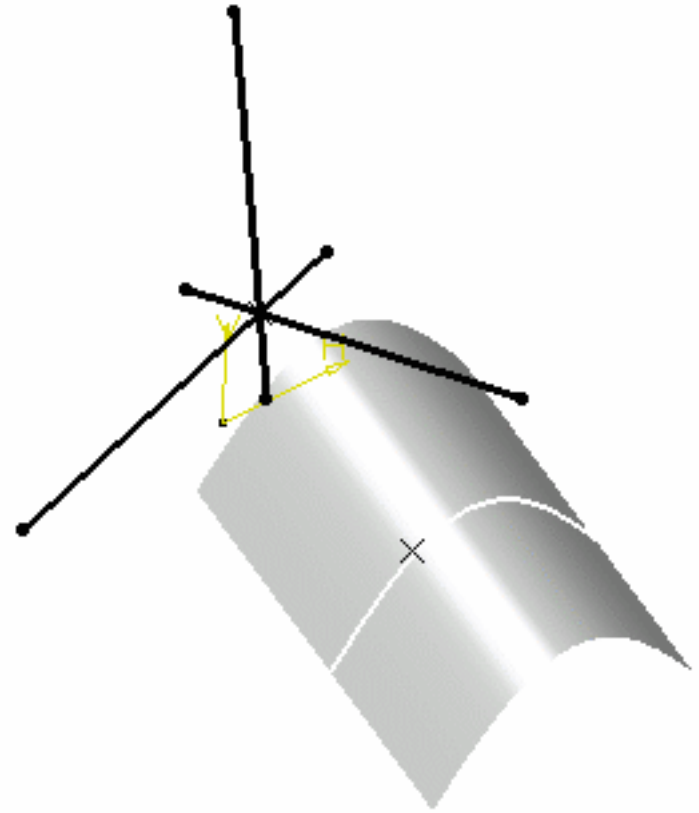
- Select a **Point** on the curve.
- Enter an **Angle** value.

A line is displayed at the given angle with respect to the tangent to the reference curve at the selected point. These elements are displayed in the plane tangent to the surface at the selected point.
You can click on the **Normal to Curve** button to specify an angle of 90 degrees.
Proposed **Start** and **End** points of the line are shown.



- Specify the **Start** and **End** points of the new line.
The corresponding line is displayed.

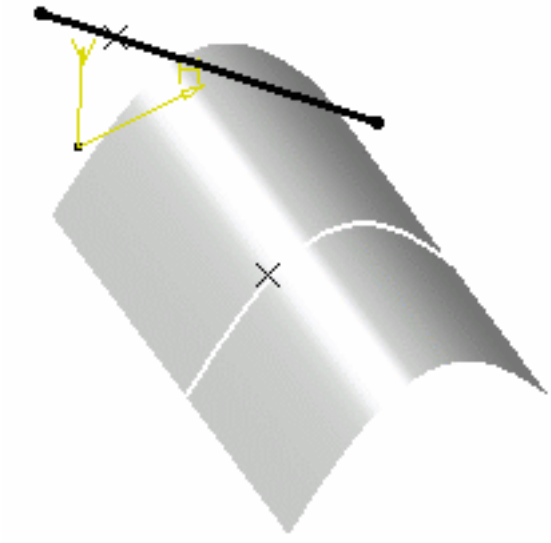
- Click the **Repeat object after OK** if you wish to create more lines with the same definition as the currently created line.
  In this case, the Object Repetition dialog box is displayed, and you key in the number of instances to be created before pressing OK.



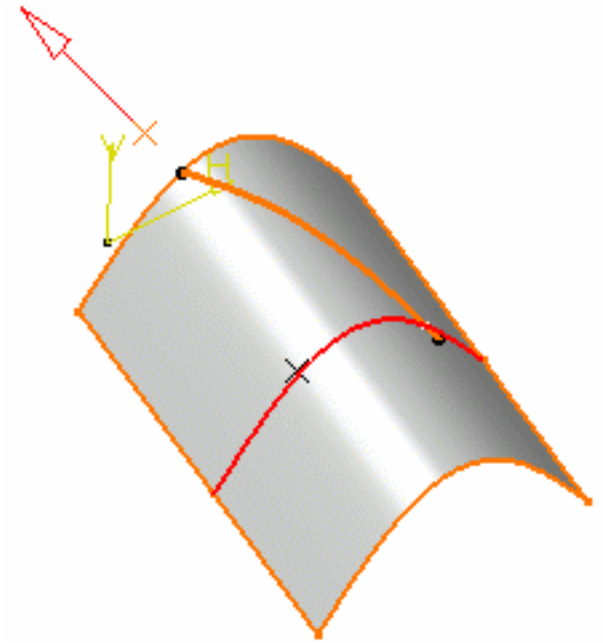
As many lines as indicated in the dialog box are created, each separated from the initial line by a multiple of the **angle** value.

You can select the **Geometry on Support** check box if you want to create a geodesic line onto a support surface.

The figure below illustrates this case.



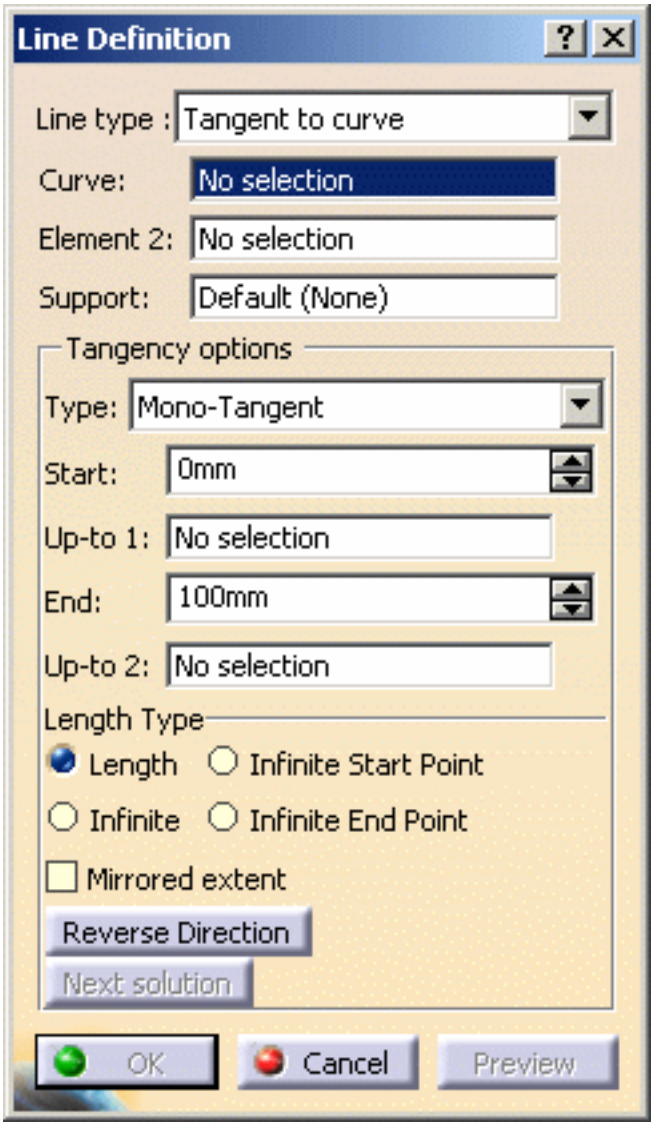
Geometry on support option not checked



Geometry on support option checked

This line type enables to edit the line's parameters. Refer to Editing Parameters to find out how to display these parameters in the 3D geometry.

Tangent to curve

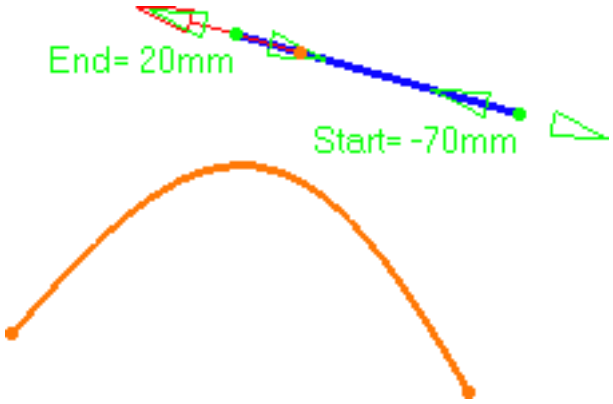


- Select a reference **Curve** and a **point** or another **Curve** to define the tangency.
  - if a point is selected (mono-tangent mode): a vector tangent to the curve is displayed at the selected point.
  - If a second curve is selected (or a point in bi-tangent mode), you need to select a support plane. The line will be tangent to both curves.

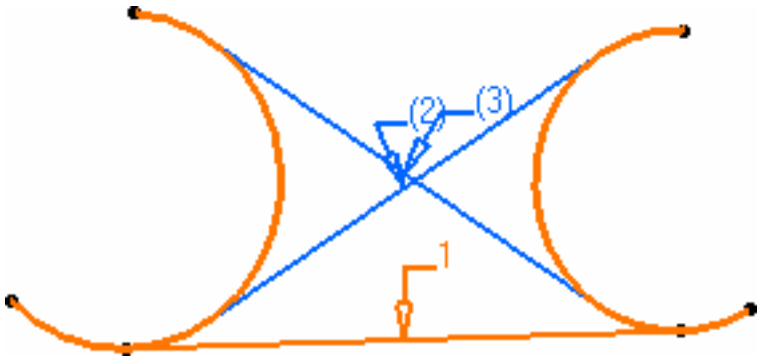
- If the selected curve is a line, then the **Support** is set to Default (Plane).

- If an explicit **Support** has been defined, a contextual menu is available to clear the selection.

When several solutions are possible, you can choose one (displayed in red) directly in the geometry, or using the **Next Solution** button.

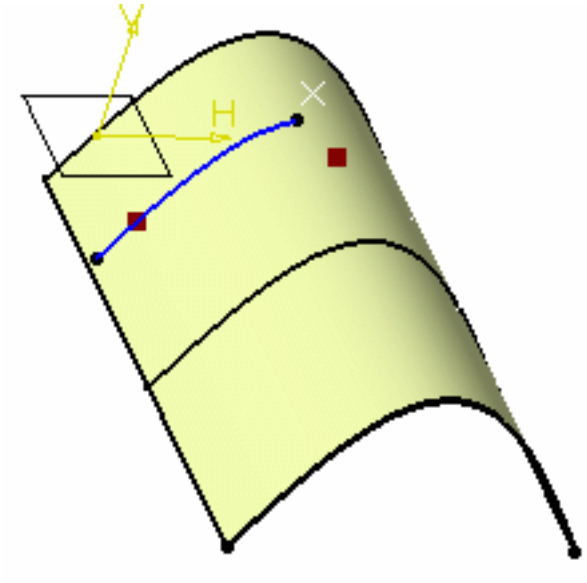


Line tangent to curve at a given point

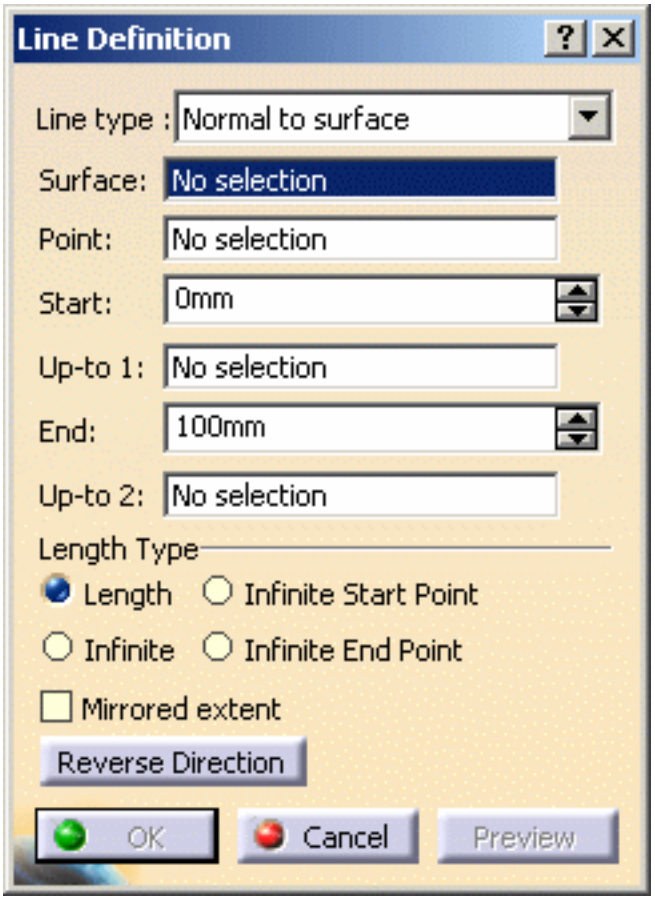


Line tangent to two curves

- Specify **Start** and **End** points to define the new line.
  The corresponding line is displayed.

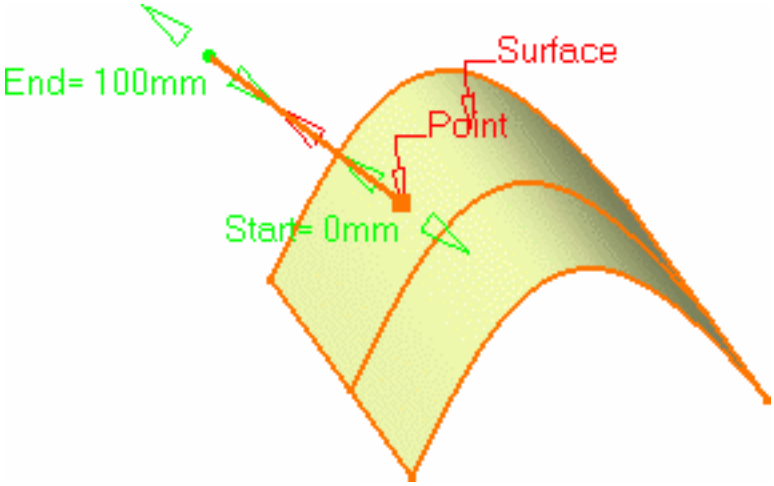


Normal to surface

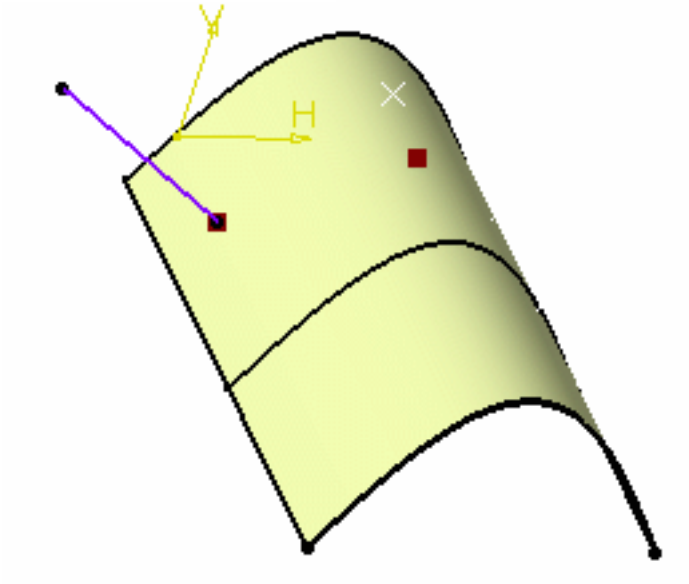


- Select a reference **Surface** and a **Point**.
  A vector normal to the surface is displayed at the reference point.
  Proposed **Start** and **End** points of the new line are shown.

If the point does not lie on the support surface, the minimum distance between the point and the surface is computed, and the vector normal to the surface is displayed at the resulted reference point.

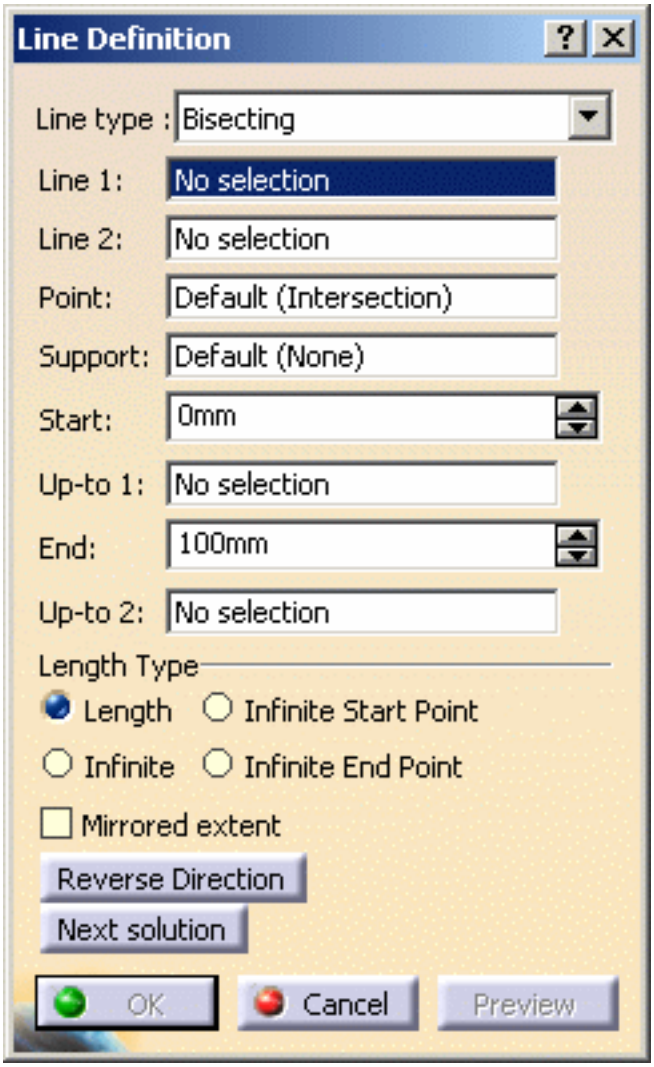


- Specify **Start** and **End** points to define the new line.
  The corresponding line is displayed.

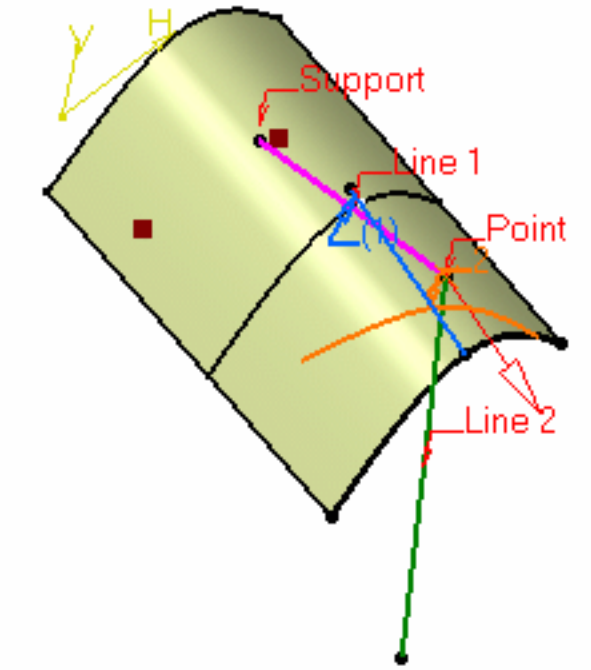


Bisecting

- Select two lines. Their bisecting line is the line splitting in two equals parts the angle between these two lines.
- Select a point as the starting point for the line. By default it is the intersection of the bisecting line and the first selected line.



- Select the support surface onto which the bisecting line is to be projected, if needed.
- Specify the line's length in relation to its starting point (**Start** and **End** values for each side of the line in relation to the default end points).
  The corresponding bisecting line, is displayed.
- You can choose between two solutions, using the **Next Solution** button, or directly clicking the numbered arrows in the geometry.



3. Click **OK** to create the line.

The line (identified as Line.xxx) is added to the specification tree.

- Regardless of the line type, **Start** and **End** values are specified by entering distance values or by using the graphic manipulators.
- **Start** and **End** values should not be the same.
- Check the **Mirrored extent** option to create a line symmetrically in relation to the selected **Start** point.
  It is only available with the **Length** Length type.
- In most cases, you can select a support on which the line is to be created. In this case, the selected point(s) is projected onto this support.
- You can reverse the direction of the line by either clicking the displayed vector or selecting the **Reverse Direction** button (not available with the point-point line type).

Creating a line up to an element

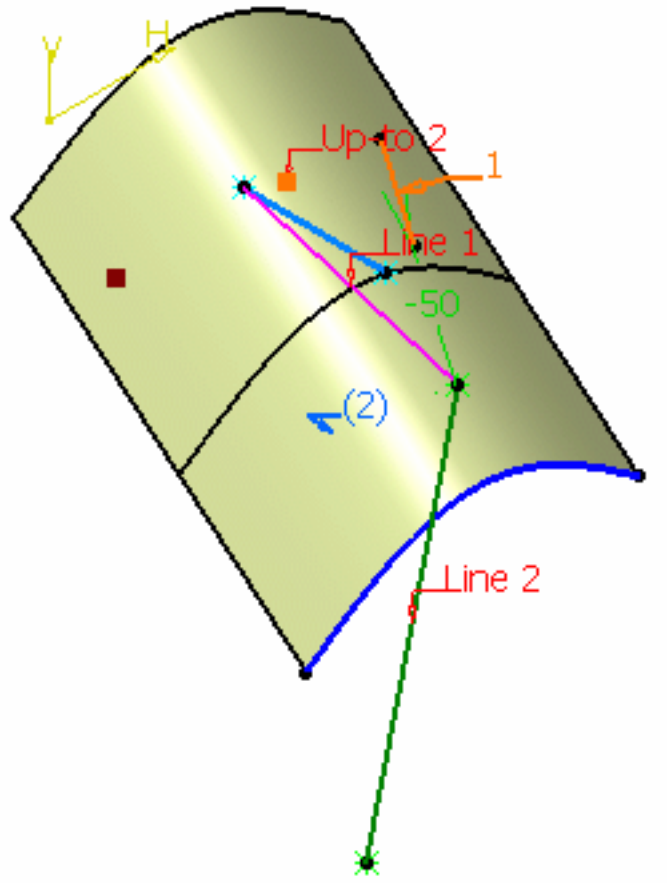
This capability allows you to create a line up to a point, a curve, or a surface.

- It is available with all line types, but the Tangent to curve type.

Up to a point

- Select a point in the **Up-to 1** and/or **Up-to 2** fields.

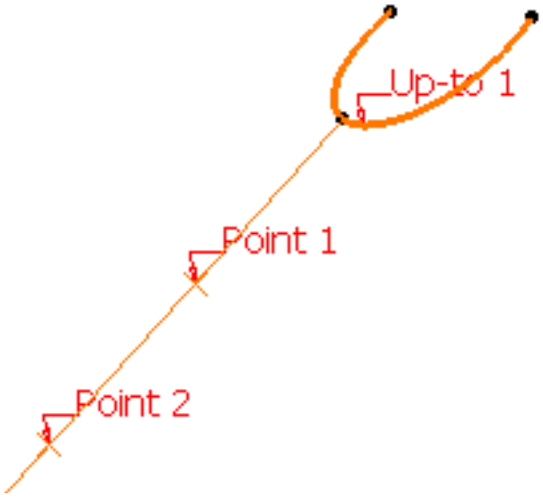
Here is an example with the Bisecting line type, the **Length** Length type, and a point as **Up-to 2** element.



Up to a curve

- Select a curve in the **Up-to 1** and/or **Up-to 2** fields.

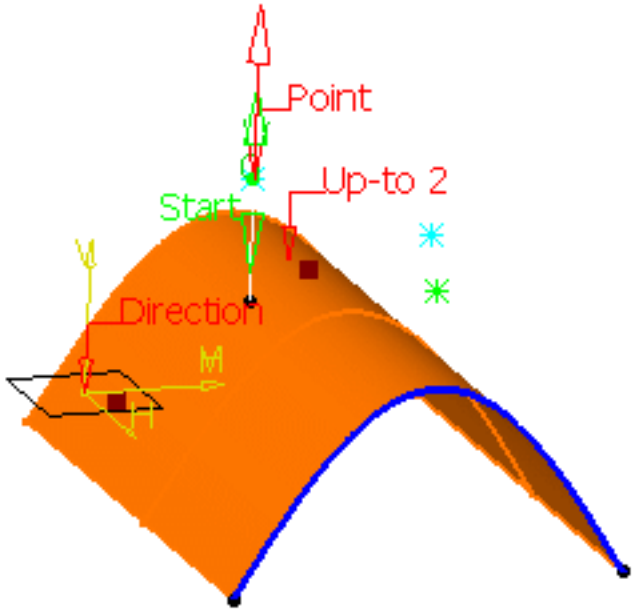
  Here is an example with the Point-Point line type, the **Infinite End** Length type, and a curve as the **Up-to 1** element.



Up to a surface

- Select a surface in the **Up-to 1** and/or **Up-to 2** fields.

  Here is an example with the Point-Direction line type, the **Length** Length type, and the surface as the **Up-to 2** element.



- If the selected Up-to element does not intersect with the line being created, then an extrapolation is performed. It is only possible if the element is linear and lies on the same plane as the line being created.
  However, no extrapolation is performed if the Up-to element is a curve or a surface.
- The **Up-to 1** and **Up-to 2** fields are grayed out with the **Infinite** Length type, the **Up-to 1** field is grayed out with the **Infinite Start** Length type, the Up-to 2 field is grayed out with the **Infinite End** Length type.
- The **Up-to 1** field is grayed out if the **Mirrored extent** option is checked.
- In the case of the Point-Point line type, **Start** and **End** values cannot be negative.

Defining the length type

- Select the Length Type:
  - **Length**: the line will be defined according to the **Start** and **End** points values
  - **Infinite**: the line will be infinite
  - **Infinite Start Point**: the line will be infinite from the **Start** point
  - **Infinite End Point**: the line will be infinite from the **End** point

By default, the Length type is selected.
The **Start** and/or the **End** points values will be greyed out when one of the **Infinite** options is chosen.

Reselecting automatically a second point

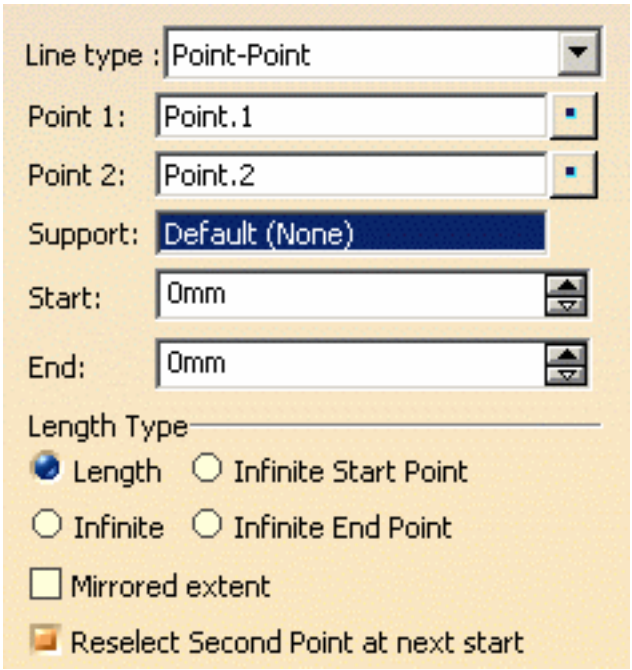
This capability is only available with the Point-Point line method.

1. Double-click the **Line** icon  .

   The Line dialog box is displayed.

2. Create the first point.

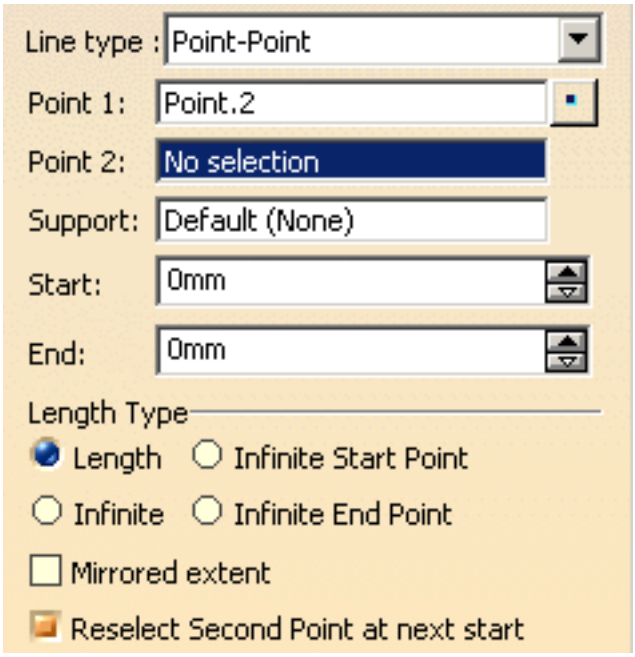
The **Reselect Second Point at next start** option appears in the Line dialog box.



3. Check it to be able to later reuse the second point.
4. Create the second point.
5. Click OK to create the first line.

The Line dialog box opens again with the first point initialized with the second point of the first line.

6. Click OK to create the second line.



To stop the repeat action, simply uncheck the option or click Cancel in the Line dialog box.

- Parameters can be edited in the 3D geometry. For more information, refer to the Editing Parameters chapter.
- You can isolate a line in order to cut the links it has with the geometry used to create it. To do so, use the **Isolate** contextual menu. For more information, refer to the Isolating Features chapter.

Creating Planes

This task shows the various methods for creating planes:

- offset from a plane
- parallel through point
- angle/normal to a plane
- through three points
- through two lines
- through a point and a line
- through a planar curve
- normal to a curve
- tangent to a surface
- from its equation
- mean through points

Open the Planes1.CATPart document.

1. Click the **Plane** icon .

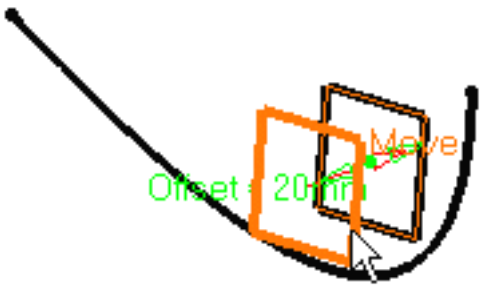
The Plane Definition dialog box appears.

2. Use the combo to choose the desired **Plane type**.

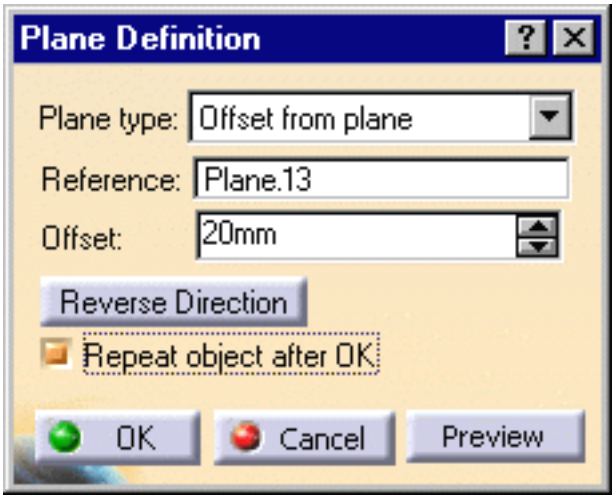
Once you have defined the plane, it is represented by a red square symbol, which you can move using the graphic manipulator.

