- Transforming
  - Creating Mirror/Copy Elements
  - Creating Translate/Copy Elements
- Copying/Pasting elements
- Renaming Features
- Editing Fasteners' Properties

Advanced Tasks

- Running the Fastening Rules Analysis
- Searching Automotive BiW Features
- Integration With Drafting
- Measure Tools

Automotive Body In White Fastening Interoperability

- Optimal CATIA PLM Usability for Automotive BiW Fastening

Workbench Description

- Menu Bar
- Automotive BiW Fasteners Toolbar
- Welding Toolbar
- Adhesive Toolbar
- Sealant Toolbar
- Unspec Toolbar
- Mechanical BiW Fasteners Toolbar
- Analysis Toolbar
- Tools Toolbar
- Measure Toolbar
- Specification Tree

Customizing

- General Settings
- BiW Fastening Application Display User Settings

Index

Overview

Welcome to the *Automotive Body in White Fastening User's Guide* !
This guide is intended for users who need to become quickly familiar with the product.

This overview provides the following information:

- Automotive Body in White Fastening in a Nutshell
- Before Reading this Guide
- Getting the Most Out of this Guide
- Accessing Sample Documents
- Conventions Used in this Guide

Automotive Body in White Fastening in a Nutshell

The Automotive Body in White Fastening application is a product dedicated to the design of Automotive Body In White Fasteners. It supports welding technologies and mechanical clinching, along with Adhesives, Sealers, and Mastics.

For each of these general fastening technologies, specific Fastening process codes along with their matching relevant parameters can be set-up in an Application Parameters Start-Up file. This file can be fully customized by the customer, according to company or industry specific standards requirements. Its feature-based approach offers both a highly structured and intuitive design environment.

Fasteners can be gathered so that to connect specific contact zones of the parts to join. You can create, edit and delete spot like fasteners.

In addition to placing the fasteners, reports can be issued from the application in order to list: data can be exported (respectively imported) to a neutral ASCII text format, providing opening with customer legacy systems.

As a scalable product, Automotive Body in White Fastening can be used in cooperation with other products such as *Assembly Design* and *Generative Shape Design*.

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.

You may also like to read the following complementary product guides:

- *Part Design*,
- *Assembly Design*,
- *Generative Shape Design*

Getting the Most Out of this Guide

To get the most out of this guide, we suggest that you start reading and performing the step-by-step Getting Started tutorial.

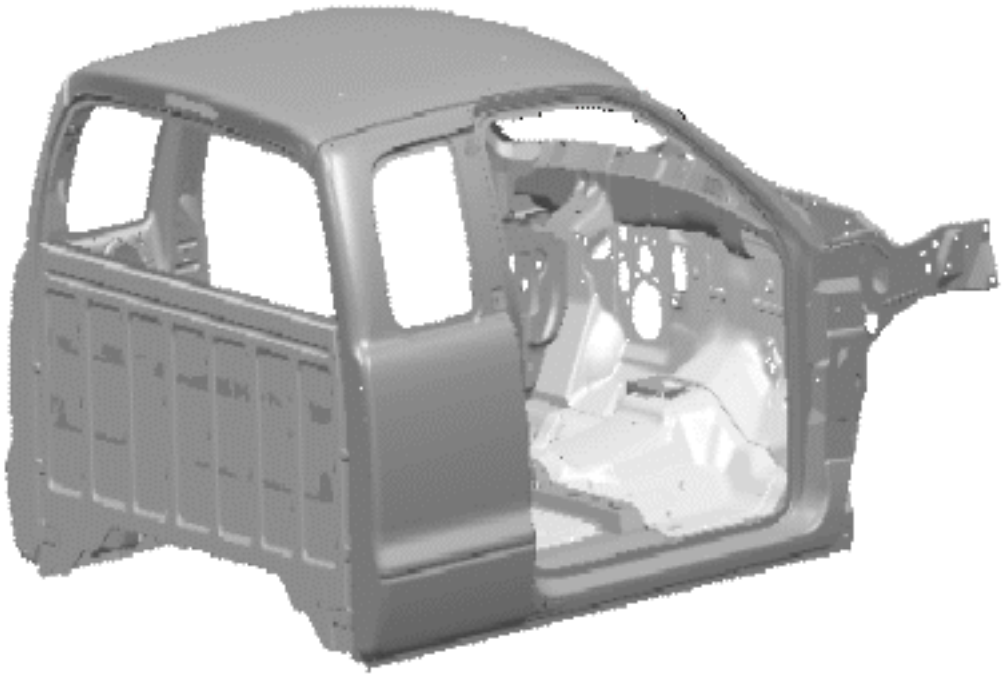
Once you have finished, you should move on to the Basic Tasks and Advanced Tasks sections, which deal with handling all the product functions.

The Workbench Description section, which describes the Automotive BiW Fastening 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 on accessing sample documents, refer to Accessing Sample Documents in the *Infrastructure User's Guide*.



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

Index

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)

What's New?

New Functionality

Measure Tools

Enhanced Functionalities

Getting Started

Customizing the BiW Fastening Application Standard File
- New discretization parameter for curvebeads
- New Shape Definition attributes for BiW fasteners: Shank, Shank with Head, Shank with Head and Foot

Defining Symbols
- New symbols: 136, 137, 138, 139

Setting up a CATIA Reference Product with BiW Fastening Parameters
- New design rules: Joined thickness normal alignment, Maximum/Minimum thickness ratio, Spot center distance to flange edge, Spot border distance to flange edge, Diameter overhang

Basic Tasks

Creating SpotPoints
- The ''On Support Surface'' location method is replaced by ''On Surface'' to support the location of a spotpoint on any selected surface belonging to a product component of the BiW joint
- You can select a circle when using the On Surface, On Point and Explicit location methods
- Using the Bitangent Circle Center location method, if the curves are non coplanar, the center of gravity is computed
- New K Axis option to define the orientation of the spotpoint
- It is now possible to associate an annotation to a spotpoint
- It is now possible to assign a layer to a spot fastener
- It is now possible to apply visualization filters on spot fasteners

Creating Mechanical SpotPoints
- New process type: rivet

Creating Curvebeads
- New location method: Intersect
- New parameters: K axis orientation and discretization
- It is now possible to associate an annotation to a curvebead
- It is now possible to assign a layer to a spot fastener
- It is now possible to apply visualization filters on spot fasteners

Creating Structural Reports
- New attributes: Layer number, K flag, I flag, X, Y, Z coordinates, normal reference, and tangent vector
- Deactivating features are no longer reported

Creating Flat Reports
- A new version of the flat report is available

New attributes: Layer number, X, Y, Z coordinates, normal reference, and tangent vector

Deactivated features are no longer reported

Exporting

New attributes: Layer number, K flag, I flag, X, Y, Z coordinates, normal reference, and tangent vector

Discretization value when exporting a curvebead

Deactivated features are no longer reported

Isolating Features

You can now isolate all elements from the parameters in the specification tree

Advanced Tasks

Running the Fastening Rules Analysis

New rules: Thicknesses normals alignment, Maximum/Minimum thickness ratio, Fastener center distance to flange edge, Fastener border distance to flange edge, Diameter overhang

Integration With Drafting

New 2D symbols associated to 3D symbols

Customizing

General Settings

New option to allow the creation of unsaved data from Enovia

BiW Fastening Application Display User Settings

New Fastener axis orientation parameters to be displayed in the 3D geometry

Getting Started

The following tutorial aims at giving you a feel of what you can do with Automotive Body in White Fastening. It provides a step-by-step scenario showing you how to use key capabilities.

The tasks proposed in this section are:

Customizing the BiW Fastening Application Standard File
Adding a New Process Type
Defining Symbols
Entering the Workbench
Setting up a CATIA Reference Product with BiW Fastening Parameters

All together these tasks should take 20 minutes to complete.

Customizing the BiW Fastening Application Standard File

The Automotive BiW Fastening Application natively supports the most common BiW joining process categories: Welding, Adhesive, Sealant, and BiW Mechanical. An additional "Unspecified" Process Category is supplied in order to classify specific technologies that do not belong to the application natively supported process categories.
For each general process category, specific fastening process codes along with their matching relevant parameters can be set-up in the Application Parameters Start-Up standard file named **GBF_STD**.
Here are the steps to customize the standard file:

- Locating the CATIA BiW Fastening Application Standard file in the CATIA run-time environment
- Customizing the BiW Fastening Application Standard GBF_STD.xls file
- What can be modified?
- Saving the GBF_STD excel file and keeping its name

Locating the CATIA BiW Fastening Application Standard file in the CATIA run-time environment

The "GBF_STD" Standard file is located according to the reffiles/GbfStandards path from CATIA run time view root directory (intel_a for Windows, aix_a for IBM UNIX, irix_a for SGI UNIX, solaris_a for SUN Unix, hpux_a for HP UNIX). Two extensions are provided for this file:

- .xls for Windows Operating System run time usage
- .txt for UNIX operating system run time usage

For example, you will find in the directory:
MyCompanyRootRuntimeview/intel_a/reffiles/GbfStandards, the following files:

- GBF_STD.xls which is the master file that can be edited with the Excel Windows Editor
- GBF_STD.txt which must be generated from the master GBF_STD.xls file using the Save as *.txt command in Excel


Customizing the BiW Fastening Application Standard GBF_STD.xls file

The GBF_STD.xls standard file can be customized by the company data administrator, according to his company or industry specific standards requirements.
This file features two columns:

- the first column identifies the parameter name
- the second column specifies the list of possible values of the identified parameter : value1, <value2>, value3. (brackets <>, specify the default value: i.e. value2 is the default value of the previous list)

Rows are divided into six chapters delimited by excel row cells and filled in different colors.

- The first chapter (light green fill-in) specifies the list of CATIA application parameters supported whatever the fastener process category can be. This chapter is itself subdivided into three sections:
  - A: CATIA V5 BiW Fastening Application parameters
    - Number of digits for fixed-length
  - B: CATIA V5 BiW Joint and Joint Body Parameters
  - C: CATIA V5 BiW Fastener parameters
    - 1. Life Cycle parameters
    - 2. Fastener type parameters
    - 3. Fastener Parameters
- The five additional chapters enable to set-up the list of the fasteners parameters respectively for each specific process type and for each supported process category:
  - Welding (yellow fill-in)
  - Adhesive (cyan fill-in)
  - Sealant (magenta fill-in)
  - BiW Mechanical (green fill-in)
  - Unspecified (brown fill-in)

These chapters are themselves subdivided into sections C-1, C-2, C-3.

What can be modified?

Pre-defined values of each BiW Fastening general parameter identified in the sections C-2, C-3 of chapter 1

Modifying the list of values (column 2) for each row parameter of Chapter 1, you can customize:

- the applicable process categories (PCATS parameter) value list: WLD (Welding), ADH (Adhesive), SEA (Sealant), BWM (BiW Mechanical), U (Unspecified Process Category). For example, you can narrow this list to: WLD, BWM if you plan to use only Welding and BiW mechanical fastening technologies.

- the list of applicable fastener types (FTY parameter):
  - SpotPoint
  - SpotProjection
  - CurveBead

Note that if you do not set up the Fastener Types in the standard file, then the corresponding Fastener commands will be appear grayish meaning that they cannot be activated.

- the assigned color for each process category (WLD_Color parameter, ADH_Color parameter, transparent, etc...).

WLD_Color	**TRANSPARENT**

- the assigned symbol to thickness count if necessary:
  - By default, the code 111 is assigned to 1T Thickness Count -> 
  - By default, the code 151 is assigned to 2T Thickness Count -> 
  - By default, the code 113 is assigned to 3T Thickness Count -> 
  - By default, the code 114 is assigned to 4T Thickness Count -> 

/ / Set-up the attached symbol to Thickness Count	
> NTH_1_SYM	**111**
NTH_2_SYM	**151**
NTH_3_SYM	**113**
NTH_4_SYM	**114**

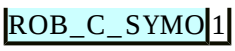
Please refer to the Symbols table for the available list of symbols.

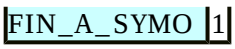
- the list of possible process categories for each fastener (SpotPoint_PCA parameter): this list must be set up as a subset of the PCATS parameter values.
  For example, you can narrow the list to WLD if you plan to use only Spot Welding technologies.

- the lists of possible process types values for each process category (PCAName_PTYPS parameter). For instance:
  - the list of Welding process types (WLD_PTYPS parameter): <21>;14;141;52;UNSW

- the list of possible values of each CATIA BiW Fastener attribute:
  - robustness (ROB parameter): A,B,C,D,U
  - regulation (REG parameter): A,B,C,D,U
  - finish (FIN parameter): A,B,C,D,U
  - material (MAT parameter): MAT1, MAT2, etc.
    For each fastener attribute, you can choose your own set of code values (except for the discretization).
    For example, you can choose to customize:
    - the list of values of the robustness attributes to: A,B,C,D,E,U
    - the list of values of the regulation attributes to : A,B

- the specific "relevant" value for each classification parameters: Robustness, Finish, Regulation, etc. and assign to this value a Symbol Mark Code:
  - Example1: Assign Mark code "1" to Robustness parameter value C.
  - Example2: Assign Mark code "1" to Finish parameter value A





Each attribute value (A, B, etc) is the coded value of the attribute as it will be stored in the Data Model.

You can provide a end user translation for each parameter coded value in the CATIA NLS resource file CATBfmAttributesNLS.txt . In this file, each parameter coded value is identified by a key built by concatenating the name of the parameter and the coded value, separated by the "_" string separator. The translation of the parameter is provided by setting the parameter value key to its translation text.
For example the line: ROB_A = "Critical" provides the end-user with the "Critical" translation of the coded value A of the Robustness parameter (ROB).

The CATBfmAttributesNLS.txt file is located in your runtime environment in **MyCompanyRootRunTimeView/intel_a/resources/msgcatalog**.


You must not remove any parameter belonging to the Chapter 1 rows.

The list of pre-defined fastener process type parameters belonging to chapters 2 to 6

You can modify this list, either by:

- modifying the list of values for each row containing the pre-defined specific fastener process type parameters, or
- removing the pre-defined specific fastener process type parameters you are not interested in, or
- adding your company specific fastener process type parameters consistently with the list of process type values set up in Chapter 1

Let's take an example for a spotpoint of Process Category = Welding and Process Type = Resistance. You can set:

- the list of specific process types code values (SpotPoint_WLD_PTY parameter):
  <21>;14;141;52;UNSW
  In this list, the code value <21> matches the ISO code for Resistance Welding.
- the list of the applicable standard attributes (SpotPoint_21_APAT parameter): you can specify the list of attributes you consider as relevant for this process type. The attributes strings must be chosen among the CATIA list: ROB;REG;FIN;GFL;IFL;MID.
  For a curve fastener, its discretization parameter (DIS) is an added applicable attribute.

- the applicability or the non applicability of an add-on material (SpotPoint_21_AMAT parameter).
- the list of possible Shape Definition values (ASDF) of each parameter. The attribute strings must be chosen among the values: PT3 (3DPoint);HSP (Hemisphere);SH (Shank);SHH(Shank with Head);SHF(Shank with Head and Foot).
- the list of possible string code values of each parameter (SpotPoint_21_ROB, SpotPoint_21_REG, etc). For a curve fastener, its discretization parameter (CurveBead_PTY_DIS values) must be chosen among the SAG and STEP values.
- the assignment of a graphic symbol (SpotPoint_21_SYM parameter)
  Refer to the Symbols table for the available list of symbols.

Saving the GBF_STD excel file and keeping its name

- The data administrator saves this excel file by keeping its original name (GBF_STD.xls file for WINDOWS or GBF_STD.txt (Tab delimiter) for UNIX) and extension (.xls), as the name will be recognized by the application.
- On Unix, the GBF_STD.txt file must be generated from the master GBF_STD.xls file or by using the Save as *.txt command in Excel.

Here is an example of how to add a new process type named "MyWldType" for a Welding SpotPoint.

Adding a New Process Type

This task shows how to add a new process type named "MyWldType" for a Welding SpotPoint.

1. Add MyWLDType value in the list of the Welding Process Type possible values of the WLD_PTYPS parameter in the Section C-2, Chapter 1 (light-green fill-in).

// Set-Up Welding Process Types	reference Codes
WLD_PTYPS	<21>;14;141;52;UNSW;MyWLDType

2. Add MyWLDType value in the list of the Welding Process Type possible values for a Welding SpotPoint of the SpotPoint_WLD_PTY parameter in the section C-2, Chapter 2 (Welding Section, yellow fill-in)

// SECTION C-2: WELDING FASTENER Type PARAMETERS	
//	
//Set-Up Welding Process Types for Each applicable CATIA V5 BiW Fastener Shape Type	ISO Norm reference Codes
SpotPoint_WLD_PTY	<21>;14;141;52;UNSW;MyWLDType

3. Define which Fastener parameters are applicable to this SpotPoint and their values in the Section C-3, Chapter 2 (Welding Fastener Parameters Section, yellow fill-in).

#	
# SECTION C : WELDING FASTENER PARAMETERS	PARAMETERS VALUES
#	
#	
# SECTION C-1 : WELDING FASTENER Life Cycle PARAMETERS	
#	
# SECTION C-2 : WELDING FASTENER Type PARAMETERS	.
#	
# Set-Up the Welding Process Types for each applicable CATIA V5 BiW Fastener Shape Type	ISO Norm reference Codes
SpotPoint_WLD_PTY	<21>;14;141;52;UNSW;
SpotProjection_WLD_PTY	<23>;UPRJ
CurveBead_WLD_PTY	<13>;131;14;141;22;24;25;52;91;UNSW;
# SECTION C-3 : WELDING FASTENER PARAMETERS	Please, specify below the list of possible Code values for each CATIA Function attribute; Please, note that you have to provide the NLS translation of each code value in the CATBfmAttributesNLS.CATNls file .
#	
# ISO Code 21: Resistance Spot Welding	
# Set-up Function Attributes applicability (APAT) | Material Attribute applicability (AMAT) | Shape Definition applicability (ASDF)	Please specify below, the list of applicable Attributes that have to be set for a Resistance (21) Spot Point
SpotPoint_21_APAT	ROB;REG;FIN;GFL;IFL;MID
SpotPoint_21_AMAT	false
SpotPoint_21_ASDF	<PT3>
# Set-up the 21 Process Type Graphics Symbol	
SpotPoint_21_SYM	101
# Set-up the values of each applicable attribute	Please specify for each row, the list of applicable Code Values for each attribute
SpotPoint_21_ROB	<A>;B;C;D;U
SpotPoint_21_REG	<A>;B;C;U
SpotPoint_21_FIN	<A>;B;C;D;U

Please refer to the Symbols table to have further information on existing symbols.

4. In addition, you have the possibility to translate the coded value MyWLDType. You can provide this translation by modifying the CATBfmAttributesNLS.CATNls files located in resources/msgcatalog/

Here is an example using English.
In the appropriate file, add PTY_MyWLDType key like this:

```
// Process Type Attribute Code Values
// -----------------------------------------------------------------------------
PTY_14   = "TIG (14)";
PTY_141  = "TIG (141)";
PTY_21   = "Resistance (21)";
PTY_52   = "Laser (52)";
PTY_MyWLDType = "My Process Type Name";
//
PTY_UNSW = "Unspecified Welding";
```

5. Check the modifications in CATIA V5: import this standard and create a Welding SpotPoint, with the added process Type MyProcessTypeName.

Defining Symbols

The following table lists all the symbols used in the Automotive Body In White Fastening workbench.

Symbol	Value
	100
	101
	110
	111
	112
	113
	114
	121
	122
	123
	124
	125
	131
	132
	133
	134
	135
	136
	137
	138
	139
	140
	141
	142


151


152


153


154


161


162

171

172

173


200


201


211


221


231


241


251


301


311


321


331


341

Entering the Workbench

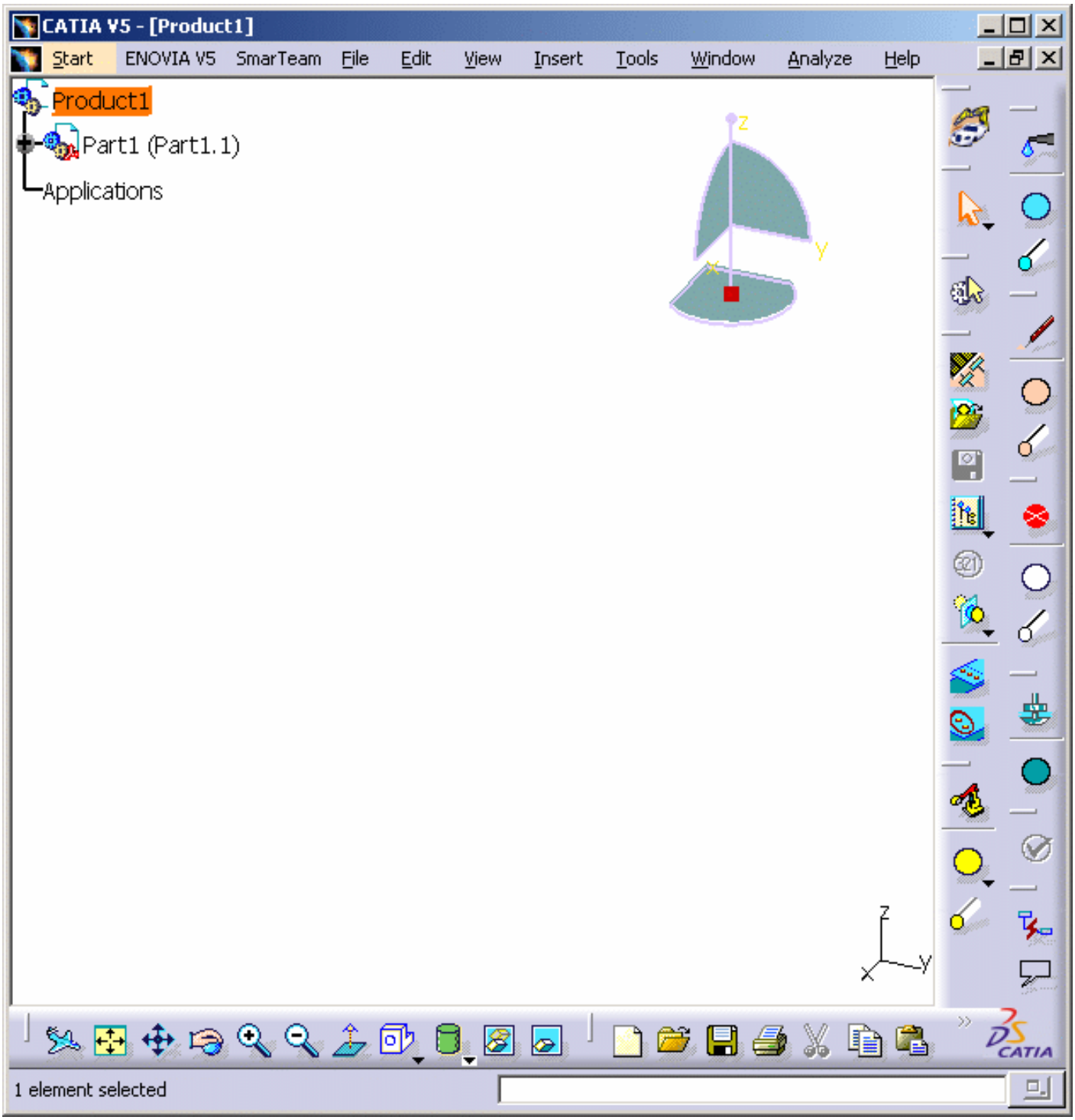
The first task will show you how to enter the Automotive BiW Fastening workbench.

The only pre-requisite for this task is to have a current CATIA V5 session running.

1. Choose **Automotive BiW Fastening** from the **Start -> Shape** menu, or click the Automotive BiW Fastening icon from the Welcome to CATIA V5 dialog box.

   The Automotive BiW Fastening workbench is displayed and ready to use.

The workbench looks like this:



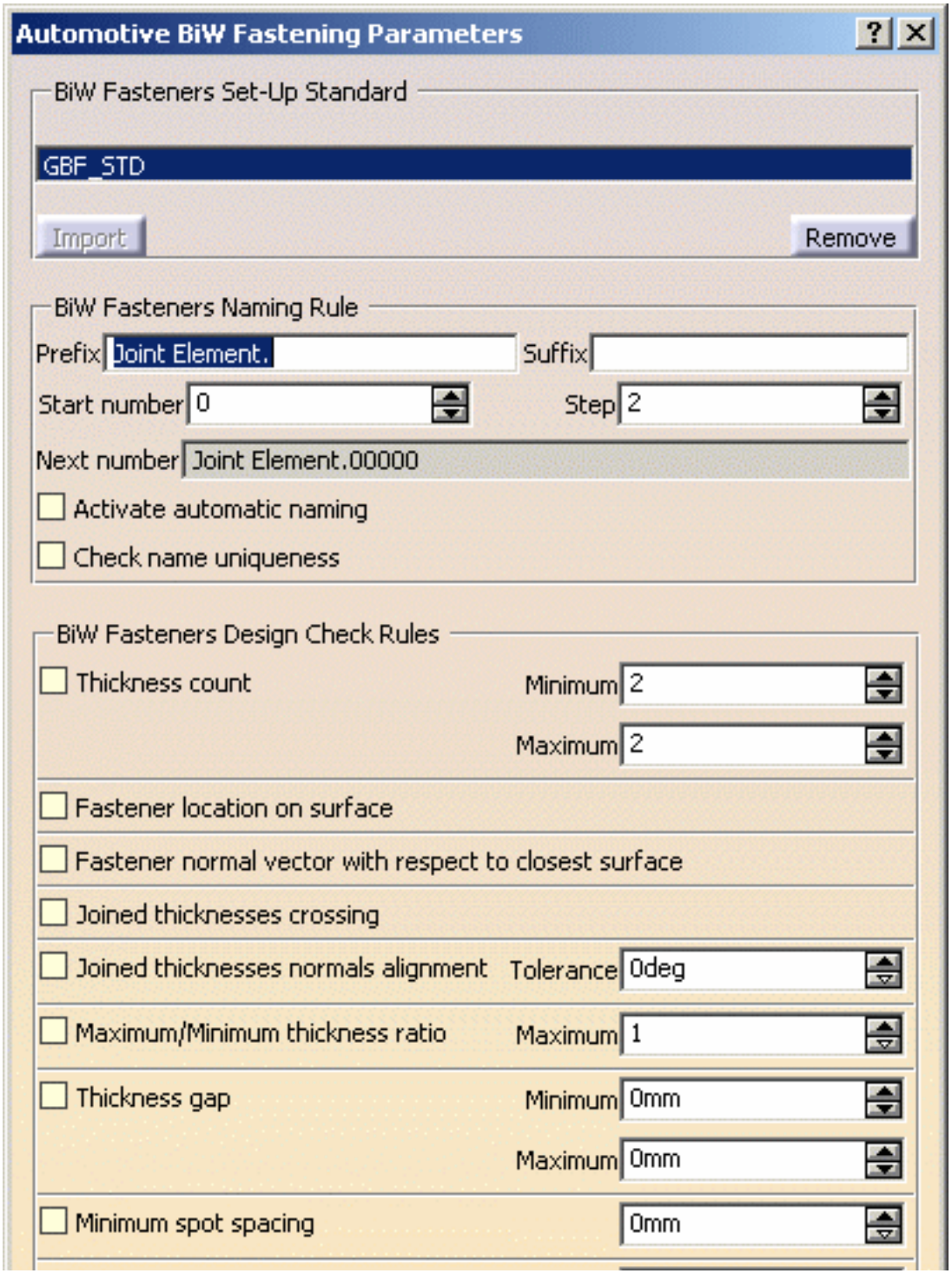
If you wish to use the whole screen space for the geometry, remove the specification tree clicking off the **View -> Specifications Visible** menu item or pressing F3.

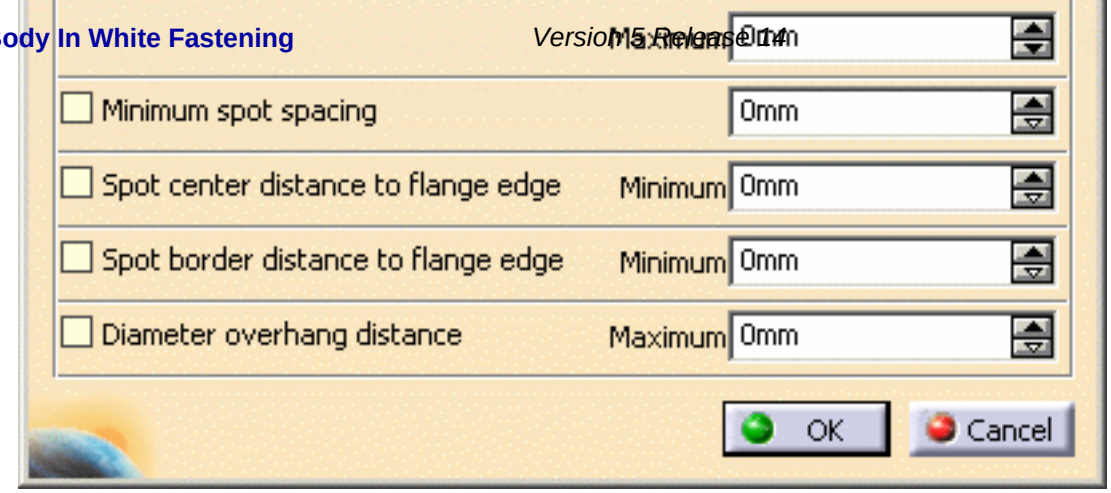
Setting up a CATIA Reference Product with BiW Fastening Parameters

It is essential to define Automotive BiW Fastening parameters in a product in order to create BiW features within this product. This task shows you how to set up the Automotive BiW Fastening parameters in a reference product. Nevertheless, Automotive BiW Fastening parameters are automatically defined when you create BiW Features (BiW Joint, BiW Joint Body BiW Fasteners) in a reference product.

1. Click the **Automotive BiW Fastening Parameters** icon.

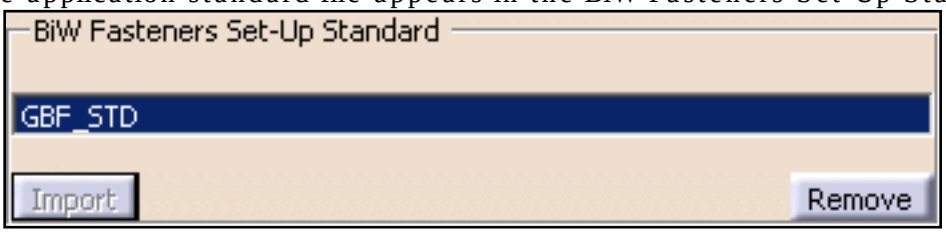
The Automotive BiW Fastening Parameters dialog box is displayed.





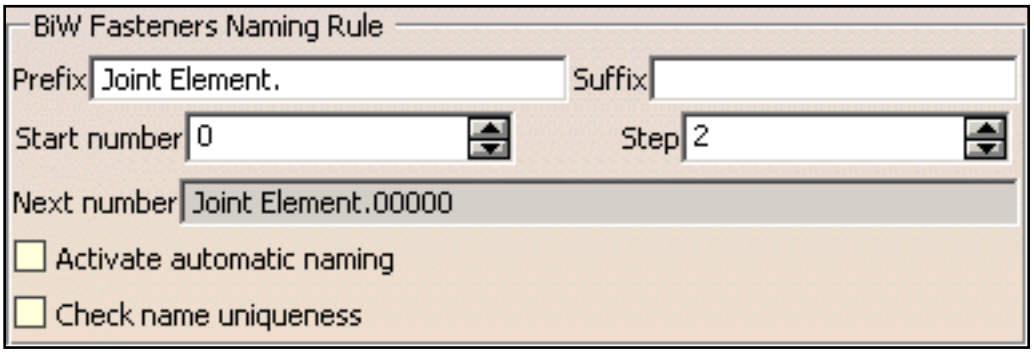
2. Click the **Import** button to import the default standard file.

The application standard file appears in the BiW Fasteners Set-Up Standard window.



You can set up BiW Fastening rules:

Naming Rules



You can define a naming rule to automatically name fasteners that will be further created. The naming rule consists in an alphanumerical prefix followed by a numerical part (the "number"), then by another alphanumerical suffix. The number has a fixed length as defined in the default standard file.

3. Key in the naming rule's **Prefix** string. Each fastener's name will start with this prefix.

   A default prefix, Joint Element. is proposed.

4. Key in the naming rule's **Suffix** string. Each fastener's name will end with this suffix.

Prefix and suffix strings can contain blanks.

5. Define the naming rule's **Start number** using the spinners.

   Modifying this number automatically resets the **Next number** field that shows what the name of the next fastener to be created will be, if the naming rule is active.

6. Define the **Step** to be added to the current number to obtain the next number.

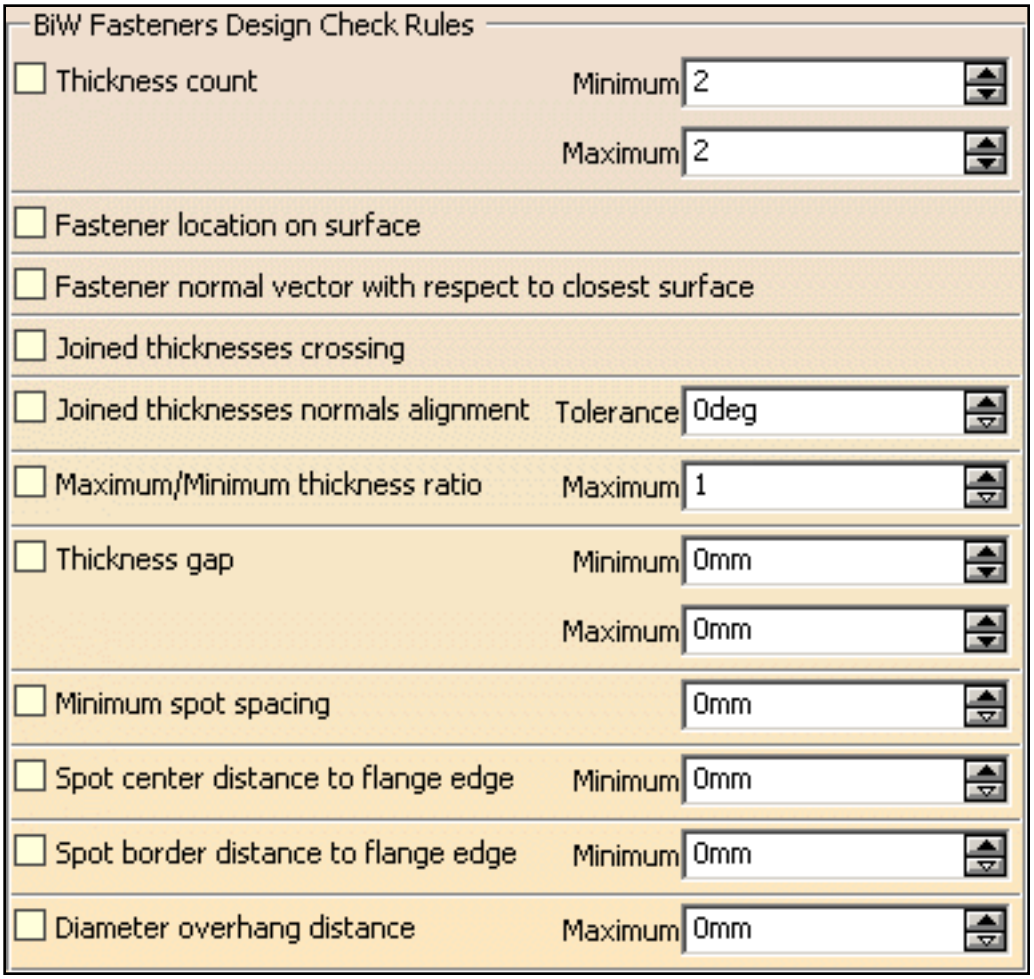
   The modification of the step does not change the value of the **Next number**, but will be effective for the computation of the next one. The Next number field displays the

- The **Activate automatic naming** check box allows to activate the naming rule using the above parameters for all fasteners to be next created.
- The **Check name uniqueness** check box allows to check, each time a fastener is created or modified, that its name is not already defined in the product structure.

  If the name is already given to another fastener, an error message is issued and you are prompted to manually set the ID.

For instance, if the prefix is set to ''My Fastener'', the suffix to ''SSS'', the step to 10 and the start number to 100, the next fastener's name will be: ''My Fastener 01 00100 SSS''.
Note that quotes only show blanks, they will not be part of the resulting fastener's name.

Check Rules



You can define design rules to automatically check parameters at creation or modification time.

- The **Thickness count** option enables to check the number of thicknesses when creating a jointbody.

  This number should be not be greater than the maximum thickness count defined here.

- The **Fastener location on surface** option enables to detect all fasteners that are not located on surfaces.

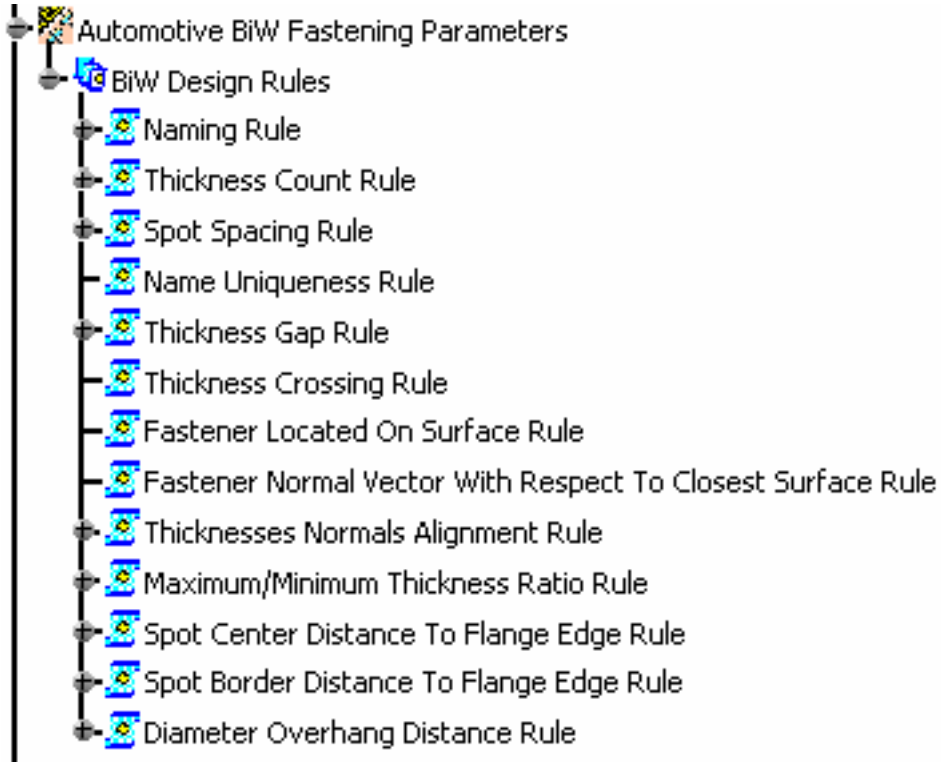
- The **Fastener normal vector with respect to the closest surface** option enables to check the fastener's normal vector and the fastener's normal vector at the projection on the closest surface.

- The **Joined Thicknesses Crossing** option enables to check whether the fastener crosses all the specified thicknesses of the jointbody.

- The **Joined thicknesses normals alignment** option enables to specify an angular accuracy (in degree) between the normal of the joint element and those of all crossed thicknesses.
- The **Maximum/Minimum thickness ratio** option enables to compute the maximum tolerance between the minimum thickness and the maximum thickness.
- The **Thickness gap** option enables to check the gap value between two joined components crossed by a given fastener (this accounts for the thickness defined on each zone).
  This value is included between a minimum and a maximum value defined here.
- The **Minimum spot spacing** option enables to check the minimum spacing between two spotpoints.
  Selecting curves is not taken into account in any case (for instance if nothing is selected).
  This number should be not be less than the minimum spacing defined here.
- The **Spot center distance to flange edge** option enables to specify a minimum distance (in mm) from the fastener's center location to the closest flange edge.
- The **Spot border distance to flange edge** option enables to specify a minimum distance (in mm) from the fastener's border location to the closest flange edge.
- The **Diameter overhang distance** option enables to specify a maximum overhang distance (in mm) of the fastener's diameter to the closest flange edge.

If one of the above rules fails, an error message is issued and the joint element cannot be created.

7. Click **OK** to validate the parameters and close the dialog box.
   The feature is added in the specification tree.
   Had you activated all the rules, the specification tree looks like that:



- You can edit the rules by double-clicking on one of them in the specification tree: the Automotive BiW Fastening Parameters opens.
- If you expand the tree (for the Naming Rules for instance), you can edit the parameters associated to the rule by double-clicking them.

Basic Tasks

Creating Joints
Creating Joint Bodies
Creating Spot Fasteners
Displaying Process Category Parameters
Creating Curvebeads
Using Tools
Reporting
Exporting
Importing
Isolating Features
Transforming
Copying/Pasting elements
Renaming Features
Editing Fasteners' Properties

Creating Joints

This task enables to create a joint, that is a set of two or more parts, then edit it.

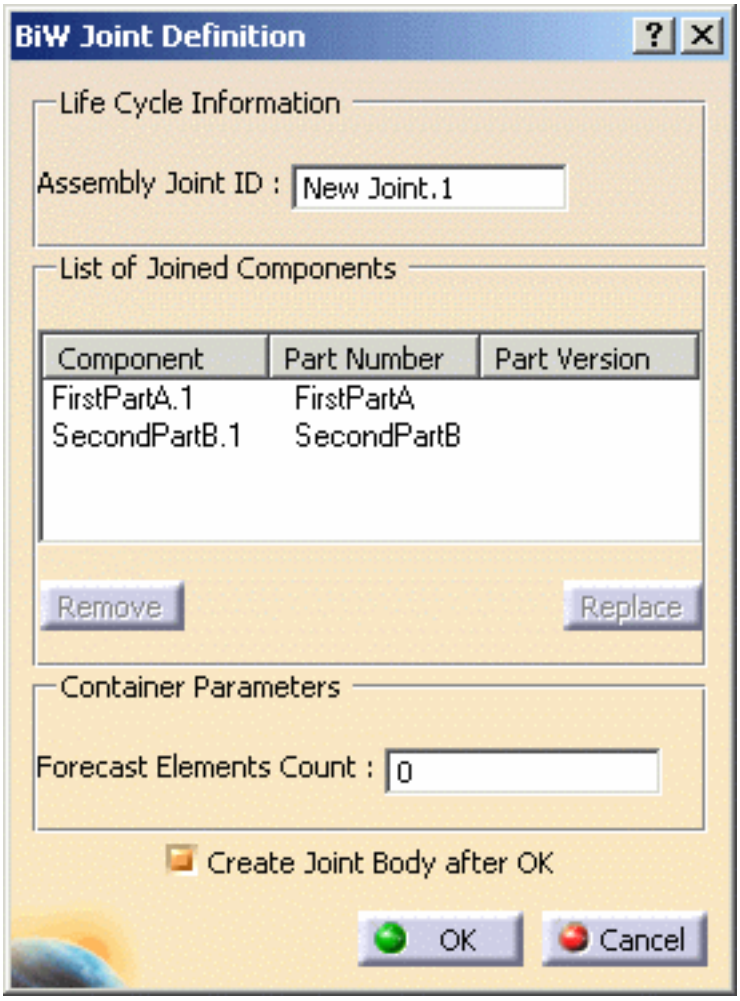
Open the Joint1.CATProduct document.

There are several ways to create a joint:

- select the components then click the **BiW Joint** icon.
- click **BiW Joint** icon and select the components (this is the example for our scenario).

1. Click the **BiW Joint** icon from the **Automotive BiW Fasteners** toolbar.

The BiW Joint Definition dialog box is displayed.



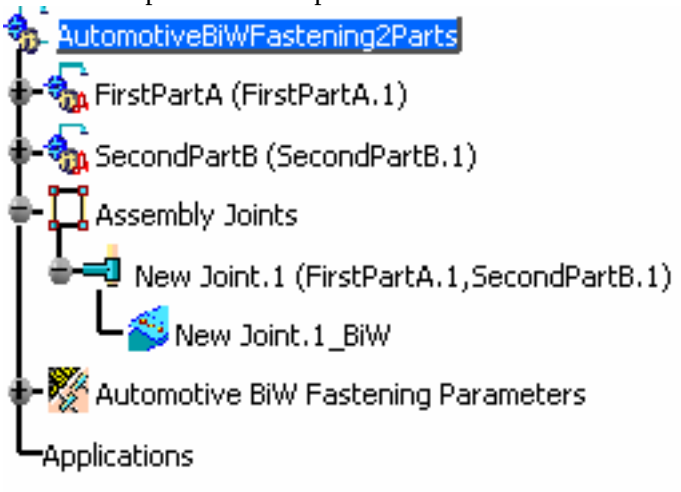
2. Give a new name for the Assembly Joint, "New Joint.1" for example.
3. Select First part as the first component to join.
4. Select Second part as the second component to join.

The list displays the name of these components as well as their part number. The Part Version field shows possible revision numbers.

5. Click OK to create the joint.

- The **Forecast Elements Count** field allows you to define a number of fasteners this joint is supposed to contain for further verifications.
- The **Create Joint Body after OK** button is automatically selected. Deselect it to only create the joint.

"New Joint.1" appears as an entity below the Assembly node in the specification tree. The name of the parts it comprises is indicated between brackets.

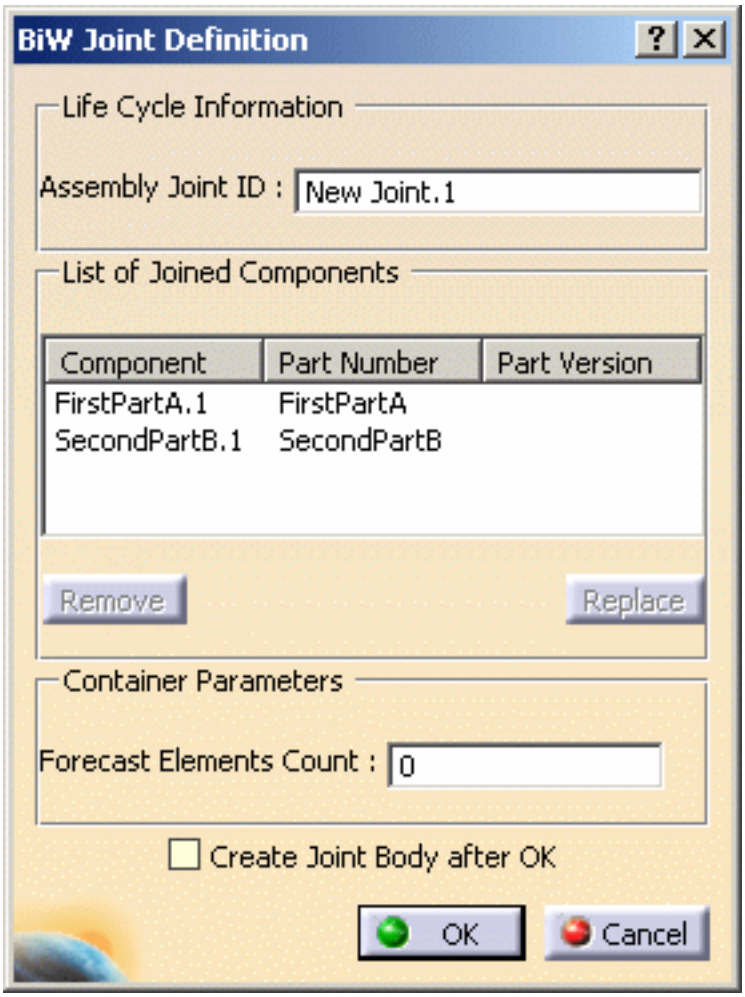


Removing a Component

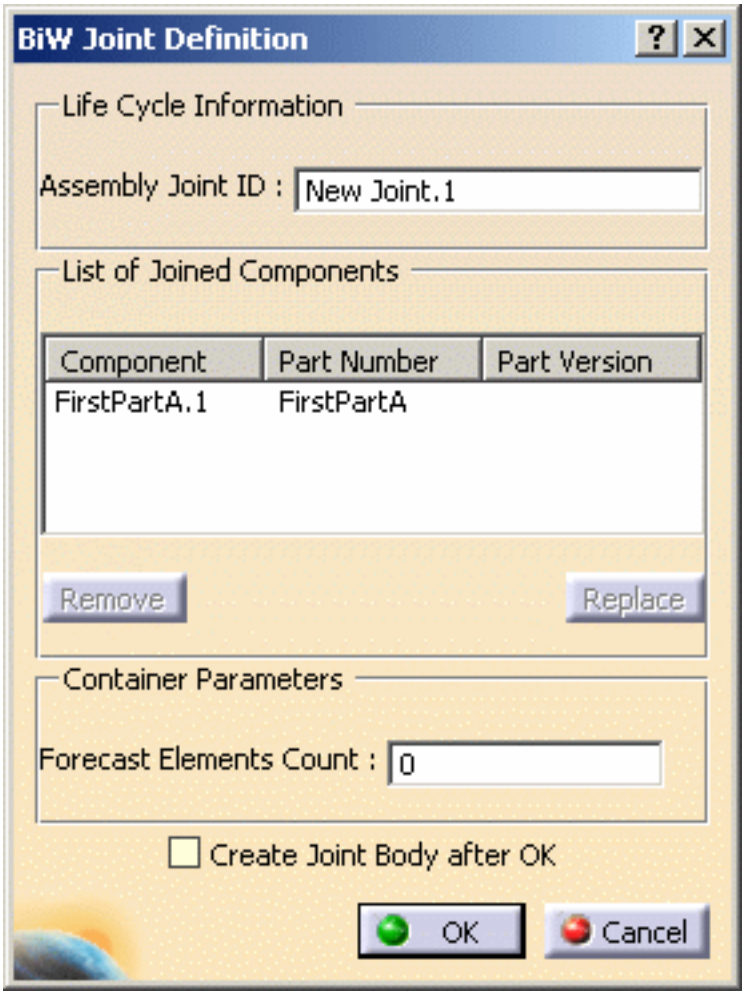
1. Double-click the New Joint.1 from the specification tree.

The **BiW Joint Definition** dialog box is displayed.

2. Select the component you wish to remove.
3. Click the **Remove** button.
4. Click OK in the BiW Joint Definition dialog box.



The selected component is deleted from the BiW Joint Definition dialog box.