Offset from plane

- Select a reference **Plane** then enter an **Offset** value.



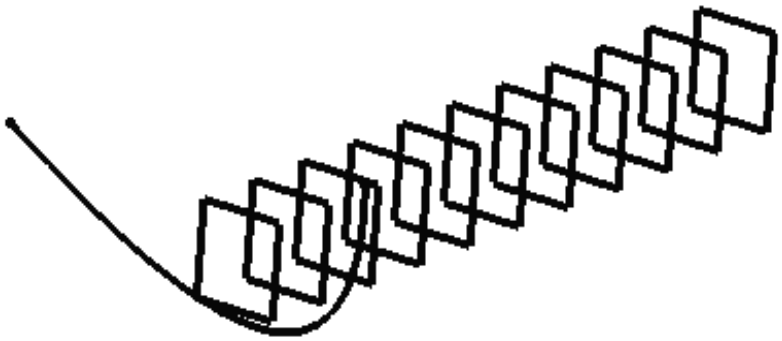
A plane is displayed offset from the reference plane.



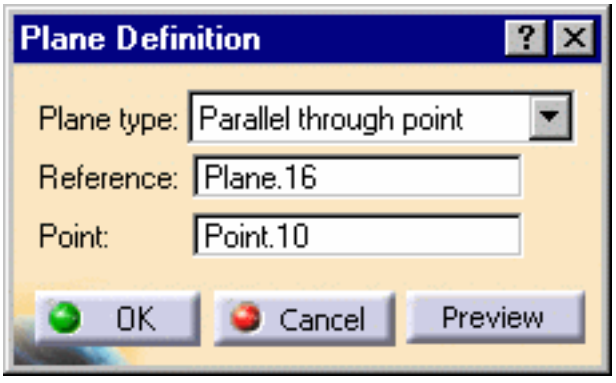
Use the **Reverse Direction** button to reverse the change the offset direction, or simply click on the arrow in the geometry.

- Click the **Repeat object after OK** if you wish to create more offset planes .
  In this case, the **Object Repetition** dialog box is displayed, and you key in the number of instances to be created before pressing OK.

As many planes as indicated in the dialog box are created (including the one you were currently creating), each separated from the initial plane by a multiple of the **Offset** value.

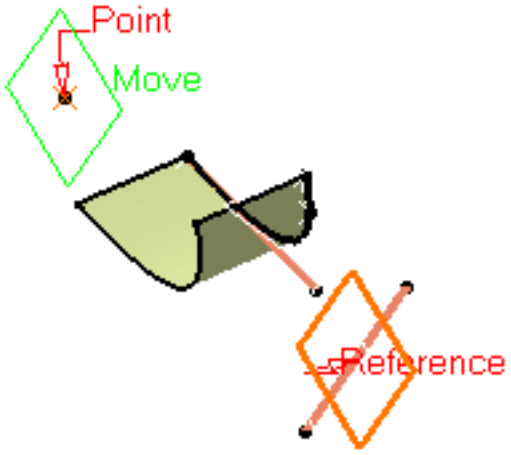


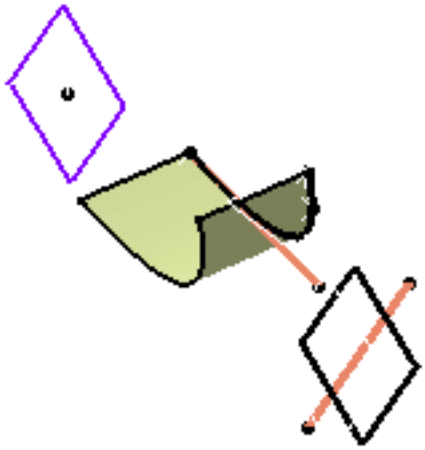
Parallel through point



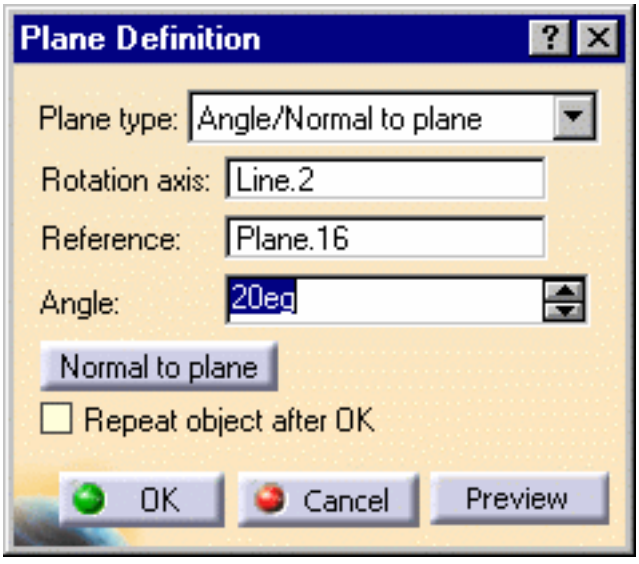
- Select a reference **Plane** and a **Point**.

A plane is displayed parallel to the reference plane and passing through the selected point.



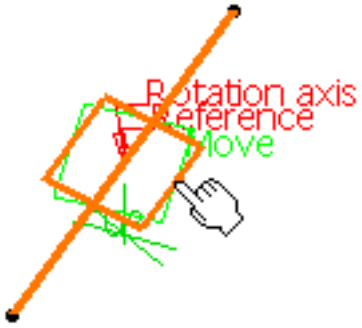


Angle or normal to plane



- Select a reference **Plane** and a **Rotation axis**.
  This axis can be any line or an implicit element, such as a cylinder axis for example. To select the latter press and hold the Shift key while moving the pointer over the element, then click it.
- Enter an **Angle** value.

A plane is displayed passing through the rotation axis. It is oriented at the specified angle to the reference plane.



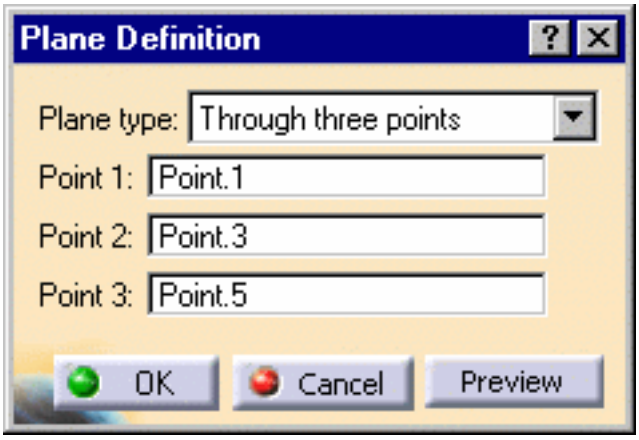
- Click the **Repeat object after OK** if you wish to create more planes at an angle from the initial plane.
  In this case, the **Object Repetition** dialog box is displayed, and you key in the number of instances to be created before pressing OK.

As many planes as indicated in the dialog box are created (including the one you were currently creating), each separated from the initial plane by a multiple of the **Angle** value.

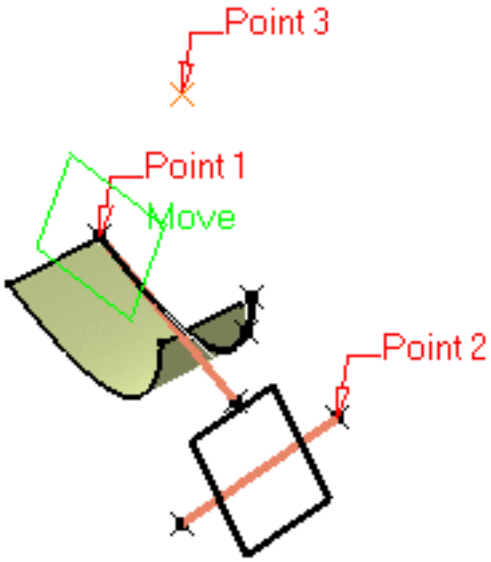
Here we created five planes at an angle of 20 degrees.

This plane type enables to edit the plane's parameters. Refer to Editing Parameters to find out how to display these parameters in the 3D geometry.

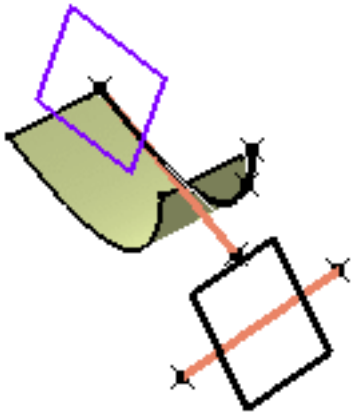
Through three points



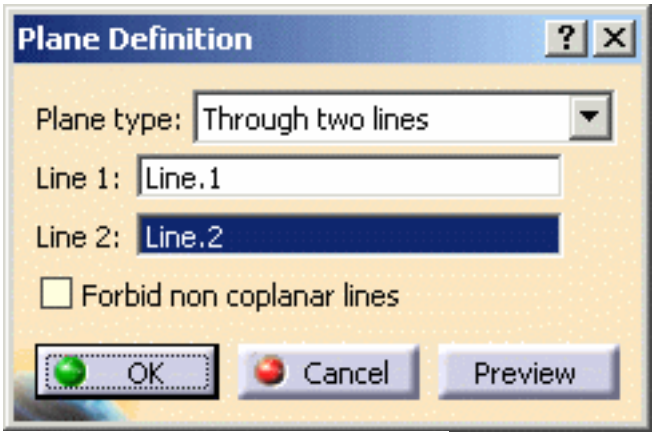
- Select three points.



The plane passing through the three points is displayed. You can move it simply by dragging it to the desired location.

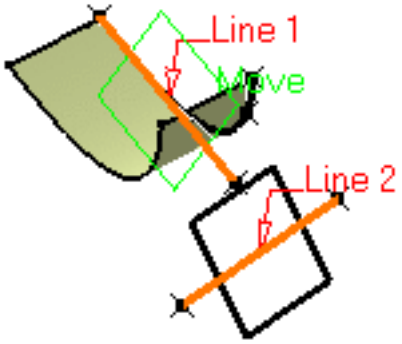


Through two lines

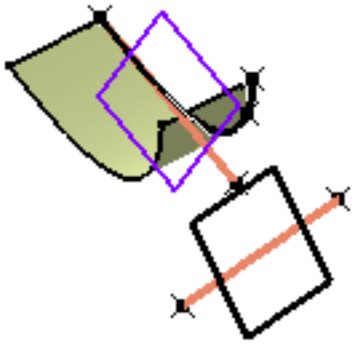


- Select two lines.

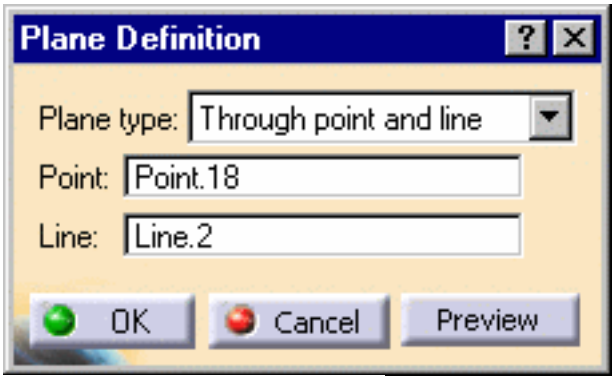
The plane passing through the two line directions is displayed.
When these two lines are not coplanar, the vector of the second line is moved to the first line location to define the plane's second direction.



Check the **Forbid non coplanar lines button** to specify that both lines be in the same plane.

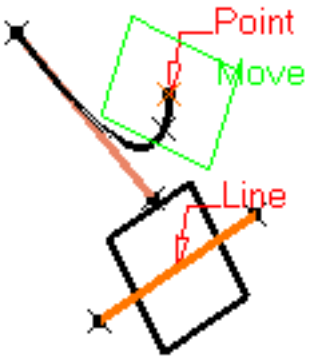


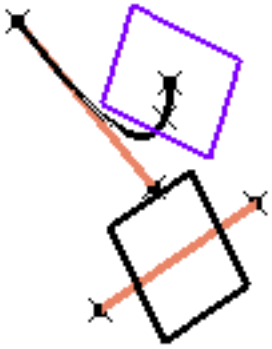
Through point and line



- Select a **Point** and a **Line**.

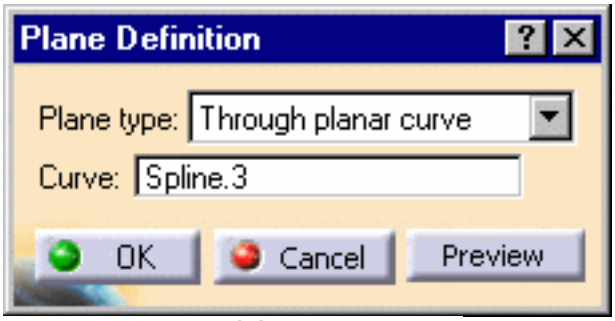
The plane passing through the point and the line is displayed.

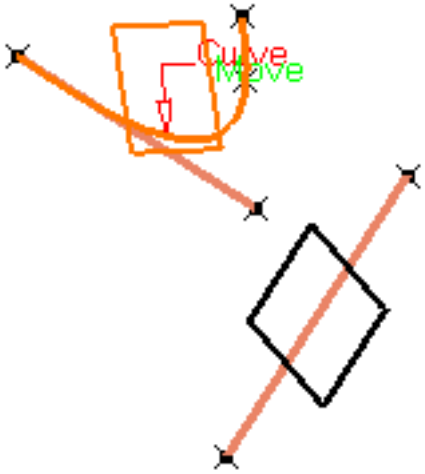




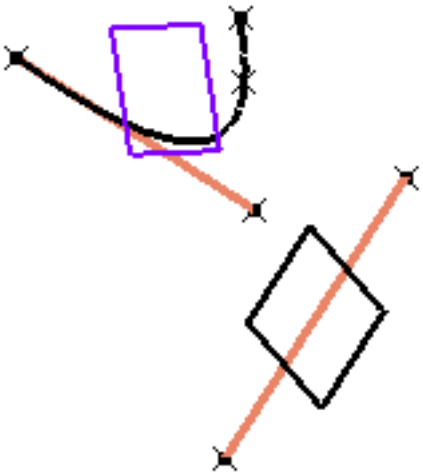
Through planar curve

- Select a planar **Curve**.



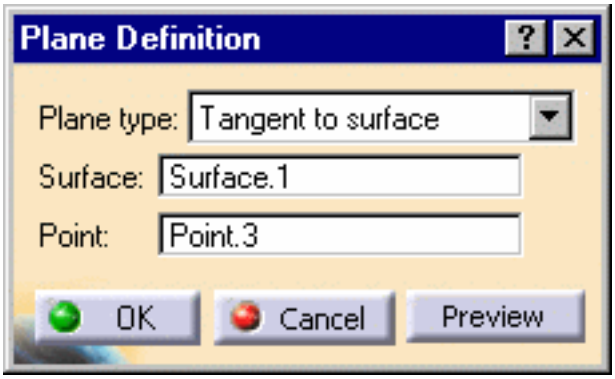


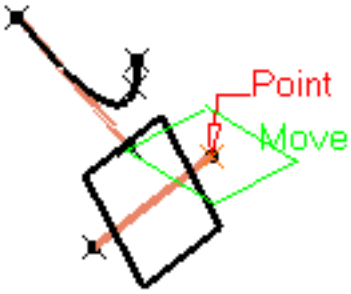
The plane containing the curve is displayed.



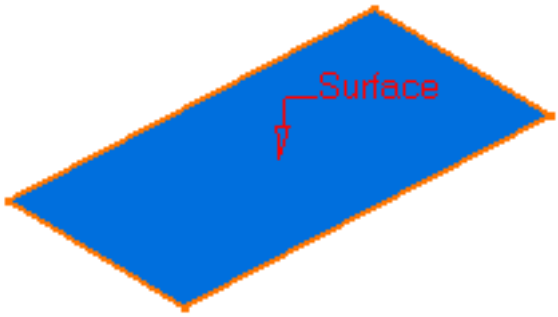
Tangent to surface

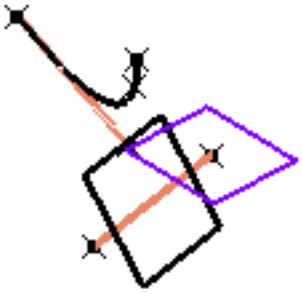
- Select a reference **Surface** and a **Point**.

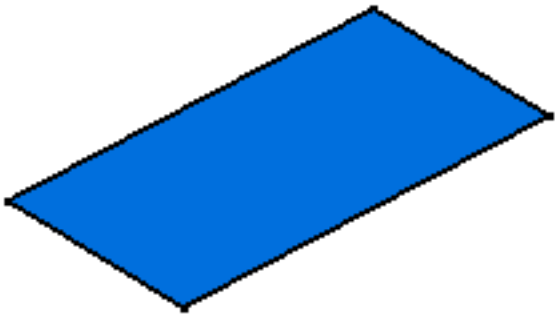




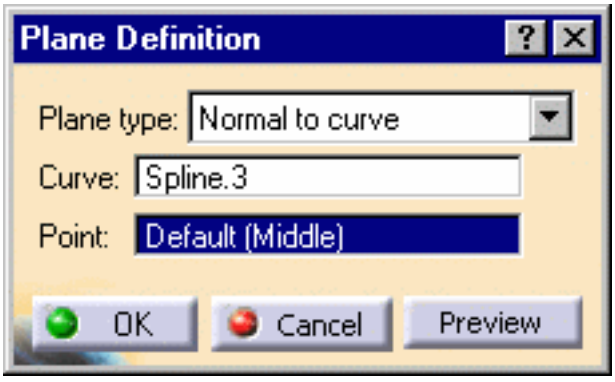
A plane is displayed tangent to the surface at the specified point.



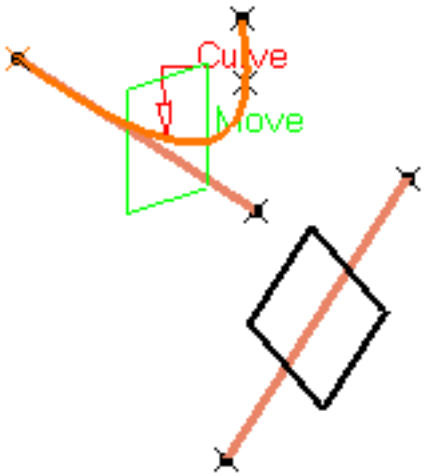




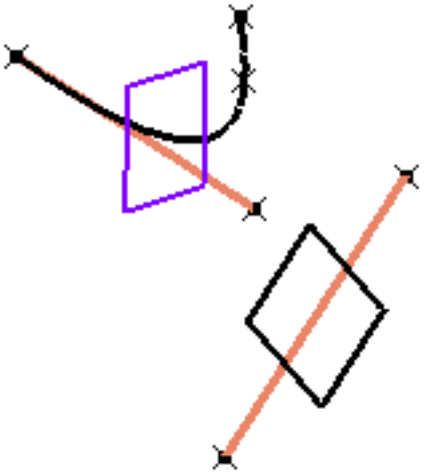
Normal to curve



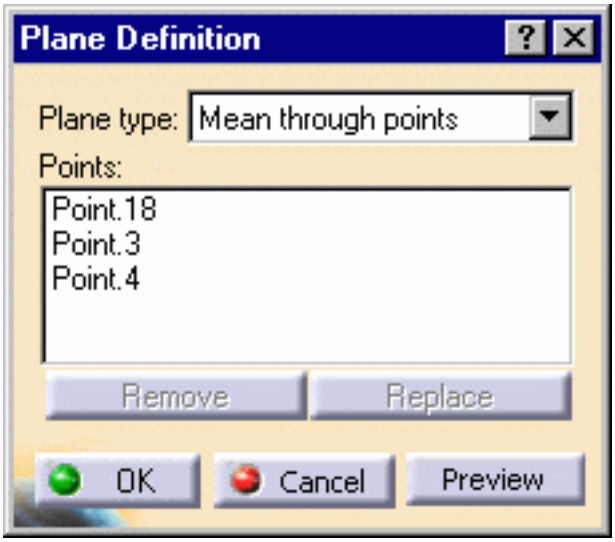
- Select a reference **Curve**.
- You can select a **Point**. By default, the curve's middle point is selecte.



A plane is displayed normal to the curve at the specified point.

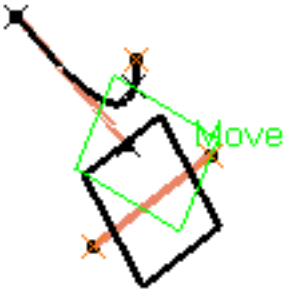


Mean through points

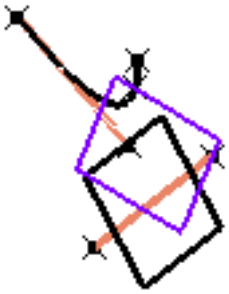


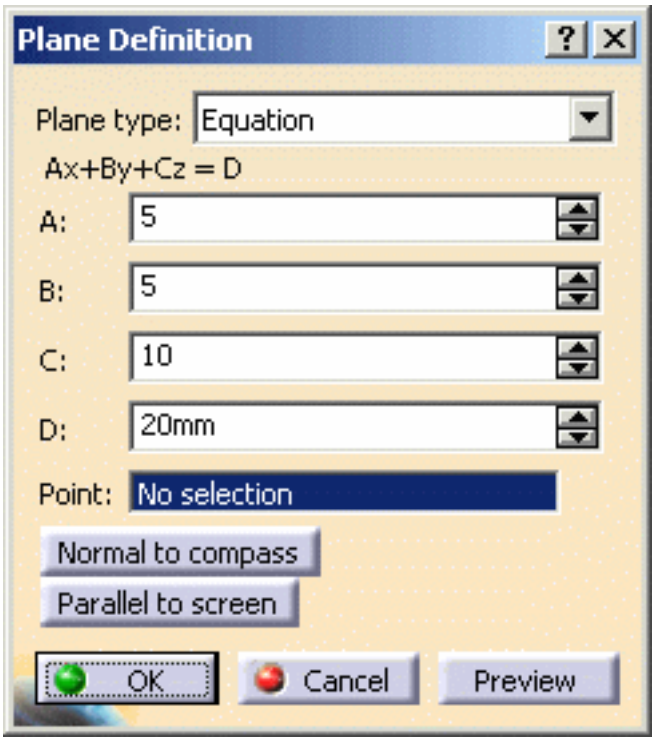
- Select three or more points to display the mean plane through these points.

It is possible to edit the plane by first selecting a point in the dialog box list then choosing an option to either:



- **Remove** the selected point
- **Replace** the selected point by another point.

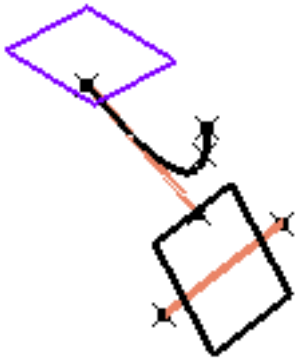




Equation

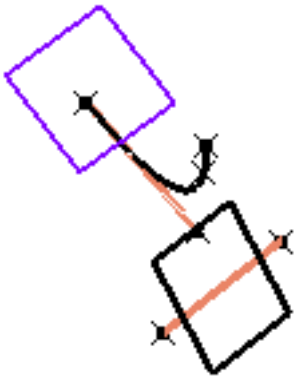
- Enter the **A**, **B**, **C**, **D** components of the **Ax + By + Cz = D** plane equation.

Select a point to position the plane through this point, you are able to modify **A**, **B**, and **C** components, the **D** component becomes grayed.



Use the **Normal to compass** button to position the plane perpendicular to the compass direction.

Use the **Parallel to screen** button to parallel to the screen current view.



3. Click **OK** to create the plane.

The plane (identified as Plane.xxx) is added to the specification tree.

- Parameters can be edited in the 3D geometry. For more information, refer to the Editing Parameters chapter.
- You can isolate a plane in order to cut the links it has with the geometry used to create it. To do so, use the **Isolate** contextual menu. For more information, refer to the Isolating Features chapter.

Creating the Electrical Bundle

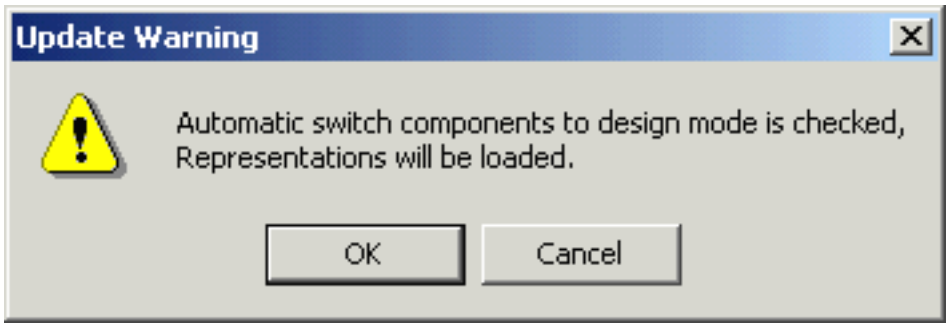
This task shows you how to create an electrical bundle.
A bundle or electrical bundle is a document containing wires.

Open a document containing a geometrical bundle composed of electrical devices and bundle segments.

If you work in visualization mode, since routing is possible with this mode, you need to update your document at opening to load the publication.

a. Click the **Update** button .

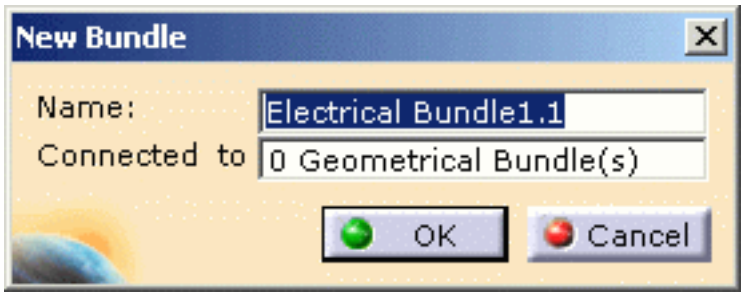
An update warning opens:



b. Click **OK** to validate.

1. Select the **New Bundle** button .

The New Bundle dialog box opens:

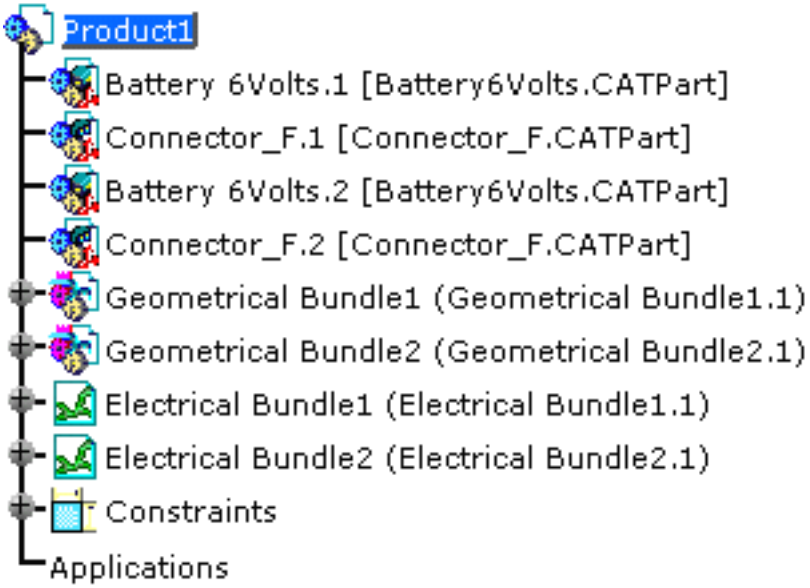


2. Change the bundle name if needed.

3. Select a geometrical bundle in the specification tree that you want to be connected to the new bundle.

4. Click **OK** to validate.

The new bundle is automatically created under the active product.
It is added to the specification tree.



An electrical bundle is associated to a geometrical bundle by the wires it contains.

If the geometry is not loaded, check the Product Structure settings.

An alternative to display the geometry is to choose the **Representations -> Activate Terminal Node** item. Right-click **Product1** to use the contextual menu or select **Edit -> Representations**.

Selecting External Data Systems

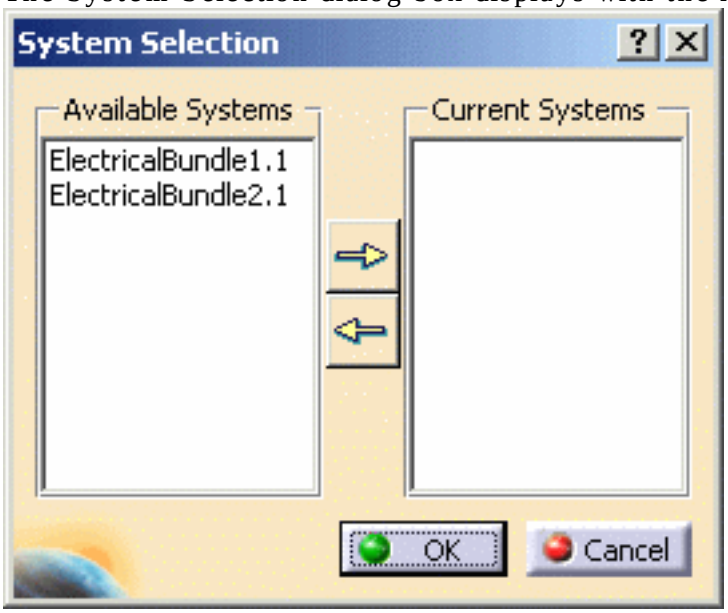
This task shows how to select the external system before routing.

The Electrical Bundle has been defined in the previous task.

Make sure you have set up the option to locate the iXF systems repository.

1. Select the **Select External Systems** button .

   The System Selection dialog box displays with the XML files available:



2. Select one or more system files and click the right arrow .
3. Click **OK** to validate.

   The component list is filled up with this data and available for routing.