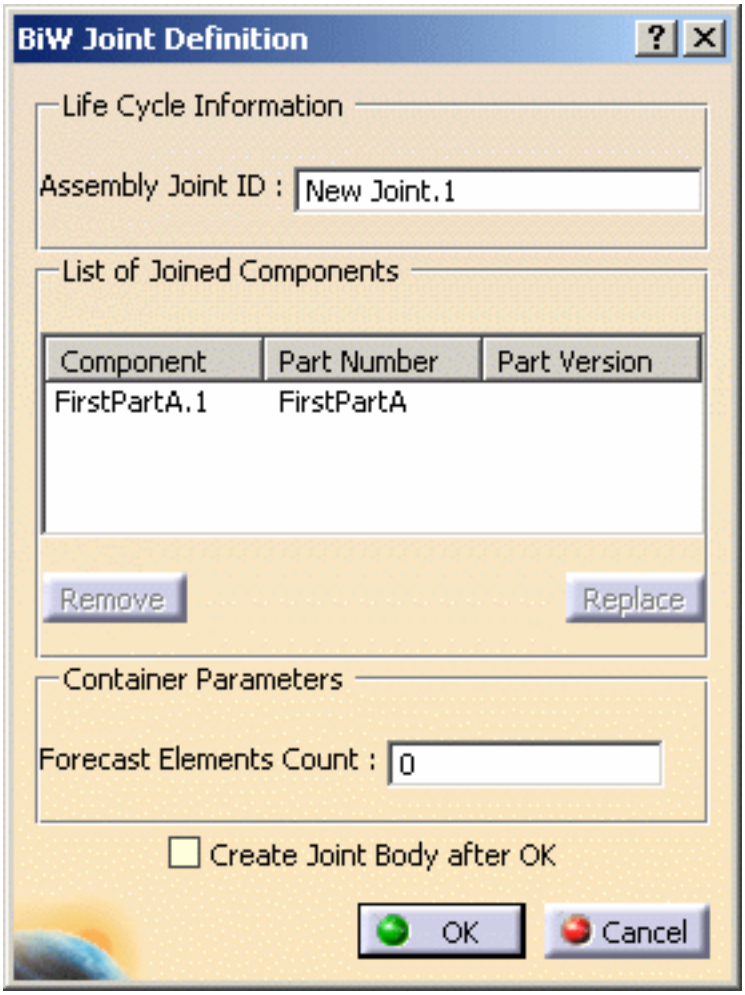
Replacing a Component

1. Double-click the New Joint.1 from the specification tree.

   The BiW Joint Definition dialog box is displayed.

2. Select the component you wish to replace.

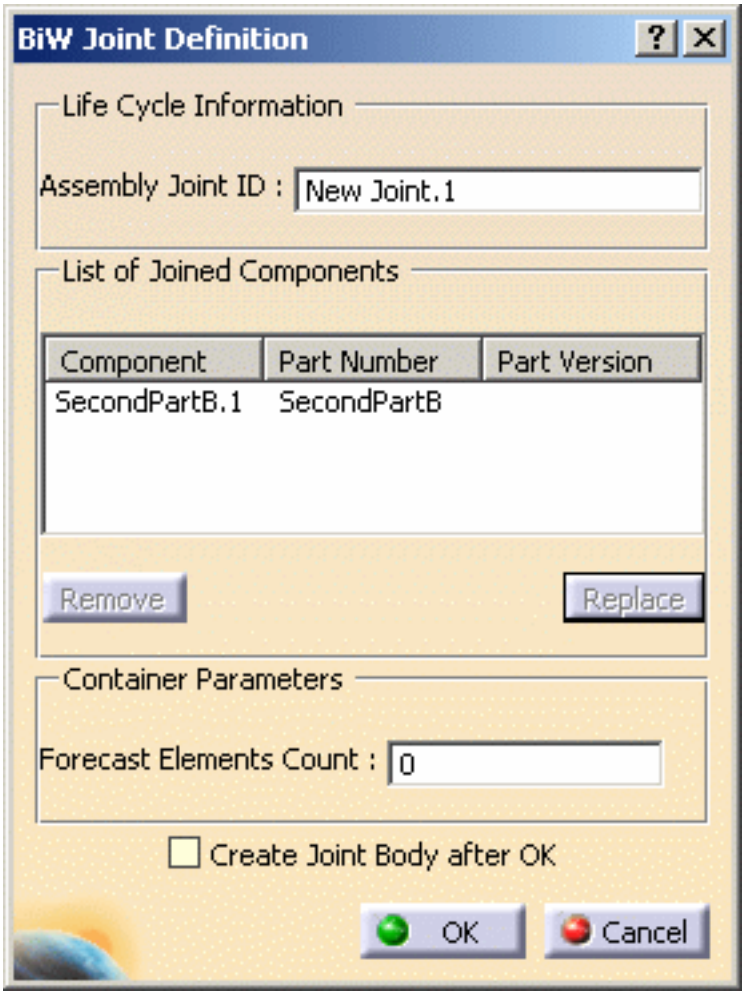
3. Click the **Replace** button.



4. Select the new component either from the geometry area or the specification tree.

   The part component has been updated in the dialog box.

5. Click OK in the BiW Joint Definition dialog box.



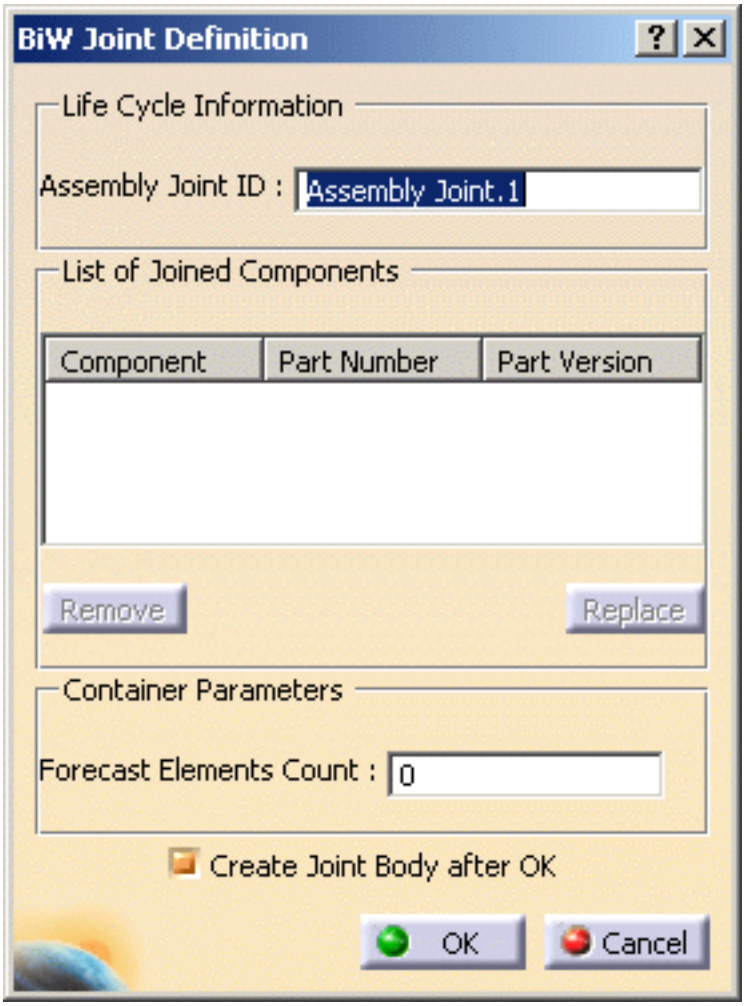
Creating a joint with no components

Open the Joint1.CATProduct document.

1. Click the **BiW Joint** icon from the Automotive BiW Fasteners toolbar.

The BiW Joint Definition dialog box is displayed.

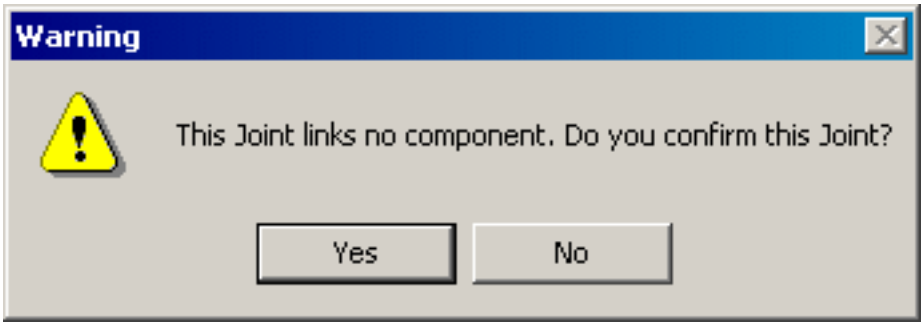
2. Click OK to create the joint.



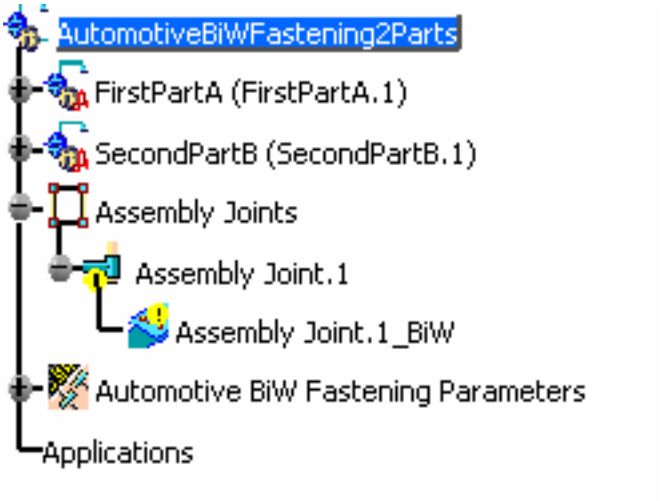
A Warning dialog box is displayed.

3. Click Yes in the Warning dialog box to confirm the operation.

When creating a duplicated Joint, the same Warning dialog box appears.



- "Assembly Joint.1" appears as an entity below the Assembly node in the specification tree.
- A mask is displayed on its icon to identify that it is not valid.

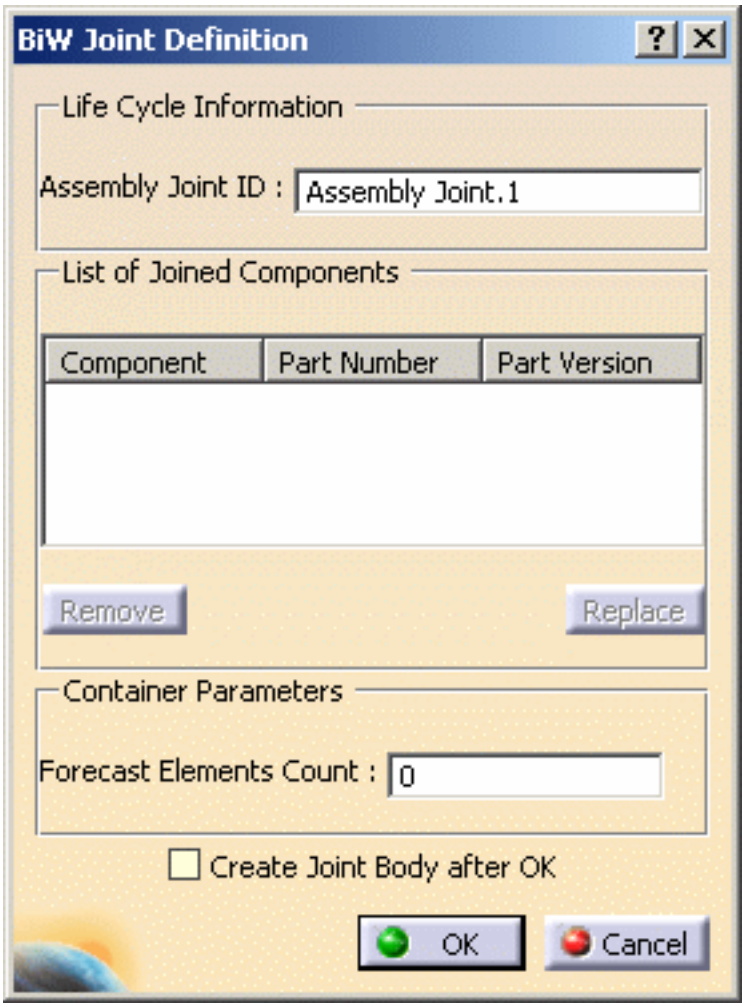


Adding Components

Create a joint with no components for instance.

1. Double-click the Assembly joint.1 from the specification tree.

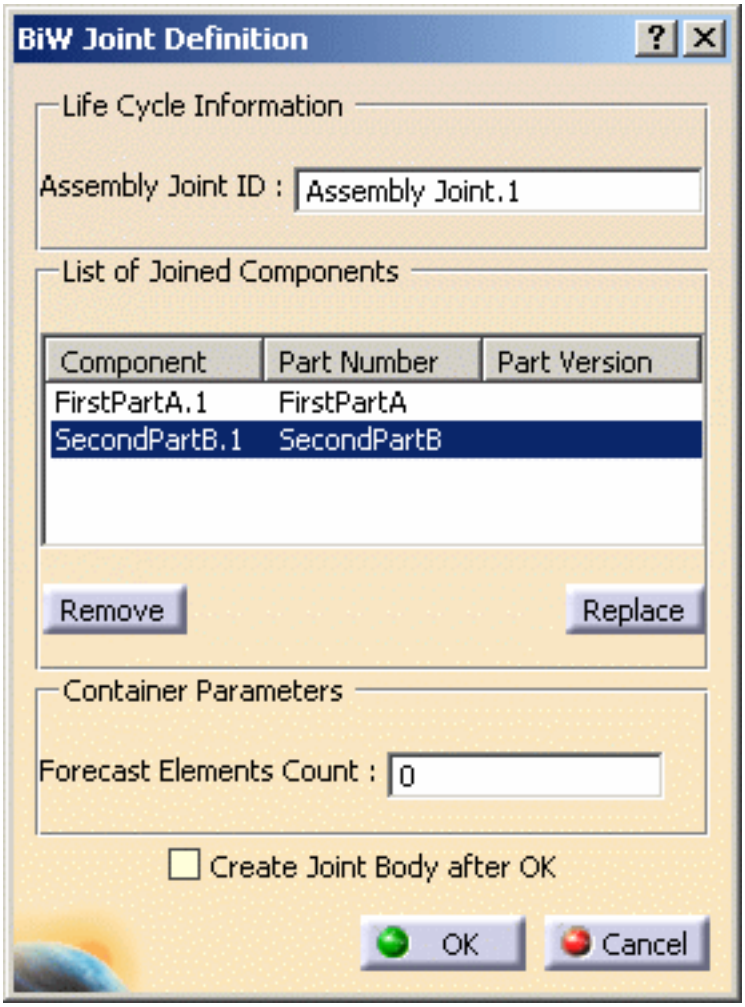
The BiW Joint Definition dialog box is displayed.



2. Select the components you wish to add, for instance FirstPartA and SecondPartB either from the geometry or the specification tree.

The selected components are automatically displayed in the dialog box.

3. Click OK to create the joint.



Creating Joint Bodies

This task shows how to create a joint body, that is a container for joint elements.

Open the Joint1.CATProduct.

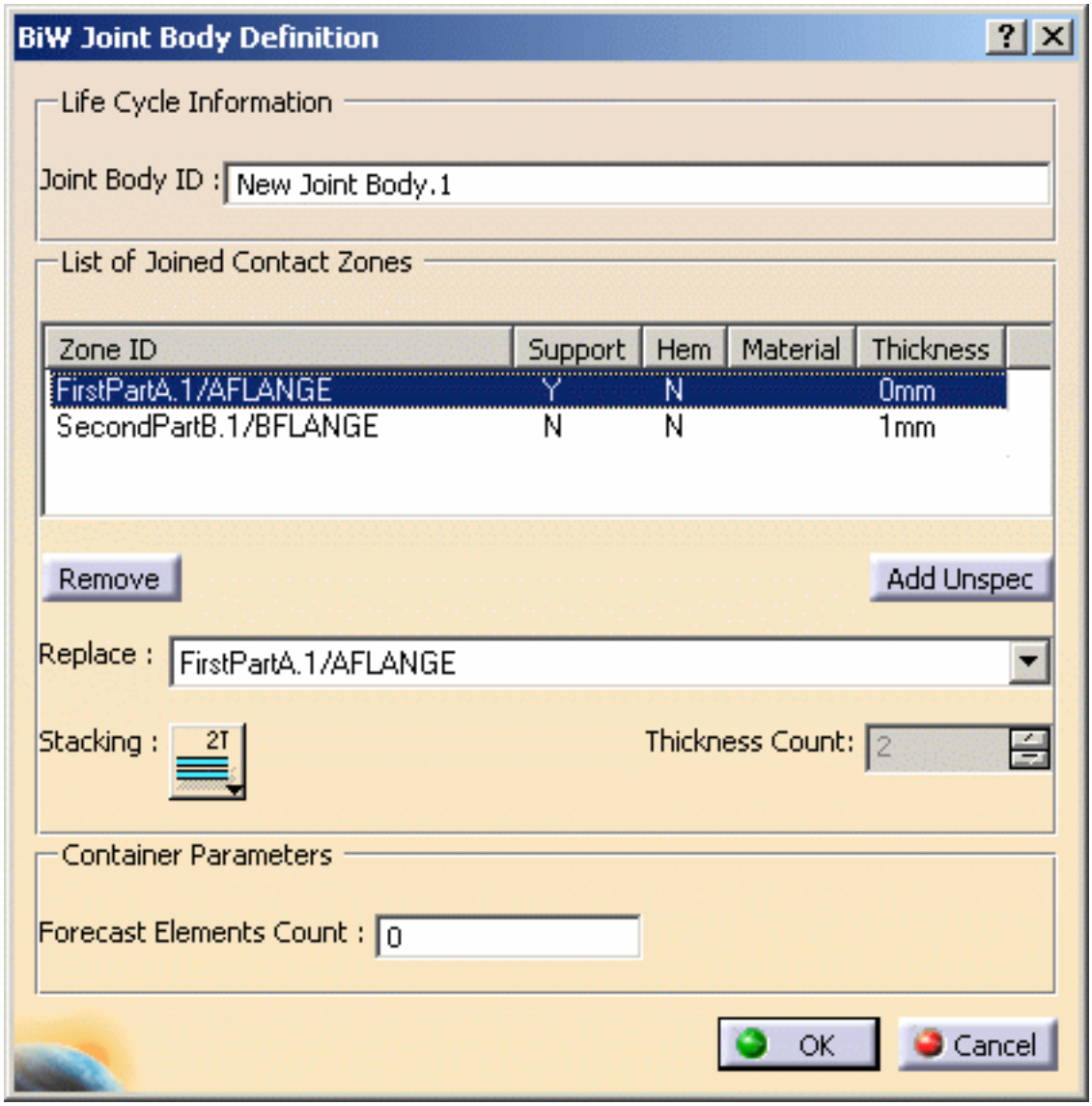
There are several ways to create joint bodies:

- select the publications, and click the **Joint Body** icon  . If no joint was created previously, it is automatically created; it links the corresponding components. The joint body links the selected publications.
- select the components, and click the **Joint Body** icon  . If no joint was created previously, it is automatically created; it links the corresponding components. The connected zones of this joint body are not specified.
- a joint is already created: select it in the specification tree, and click the **Joint Body** icon  .
- a joint is already created. Click the **Joint Body** icon  and select the joint in the specification tree.
- during the creation of a joint, if the **Create Joint Body after OK** button is selected, when you click OK, the BiW Joint Body Definition dialog box automatically displays (this is the example for our scenario).

1. Click OK in the BiW Joint definition dialog box.

The Joint Body Definition dialog box opens.

2. Give a new name for the Joint Body, "New Joint Body.1" for example.



The List of joined Contact Zones display the Zone ID.

4. Define whether the zone is the **Support** zone.

5. Define whether the zone is a **Hem**.

The material and the thickness can be defined using the **Thin Parts Attribute** command in the Generative Shape Design workbench.
Please refer to the *Applying a Thickness* chapter in the Generative Shape Design documentation.

6. Define the stacking and the thickness count depending on the number of joined zones.

You can click the down arrow to display the stacking toolbar and choose the stacking type:

- Lap
- Hem
- Unspecified: you are able to modify the thickness count using the spinners, depending on the number of joined zones

The Forecast Elements Count field allows you to define a number of fasteners this joint body is supposed to contain for further verifications.

7. Click OK to create the joint body.

"New Joint Body.1" appears below "New Joint.1" (created in the previous task) in the specification tree.

When creating a Joint Body with one zone, a second one is automatically created. You can edit the second zone and delete it if needed.

Managing multi-zones, whether published or not, is possible: one joint body can join several zones for a same component.

Removing a Zone

1. Double-click the New Joint Body.1 from the specification tree.

   The BiW Joint Body Definition dialog box is displayed.

2. Select the zone you wish to remove.
3. Click the **Remove** button to remove this zone.

   A zone should be first selected in the list, and one zone at least must be connected for a linked component. If not, the zone can not be removed.

4. Click OK in the BiW Joint Definition dialog box.

Adding an Unspecified Zone

1. Double-click the New Joint Body.1 from the specification tree.

   The BiW Joint Body Definition dialog box is displayed.

2. Click the **Add Unspec** button to add an unspecified zone in the zones list for the selected corresponding component.

   A zone should be first selected in the list.

Replacing a Zone

1. Double-click the New Joint Body.1 from the specification tree.

   The BiW Joint Body Definition dialog box is displayed.

2. Select a zone to replace by another one, either by:
   - selecting a zone published by the corresponding component in the specification tree or in the 3D geometry, or
   - selecting the name of the zone in the combo list.

Creating Spot Fasteners

Creating Welding SpotPoints
Creating Adhesive SpotPoints
Creating Sealant SpotPoints
Creating BiW Mechanical SpotPoints
Creating Unspecified SpotPoints
Creating SpotProjection Welds
Creating Multi-Selected Spots
Repeating SpotPoints

When working with spot fasteners, it is possible to:

- valuate spot fasteners' attributes of type ''Dimension'' from a measure. However these dimensions are not associative, that is to say they are not Knowledge-integrated.

- create and associate an annotation to a spot fastener.
  For further information, refer to the *3D Functional Tolerancing and Annotation* documentation.

- assign a layer to a spot fastener and published joined contact zones. A layer number is assigned to each published zone.
  For further information, refer to the Using the Graphic Properties Toolbar chapter in the *CATIA Infrastructure User Guide*.

- apply visualization filters, that is a group of layers, to visualize (or not) only the spot fasteners located on the layers in the filter.
  For further information, refer to the Using Visualization Filters chapter in the *CATIA Infrastructure User Guide*.

Creating Welding SpotPoints

This task shows how to create a welding spotpoint.

There are several ways to create a spotpoint:

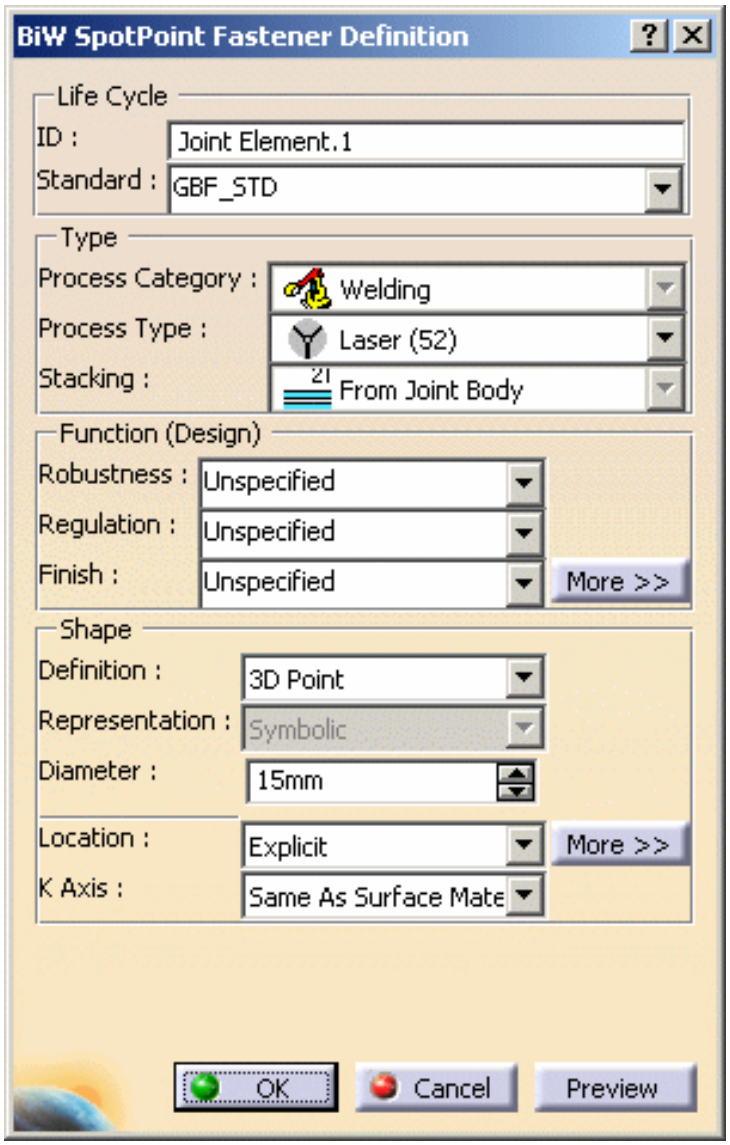
- a joint and a jointbody are already created: click the desired icon and select the jointbody in the selection tree.
- a joint and a jointbody are already created: select the jointbody in the selection tree and click the desired icon (here is the example used for our scenario).
- no jointbody is created. Select the components or the publications, and click the desired icon: a joint and a jointbody are created if needed.

Open the SpotPoint1.CATProduct document.

Make sure the SpotPoint Fastener Type is set up in the standard file.

1. Select the jointbody in the specification tree.
2. Click the **BiW Welding SpotPoint**  icon from the Welding toolbar.

The BiW SpotPoint Fastener Definition panel opens.



3. Specify whether you wish to use the existing standard or not.

   If a standard has been imported, a spotpoint is created using this standard. If not, you are able to define your own values for each attribute.

If the last location method is different from **Explicit**, the ABF application creates a specification part associated to the Assembly Joint if this specification part does not already exist.

4. Define the following parameters:

- Type
- Functional Parameters
- Shape
- Location
- K Axis

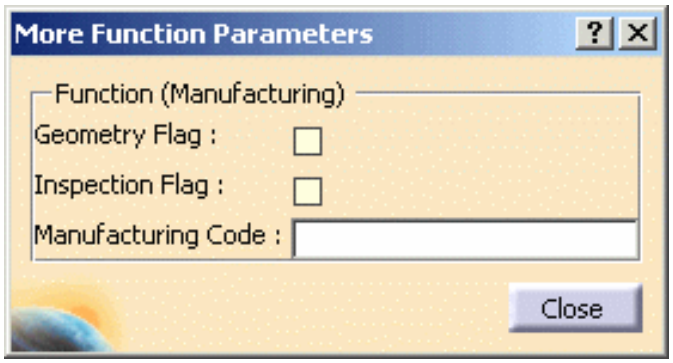
Type

5. Select the process type.

Functional Parameters

6. Define the design parameters:

- Robustness
- Regulation
- Finish
- **More>>:** allows you to define the Manufacturing parameters.



Shape

7. Specify the material (if neccessary).
8. Specify the shape of the spotpoint.
9. Define its diameter.

Location

10. Select the location:

- On Surface
- On Point
- From Curve on Surface
- Along Curve
- Intersect
- Bitangent Circle Center
- Explicit

On Surface

1. Select a surface or a point to indicate the spotpoint location.

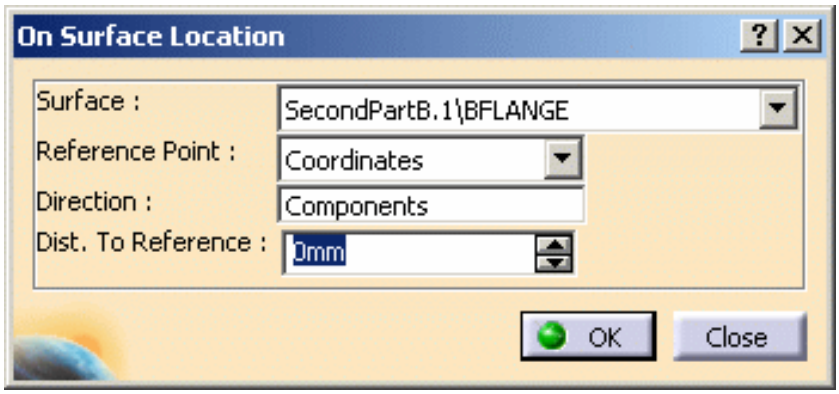
   You can as well select a circle instead of a point: the center of the circle is computed to get the reference point.

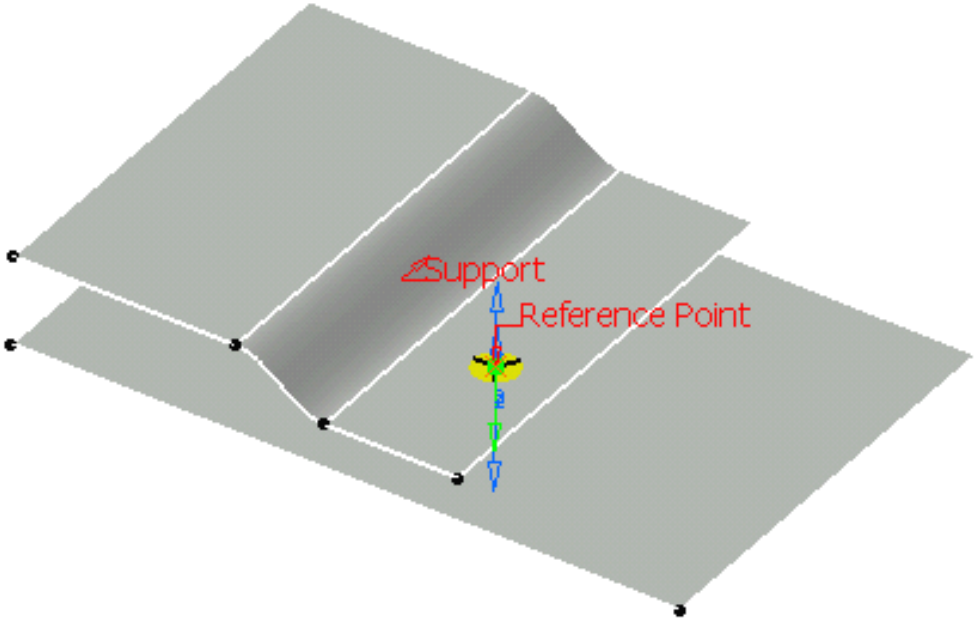
   The spotpoint will be created on the selected zone, which may not be a support zone of the joint body.

2. Optionally select a direction.

3. Click **More>>** to:

   - display the Reference Point,
   - display the Direction
   - optionally modify the distance to the reference point.



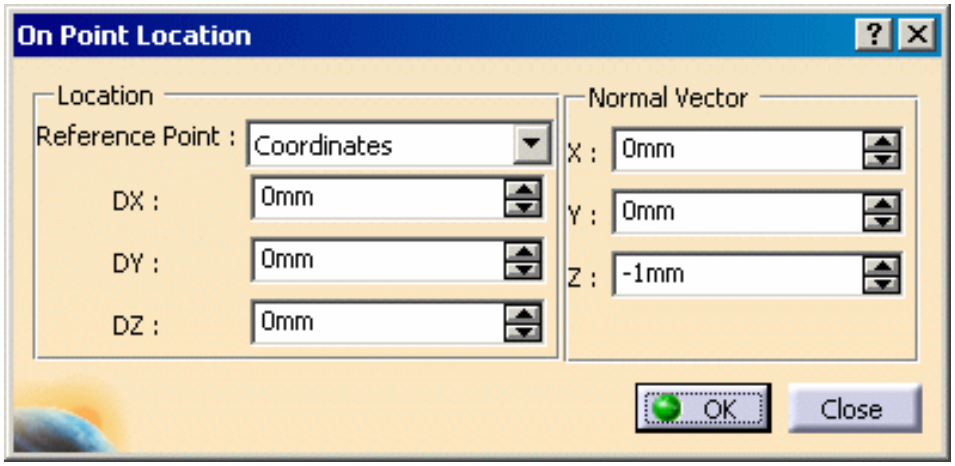


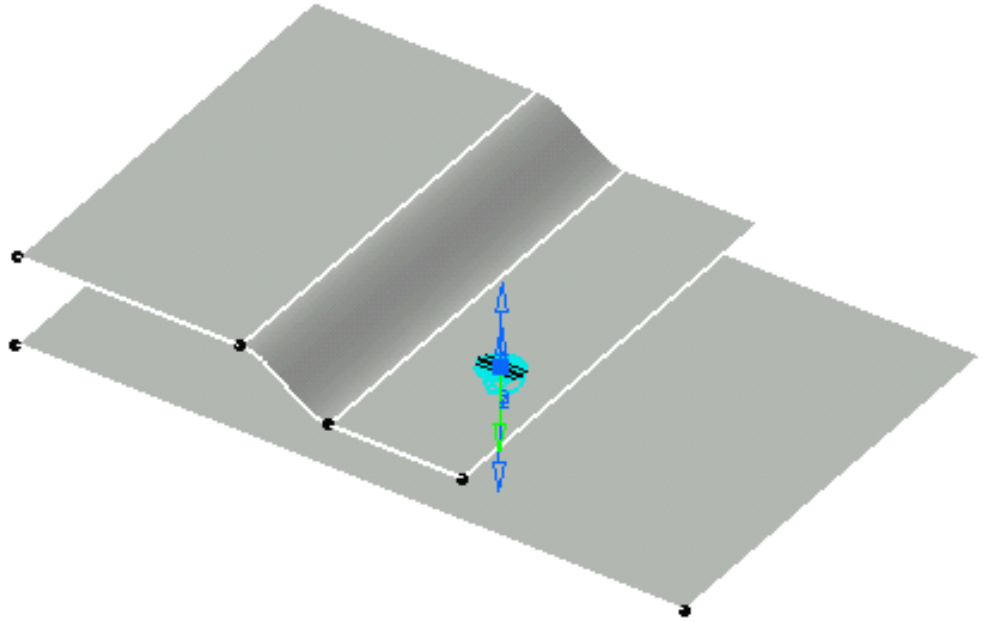
On Point

1. Select a point.

   You can as well select a circle instead of a point: the center of the circle is computed to get the reference point.

2. Click **More>>** to:
   - optionally modify the reference point type (coordinates or axis origin)
   - optionally modify the distance to the reference point
   - optionally modify the normal vector if the reference point do not lie on the surface.





From Curve On Surface

1. Select a curve lying on a surface.

   Most of the times, a surface is selected by default.

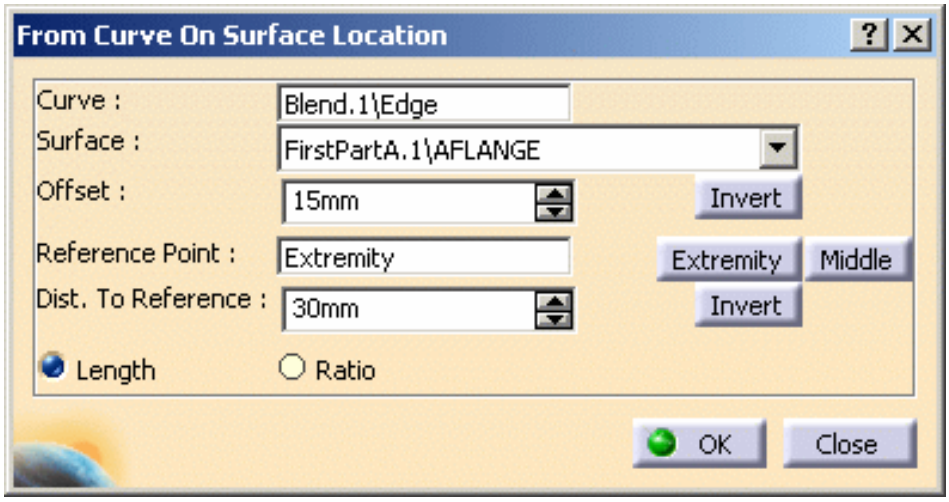
2. If needed, select a surface.

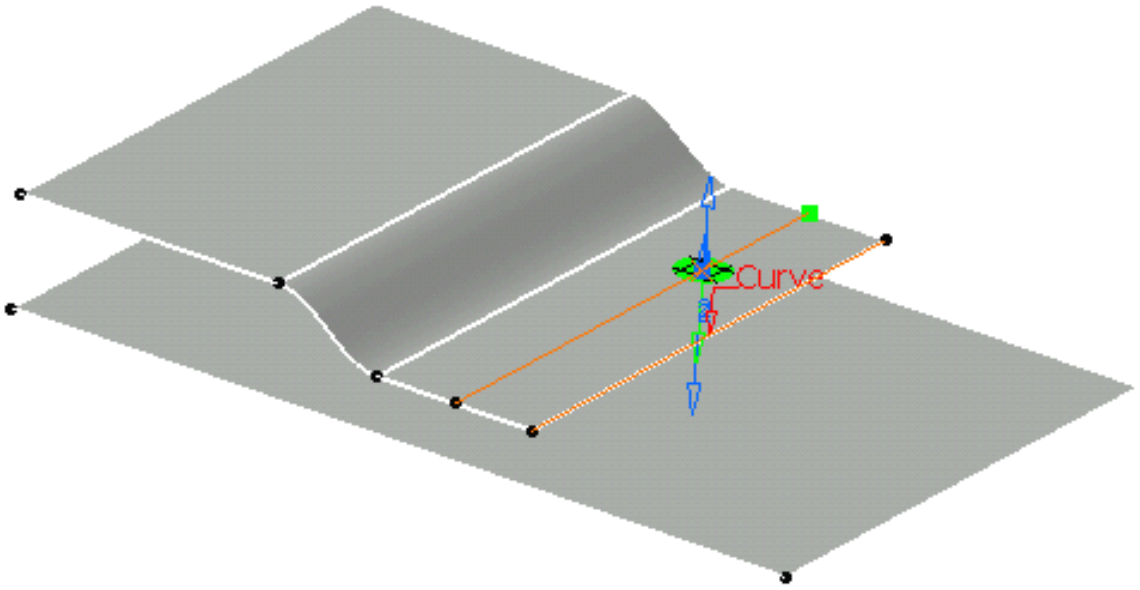
You can select a published surface from the drop-down list or choose No Selection if the surface to be selected cannot be found and select the desired surface in the 3D geometry.

3. Modify the offset.
4. Modify the Reference Point (extremity or middle of the curve).
5. Modify the distance to the reference point.
6. Define the distance type (length or ratio)

Note that steps 2 to 6 are optional.

For instance, define 15mm as the offset, select the extremity as the reference point and choose a distance of 30mm from the reference point.



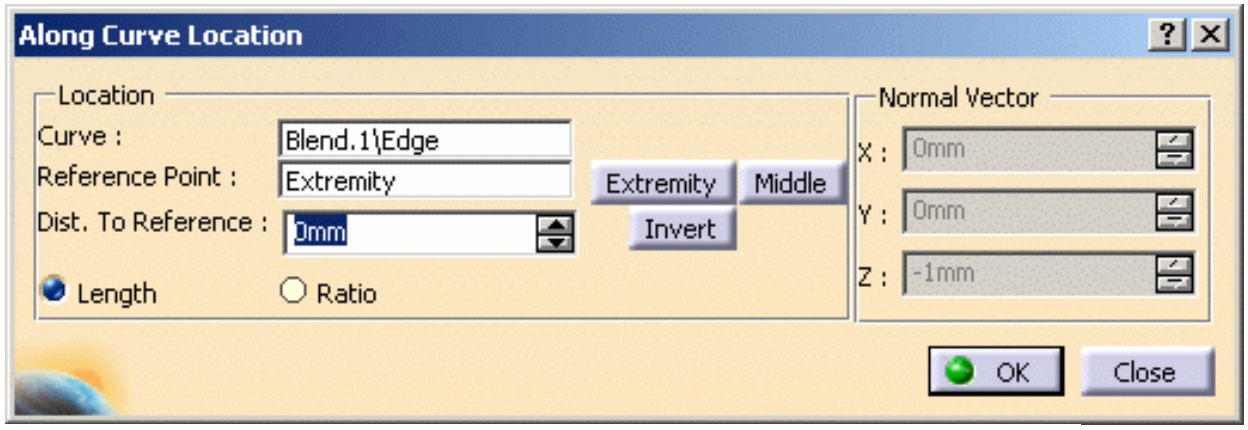


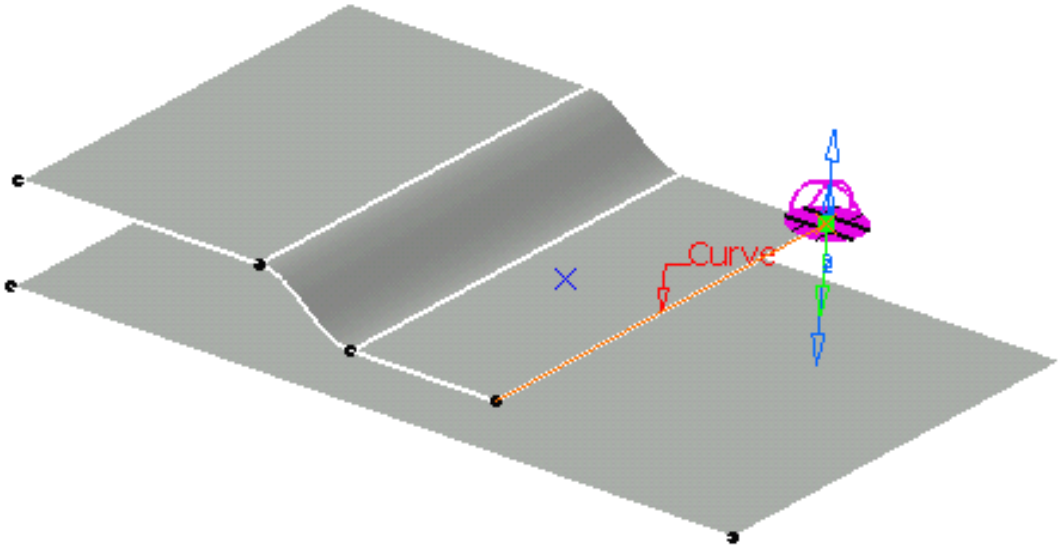
The **Repeat object after OK** button appears in the BiW SpotPoint Fastener Definition. Refer to the Repeating Spots to create more spotpoints using the currently created spot point as reference.

Along Curve

1. Select a curve.
2. Select the reference point (extremity or middle of the curve)
3. Modify the distance to the reference point.
4. Define the distance type (length or ratio)
5. Modify the normal vector if the reference point does not lie on the surface.

Note that steps 2 to 5 are optional.





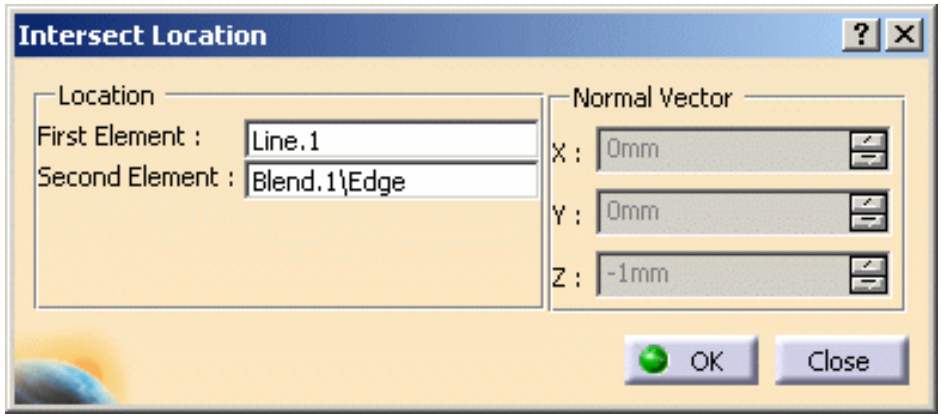
The **Repeat object after OK** button appears in the BiW SpotPoint Fastener Definition. Refer to the Repeating Spots to create more spotpoints using the currently created spot point as reference.

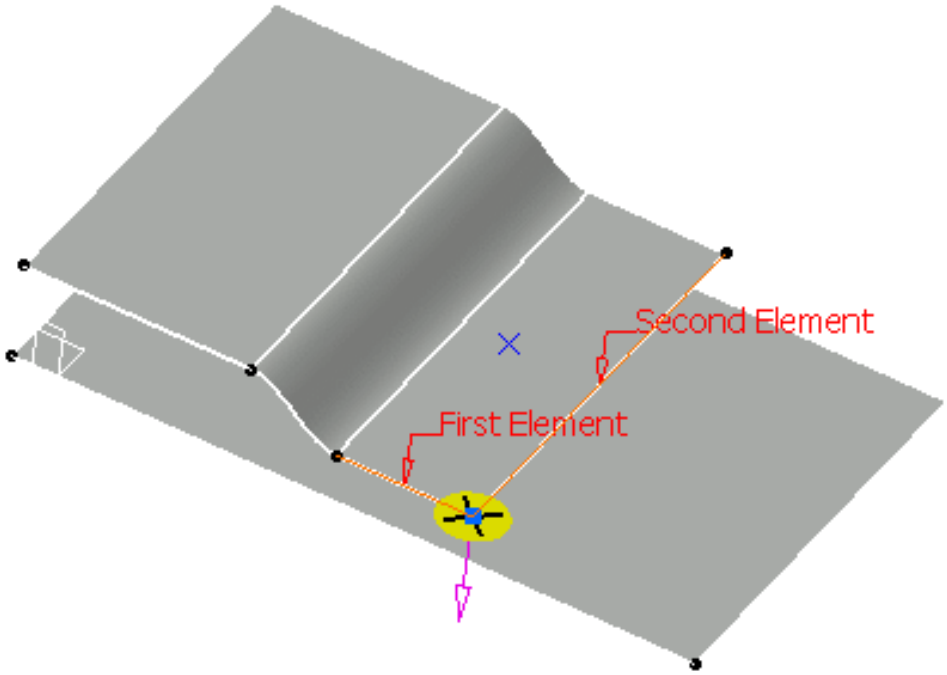
Intersect

1. Select the first and the second element.

   Elements can either be two curves or a curve and a surface.

2. Click **More>>** to optionally modify the normal vector if the reference elements do not lie on a surface.



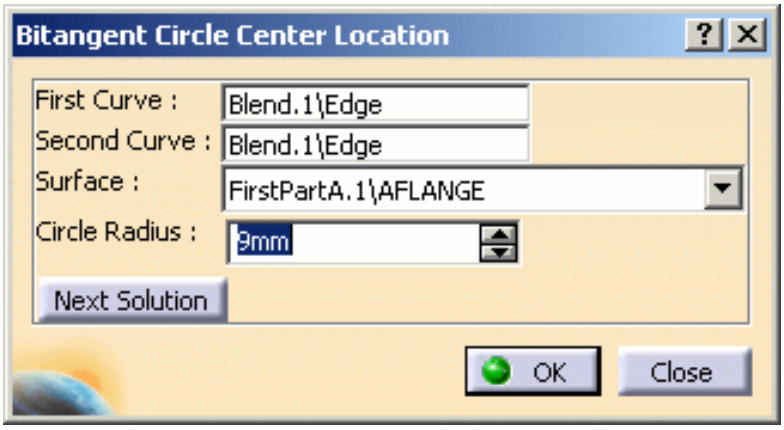


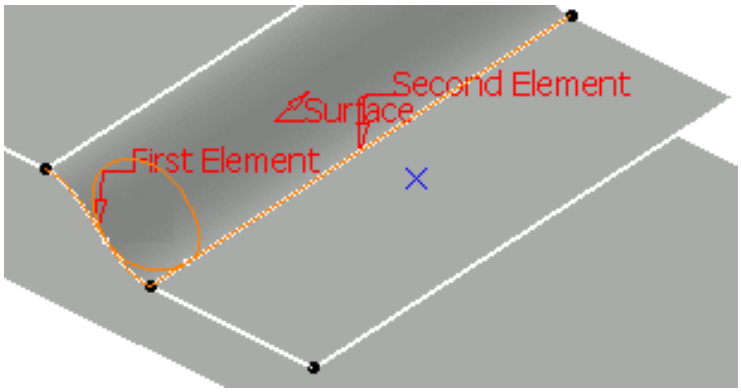
Bitangent Circle Center

1. Select the first and the second curves.

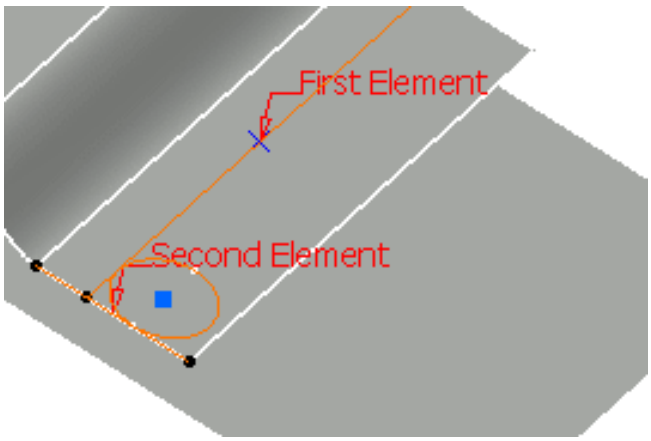
   They must not be parallel.

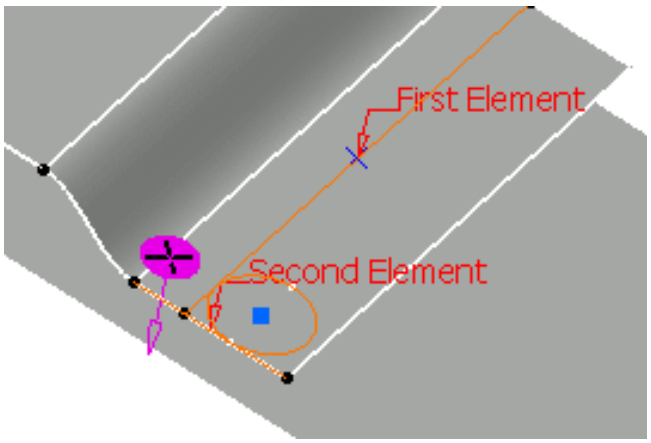
2. Click **More>>** to:
   - optionally select a support surface in the drop-down list or in the 3D geometry if the curves are not coplanar. In this case, the center of gravity of the circle is automatically computed.
   - define the circle radius
   - optionally modify the normal vector if the reference elements do not lie on a surface.