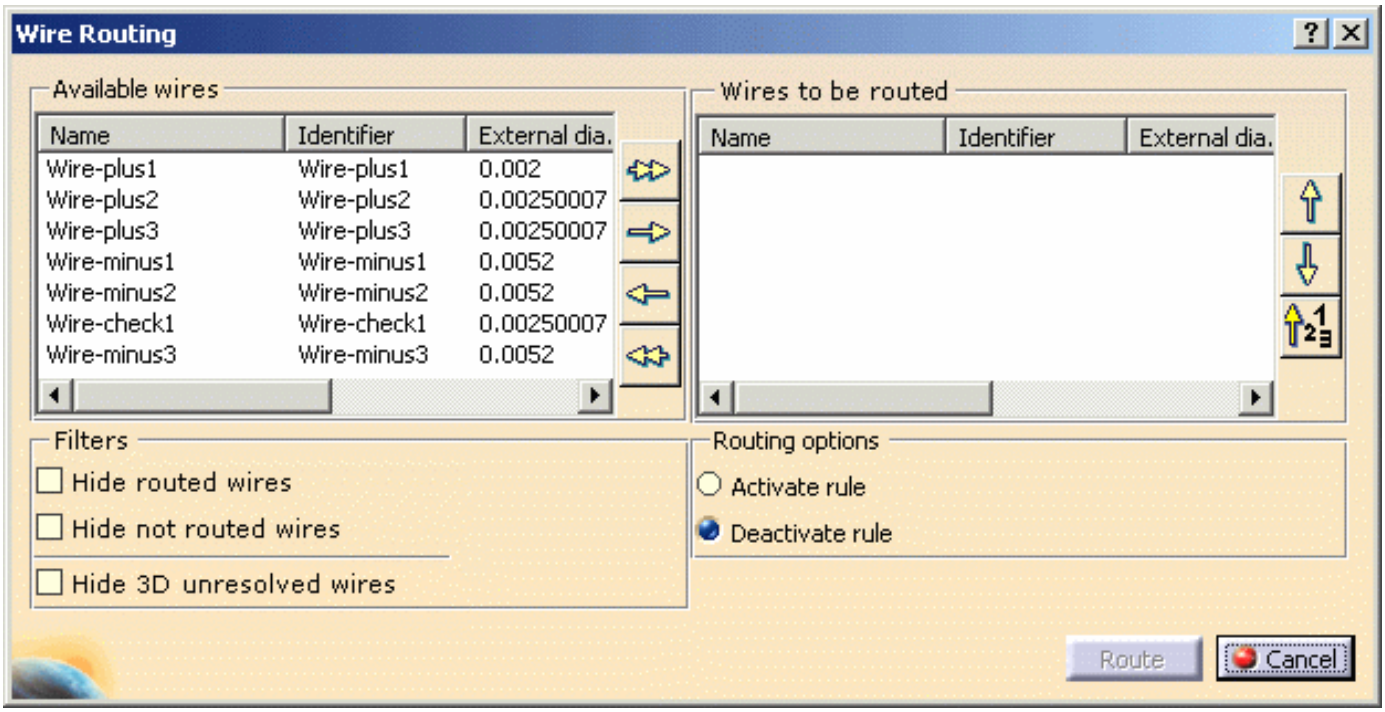
If a system has already been selected, data is reloaded.

Routing Wires from External Data

This task shows you how to route wires.

The system has been selected in the previous task.

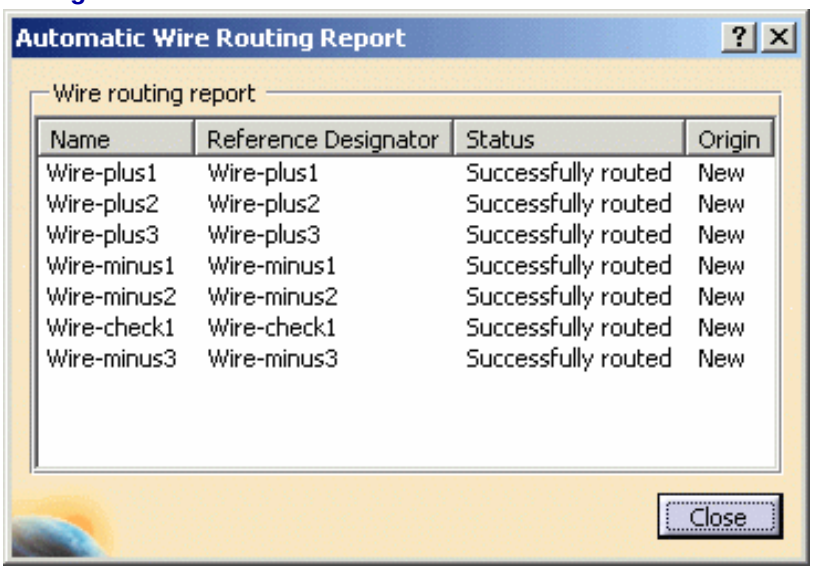
1. Select the **Automatic Wire Routing** button .
2. The Wire Routing dialog box opens with connectivity and attribute information:



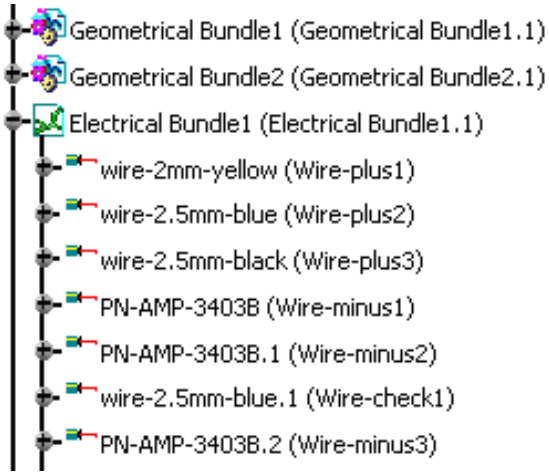
3. Select wires and click the right arrow .
   The selected wires shift to the right column: they will be routed.

   Using the double right-arrow select all the wires and send them in the right column.

4. Click **Route**.
   The Automatic Wire Routing Report dialog box is displayed:

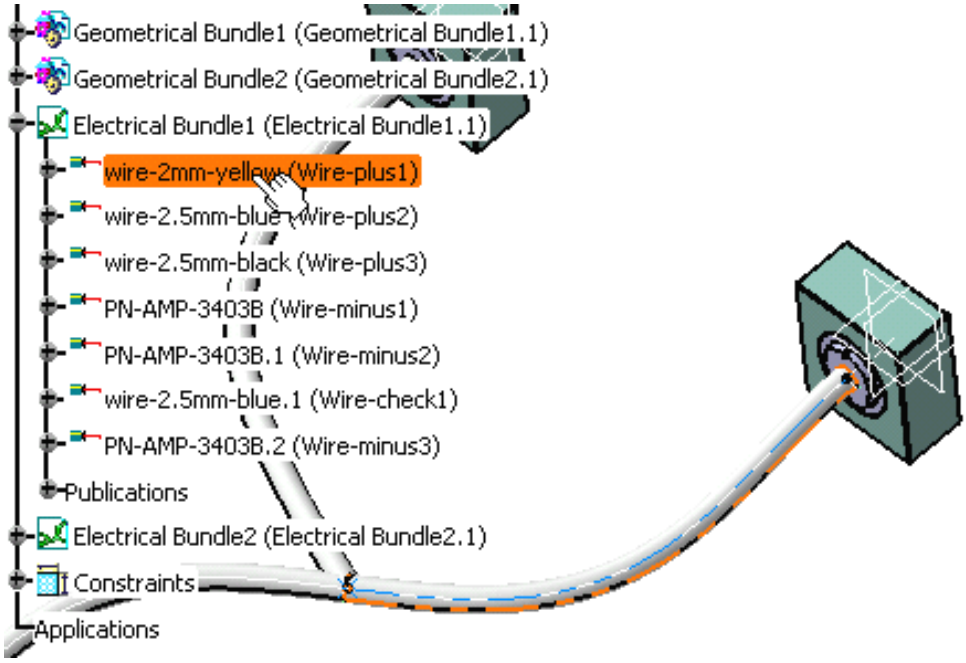


The specification tree is updated, showing the wires routed:



5. Close the report window.

Place the mouse pointer over a wire in the specification tree to highlight in the geometry the bundle segments containing the selected wire.


Geometrical Bundle1 (Geometrical Bundle1.1)
Geometrical Bundle2 (Geometrical Bundle2.1)
Electrical Bundle1 (Electrical Bundle1.1)
wire-2mm-yellow (Wire-plus1)
wire-2.5mm-blue (Wire-plus2)
wire-2.5mm-black (Wire-plus3)
PN-AMP-3403B (Wire-minus1)
PN-AMP-3403B.1 (Wire-minus2)
wire-2.5mm-blue.1 (Wire-check1)
PN-AMP-3403B.2 (Wire-minus3)
Publications
Electrical Bundle2 (Electrical Bundle2.1)
Constraints
Applications

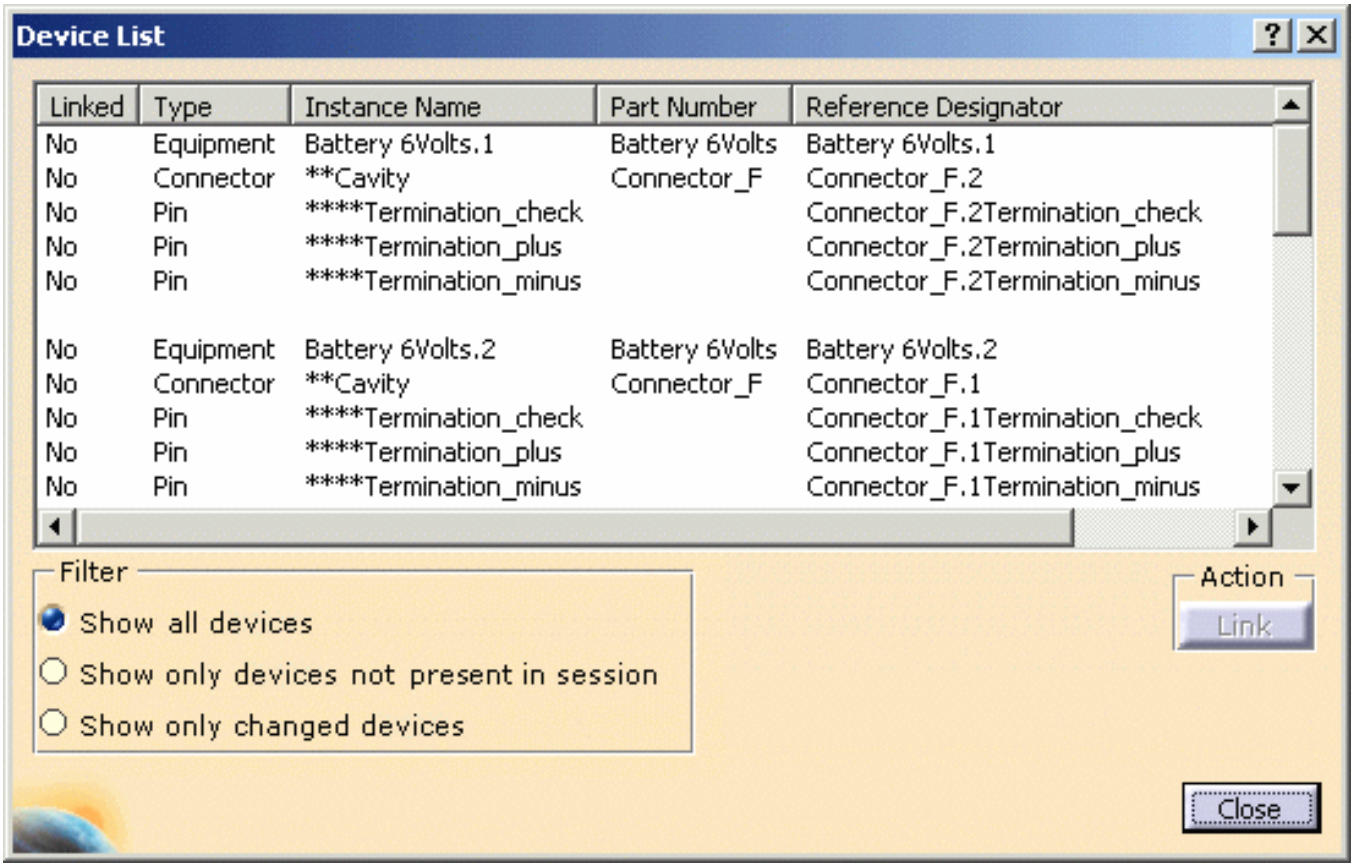
Managing Links from External Data

This task explains how to link electrical components from the external device list to physical devices.

The system has been selected in the previous task.

1. Click the **Manage Links** button.

   The device list displays:



   The components you can link are:

   - equipments
   - connectors
   - pins.

   You can filter the list:

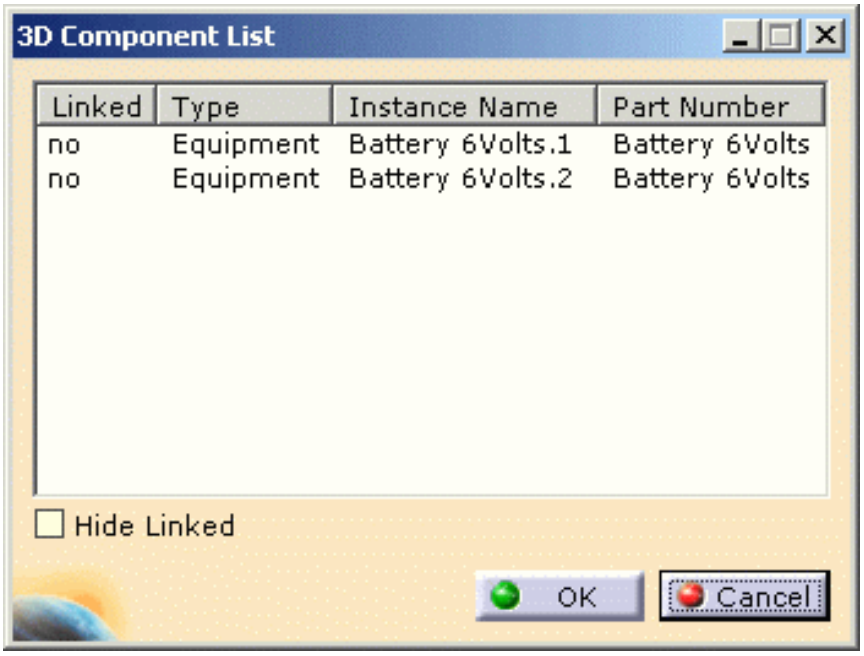
   - **Show all devices:** the list displays all the devices referenced in the external data file, even if they are not placed in the geometry.
   - **Show only devices present in session:** the list displays only the devices placed in the geometry.
   - **Show only changed devices:** the list displays only the devices which part numbers have been changed in the external data file.

2. Select a component.

   If the component is not linked to a 3D element, the **Link** button becomes available.

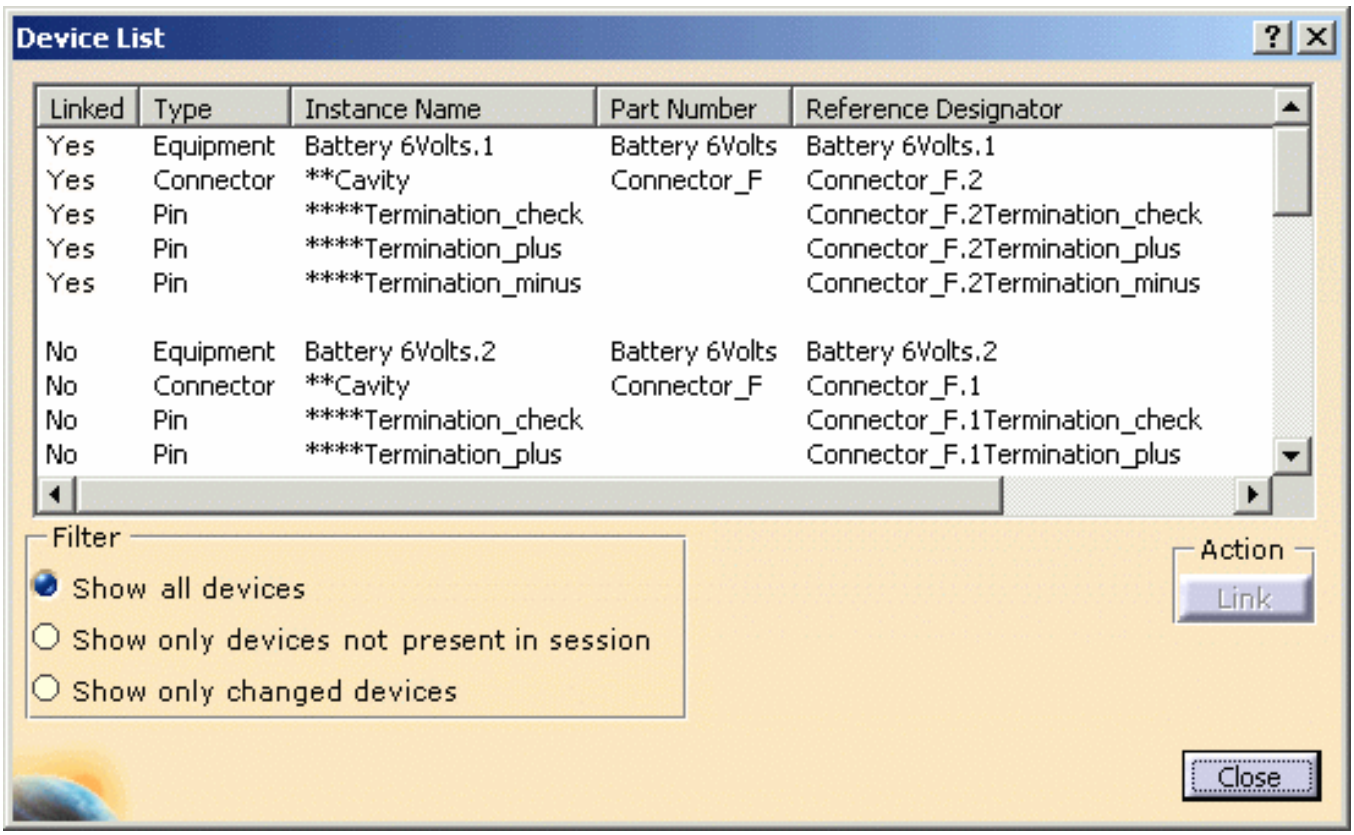
3. Click **Link**.

The 3D Component List opens.



4. Select an equipment which is not linked and click **OK**.

An alternative is to select the component in the specification tree.
The equipment is linked, together with the connector that belongs to it.
The component list is updated: the equipment, connector and pins are shown as linked in the device list.



If all the sub-element part numbers are filled up in the device list with the correct Reference Designator, they are automatically linked.

5. Click **Close** when you are done.

Viewing Related Objects

This task shows how to use the **Related Objects** viewer to navigate through the objects connected to the selected object.

You can focus on an object and see how it was constructed via its related objects.
The related objects command identifies the parent, any children or connected objects and the relationship between objects. It is available when none of the icons of the workbench are activated that is to say when you are in **Select** command .

Accessing related objects can be done in two ways:

- by clicking the **Related Objects** icon in the toolbar
- from the contextual menu, by selecting **Related Objects**.

The different options available are:

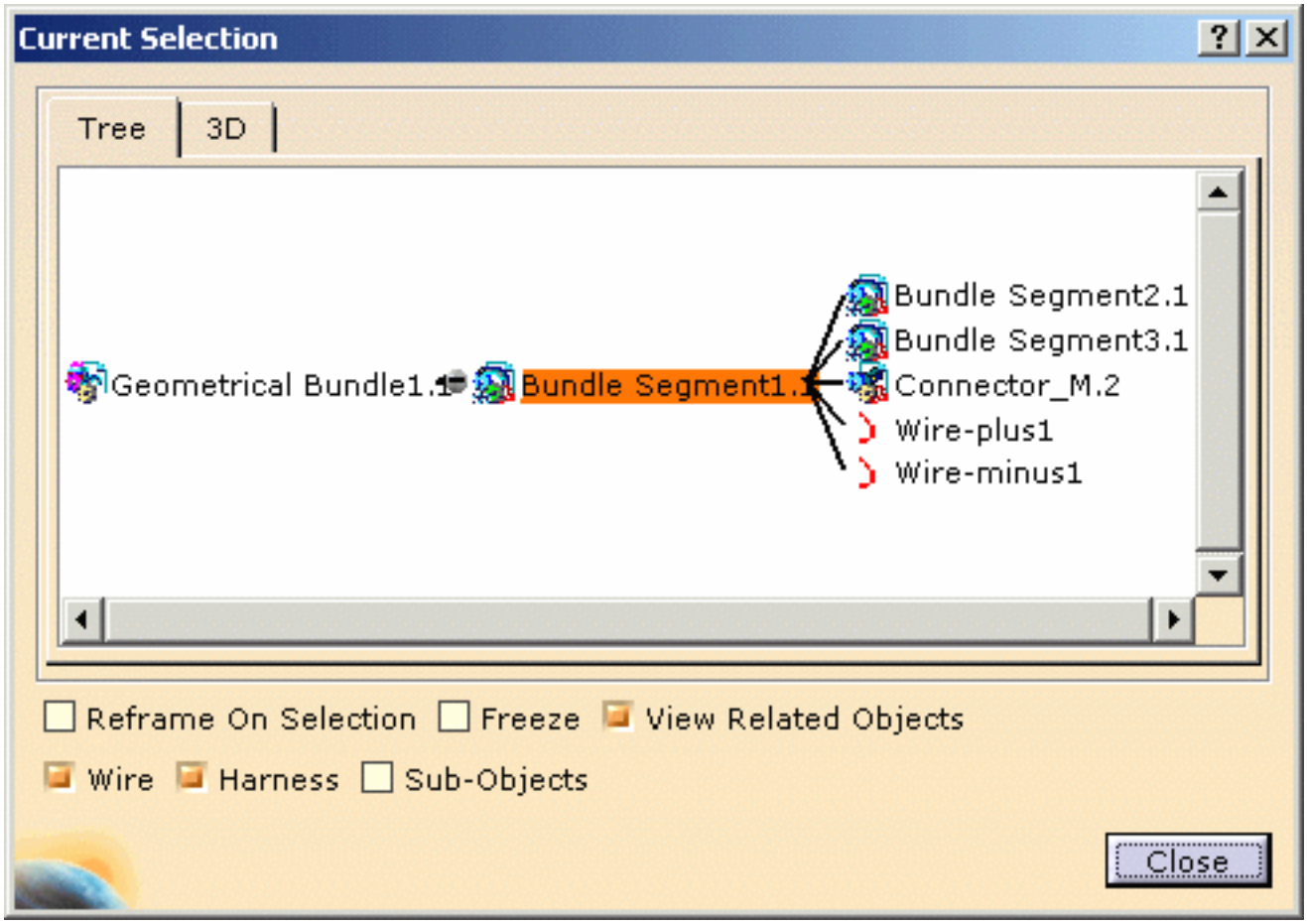
Reframe on selection:	reframes the main 3D window on the Related objects selection.
Reframe on selection:	the main window selection corresponds to the Related objects selection. The main window display is not reframed.
Freeze:	freezes the contents of the Related objects window. You can still navigate in the main window: the Related objects view will not be updated.
Freeze:	the Related objects selection corresponds to the main window selection.
View related objects:	displays the parent, any children or connected objects.
View related objects:	only displays the selected objects.
Wire:	displays the wires contained in the bundle segment, the bundle segments and devices connected to this bundle segment.
Wire:	hides the wires contained in the bundle segment, shows the bundle segments and devices.
Harness:	displays the relationship to the harness: connectors, equipments, bundle segments, wires.
Harness:	hides the children bundle segments, only shows the wires.
Sub objects:	displays the electrical contents.
Sub objects:	hides the electrical contents.

On a complex electrical system, the **3D view** allows you to limit display to a specific area thus enlightening the information regarding this area.

1. Select an object: a bundle segment for example.

2. Click the **Related Objects** button  .

   The Related Objects dialog box appears. The geometry area and the specification tree are reframed on the object selected.

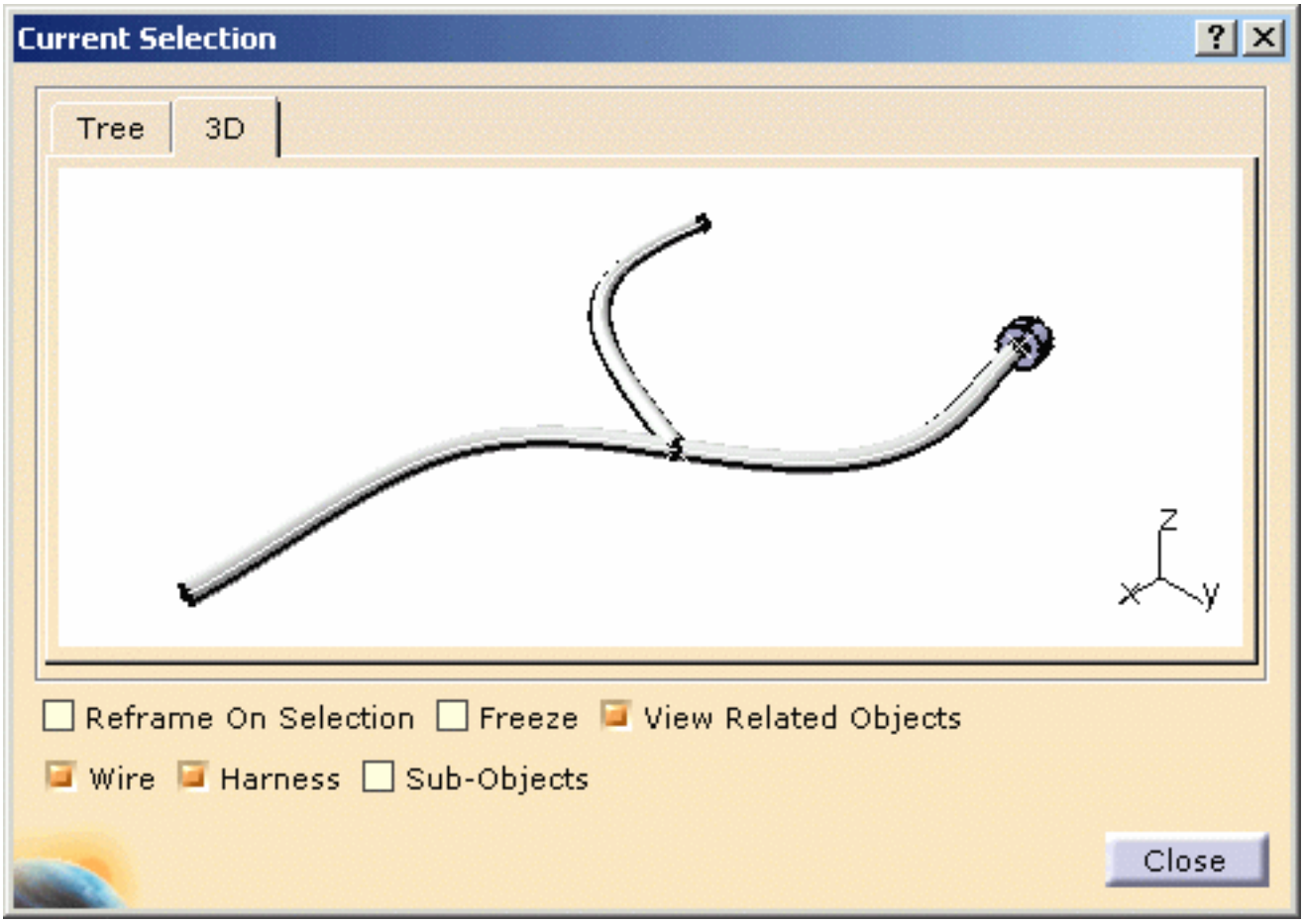


3. Select an object in this window.

   The dialog box focuses on the object selected: **Bundle Segment1.1** (center of the window) and shows the parent and the connected objects:

   - on the left is the parent object (Assembly meaning).
     It represents the container object.
   - on the right are the children objects (Assembly meaning), connected to the **Bundle Segment1.1**. They represent the contents.

4. Click the **3D** tab to display the geometry.



5. Close the dialog box to exit the **Related Objects** viewer.

Defining the Harness Flattening Parameters

This task explains how to define the flattening parameters.

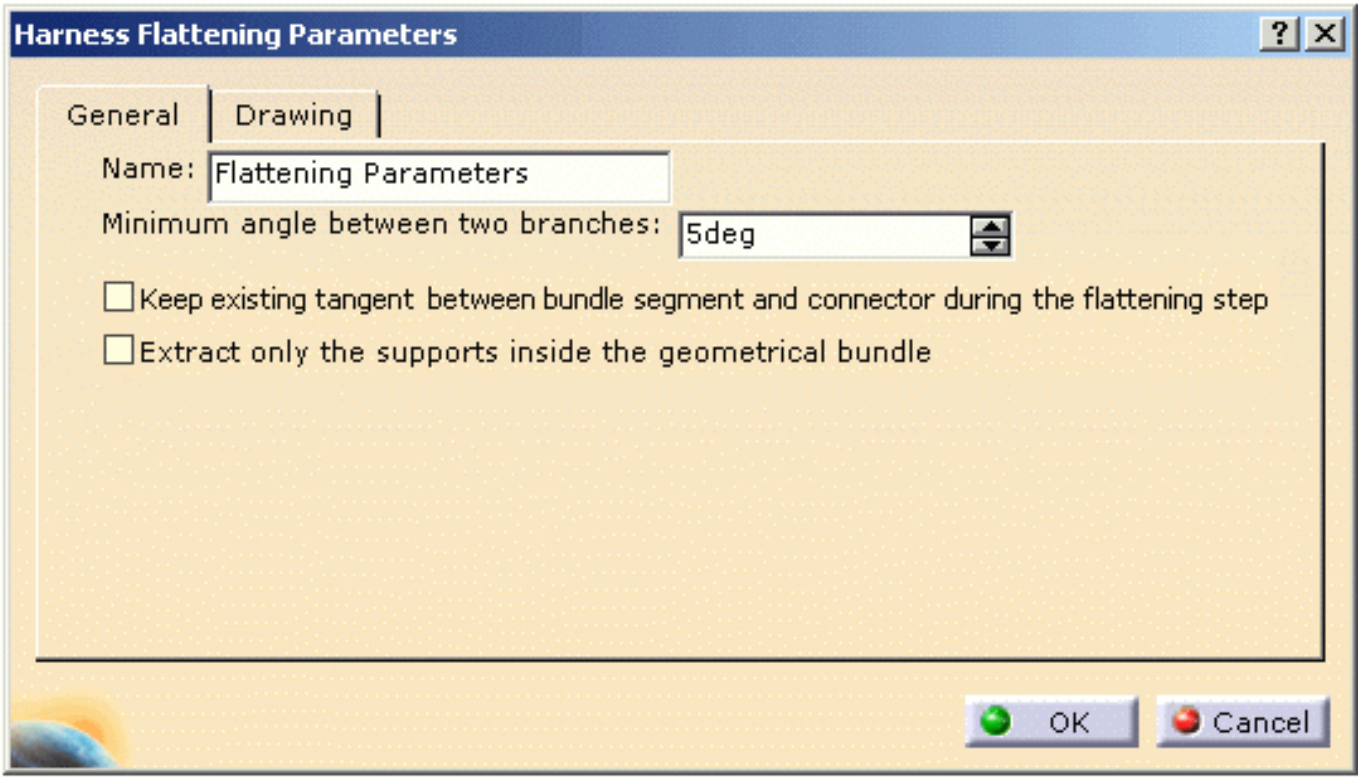
This should be done before using any other function of the **Harness Flattening** toolbar.