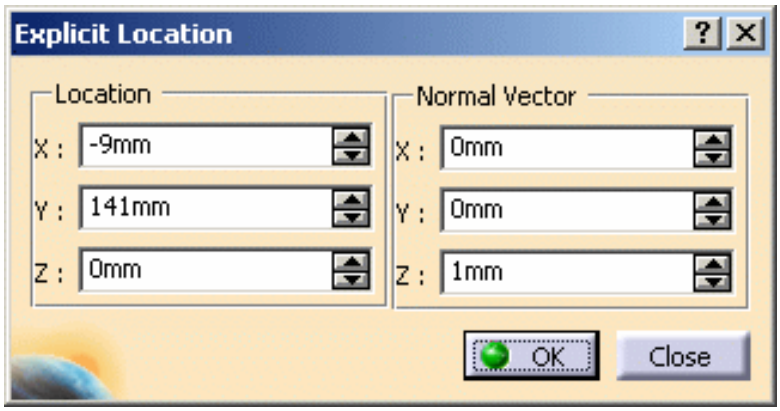
When several solutions are possible, click the **Next Solution** to move to another solution. When you click OK in the Bitangent Circle Center Location dialog box, the chosen spotpoint is created, and the next solution is proposed for creation. Click OK to create the second spotpoint, and so on until the last solution. You can click Cancel in the BiW SpotPoint Fastener Definition dialog box to only create one spotpoint or not to create any more spotpoint.



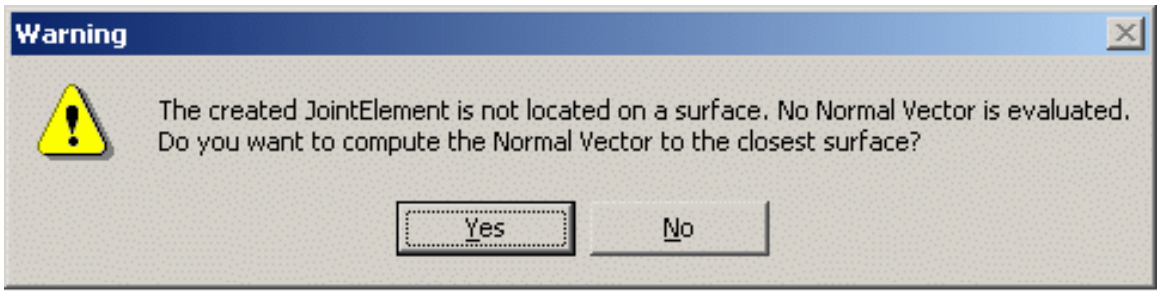


Explicit

1. Select a surface or a point where the spotpoint is to be created.
   You can as well select a circle instead of a point: the center of the circle is computed to get the reference point.
2. Click **More>>** to:
   - optionally modify the spotpoint location
   - optionally modify the normal vector if the reference point do not lie on the surface.

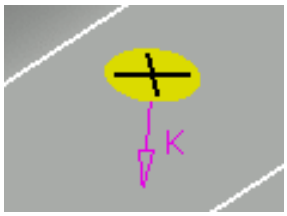


- When the location method switches to Explicit, the specification part is automatically deleted if it is empty.
- The fastener normal vector is automatically calculated if the fastener is based on a surface. Otherwise, the normal parameters need to be defined in the Normal Definition dialog box. In this case, if you do not manually define the fastener normal vector and validate this Joint Element, a warning panel is issued, asking you whether you wish to calculate it or not.



K Axis

11. Define the orientation of the spotpoint by setting the **K axis** to either:
    - **Same as Surface Material**
    - **Opposite to Surface Material**



To visualize the K axis, the Fastening Axis Orientation Show option should be checked in **Tools -> Options**.

12. Click OK to create the spotpoint.

    The spotpoint (identified as Joint Element.xxx) is added to the specification tree, under the Joint Body node.

Note that:

- the selection of geometrical elements must be done within the linked parts, except when using such options as Datum mode and Explicit locations.
- when selecting a geometric specification (curve or point), it is recommended to select published elements in order to guaranty associativity between elements.
- to authorize the selection of only published elements, check the following option using **Tools -> Options -> Infrastructure -> Part Infrastructure -> General -> Only use published elements for external selection keeping links**.
- when the **On Surface** and **Explicit** methods are activated, the application will ignore the active Part Infrastructure setting **Only use published elements for external selection** and will enable the usage of non published external geometry.

Creating Adhesive SpotPoints

This task shows how to create a Body in White Adhesive SpotPoint.

There are several ways to create a spotpoint:

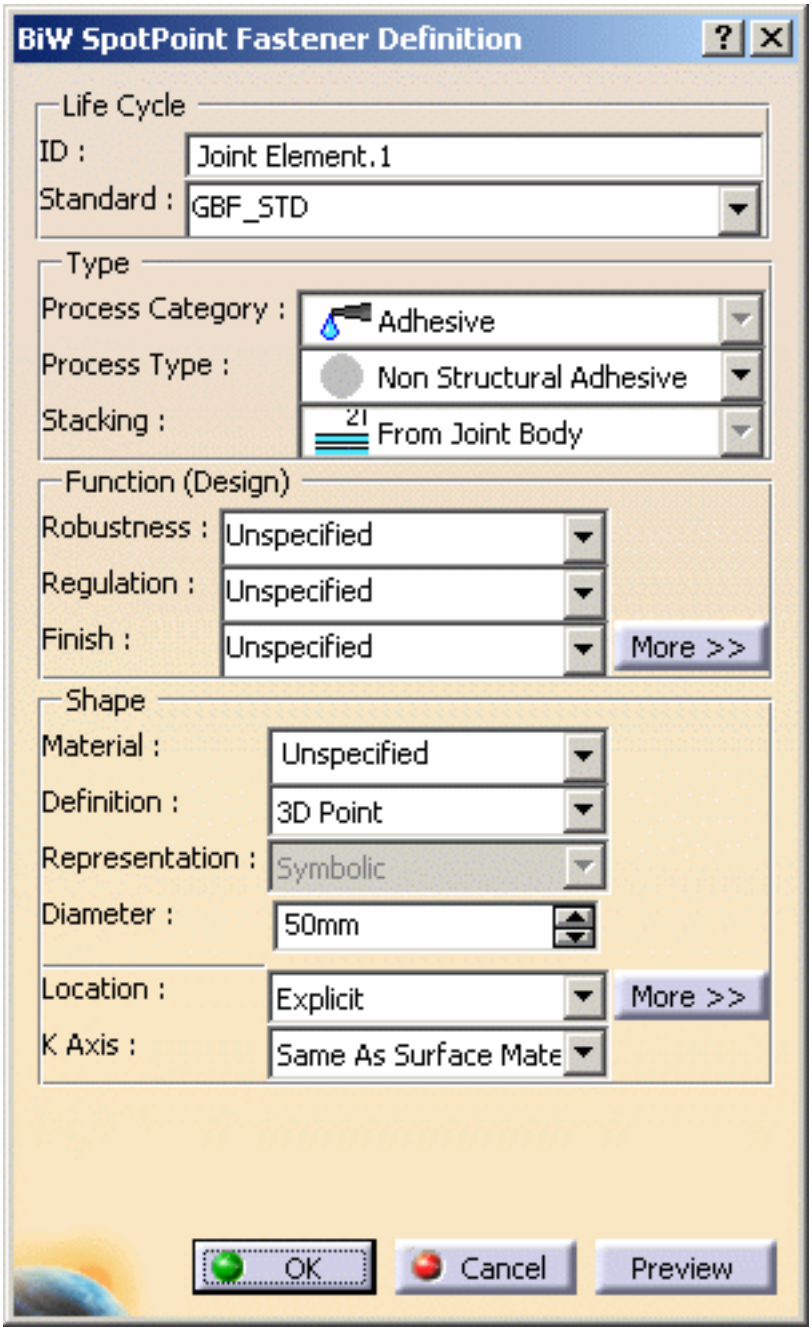
- a joint and a jointbody are already created: click the desired icon and select the jointbody in the selection tree.
- a joint and a jointbody are already created: select the jointbody in the selection tree and click the desired icon (here is the example used for our scenario).
- no jointbody is created. Select the components or the publications, and click the desired icon: a joint and a jointbody are created.

Open the SpotPoint1.CATProduct document.

Make sure the SpotPoint Fastener Type is set up in the standard file.

1. Select the jointbody in the specification tree.
2. Click the **BiW Adhesive Point** icon  .

The BiW SpotPoint Fastener Definition panel opens.



3. Specify whether you wish to use the existing standard or not.

   If a standard has been imported, a spotpoint is created using this standard. If not, you are able to define your own values for each attribute.

4. Define the spotpoint's following parameters:

- Type
- Functional parameters
- Shape
- Location
- K axis

5. Click OK to create the spot point.

The spot point (identified as Joint Element.xxx) is added to the specification tree, under the Joint Body node.

Creating Sealant SpotPoints

This task shows how to create a Body in White Sealant SpotPoint.

There are several ways to create a spotpoint:

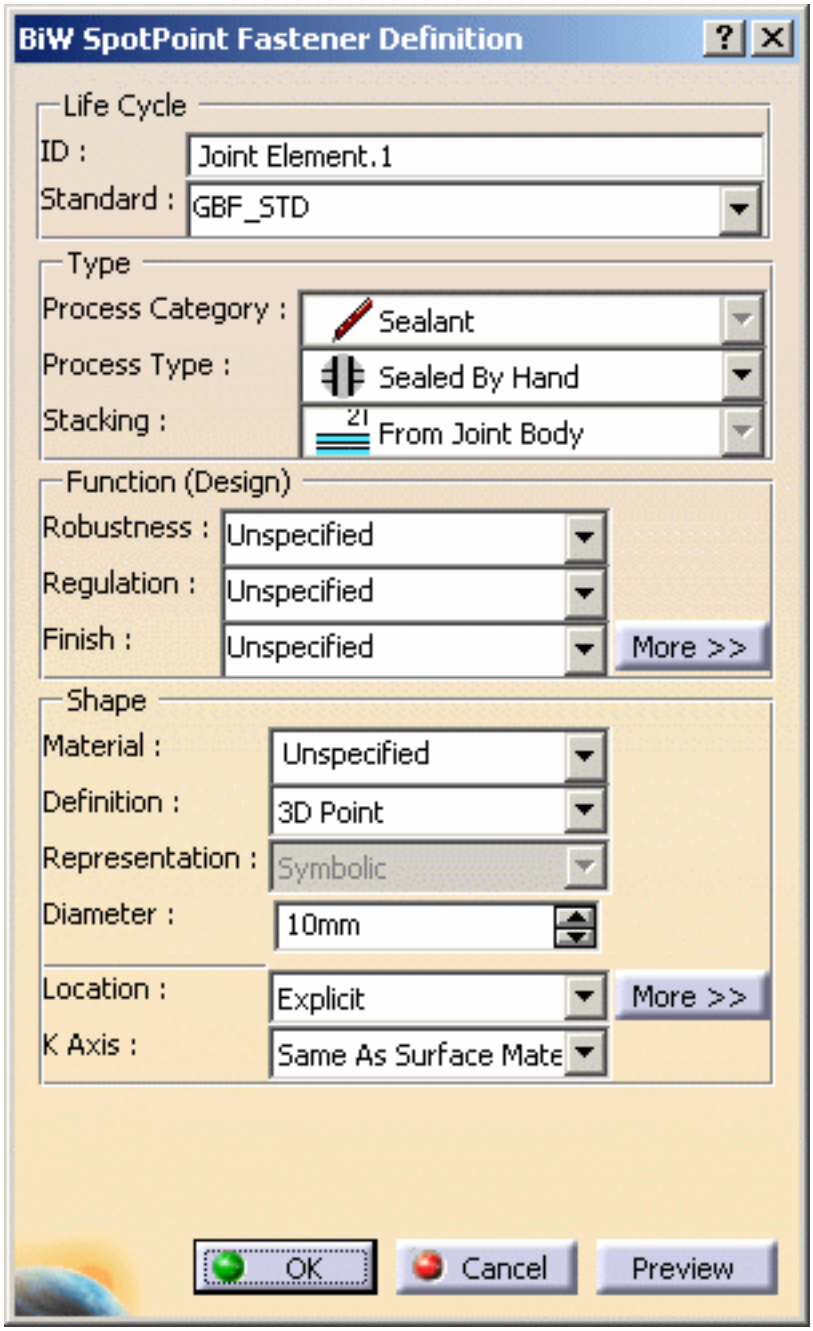
- a joint and a jointbody are already created: click the desired icon and select the jointbody in the selection tree.
- a joint and a jointbody are already created: select the jointbody in the selection tree and click the desired icon (here is the example used for our scenario).
- no jointbody is created. Select the components or the publications, and click the desired icon: a joint and a jointbody are created.

Open the SpotPoint1.CATProduct document.

Make sure the SpotPoint Fastener Type is set up in the standard file.

1. Select the jointbody in the specification tree.
2. Click the **BiW Sealant Point** icon  .

The BiW SpotPoint Fastener Definition panel opens.



3. Specify whether you wish to use the existing standard or not.

   If a standard has been imported, a spotpoint is created using this standard. If not, you are able to define your own values for each attribute.

4. Define the spotpoint's following parameters:

- Type
- Functional parameters
- Shape
- Location
- K axis

5. Click OK to create the spot point.

The spot point (identified as Joint Element.xxx) is added to the specification tree, under the Joint Body node.

Creating Mechanical SpotPoints

This task shows how to create a Body in White Mechanical SpotPoint.

There are several ways to create a spotpoint:

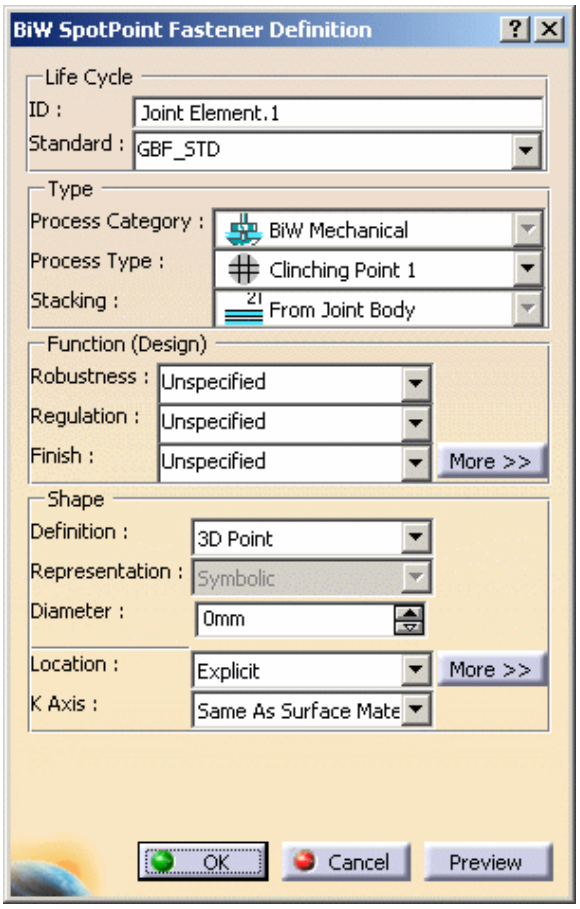
- a joint and a jointbody are already created: click the desired icon and select the joint body in the selection tree.
- a joint and a jointbody are already created: select the jointbody in the selection tree and click the desired icon (here is the example used for our scenario).
- no jointbody is created. Select the components or the publications, and click the desired icon: a joint and a jointbody are created.

Open the SpotPoint1.CATProduct document.

Make sure the SpotPoint Fastener Type is set up in the standard file.

1. Select the jointbody in the specification tree.
2. Click the **BiW Mechanical Point** icon .

The BiW SpotPoint Fastener Definition panel opens.



3. Specify whether you wish to use the existing standard or not.

   If a standard has been imported, a spotpoint is created using this standard. If not, you are able to define your own values for each attribute.

4. Define the spotpoint's following parameters:

- Type
- Functional parameters
- Shape
- Location
- K axis

In our example, we used a particular Process Type: the rivet and particular shape definitions: shank, shank with head, shank with head and foot.

Hemisphere and Shanks definition are only available with Unspecified and Rivets process types.

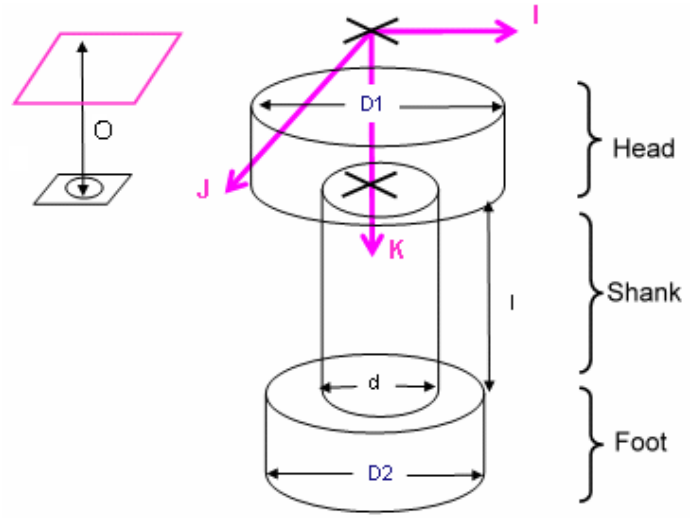
O = shank base plane offset from selection point

L1 = head length
D1 = head diameter

d = shank diameter (shape diameter)
l = shank length

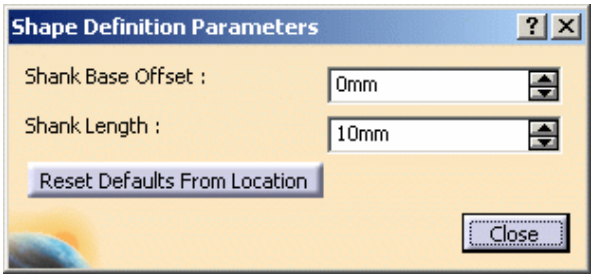
L2 = foot length
D2 = foot diameter

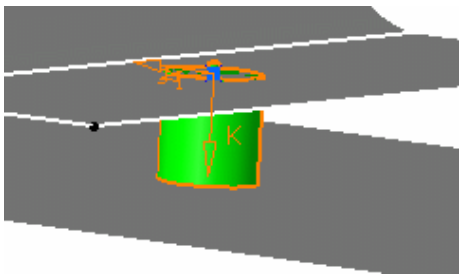
Default values:
D1 = D2 = dX2
L1 = L2 = d/3



Shank

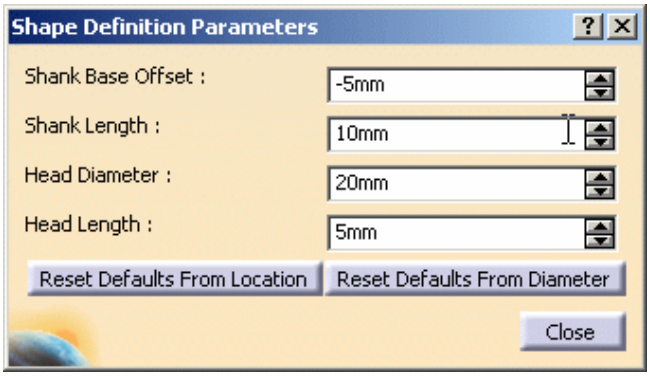
1. Define the **Shank Base Offset**.
2. Define the **Shank Length**.

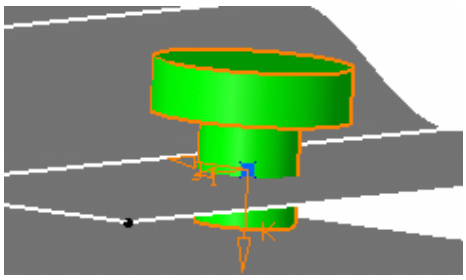




Shank with Head

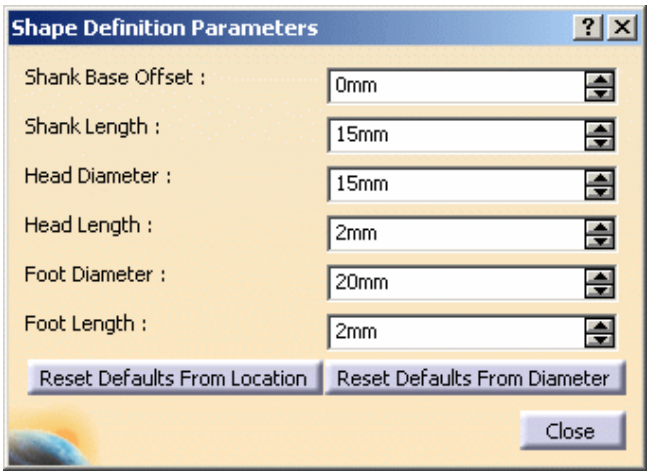
1. Define the **Shank Base Offset**.
2. Define the **Shank Length**.
3. Define the **Head Diameter**.
4. Define the **Head Length**.

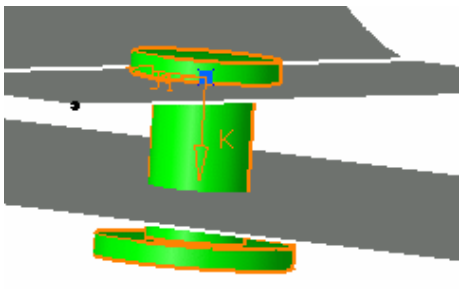




Shank with Head and Foot

1. Define the **Shank Base Offset**.
2. Define the **Shank Length**.
3. Define the **Head Diameter**.
4. Define the **Head Length**.
5. Define the **Foot Diameter**.
6. Define the **Foot Length**.



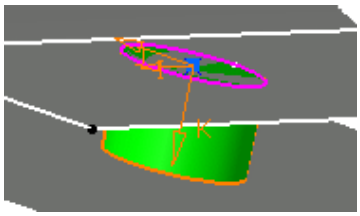


- The **Reset Defaults From Location** button automatically reset the shank base offset and the shank length values according to the location of the spotpoint (computation of the crossed thicknesses), taking into account the gap between the crossed joined components.
- The **Reset Defaults From Diameter** button automatically reset the default values of the head diameter and the head length values.

9. Define the shank diameter.

10. Define the spotpoint's location.

Selecting a circle to create a rivet automatically initializes the shank diameter.



Rivets cannot apply on non planar circles.

11. Define the K axis:
    - **Same as Surface Material**
    - **Opposite to Surface Material**

12. Click OK to create the spot point.

The spot point (identified as Joint Element.xxx) is added to the specification tree, under the Joint Body node.

Creating Unspecified SpotPoints

This task shows how to create a Body in White SpotPoint.

There are several ways to create a spotpoint:

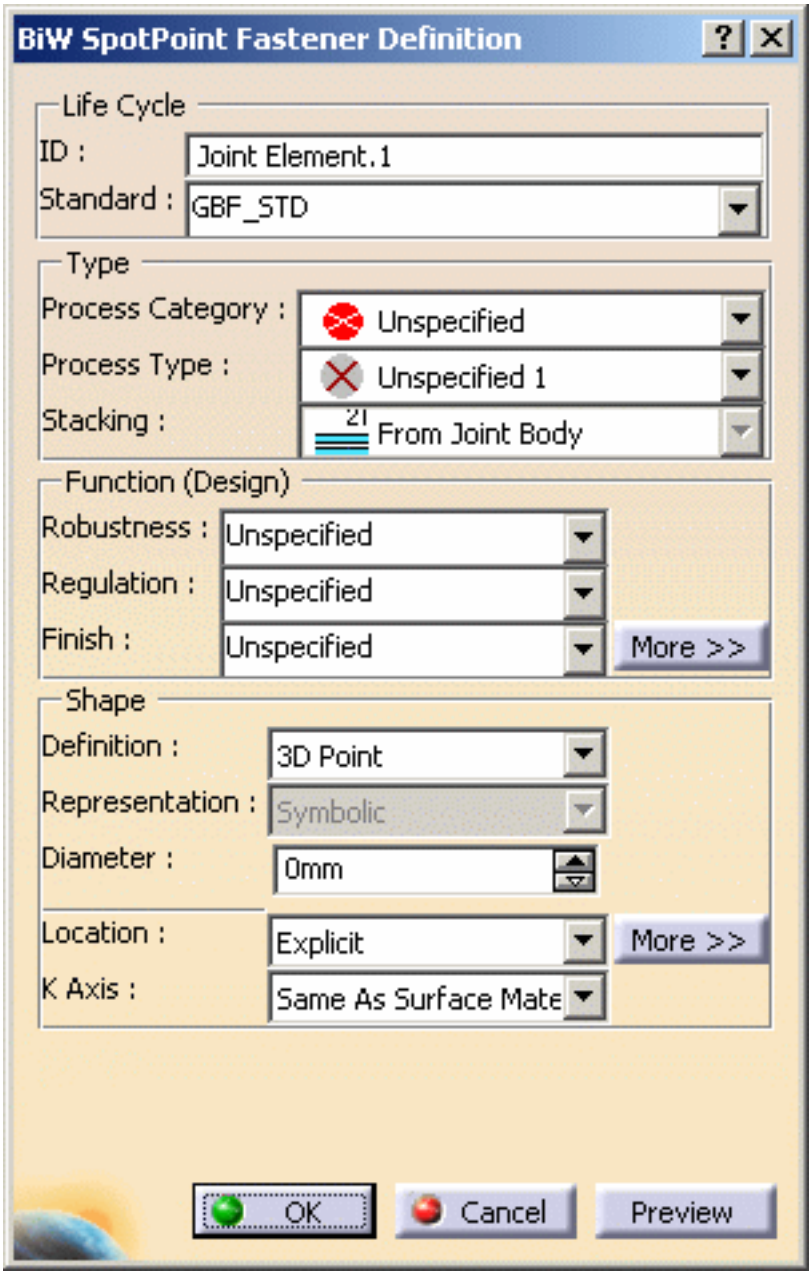
- a joint and a jointbody are already created: click the desired icon and select the joint body in the selection tree.
- a joint and a jointbody are already created: select the joint body in the selection tree and click the desired icon (here is the example used for our scenario).
- no joint body is created. Select the components or the publications, and click the desired icon: a joint and a joint body are created.

Open the SpotPoint1.CATProduct document.

Make sure the SpotPoint Fastener Type is set up in the standard file.

1. Select the jointbody in the specification tree.
2. Click the **BiW SpotPoint** icon  .

The BiW SpotPoint Fastener Definition dialog box opens.



3. Specify whether you wish to use the existing standard or not.

   If a standard has been imported, a spotpoint is created using this standard. If not, you are able to define your own values for each attribute.

4. Define the spotpoint's following parameters:

- Type
- Functional parameters
- Shape
- Location
- K axis

5. Click OK to create the spotpoint.

The spotpoint (identified as Joint Element.xxx) is added to the specification tree, under the Assembly Joint Body node.

Creating SpotProjection Welds

This task shows how to create a welding spotprojection, which one of the connected zones is specified as a projection zone.
There are several ways to create a spotprojection:

- a joint and a jointbody are already created: click the desired icon and select the jointbody in the selection tree.
- a joint and a jointbody are already created: select the jointbody in the selection tree and click the desired icon (here is the example used for our scenario).
- no jointbody is created. Select the components or the publications, and click the desired icon: a joint and a jointbody are created if needed.

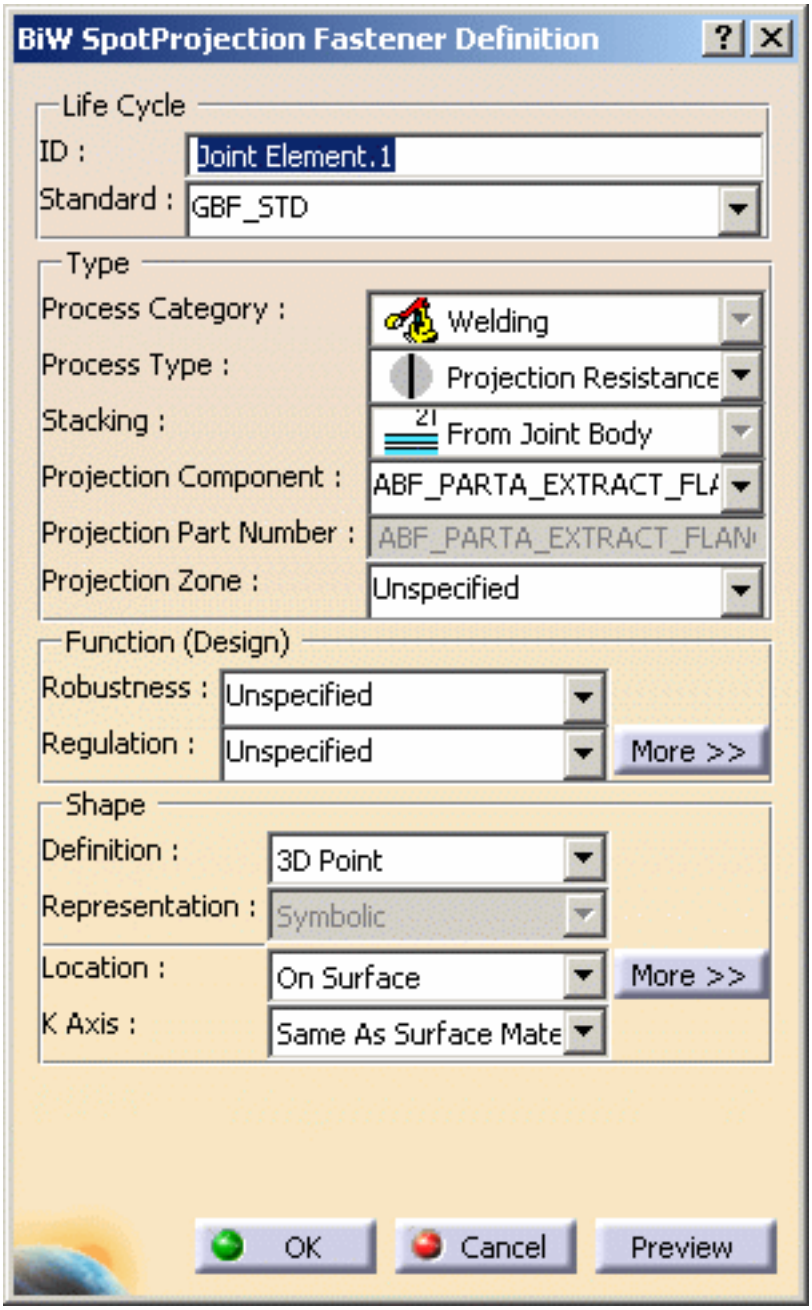
Open the Projection1.CATProduct document.

1. Select the jointbody in the specification tree.

2. Click the **BiW Welding SpotProjection** icon in the Welding toolbar from the Spot Welding sub-toolbar.

Make sure the SpotProjection Fastener Type is set up in the standard file.

The BiW SpotProjection Fastener Definition dialog box opens.



3. Specify whether you wish to use the existing standard or not.

   If a standard has been imported, a spotprojection is created using this standard. If not, you are able to define your own values for each attribute.

4. Define the Projection type:

- **Projection Component**: specifies one of the Joints' components
- **Projection Part Number**
- **Projection Zone**: specifies one of the Joint Body zones and must belong to the component specified in the Projection Component field.

5. Define the design of the spotprojection:

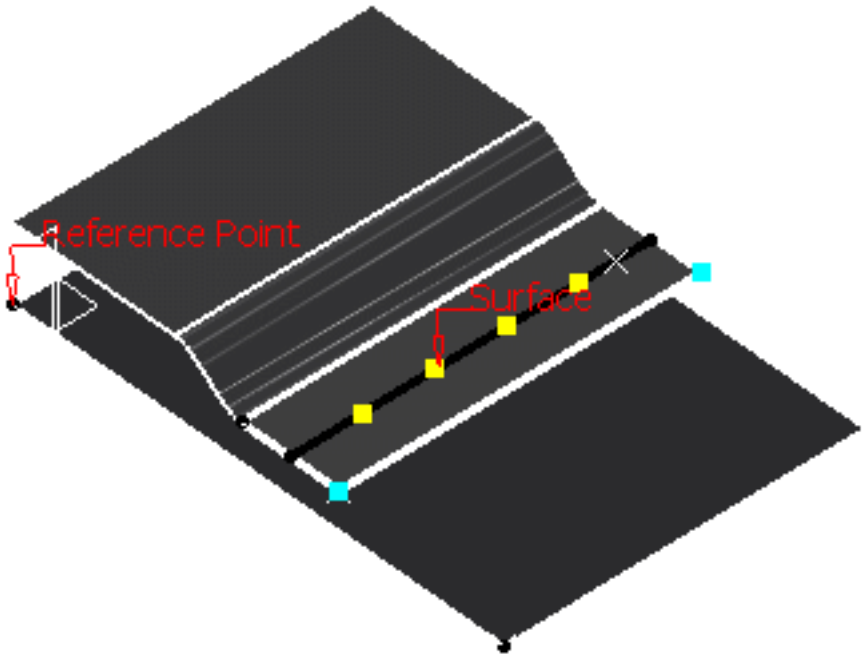
- Robustness
- Regulation
- **More>>:** allows you to define the Manufacturing parameters.

The shape is necessary a 3D point.

6. Select the location of the point:

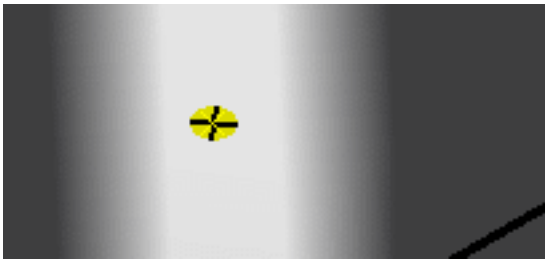
- On Surface (as in our scenario)
- On Point
- From Curve on Surface
- Along Curve
- Explicit

For further information on the location methods, you can click the **More>>** button.



7. Click **OK** to create the spotprojection.

   The SpotProjection (identified as Joint Element.xxx) is added to the specification tree, under the Joint Body node.

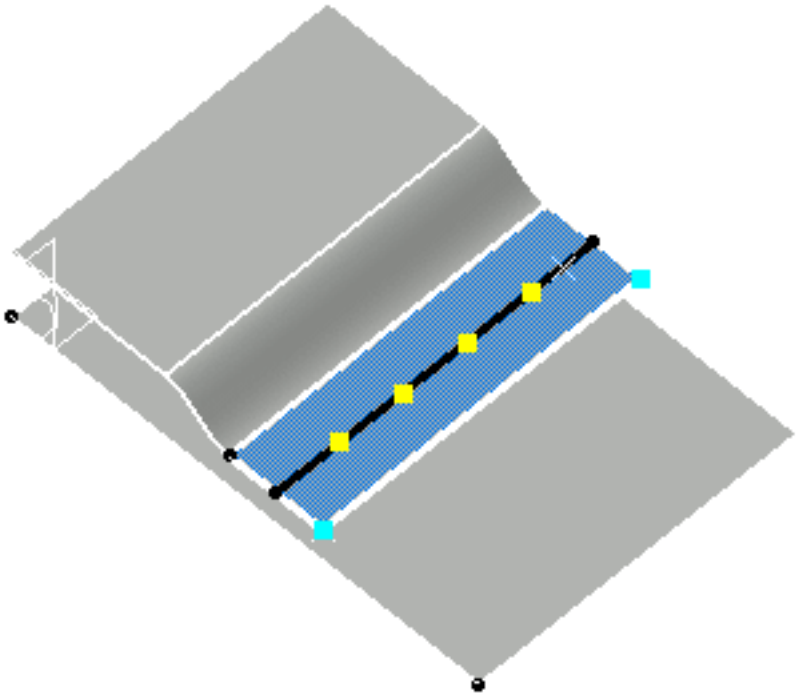


Creating Multi-Selected Spots

This task shows how to create multi-selected spot fasteners.

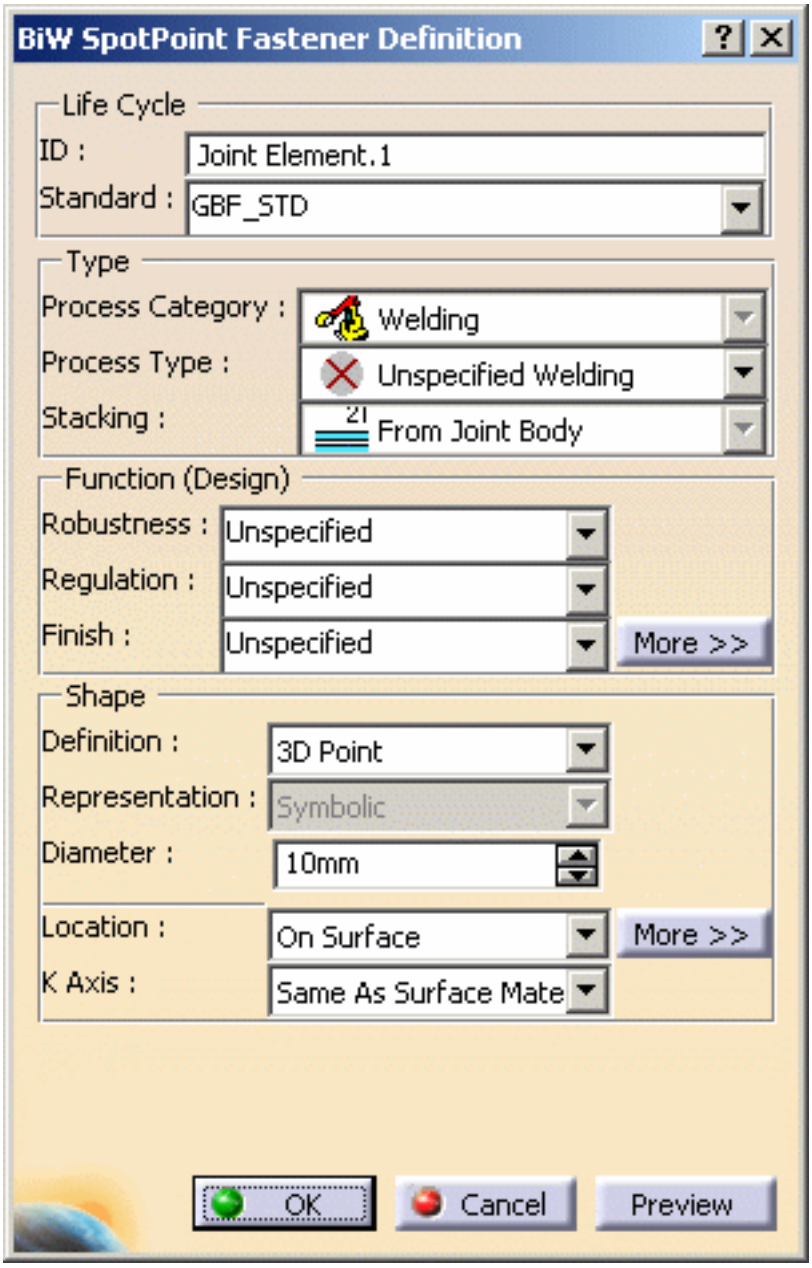
Open the MultipleSpotPoints1.CATProduct document.

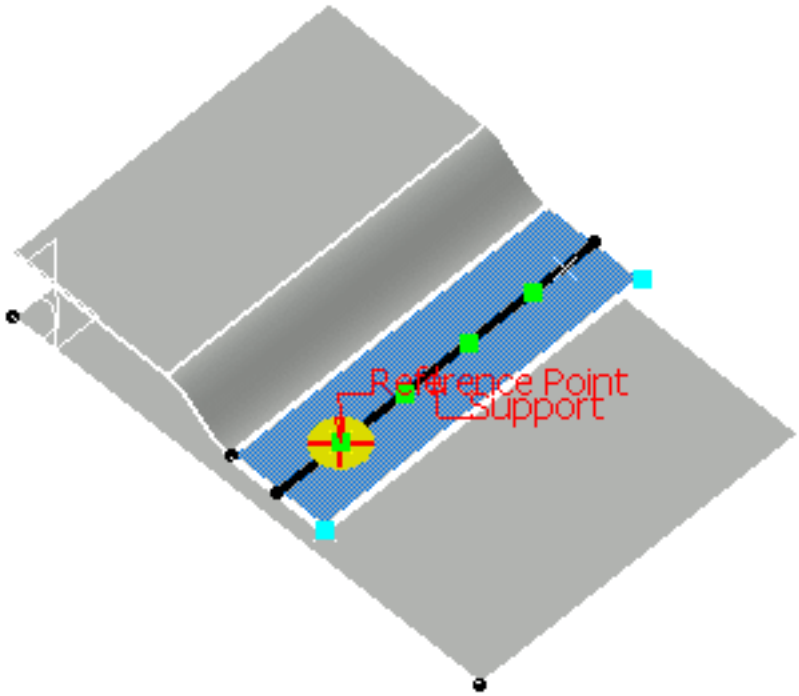
1. Select the points located along the curve using the **Ctrl** key.



2. Still holding the **Ctrl** key, select the joint body from the specification tree.
3. Select the **BiW Welding SpotPoint** icon.

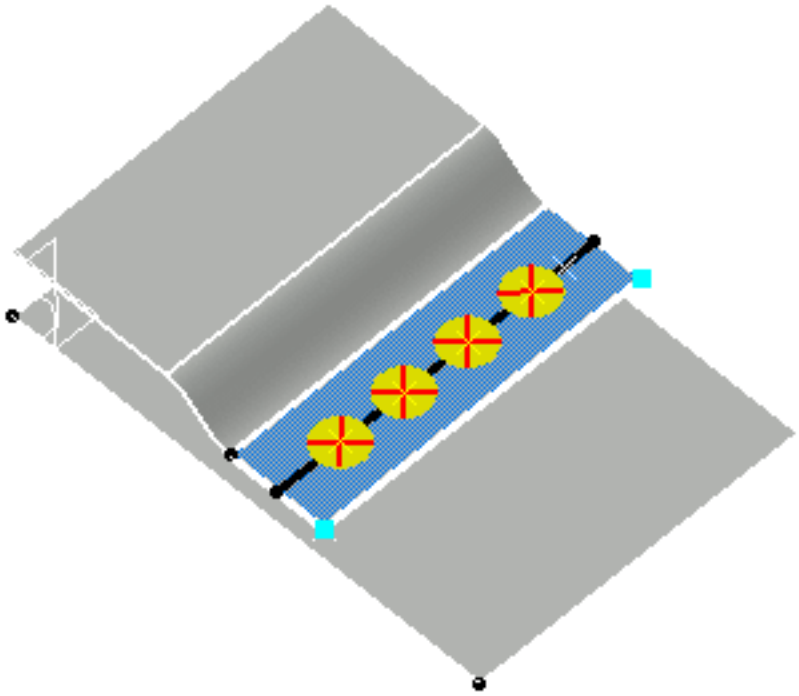
The BiW SpotPoint Fastener Definition dialog box displays as well as a preview of the reference spotpoint.


BiW SpotPoint Fastener Definition
Life Cycle
ID :
Joint Element.1
Standard :
GBF_STD
Type
Process Category :
Welding
Process Type :
Unspecified Welding
Stacking :
From Joint Body
Function (Design)
Robustness :
Unspecified
Regulation :
Unspecified
Finish :
Unspecified
More >>
Shape
Definition :
3D Point
Representation :
Symbolic
Diameter :
10mm
Location :
On Surface
More >>
K Axis :
Same As Surface Mate
OK
Cancel
Preview



5. Click OK to validate the creation of multiple spotpoints.

Multiple spotpoints have been created.



This capability is only available with the **On Point**, **On Support Surface**, and **Explicit** location options.

Repeating Spots

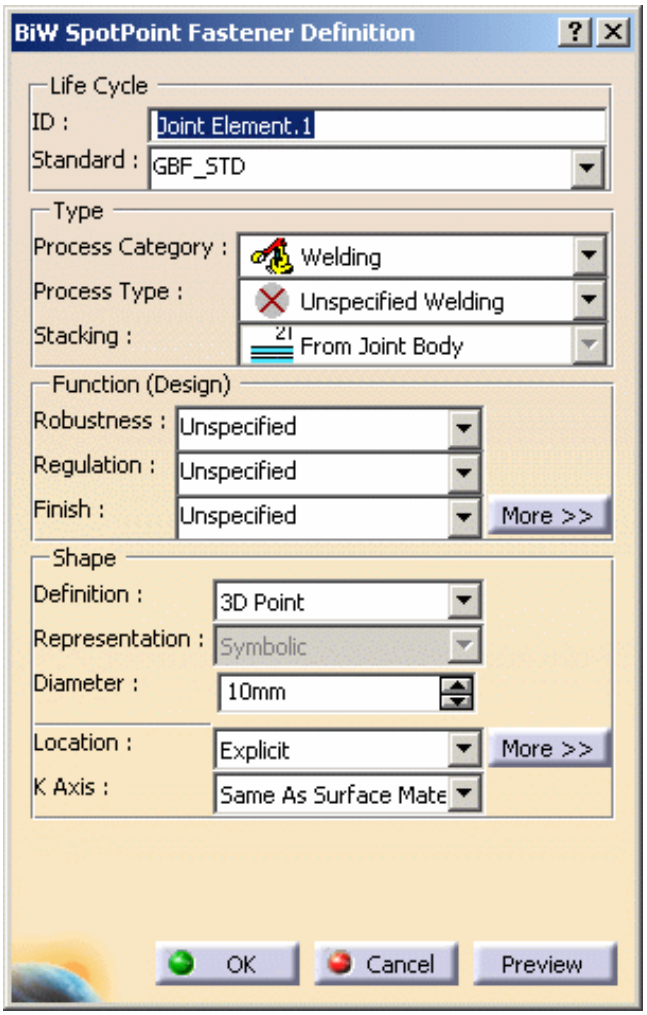
This task shows you how to repeat spots along a selected curve.

Open the Repeating1.CATProduct document.

1. Double-click Joint Element.1 from the specification tree.

The BiW SpotPoint Fastener Definition dialog box opens.

2. Select the **Along Curve** location.



3. Check the **Repeat after OK** button to create more spot points using the currently created spot point as reference.

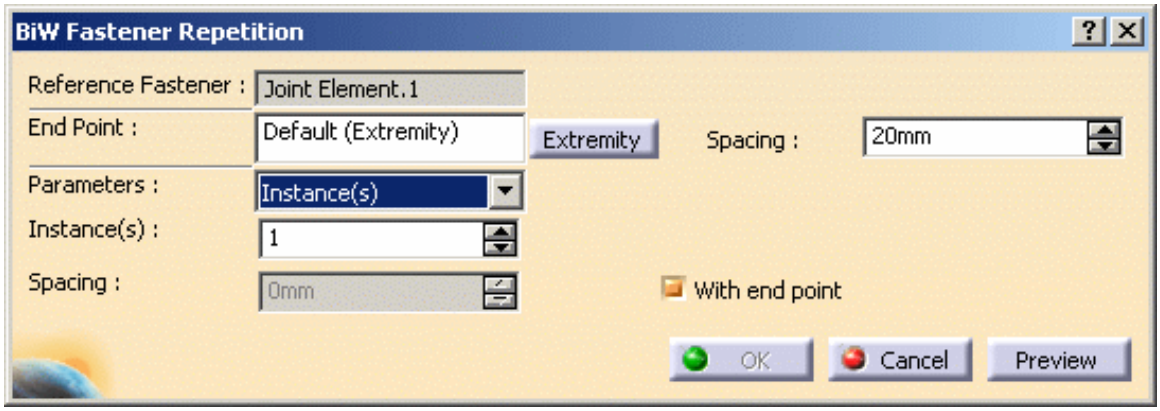
4. Click OK.

The BiW Fastener Repetition dialog box appears.

It shows:

- the reference fastener referring to the currently created spotpoint.
- the End Point being the extremity of the curve by default.
  - You can choose the other extremity by clicking the **Extremity** button as well as selecting a point or a spotpoint.
  - You can define the distance between the last created spot and its End Point in the **Spacing** field. By default the distance is computed using the **Dist. To Reference** value of the Reference Fastener from the Along Curve Location dialog box.

- Uncheck the **With end point** if you do not want the last created spotpoint to be located on the end point. This option is automatically grayed out if the End Point is a spotpoint and as long as the spacing is 0mm. You can manage the spacing and the end point when using the Instances parameter.



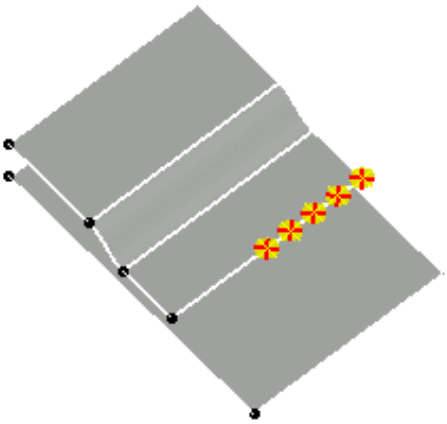
5. Define the Parameters:

- **Instance(s)**: specifies the number of instances to be created and the last created spot corresponds to the curve End point. The spacing between two successive spots is computed.
- **Spacing**: specifies the spacing value between two consecutive created spot points.
- **Instances & Spacing**: specifies the number of instances as well as the spacing value. The spot points are created towards the end point.
  In this case, spot points can be created after the end point.

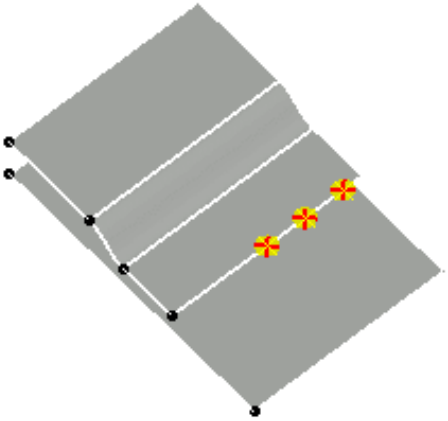
6. According to what you chose in the Parameters field, either define a value in the Instances field or in the Spacing one or in both of them.

When entering a value in the Instance field, the distance between two successive points is automatically calculated, spot points are equidistant and the last repeated spot point is positioned on the curve end point.

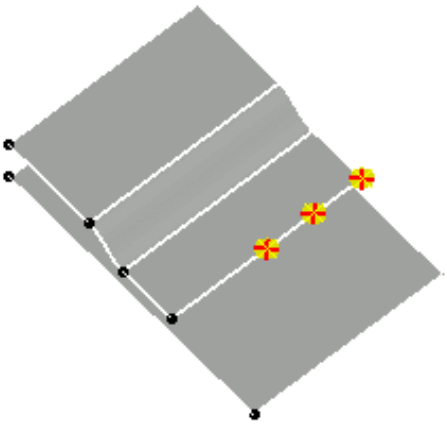
7. Click OK to validate the repetition.



Instances (4)



Spacing (20mm)



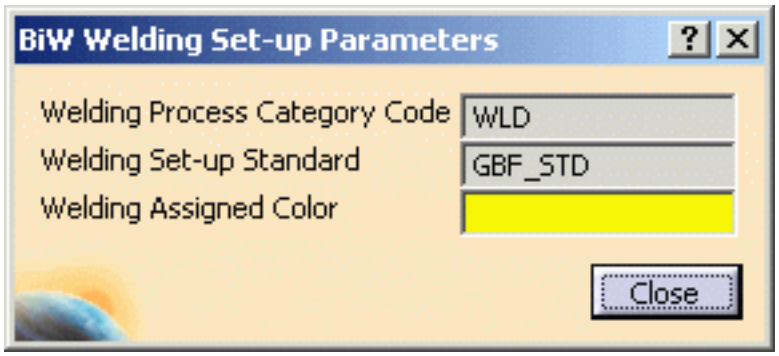
Instances and Spacing (2 & 25mm)

This capability is only available with the **Along Curve** and the **From Curve On Surface** options.

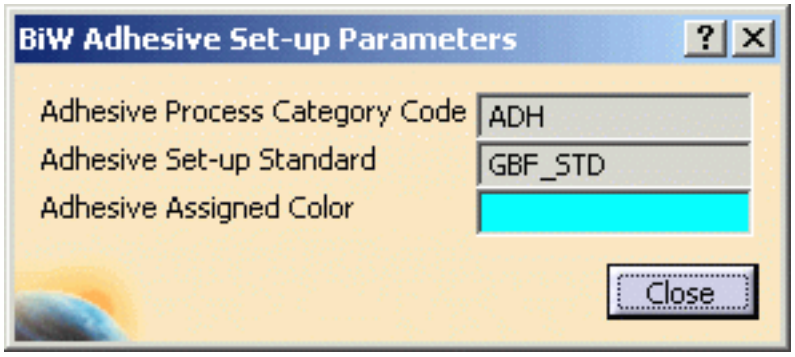
Displaying Process Category Parameters

This task shows you how to display the parameters for each process category by clicking the parameters icon available on every appropriate toolbar.
The dialog box displays the process category code, the imported standard (if it has been previously defined), and the assigned color.

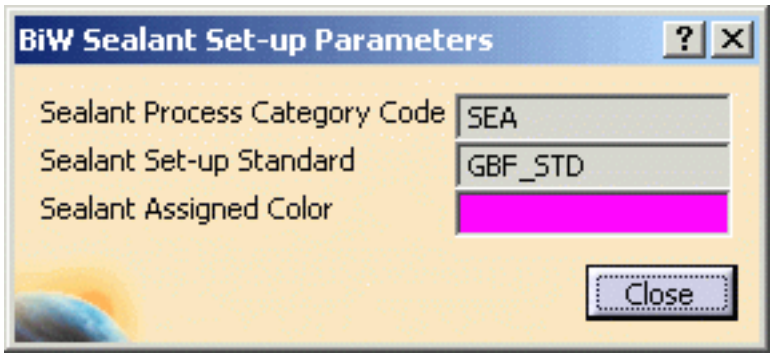
- Welding 



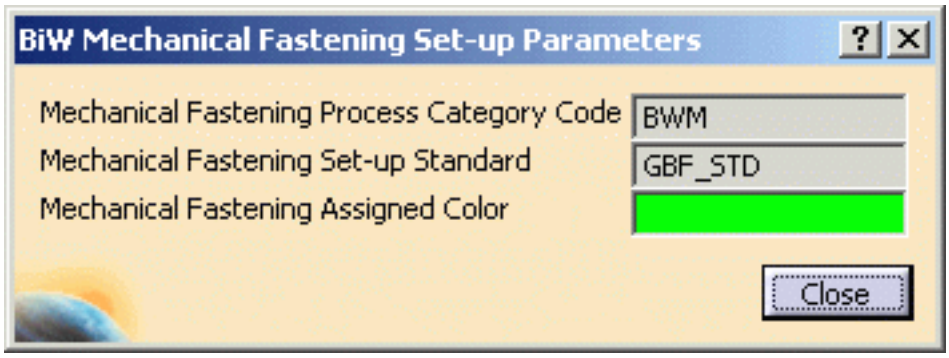
- Adhesive 



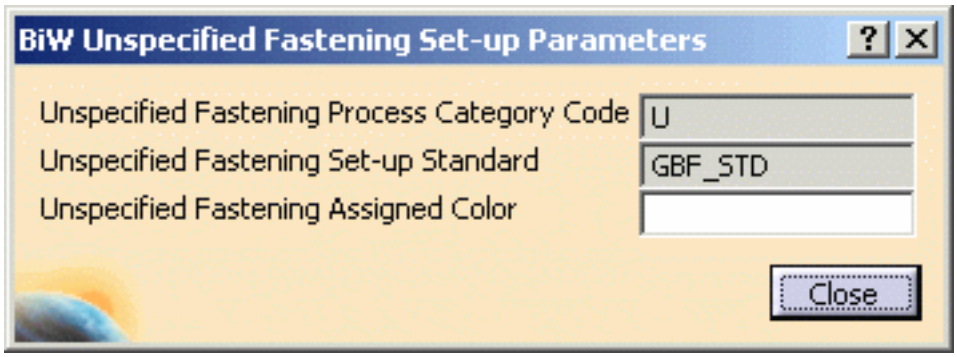
- Sealant 



- Mechanical 



- Unspecified 



Creating CurveBeads

Creating Welding Curvebeads
Creating Adhesive Curvebeads
Creating Sealant Curvebeads
Creating Unspecified Curvebeads
Repeating CurveBeads

It is possible to valuate curve fasteners' attributes of type Dimension from a measure. However these dimensions are not associative, that is to say they are not integrated in Knowledge.

- It is possible to create and associate an annotation to a curve fastener.
  For further information, refer to the *3D Functional Tolerancing and Annotation* documentation.

- assign a layer to a curve fastener and published joined contact zones. A layer number is assigned to each published zone.
  For further information, refer to the Using the Graphic Properties Toolbar chapter in the *CATIA Infrastructure User Guide*.

- apply visualization filters, that is a group of layers, to visualize (or not) only the curve fasteners located on the layers in the filter.
  For further information, refer to the Using Visualization Filters chapter in the *CATIA Infrastructure User Guide*.

Creating Welding CurveBeads

This task shows how to create a BiW Welding CurveBead.

There are several ways to create a curvebead:

- a joint and a jointbody are already created: click the desired icon and select the joint body in the selection tree.
- a joint and a jointbody are already created: select the joint body in the selection tree and click the desired icon (here is the example used for our scenario).
- no joint body is created. Select the components or the publications, and click the desired icon: a joint and a joint body are created if needed.

Open the ABF_CurveBead.CATProduct document.

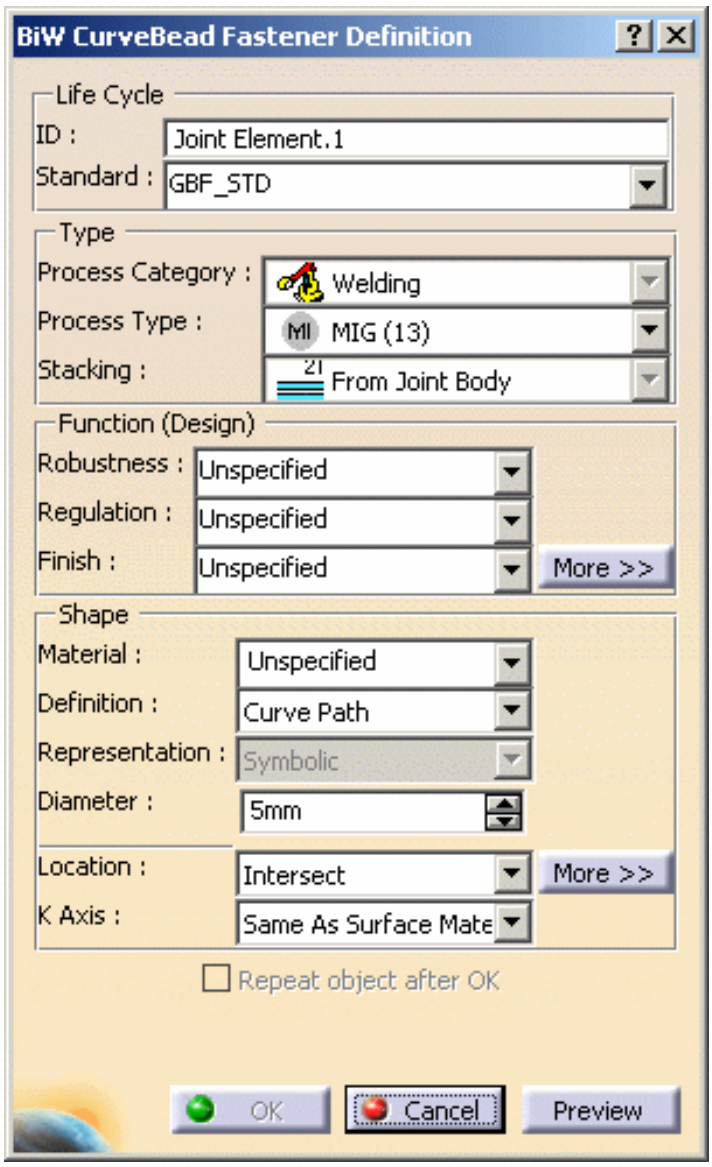
Make sure the Curvebead Fastener Type is set up in the standard file.

1. Select the joint body in the specification tree.
2. Click the **BiW Welding CurveBead** icon from the **Welding** toolbar.

The BiW CurveBead Fastener Definition dialog box opens.

3. Specify whether you wish to use the existing standard or not.
   If a standard has been imported, a curvebead is created using this standard. If not, you are able to define your own values for each attribute.

If the last location method is different from **Explicit**, the ABF application creates a specification part associated to the Assembly Joint if this specification part does not already exist.



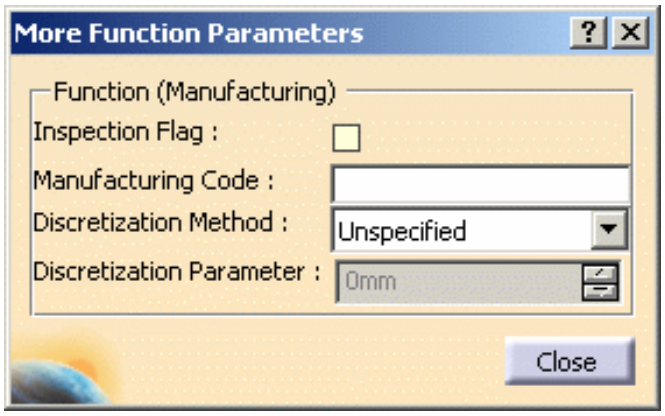
Type

4. Select the process type.

Functional parameters

5. Define the functional parameters of the curvebead:

- Robustness
- Regulation
- Finish
- **More>>:** allows you to define Manufacturing parameters.



- **Discretization Method**:
  - **Unspecified**: no discretization method is used. The default visualization is set in **Tools -> Options -> General -> Display -> Performances -> 3D Accuracy**.
  - **Sag:** segments are defined on the curve according to the tolerance value set in the discretization parameter.
  - **Step:** equidistant points are created on the curve according to the value set in the discretization parameter.
- **Discretization Parameter**: distance value for each curvebead depending on the chosen discretization method.

Both parameters will be used for the visualization of the curvebead and for the export/report.

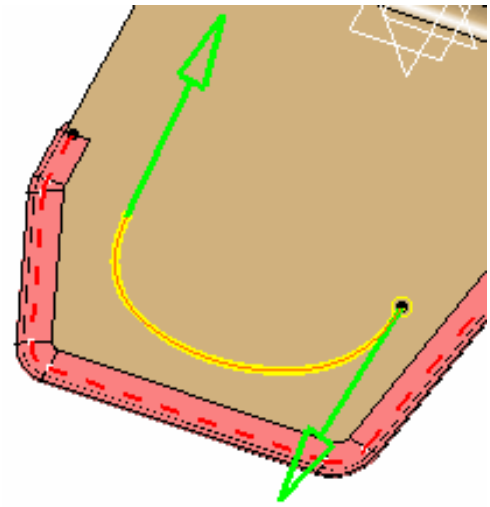
Shape

6. Specify the material.
7. Specify the definition:

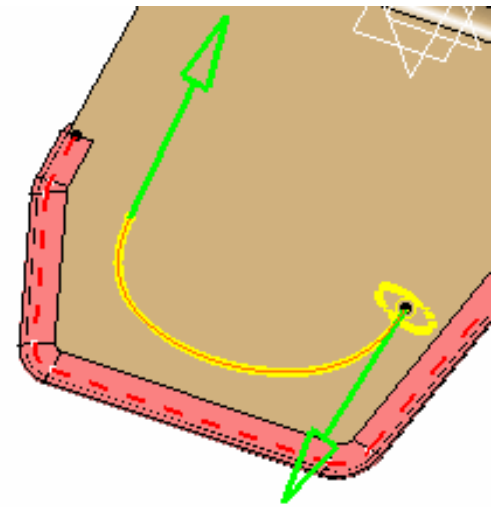
- curve path
- cylinder path
- half cylinder path, etc.

8. Define the diameter in the following cases:

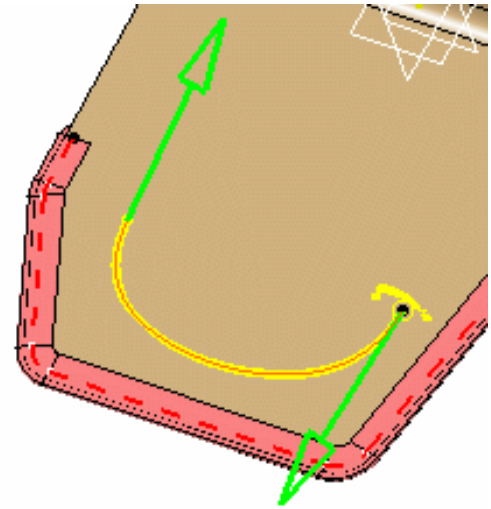
- Curve Path
- Cylinder Pipe
- Half Cylinder Pipe



Curve Path



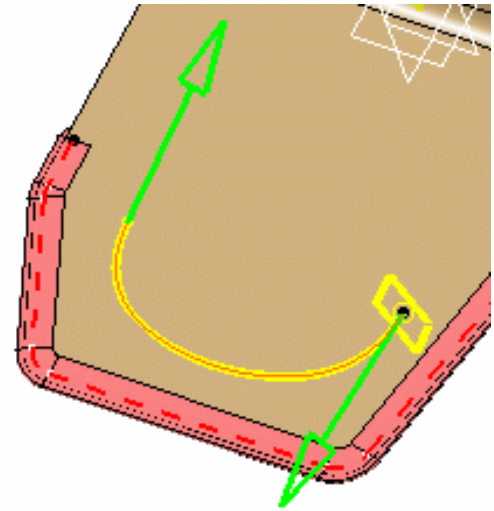
Cylinder



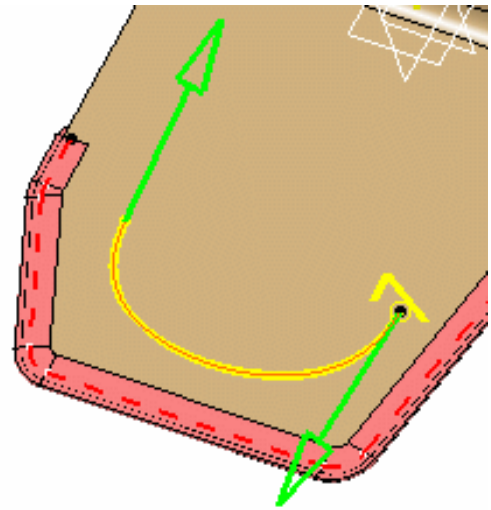
Half Cylinder

9. Define the base and height in the following cases:

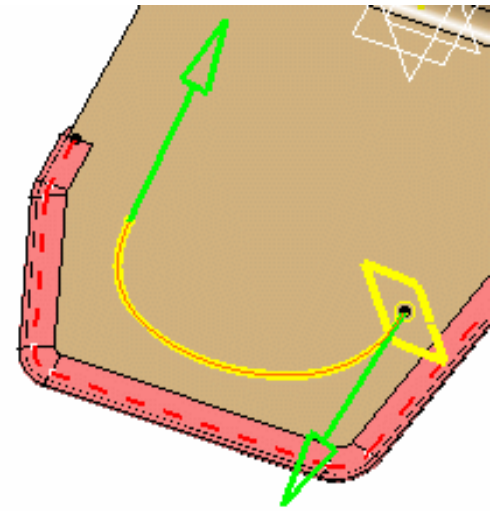
- Diamond Pipe
- Half Diamond Pipe
- Rectangle Pipe
- Half Rectangle Pipe



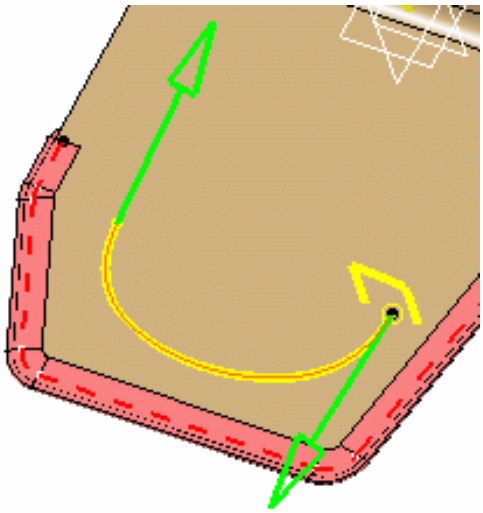
Diamond Pipe



Half Diamond Pipe



Rectangle Pipe



Half Rectangle Pipe

Location

10. Select the location:

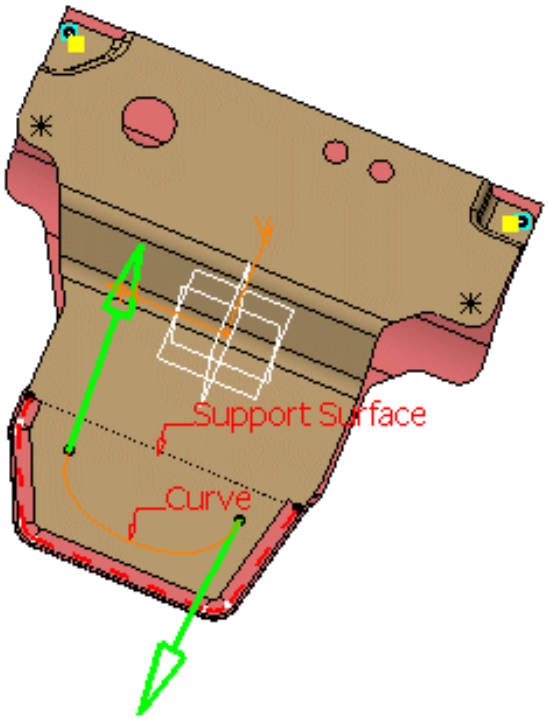
- Along Curve
- From Curve On Surface
- Intersect
- Explicit

According to the location you choose, different dialog boxes display, except when selecting the Explicit mode.

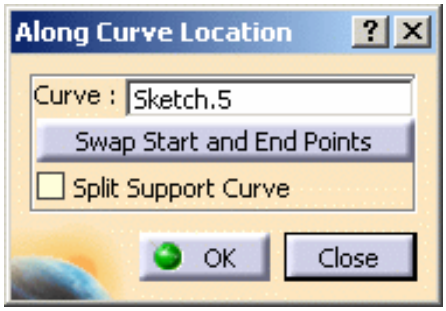
Along Curve

1. Select the curve.

Green arrows appear in the 3D geometry to let you know the end and start point of the curvebead.

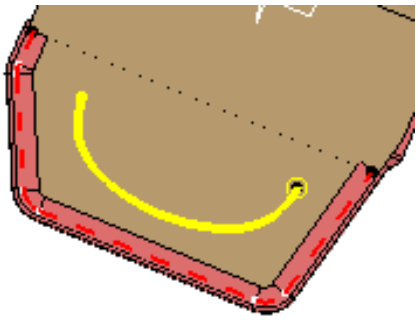


2. Click the **More>>** button to display further information.
3. From the Along Curve Location dialog box, click the **Swap Start and End Points** button to change the curve direction.



The **Split Support Curve** enables to trim the curvebead.

The curvebead will be created along the original curve.

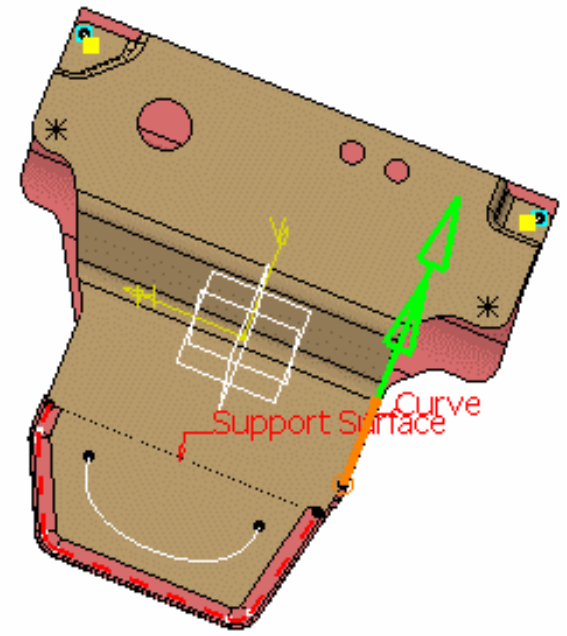


From Curve On Surface

1. Select a curve, for instance a part edge.

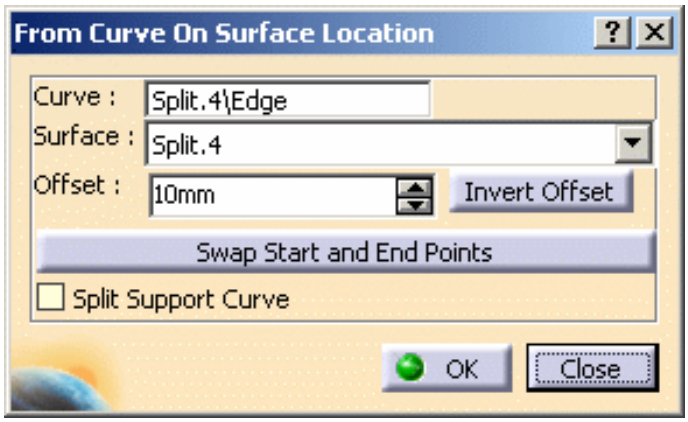
   Most of the times, a surface is selected by default.

   Green arrows appear in the 3D geometry to let you know the end and start point of the curvebead.
2. If needed, select a surface.



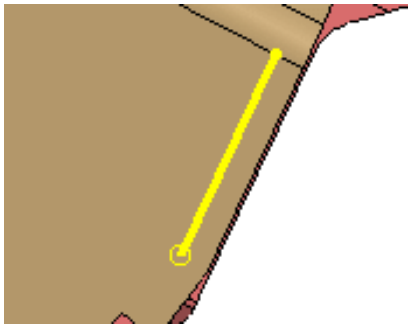
If the surface could not be found, or if you want to select another surface, choose a surface in the drop-down list. To choose an unspecified zone, select **Other (Zone Unspec)** and then select the zone in the 3D geometry.

3. Optionally modify the offset. In our example, we defined an offset of 10mm.
4. Click the **Swap Start and End Points** button to change the curve direction.
5. Click the **Inverse Offset** button changes the offset curvebead position according to the original curvebead.

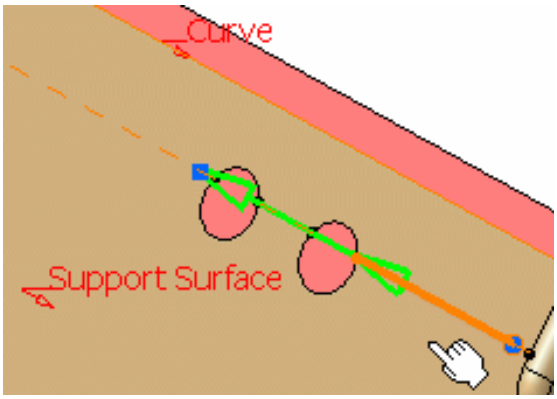


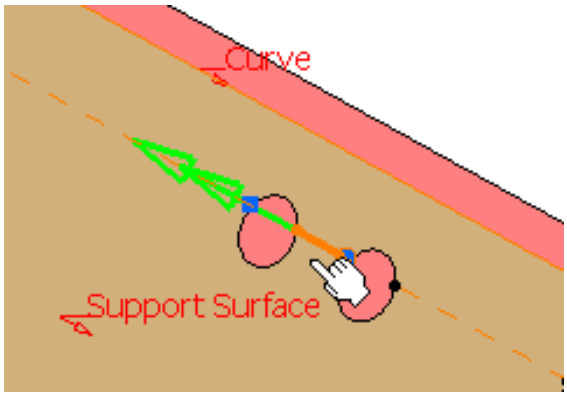
The Split Support Curve enables to trim the curvebead.

The curvebead will be created along the original curve offset.



If several solutions are possible (for instance if there is a hole on the surfaces), you need to select the surface in the 3D geometry that is the closest to the curvebead to be created.

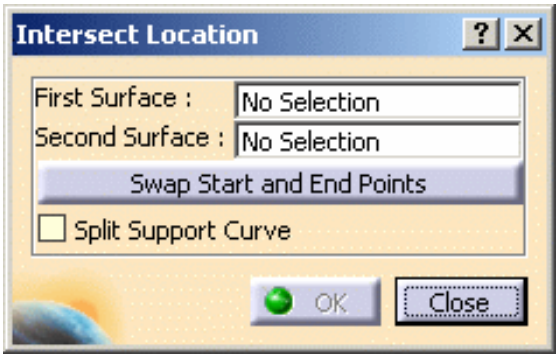


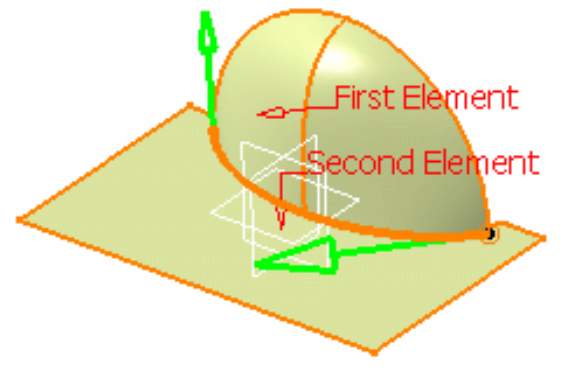


Intersect

Open the Intersect1.CATProduct document.

1. Click the **More>>** button to display further information.





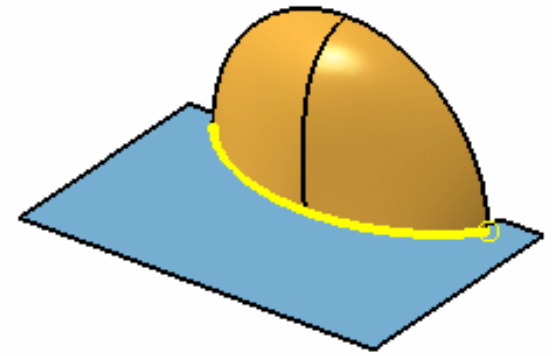
2. Successively select two surfaces, either in the 3D geometry or in the specification tree.
   The curvebead is located at the intersection of the two surfaces.

- You can as well select a plane as the surface.
- Surfaces must belong to the joined components but may be different from the joint body zones.

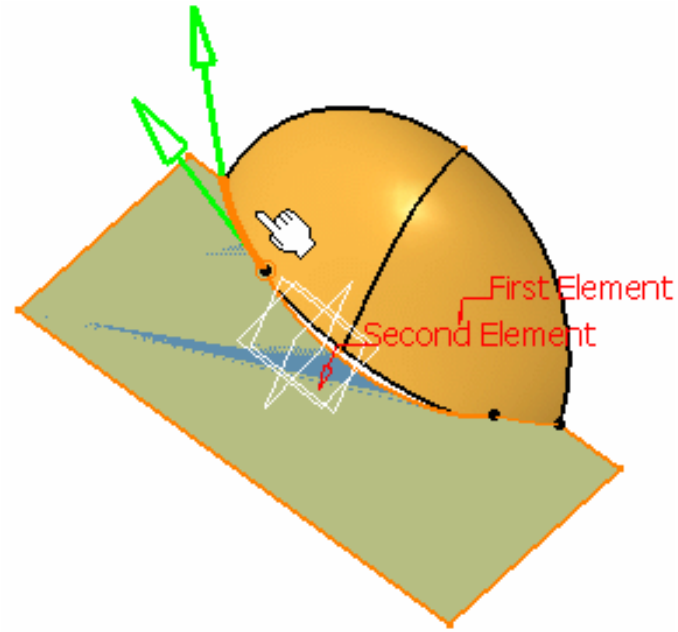
3. Click the **Swap Start and End Points** button to change the curve direction.

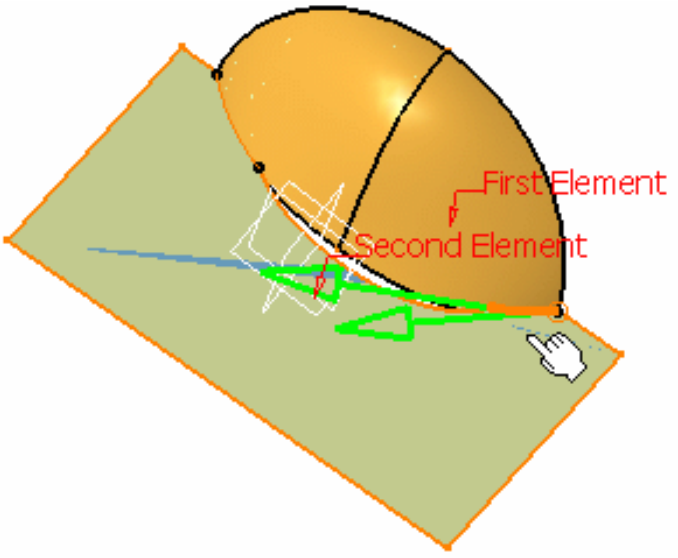
The Split Support Curve enables to trim the curvebead.

The curvebead will be created from the intersection of the two surfaces.
The curvebead's normal is the normal to the first selected surface at the curvebead's start point



If several solutions are possible (for instance if there is a hole between the two surfaces), you need to select the surface in the 3D geometry that is the closest to the curvebead to be created.





Explicit

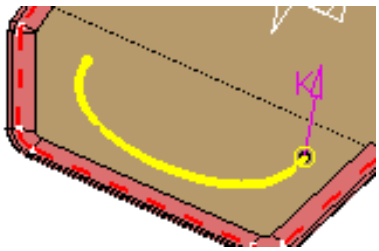
- You need to select first one of the three other options (Along curve, from curve on surface, or Intersect) to be able to select the curve position.
- When selecting the Explicit mode, the curve location is kept and it cannot be modified.
  The curve location is then a polyline, which is recalculated according to the selected curve, and does not reference the specification part anymore.

When the location method switches to Explicit, the specification part is automatically deleted if it is empty.

- The polyline is computed according to the discretization parameters. If it is set to Unspecified, the discretization value is calculated from the options defined in **Tools -> Options -> General -> Display -> Performances -> 3D Accuracy**.
- The **Isolate** command has the same behavior as the Explicit method: you can select the **Isolate** item from the contextual menu once the curve is created, by right-clicking the joint element.

K Axis

11. Define the orientation of the curvebead by setting the **K axis** to either:
    - **Same as Surface Material**
    - **Opposite to Surface Material**



To visualize the K axis, the K Axis Only option should be checked in **Tools -> Options**.

12. Click **OK** to create the curvebead.

The curvebead (identified as Joint Element.xxx) is added to the specification tree, under the Assembly Joint Body node.

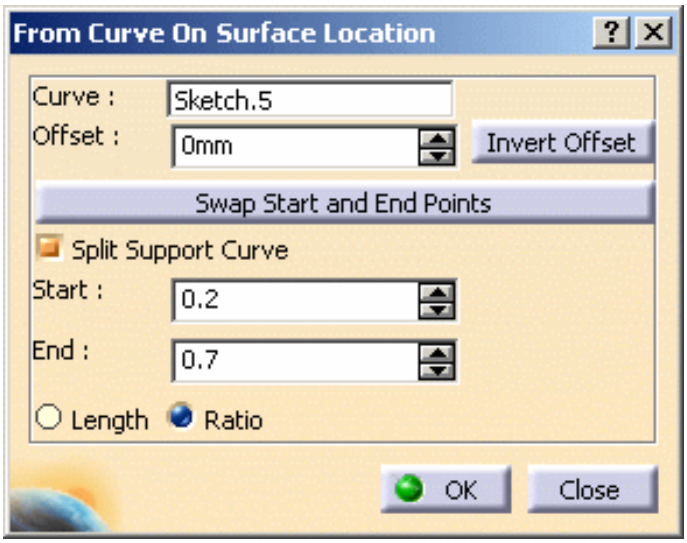
Trimming a Curvebead

1. Select the curvebead location. In our scenario, we selected the **From Curve On Surface** location.
2. Verify that the **Split Support Curve** button is checked.

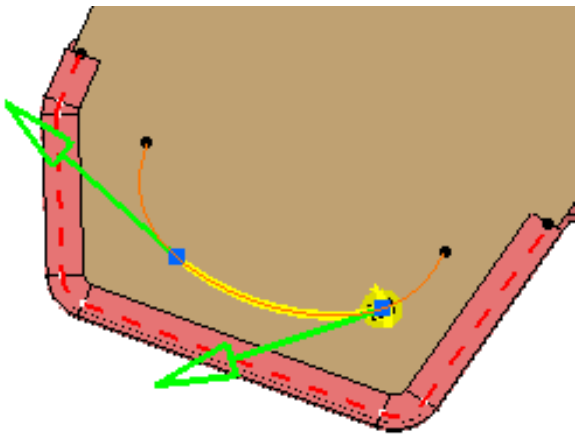
- This option is only available with all the location methods but the Explicit location method.
- This option must be activated to be able to define the Start and End points values.

The **Repeat object after OK** button appears in the BiW CurveBead Fastener Definition. Refer to the Repeating CurveBeads to create more curvebeads using the currently created spot point as reference.

The From Curve On Surface Location dialog box opens.

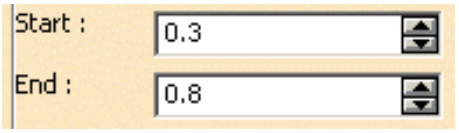


3. Enter 0.2 for the Start point.
4. Enter 0.7 for the End point.
5. Click **Preview** in the BiW CurveBead Fastener Definition dialog box to preview the curvebead.

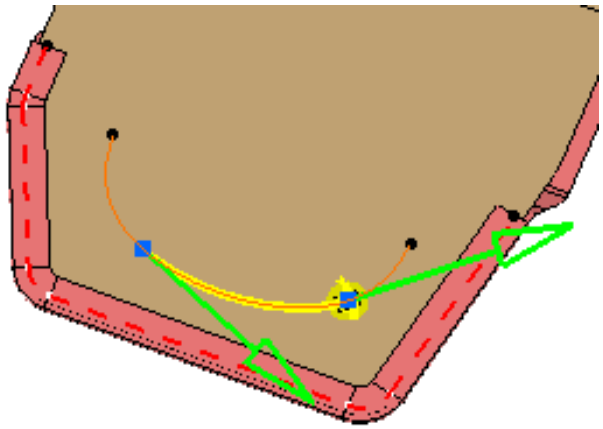


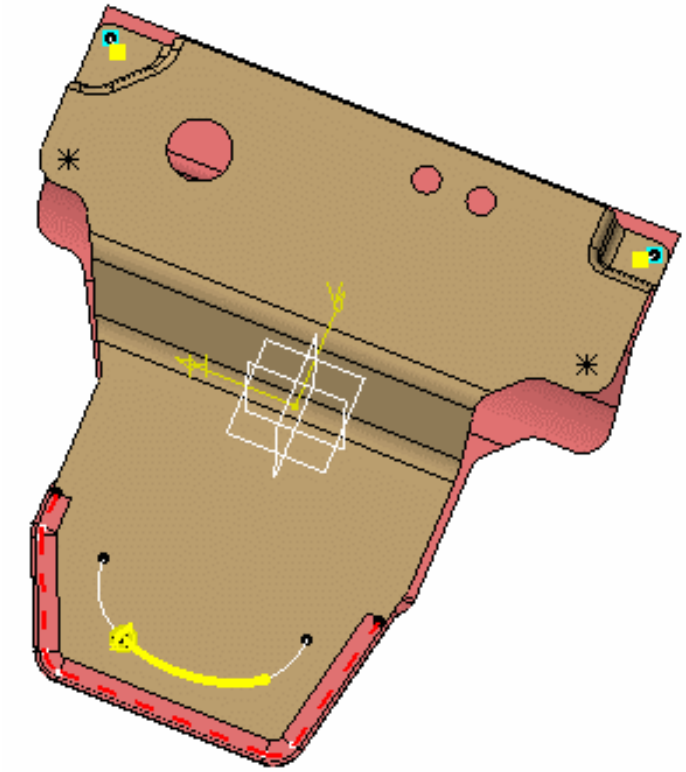
6. Click the **Swap Start and End Points** button.

The Start and End point coordinates are automatically recalculated and new values are displayed within the dialog box, but the curvebead position within the geometry does not change.



The curvebead is modified with its start and end points recalculated.





7. Click OK to create the trimmed curvebead.

Note that:

- the selection of the curve must be done within the linked parts, except when for the Explicit location method.
- When selecting the curve (except for the Explicit location method), it is recommended to select published elements in order to guaranty associativity between elements.
- to authorize the selection of only published elements, check the following option using **Tools -> Options -> Infrastructure -> Part Infrastructure -> General -> Only use published elements for external selection keeping links**.
- when the **On Support Surface** and **Explicit** methods are activated, the application will ignore the active Part Infrastructure setting **Only use published elements for external selection** and will enable the usage of non published external geometry.

Creating Adhesive CurveBeads

This task shows how to create a BiW Adhesive CurveBead.

There are several ways to create a curvebead:

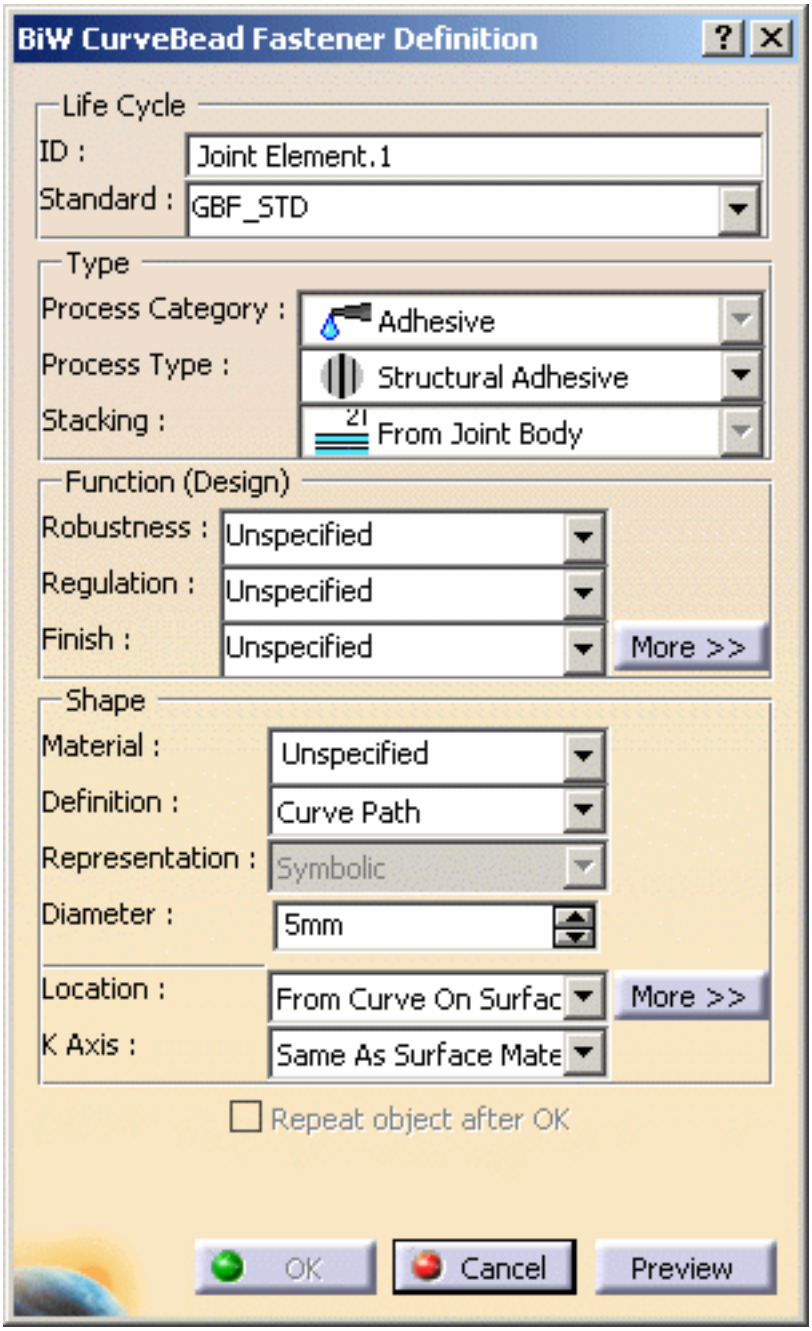
- a joint and a jointbody are already created: click the desired icon and select the joint body in the selection tree.
- a joint and a jointbody are already created: select the joint body in the selection tree and click the desired icon (here is the example used for our scenario).
- no joint body is created. Select the components or the publications, and click the desired icon: a joint and a joint body are created if needed.

Open the ABF_CurveBead.CATProduct document.

Make sure the Curvebead Fastener Type is set up in the standard file.

1. Select the joint body in the specification tree.
2. Click the **BiW Adhesive CurveBead**  icon from the Adhesive toolbar.

The BiW CurveBead Fastener Definition dialog box opens.



3. Specify whether you wish to use the existing standard or not.

   If a standard has been imported, a curvebead is created using this standard. If not, you are able to define your own values for each attribute.

4. Define the curvebead's following parameters:

- Type
- Functional parameters
- Shape
- Location
- K Axis
- Swapping Start and End Points

5. Click **OK** to create the curvebead along the original curve.

   The curvebead (identified as Joint Element.xxx) is added to the specification tree, under the Joint Body node.

Creating Sealant CurveBeads

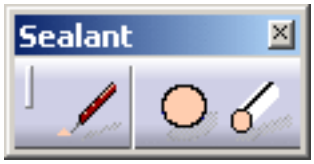
This task shows how to create a BiW Sealant CurveBead.
There are several ways to create a curvebead:

- a joint and a jointbody are already created: click the desired icon and select the joint body in the selection tree.
- a joint and a jointbody are already created: select the joint body in the selection tree and click the desired icon (here is the example used for our scenario).
- no joint body is created. Select the components or the publications, and click the desired icon: a joint and a joint body are created if needed.

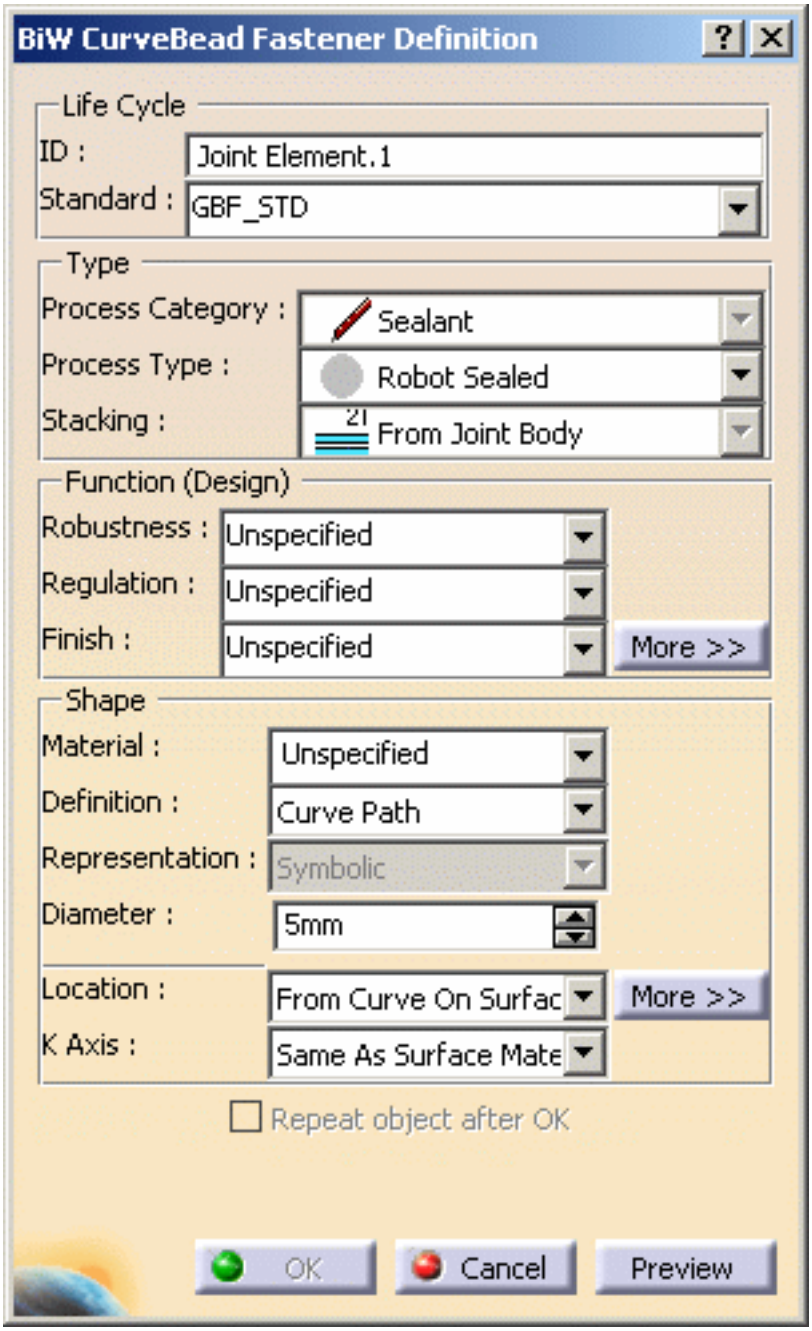
Open the ABF_CurveBead.CATProduct document.

Make sure the Curvebead Fastener Type is set up in the standard file.

1. Select the joint body in the specification tree.
2. Click the **BiW Sealant CurveBead** icon from the Sealant toolbar.



The BiW CurveBead Fastener Definition dialog box opens.



3. Specify whether you wish to use the existing standard or not.

   If a standard has been imported, a curvebead is created using this standard. If not, you are able to define your own values for each attribute.

4. Define the curvebead's following parameters:

- Type
- Functional parameters
- Shape
- Location
- K Axis
- Swapping Start and End Points

5. Click **OK** to create the curvebead along the original curve.

   The curvebead (identified as Joint Element.xxx) is added to the specification tree, under the Joint Body node.

Creating Unspecified CurveBeads

This task shows how to create a BiW Unspecified CurveBead.
There are several ways to create a curvebead:

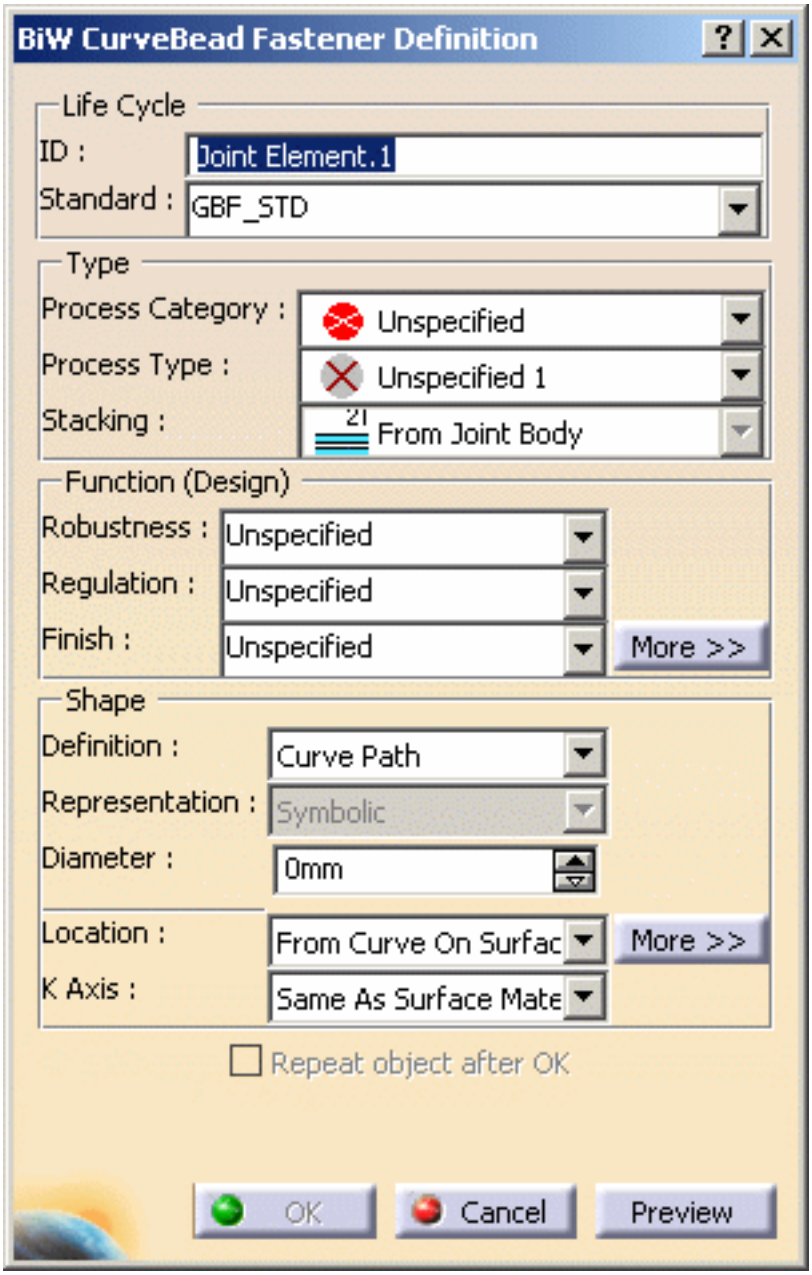
- a joint and a jointbody are already created: click the desired icon and select the joint body in the selection tree.
- a joint and a jointbody are already created: select the joint body in the selection tree and click the desired icon (here is the example used for our scenario).
- no joint body is created. Select the components or the publications, and click the desired icon: a joint and a joint body are created if needed.