Open a new product document using the **File -> New...** command.

Choose the **Product** type.

1. Click the **Harness Flattening Parameters** button .

The Harness Flattening Parameters dialog box opens on the **General** tab:



This tab lets you define:

Name

In the specification tree, this icon is displayed with the name chosen in this field.

Flattening Parameters is the default value.

Minimum angle between two branches

During the flattening process, this minimum angle is applied every time an angle between two branches is null in the 3D design. That way, bundle segments are never superposed upon one another and can easily be made out.

The default value is 5deg.

Keep existing tangent between bundle segment and connector during the flattening step

This option gives a better flattening shape for flattening.
By default this option is not checked.

Extract only the supports inside the geometrical bundle

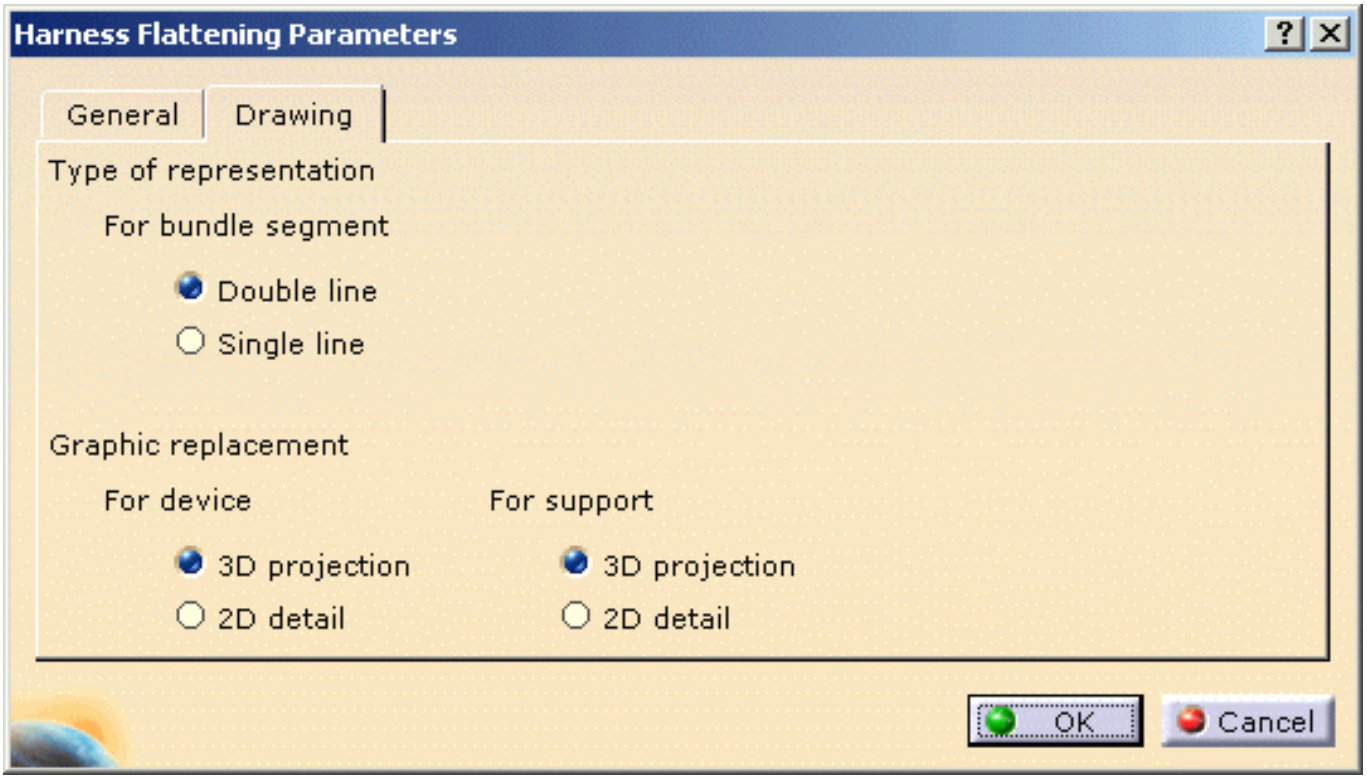
The supports which do not belong to the geometrical bundle will not appear in the extracted document.
This option allows you to manage two types of support:

- The supports which must be assembled with the harness during the manufacturing step.
  This type of support is created inside the geometrical bundle.

- The supports which are linked to the structure.
  These supports are added to the harness during the installation step.
  This type of supports is created outside the geometrical bundle.

By default this option is not checked.

2. Select the **Drawing** tab.



This tab lets you define:

- the type of representation for **bundle segments:**
  - with **Double line**
    It is the default value.
  - or **Single line**.
- the graphic representation you want to use when replacing **devices** or **supports**:
  - keep the **3D projection**.
    It is the default value.
  - or use a **2D detail** previously stored in a catalog.

    In this case, the catalog name and path must be defined in the dedicated option.

3. Select your options and click **OK** to validate.

The parameters you have entered will be automatically applied to the other functions available in this workbench. You will be able to modify them at any time during your session by double-clicking the **Flattening Parameters** icon in the specification tree.

Extracting 3D Data

This task explains how to extract the geometrical and electrical bundles, with all the devices that are associated to them, in order to duplicate the information they contained in the new document.

The 3D document containing a geometrical bundle and an electrical bundle is open as well as the new product document.

Make sure the source document has not been modified, otherwise you will be prompted to save it before extracting the data.

In the new product document:

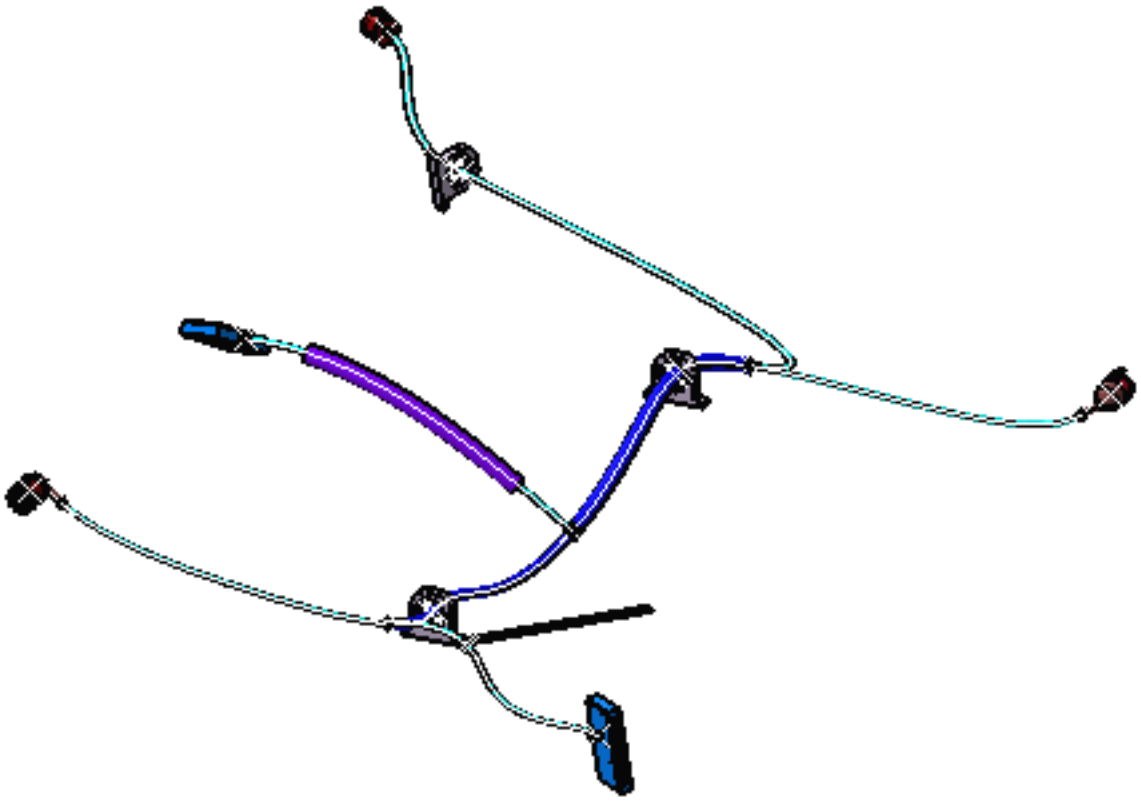
1. Click the **Extract** button  .

   You are prompted to select an electrical or a geometrical bundle.

   It enables you to extract the information contained in the geometrical or the electrical bundles from your first document.

   When duplicating an electrical bundle, all the links with the geometrical bundles associated to it are maintained.

2. Select the geometrical bundle you want to extract from the source document, whether in the geometry or in the specification tree.



When the extraction is performed the source document automatically closes.

3. Save your data.

Flattening the Harness

This task explains how to flatten out bundle segments.

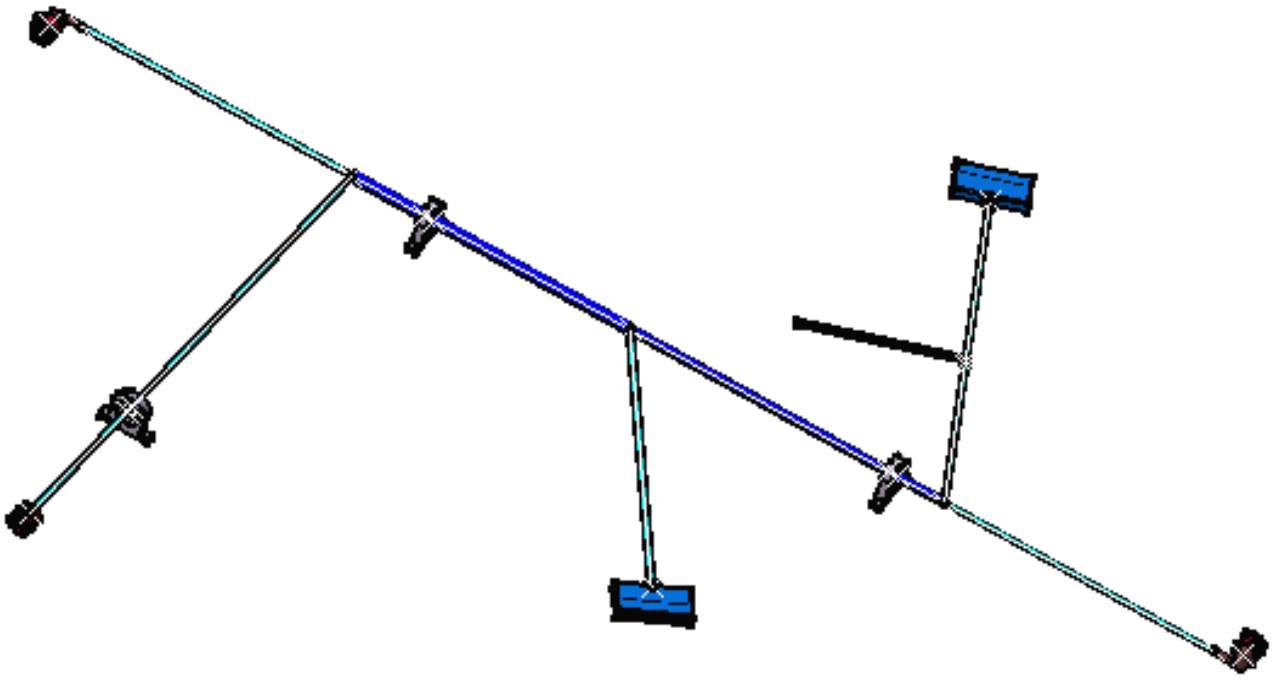
The document contains the data extracted in the previous task.

You can now flatten the whole geometrical/electrical bundle.

To do so:

1. Click the **Flatten** button .
2. Select the geometrical bundle, in the specification tree or in the geometry.
3. Click **OK** to validate.

   The result looks like this:



Rotating Bundle Segments

This task shows you how to rotate a bundle segment.

Two types of rotation can be considered: you can

- rotate a bundle segment around a bundle segment extremity, whatever the direction.
- bend a bundle segment whatever the selected plane.

Rotating Bundle Segments whatever the Selected Plan

This task shows you how to rotate a bundle segment around a bundle segment extremity whatever the direction.

The document contains the flattened harness obtained in the previous task.

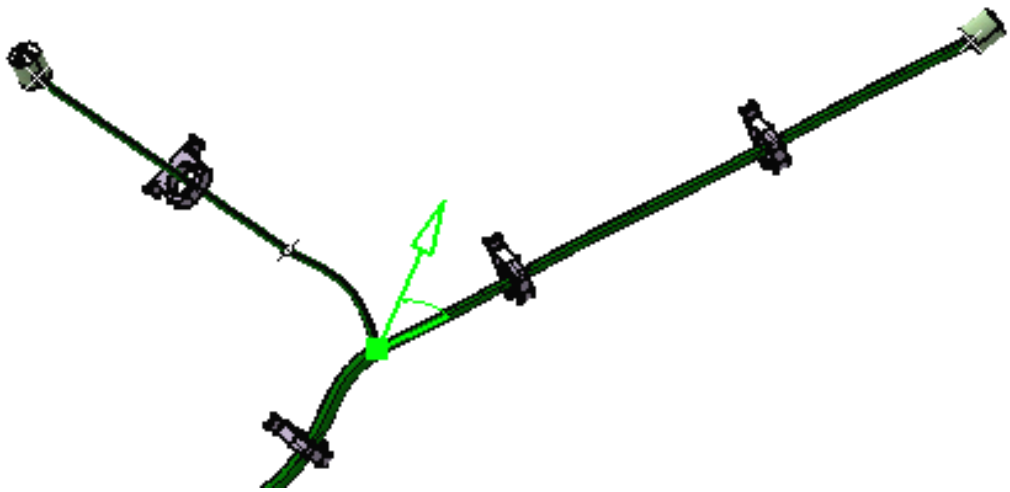
1. Click the **Rotate** button .

   You are prompted to select a bundle segment extremity in the geometry. It is impossible to select the bundle segment in the specification tree, since the exact position of the selected point has to be known.

2. Select an extremity.

   A green arrow and a dialog box appear at the same time.

   a. A green arrow indicating the bundle direction appears on the geometry. The user can modify the angle by selecting the green arrow directly.



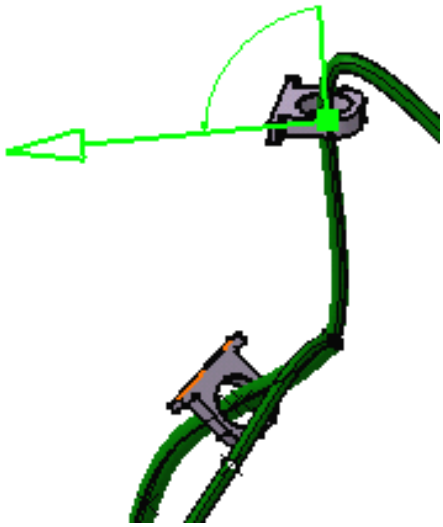
   b. The Define Direction dialog box pops up.

The default plane is the one you have defined in the Harness Flattening Parameters dialog box at the beginning of your session.

3. Whether you indicate the bundle main direction or you specify its angle and direction values one at a time.

   a. Enter the main direction.

      In this example, you can select a geometrical line or a pad edge as the main direction.



   b. Enter an angle value.

      As you are changing the angle value, the green arrow is moving to show you the direction the bundle segment is about to take.

   c. Select two directions to retrieve the angle value.

      As above, you can select a geometrical line or a pad edge. Once the two directions have been defined, the angle between them is automatically calculated.

Note that you can modify the direction of the bundle directly on the geometry by clicking the arc of circle around the selected point.

4. Click **Apply**.

The entered values are applied but the dialog box remains open and you can still modify the inputs.

5. Click **OK** to validate.

The result looks like this.

Bending Bundle Segments

This task shows you how to rotate a bundle segment around a bundle segment extremity whatever the direction.

The document contains the flattened harness obtained in the previous task.

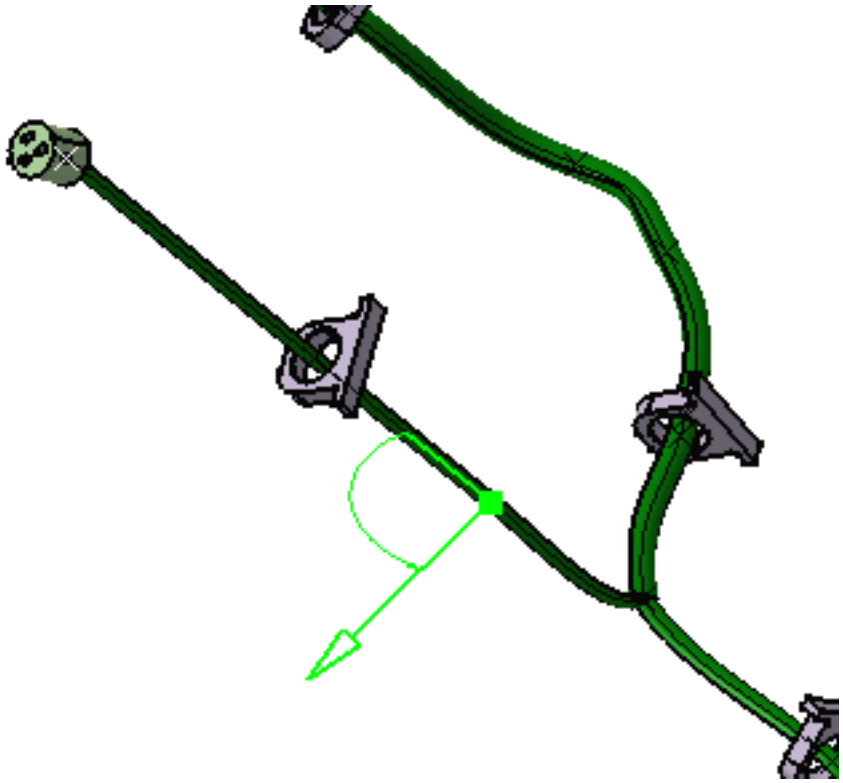
1. Click the **Rotate** button .
2. Select an intermediate point on a bundle segment.

   The bundle segment must be selected in the geometry and not in the specification tree, since the exact position of the selected point has to be known.

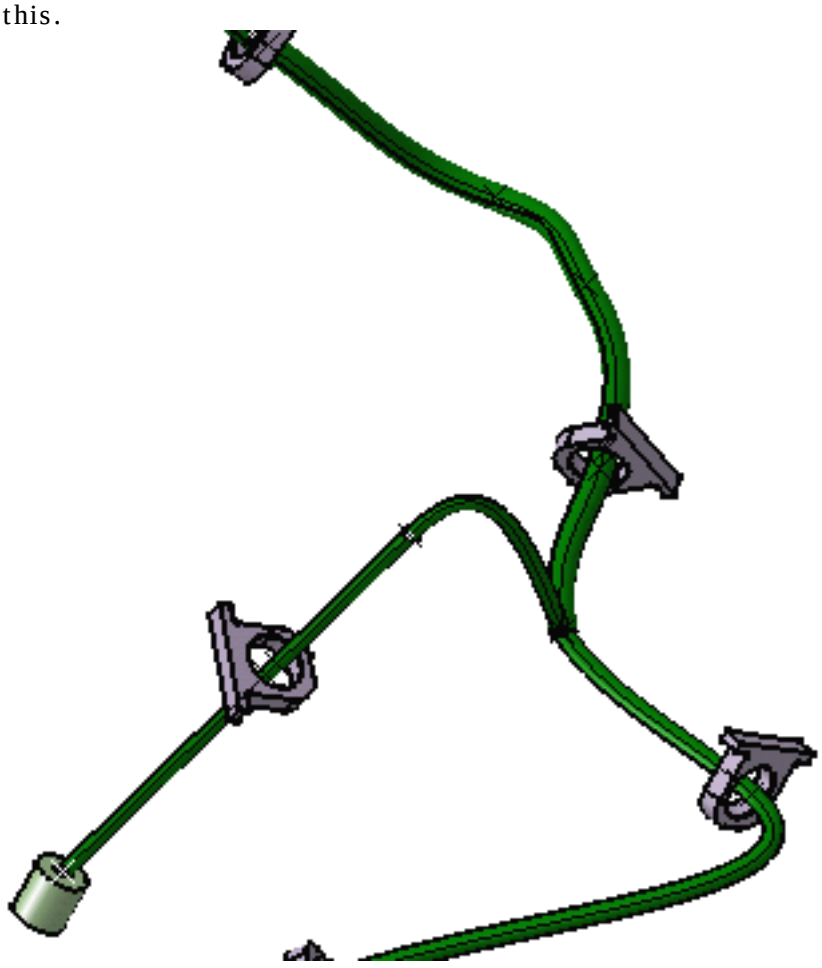
   a. The Define Direction dialog box pops up.

   b. A green arrow indicating the bundle direction appears in the geometry.

   You can also click the green arrow and circle arc to change the direction the bundle segment is about to take.



3. As in the previous task, either you select the main direction, or you indicate the angle or you specify two directions and the angle will be automatically calculated.
4. Click **OK** to validate.
5. The result looks like this.



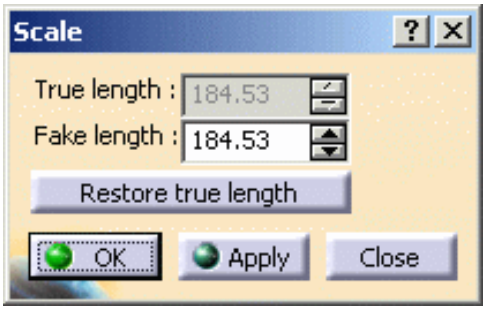
Scaling Bundle Segments

This task shows you how to enter a bundle segment fake length. Working with fake lengths enables you to fit the whole harness in the board.

The document contains the flattened harness obtained in the previous task.

1. Click the **Scale** button .

   You are prompted to select a bundle segment in the geometry.
   The Scale dialog box pops up.
   The bundle segment true length is indicated.



2. Enter the fake length you want.

3. Click **OK** to validate.

   The support relative positions are maintained when working with fake lengths.
   If you want to work with true lengths again, press the **Restore true length** button.

Synchronizing the Environment

This task shows you how to synchronize automatically your 3D flattened geometry at any time during your session. You can add the missing components or remove the additional components, and also synchronize the bundle segment structure.

Make sure you have set up the option for the synchronization report repository.

The document containing the flattened harness is open from the previous task.

Open the initial document as well.

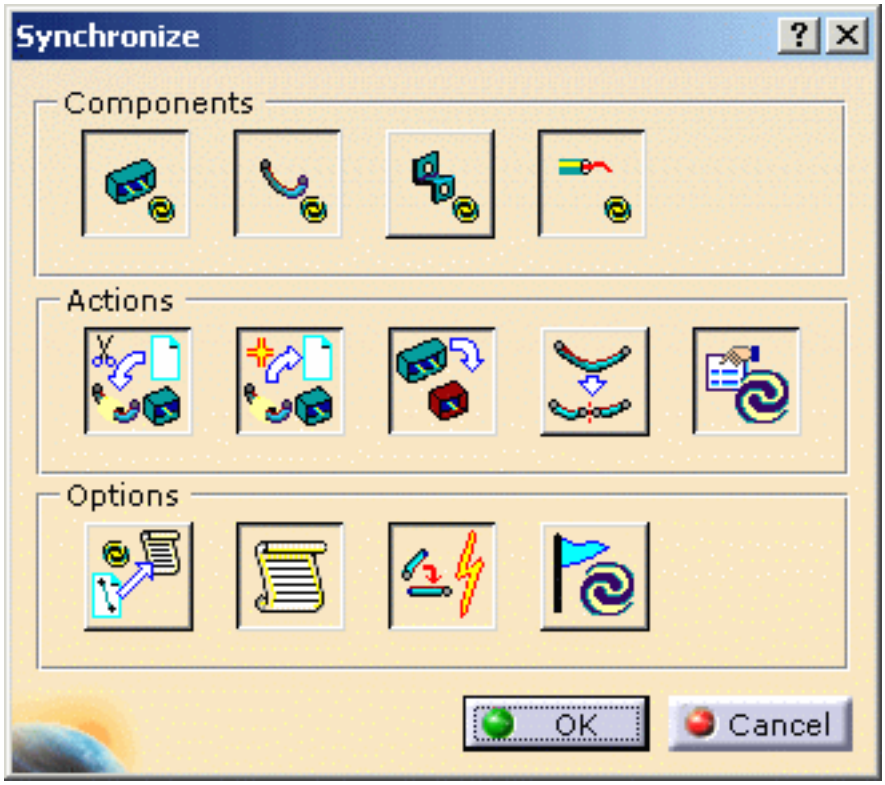
1. In the initial document, select a bundle segment and modify its length, diameter, bend radius, instance name or color.
2. Save your document.
3. Switch back to the flattened harness.
4. Click the **Synchronize** button  .

   - If there is no selection in the specification tree, the first electrical bundle modified will be synchronized. But, if there is more than one electrical bundle, you must select one otherwise a warning will be displayed.
   - If there is no electrical bundle, all the geometrical bundles will be selected and updated.
   - You can do a multi-selection (devices, supports, etc. at the same time).

   The Synchronize dialog box pops up:

   During the synchronization step, only the selected options will be performed. This will optimize the synchronization length.

   The html report will be generated accordingly and will show only the selected options.



This dialog box lets you define what you want to synchronize:

Components

- **Components** allows you to choose which components you want to update:
  - Devices
  - Bundle segments
  - Supports
  - Wires.

You can select at the same time as many components as needed.

Actions

- **Actions** allows you to customize the actions during the synchronization step:
  - Remove the components deleted in 3D
  - Add the components added in 3D
  - Replace the reference of components changed in 3D (it can only be used with the **Supports** or **Devices** options activated.)
  - Synchronize the bundle segment structure (it can only be used with the **Bundle Segments** option activated.)

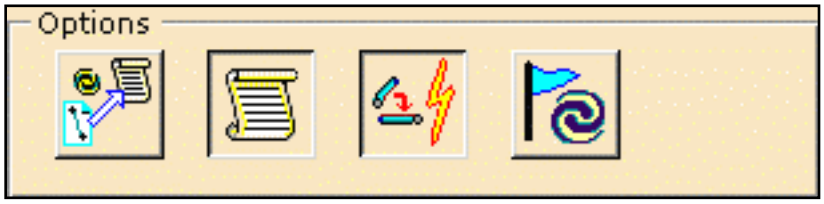
    It allows to:

    - add or remove intermediate bundle segments
    - replace one or more bundle segments with one or more different bundle segments.

  - Synchronize the attributes.
    For the synchronization of the position of supports, you must select **Bundle Segments** instead of **Supports**.

You can select as many actions as needed.

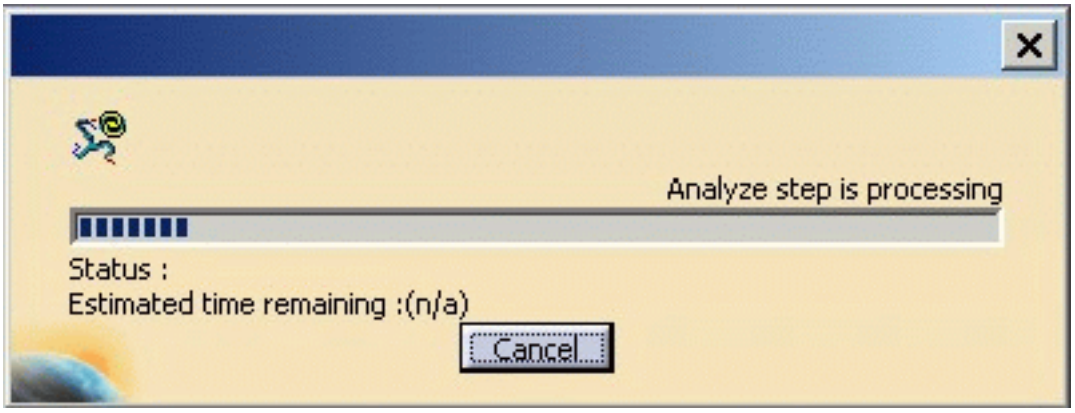
Options



- **Options**:
  - Simulation (generates a report without applying changes to the flattened document.)
  - Html report (generates the report in the **Tools -> Options** predefined folder.)
  - Automatic flattening on the added components
  - Reset the synchronization attributes.
    (all components will be set to the **False** value again, so only the newly changed components will be set to **True**. You can use Edit/Search to retrieve quickly all components.)

5. Click **OK** to validate.

A status bar appears showing the synchronization progress:



Here are the different steps that can be seen through the synchronization process according to the options previously selected:

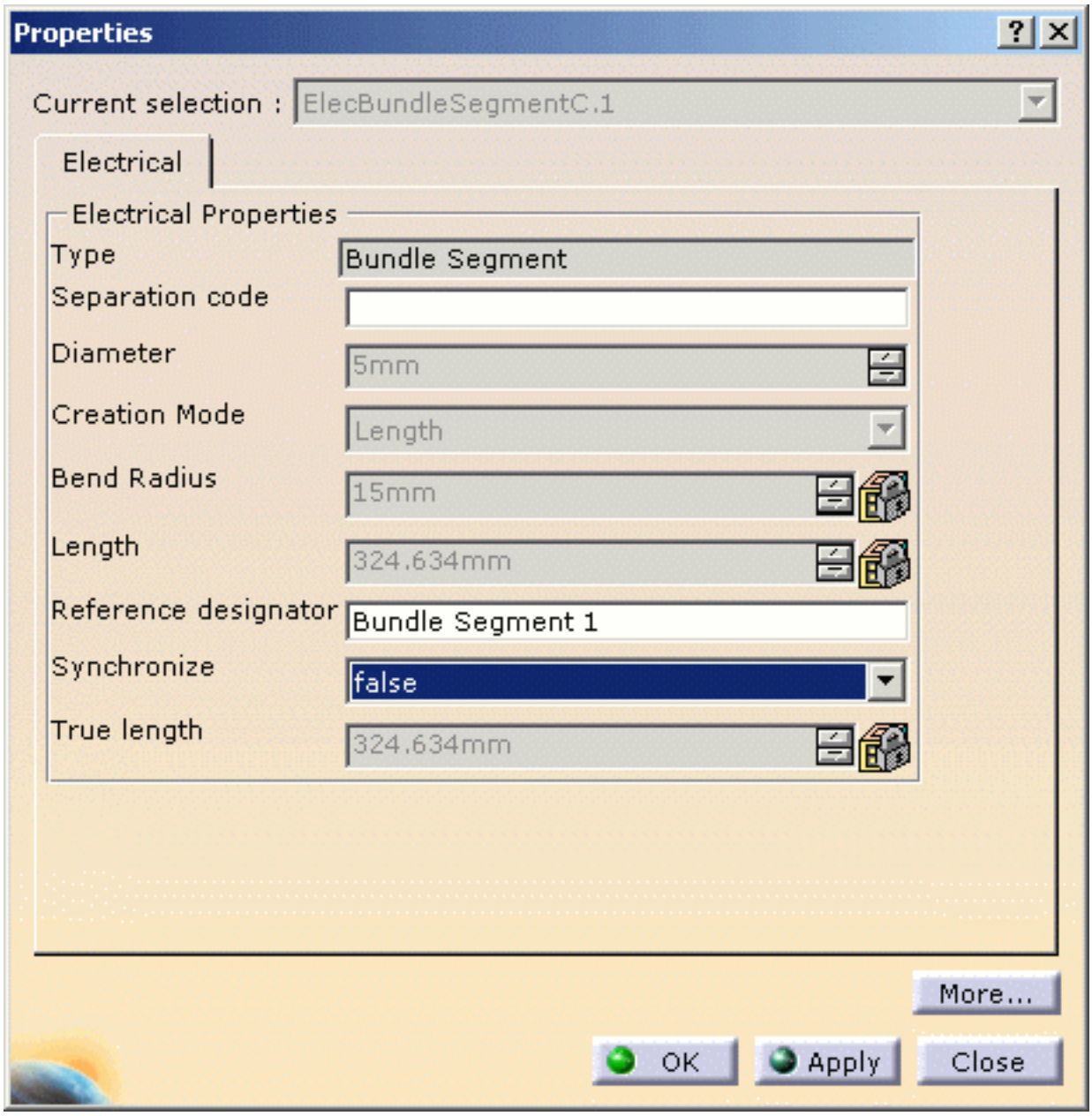
- Analyze step is processing
- Replaced components synchronization is processing
- Removed components synchronization is processing
- Bundle segment structure synchronization is processing
- Added components synchronization is processing
- Attributes synchronization is processing
- Generating Html report

An HTML file is generated in the synchronization report repository if **Html report** is checked.