Open the ABF_CurveBead.CATProduct document.

Make sure the Curvebead Fastener Type is set up in the standard file.

1. Select the joint body in the specification tree.
2. Click the **BiW CurveBead**  icon from the Unspec Process toolbar.

The BiW CurveBead Fastener Definition dialog box opens.



3. Specify whether you wish to use the existing standard or not.

   If a standard has been imported, a curvebead is created using this standard. If not, you are able to define your own values for each attribute.

4. Define the curvebead's following parameters:

- Type
- Functional parameters
- Shape
- Location
- K Axis
- Swapping Start and End Points

5. Click **OK** to create the curvebead along the original curve.

   The curvebead (identified as Joint Element.xxx) is added to the specification tree, under the Joint Body node.

Repeating CurveBeads

This task shows how to repeat BiW curvebeads along a selected curve.

Open the ABF_CurveBead02.CATProduct document.

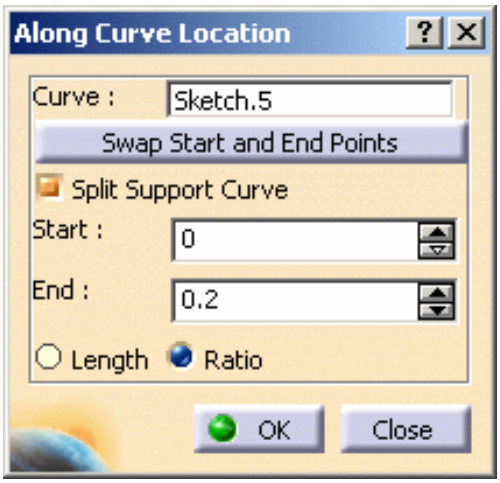
1. Double-click the Joint Element.1 from the specification tree.

The BiW CurveBead Fastener Definition dialog box opens.

2. Select the **Along Curve** option in the location field.
3. Click the **More>>** button to define the location if the dialog box is not already displayed.

The Along Curve Location dialog box opens.

4. Check the **Split Support Curve** button.
5. Define the **Start** and **End** parameters, for instance enter 0 for the Start point and 0.2 for the End point.
6. Click Close.



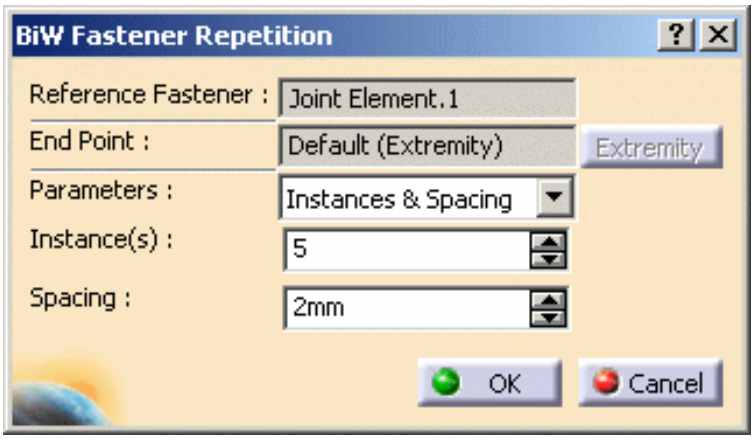
The **Repeat object after OK** option, in the BiW CurveBead Fastener Definition dialog box, is now active.

7. Check the **Split Support Curve** button to be able to display the BiW Fastener Repetition dialog box.
8. Click OK.

The BiW Fastener Repetition dialog box opens.

It shows:

- the reference fastener referring to the currently created spot point.
- the End Point being the extremity of the curve by default. You can choose the other extremity by clicking the Extremity button as well as defining another point on the curve.



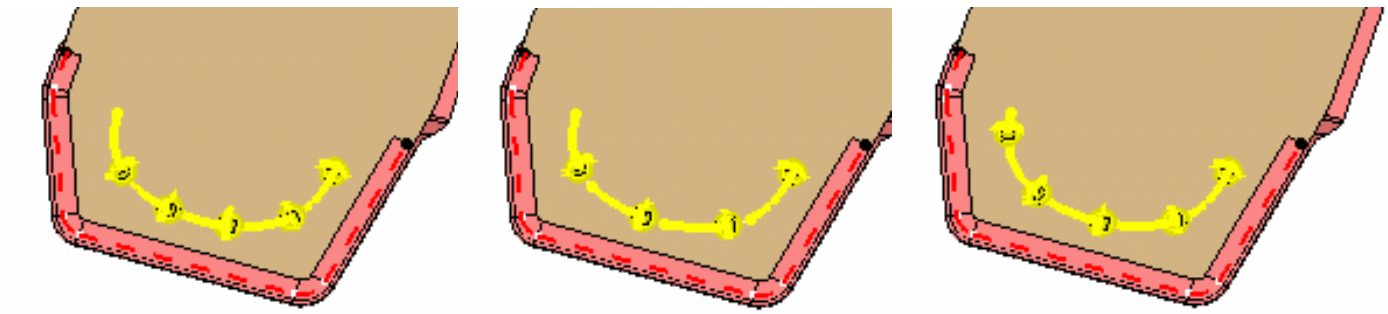
9. Define the parameters:

- **Instance(s)**: defines the number of elements to be repeated.
- **Spacing**: defines the spacing between two successive elements to be repeated. As many curvebeads as possible are created.
- **Instances and Spacing**: defines both the number and the spacing between the elements to be repeated.

10. According to what you chose in the Parameters field, either define a value in the Instances field or in the Spacing one or in both of them.

When entering a value in the Instance field, the distance between two successive points is automatically calculated, curvebeads are equidistant and the last repeated curvebead is positioned on the curve end point.

11. Click OK to validate the repetition.



Instances (4) *Spacing (15mm)* *Instances and Spacing (4 & 7 mm)*

The repeated curvebeads are created along the selected curve as defined in the BiW Fastener Repetition dialog box.

- All repeated curvebeads are exactly the copy of the reference curvebead, therefore have the same length as the reference curvebead.
- All repeated curves are located on the same curve as the reference element.

This capability is available with all the location methods but the Explicit location method.

Using Tools

Creating Features using the Datum Mode
Displaying Joined Parts in a Balloon

Creating Features Using the Datum Mode

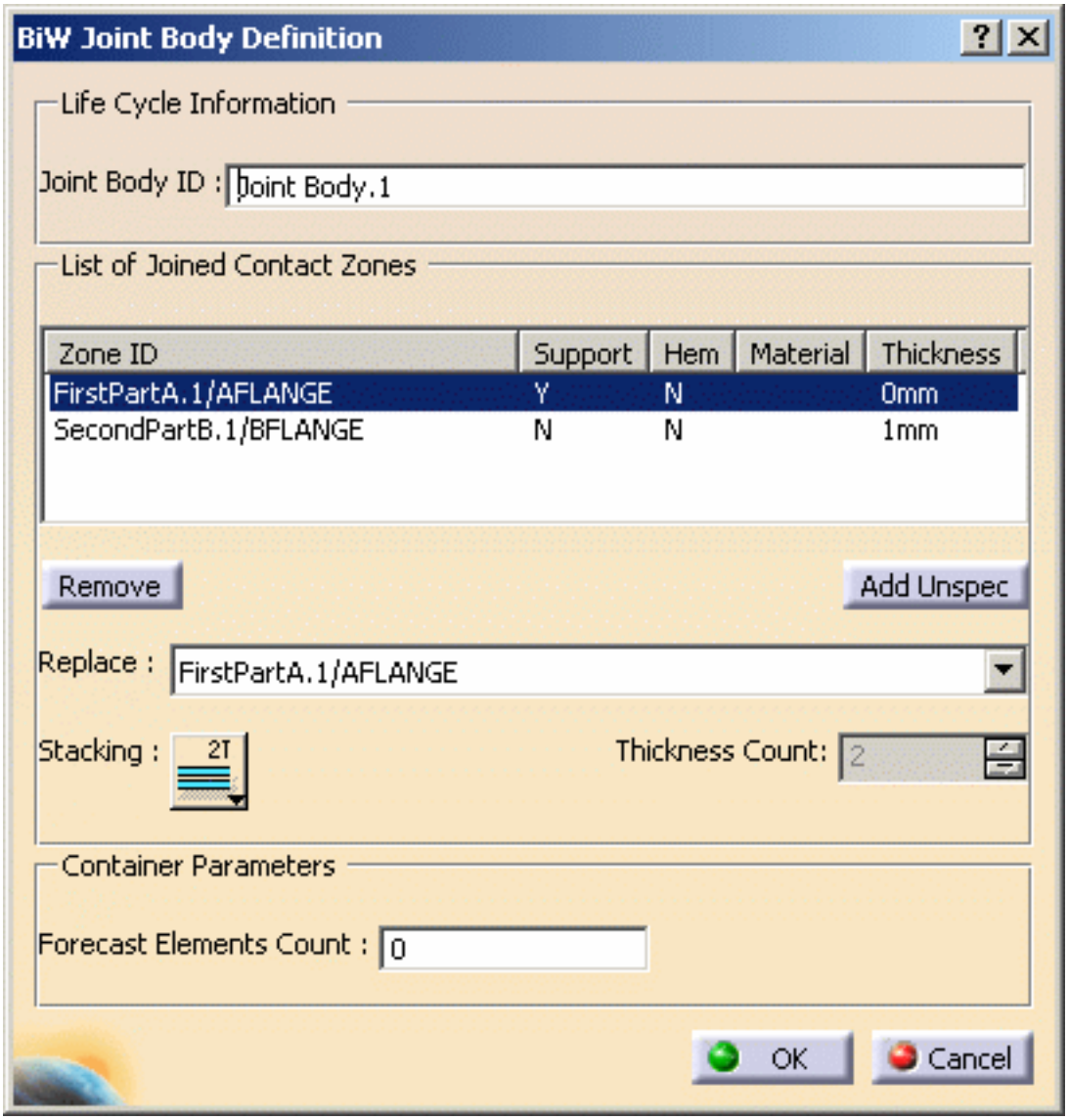
This task shows how to set joint bodies as unspecified and BiW features as explicit.

Open the Datum1.CATProduct document.

Creating Unspecified Joint Bodies

1. Click the **Datum Mode**  icon from the BiW Tools toolbar.
2. Select a zone in FirstPartA and SecondPartB.
3. Click the **Joint Body**  icon from the **Automotive BiW Fasteners** toolbar.

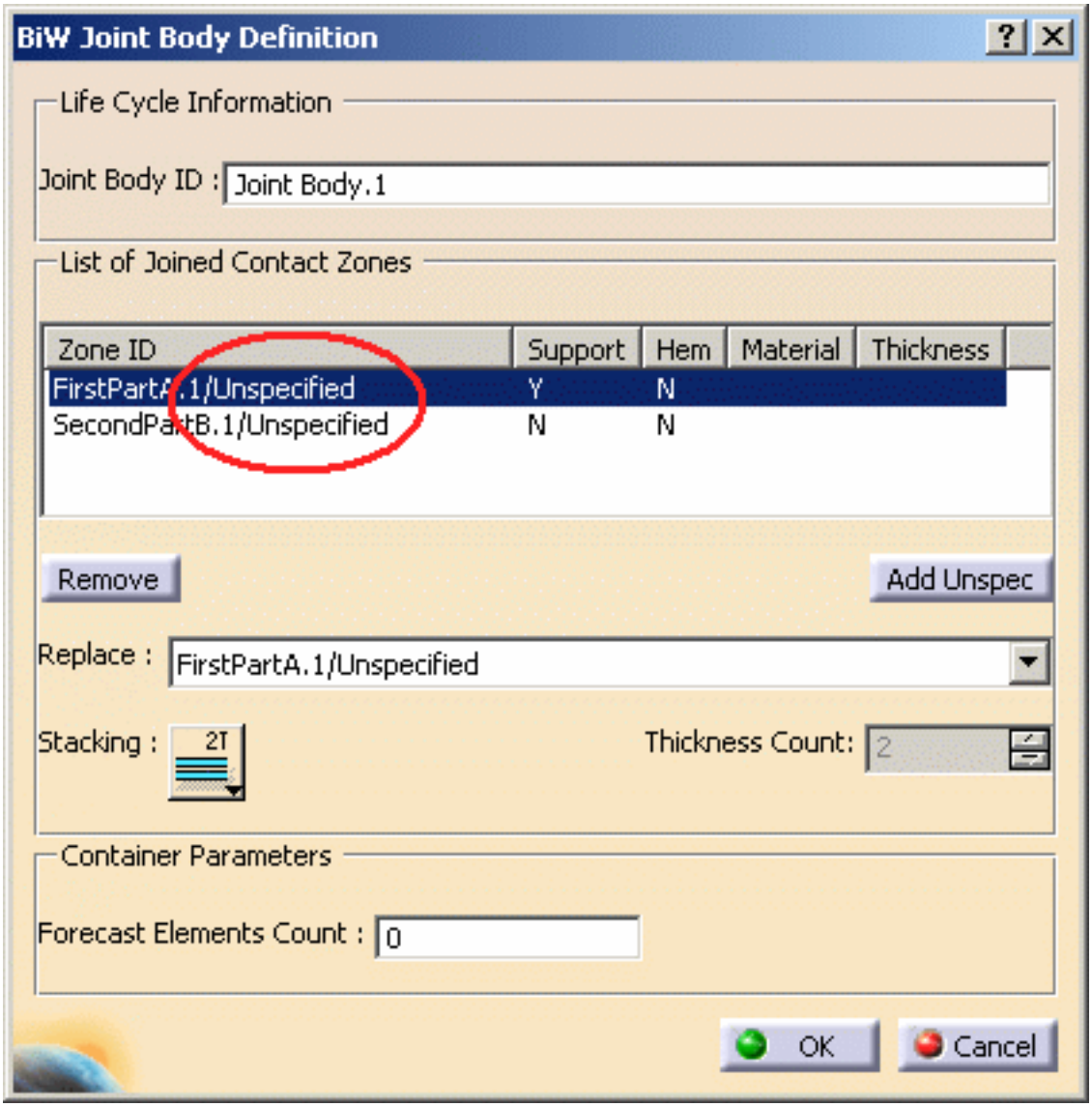
   The BiW Joint Body Definition dialog box opens.



4. Click OK to validate the unspecified joint body creation.

5. Double-click the joint body from the specification tree.

The BiW Joint Body Definition dialog box opens again and you can see that its zones have been swapped to Unspecified.

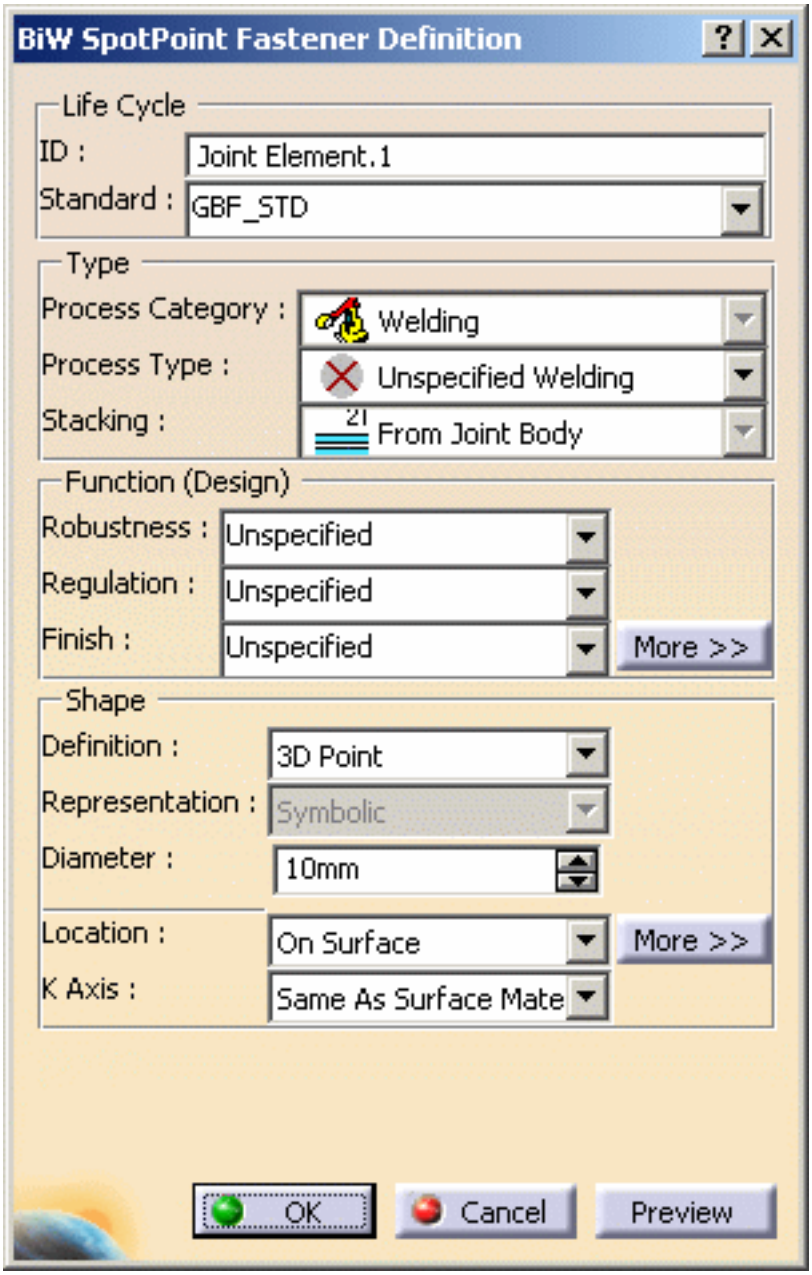


Creating Explicit Joint Elements

1. Click the **Datum Mode** icon from the Tools toolbar.
2. Select a zone in FirstPartA and SecondPartB.
3. Select the joint body from the specification tree.
4. Select a joint element, for instance the **BiW Welding SpotPoint** .

   The BiW SpotPoint Fastener Definition dialog box opens.

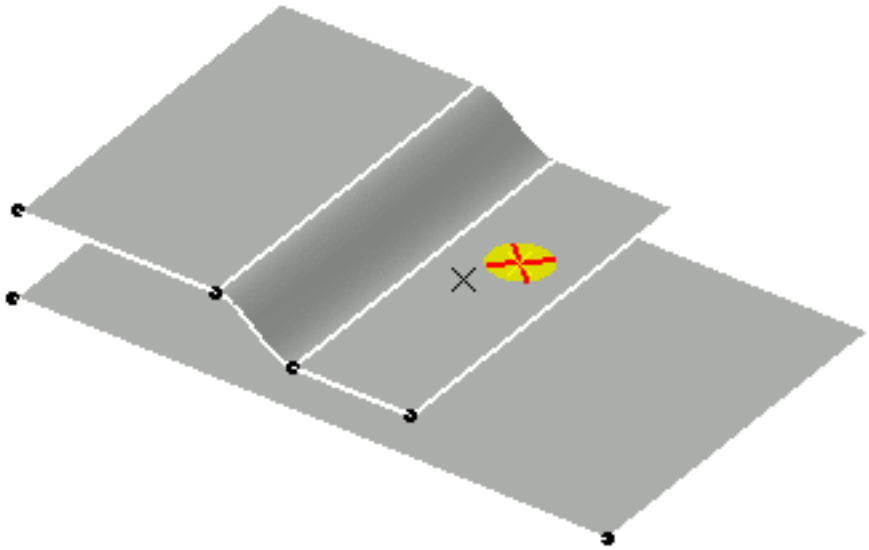
5. Select a location method, for instance **On Surface**.



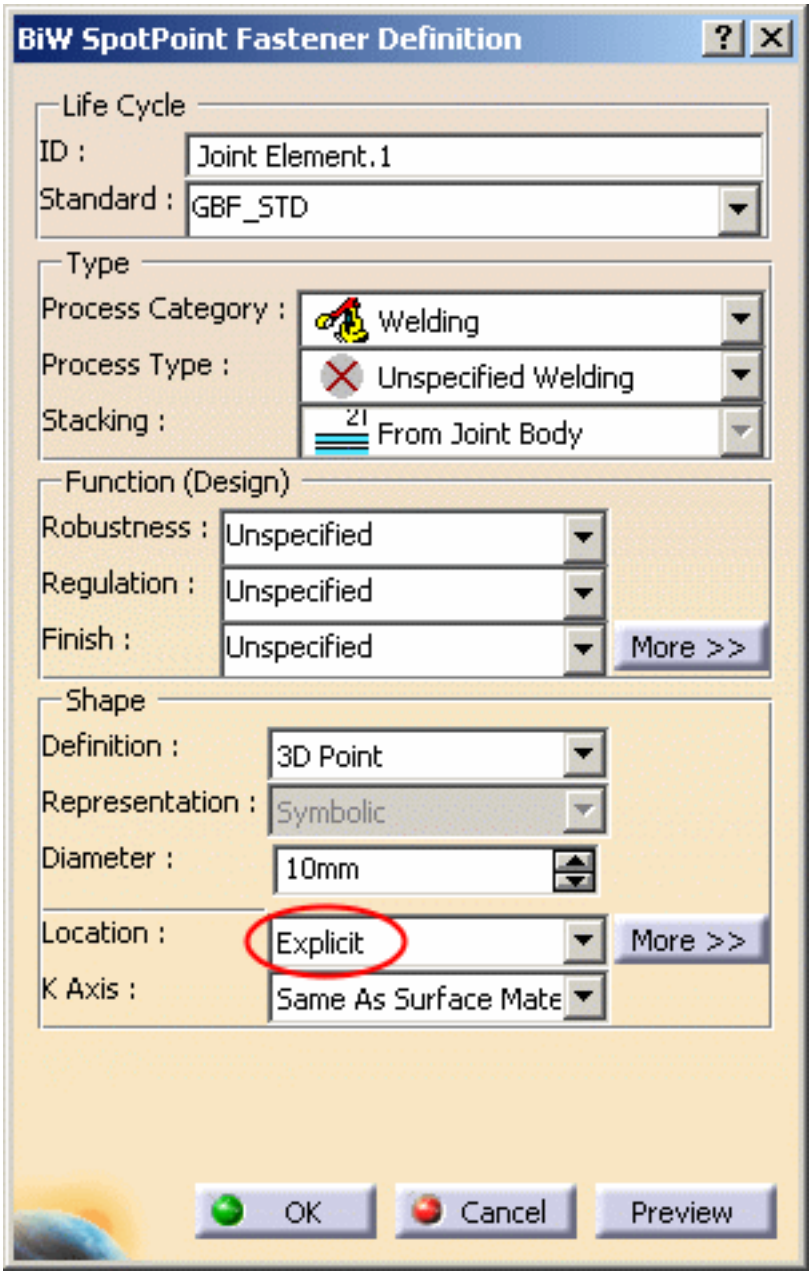
6. Click OK to create the joint element.

   The spotpoint is created.

7. Double-click the spotpoint either from the specification tree or the geometry.



The BiW SpotPoint Fastener Definition dialog box opens once again and you can see that its location has been swapped to Unspecified.



- The specification part is not deleted even if it is empty. It can be deleted by performing an isolate on the joint.
- When Datum Mode is activated, the ABF application will ignore the active Part Infrastructure setting **Only use published elements for external selection** and will enable the usage of non published external geometry.

Displaying Joined Parts in a Balloon

This task shows you how to display the name of the joined parts in a balloon.

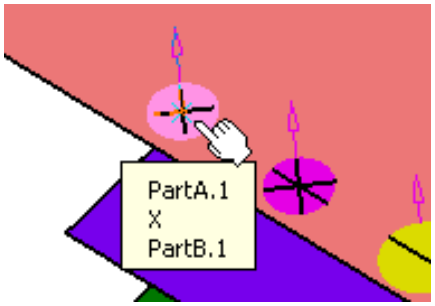
Open the Product0.CATProduct document and click the **Display Joined Parts in a Balloon** icon from the BiW Tools toolbar.

Displaying Joined Parts in a Balloon from a Fastener in the 3D Geometry

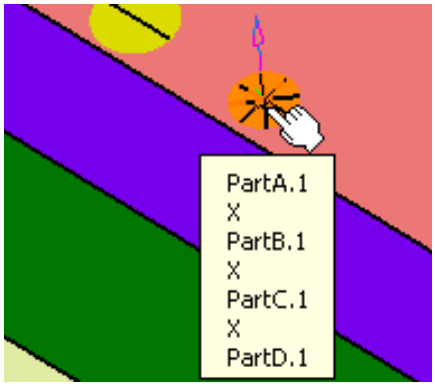
1. Point out a fastener symbol in the 3D geometry.

   The instance names of the joined parts are displayed in a balloon.

   Each joined part is separated by the cross symbol X.



Two joined parts



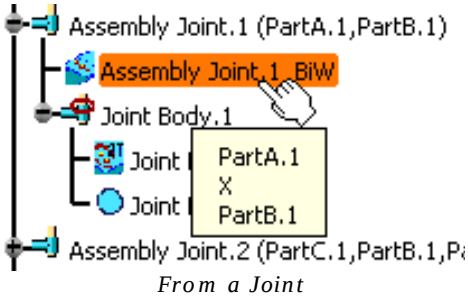
Four joined parts

Displaying Joined Parts in a Balloon from a Joint, a Joint Body, or a Fastener in the Specification Tree

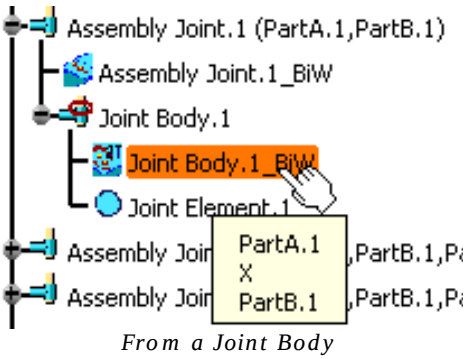
1. Point out a joint, a joint body, or a fastener in the specification tree.

   The instance names of the joined parts are displayed in a balloon.

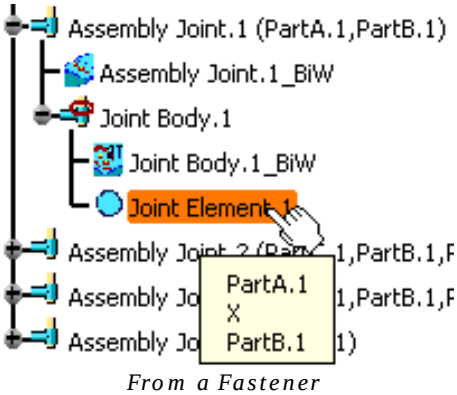
   Each joined part is separated by the cross symbol X.



From a Joint



From a Joint Body



From a Fastener

- The order of the joined parts in the balloon is the same as in the joint in the specification tree.
- This command can be launched while in another command.

The joined part numbers are not displayed.

Reporting

Creating Structural Reports
Creating Flat Reports
Reporting Using CATUtil
Reporting Using a Batch

Creating Structural Reports

This task shows how to report data into a .html file.

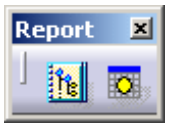
A .CATProduct document must be loaded.
You can also use the ABF_AllTypeFast_01.CATProduct document.

1. Select the objects for which you wish to generate a report. The object(s) and the parent of higher level are selected.

   You can select them either in the specification tree or in the 3D geometry.

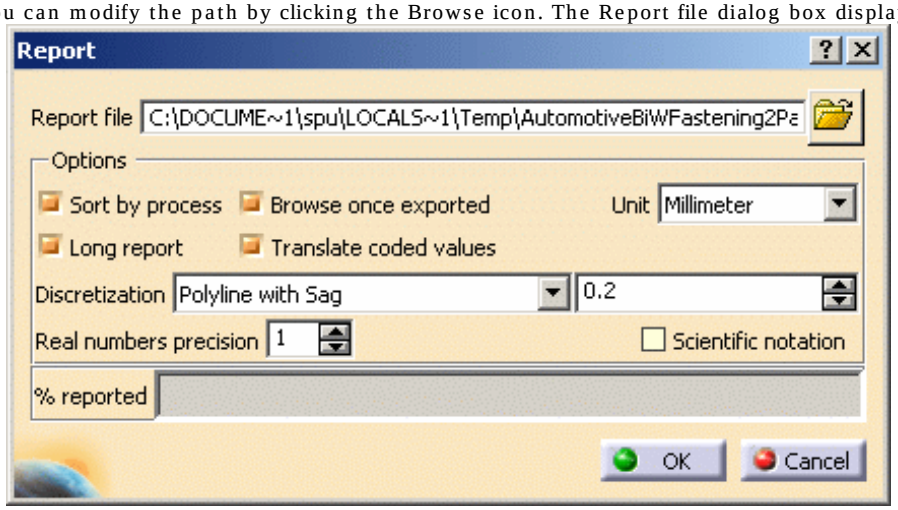
You can modify the selection once in the command.

2. Click the **BiW reporting** icon from the **Automotive BiW Fasteners** toolbar.



The Report dialog box opens.

The directory for the report file is the one defined in Customizing General Settings.
You can modify the path by clicking the Browse icon. The Report file dialog box displays letting you choose the file.



The dialog box enables you to customize the report through several options:

- Sort by process: fasteners are grouped by process category and type. If a standard has been imported, the process types are ordered accordingly.
- Long report: projection results and specific process category data are added for each fastener
- Browse once exported: the file opens in your current html browser once you click OK.
- Translate coded values: attribute values belonging to a predefined list of codes are translated according to the NLS set-up. For instance if the value of the joint element's finish attribute is "B", using this option will display the translation "Class B" in the report.
- Real numbers precision: corresponds to the number of decimal digits.
- Unit: either millimeter or inch
- Scientific notation: power of 10
- Discretization: specify the discretization method:
  - Polyline with sag: define the sag value used in the exported file.
  - Polyline with step: define the value used in the exported file.

3. Click **OK**.

   The progress bar shows you the remaining exported percentage.

The CATIA BiW Fasteners report opens. It displays:

- General information:
  - the date
  - the product name
  - the unit

Date	23 April 2004
Product	ABF_AllTypeFast_01
Unit	1 mm (Millimeter)

- a summary containing (number of features) fasteners sorted by process type and process category.

Summary

BiW Joint count	1
BiW Spot Point / Adhesive / Structural Adhesive count	1
BiW Spot Point / BiW Mechanical / Punch Rivet count	1
BiW Curve Bead / Sealant / Robot Sealed count	1
BiW Spot Projection / Welding / Projection Resistance (23) count	1
BiW Fastener count	4

Fasteners belonging to a same process type or process category are displayed in the same order as they appear in the specification tree.

The following tables are contained into domains:

- Joint attributes
  - joined components
    - Part path ID, Part Number, Version, Material
- joint bodies
  - BiW Body Name, Stacking, Thicknesses Count, Zone ID, Layer number for each joined contact zone, etc.
- BiW Fasteners sorted by process type and process category (if option checked)
  - general attributes: BiW Joint and Joint Body Names, Process Category, K Flag, I Flag, Layer number, X, Y, Z coordinates of the fastener's reference tangent vector (U), etc.
  - projection attributes (only in long reports): Zone path ID, Normal and projection point coordinates, Material, Thicknesses.
- Curvilinear fasteners discretization attributes (only in long reports)
  - Point Count, for each point the localization coordinates (X, Y, Z), the reference normal (W) and the tangent vector (U).
- Projection fasteners attributes (only in long reports)
  - ID, Projection zone

- You can only report BiW objects.
- A warning message is issued in case errors occur during the report process. They are logged in a .xml file. Refer to the Import chapter for more information.
- One single log file is issued per structural report.
- Deactivated features are not reported; they appear in the log file.
- If no object is selected, all the active BiW objects as long as the sub-products are reported.
- The products coordinates are reported relatively to the active product coordinates.
- The header always displays the active product information even if the report is for an assembly.
- When you have just created a feature, the latter is taken into account when clicking the BiW reporting icon. Expand the specification tree to ensure you selected the correct objects.

Creating Flat Reports

This task shows how to report data into a flat table of tab-delimiting text.

A .CATProduct document must be loaded.
You can also use the ABF_AllTypeFast_01.CATProduct document.

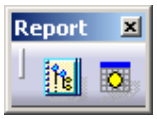
1. Select the joint elements for which you wish to generate a report. The element(s) and the parent of higher level are selected.

   You can select them either in the specification tree or in the 3D geometry.

   If you select a joint or a joint body, its children fasteners are highlighted in the specification tree.

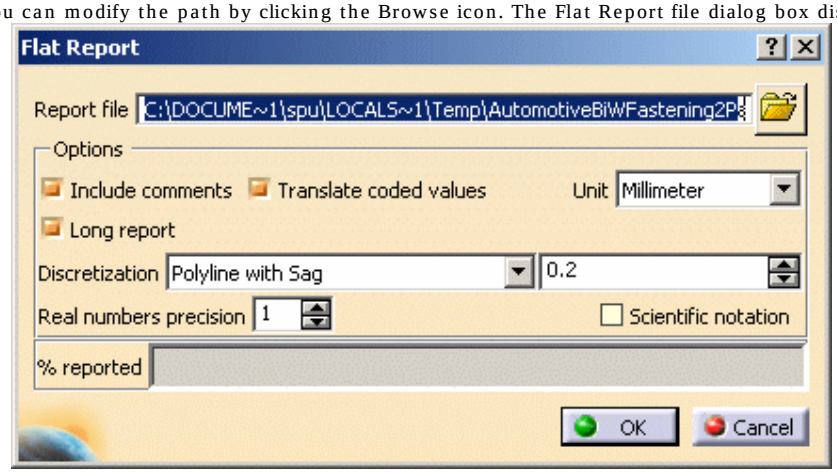
You can modify the selection once in the command.

2. Click the **BiW flat reporting** icon from the BiW Fasteners Report sub-toolbar of the Automotive BiW Fasteners toolbar.



   The Flat Report dialog box opens.

The directory for the report file is the one defined in Customizing General Settings.
You can modify the path by clicking the Browse icon. The Flat Report file dialog box displays letting you choose the file.



The dialog box enables you to customize the report through several options:

- Include comments: specifies that the report should start with the commented header block showing keywords description.
- Long report: projection results and specific process category data are added for each fastener
- Translate coded values: attribute values belonging to a predefined list of codes are translated according to the NLS set-up. For instance if the value of the joint element's finish attribute is "B", using this option will display the translation "Class B" in the report.
- Real numbers precision: corresponds to the number of decimal digits.
- Unit: specifies the unit for output length values which can either be millimeter or inch.
- Discretization: the value affects long reports only as discretization points of curvebead are not included in short reports.
  - Polyline with sag: define the sag value used in the exported file.
  - Polyline with step: define the step value used in the exported file.
- Scientific notation: power of 10.

3. Click **OK**.

   The progress bar shows you the remaining exported percentage.

4. Open the .txt file.

The file opens. It displays:

- one row per exported fastener containing all its attributes plus its parents ones.
- the joint element data (ID, type, diameter, etc...).
- the parents data (Joint Body ID, Joint ID).
- the joint crossed thicknesses information (count, part number, material).

Fasteners belonging to a same process type or process category are displayed in the same order as they appear in the specification tree.

General Structure of the Flat Report

Structure of the Commented Block

The flat report begins with an optional commented block containing the following information:

- A comprehensive description of the keywords used as column titles.

```
//===========================================================
//CATIA V5 BiW Automotive Product Flat Report Format
//===========================================================
//Keyword definitions:
//--------------------
//       HEADER  Header description
//       VERSION Version of the CATIA BiW Fastening data exchange neutral format
//       DATE    Date when exported
//       PRODUCT Product name
//       UNIT    Unit used for length values
//       MATHDEF Mathematical definition (or discretization method) of a curvilinear fast
//       POL_DISCRETIZATION_PARAM        Parameter value used to discretize a curvilinear
//       BfmJointElement BiW Fastener
//       JEID    ID
//       PCA     Process Category
//       PTY     Process Type
//       FTY     Fastener feature type
//       SDF     Shape definition (0=Point3D,1=Hemisphere inwards material,3=Hemisphere
//       ROB     Robustness
//       REG     Regulation
//       FIN     Finish
//       GFL     Geometry Flag
//       IFL     Inspection Flag
//       MID     Manufacturing Code
//       MAT     Material
//       DIA     Diameter
```

- Some header information such as the product name and the report date issue.

```
//===========================================================
//       HEADER_START
//               VERSION 3.1
//               DATE    18 November 2003
//               PRODUCT ABF_WSpot_ABead_01
//               UNIT    1 mm (Millimeter)
//               MATHDEF Polyline (Sag criterium)
//               POL_DISCRETIZATION_PARAM        0.2
//       HEADER_END
//===========================================================
```

- The column titles as keywords described above.

Version 3

```
//JEID  JBID     JID      PCA      PTY      FTY      SDF      ROB      REG      FIN      GFL      IFL
Joint Element.1 Joint Body.1    Assembly Joint.1          Welding Resistance (21) BiW Spot Point
Joint Element.2 Joint Body.1    Assembly Joint.1          Adhesive         Structural Adhesive
```

Version 3.1

```
//JEID  JBID     JID      PRD_PN  PRD_VERS         PRDINST_PATHID  PCA      PTY
Joint Element.1 Joint Body.1    Assembly Joint.1          ABF_WSpot_ABead_01
Joint Element.2 Joint Body.1    Assembly Joint.1          ABF_WSpot_ABead_01
```

Version 4

```
//JEID  JBID     JID      PRD_PN  PRD_VERS         PRDINST_PATHID  LAYER   PCA
Joint Element.1 Joint Body.1    Assembly Joint.1          ABF_AllTypeFast_01
Joint Element.2 Joint Body.1    Assembly Joint.1          ABF_AllTypeFast_01
```

Structure of the Results Block

The flat report contains different attributes which are spread over tab separated columns:

- Fasteners' attributes
  - ID, type, diameter, layer number, etc.
- Fasteners parents' attributes
  - Joint and joint bodies
- Joined contact zone attributes
  - Path ID, thickness, material, layer number, etc.
- Additional crossed thicknesses attributes (only in long reports)
  - PROJ, Normal and projection point coordinates
- Curvilinear fasteners discretization attributes (only in long reports)
  - Point Count, for each point the localization coordinates (X, Y, Z), the reference normal (W) and the tangent vector (U) coordinates.
- Projection fasteners attributes (only in long reports)
  - ID, Projection zone

- You can only report BiW objects.
- A warning message is issued in case errors occur during the report process. They are logged in a .xml file.
  Refer to the Import chapter for more information.
- One single log file is issued per flat report.
- Deactivated features are not reported; they appear in the log file.
- A selection of BiW fasteners to be reported can be made prior to entering the command. This selection can be modified at all time within the command.
- If no BiW entity is selected, the flat report will contain all fasteners found in the active product, including those located within all sub-product instances. In this case, the whole product structure is scanned recursively, starting from the active product.
- The products coordinates are reported relatively to the active product coordinates.
- The header always displays the active product information even if the report deals with an assembly.

Reporting Using CATUtil

This task shows you how to work with a batch, whose inputs are defined through a dialog box, in order to generate all types of report files (report, flat report, export).
A batch V5 is an non interactive program requiring an xml file that includes a parameter File (containing its inputs and outputs)

and that provides:

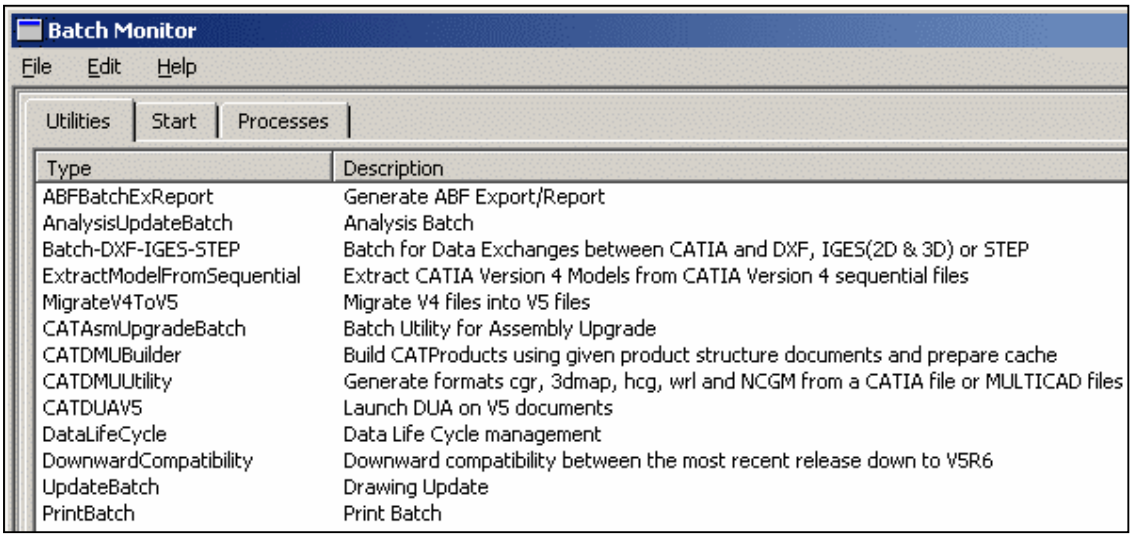
- Batch process (export or report) result file (.txt or .xml formats)
- Log output xml file

No product needs to be open.
Launch CATUtil. There are two ways to do it:

a. Select **Tools -> Utility**.

b. From a MS-DOS window, change to the default folder where the product is stored, then type the **CATUtil** command.

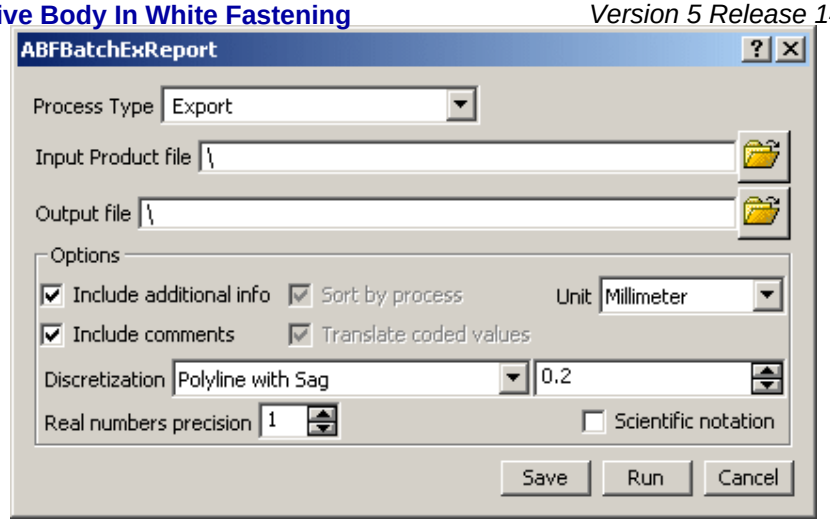
The Batch Monitor dialog box displays, listing all available batches.



Creating a New Parameter File

1. From the Utilities tab, right-click ABFBatchExReport and select the **New parameters file** contextual menu.

The ABFBatchExReport dialog box opens.



2. Select the Process Type:
   - **Export**
   - **Report**
   - **Flat Report**

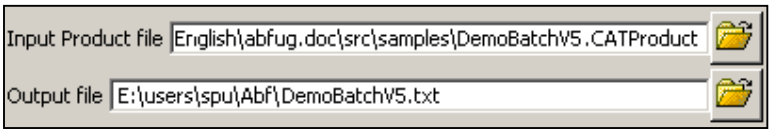
3. Select a .CATProduct document as the **Input Product file**. The default path is the one of the CATBatch_HOME variable.

4. Select a .txt file as the **Output file**.

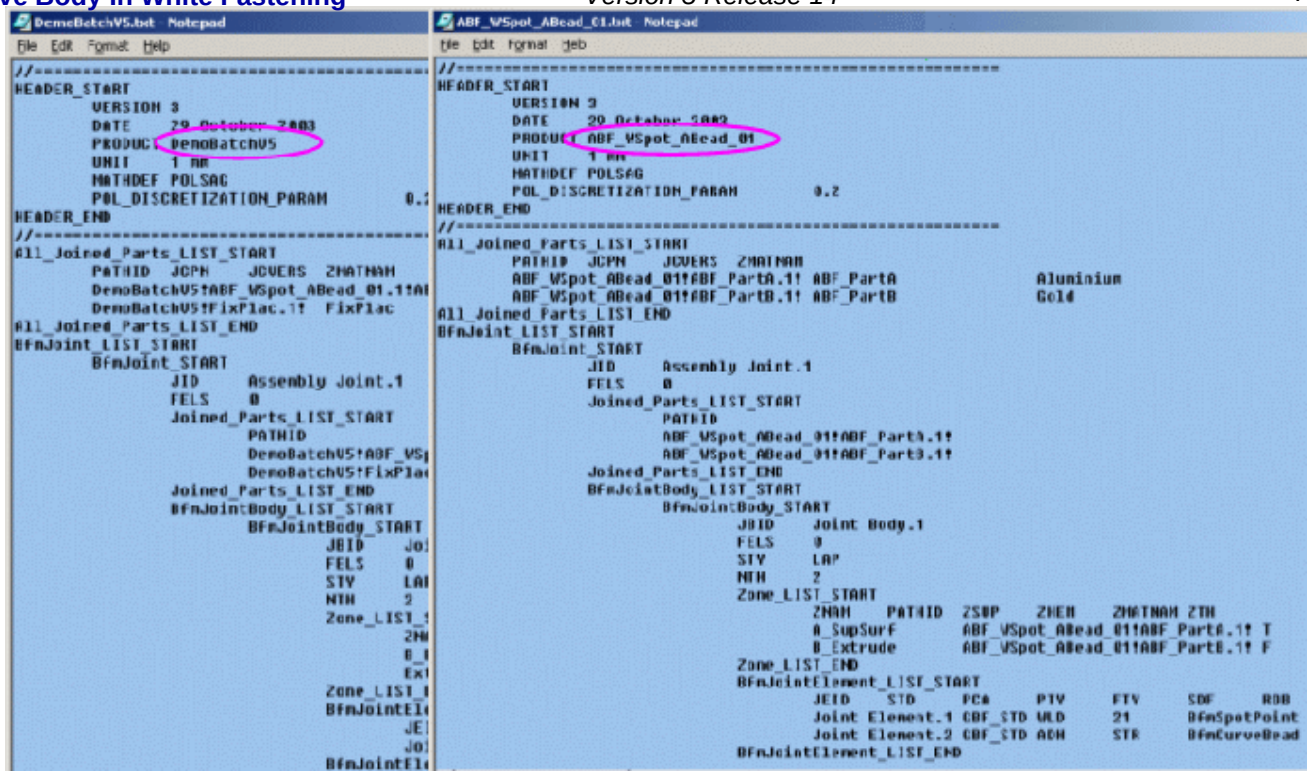
   The output file is associated to the Input Product file and its name is the same as the Input Product file.

   If no output file is selected, the path is defined as in **Tools -> Options -> Shape -> Automotive BiW Fastening -> General** tab.

   One output file is generated per reference product.



Here is an example of an output file with a product and a sub-product.



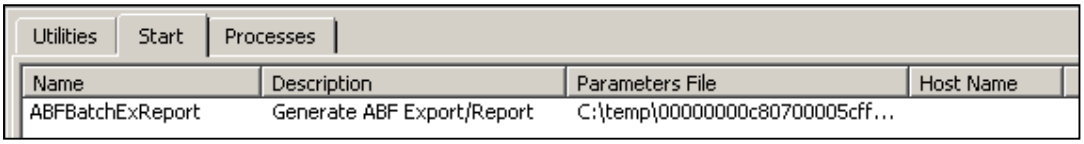
5. Define the options.

   For further information, please refer to the Export chapter.

Saving the Export Batch

Click **Save** to save the xml file in a directory.

The xml parameter file is automatically generated and appears as an argument for the start of the ABFBatchExReport utility.



Running the Export Batch

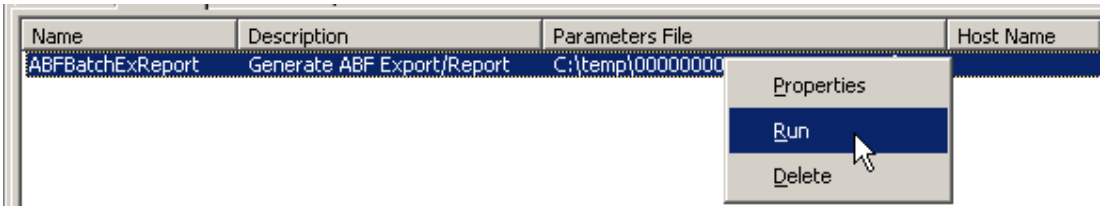
From the ABFBatchExReport dialog box

Click **Run** to run the batch process.

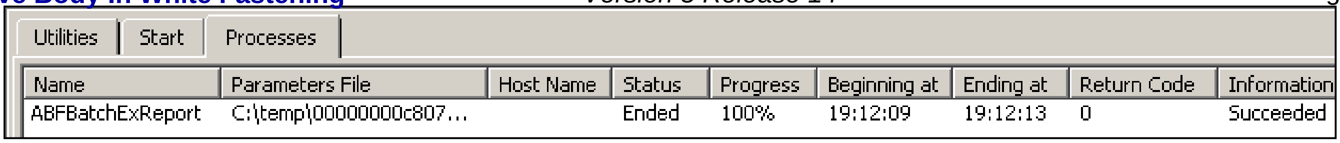
An xml parameter is generated in a default repository.

From the Start tab

Right-click **Run** to generate the report again.



You can also generate a report using a batch with a parameter file (CATBatchXMLFile)



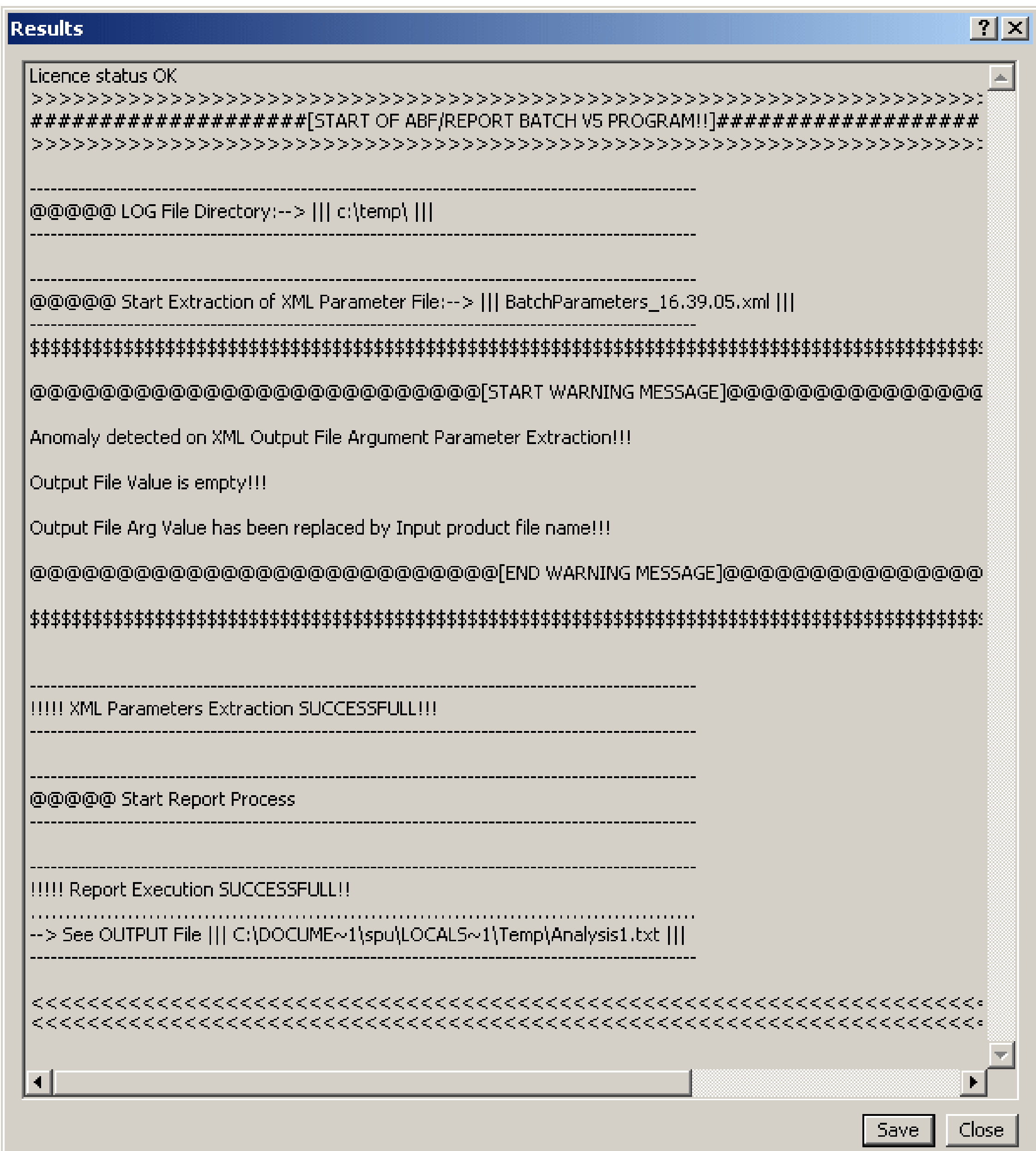
The Process tab displays a status of how the process went on.

The following information is displayed:

- name of the batch
- name of the parameters file used to run the batch
- status of the batch ("Ended", etc.)
- start and end time, respectively in the "Beginning at" and "Ending at" columns
- return code: "0" means that the batch executed correctly, "-1" means that an error occurred
  result of the batch execution (either "Succeeded" or "Failed")
- batch identification number

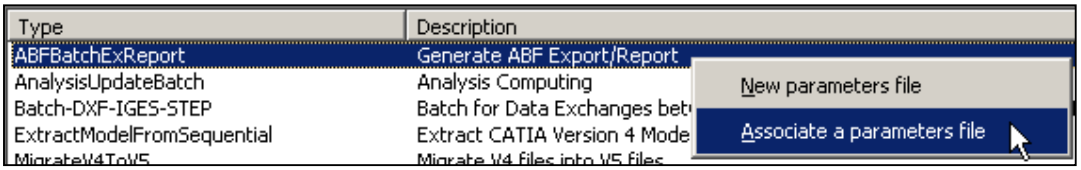
The batch ID corresponds to the directory, named after the batch ID, where the log is stored.
In case warnings are detected, a warning log is added to the existing log file.

8. Right-click the ABFBatchExReport and select the **Results** contextual menu.



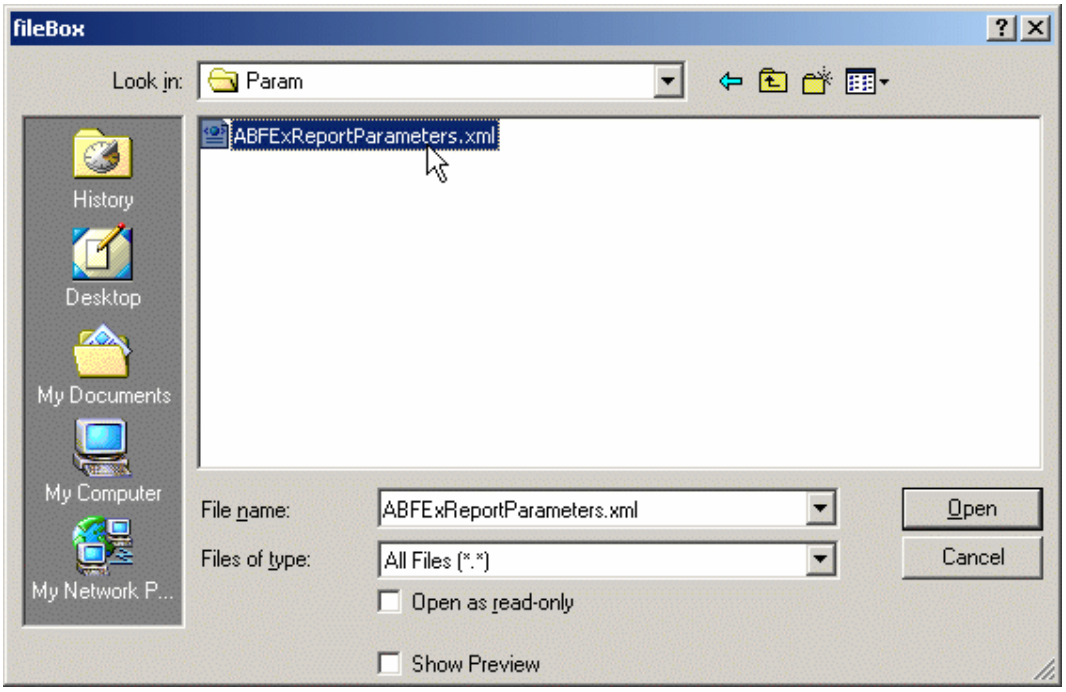
Associating an Existing Parameter File

1. From the Utilities tab, right-click ABFBatchExReport and select the **Associate a parameters file** contextual menu.

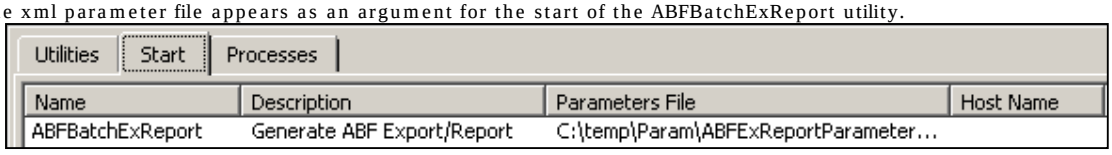


The fileBox dialog box displays.

2. Select an xml parameter file.
3. Click Open.

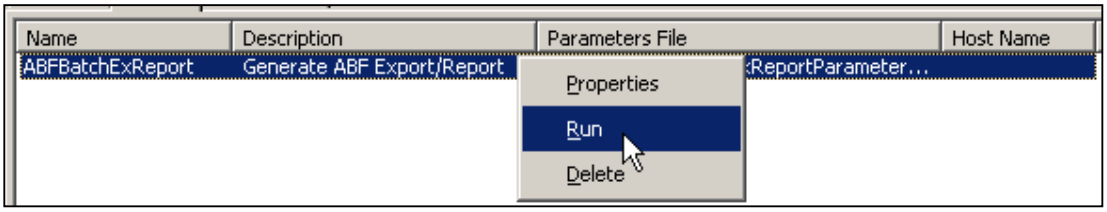


The xml parameter file appears as an argument for the start of the ABFBatchExReport utility.



Running the Export Batch

4. Right-click **Run** to generate the report again.



The Process tab displays a status of how the process went on.

Reporting Using a Batch

This task shows you how to work with a batch in order to generate all types of report files (report, flat report, export).
A batch V5 is an non interactive program which is based on an xml file that includes a parameter File (containing its inputs and outputs).

No product needs to be open.

1. From a command window, launch the following command: **CATBatch ''path of ABFExReportParam_Default.xml file'' _lic setting**.

   A default file can be found in reffiles/AbfBatchParam (intel_a for Windows, aix_a for IBM UNIX, irix_a for SGI UNIX, solaris_a for SUN Unix, hpux_a for HP UNIX).

```
<?xml version="1.0" encoding="UTF-8" ?>
<!DOCTYPE root (View Source for full doctype...)>
- <root batch_name="ABFExReportDesc">                                   → Descriptor File
  - <inputParameters>
      <file id="Product file" filePath="" upLoadable="RightNow" type="bin" />   → Process Type
      <simple_arg id="Process type (Exp,RepS,RepF)" value="Exp" />              → Export/Report Output File Name
      <simple_arg id="Output file" value="ABF_Batch_Output_Default" />
      <simple_arg id="Include additional info (Yes,No)" value="No" />
      <simple_arg id="Include comments (Yes,No)" value="No" />
      <simple_arg id="Sort by process (Yes,No)" value="No" />
      <simple_arg id="Translate coded values (Yes,No)" value="No" />
      <simple_arg id="Inches unit (Yes=Inch,No=mm)" value="No" />
      <simple_arg id="Discretization method (PolSag,PolStep)" value="PolSag" />
      <simple_arg id="Discretization sag" value="0.2" />
      <simple_arg id="Real numbers precision (Decimal digit number)" value="1" />
      <simple_arg id="Scientific notation (Yes,No)" value="No" />
    </inputParameters>                                                      Export/Report Customizable Options
  - <outputParameters>
      <folder id="Output directory" folderPath="" upLoadable="RightNow" type="bin" />   → Export/Report Output Directory
    </outputParameters>
  </root>
```

This file is composed of two main parts:

- Input Parameters
- Output Parameters

Input Parameters

This part is itself divided into two main categories:

- File Part: it includes:
  - the file id (called ''Product File'' and must not be modified by the user), and
  - the filepath (complete path of the product to be filled by the user)
- Argument Part: it includes:
  - the Process Type (Export (Exp), Report (RepS), Flat Report (RepF)) and the output file name, then
  - all the export/report options.

  If an option is wrong, the report is generated with a warning (except for the Process Type)

Output Parameters

It contains the export/report output directory defined by the user:

- folder id: called ''Output Directory'' (and must not be modified by the user)
- folderpath: directory path of the product to be filled by the user

A log file is open each time the batch command is launched. It is stored in the same directory as the CATBATCH_HOME variable.

This result file can either be a .txt or .html file.

- If the product contains several sub-products, as many files as there are sub-products are generated. Each file name corresponds to a product name.
- In case warnings are detected, a warning message is added to the existing log file:

```
Licence status OK
>>>>>>>>>>>>>>>>>>>>>>>>>>>>>>>>>>>>>>>>>>>>>>>>>>>>>>>>>>>>>>>>>>>>>>>>>>>>>>>>>>>>>>>>>>>>
####################[START OF ABF/REPORT BATCH V5 PROGRAM!!]####################
>>>>>>>>>>>>>>>>>>>>>>>>>>>>>>>>>>>>>>>>>>>>>>>>>>>>>>>>>>>>>>>>>>>>>>>>>>>>>>>>>>>>>>>>>>>>

--------------------------------------------------------------------------------------------
@@@@@ LOG File Directory:--> ||| E:\users\fbw\ABF\Dev\ImportExport_ReqTMC |||
--------------------------------------------------------------------------------------------

--------------------------------------------------------------------------------------------
@@@@@ Start Extraction of XML Parameter File:--> ||| DemoBatchV5_Param.xml |||
--------------------------------------------------------------------------------------------
$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$

@@@@@@@@@@@@@@@@@@@@@@@@@@[START WARNING MESSAGE]@@@@@@@@@@@@@@@@@@@@@@@@@@

Anomaly detected on XML Input Argument Parameters Extraction!!!

Bad Input for <Discretization sag> Argument Value!!!

Argument Value has been replaced by the default one:'0.2'!!!

@@@@@@@@@@@@@@@@@@@@@@@@@@@[END WARNING MESSAGE]@@@@@@@@@@@@@@@@@@@@@@@@@@@

$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$


--------------------------------------------------------------------------------------------
!!!!! XML Parameters Extraction SUCCESSFULL!!!
--------------------------------------------------------------------------------------------

--------------------------------------------------------------------------------------------
@@@@@ Start Report Process
--------------------------------------------------------------------------------------------

--------------------------------------------------------------------------------------------
!!!!! Report Execution SUCCESSFULL!!
............................................................................................
--> See OUTPUT File ||| E:\users\fbw\ABF\Dev\ImportExport_ReqTMC\Output\DemoBatchV5.txt |||
--> See OUTPUT File ||| E:\users\fbw\ABF\Dev\ImportExport_ReqTMC\Output\ABF_WSpot_ABead_01.txt
--------------------------------------------------------------------------------------------

<<<<<<<<<<<<<<<<<<<<<<<<<<<<<<<<<<<<<<<<<<<<<<<<<<<<<<<<<<<<<<<<<<<<<<<<<<<<<<<<<<<<<<<<<<<<
####################[END OF ABF/REPORT BATCH V5 PROGRAM!!]####################
<<<<<<<<<<<<<<<<<<<<<<<<<<<<<<<<<<<<<<<<<<<<<<<<<<<<<<<<<<<<<<<<<<<<<<<<<<<<<<<<<<<<<<<<<<<<
```

Exporting

This task shows how to export fastening data in a neutral format file (.txt for example).

A .CATProduct document must be loaded.
You can also use the ABF_AllTypeFast_01.CATProduct document.

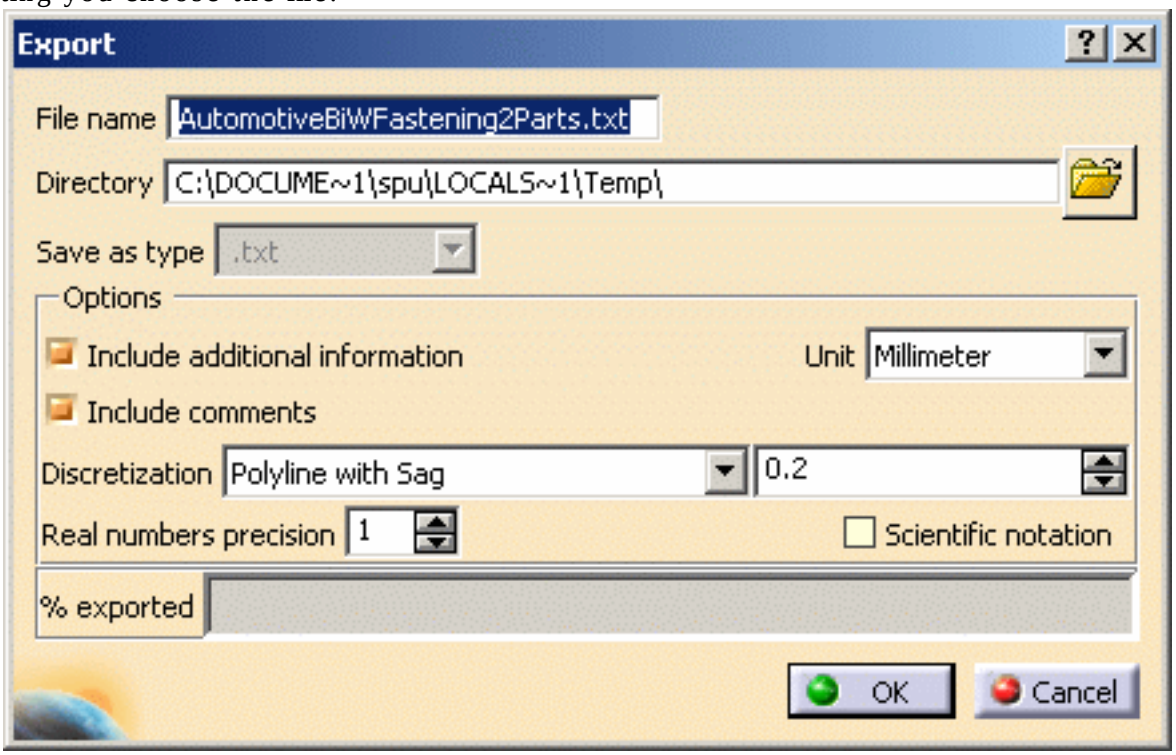
1. Click the **Automotive BiW Fastening data export** icon.

The Export dialog box opens.

It shows:

- the neutral file name
- the directory for the neutral file as defined in Customizing General Settings.

You can modify the path by clicking the **Browse** icon. The Select directory dialog box displays letting you choose the file.

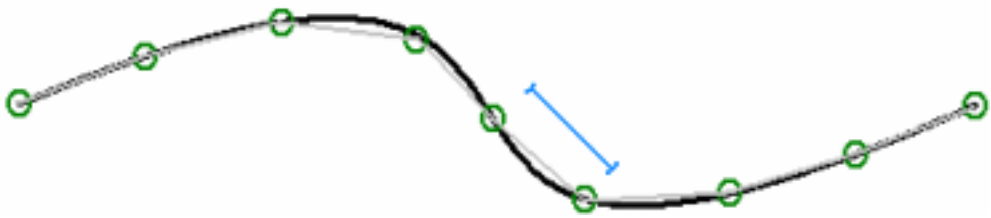


The dialog box enables you to customize the neutral file through several options:

- Include additional information: projection results for example
- Include comments: corresponds to the glossary at the beginning of the neutral file
- Real numbers precision: corresponds to the number of decimal digits
- Unit: either millimeter or inch

- Discretization:
  - Polyline with sag: segments are defined on the curve.
    The maximum sag value calculated from the options set in **Tools -> Options -> General -> Display -> Performances -> 3D Accuracy**.

  - Polyline with step: equidistant points are created on the curve.



When exporting a curvebead, the discretization value set in the Export dialog box only prevails if the Unspecified method is chosen. Otherwise, the value set when creating a curvebead using the sag or step discretization method prevails.

- Scientific notation: power of 10

2. Select one or more elements in the reference product to be included in the export file.

   They can be:

   - joints: the selected joint and all its children (joint bodies and joint elements if any) are highlighted in the specification tree
   - joint bodies: the selected joint body, its parent and children (joint and joint elements if any) are highlighted in the specification tree
   - joint elements: the selected joint elements and its parent (joint and joint body) are highlighted in the specification tree

You can also select several elements from two sub-products. In this case, the File name field is grayed out. As many export files are created as there are sub-products. The name of each export file corresponds to the name of the sub-product.
If no element is selected, the export applies to the whole active product.

3. Click OK.

   The progress bar shows you the remaining exported percentage.

4. Open the file as per the path indicated. Its name is the same as the reference product's. You may change it if you wish.

   The format is tab separated text, allowing to open it in a spreadsheet application.

   The structure of the document is the following:

- Comments
- Header
- Joined parts attributes
  - Path ID, JCPN, ZMATNAM, etc.
- Joint attributes
  - Joint part
  - Joint body attributes
    - Zone attributes (ZNAM, Layer, etc.)
    - Joint element attributes (ID, Process Category, Process Type, Layer, K Flag, IF Flag, Reference Normal (W) and Tangent Vector (U) coordinates, etc.)
    - Curvilinear fasteners discretization attributes (Point Count, for each point the localization coordinates (X, Y, Z), the reference normal (W) and the tangent vector (U) coordinates.
    - Projection fasteners attributes (ID, Projection Zone)
    - INFO attributes (only in long reports: Zone Path ID, Normal and projection point coordinates)
    - INFO1 attribute (Length, Volume)

One neutral file associated to the reference product is created. If the selected features belong to different products, there will be as many export files as there are different products.

Elements are displayed in the same order as they appear in the specification tree.

The BiW Fasteners specifications are exported, except for their links to the 3D geometry. For example, if the location method is **Point on Surface**, this specification will not show in the neutral file; only the coordinates of the point will be displayed.

- A warning message is issued in case errors occur during the export process.
  Please refer to the Import chapter for more information.
- One single log file is issued per export.
- Deactivated features are not exported; they appear in the log file.
- On a Windows workstation, you cannot export a file if it is already open.

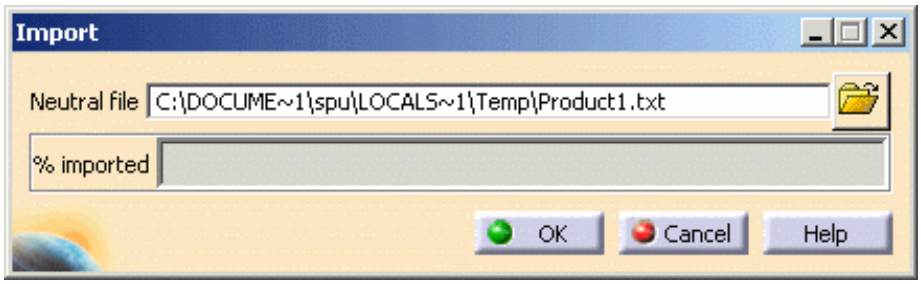
Importing

This task shows how to import data from a neutral file.

A neutral file must exist.

1. Click the **Automotive BiW data import** icon .

   The Import dialog box opens.

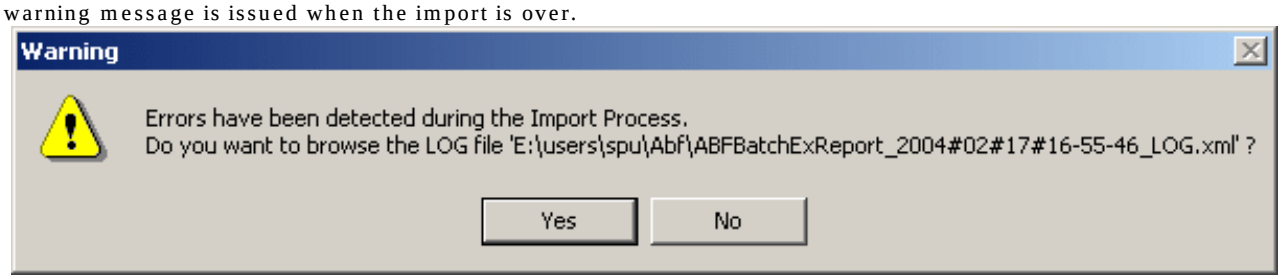


The directory for the neutral file is the one defined in Customizing General Settings and is the same as the export directory.

2. Click OK.

   The progress bar shows you the remaining imported percentage.

May the names of the joined part numbers or the root product differ from those used in the neutral file, the import process still goes on. If the joined part ID differs from the neutral file, the import is possible only if an associated part number is found in the session.
A warning message is issued when the import is over.



All inconsistencies found during the import process are logged in a .xml file. The path to this file is the same as the neutral file's. Its ID corresponds to: ProductName(Part Number)_date(year#month#day#time)_LOG.xml
One single log file is issued per import.

```
<?xml version="1.0" encoding="UTF-8" ?>
- <head>
  - <Import>
    - <Head>
      - <![CDATA[
          ================================================================================
          CATIA V5 BiW Automotive Import WARNING LOG
          ================================================================================
          VERSION: 1
          DATE: 18 November 2003
          IMPORT_FILE_PATH: E:\users\spu\Abf\Batch\AutomotiveBiWFastening2Parts.txt
          CURRENT_PRODUCT: AutomotiveBiWFastening2Parts
        ]]>
      </Head>
    - <Warnings>
      - <![CDATA[
          ### WARNING_NUMBER 1 ###
          Syntax Error found in Import file:
           E:\users\spu\Abf\Batch\AutomotiveBiWFastening2Parts.txt, line 89.
          The Part Number attribute <FirstPartA> in the neutral file has not been
          found in Product Session.

        ]]>
      </Warnings>
    </Import>
</head>
```

- All BiW Fasteners have an "Explicit" location method after they have been imported. They are located only by their coordinates but the links to the 3D geometry are not kept.
  See Exporting.
- Rules Check is not active when importing data.

Isolating Features

This task shows how to isolate joints (joints, joint bodies and joint elements) either individually or globally.

Open the Isolate1.CATProduct document.

Isolating Element by Element

1. Right-click a joint element for instance from the specification tree and select the **Joint Element.1 object -> Isolate** contextual command.

2. Double-click the joint element either from the specification tree or the geometry.

   The BiW SpotPoint Fastener Definition dialog box opens.

   As you can see, the joint element location has changed to explicit.

Isolating All Elements

1. Right-click the Automotive BiW Fastening Parameters from the specification tree and select the **Automotive BiW Fastening Parameters object -> Isolate Joints** contextual command.

2. Double-click a joint element of your choice either from the specification tree or the geometry.

   The BiW SpotPoint Fastener Definition dialog box opens.

   As you can see, the joint element location has changed to explicit.

- The isolated element has no more specific geometry.
- When applying this capability to a joint element, it is automatically set to the explicit mode.
- When applying this capability to a joint body, all its contained joint elements are automatically set to the explicit mode and its zones set as unspecified.
- When applying this capability to a joint, all its contained joint bodies are automatically set to the unspecified mode and its associated Specification Part is automatically destroyed.
- When applying this capability from the parameters, all joints belonging to the corresponding reference product are automatically isolated.

Transforming

Creating Mirror/Copy Elements
Creating Translate/Copy Elements

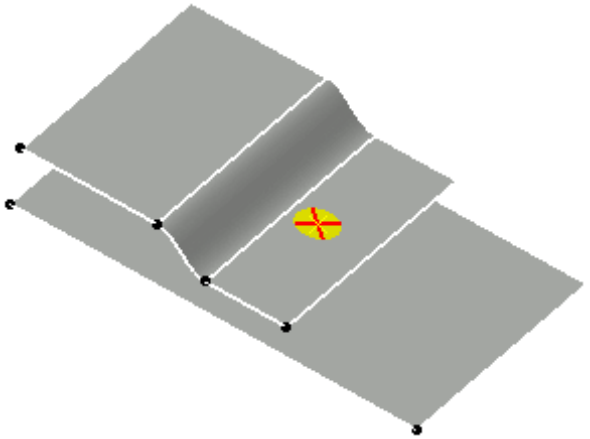
Creating Mirror/Copy Elements

This task shows how to create a Mirror/Copy element, either within the same product or two different products.

In this scenario an example with a Joint Element is given, but you can also select a Joint Body or an Assembly Joint.

Open the Joint2.CATProduct document.

The product is displayed as shown here with one joint element created.

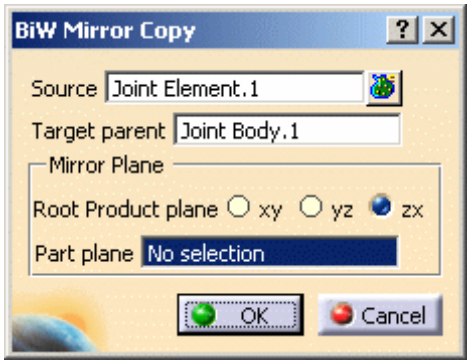


1. Select the joint element from the specification tree.
2. Select the **Mirror Copy** icon from **Transformed copies** sub-toolbar of the **Automotive BiW fasteners** toolbar.

By default, the **Target parent** object is the same as the one of the **Source** object, here Joint Body.1.

If the source object and the target parent belong to two different products, you need to explicitly select the target parent in a product different from the source's object.

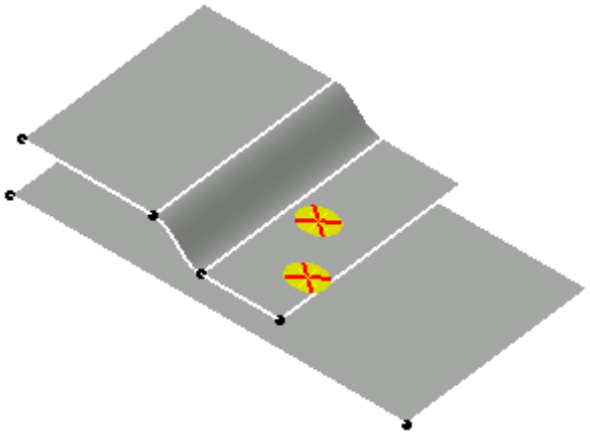
The BiW Mirror Copy dialog box opens.



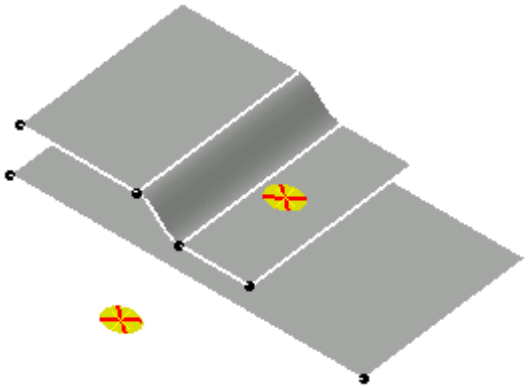
3. Select a Root Product Plane or a part zone on which you wish to paste the body.

4. Click OK.

   The mirrored SpotPoint is created.



In the above example, we selected xy as the Root Product Plane.

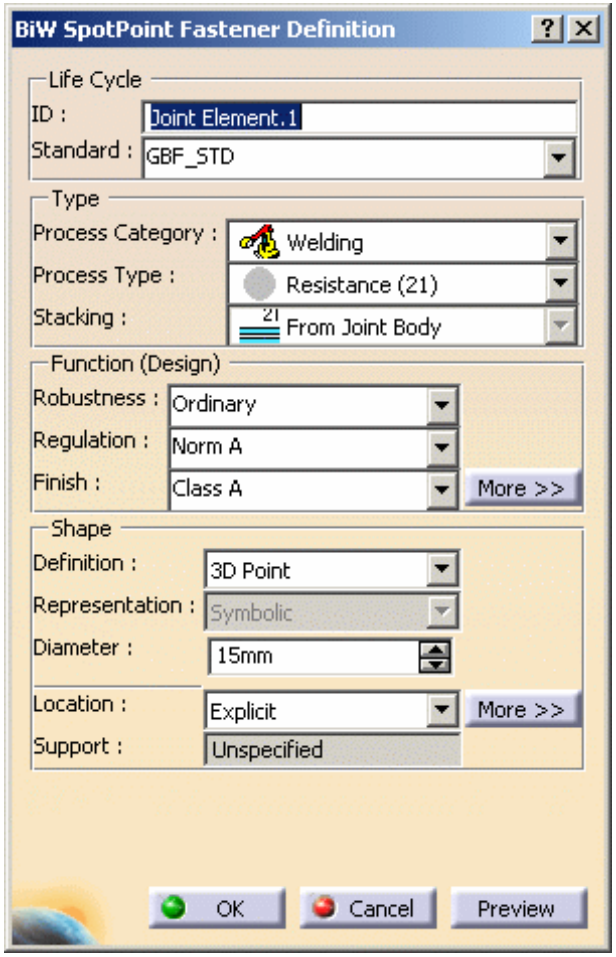


In the above example, we selected Blend.1 as the Part plane.

5. Double-click this new element from the specification tree.

The BiW SpotPoint Fastener Definition dialog box opens.

The SpotPoint is created as explicit.

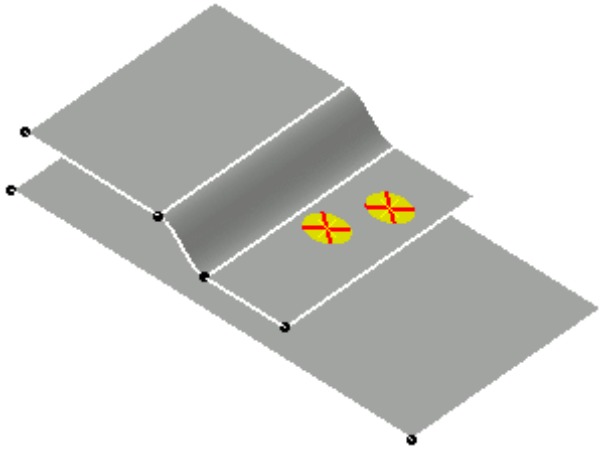


6. Select the **On Support Surface** location.

7. Select Blend.1.

8. Click OK in the dialog box.

The SpotPoint is relocated upon the selected surface.



- Mirrored objects follow the same rules as pasted objects, with an additional symmetry applied on explicit coordinates of Joint Elements.
- When the source element is a joint, the Parent target allows to multi-select parts in the target product to be reconnected.
- When the naming rule is not active, the copied elements have the same name as the source elements.

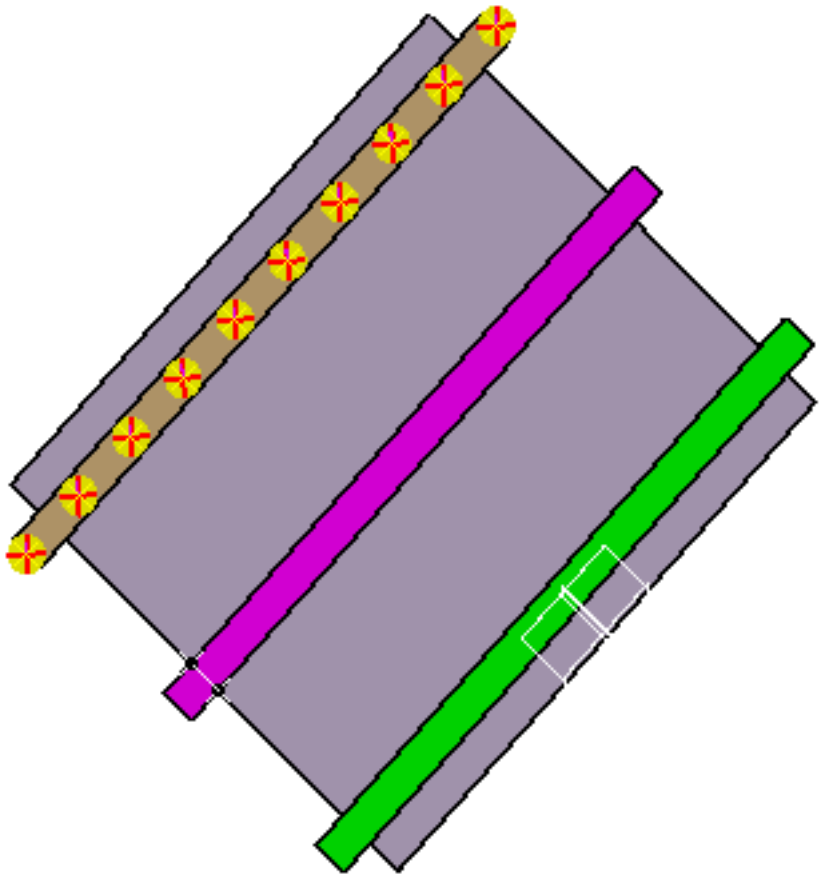
Creating Translate/Copy Elements

This task shows how to create a Translate/Copy element, either within the same product or onto different products.

In this scenario an example with Joint Elements is given, but you can also select a Joint Body or an Assembly Joint.

Open the Translate.CATProduct document.

The product is displayed as shown here with several joint elements created.

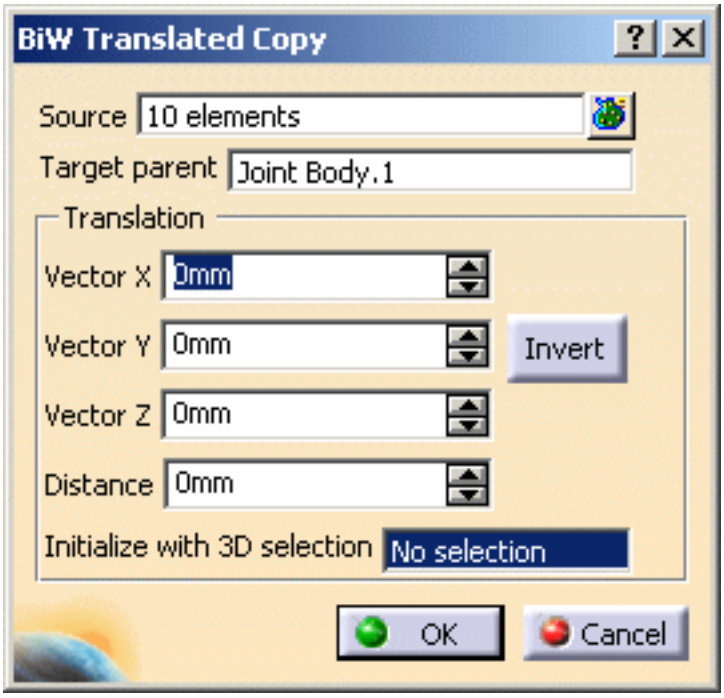


1. Select the joint element from the specification tree.
2. Select the **Translates BiW object Copies** icon from **Transformed copies** sub-toolbar of the **Automotive BiW fasteners** toolbar.

By default, the **Target parent** object is the same as the **Source** objects, here Joint Body.1.

If the source objects and the target parent belong to two different products, you need to explicitly select the target parent in a product different from the source's objects.

The BiW Translated Copy dialog box opens.



3. Select the Translation vectors.

   You can either:

   - manually select the X, Y, and Z vectors using the spinners, or
   - select a geometrical element to initialize with the selection and provide a direction.

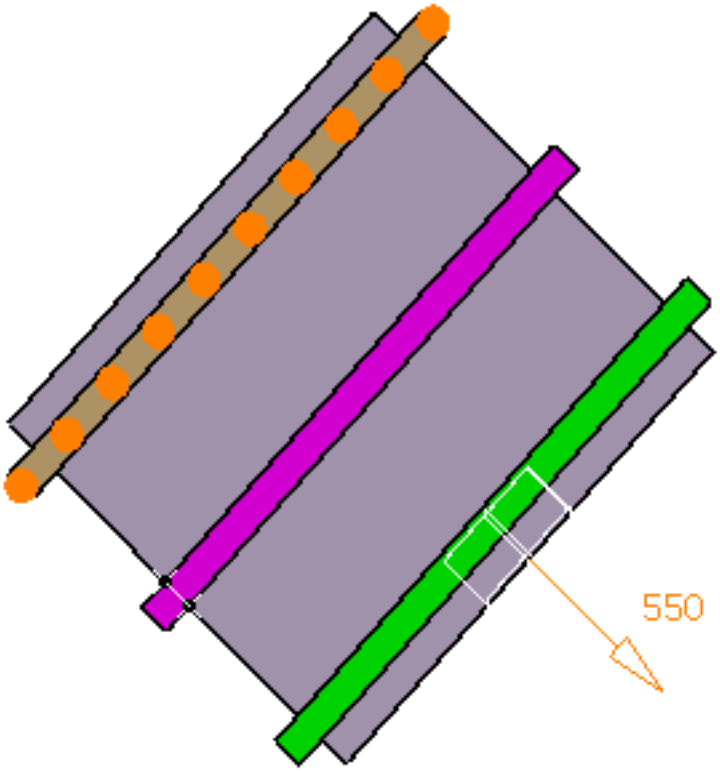
     This element can either be a surface or a plane (the normal to this surface or plane is computed to obtain the translation direction), or a line or an edge (the line or edge direction gives the translation direction)

4. Define the Distance to specify the length of the translation vector.

Whenever vector values are modified, the Distance value is automatically updated as the length of the vector, and vice-versa. Therefore the distance value is always equal to the corresponding vector value.

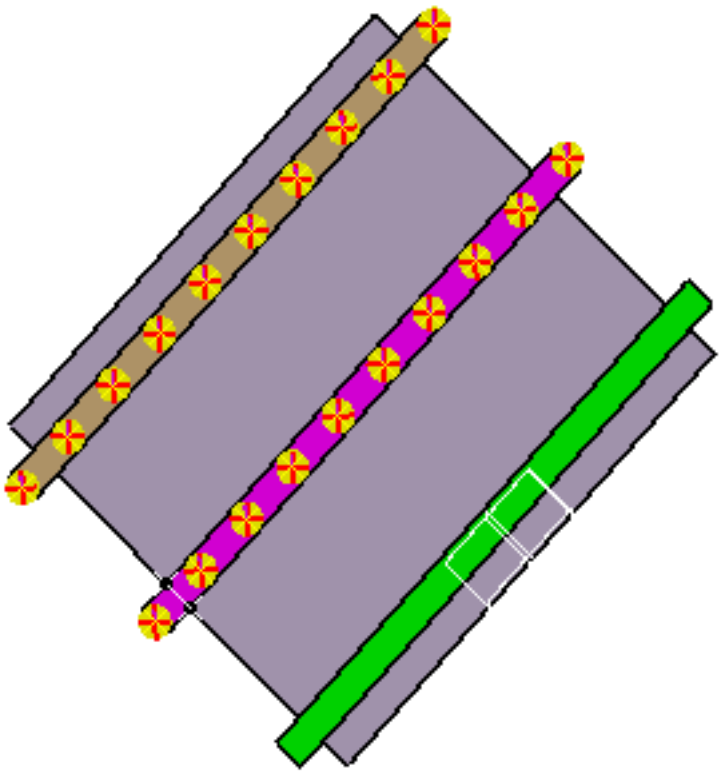
In our scenario we define a distance of 550mm.

5. Click the **Invert** button to reverse the direction vector.



6. Click OK.

The translated joint elements are created.



When the naming rule is not active, the copied elements have the same name as the source elements.

Copying/Pasting Elements

This task shows how to copy/paste elements such as Joint Elements, Joint Bodies, and Assembly Joints in different products.

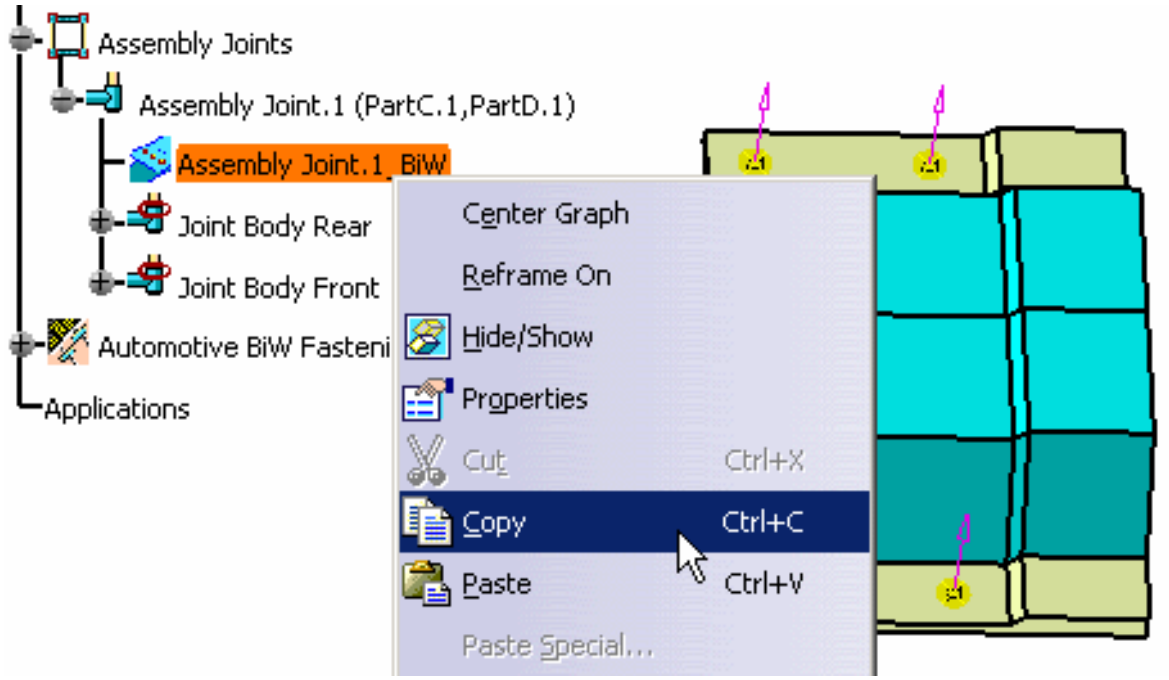
Open the ProductA.CATProduct and the ProductB.CATProduct documents.

1. Select **Tile Vertically** from the **Window** menu to display them both.

Copying/pasting Joints

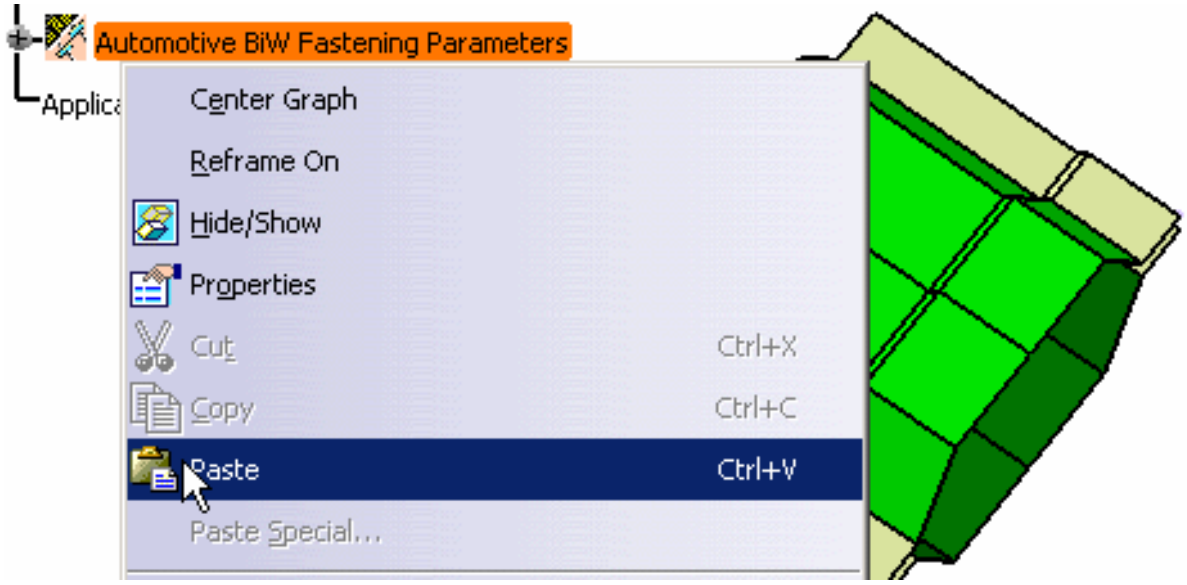
An ABF Parameters object must be created first on the target product. See Setting up BiW Fastening Parameters.

2. Right-click the source Joint you wish to copy.
3. Select **Copy** from the contextual menu.

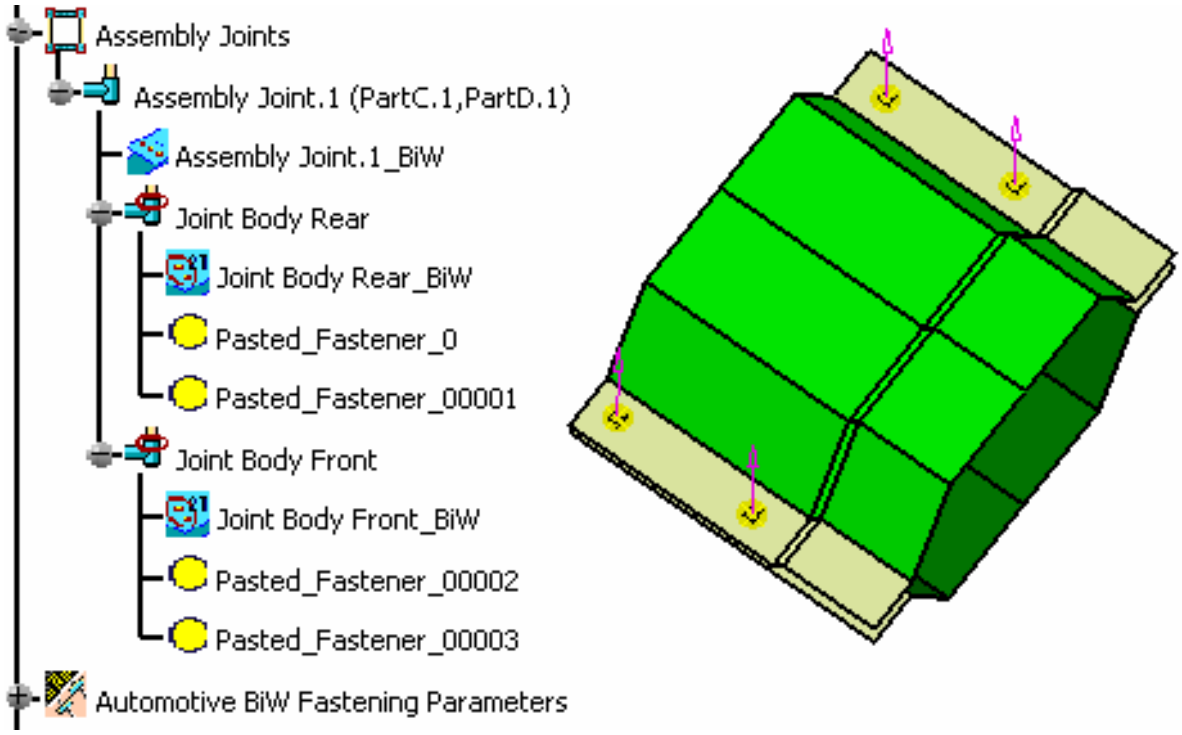


4. Right-click the target ABF Parameter object on which you wish to paste the Joint.

5. Select **Paste** from the contextual menu.



A new Joint is created based on the source joint and new icons are displayed in the specification tree.