For more information about the html report, see Generating the HTML Report during the Synchronization.

The extracted document is updated if **Simulation** is not checked:

- The attributes that have been modified will show the **Synchronize** property set to **True**.
- Once the attributes have been reset, the **Synchronize** property will turn back to **False**.



Thus you can visualize what electrical attributes have been modified in the initial model.

Generating the HTML Report during the Synchronization

This file explains the contents of the Html Report created during the synchronization.

The HTML report is composed of accurate fields, which describe the changes that occurred during the synchronization.

Only the tables which have modified components are shown, except for the **Synchronization Parameters** table, which is always present.
They rely on the fields that can be found in the **Synchronize** dialog box.

- **Synchronization Parameters**:
  - Option:
    - Simulation
    - Automatic flattening on added components
    - Reset the synchronization attributes.
  - Action:
    - Remove the components
    - Add the components
    - Replace the reference of components
    - Synchronize the bundle segment structure
    - Synchronize the attributes.
  - Components:
    - Bundle segments
    - Devices
    - Supports
    - Wires.
- **List of Removed Components**:
  - Bundle Name
  - Bundle Segments
  - Devices
  - Supports
  - Wires.
- **List of Added Components**:
  - Bundle Name

- Bundle Segments
- Devices
- Supports
- Wires.

- **List of Replaced Components**:
  - Bundle Name
  - Devices
  - Supports.

- **List of Bundle Segment Structure Modifications**:
  - Bundle Name
  - Modification Number
  - Before
  - After.

- **List of Bundle Segment Attribute Modifications**:
  - Bundle Name
  - Name
  - Reference Designator
  - Length
  - Diameter
  - Bend Radius
  - Separation Code
  - Color.

- **List of Device Attribute Modifications**:
  - Bundle Name
  - Instance Name
  - Reference Designator
  - Sub Type
  - Part Number.

- **List of Support Attribute Modifications**:
  - Bundle Name
  - Instance Name
  - Part Number
  - Support Position.

- **List of Wire Attribute Modifications**:
  - Bundle Name
  - Instance Name
  - Reference Designator

- Sub Type
- Diameter
- Linear Mass
- Bend Radius
- Separation Code
- Color
- Modified Route.

- **List of Elements Showing an Error**:
  - Bundle Name
  - Instance Name
  - Error Message.

In the **Synchronization Parameters** table, the options that have been selected in the **Synchronize** dialog box are set to YES, the others are set to NO:

Synchronization Parameters

Options		Actions		Components	
Simulation	NO	Remove components	YES	Bundle segments	YES
Automatic flattening on added components	YES	Add components	YES	Devices	YES
Reset the synchronization attributes	NO	Replace component reference	YES	Supports	NO
		Synchronize bundle segment structure	NO	Wires	NO
		Synchronize attributes	YES		

In all the fields of the attribute and structure modification list, two lines appear:

- The first line indicates the status of the component before synchronization.
- The second line indicates the parameters that have been changed with the synchronization.

List of Wire Attribute Modifications

Bundle Name	Instance Name	Reference Designator	Sub Type	Diameter	Linear Mass	Bend Radius	Separation Code	Color	Modified Route

Electrical Bundle2.1	Wire-plus1	Wire-plus1		2mm	0kg_m	20mm	Attribute unset	yellow	
				5mm					NO

The added, replaced, removed component fields have one line per element indicating the modifications.

List of Added Components

Bundle Name	Bundle Segments	Devices	Supports	Diameter	Wires
Electrical Bundle2.1					Wire-minus1
					Wire-minus2
					Wire-minus3

Find hereafter the list of the errors that can occur during the synchronization and that will be reported in the Html file:

- "The 3D harness flattening and 3D harness design link is broken"
- "Error during the Analyze"
- "Error during the Remove Component Synchronization"
- "Error during the Bundle Segment Structure Synchronization"
- "Error during the Add Component Synchronization"
- "Error during the Attribute Access"
- "Error during the Replace Component Synchronization"
- "Error during the Bundle Segment Attribute Synchronization - Different Bundle Segment Structure"
- "Error during Attribute Synchronization - Internal link failed to 3D harness flattening from 3D harness design"
- "Error during the Bundle Segment Attribute Synchronization - Different Number of Supports"
- "Error during the Bundle Segment Attribute Synchronization - Different Supports Configuration"
- "Error during the Add Component - The original component document is not saved"
- "Error during Wire Route Synchronization"
- "Error during the Device Move"
- "Error during Bundle Segment Move"

List of Elements Showing an Error

Bundle Name	Instance Name	Error Message

Geometrical Bundle2.1	Connector_F.1	Error during Add Component Synchronization
	Connector_F.2	Error during Add Component Synchronization

Note: The Html report is generated with a default Cascading Style Sheet document (.css) that can be personalized.

Using the Drawing Capabilities

You can use these functionalities only if a Drafting license is available.

These functionalities allow you to take advantage of the Interactive Drafting capabilities:

- You can easily create a drawing of the flattened harness.
- You can use the various drawing representation options.
- The Wire Annotation command in the Interactive Drafting workbench allows you to create a wire annotation on bundle segments and connectors.
- You can now gain in efficiency by creating dimensions for technological features such as electrical harness. For a drawing created with a flattened document, you can have dimensions only on bundle segments.
- The Interactive Drafting workbench lets you define and store text templates into catalogs to be used when creating texts associated to technological objects:
  - Create Text Templates.
  - Store Text Templates into a Catalog.
  - Annotate Drawings using Text Templates.

Customizing a Drawing View

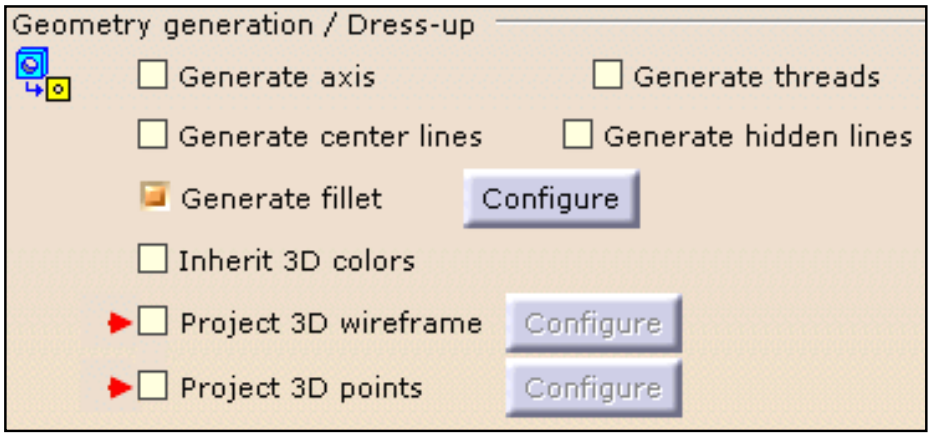
This task shows you how to generate a drawing view, using various representation options.

The document contains the harness obtained in Flattening the Harness.

Make sure the **Drafting -> View** options are set as follows:
Open the **Tools -> Options...** menu, then **Mechanical Design**:

In the Geometry generation / Dress-up frame, check the following options:

- **Project 3D wireframe**
- **Project 3D points**



For more information, refer to the View tab documentation.

1. Open the **Tools -> Options...** menu, then in the **Equipment & Systems** category, choose the **Electrical 3D Design** item to define a graphic replacement catalog.

   To do so:

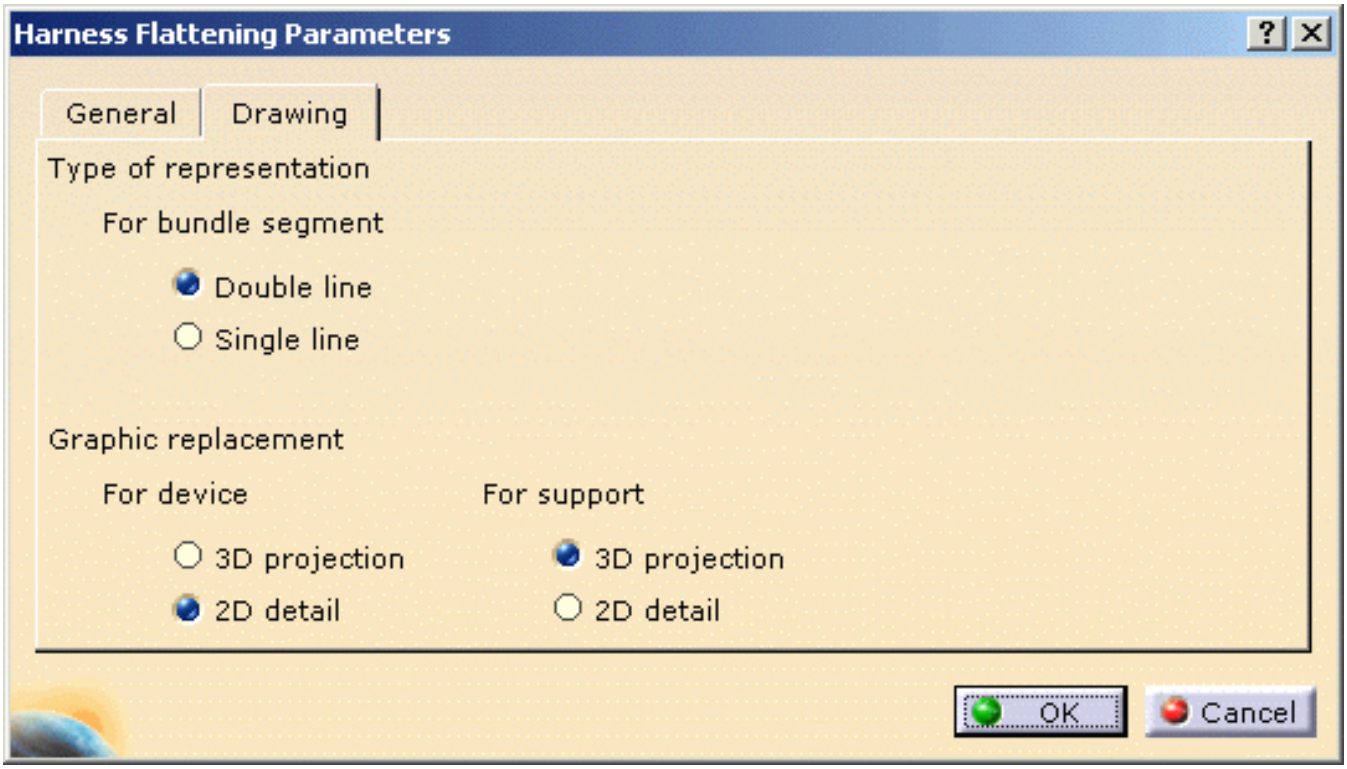
   - click the **Add** button
   - browse to the catalog.
   - click OK to validate.

   A mapping will be done between the connector external reference (or the part number if the external reference is not valuated) and the name of the 2D detail.

   - If an equivalence is found, the corresponding 2D detail will replace the connector in the drawing.
   - If not, the 3D projection will be displayed.

2. In the **Drawing** tab, select:

   - the **double line** for **bundle segment** representation
   - the **2D detail** option for **device** graphical representation



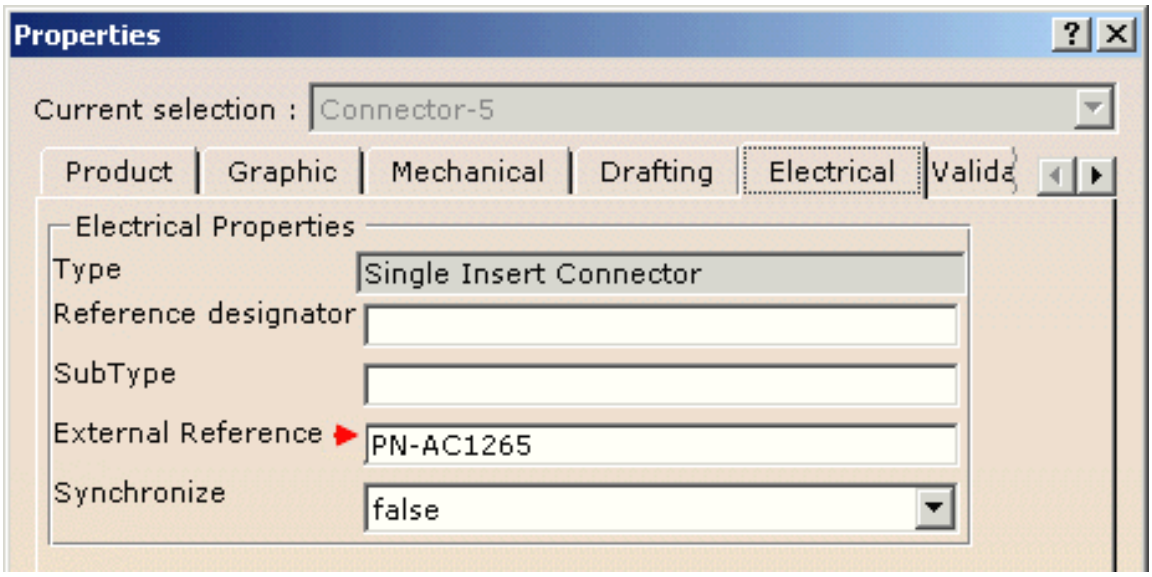
3. Click **OK** to validate.

4. Perform the mapping:

   a. Right-click a connector to display the properties,

   b. Click the **More...** button to display the **Electrical** tab.

   c. Enter a value in the **External Reference** field.



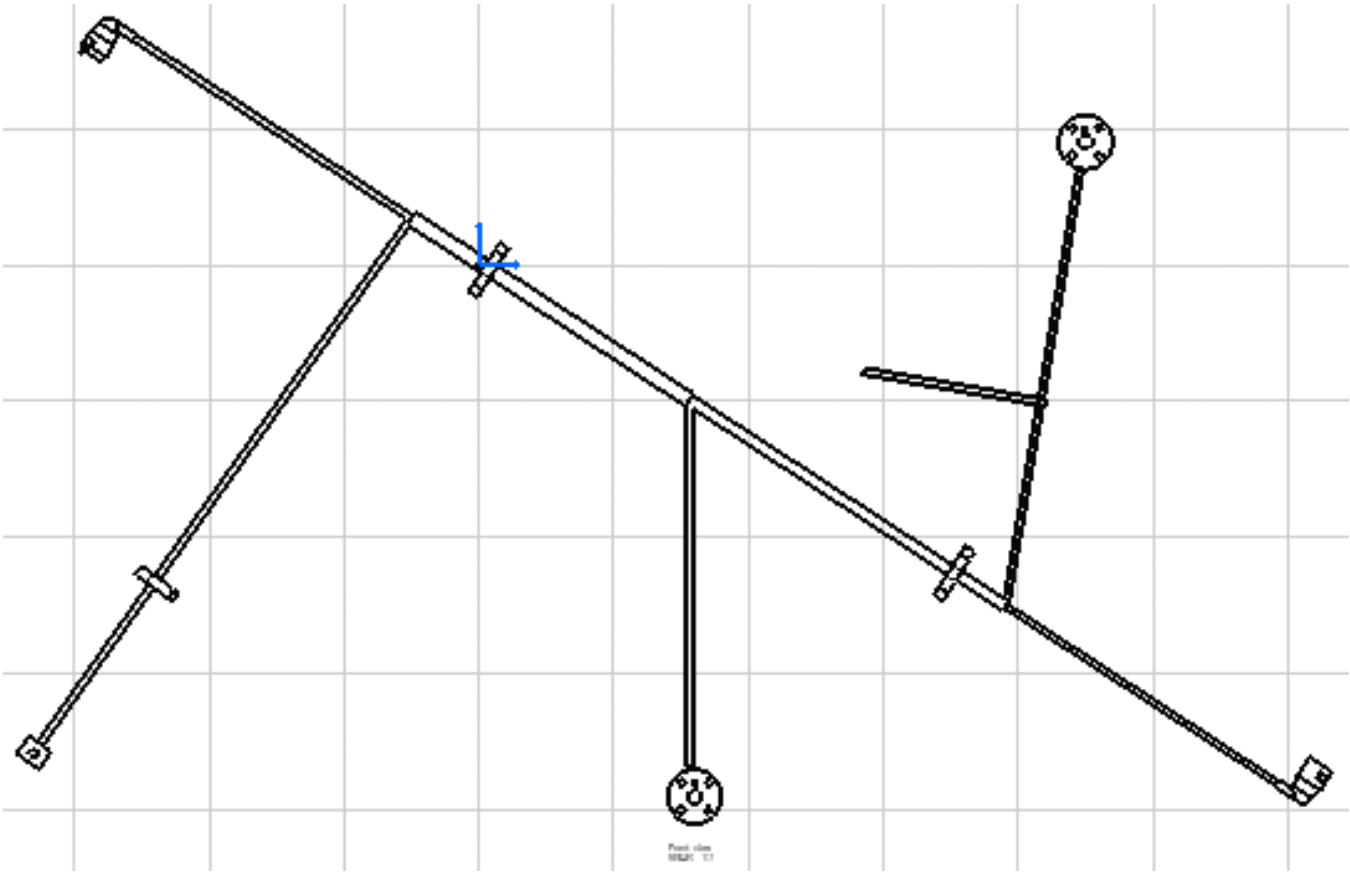
   d. Click **OK** to validate.

   Once this is done, you will generate the drawing.

5. Open a new drawing document.

6. Click the **Front View** button and select a face of the connector as **Plane**.

The drawing document updates according to this choice:



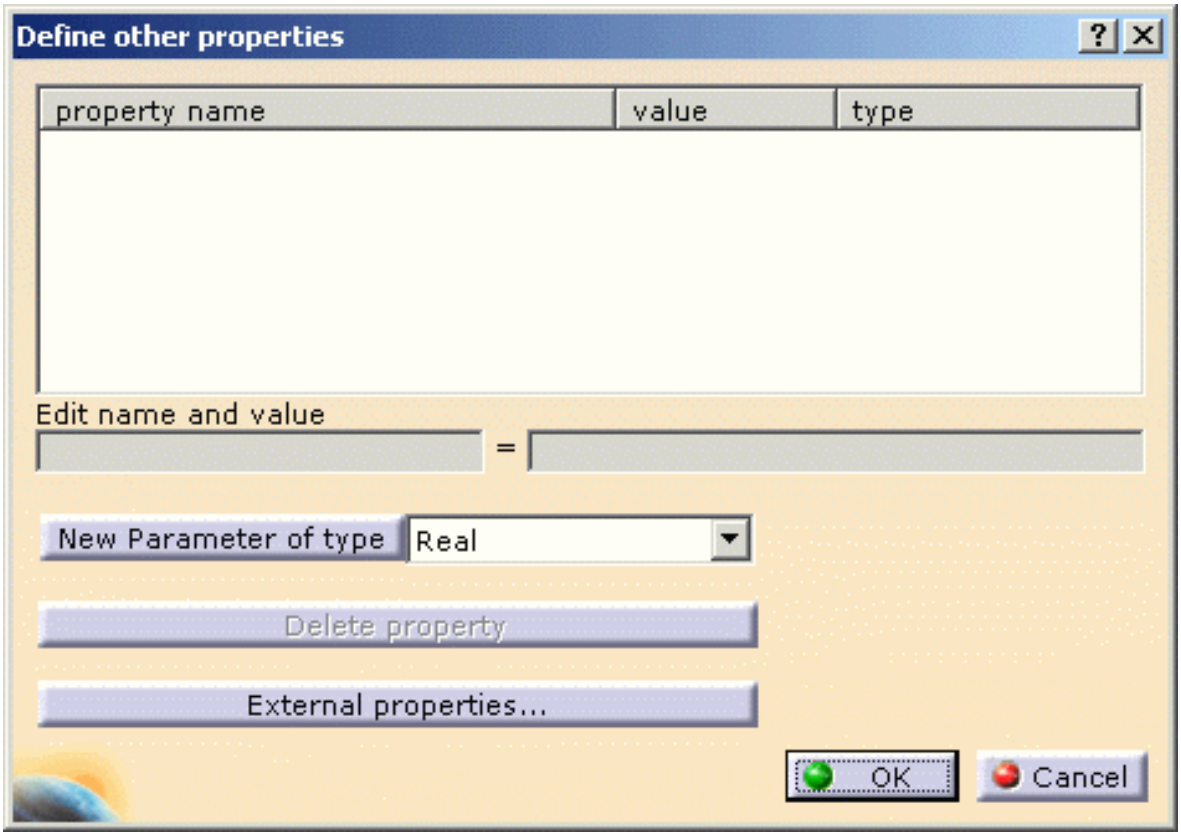
The connectors have been replaced with the chosen 2D detail and bundle segments are displayed with a double line.

Note that the **External Reference** attribute does not exist for the support and you need to create it.

To do so:

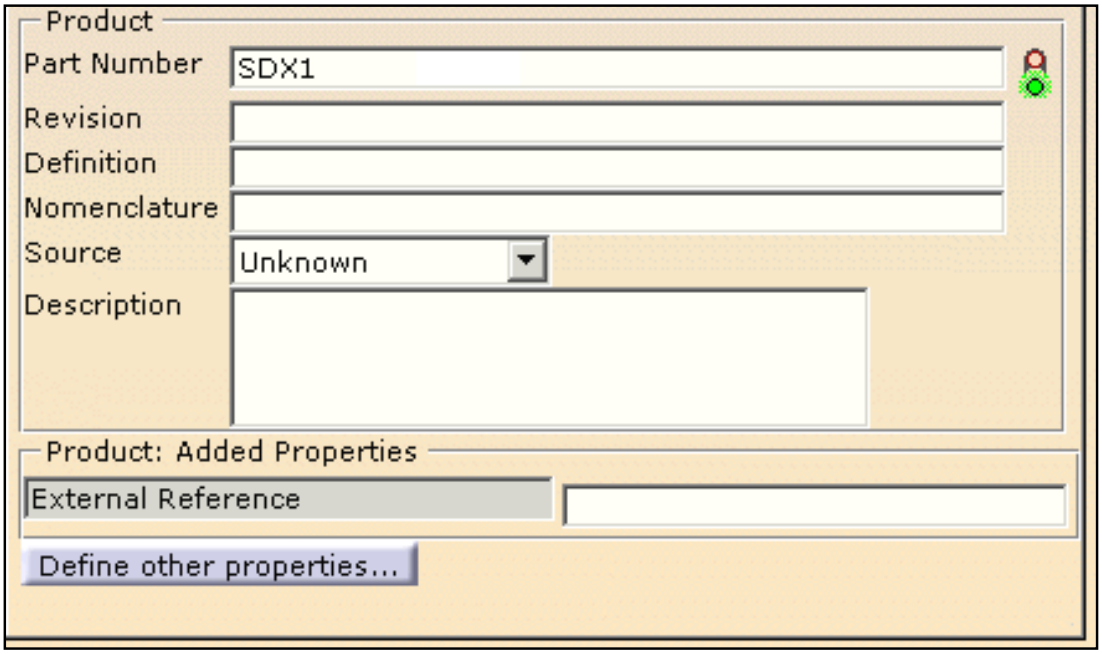
- Right-click the support to display the **Properties**.
- At the bottom of the **Product** tab, click the **Define other properties...** button.

  The dialog box displays:



- Select String in the combo and click the **New Parameter of type** button.
- Edit the name: External Reference

  This property is added to the support.



- You can now enter a value to map a catalog 2D detail.

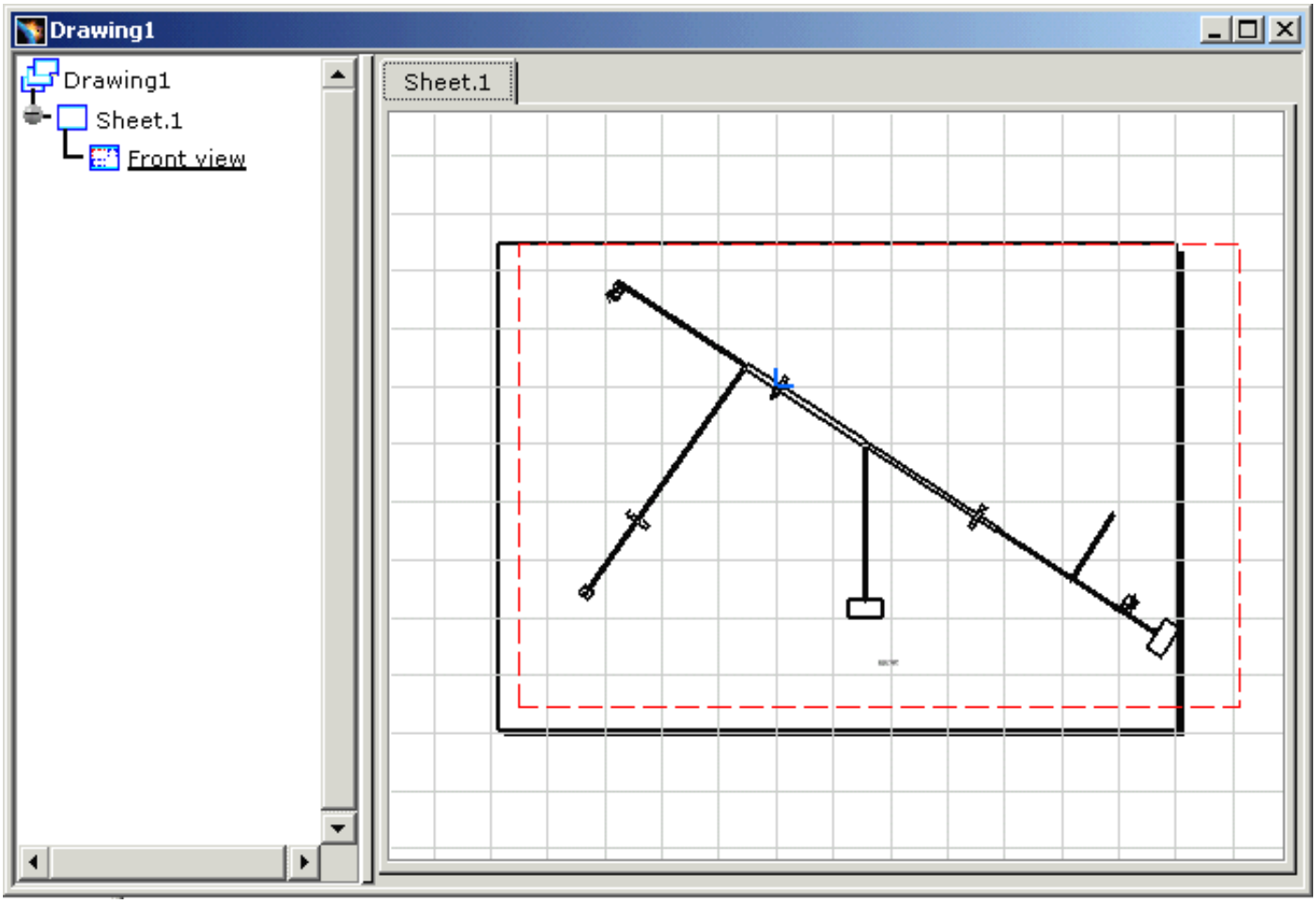
Generating a Drawing

You can perform this scenario only if a Drafting license is available.

This task explains how to generate a drawing from the flattened harness.

The document contains the flattened harness obtained in the previous task.