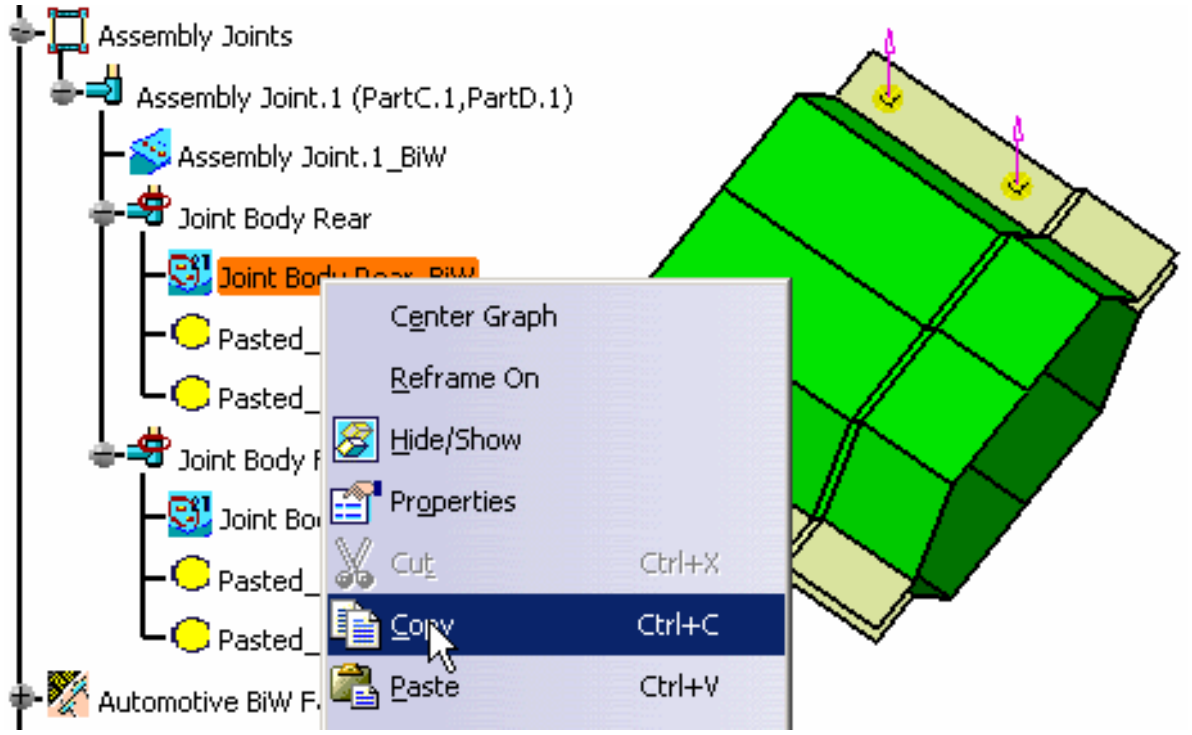
Note that:

- The Joint Bodies contained in the source Joint are also pasted onto the pasted Joint.
- The pasted Joint connects parts in the target product that have the same instance name as parts connected by the source Joint.
- If a part connected by the source Joint is not found in the target product assembly structure, then it is not referenced by the pasted Joint.

Copying/pasting Joints Bodies

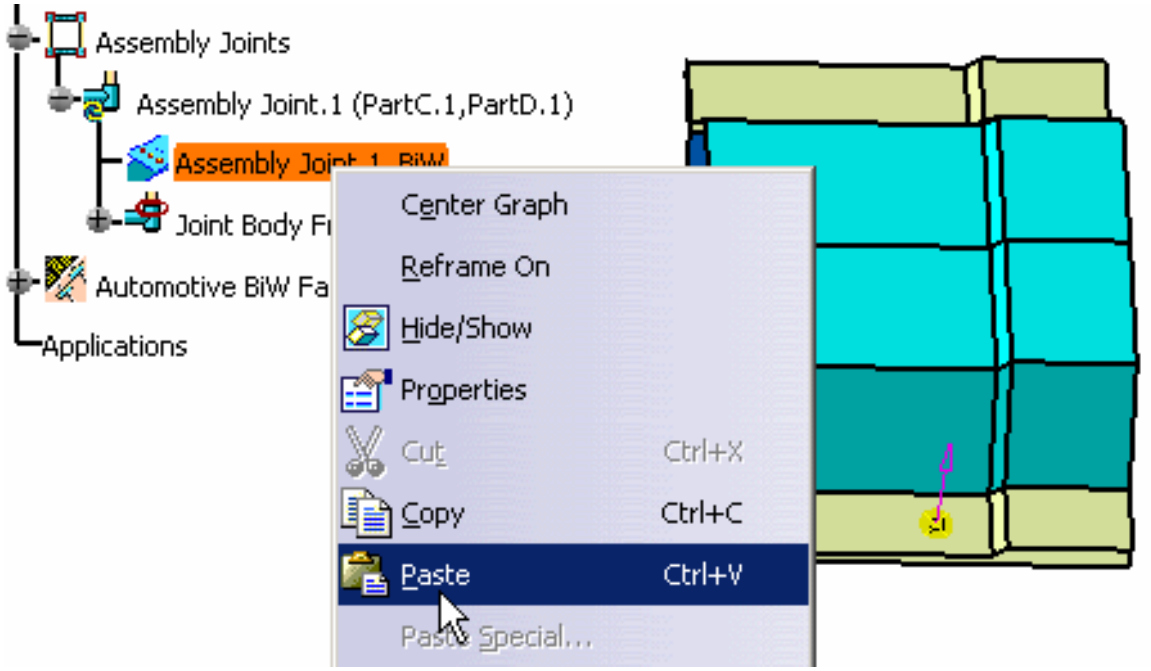
A Joint must be created first on the target product.

1. Right-click the source Joint Body you wish to copy.
2. Select **Copy** from the contextual menu.

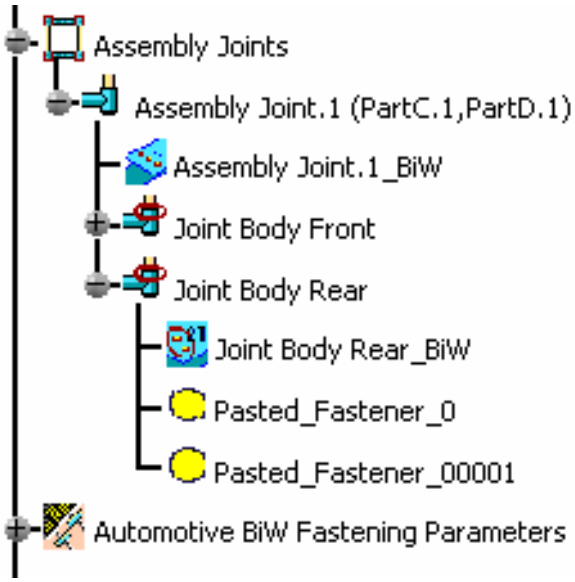


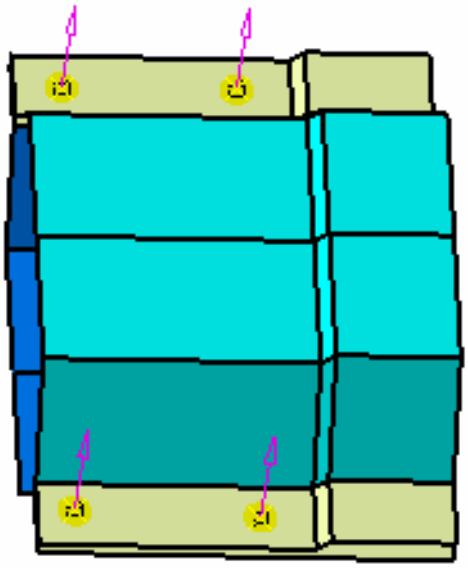
3. Right-click the target Joint on which you wish to paste the Joint Body.

4. Select **Paste** from the contextual menu.



A new Joint Body is created based on the source joint body and new icons are displayed in the specification tree.





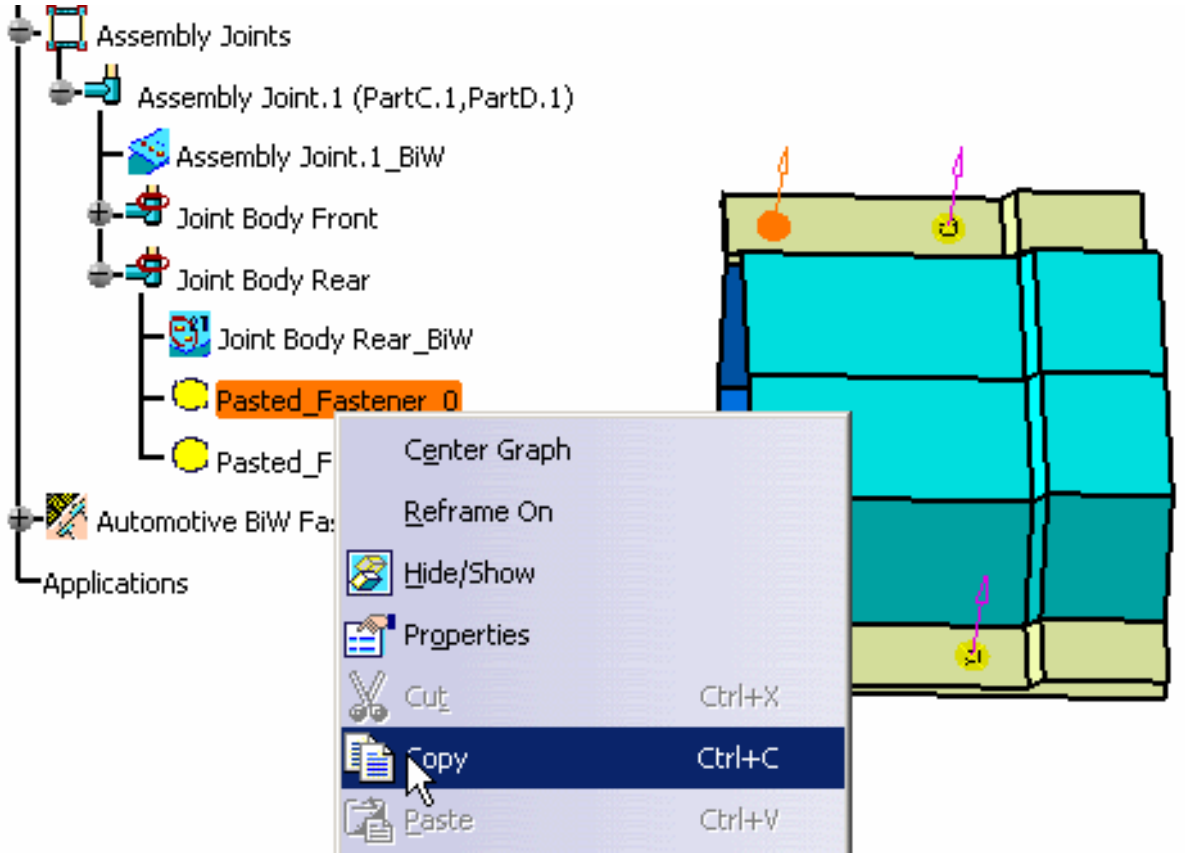
Note that:

- The Joint Elements contained in the source Joint Body are pasted onto the target Joint.
- The pasted Joint Body refers to the same zones as it is defined in the source Joint Body.
- If a published zone cannot be found on the target Joint, an unspecified zone is created instead.

Copying/Pasting Joints Elements

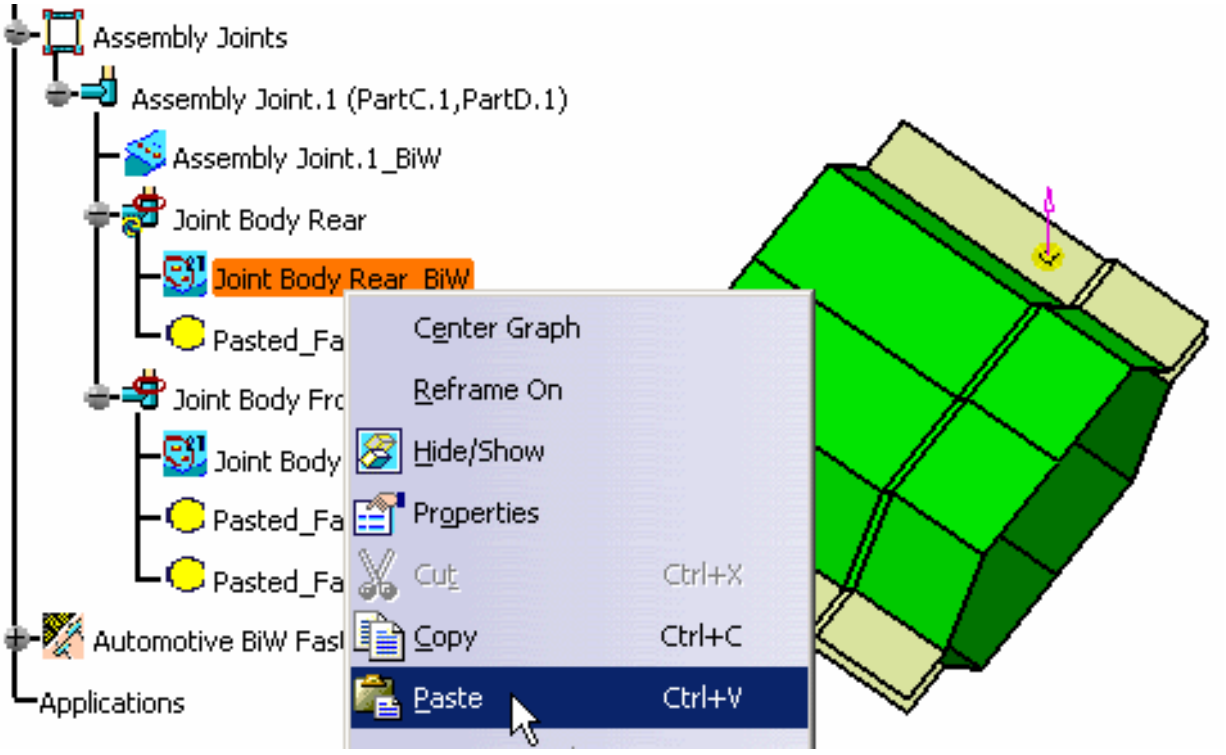
A Joint Body must be created first on the target product.

1. Right-click the source Joint Element you wish to copy.
2. Select **Copy** from the contextual menu.

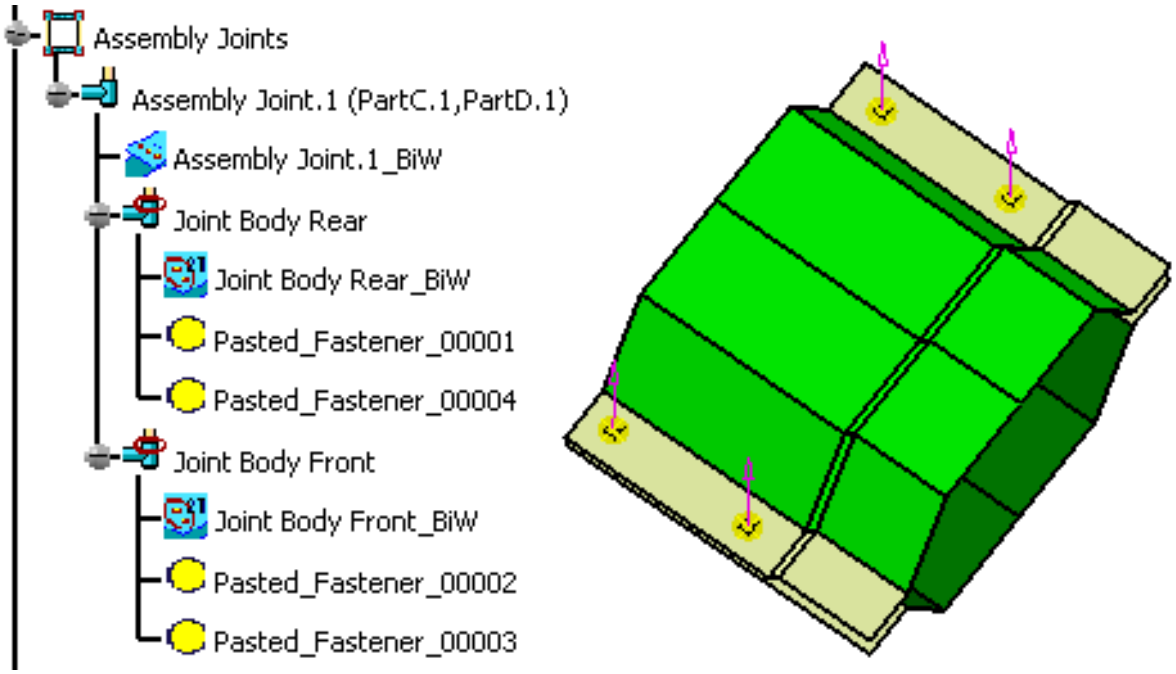


3. Right-click the target Joint Body on which you wish to paste the Joint Element.

4. Select **Paste** from the contextual menu.



A new Joint Element is created based on the source Joint Element and new icons are displayed within the specification tree.



Note that:

- The pasted Joint Element is explicit which means that its position is not specified by a geometry.
- The source Joint Element and the pasted one have the same position respectively to their roots products.
- The source Joint Element and the pasted one have the same normal vector, except if the pasted Joint Element is based on a surface which normal vector is different.
- The pasted Joint Element is named after the target's product Naming Rule if it is activated. If it is not activated, the pasted Joint Element has the same name as the source element.

Renaming Features

This task shows how to rename a selection of fasteners.

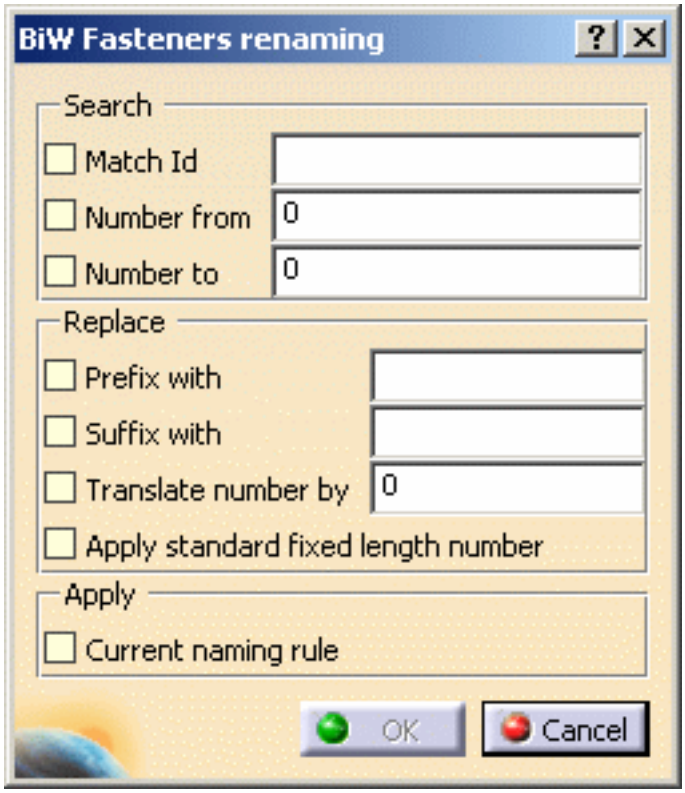
Open a .CATProduct document.

1. In the specification tree or in the 3D geometry, select the fastener(s) you want to rename.

   If you select a joint or a joint body, its children fasteners are highlighted in the specification tree.

2. Click the **Renames BiW Fasteners** icon .

   The BiW Fasteners renaming dialog box opens.



Search

- **Match ID**: enter the string that matches an ID in the specification tree. This field may be empty or unchecked if you do not want to apply any selection filter.

  You can use the * (star) special character to specify any sub-string.

  You can use the # (pound) special character to specify a numerical sub-string: it indicates the location of the fastener number among all selected fasteners that match the ID.

  Identification rules of the fastener number, prefix, and suffix:

  - Field entries are case sensitive.
  - If the fastener number cannot be found, the prefix equals the ID, the suffix is empty and the number is 0.
  - If the # special character is used, its location determines the fastener number.
  - If no # special character is used, the fastener number is the first numerical sub-string found in the ID.

    Here is an example with a joint called *Joint 32 Roof Top 1*.

Match ID	How is it interpreted?
<empty>	Prefix: "Joint" Number: 32 Suffix: "Roof Top 1"
*	Prefix: "Joint" Number: 32 Suffix: "Roof Top 1"
* 32 *	Prefix: "Joint" Number: 32 Suffix: "Roof Top 1"
* 32 * #	Prefix: "Joint 32 Roof Top" Number: 1 Suffix: /

  - Once the fastener number is computed, the prefix and the suffix are deduced as the surrounding sub-strings.
  - If the numerical suffix and the fastener number follow, they cannot be deduced.

    Here is an example with *Joint 325689*.

Match ID	Result expected	Result obtained	Feedback

Joint #89	Prefix: "Joint" Number: 3256 Suffix: 89	Prefix: "Joint" Number: 325689 Suffix: /	No fastener corresponds to the request
Joint #	Prefix: "Joint" Number: 3256 Suffix: 89	Prefix: "Joint" Number: 325689 Suffix: /	No fastener corresponds to the request

To obtain a result, you need to rename the fastener as Joint 3256 89:

Joint 3256 89, or **Joint # 89**, or **Joint # ***	Prefix: "Joint" Number: 3256 Suffix: 89	Prefix: "Joint" Number: 3256 Suffix: 89	Request OK

- In case there are several identical sub-strings in the fastener name, the system stops as soon as a criterion is verified. This may lead to unpredictable results.

- **Number from**: enter a number to filter out fasteners whose fastener number is lower or equal to the number defined here.

- **Number to**: enter a number to filter out fasteners whose fastener number is higher or equal to the number defined here.

Here are various examples illustrating the Match ID behavior:

Fastener Name to obtain	Replace	Match ID
Joint Element 1	Prefix: Joint Element Suffix: / Number: 1	**<empty>**, or ***Joint *#**
Joint Element 1 xxx	Prefix: Joint Element Suffix: xxx Number: 1	**<empty>**, or ***Joint *#***
61 10 21	Prefix: 61 Suffix: 21 Number: 10	**61 # 21**
61 10 21	Prefix: 61 10 Suffix:/ Number: 21	***10 #**
aa21aa31	Prefix: aa Suffix: aa31 Number: 21	**<empty>**, or ***aa#***

aa21aa31	Prefix: aa21aa Suffix: / Number: 31	**aa21aa#**
aa21aa31	Prefix: aa21aa Suffix: / Number: 31	**a*1aa#**
aa21aa31	Prefix: aa21aa Suffix: / Number: 21	***#***

Replace

- enter the **Prefix** that will replace the one initialized with the common prefix of all selected fasteners (only the fasteners that match the prefix to be found will be renamed).

- enter the **Suffix** that will replace the one initialized with the common suffix of all selected fasteners (only the fasteners that match the suffix to be found will be renamed).

- enter the **Translate number** value to offset the joint element number (for all selected fasteners, the joint element number is extracted by finding the rightmost numerical characters of the identifier (the result is zero if the identifier does not end with numerals), offsetting it by the entered value and replacing it within the identifier).

- Check the **Apply standard fixed length number** option to replace the number ID with the equivalent fixed-length number as defined in the Parameters chapter.

Apply

- Check the **Current naming rule** option to apply the naming rule as defined in the Parameters chapter.

  If any of the above Replace options were checked, they are automatically unchecked.

3. Click OK.

   Modifications are automatically taken into account: make sure the rule is configured to answer your need as set previously.

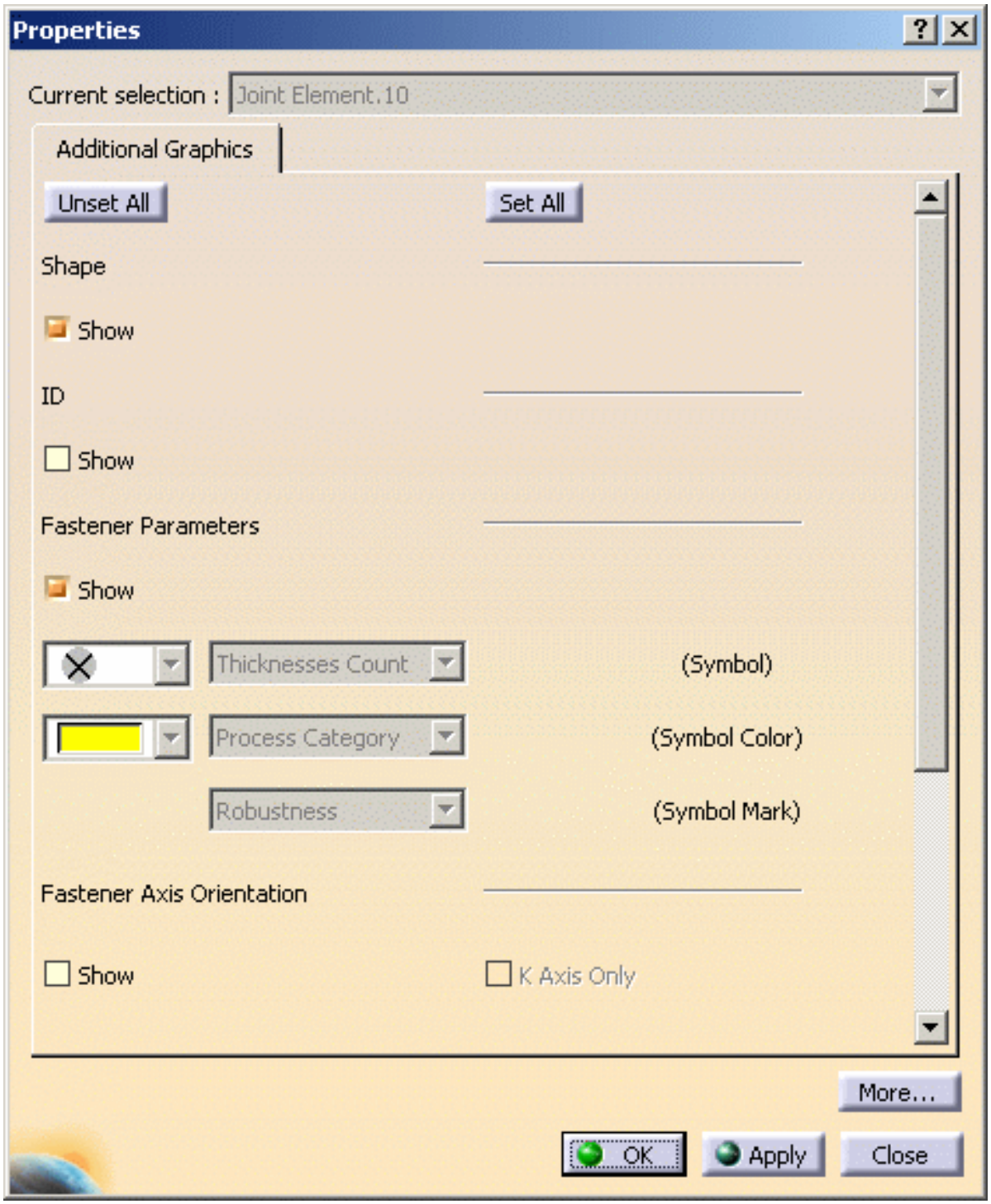
You can also change a fastener's name directly in the specification tree by double-clicking the fastener to edit it.

Editing Fasteners' Properties

This task shows you how to control the display of the elements you create in the geometry area.

1. Right-click a fastener in the specification tree or in the 3D geometry and select **Properties**.

   The Properties dialog box opens.



3. Click the **Unset All** button to uncheck all the options, except for the Shape Representation option. Similarly click the **Set All** button to check all options.

4. Click the **Show** buttons to visualize the different options.

The same options can be chosen using the **Tools -> Options** command.

If the graphic symbols (symbol and color) are already assigned to a Thickness Count or Process Type (for the symbol) or a Process Category (for the color) as defined in **Tools -> Options**, they cannot be modified. In that case, spinners are grayed.

If the graphic symbols are assigned to the Unspecified type, you are able to modify them using the combo box.

Note that, in visualization mode:

- when selecting at least one of the options contained either in the **Support Contact zone Material Orientation** or in the **Contact Zones** within the **Properties** dialog box,
- and when selecting **Tools -> Options -> Mechanical Design -> Assembly Design -> General**, the **Automatic Switch to Design Mode** option,

then all joint components are automatically switch to Design mode.

The **Automatic Switch to Design Mode** option must be activated to visualize the desired options.
If you do not wish to have all joint components automatically switch to Design mode, then uncheck the options mentioned above.

Refer to the Display chapter to have further information about the different options.

Advanced Tasks

Running the Fastening Rules Analysis
Searching Automotive BiW Features
Integration With Drafting
Measure Tools

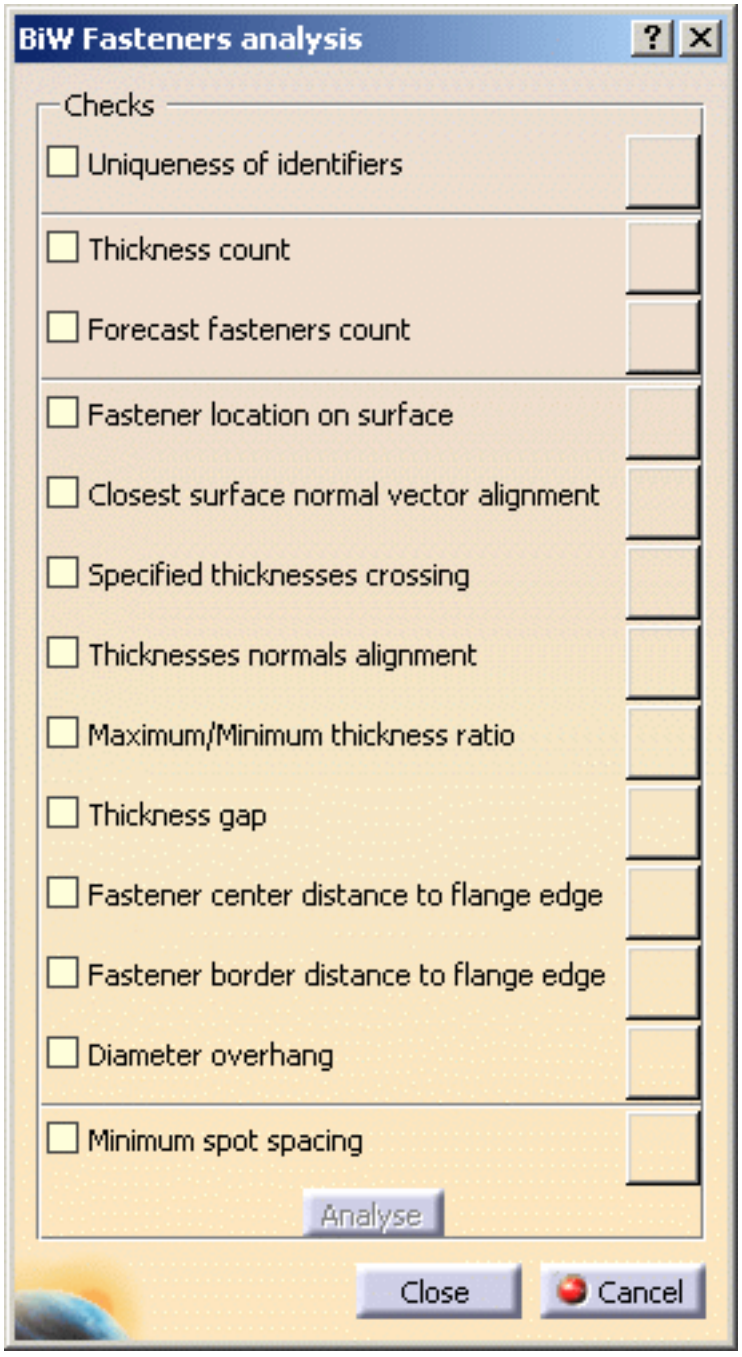
Running the Fastening Rules Analysis

This task shows you how to launch an analysis to check the rules' compliance as defined in the BiW Fastening Parameters.

Open the Analysis1.CATProduct document.

1. Click the **Check** icon .

   The BiW Fasteners analysis dialog box opens.



2. Check the rules on which you want to run the analysis.

- **Uniqueness of identifiers**: checks that all joint elements have different names.
- **Thickness count**: checks that the number of thicknesses is compliant with the number set in the BiW parameters.
- **Forecast fasteners count**: checks that the number of joint elements the joint body is supposed to contain for further verifications is compliant with the number set when creating the joint and the jointbody.
- **Fastener location on surface**: checks that joint elements are located on surfaces.

This rule applies for spotpoints only.

- **Closest surface normal vector alignment**: checks the deviation angle between the joint element's normal vector of the joint element's normal vector at the projection on the closest surface.
- **Specified thicknesses crossing**: checks that joint elements cross all the specified thicknesses of the jointbody.

This rule applies for spotpoints only.

- **Thicknesses normals alignment**: checks that the thickness normal to the crossed joined components is aligned with the joint element's normal.
- **Maximum/Minimum thickness ratio**: checks that the ratio between the minimum and the maximum thicknesses of the crossed zones is lower or equal to the maximum defined ratio, even if all joint elements do not intersect all the specified joined components.

- In case the maximum ratio is set to 1, the rule checks that all crossed thicknesses are equal.
- If one or more thicknesses are null, the ratio is infinite.
- For curvebeads, specified crossed thicknesses are computed at the start location.

- **Thickness gap**: checks that the gap value between two joined components crossed by a given joint element is compliant with the value set in the BiW parameters.
- **Fastener center distance to flange edge**: checks that all joint elements on crossed surfaces intersect all specified components and computes the minimum distance from the fastener's center location to the closest flange edge.
- **Fastener border distance to flange edge**: checks that all joint elements on crossed surfaces intersect all specified components and computes the minimum distance from the fastener's border location to the closest flange edge.
- **Diameter overhang**: checks the maximum overhang of the fastener's diameter to the closest flange edge.

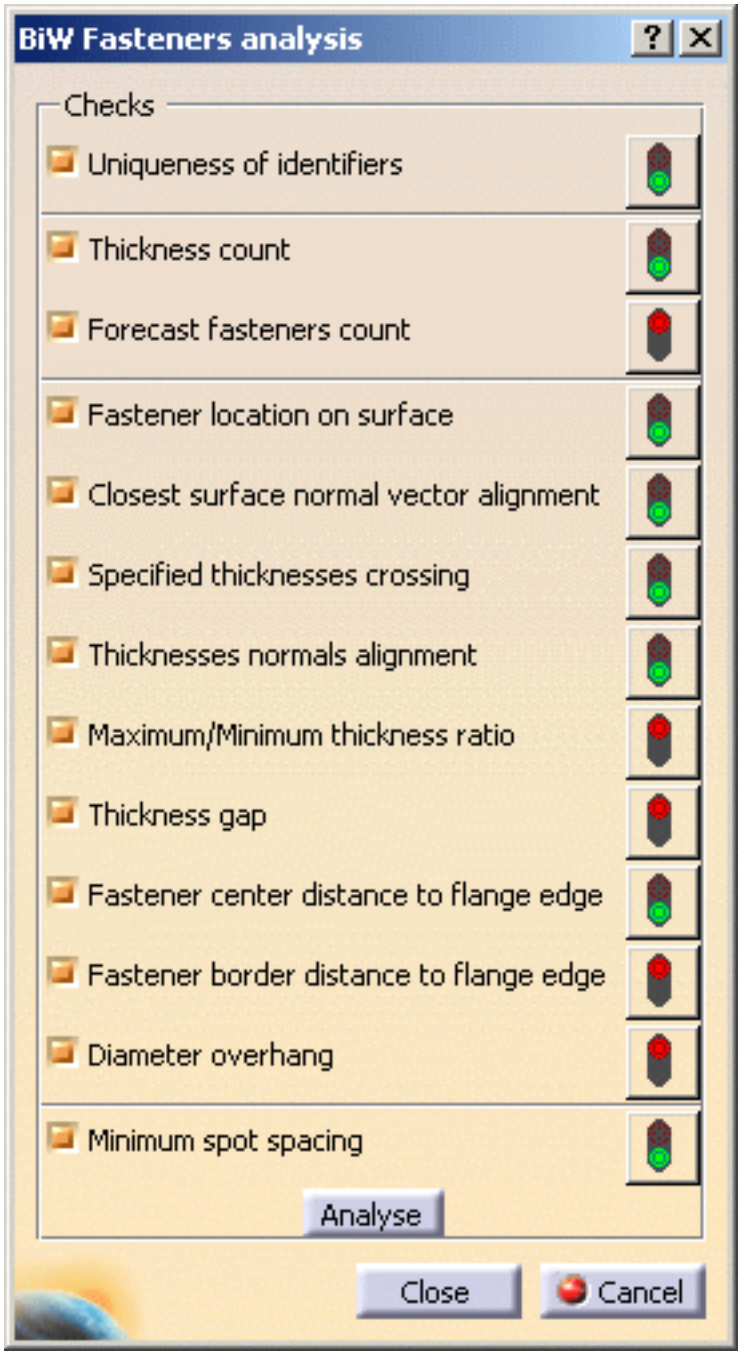
- In case two or more surfaces with common edges are crossed, you need to perform a join so their free edges are considered as those of one single surface.
- Internal edges and tangent surfaces are ignored, but holes are not.
- For rivets, the diameter taken into account is the shank diameter.
- The three above rules apply for spotpoints and spotprojection welds only, not curvebeads.

- **Minimum spot spacing**: checks that the minimum spot spacing between two spotpoints is compliant with the value set in the BiW parameters.

This rule applies for spotpoints only.

3. Click the **Analyse** button.

   Besides each rule, a sign gives a feedback whether the check passed or failed.



4. Click the red sign corresponding to the rule that failed. The corresponding check failure analysis window opens, along with the list of selected failing elements. You can select several elements in the list to highlight them in the specification tree and 3D geometry.

- The Closest surface normal vector alignment check failure report displays either "N/A" when no normal vector was found on the joint element (which is the case when you open a pre-V5R12 model) or the deviation angle between the normal vector of the joint element and the normal vector at the projection on the closest surface. You can then select the Joint Elements you want to "heal" and either with the "Apply remedial action" contextual menu or push button, reset the normal vector according to the normal vector at the projection on the closest surface.
- The Fastener location on surface check failure report displays either "N/A" if the joint element cannot be projected on any surface or the distance to the closest surface. Same as before, applying the Remedial action on a joint element will project it on the closest surface.
- The Fastener center distance to flange edge, Fastener border distance to flange edge, or Diameter overhang ratio check failure report displays the distance of the failed joint elements and highlights the closest edge involving the failure in the 3D geometry.

- The BiW Fasteners Analysis command runs on the current selection. If the latter is empty, the whole product structure is taken into account.
- When you exit the command, elements in error are still highlighted.
- You can right-click the highlighted element(s) to **Reframe on** and **Center Graph**.

Searching Automotive BiW Features

This task shows how to use the Search capabilities on Automotive BiW Features, in order to detect any specific kind of feature.

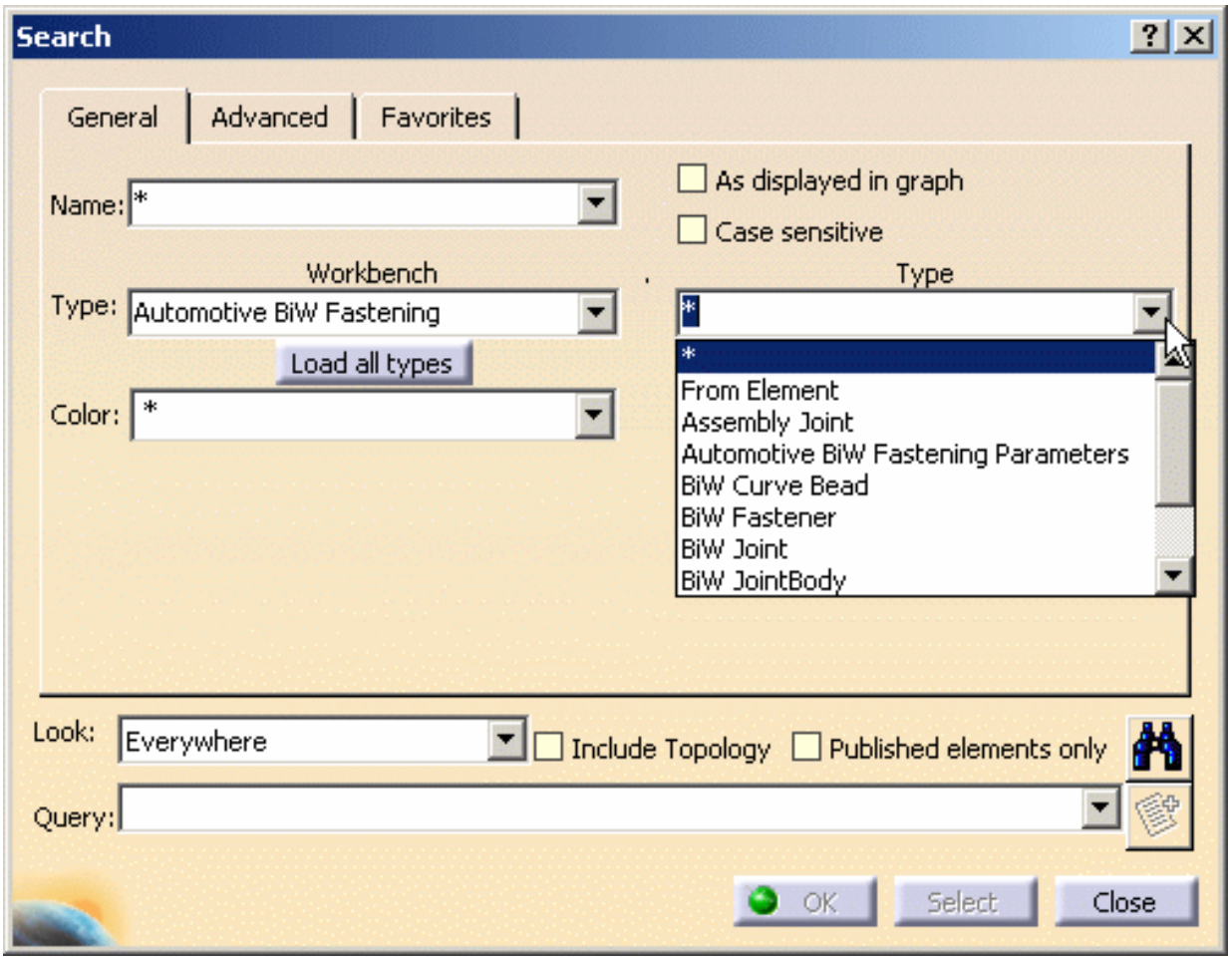
Open the PowerCopyStart.CATProduct document.

1. Select the **Edit** -> **Search** menu item.

The Search dialog box opens.

2. From the Type Workbench list choose Automotive BiW Fastening.

You can then display the list of Automotive BiW Fastening from the **Type** list:

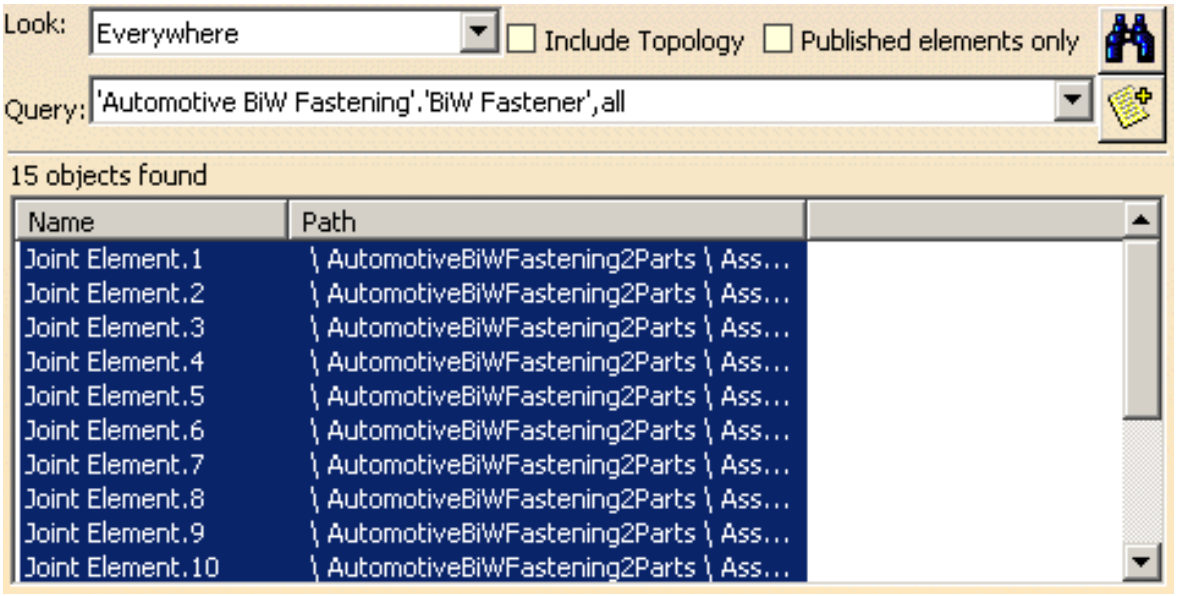


3. Select the type of feature you wish to find within the open .CATPart document.

   Here we chose BiW Fastener.

4. Click the **Search** icon .

The list of all elements of the selected type is displayed in the objects found field:



You can select an element from the list, it will be highlighted in the geometry area.

To find out more on the search capabilities, refer to Selecting Using the Search... Command (General Mode) and Selecting Using the Search... Command (Favorites and Advanced Modes) from the *Infrastructure User's Guide*.

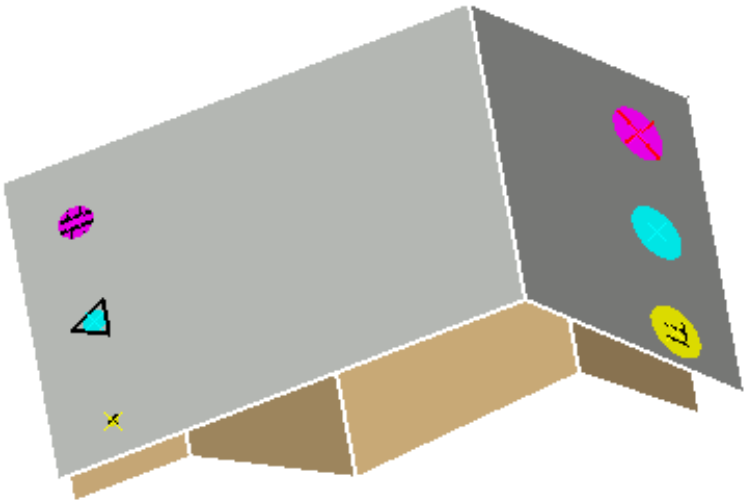
Integration With Drafting

This task shows you how to generate a .CATDrawing document from a .CATProduct document containing BiW spot fasteners.

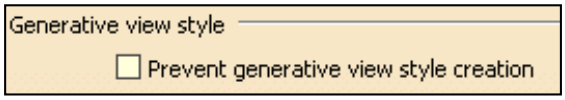
Open the DraftingIntegration1.CATProduct document.

1. Make sure the Normal Vector Orientation is defined.

   Refer to the BiW Fastening Application Display User Settings chapter for more information.

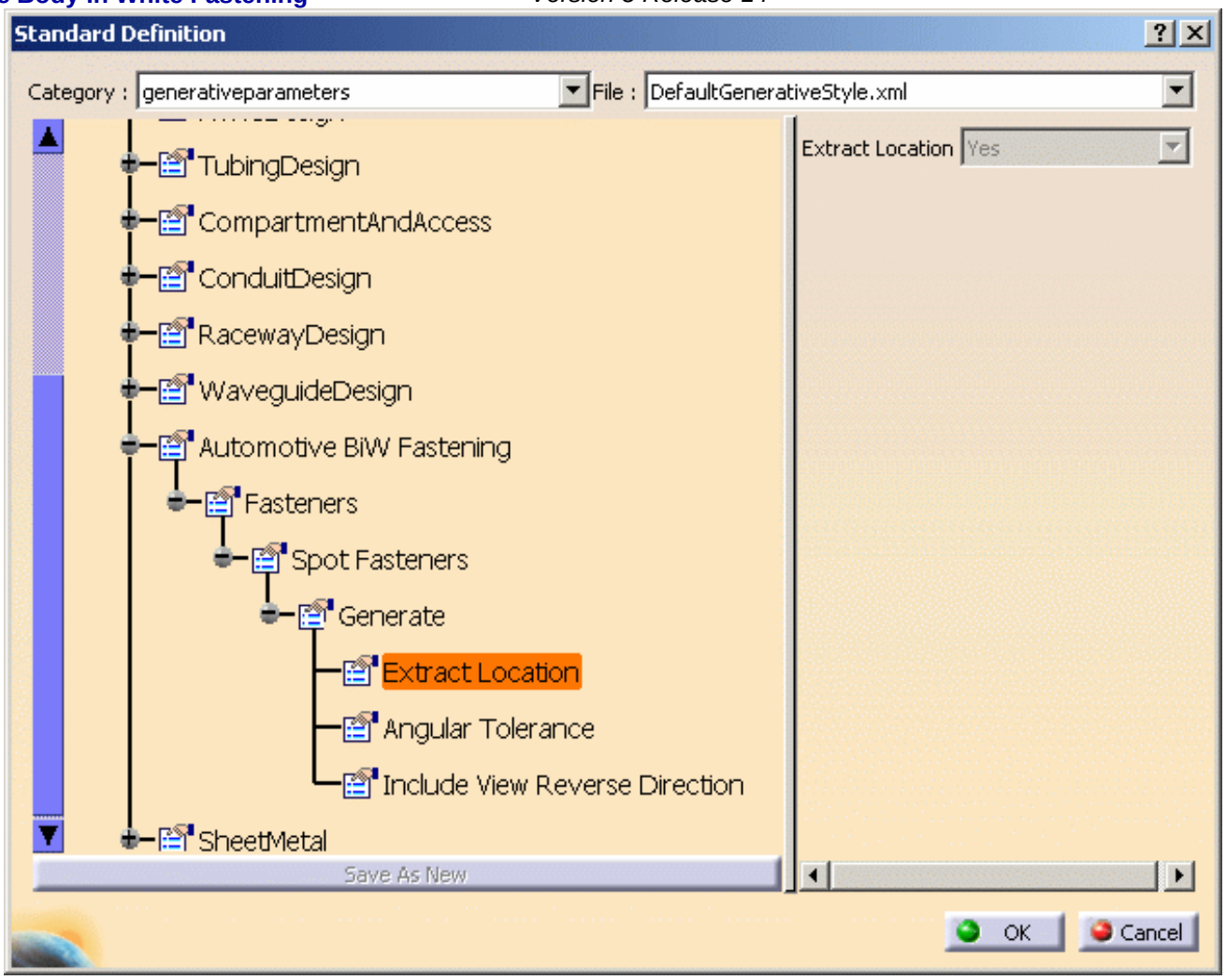


Make sure the **Prevent generative view style creation** option is deselected from **Tools -> Options -> Mechanical Design -> Drafting -> Administration** tab in order to be able to create and use generative view styles.