1. Open a new product document using the **File -> New...** command.
   Choose the **Drawing** type and click **OK** to validate the default New Drawing parameters.
   CATIA switches to the Drafting workbench.
2. Click the **Front View** button .
   You are prompted to select a reference plane on the 3D geometry.
3. Click a plane in the flattened harness document: for example the face of a connector.
   The drawing document updates according to this choice:

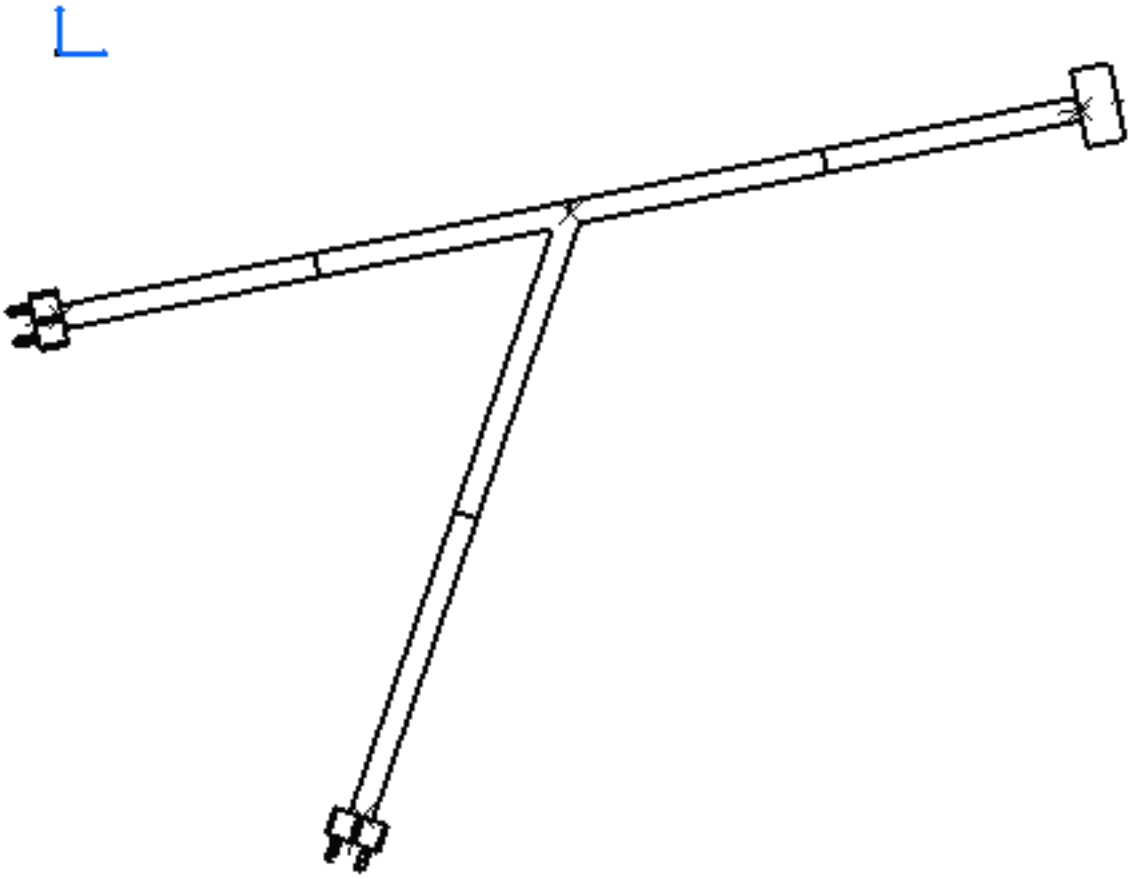


Creating Wire Annotations

A dedicated command is available in the drafting workbench to create a wire annotation on bundle segments and connectors. The wire attributes in the annotation can be customized.

The document contains the flattened harness obtained in the previous task.

1. Generate a drawing like it is explained in Generating a Drawing.



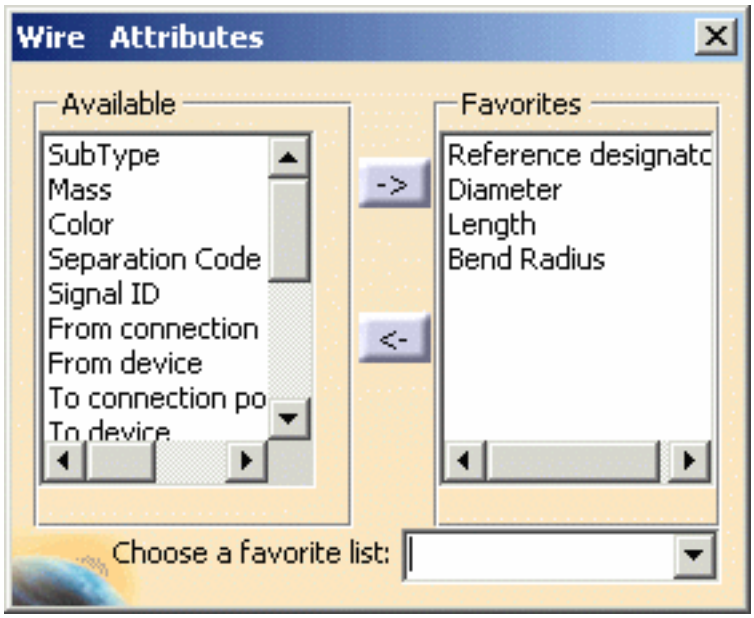
2. In the Drafting workbench, click the **Wire Annotation** button .

   Two dialog boxes open:

- The **Tools Palette,** which allows you to show or hide the wire annotation dialog box:
  - When it looks like this, the **Wire Attributes** dialog box is visible.

  - When it looks like that, the **Wire Attributes** dialog box is not visible: Click the **Tools Palette** to make it visible.

- The **Wire Attributes** dialog box, which allows you to select the attributes you want to see as annotations:



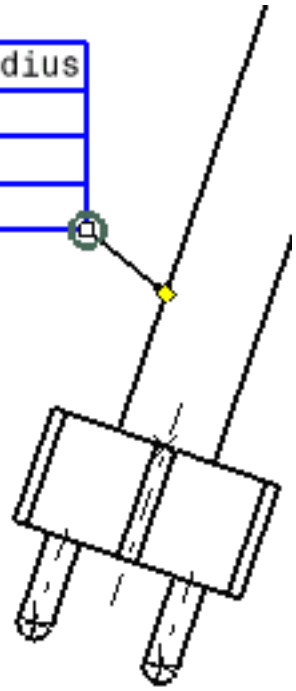
  - To add an attribute from **Available** to **Favorites**, select the attribute of interest then click the right arrow  .
  - To remove an attribute from **Favorites**, select it in the list then click the left arrow  .
  - By default, the last **Favorites** used are available when the dialog box is re-opened.
  - If you want to save different favorite lists: select the attributes needed, enter a name in the input field then validate. You can then display your favorite list in the combo box.

3. Choose **Reference designator**, **Diameter**, **Length** and **Bend Radius** for example.

4. Then, select a bundle segment or a connector in the geometry.

A table is created, showing all the attributes that you have selected in **Favorites**:

Reference designator	Diameter	Length	Bend Radius
Signal-checkWire2.1	2mm	742.13mm	10mm
Signal-minusWire3.1	4mm	347.108mm	10mm
signal-plusWire3.1	2mm	443.554mm	9mm



If you want to add another wire annotation, you have to click again the **Wire Annotation** button before selecting a bundle segment or a connector.

There is an automatic update of the modifications except if you remove or add some components.

If you want to customize the graphical representation of the table, right-click to display the contextual menu then select **Properties**.

Creating Intra-Technological Feature Dimensions

This task will show you how to create dimensions for technological features such as electrical harness.

You need an Electrical Harness Assembly license for the purpose of this scenario as we will be dimensioning Electrical Harness Assembly features. Intra-technological feature dimensioning is also available for other applications such as Structure Functional Design or Ship Structure Detail Design. For more information on the availability of technological feature dimensioning for a given workbench, refer to the related documentation.

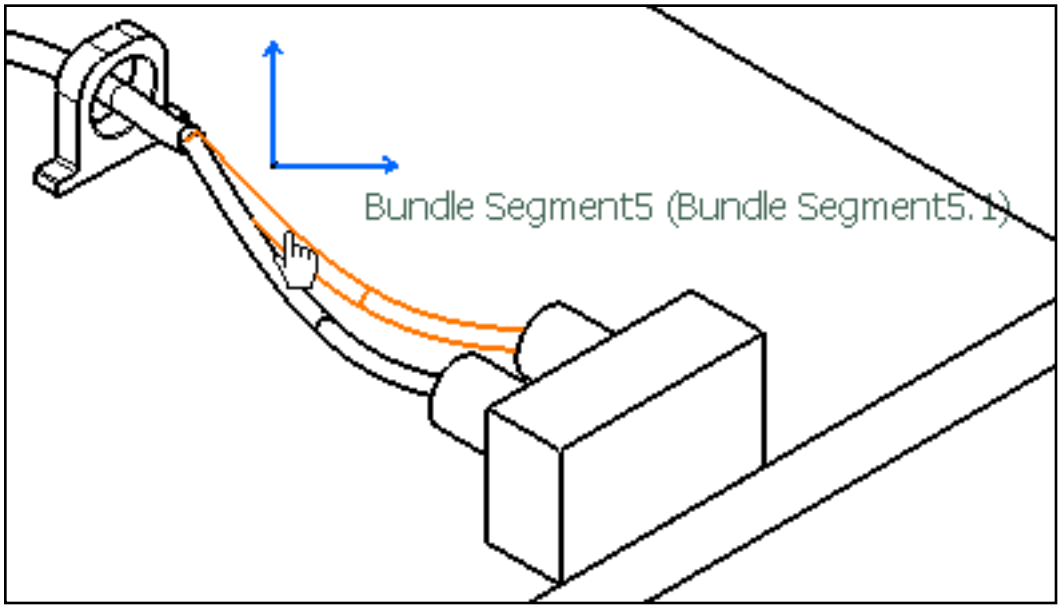
Refer to Before you Begin for general information about technological feature dimensions.

Open the ElectricalAssembly.CATProduct document and make sure it is loaded in the Electrical Harness Assembly workbench (if necessary, select **Start -> Equipment & Systems -> Electrical Harness Assembly** to launch the workbench). Open the ElectricalAssembly.CATDrawing document.

1. Click the **Multiple Intra Technological Feature Dimensions** icon  from the Dimensioning toolbar, Technological Feature Dimensions sub-toolbar.

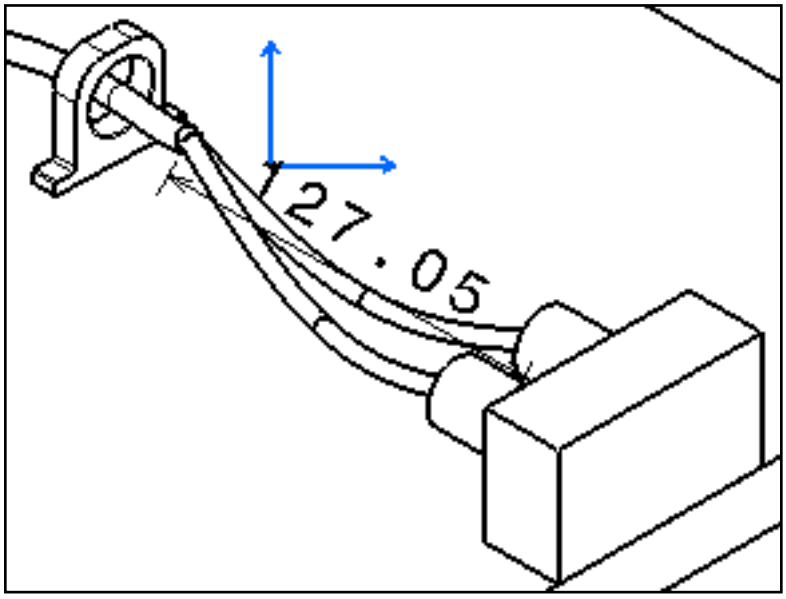
You can also click the **Technological Feature Dimensions** icon  and then select the **Multiple Intra Technological Feature Dimensions** icon  from the Tools Palette.

2. Select the feature that you want to dimension. Note that the name of a feature is displayed as a help as you move the cursor over it.



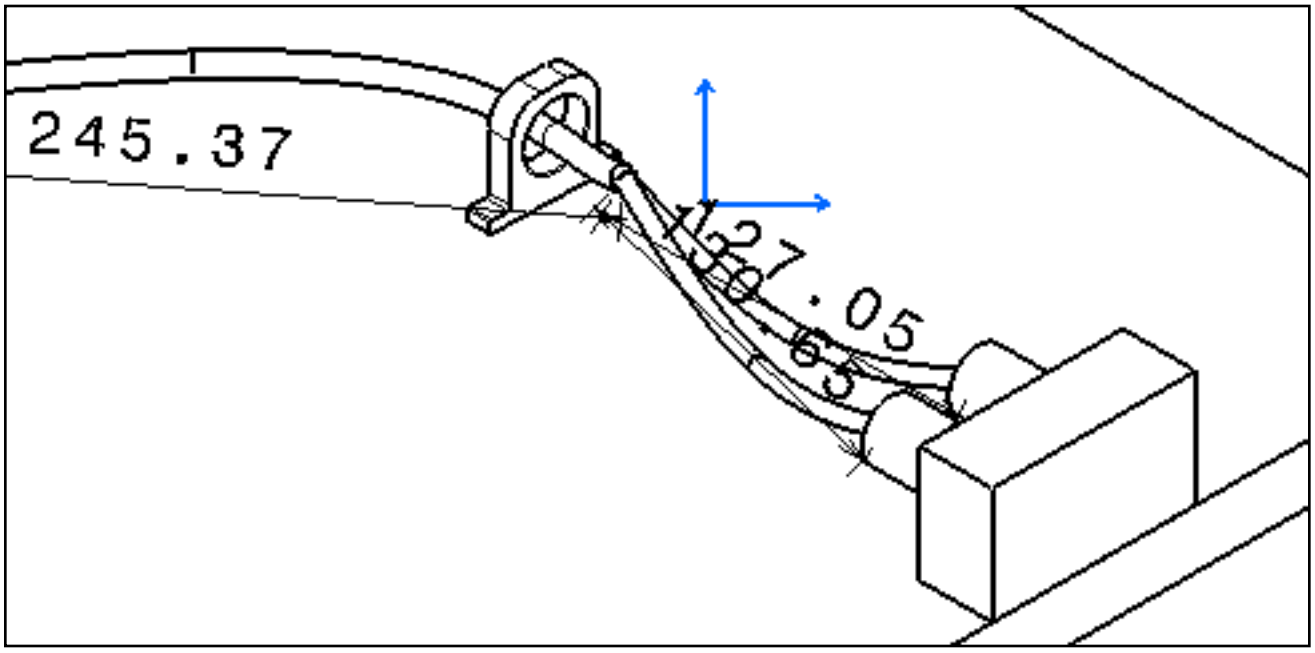
The dimension is created as specified by the feature. In this specific example, the bundle segment specifies that the dimension should provide its overall length.

The dimension creation command remains active.



3. Repeat step 2 for each additional feature that you want to dimension.
4. End the dimension creation by clicking anywhere in the drawing (but not on a technological feature) or by lining-up the dimension. The intra-feature dimensions are created as specified by the feature.

You can now handle the dimension(s) just like any other dimension.



Creating Text Templates

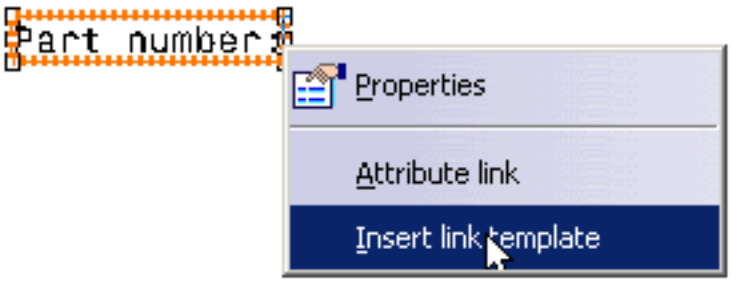
This task will show you how to create text templates.

Before you begin, you need to make sure that the package corresponding to the type of object for which you want to create a template is correctly loaded. For the purpose of this scenario, you will load the Product package. Go to **Tools -> Options -> General -> Parameters and Measure** and click on the **Language** tab. Check **Load extended language libraries** and uncheck **All packages**. From the **Available Packages** list, select **ProductPackage** and click on the right arrow to add it to the **Packages to load** list. Click **OK**, and then exit and re-start the software.

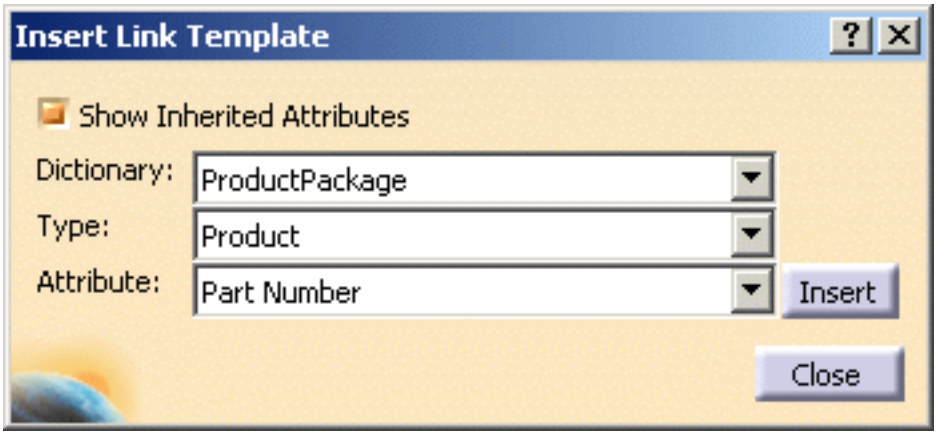
Create a new drawing.

1. Click the **Text** icon  from the Annotations toolbar.

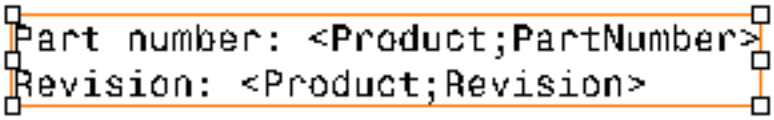
2. Click anywhere in the drawing. A green frame appears, as well as the Text Editor dialog box.
3. In the Text Editor dialog box, type *Part number:*.
4. Without closing the Text Editor dialog box, right-click the frame and select **Insert link template** from the contextual menu which is displayed.



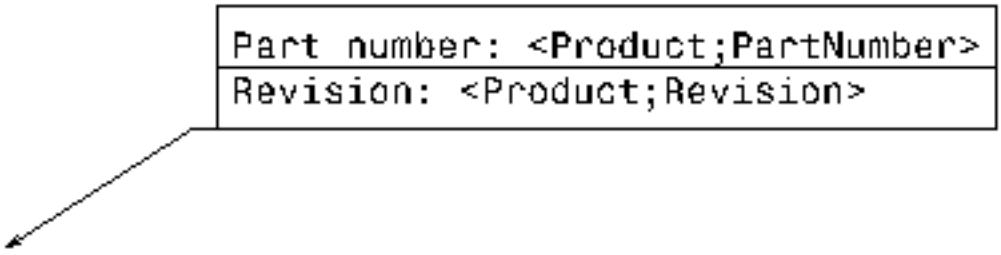
5. In the Insert Link Template dialog box which is displayed, select the **ProductPackage** dictionary, the **Product** type and the **PartNumber** attribute, and click **Insert**.



6. Back in the Text Editor dialog box, press the Enter key and type *Revision:*.
7. Back in the Insert Link Template dialog box, select the **Revision** attribute (leave the other fields as is), click **Insert** and then **Close**.
8. Click OK in the Text Editor dialog box. The text template is now created.



9. Make sure the text template is selected and click the **Frame** icon in the Text Properties toolbar.
10. From the Frames sub-menu, choose the **Scored Rectangle** frame.
11. Right-click the text template, and select **Add Leader** from the contextual menu.
12. Click in the drawing to end the leader creation. The text template is now set.



13. Right-click the text template, and select **Properties** from the contextual menu.
14. Click the **Feature Properties** tab in the Properties dialog box which is displayed.
15. In the **Feature Name** field, type *Part number & Revision* and click **OK**. You will use this feature name to identify this text template in the future.
16. Create another text by repeating steps 1 to 3, this time typing *Part name:* in the Text Editor dialog box.
17. Repeat steps 4 and 5, this time selecting the **Name** attribute in the Insert Link Template dialog box.
18. Click **Close** in the Insert Link Template dialog box and then **OK** in the Text Editor dialog box. The text template is now created.
19. Make sure the text template is selected and in the Graphic Properties toolbar, choose green from the **Color** list. The text template is now set.

Part name: <Product;Name>

20. Repeat steps 13 to 15, this time typing *Part name* in the **Feature Name** field. You will use this feature name to identify this text template in the future.
21. Select **File -> Save As** and save the drawing as a .CATDrawing document.

Now that your text templates are defined, you need to store them in a catalog.

Storing Text Templates in a Catalog

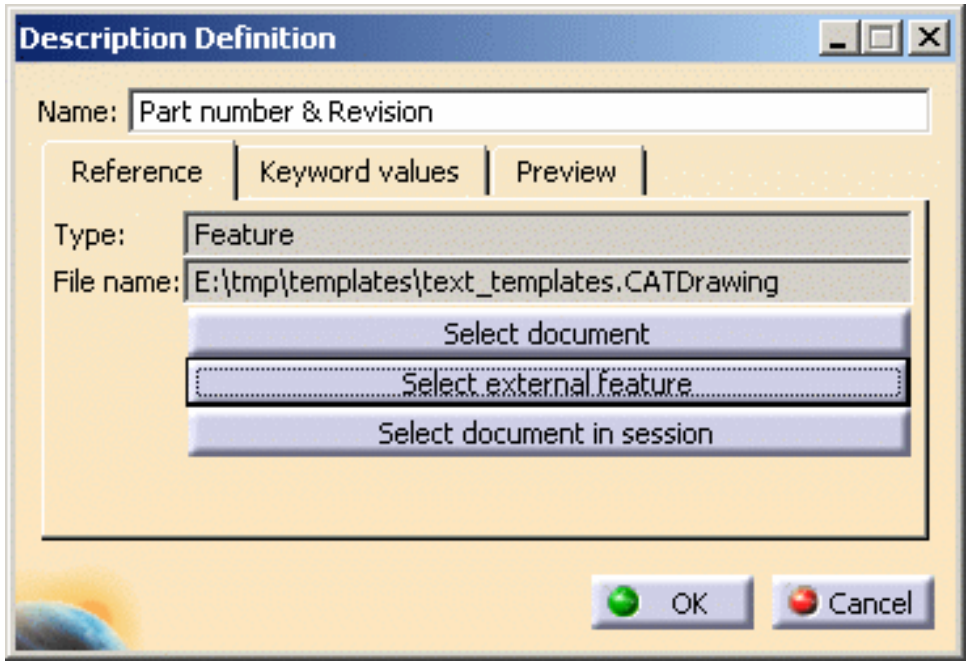
This task will show you how to store text templates in a catalog.

For more information on catalogs, refer to the Using Catalogs chapter in the *Infrastructure User's Guide*.

Open the TextTemplates.CATDrawing document.

1. Select **File -> New**.
2. In the New dialog box, select **CatalogDocument** from the list of types and click **OK**. The Catalog Editor workbench is launched and a new catalog is created.
3. In the left-hand pane, double-click **Chapter.1** to activate it.
4. Select **Insert -> Add Family...**. The Component Family Definition dialog box is displayed.
5. Type *Text templates* in the **Name** field.
6. Make sure **Standard** is selected in the **Type** field, and click **OK**. The family is created.

7. For more convenience, select **Window -> Tile Horizontally** to display your Catalog Editor and Drafting windows at once.
8. In the Drafting window, select one of the text templates, e.g. Part number & Revision.
9. In the left-hand pane of the Catalog Editor window, double-click **Text templates** to activate it.
10. Select **Insert -> Add Component...**. The Description Definition dialog box is displayed.
11. On the Reference tab, click the **Select external feature** button. The dialog box is updated with information about the selected text template, i.e. Part number & Revision.

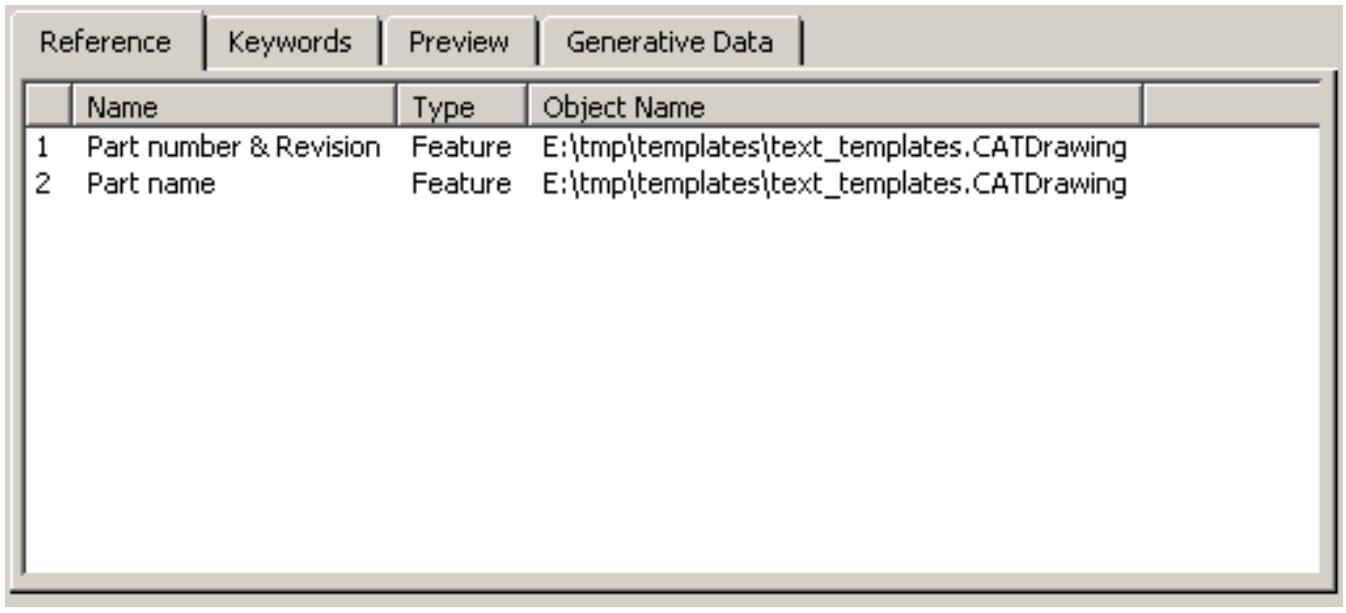


12. Click **OK**. The selected text template is listed on the Reference tab, in the right-hand pane of the Catalog Editor window.

13. Go back to the Drafting window and select the other text templates, e.g. Part name.

14. Return to the Catalog Editor window and repeat steps 10 and 11. The dialog box is now updated with information about the Part name text template.

15. Click **OK**. Both selected text templates are now listed on the Reference tab, in the right-hand pane of the Catalog Editor window.



16. Select **File -> Save As** and save the catalog as a .catalog document.

Annotating Drawings Using Text Templates

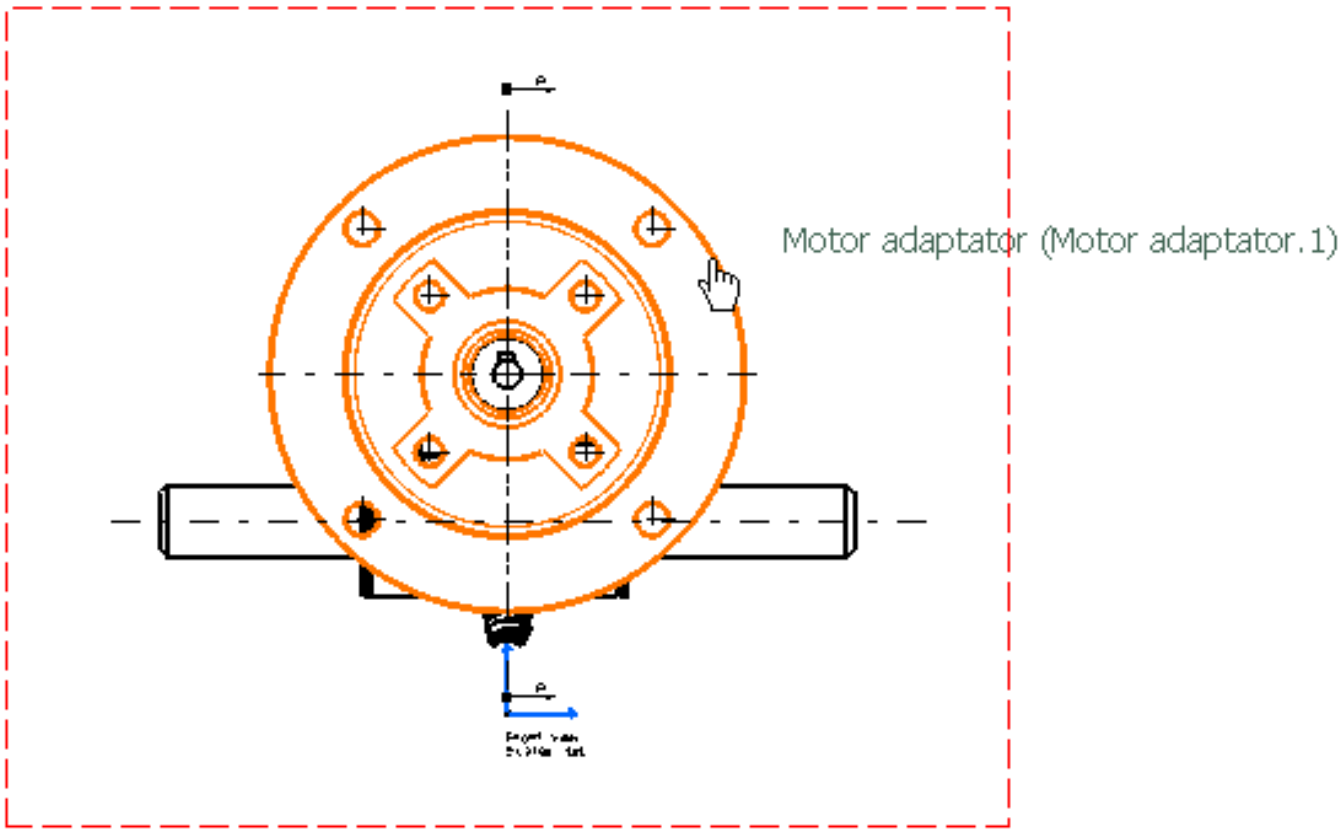
This task will show you how to annotate drawings using text templates stored in a catalog.

Before you begin, you need to make sure that the package corresponding to the type of object for which you want to create a template is correctly loaded. For the purpose of this scenario, you will load the Product package. Go to **Tools -> Options -> General -> Parameters and Measure** and click on the **Language** tab. Check **Load extended language libraries** and uncheck **All packages**. From the **Available Packages** list, select **ProductPackage** and click on the right arrow to add it to the **Packages to load** list. Click **OK**, and then exit and re-start the software.

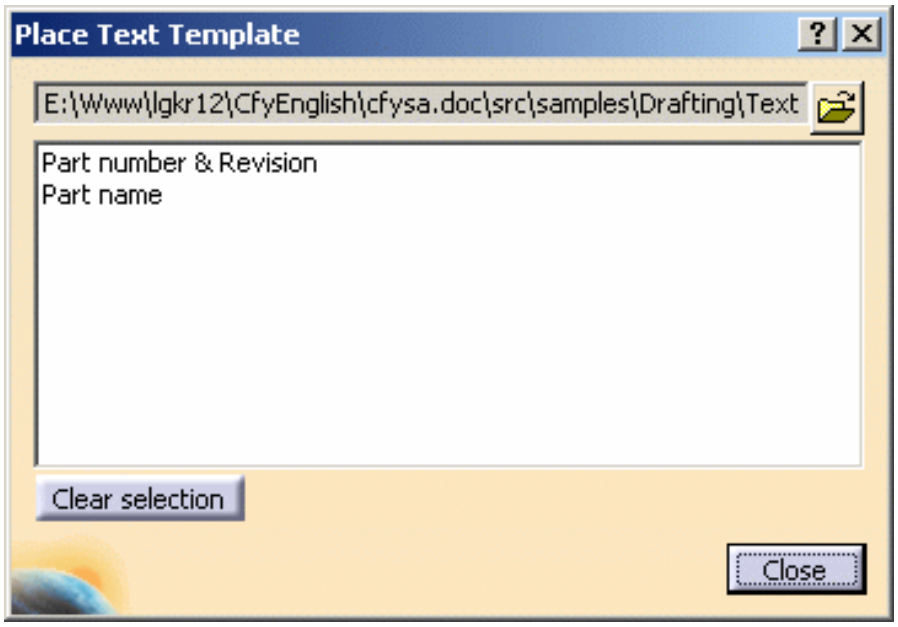
Open the GEAR-REDUCER.CATDrawing document.

1. Click the **Text Template Placement** icon  from the Annotations toolbar.

2. In the Place Text Template dialog box, browse to select the TextTemplates.catalog document. This document is located in your documentation installation folder (by default, this folder is `C:\Program Files\Dassault Systemes\XXXdoc\online\`), in `cfysa_C2\samples\Drafting`. Leave the Place Text Template dialog box open to perform the next steps.
3. On any view, select the part that you want to annotate, making sure that you click where you want the anchor point of the annotation to be located. Note that the name of a part is displayed as a help as you fly the cursor over it.

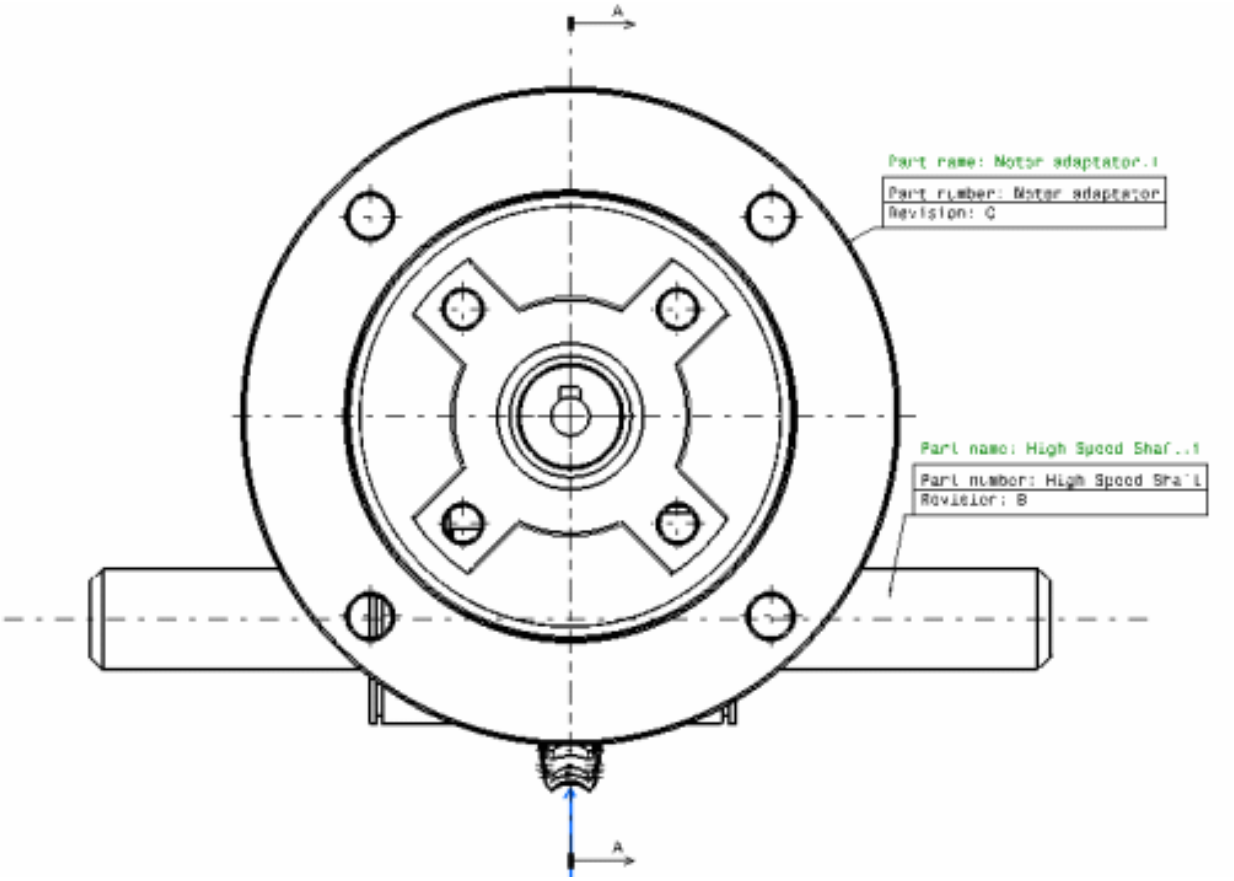


The Place Text Template dialog box now lists all the templates available in the selected catalog and which can be applied to the selected object.



4. In the Place Text Template dialog box, select the text template that you want to apply, Part number & Revision for example. The annotation is created at the point you clicked when selecting the part to annotate, and contains information retrieved from the 3D part. Note that this annotation is associative to the 3D part.
5. If you want, select the other text template (Part name). Note that this annotation will also be created at the point you clicked, so it will overlap the first annotation. For better results, you will have to move it afterwards.
6. Repeat steps 3 to 5 for other parts that you want to annotate.

Note that the last template you selected in the Place Text Template dialog box remains active when annotating other parts. You can de-activate it by clicking the **Clear selection** button.



7. When you're done, click **Close** to close the Place Text Template dialog box.

You can also multi-select the parts that you want to annotate (using the Ctrl key) prior to clicking the **Text Template Placement** icon.

Defining the Report Format

This task shows you how to define the report format. You use this function, together with the function described in Generating a Report, to get the values of properties of objects in a document.

Examples from the Piping Design workbench are used here.
Substitute the appropriate resource or directory when working in another workbench.

Before you generate a report you need to define its format. This means deciding which properties you are interested in. This report format is kept in a file which you can use to generate reports from other documents.

To use this function you must first set up an option:

a. Click **Tools -> Options -> General -> Parameters and Measure**

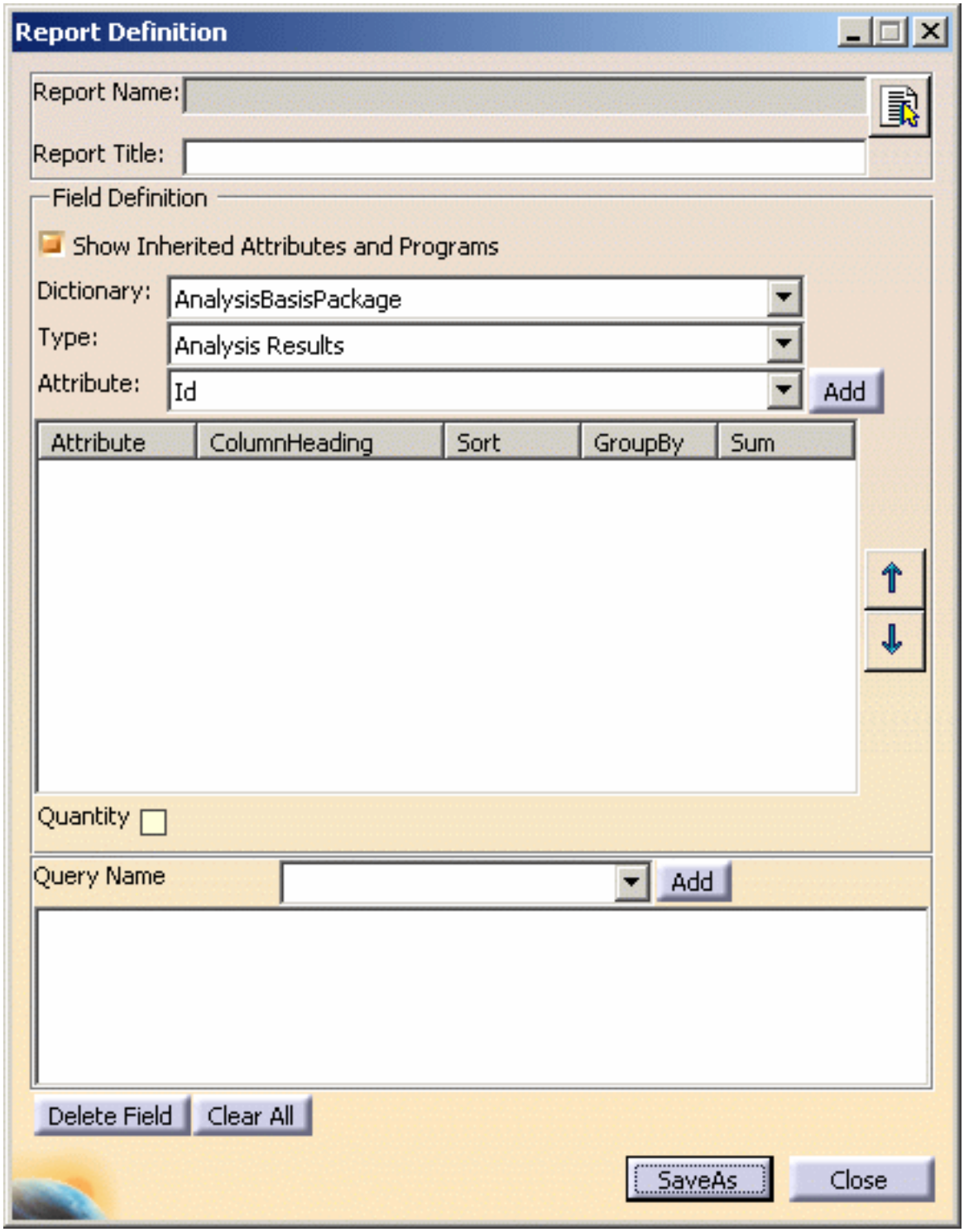
b. Select the **Language** tab.

c. Under **Language** check **Load extended language libraries**.

d. Click **OK** to validate.

1. Click **Tools -> Report -> Define**.

   The Report Definition dialog box displays.



2. Enter a **Report Name** and select a directory location.

3. Enter a **Report Title** - you can select anything but you must enter a title.

4. Check the **Show Inherited Attributes** box if you want to.

5. Click the down arrow and select the **Dictionary** related to your program.

6. Select the **Type** of object.

   The list of attributes you see in Step 7 depends on the type you select here. However, when you

generate a report you will get values for all objects in the document that have the attributes in your report format. If you want to limit the objects for which you get a report you must create a query (Step 9).

7. Select an attribute in the **Attribute** field and click the **Add** button.
   The attribute is added in the window.
   Add as many attributes as you want to.

8. You can further refine your report by using the **Edit -> Search** function to define a query.
   This allows you to generate a report on a narrower selection of check valve, say, of a certain size, instead of all check valves in your document. The queries you create are available for selection when you click the arrow in the **Query Name** field above.
   Detailed instructions on using the **Search** function can be found in the *Infrastructure User's Guide* under Basic Tasks - Selecting Objects.
   Briefly:

   a. Click **Edit -> Search** to bring up the **Search** dialog box.

   b. Select the **Advanced** tab, then select a workbench, type of object and attribute you are interested in.

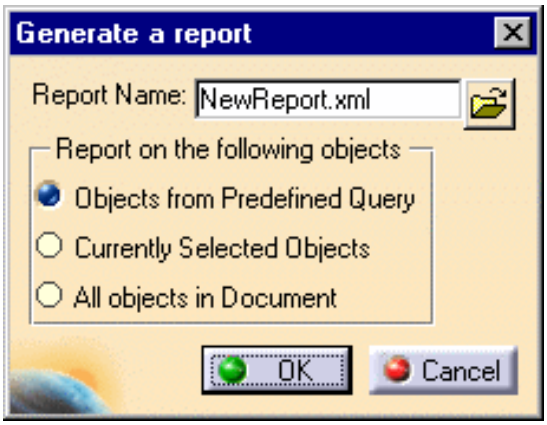
   c. Clicking the **Add to Favorites** button brings up the **Create a Favorite Query** dialog box, where you can name the query and save it.

Generating a Report

This task explains how to generate a report listing values of selected properties. Before you do this you need to define the report format.

Examples from the Piping Design workbench are used here.
Substitute the appropriate resource or directory when working in another workbench.

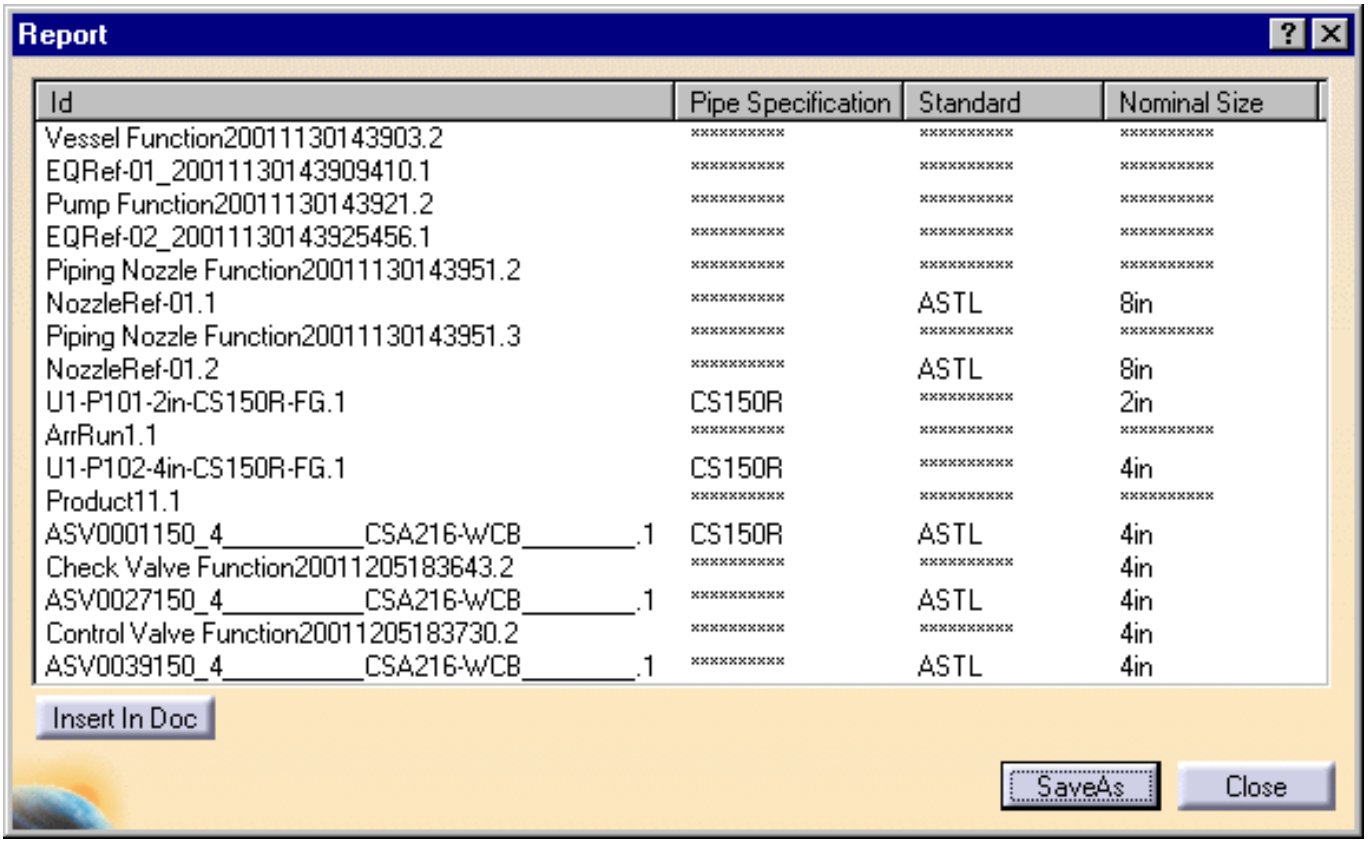
1. Click **Tools -> Report -> Generate**.



The Generate Report dialog box displays.

2. Click the **Open** button and select the format you want to use for your report, in this case NewReport.
3. If you had defined a query in your report format then check **Objects From Predefined Query**.
4. If you select one or more objects in the document then check **Currently Selected Objects**.
5. Check **All Objects in Document** if you want a report on all objects in your document.
6. Click **OK** and select a format, such as HTML, when you are prompted.

The report is generated. It shows values for all properties defined in your report format for all objects in the document that have them.
Where an object does not have a property the report displays asterisks.



7. Click **Insert in Document** if you want to display these values in your document.

   - To generate the report from a schematic and insert it in a schematic, click the **Insert in Document** button and click anywhere in your drawing.
   - To generate the report from a 3-D document and insert it in a schematic, click the **Insert in Document** button and select the sheet or view *in the specifications tree*.
     Do not select a point in the sheet. The data will be placed at the origin of the sheet or view, and can be moved to another location.

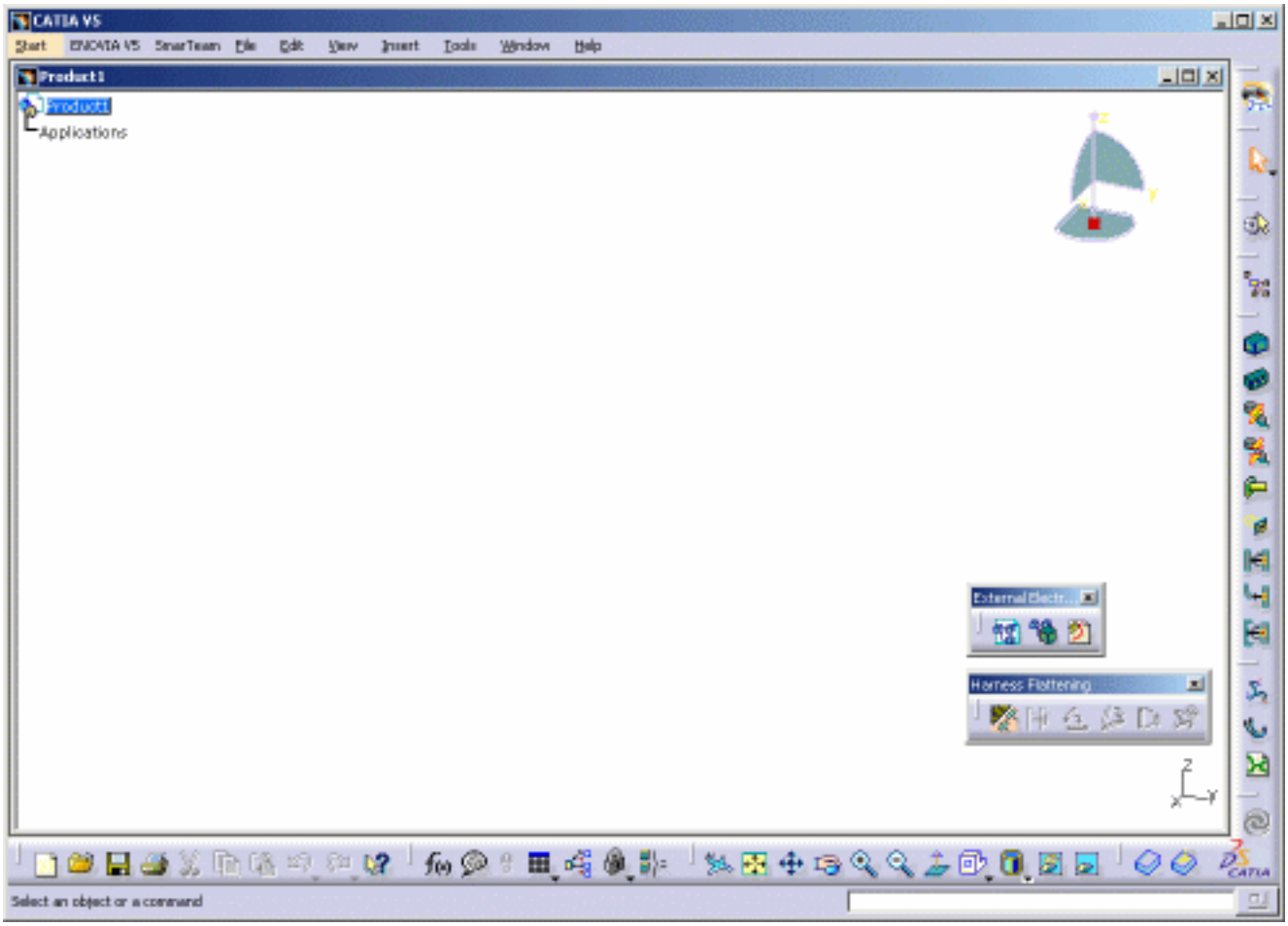
8. Click **Save As** to save the report.

   Specify a file name and location.

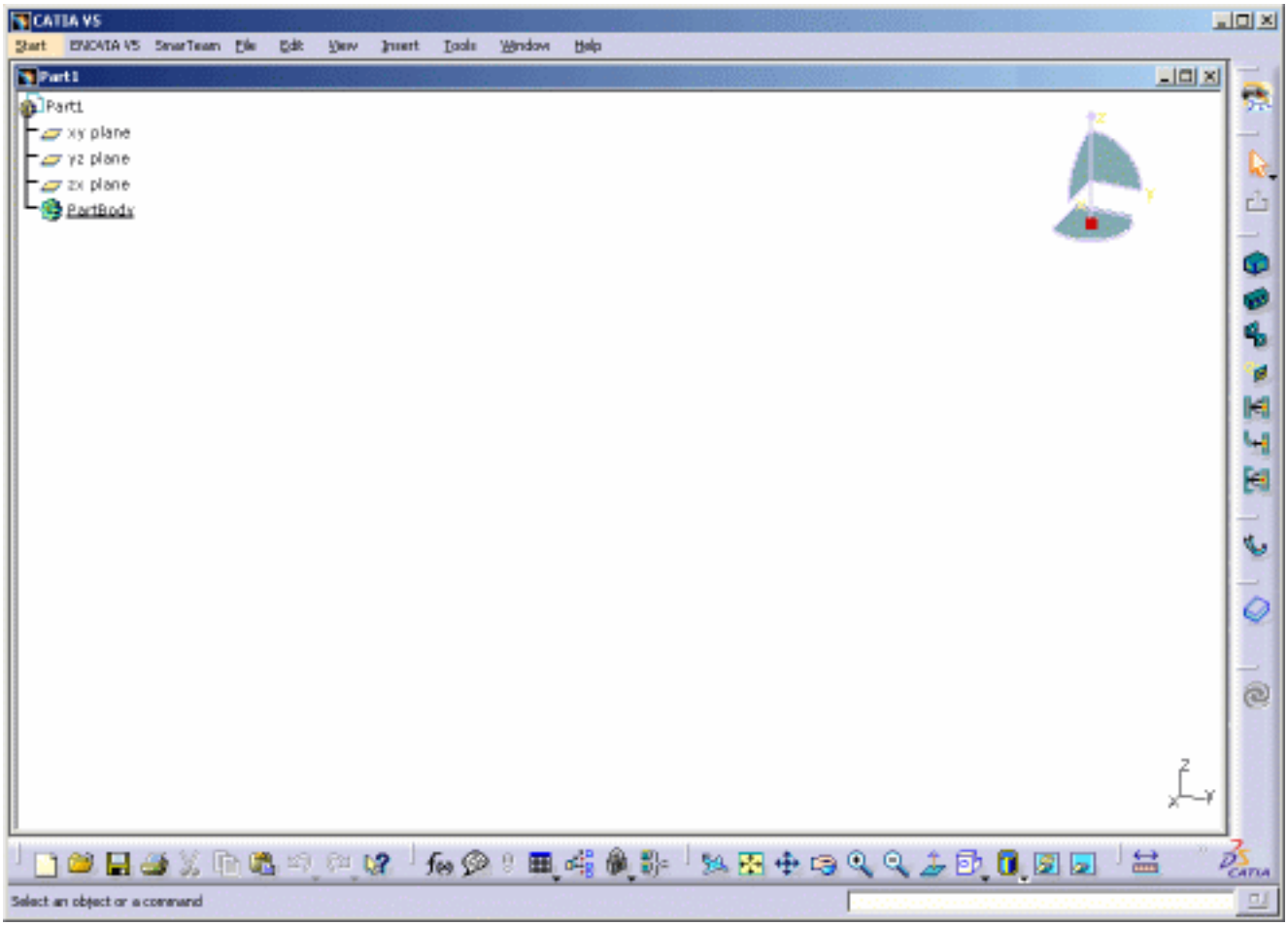
Workbench Description

Electrical 3D Design and Documentation application windows look like this:

Electrical 3D Design & Documentation - Assembly



Electrical 3D Design & Documentation - Part



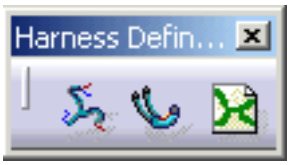
Assembly Toolbars
Part Toolbars

Assembly Toolbars

This section describes the various icons available in the Electrical 3D Design Assembly workbench.
The toolbars are located on the right in the default set-up except for the Catalog Browser, which is located in the horizontal bottom toolbar.

Electrical Device Definition

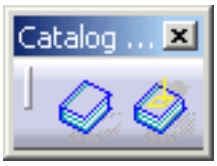
Harness Definition



External Electrical System

Harness Flattening

Catalog



Related Objects

 See Viewing Related Objects

 See Defining an Equipment

 See Defining a Connector

 See Defining a Cavity

 See Defining a Termination

 See Defining a Bundle Connection Point

 See Defining a Connector Connection Point

 See Defining a Cavity Connection Point

 See Connecting Devices

 See Disconnecting Devices

 See Creating a Geometrical Bundle

 See Defining Bundle Segments

 See Creating an Electrical Bundle

 See Selecting External Systems

 See Managing Links

 See Routing Wires from External Data

 See Defining the Harness Flattening Parameters

 See Extracting Data


See Flattening the Harness


See Rotating Bundle Segments


See Scaling Bundle Segments


See Synchronizing the Environment


See Using a Catalog, Using Catalogs


See Using Smart Placement from Catalog

Part Toolbars

This section describes the various icons available in the Electrical Library workbenches.
The toolbars are located on the right in the default set-up except for the Catalog Browser, the Measure and the Update icons which are located in the horizontal bottom toolbar.

Electrical Device Definition

Bundle Segment Definition

Catalog

 See Defining an Equipment

 See Defining a Connector

 See Defining a Cavity

 See Defining a Termination

 See Defining a Bundle Connection Point

 See Defining a Connector Connection Point

 See Defining a Cavity Connection Point


See Creating a Support


See Defining Bundle Segments


See Using Catalogs

Customizing Electrical 3D Design & Documentation

Before you start your first working session, you can customize the way you work to suit your habits.

This type of customization is stored in permanent setting files: these settings will not be lost if you end your session.

Other information can be considered as customization:

Electrical Data Exchange Format deals with electrical specifications from external data.

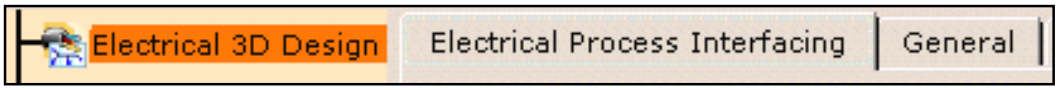
1. Select the **Tools -> Options** command.

   The Options dialog box opens.

2. Select the **Equipment & Systems** category in the left-hand box.

3. Click the **Electrical 3D Design** workbench.

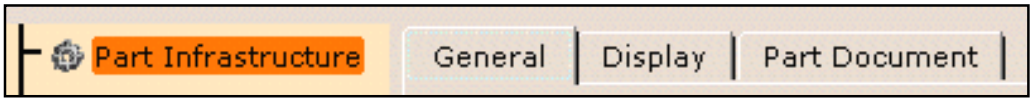
   The options, organized in tab pages appear.



4. Select the tab containing the options to be customized:

   - The Electrical Process Interfacing tab lets you define the external data file repository
   - The General tab lets you set the automatic compass
   - The Harness Flattening tab lets you define the synchronization report repository
   - The Harness Management tab lets you set the bundle segment creation options
   - The Wire Routing tab lets you customize the options to optimize the routing.

5. A tab located in the **Infrastructure** category, in the Part Infrastructure workbench, also interfere with Electrical 3D Design:



   - General

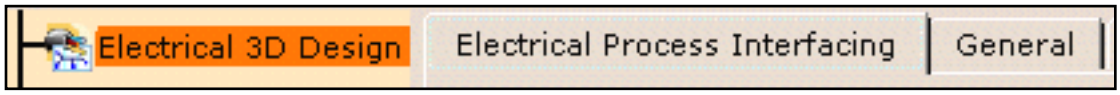
6. Another tab, located in the **Mechanical Design** category, in the Drafting workbench, also interfere with Electrical 3D Design:

   - View

7. Set options in theses tabs according to your needs.

8. Click **OK** when done.

Electrical Process Interfacing



This page deals with the options concerning:

- the electrical iXF repository path

Electrical iXF Data Repository

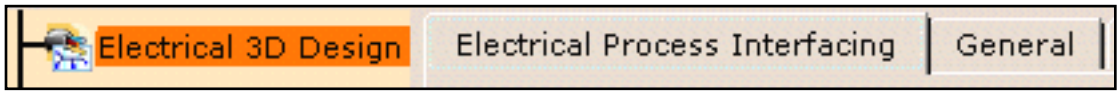
System repository

You are required to identify the path of the folder in which the XML files available are stored.

Use  to locate the iXF systems repository.

By default the field is empty.

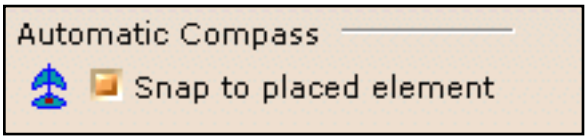
General



This page deals with the options concerning:

- the automatic compass

Automatic Compass



Snap to placed element

if this option is checked, you take advantage of the compass:

when you place an element, the compass snaps to this element allowing you to modify the orientation and location.

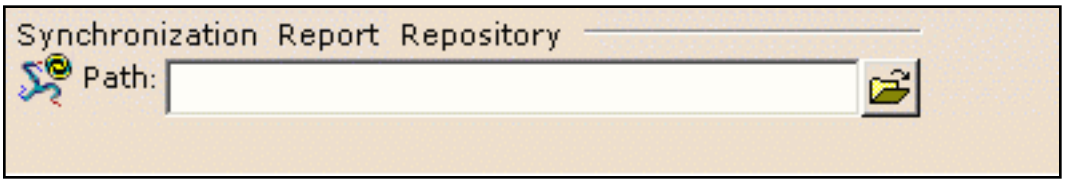
By default this option is checked.

Harness Flattening

This page deals with the options concerning:

- the synchronization report repository
- the graphic replacement catalog

Synchronization Report Repository

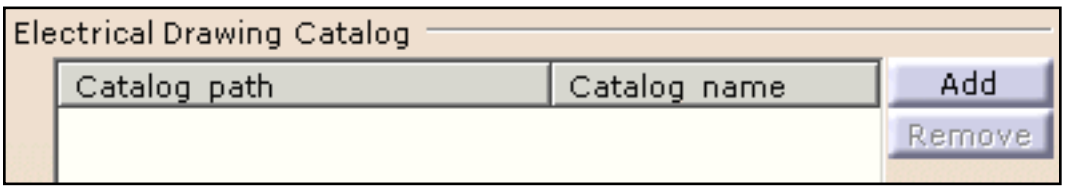


Path

Indicate the folder path where you want to save the synchronization report html files.

By default, this field contains: ...\DassaultSystemes\CATTemp.

Graphic Replacement Catalog



Catalog path and Catalog name

Indicate the catalog path and name where the drawing 2D details are stored.

By default, this field is empty.

Harness Management

This page deals with the options concerning:

- the bundle segment creation

Bundle Segment Creation

Bundle segment color

Use the color chooser to define the bundle segment color at creation.

Wire Routing

This page deals with the options concerning:

- the separation code file
- the bundle segment bend radius update

Separation Code File

This option is used to define separation code rules to optimize the routing.

No Rule

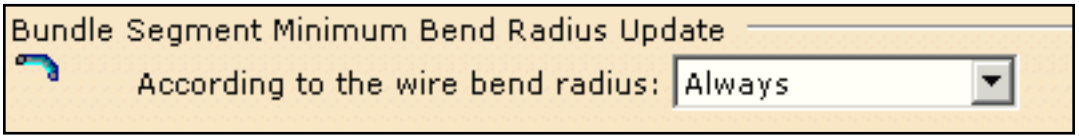
The separation code is not used. The routing is done according to the shortest route found.

It is the default value.

File Based

The separation codes may be **File Based**.
In this case, define the path to access the compatibility table by clicking the **Browse** button to choose the separation code file.
This file is used during the automatic routing.

Bundle Segment Bend Radius Update



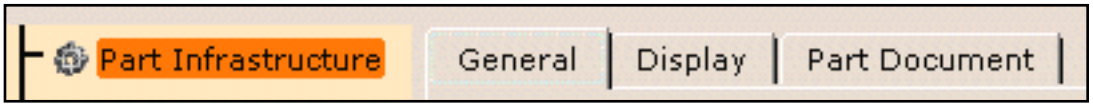
According to the wire bend radius

Three modes are available:

- **Never**: no update of the bundle segment minimum bend radius is performed.
- **Always**: the greatest bend radius of the wires routed in the bundle segment determines the segment bend radius to be applied.
- **Conditional**: the wire bend radius is taken into account for update when it is greater than the bundle segment bend radius.

The default mode is **Always**.

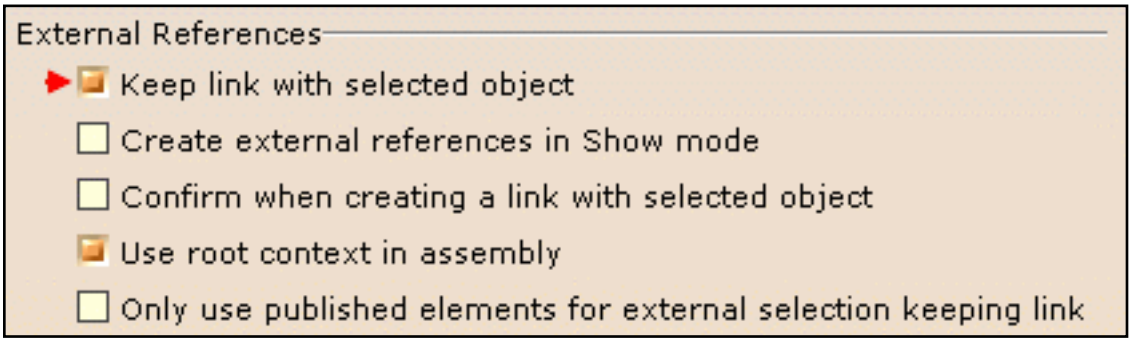
Part Infrastructure for Electrical 3D Design



This page deals with the following option:

- the external references: keep link with selected object

Part Infrastructure General option



Click here to know more about the Part Infrastructure **General** options.

Keep link with selected object

You need to select this option to take advantage of the associativity between the construction points or part body and the bundle segment.

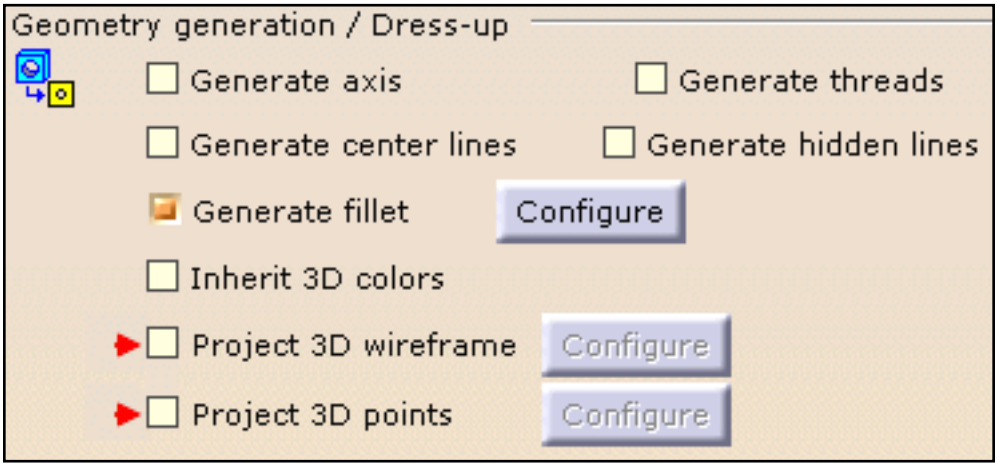
By default, this option is cleared.

Drafting for Electrical 3D Design

This page deals with the following option used with Electrical 3D Design:

- Geometry generation / Dress-up

Drafting View options



Click here to know more about the Drafting **View** options.

Project 3D wireframe

You need to select this option to take advantage of the line type personalization used to display the bundle segments as single line in the drawings you will generate.

By default, this option is cleared.

Project 3D points

You need to select this option to display construction points in the drawings you will generate..

By default, this option is cleared.

Electrical Data Exchange Format

To complete the end-to-end process, it is possible to get electrical specifications from external data. Those specifications consist of:

- a device list containing the device attributes and the assembly connectivity
- and a wire or equipotential list containing their attributes and the from-to connectivity.

This information will be used by CATIA Electrical products to implement in the digital mock-up the electrical systems driven by any electrical authoring tool (schematics or database for example).
You can either access this information through CAA APIs (refer to CAA documentation) or through an XML file. This method (using an XML file) is described in Electrical Integration from External Data.

You will find hereafter the XML schema to create your own interface to CATIA Electrical Tools.

The exchange data model is different from the data model of CATIA Electrical solutions. Actually, this data model consists of pertinent information that needs to transit between Electrical specification tools (schematics, etc.) and CATIA. It only aims at implementing those specifications in the mock-up. So only this pertinent information (objects, attributes and connectivity) is described in the following XML schema and not all the information stored in the CATIA electrical product documents.

In the context of CATIA P1, the CAA APIs are not supported.

As well, several objects are not managed in CATIA P1:

- equipotential
- connector shell
- splice

even if defined in the XML file: the same XML schema is shared for P1 and P2 products.

Preamble

iXF is a format defined by Dassault Systèmes, which is used for the data exchange in XML within CATIA.

Succinctly, iXF is based on the SOAP format and relies on the XML and the XML schema concept. It makes it possible to describe a grammar (a specific data model) expressing the object, class and behavior concepts as well as documents containing data conforming to the defined grammar.

A detailed description of the iXF format can be found at the following address: http://www.ixfstd.org/

Describing the iXF Electrical Schema
Considering the iXF Schema in Greater Depth

Describing the iXF Electrical Schema

The electrical schema, defined for the data exchange between CATIA V5 electrical products and external applications (CAA partners applications, etc.), describes a subset of electrical objects together with their relations.

This schema is based on the fact that an object is defined as a class, which one is associated to a behavior set. Thus, an electrical connector corresponds to the **Connector** class, to which the **Connector** and **Product** behaviors are associated.

Electrical Objects

The following classes with their associated behavior describe the electrical objects within the iXF Electrical Schema.

- Harness
  - Harness
  - Product
- Wire
  - Wire
  - Product
- Equipotential
  - Equipotential
- Equipment
  - Equipment
  - Product
  - Function
- ConnectorShell
  - ConnectorShell
  - Product
- Connector
  - Connector
  - Product
- Splice
  - Splice
  - Product
- Pin
  - Pin
  - Product
- Cavity
  - Cavity

All these classes derived from the **Object** abstract class. This one is not to be used as is but allows you to define the **Name** attribute for all the classes deriving from the **Object** class.

The units for all the attributes are given in the standard MKS system.

The following behaviors, associated to the electrical objects are defined in the iXF electrical schema (behavior named **ClassBehavior**):

- Harness
  - Attribute: SubType
- Wire
  - Attributes: InnerDiameter, OuterDiameter (mandatory), BendRadius, Length, Color, LinearMass, SeparationCode, SubType, SignalId
- Equipment
  - Attribute: SubType
- ConnectorShell
  - Attribute: SubType
- Connector
  - Attributes: SubType, Color, MatingConnector
- Splice
  - Attribute: SubType
- Pin
  - Attribute: SubType
- Cavity
  - Attribute: SubType
- Product
  - Attribute: PartNumber
- Equipotential
  - Attributes: EstimatedDiameter, SeparationCode, RoutingPriority, WirePartNumber, SubType
- Function
  - Attributes: System_Type (mandatory if the **Function** behavior is defined), Description, Localization

At last, all the objects expressed in a iXF document have an attribute identifying in a unique way each object within the project, except for the objects of **Link** type, which have a unique identifier within the document.

To illustrate how to describe an electrical object with the iXF format, let's take the example of an electrical connector:

```
- <ixf:object xsi:type="tns:Connector" id="V242">
    <tns:Name>Motor,Window_Driver</tns:Name>
  - <NS1:Connector>
      <NS1:Color>Yellow</NS1:Color>
      <NS1:SubType>Single Insert Connector</NS1:SubType>
    </NS1:Connector>
  - <NS1:Product>
      <NS1:PartNumber>5584555 -5W</NS1:PartNumber>
    </NS1:Product>
  </ixf:object>
```

This electrical connector of **Connector** class has:

- as identifier: **V242** (attribute 'id')
- as name: **Motor,Window_Driver** (attribute 'Name')
- as reference: **5584555 -5W** (attribute 'PartNumber' via its **Product** behavior)
- as subtype: **Single Insert Connector** (attribute 'SubType' via its **Connector** behavior)
- as color: **Yellow** (attribute 'Color' via its **Connector** behavior)

Relations between Electrical Objects

Using a Link type behavior

Nearly all the relations between electrical objects take the form of objects with a Link type behavior To know about the standard behavior definition, refer to http://www.ixfstd.org/std/ns/core/classBehaviors/links/1.0.

The **Link** object classes of the electrical schema are the following:

- WireLink
- EquipotentialLink
- DeviceLink
- HarnessLink

These object classes allow you to define:

a. the connectivity of the wire and equipotential objects, that is to say their connections with the electrical components (instantiated standard parts).

b. the aggregation relations between the electrical components (instantiated standard parts).
For example a connector with pins or an equipment with connectors and pins.

c. the harness composition (wires and electrical components)

These links take the form of identifiers.
For example: a wire, which identifier is **W1**, connected to two electrical connectors, which identifiers are **C1** and **C2**:

```
- <ixf:object xsi:type="tns:WireLink" id="ID_WL1">
  - <NS2:link>
      <NS2:object1 href="#W1" />
      <NS2:object2 href="#C1" />
    </NS2:link>
  </ixf:object>
- <ixf:object xsi:type="tns:WireLink" id="ID_WL2">
  - <NS2:link>
      <NS2:object1 href="#W1" />
      <NS2:object2 href="#C2" />
    </NS2:link>
  </ixf:object>
```

Using a Specific Attribute

The relation between two connectors (mating connector, connector) is not managed by a Link type object as above but using a specific attribute named **MatingConnector** (behavior attribute of **Connector** type). This attribute is optional and is valuated with the identifier of the mating connector on both sides of the connection.

Let see an example where two connectors are connected together. The connectivity between them is described as follows:

```
- <ixf:object id="ConnectorIdentifier1" xsi:type="tns:Connector">
    <tns:Name>PB1R-BNL2.1</tns:Name>
  - <NS1:Connector>
      <NS1:MatingConnector>ConnectorIdentifier2</NS1:MatingConnector
      >
    </NS1:Connector>
  - <NS1:Product>
      <NS1:PartNumber>PB1R-BNL2</NS1:PartNumber>
    </NS1:Product>
  </ixf:object>

- <ixf:object id="ConnectorIdentifier2" xsi:type="tns:Connector">
    <tns:Name>PB1L-BNL1.1</tns:Name>
  - <NS1:Connector>
      <NS1:MatingConnector>ConnectorIdentifier1</NS1:MatingConnector
      >
    </NS1:Connector>
  - <NS1:Product>
      <NS1:PartNumber>PB1L-BNL1</NS1:PartNumber>
    </NS1:Product>
  </ixf:object>
```

Considering the iXF Schema in Greater Depth

The iXF electrical schema is split in four parts (four files):

- the first one: **ElectricalSchema.xsd** refers to the other three and describes the Electrical Object Classes.
- the second one: **IXF_CB_NS1.xsd** describes strictly the Electrical Behaviors.
- the third one: **IXF_CB_NS2.xsd** describes the **Link** type Behaviors (iXF standard reference).
- and last but not least, the fourth one: **IXF_CB_NS3.xsd** describes the Functional Behaviors.

Electrical Object Classes

```
- <schema xmlns="http://www.w3.org/2001/XMLSchema"
    targetNamespace="IXF_Schema.xsd" xmlns:tns="IXF_Schema.xsd"
    xmlns:ixf="http://www.ixfstd.org/std/ns/core/1.0"
    xmlns:xsd="http://www.w3.org/2001/XMLSchema"
    xmlns:SOAP_ENV="http://schemas.xmlsoap.org/soap/envelope/"
    xmlns:NS1="CATIA/V5/Electrical/1.0"
    xmlns:NS2="http://www.ixfstd.org/std/ns/core/classBehaviors/links/1.
    0" xmlns:NS3="CATIA/V5/ElecFunctionalBehavior/1.0">
    <import namespace="http://www.ixfstd.org/std/ns/core/1.0"
      schemaLocation="http://www.ixfstd.org/std/schema/core/1.0/core.
      xsd" />
    <import namespace="http://schemas.xmlsoap.org/soap/envelope/"
      schemaLocation="http://schemas.xmlsoap.org/soap/envelope/" />
    <import namespace="CATIA/V5/Electrical/1.0"
      schemaLocation="IXF_CB_NS1.xsd" />
    <import
      namespace="http://www.ixfstd.org/std/ns/core/classBehaviors/lin
      ks/1.0" schemaLocation="IXF_CB_NS2.xsd" />
    <import namespace="CATIA/V5/ElecFunctionalBehavior/1.0"
      schemaLocation="IXF_CB_NS3.xsd" />
  - <complexType name="Object" abstract="true"
      ixf:dataModelRole="ixf:class">
    - <complexContent>
      - <extension base="ixf:root">
        - <sequence>
            <element name="Name" type="string"
              ixf:dataModelRole="ixf:attribute" />
          </sequence>
        </extension>
      </complexContent>
    </complexType>
  - <complexType name="Harness" ixf:dataModelRole="ixf:class">
    - <complexContent>
      - <extension base="tns:Object">
```

```
</complexType>
- <complexType name="Harness" ixf:dataModelRole="ixf:class">
  - <complexContent>
    - <extension base="tns:Object">
      - <sequence>
          <element ref="NS1:Harness" />
          <element ref="NS1:Product" />
        </sequence>
      </extension>
    </complexContent>
  </complexType>
- <complexType name="Wire" ixf:dataModelRole="ixf:class">
  - <complexContent>
    - <extension base="tns:Object">
      - <sequence>
          <element ref="NS1:Wire" />
          <element ref="NS1:Product" />
        </sequence>
      </extension>
    </complexContent>
  </complexType>
- <complexType name="Equipotential" ixf:dataModelRole="ixf:class">
  - <complexContent>
    - <extension base="tns:Object">
      - <sequence>
          <element ref="NS3:Equipotential" />
        </sequence>
      </extension>
    </complexContent>
  </complexType>
- <complexType name="Equipment" ixf:dataModelRole="ixf:class">
  - <complexContent>
    - <extension base="tns:Object">
      - <sequence>
          <element ref="NS1:Equipment" />
          <element ref="NS1:Product" />
```

```
            <element ref="NS3:Function" />
          </sequence>
        </extension>
      </complexContent>
    </complexType>
  - <complexType name="ConnectorShell" ixf:dataModelRole="ixf:class">
    - <complexContent>
      - <extension base="tns:Object">
        - <sequence>
            <element ref="NS1:ConnectorShell" />
            <element ref="NS1:Product" />
          </sequence>
        </extension>
      </complexContent>
    </complexType>
  - <complexType name="Connector" ixf:dataModelRole="ixf:class">
    - <complexContent>
      - <extension base="tns:Object">
        - <sequence>
            <element ref="NS1:Connector" />
            <element ref="NS1:Product" />
          </sequence>
        </extension>
      </complexContent>
    </complexType>
  - <complexType name="Splice" ixf:dataModelRole="ixf:class">
    - <complexContent>
      - <extension base="tns:Object">
        - <sequence>
            <element ref="NS1:Splice" />
            <element ref="NS1:Product" />
          </sequence>
```

```
        </extension>
      </complexContent>
    </complexType>
  - <complexType name="Pin" ixf:dataModelRole="ixf:class">
    - <complexContent>
      - <extension base="tns:Object">
        - <sequence>
            <element ref="NS1:Pin" />
            <element ref="NS1:Product" />
          </sequence>
        </extension>
      </complexContent>
    </complexType>
  - <complexType name="Cavity" ixf:dataModelRole="ixf:class">
    - <complexContent>
      - <extension base="tns:Object">
        - <sequence>
            <element ref="NS1:Cavity" />
          </sequence>
        </extension>
      </complexContent>
    </complexType>
  - <complexType name="WireLink" ixf:dataModelRole="ixf:class">
    - <complexContent>
      - <extension base="ixf:root">
        - <sequence>
            <element ref="NS2:link" />
          </sequence>
        </extension>
      </complexContent>
    </complexType>
  - <complexType name="EquipotentialLink" ixf:dataModelRole="ixf:class">
    - <complexContent>
```

```
      - <extension base="ixf:root">
        - <sequence>
            <element ref="NS2:link" />
          </sequence>
        </extension>
      </complexContent>
    </complexType>
  - <complexType name="DeviceLink" ixf:dataModelRole="ixf:class">
    - <complexContent>
      - <extension base="ixf:root">
        - <sequence>
            <element ref="NS2:link" />
          </sequence>
        </extension>
      </complexContent>
    </complexType>
  - <complexType name="HarnessLink" ixf:dataModelRole="ixf:class">
    - <complexContent>
      - <extension base="ixf:root">
        - <sequence>
            <element ref="NS2:link" />
            <element ref="NS2:directedLink" />
            <element ref="NS2:treeLink" />
          </sequence>
        </extension>
      </complexContent>
    </complexType>
    <element name="Envelope" type="SOAP_ENV:Envelope" />
  </schema>
```

Electrical Behaviors

```
- <schema xmlns="http://www.w3.org/2001/XMLSchema"
    targetNamespace="CATIA/V5/Electrical/1.0"
    xmlns:tns="CATIA/V5/Electrical/1.0"
    xmlns:ixf="http://www.ixfstd.org/std/ns/core/1.0"
    xmlns:xsd="http://www.w3.org/2001/XMLSchema"
    xmlns:SOAP_ENV="http://schemas.xmlsoap.org/soap/envelope/">
    <import namespace="http://www.ixfstd.org/std/ns/core/1.0"
      schemaLocation="http://www.ixfstd.org/std/schema/core/1.0/co
      re.xsd" />
    <import namespace="http://schemas.xmlsoap.org/soap/envelope/"
      schemaLocation="http://schemas.xmlsoap.org/soap/envelope/"
      />
  - <complexType name="Harness"
      ixf:dataModelRole="ixf:class_behavior">
    - <sequence>
        <element name="SubType" minOccurs="0" type="string"
          ixf:dataModelRole="ixf:attribute" />
      </sequence>
    </complexType>
    <element name="Harness" type="tns:Harness" />
  - <complexType name="Wire" ixf:dataModelRole="ixf:class_behavior">
    - <sequence>
        <element name="InnerDiameter" minOccurs="0"
          type="double" ixf:dataModelRole="ixf:attribute" />
        <element name="OuterDiameter" type="double"
          ixf:dataModelRole="ixf:attribute" />
        <element name="BendRadius" minOccurs="0" type="double"
          ixf:dataModelRole="ixf:attribute" />
        <element name="Length" minOccurs="0" type="double"
          ixf:dataModelRole="ixf:attribute" />
        <element name="Color" minOccurs="0" type="string"
          ixf:dataModelRole="ixf:attribute" />
        <element name="LinearMass" minOccurs="0" type="double"
          ixf:dataModelRole="ixf:attribute" />
```

```
      <element name="SeparationCode" minOccurs="0"
        type="string" ixf:dataModelRole="ixf:attribute" />
      <element name="SubType" minOccurs="0" type="string"
        ixf:dataModelRole="ixf:attribute" />
      <element name="SignalId" minOccurs="0" type="string"
        ixf:dataModelRole="ixf:attribute" />
    </sequence>
  </complexType>
  <element name="Wire" type="tns:Wire" />
- <complexType name="Equipment"
    ixf:dataModelRole="ixf:class_behavior">
  - <sequence>
      <element name="SubType" minOccurs="0" type="string"
        ixf:dataModelRole="ixf:attribute" />
    </sequence>
  </complexType>
  <element name="Equipment" type="tns:Equipment" />
- <complexType name="ConnectorShell"
    ixf:dataModelRole="ixf:class_behavior">
  - <sequence>
      <element name="SubType" minOccurs="0" type="string"
        ixf:dataModelRole="ixf:attribute" />
    </sequence>
  </complexType>
  <element name="ConnectorShell" type="tns:ConnectorShell" />
- <complexType name="Connector"
    ixf:dataModelRole="ixf:class_behavior">
  - <sequence>
      <element name="MatingConnector" minOccurs="0"
        type="string" ixf:dataModelRole="ixf:attribute" />
      <element name="Color" minOccurs="0" type="string"
        ixf:dataModelRole="ixf:attribute" />
      <element name="SubType" minOccurs="0" type="string"
```

```
        ixf:dataModelRole="ixf:attribute" />
    </sequence>
  </complexType>
  <element name="Connector" type="tns:Connector" />
- <complexType name="Splice" ixf:dataModelRole="ixf:class_behavior">
  - <sequence>
      <element name="SubType" minOccurs="0" type="string"
        ixf:dataModelRole="ixf:attribute" />
    </sequence>
  </complexType>
  <element name="Splice" type="tns:Splice" />
- <complexType name="Pin" ixf:dataModelRole="ixf:class_behavior">
  - <sequence>
      <element name="SubType" minOccurs="0" type="string"
        ixf:dataModelRole="ixf:attribute" />
    </sequence>
  </complexType>
  <element name="Pin" type="tns:Pin" />
- <complexType name="Cavity" ixf:dataModelRole="ixf:class_behavior">
  - <sequence>
      <element name="SubType" minOccurs="0" type="string"
        ixf:dataModelRole="ixf:attribute" />
    </sequence>
  </complexType>
  <element name="Cavity" type="tns:Cavity" />
- <complexType name="Product" ixf:dataModelRole="ixf:class_behavior">
  - <sequence>
      <element name="PartNumber" minOccurs="0" type="string"
        ixf:dataModelRole="ixf:attribute" />
    </sequence>
  </complexType>
  <element name="Product" type="tns:Product" />
</schema>
```

Link Type Behaviors

```
- <schema xmlns="http://www.w3.org/2001/XMLSchema"
    targetNamespace="http://www.ixfstd.org/std/ns/core/classBehaviors/lin
    ks/1.0"
    xmlns:tns="http://www.ixfstd.org/std/ns/core/classBehaviors/links/1.0
    " xmlns:ixf="http://www.ixfstd.org/std/ns/core/1.0"
    xmlns:xsd="http://www.w3.org/2001/XMLSchema"
    xmlns:SOAP_ENV="http://schemas.xmlsoap.org/soap/envelope/">
    <import namespace="http://www.ixfstd.org/std/ns/core/1.0"
      schemaLocation="http://www.ixfstd.org/std/schema/core/1.0/core.
      xsd" />
    <import namespace="http://schemas.xmlsoap.org/soap/envelope/"
      schemaLocation="http://schemas.xmlsoap.org/soap/envelope/" />
  - <complexType name="link" ixf:dataModelRole="ixf:class_behavior">
    - <sequence>
        <element name="object1" nillable="true"
          type="ixf:objectReference" ixf:dataModelRole="ixf:attribute"
          />
        <element name="object2" nillable="true"
          type="ixf:objectReference" ixf:dataModelRole="ixf:attribute"
          />
      </sequence>
    </complexType>
    <element name="link" type="tns:link" />
  - <complexType name="directedLink"
      ixf:dataModelRole="ixf:class_behavior">
      <sequence />
    </complexType>
    <element name="directedLink" type="tns:directedLink" />
  - <complexType name="treeLink" ixf:dataModelRole="ixf:class_behavior">
      <sequence />
    </complexType>
    <element name="treeLink" type="tns:treeLink" />
  </schema>
```

Functional Behaviors

```
- <schema xmlns="http://www.w3.org/2001/XMLSchema"
    targetNamespace="CATIA/V5/ElecFunctionalBehavior/1.0"
    xmlns:tns="CATIA/V5/ElecFunctionalBehavior/1.0"
    xmlns:ixf="http://www.ixfstd.org/std/ns/core/1.0"
    xmlns:xsd="http://www.w3.org/2001/XMLSchema"
    xmlns:SOAP_ENV="http://schemas.xmlsoap.org/soap/envelope/">
    <import namespace="http://www.ixfstd.org/std/ns/core/1.0"
      schemaLocation="http://www.ixfstd.org/std/schema/core/1.0/co
      re.xsd" />
    <import namespace="http://schemas.xmlsoap.org/soap/envelope/"
      schemaLocation="http://schemas.xmlsoap.org/soap/envelope/"
      />
  - <complexType name="Function"
      ixf:dataModelRole="ixf:class_behavior">
    - <sequence>
        <element name="System_Type" type="string"
          ixf:dataModelRole="ixf:attribute" />
        <element name="Description" minOccurs="0" type="string"
          ixf:dataModelRole="ixf:attribute" />
        <element name="Localisation" minOccurs="0" type="string"
          ixf:dataModelRole="ixf:attribute" />
      </sequence>
    </complexType>
    <element name="Function" type="tns:Function" />
  - <complexType name="Equipotential"
      ixf:dataModelRole="ixf:class_behavior">
    - <sequence>
        <element name="EstimatedDiameter" minOccurs="0"
          type="double" ixf:dataModelRole="ixf:attribute" />
        <element name="SeparationCode" minOccurs="0"
          type="string" ixf:dataModelRole="ixf:attribute" />
        <element name="RoutingPriority" minOccurs="0" type="string"
          ixf:dataModelRole="ixf:attribute" />
        <element name="WirePartNumber" minOccurs="0"
          type="string" ixf:dataModelRole="ixf:attribute" />
        <element name="SubType" minOccurs="0" type="string"
          ixf:dataModelRole="ixf:attribute" />
      </sequence>
    </complexType>
    <element name="Equipotential" type="tns:Equipotential" />
  </schema>
```

Glossary

B C D E I P R S T W

B

bend radius The minimum bend radius allowed for the bundle segment: it corresponds to the maximum torsion possibly applied to a wire according to its physical characteristics.

bundle A document containing wires

bundle segment A geometrical subdivision of a bundle
Also called segment or BNS in V4 environment.

bundle connection point Sub-element used to place the bundle segment extremity on the electrical components.

C

cavity Sub-element allowing one associated component such as a connector. An electrical connection can be defined between a cavity and a cavity connection point.

cavity connection point Sub-element allowing the connection into a cavity. This connection corresponds to an assembly relation. The connectors allow a cavity connection point but not the equipments.

connect Establishes peculiar constraints between two electrical objects:

- the electrical signal continuity is ensured between the connected components.
- if connection points have been defined, the mechanical assembly constraints are automatically created.

connector Basic term to define the single insert connector.

connector connection point Sub-element allowing the connection with another connector connection point. Only the single insert connectors use the connector connection point.

convert into electrical device Act to add an electrical behavior to an existing component which becomes an electrical component. If the component is a reference, it becomes an electrical reference. As opposed, converting an instance only add an electrical behavior to this instance, but doesn't in any case modify the reference.

D

diameter Corresponds to the diameter of the wire together with the insulation.

disconnect Deletes the electrical connection between two electrical components.

E

electrical behavior Peculiar reactions of a component bound to additional rules adapted to the electrical domain.

electrical component A product or a part with an electrical behavior

electrical connection Constraint existing between two electrical objects. Allows connection relations as well as assembly constraints.

electrical system An electrical unit which accomplishes a specific function. Consists of equipment, connectors and signals. Described in a CATProduct document.

equipment An electrical device with one or more associated components: connectors placed in cavities. An equipment can also comprise terminations and bundle connection points.

I

instance Designation of reference placed in a context, i.e. in design mode. An instance can have additional characteristics that do not belong to the reference. For example, the instance of a wire has a length attribute in an assembly context when its reference doesn't out of this context.
As opposed, see reference.

P

properties Attributes of a component that define its electrical, mechanical, etc. characteristics.

R

reference Component model corresponding to the definition of a real object. A reference can be stored in a catalog. In the electrical context a reference corresponds either to a CATPart document being electrified or to a CATProduct document which root product is being electrified. As opposed, see instance.

S

section Corresponds to the section of the wire together with the insulation.

segment See bundle segment.

single insert connector Electrical connector male or female. It comprises terminations, also bundle connection points, only one connector connection point, only one cavity connection point.

store (a device) Place an electrical component into a catalog family.

support A mechanical object used to hold the bundle segments in position.

T

termination Sub-element ensuring the electrical signal conduction between any type of electrical component. It is indissociable from the electrical component and corresponds to a contact crimped into a cavity. Terminations are allowed on equipments and connectors. Also called pin.

W

wire Electrical wire: physical object corresponding to a signal (several wires can correspond to a signal).
A wire reference is characterized by properties such as: section, diameter, bend radius, color, linear mass, etc.

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