Generative view styles can be customized via Tools -> Standards.

2. Select **Tools -> Standards...** then the **generativeparameters** Category and the Automotive BiW Fastening Generative Style.



3D spots will be automatically generated on the drawing view depending on the ABF Drawing extraction parameters added to the CATIA Standard Default Generative View Style (GVS).

The administrator must set the generative parameter "Automotive BiW Fastening" in the DefaultGenerativeStyle.xml so that the user obtains suitable drawings using generative view styles.
Please refer to the Creating a View using Generative View Styles chapter in the *Generative Drafting* documentation.

This DefaultGenerativeStyle.xml file is located according to the resources/standard/generativeparameters path from CATIA run time view root directory (intel_a for Windows, aix_a for IBM UNIX, irix_a for SGI UNIX, solaris_a for SUN Unix, hpux_a for HP UNIX) or can be accessed via **Tools -> Standards**.

- The **Extract Location** option indicates whether the spot fastener location must be extracted or not (the default value is Yes)
- The **Angular Tolerance** option defines the maximum angle between the spot normal vector orientation and the direction of the view used for the spot extraction (the default value is 90).
- The **Include View Reverse Direction** options lets you choose whether you wish to visualize the 2D reverse spots (the default value is No).

Case A: Drafting View without associated GVS

The 3D Spot Fasteners will be generated in this view according to the Extract parameters in **Tools -> Options -> Shape -> Automotive BiW Fastening -> General tab** (Angular Tolerance, Include View Reverse Direction)

Case B: Drafting View with an associated GVS

a. Providing that the Angular Tolerance is specified (different from -1)
The 3D Spot Fasteners will be generated in this View according to the Extract parameters in GVS (Angular Tolerance, Include View Reverse Direction)
b. Providing that the Angular Tolerance is not specified (set to -1)
The 3D Spot Fasteners will be generated in this View according to the Extract parameters in **Tools -> Options -> Shape -> Automotive BiW Fastening -> General** Tab (Angular Tolerance, Include View Reverse Direction)

Example 1:
Providing that:

- the Angular Tolerance is different from -1 in the GVS parameters (45 for example),
- the Include View Reverse Direction option is checked in the GVS parameters,
- the Angular Tolerance is set to 60 in **Tools -> Options** (value not effective here),
- the Include View Reverse Direction option is unchecked in **Tools -> Options** (value not effective here),

=> the 2D reverse spots will be generated in the Drawing document according to the GVS parameters (with Angular Tolerance value = 45).

Example 2:

- Providing that:
the Angular Tolerance is set to -1 in the GVS parameters (value not effective here),
- the Include View Reverse Direction option is set to No in the GVS parameters (value not effective here),
- the Angular Tolerance is 45 in **Tools -> Options**,
- the Include View Reverse Direction option is checked in Tools -> Options,

=> the 2D reverse spots will be generated in the Drawing document according to Tools -> Options parameters (with Angular Tolerance value = 45).

If the Angular Tolerance is set to 0 (through GVS or **Tools -> Options -> Shape -> Automotive BiW Fastening -> General** Tab), **no** 3D Spot Fastener will be generated on the Drawing View.

3. Select **Top View**  from the Quick View toolbar.
4. Select **File -> New** from the menu bar.

   The New dialog box displays.
5. Select **Drawing** from the List of Types and click OK.

   The New Drawing dialog box displays.
6. Click OK.

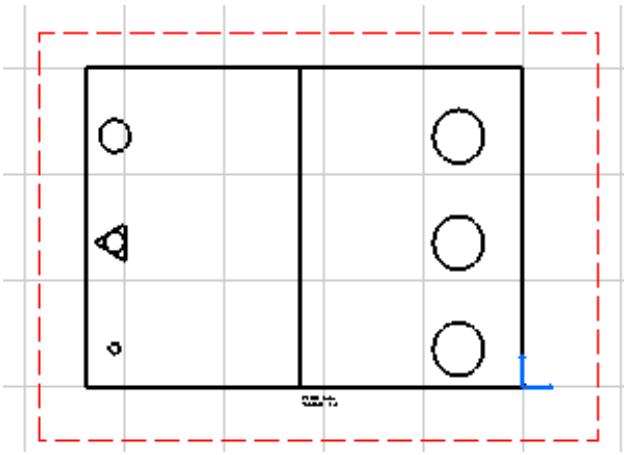
   The Generative Drafting workbench displays with a default grid.
7. Select **Window -> Tile Horizontally** from the menu bar to organize your windows horizontally.
8. In the drawing document, select **Front View**  from the View toolbar (Quick View sub-toolbar).

   The Generative View Style toolbar automatically displays with the DefaultGenerativeStyle selected by default.

9. Select the surface in the 3D document.

   3D symbol spots are visualized in the 2D drawing document and their diameters are respected.

10. Click on the drawing sheet or at the center of the blue knob to generate the view.



3D symbol spots are visualized in the 2D drawing document view and their diameters are respected (2D Spot Symbol Diameter = 3D Spot Diameter). Symbols are gathered together in the ABF_Symbols.CATDrawing reference file located according to the reffiles/GbfStandards path from CATIA run time view root directory (intel_a for Windows, aix_a for IBM UNIX, irix_a for SGI UNIX, solaris_a for SUN Unix, hpux_a for HP UNIX).

You do not need to save this file in the database because the file that is used is always the run time view's, should it be during the update process or during the drawing extraction.

You can set a fixed size for your 2D drawing spots.
Please refer to the General Settings chapter in the Customizing section.

Here are these symbols:

2D symbol	3D symbol	code
		111
		112
		113
		114
		151
		152
		153
		154
		136
		137
		138
		139
		101
		125
	any other 3D symbol	

Measure Tools

You can measure the distance between fasteners using the following command:

Measuring distances and angles: Click the Measure Between icon, set the measure type and mode in the Measure Between dialog box, then select two entities.

You can measure properties associated to a selected fastener using the following command:

Measuring properties: Click the Measure Item icon, then select an item.

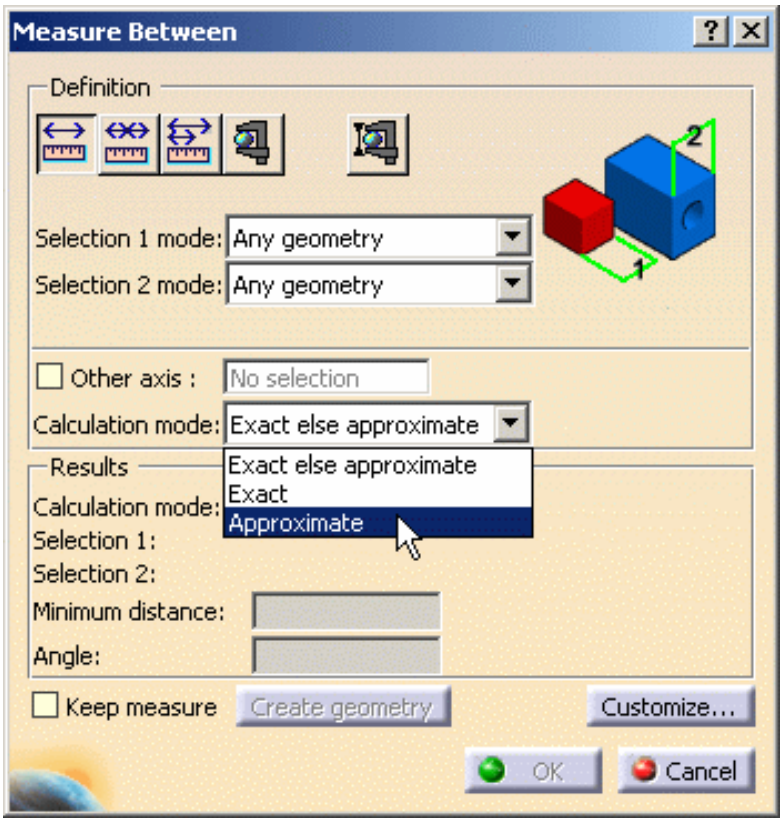
You must set the following parameters for both commands:

- Check the **Show Shape** option in **Tools -> Options -> Shape -> Automotive BiW Fastening -> Display**. Only this option must be checked.
- Choose **Any geometry** as the Selection mode, and **Approximate** as the Calculation mode, as the fastener representation is a symbolic shape representation (the Exact mode is not supported).
- Fasteners must be selected in the 3D geometry. Selecting fasteners in the specification tree is not supported.

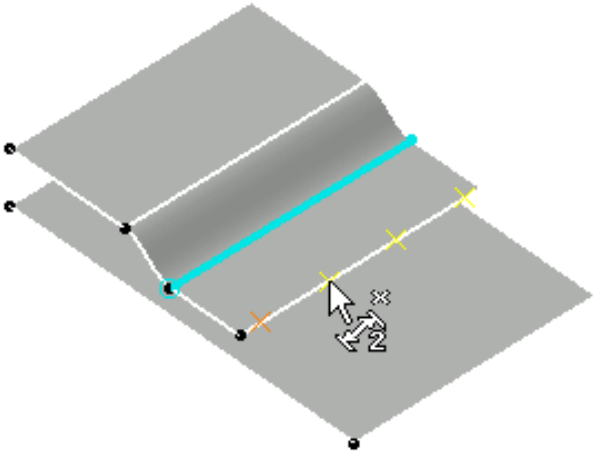
Open the Measure1.CATProduct document.

1. Select the **Measure Between** icon.

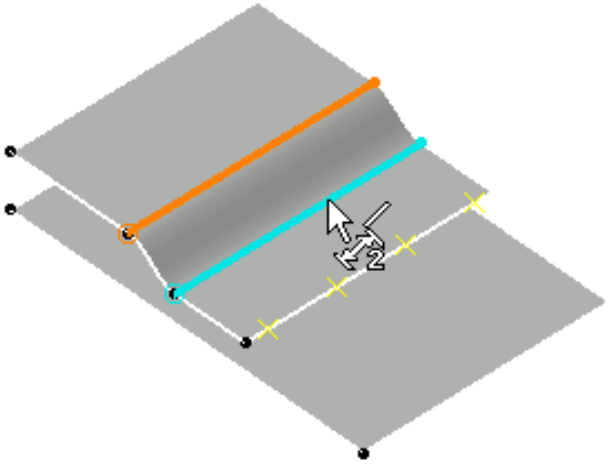
   The Measure Between dialog box appears.



2. Select two fasteners in the 3D geometry.

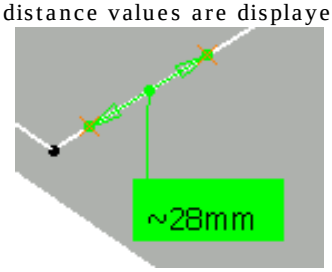


Measuring between two spotpoints

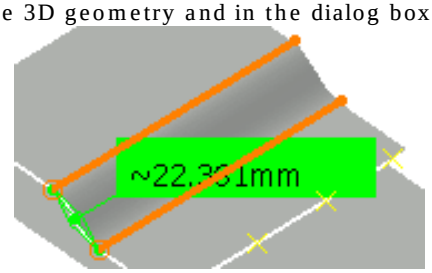


Measuring between two curvebeads

Appropriate distance values are displayed both in the 3D geometry and in the dialog box.



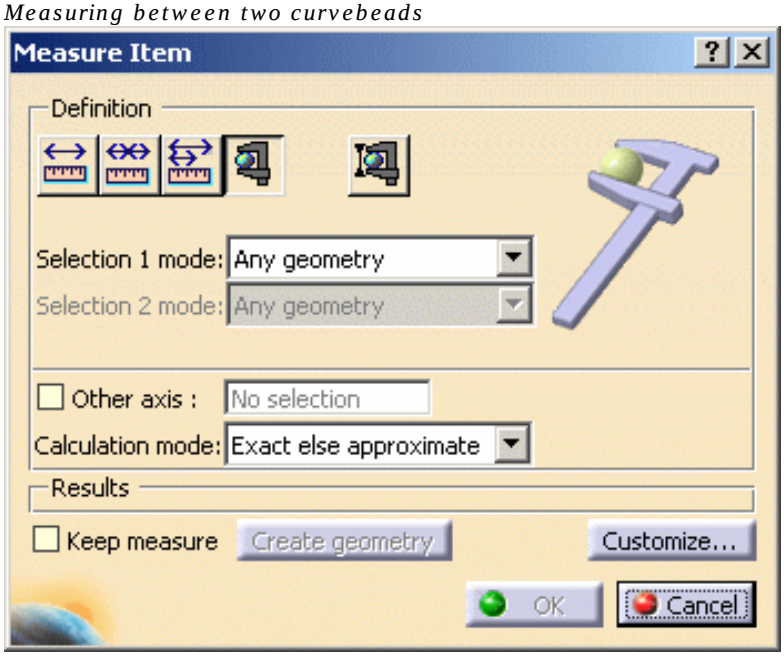
Measuring between two spotpoints



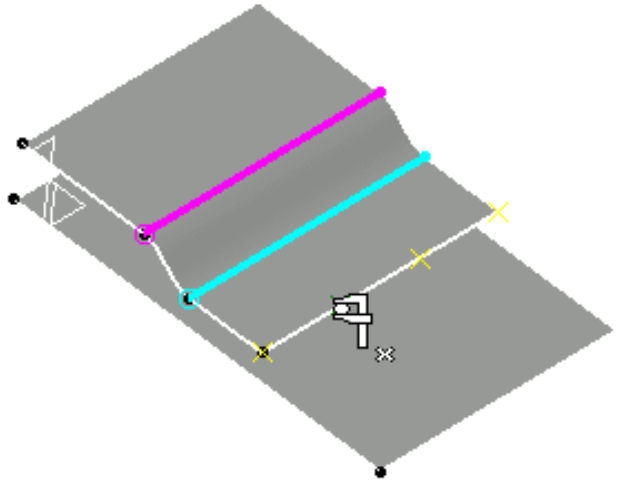
Measuring between two curvebeads

3. From the Measure Between dialog box, click the **Measure Item** icon  (you can also select this icon from the Measure toolbar).

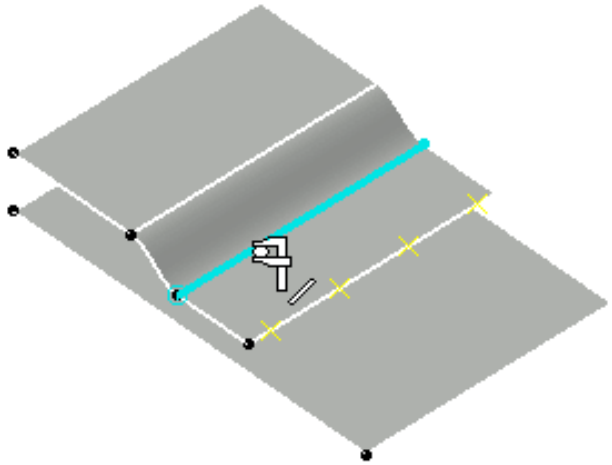
   The Measure Item dialog box appears.



4. Select the fastener in the 3D geometry.

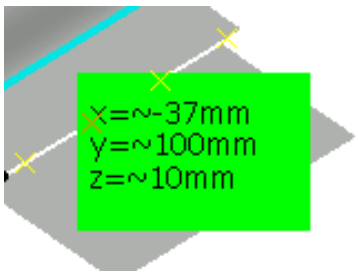


Measuring a spotpoint

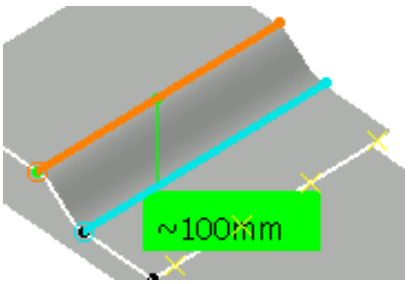


Measuring a curvebead

Appropriate values are displayed both in the 3D geometry and in the dialog box: coordinates for spotpoints and length for curvebeads.



Measuring a spotpoint



Measuring a curvebead

You can customize the properties you want to see displayed by clicking the **Customize...** option.

5. Click OK to exit the command.

Automotive Body In White Fastening Interoperability

Optimal CATIA PLM Usability for Automotive BiW Fastening

Optimal CATIA PLM Usability for Automotive BiW Fastening

When working with ENOVIA V5, the safe save mode ensures that you only create data in CATIA that can be correctly saved in ENOVIA. Therefore, in interoperability mode, some CATIA V5 commands are grayed out / hidden in the Automotive Body In White Fastening workbench.

ENOVIA V5 offers two different storage modes: Workpackage (Document kept - Publications Exposed) and Explode (Document not kept).

To ensure seamless integration, you must have both a CATIA and ENOVIA session running.

Please find below the list of the Automotive Body In White Fastening commands along with their accessibility status in Enovia V5. In some cases, restriction rules apply to certain commands.

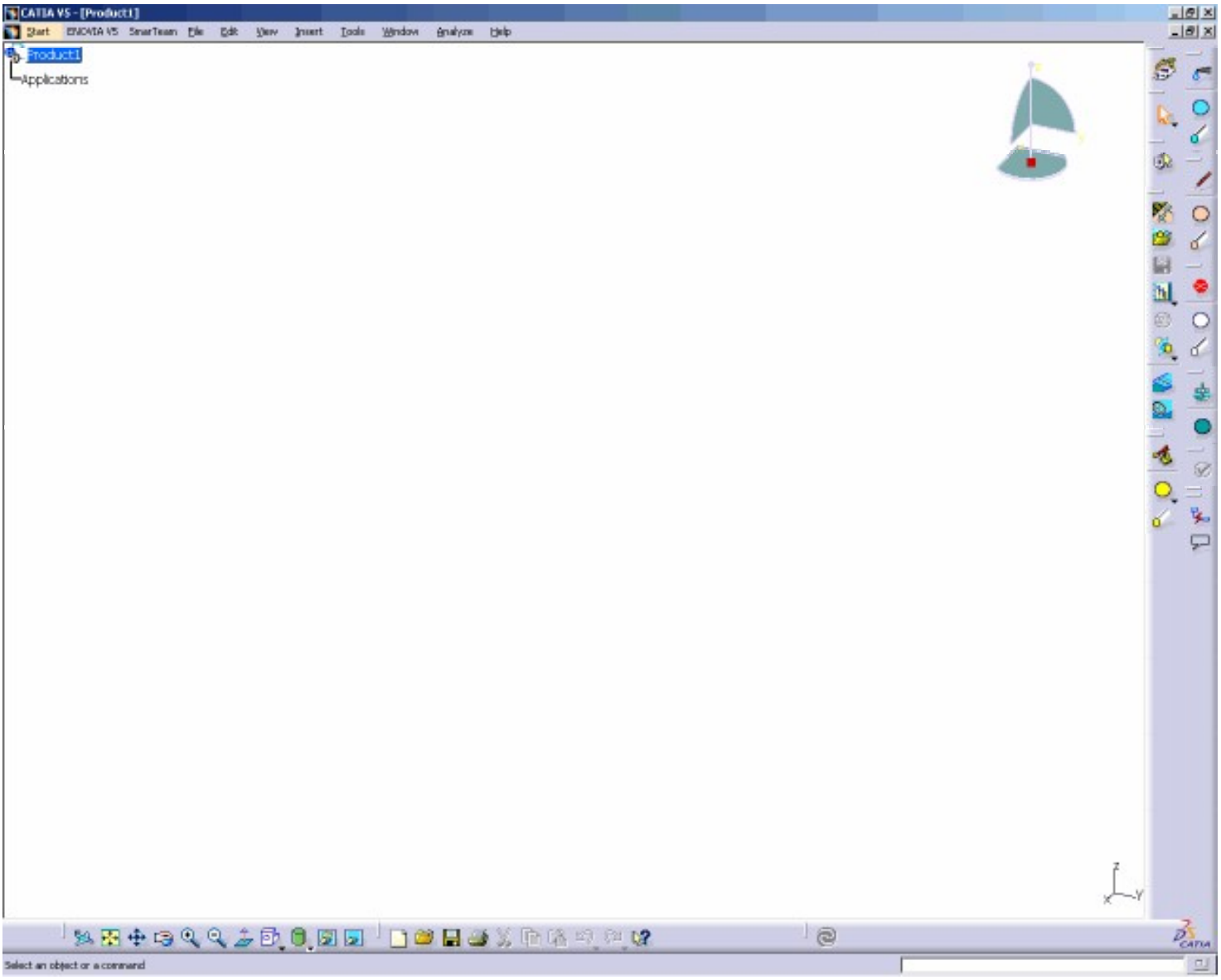
Command	Accessibility in Enovia LCA	Warning/Comment
Automotive BiW Welding Parameters	Yes	
BiW Data Import	No	
BiW Data Export	Yes	
BiW Reporting	Yes	
BiW Flat Reporting	Yes	
Renaming	No	
Mirror/Copy	Yes	If the target is not in Explode mode
Translate	Yes	If the target is not in Explode mode
BiW Joint	No	
BiW Joint Body	No	
BiW Welding SpotPoint	No	
BiW Welding SpotProjection	No	
BiW Welding CurveBead	No	
Automotive BiW Adhesive Parameters	Yes	
BiW Adhesive SpotPoint	No	
BiW Adhesive CurveBead	No	

Automotive BiW Sealant Parameters	Yes	
BiW Sealant SpotPoint	No	
BiW Sealant CurveBead	No	
Automotive BiW Mechanical Fasteners Parameters	Yes	
BiW Clinching SpotPoint	No	
Automotive BiW Unspecified Process Parameters	Yes	
BiW SpotPoint	No	
BiW CurveBead	No	
Check	No	
Update	No	
Activate/Deactivate	No	
Isolate	No	
Delete	No	
Copy	Yes	If the target is not in Explode mode

Workbench Description

This section contains the description of the icons, menus and Historical Graph that are specific to the Automotive Body In White Fastening workbench, which is shown below.

You can click the hotspots on this image to see the related documentation.



Menu Bar
Automotive BiW Fasteners Toolbar
Welding Toolbar
Adhesive Toolbar
Sealant Toolbar
Unspec Toolbar
Mechanical BiW Fasteners Toolbar
Analysis Toolbar
Tools Toolbar
Measure Toolbar
Specification Tree

Menu Bar

This section presents the tools and commands which are available in the FreeStyle Shaper, Optimizer and Profiler workbenches.
Many other operations are documented in the *Infrastructure User's Guide*.

Start | File | Edit | View | Insert | Tools | Windows | Help

File

The File menu lets you perform file creation, opening saving, printing operations

Edit

The Edit menu lets you manipulate selected objects. Refer to the *Infrastructure User's Guide* and *Part Design User's Guide*.

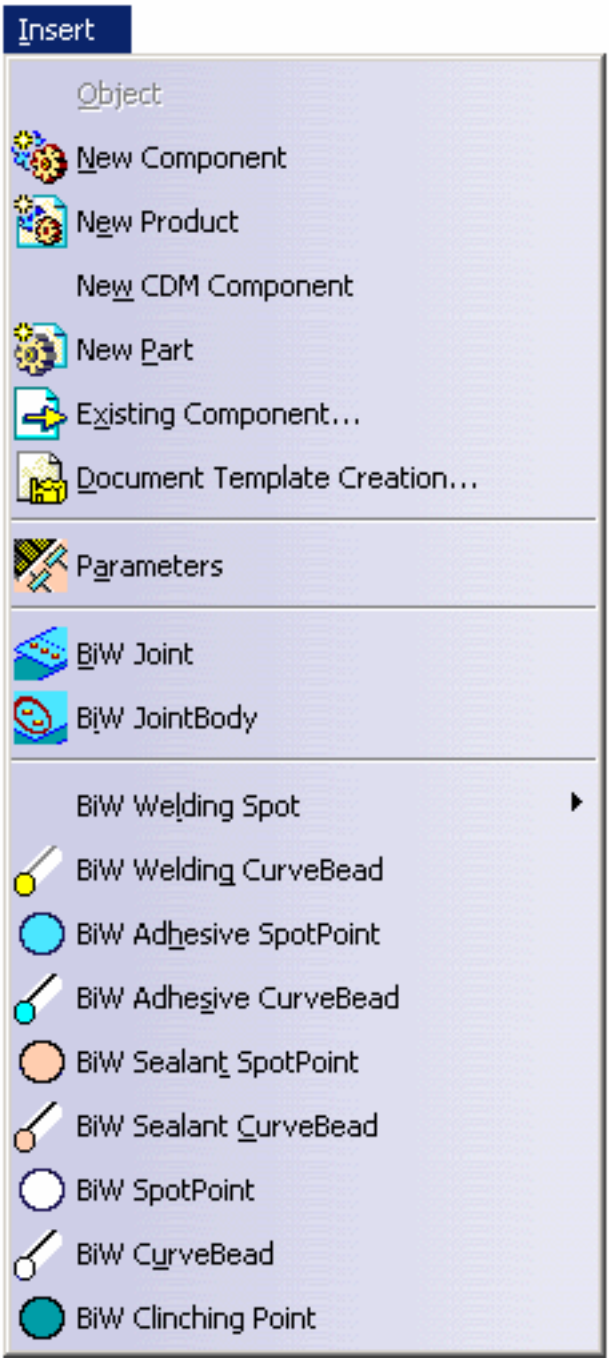
View

The View menu lets you view document contents Refer to the *Infrastructure User's Guide*.

Insert

The Insert menu lets you insert Automotive BiW Fastenings elements.

For...	See...
Object	
New Component	
New Product	
New CDM Component	*Product Structure User's Guide*
New Part	
Existing Component...	
Document Template Creation...	Creating a Part Template in the *Product Knowledge Template User's Guide*
Parameters	Defining the BiW Parameters
BiW Joint	Creating a Joint



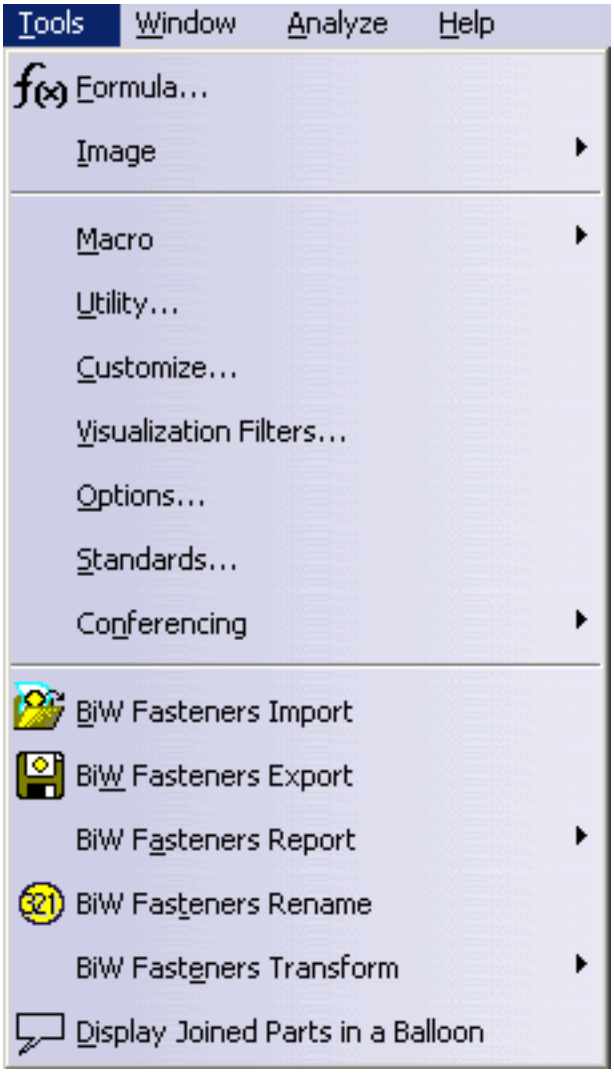
BiW JointBody	Creating a Joint Body
BiW Welding SpotPoint	Insert -> BiW Welding Spot
BiW Welding Spot	Insert -> BiW Welding Spot
BiW Welding CurveBead	Creating Welding CurveBeads
BiW Adhesive SpotPoint	Creating Adhesive SpotPoints
BiW Adhesive CurveBead	Creating Adhesive CurveBeads
BiW Sealant SpotPoint	Creating Sealant SpotPoints
BiW Sealant CurveBead	Creating Sealant CurveBeads
BiW SpotPoint	Creating BiW SpotPoints
BiW CurveBead	Creating Unspecified CurveBeads
BiW Clinching SpotPoint	Creating a Clinching SpotPoint

Insert -> BiW Welding Spot

For...	See...
BiW Welding SpotPoint	Creating a BiW Welding SpotPoint
BiW Welding SpotProjection	Creating SpotProjection Welds

Tools

The Tools menu lets you perform image capture and album management, set user preferences and manage macros. Refer to the *Infrastructure User's Guide*.



For	See...
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Image	*Infrastructure User's Guide*
Macro	
Utility...	Reporting Using CATUtil
Customize	
Visualization Filters...	
Options...	*Infrastructure User's Guide*
Standards...	
Conferencing	
BiW Fasteners Import	Importing
BiW Fasteners Export	Exporting
BiW Fasteners Report	Tools -> BiW Fasteners Report
BiW Fasteners Rename	Renaming
BiW Fasteners Transform	Tools -> BiW Fasteners Transform
Display Joined Parts in a Balloon	Displaying Joined Parts in a Balloon

Tools -> BiW Fasteners Report

For...	See.
Report	Reporting
Flat Report	Creating Flat Reports

Tools -> BiW Fasteners Transform

For...	See.
BiW Mirror Copy	Creating Mirror / Copy Elements
Translated Copy	Creating Translate / Copy Elements

Window

The Window menu lets you arrange document windows in relation one to the other. Refer to the *Infrastructure User's Guide*.

Help

The Help menu lets you get help on the currently active command, and the product in general. Refer to the *Infrastructure User's Guide*.

Automotive BiW Fasteners Toolbar

The Automotive BiW Fasteners Toolbar contains the following tools:

 See Defining the BiW Parameters

 See Importing

 See Exporting

 See Reporting

 See Creating Flat Reports

 See Renaming BiW Features

 See Creating Mirror/Copy Elements

 See Creating a Joint

 See Creating a Joint Body

Welding Toolbar

The Welding Toolbar contains the following tools:


See Displaying the Process Category parameters


See Creating a BiW Welding SpotPoint


See Creating SpotProjection Welds


See Creating Welding CurveBeads

Adhesive Toolbar

The Adhesive Toolbar contains the following tools:

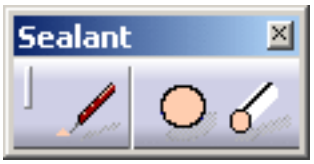
 See Displaying the Process Category parameters

 See Creating BiW Adhesive SpotPoints

 See Creating BiW Adhesive CurveBeads

Sealant Toolbar

The Sealant Toolbar contains the following tools:



 See Displaying the Process Category parameters

 See Creating BiW Sealant SpotPoints

 See Creating BiW Sealant CurveBeads

Unspec Toolbar

The Unspec Toolbar contains the following tools:

 See Displaying the Process Category parameters

 See Creating BiW SpotPoints

 See Creating BiW CurveBeads

Mechanical BiW Fasteners Toolbar

The Mechanical Toolbar contains the following tools:

 See Displaying the Process Category parameters

 See Creating a BiW Clinching SpotPoint

Analysis Toolbar

The Analysis Toolbar contains the following tool:

 See Running the BiW Fastening Rules Analysis

Tools Toolbar

The BiW Tools Toolbar contains the following tools:

See Creating BiW Features Using the Datum Mode

See Displaying Joined Parts in a Balloon

Measure Toolbar

The Measure Toolbar contains the following tools:

See Measure Tools

Specification Tree

Within the Automotive Body In White Fastening workbench, you can generate a number of elements that are identified in the specification tree by the following icons.

 Parameters

 Joint

 Joint Body

 Welding SpotPoint

 Welding SpotProjection

 Welding CurveBead

 Adhesive SpotPoint

 Adhesive CurveBead

 Sealant SpotPoint

 Sealant CurveBead

 BiW SpotPoint

 BiW CurveBead

 Clinching SpotPoint

Customizing

Before you start your first working session, you can customize the way you work to suit your habits.
This type of customization deals with permanent setting customization: these settings will not be lost if you end your session.

1. Select the **Tools -> Options** menu item.

   The Options dialog box appears.

2. Click the **Shape** category in the left-hand box.

3. Click the **Automotive Body In White** workbench.

   The General and the Display tabs are displayed.

- The General tab lets you define the application and extract parameters, as well as the export and report directories.
- The Display tab lets you define the display options.

4. Set options in these tabs according to your needs.

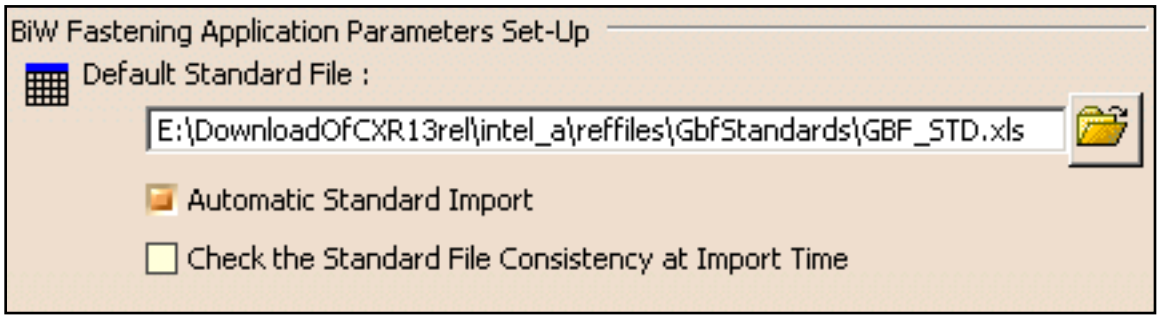
5. Click OK when done.

General Settings

This page deals with the following settings:

- Setting Up Parameters
- Defining the Export Directory
- Defining the Report Directory
- Setting Up the Extract Parameters
- Allowing the Creation of unsaved data from Enovia

Setting Up Parameters



Default Standard File

The Default Standard File path corresponds to the path of the file where the default standards are stored.

You can change the path by clicking the **Browse** icon  .

Automatic Standard Import

You can automatically import the standard by clicking on the **Automatic Standard Import** check box.
If this option is activated, any BiW feature creation (BiWJoint, BiW JointBody, BiW Fastener) automatically imports the standard.

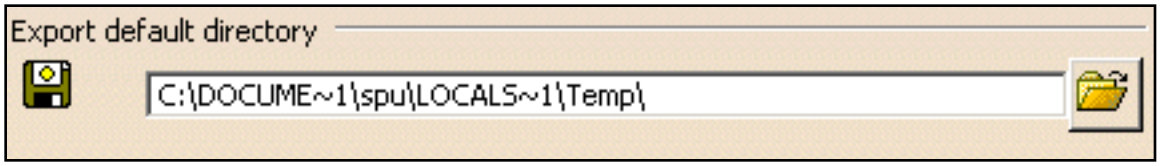
By default, this option is checked.

Check the Standard File Consistency at Import Time

You can also select the **Check the Standard File Consistency at Import Time** option to check the consistency of the standard values when importing the standard file (whose path is mentioned above)
If the standard file is not valid, it cannot be imported and an error message is issued.

By default, this option is unchecked.

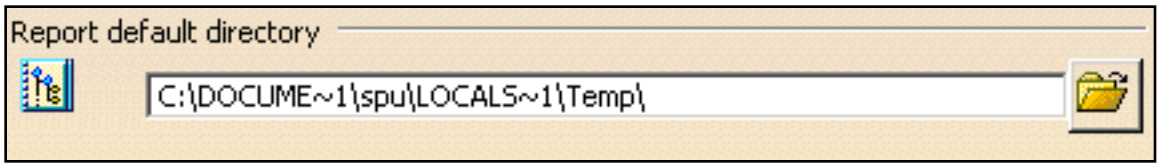
Defining the Export Directory



The Export default directory is the directory where the export file is stored.

You can change the by clicking the **Browse** icon .

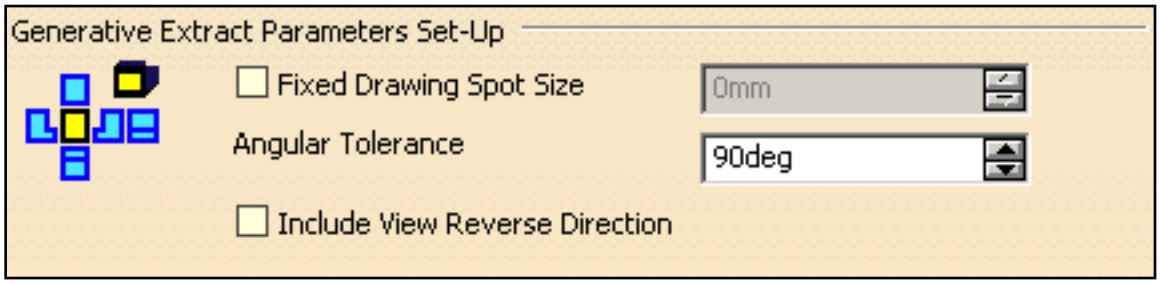
Defining the Report Directory



The Report default directory is the directory where the report file is stored.

You can change the by clicking the **Browse** icon .

Setting Up Extract Parameters



Fixed Drawing Spot Size

You can define a fixed size for the 2D drawing spots: check the **Fixed Drawing Spot Size** button and define a diameter value.
As a consequence, all spots in the 2D document will have this diameter, whatever their size in the 3D document.

By default, this option is unchecked.

Angular Tolerance

You can define a user **Angular Tolerance**.

By default, this option is set to 90deg.

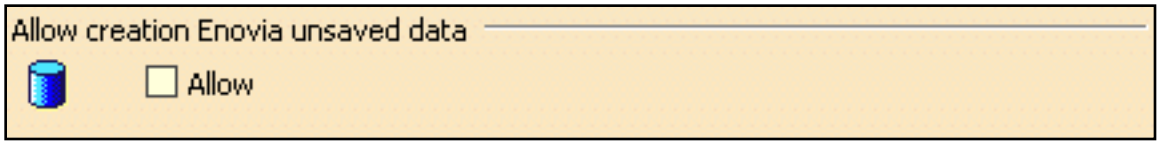
Include View Reverse Direction

You can check the **Include View Reverse Direction** checkbox to visualize 2D reverse spots.

By default, this option is unchecked.

Refer to the Drafting Integration chapter to have further information.

Allow creation Enovia unsaved data



You can allow the creation of unsaved data from Enovia by clicking the **Allow** button.
If you do not select this option, commands will be grayed out in Enovia V5.

By default, this option is unchecked.

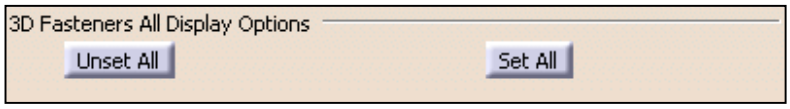
For further information, refer to the Optimal CATIA PLM Usability for Automotive BiW Fastening chapter.

BiW Fastening Application Display User Settings

This page deals with the following settings:

- 3D Fasteners All Display Options
- Shape
- ID
- Fastener Parameters
- Fastener Axis Orientation
- Support Contact Zone Material Orientation
- Contact Zones

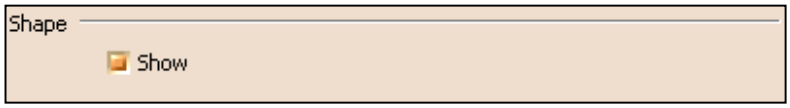
3D Fasteners All Display Options

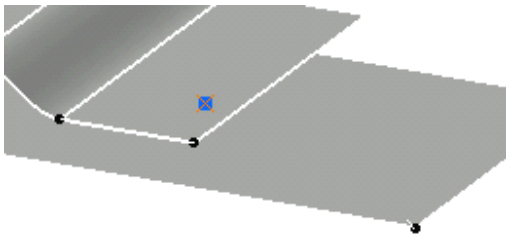


Click the **Unset All** button to uncheck all the options, except for the Shape Representation option. Similarly click the **Set All** button to check all options.

These options apply to all the fasteners contained in the product.

Shape

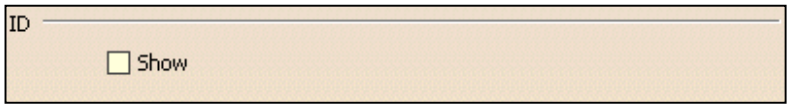


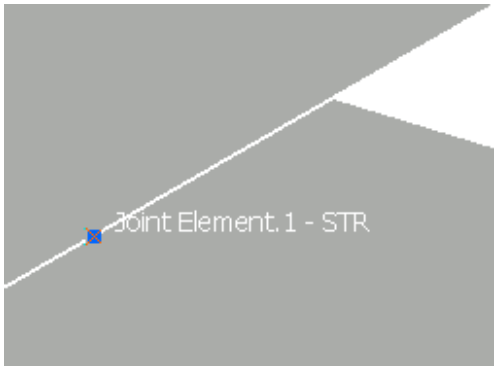


Click the **Show** button to visualize the fastener's shape.

By default, this option is checked.

ID

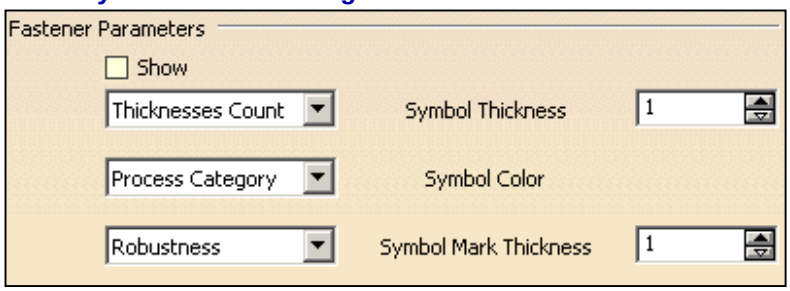




Click the **Show** button to visualize the fastener's ID.

By default, this option is unchecked.

Fastener Parameters

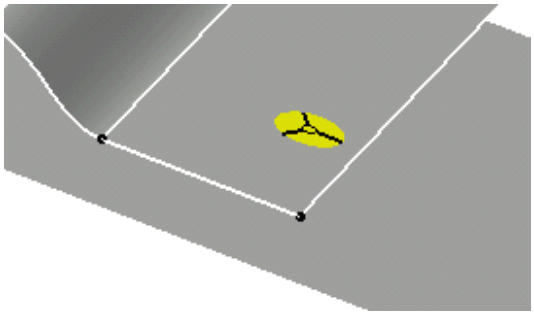


Click the **Show** button to visualize the fastener's shape.

By default, this option is unchecked.

Associating a symbol to a specification

- a process type (by default): the symbol is displayed according to the symbol code assigned to the Fastener Process Type in the standard file. By default, if no symbol is assigned to the process type in standard, or if there is no imported standard, the symbol is (101)
- an unspecified type (Unspec): the default symbol is displayed. It can be modified through the **Properties** contextual command.
- the thickness count: the symbol is displayed according to symbol code assigned to the Thickness Count in the standard file.
  By default, if no symbol is assigned to the thickness count in standard, or if there is no imported standard, the symbol is:



for 1 thicknesses (1)
for 2 thicknesses (2T)
for 3 thicknesses (3T)
for 4 thicknesses (4T)
for more than 4 thicknesses

- None: no symbol is visualized.

Once the symbol is associated with a specification, it can not longer be modified when editing the fastener's properties, except for the Unspecified type.

The example above shows that the graphic symbol is assigned to the Laser process type. The symbol color is assigned to the Welding process category.

// Symbol Dependency from Thickness Count	
NTH_2_SYM	151
NTH_3_SYM	113
NTH_4_SYM	114

For 1T:
For 2T:
For 3T:
For 4T:

The example above shows how to define a graphic symbol associated to a thickness count

Associating a color to a specification

- a process category (by default): the color is displayed according to the color code assigned to the Fastener Process Category in the standard file.

  By default, if no color is assigned to the Process Category or no standard is imported, the default color is:

  Yellow for Welding
  Cyan for Adhesive
  Magenta for Sealant
  Green for BiW Mechanical
  White for Unspecified

- an unspecified type (Unspec): the color is white whatever the Process Category.

- None: no symbol color is visualized.

In this case, the color associated with the process category will be transparent.

Once the color is associated with a specification, it can not longer be modified when editing the fastener's properties, except for the Unspecified type.

Associating marks to the specification

- Robustness
- Regulation
- Finish
- None

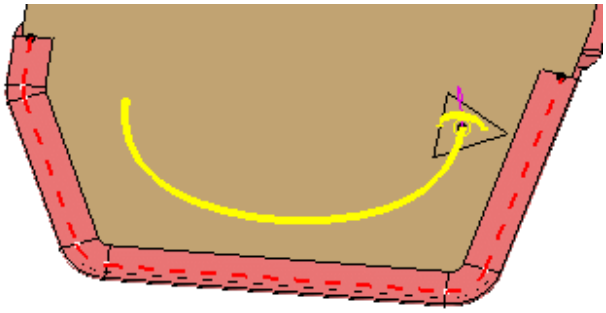
Note that:

- 1 is for assigning a mark
- 0 is for assigning no mark

For instance assign the 1 mark code for the Robustness parameter and which value set by default is C:

ROB_C_SYMO

You will get this Mark:

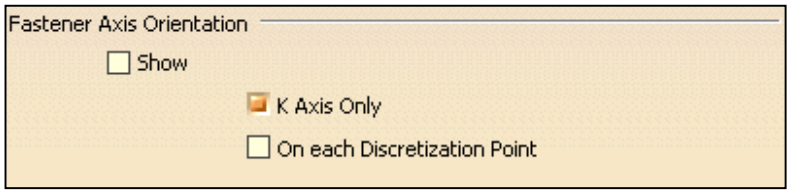


In this case all the Fasteners with the Robustness specification will get the mark symbol.

In the Standard file, Marks are assigned by default:

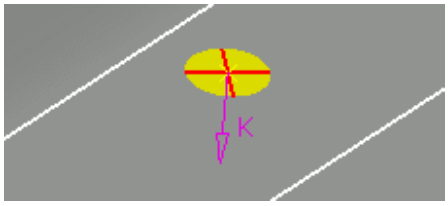
- For Robustness parameter ("Safety") enter the C value.
- For Regulation parameter ("Norm A") enter the A value.
- For Finish parameter ("Class A") enter the A value.

Fastener Axis Orientation



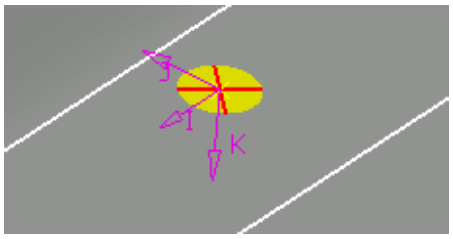
Click the **Show** button to visualize the fastener's axis orientation.

By default, this option is unchecked.

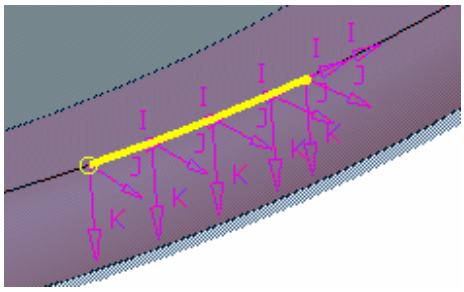


- **K Axis Only**:
  - if checked, only the K axis is visualized on the fastener's location.
  - If unchecked, a trihedron showing the fastener's axis system (I, J, K) is visualized on the fastener's location.

    For a curvebead, the location corresponds to the start point.

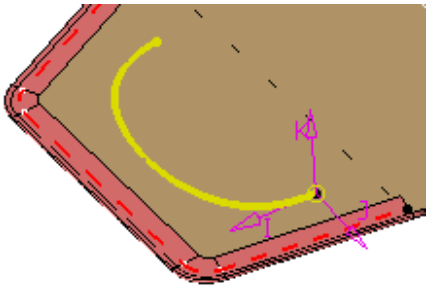


By default, this option is checked.



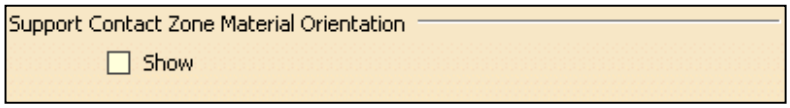
- **On each Discretization Point**: for a curvebead, it enables to visualize the axis (K or I, j, K) on each discretization point.
  - if checked: the fastener axis (K or I, j, K) is visualized on each discretization point of the curvebead.
  - if unchecked, the the fastener axis (K or I, j, K) is visualized on the start point of the curvebead only.

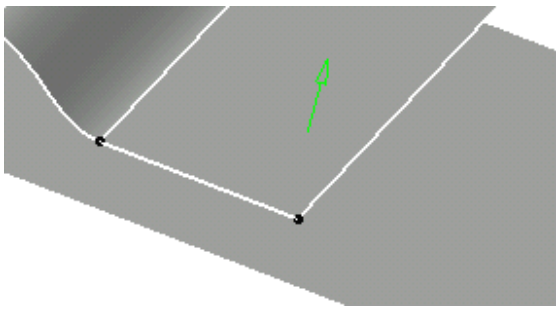
    In the examples beside, the **Fastener Axis Orientation** button is checked and the **K Axis Only** button is unchecked.



By default, this option is unchecked.

Support Contact Zone Material Orientation

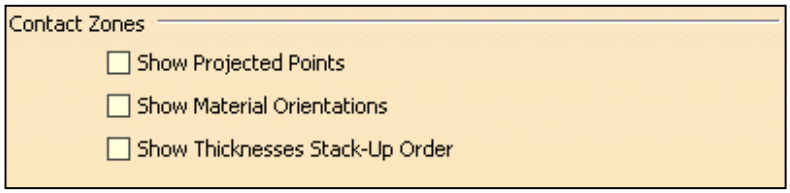




Click the **Show** button to visualize the material orientation of the support contact zone.

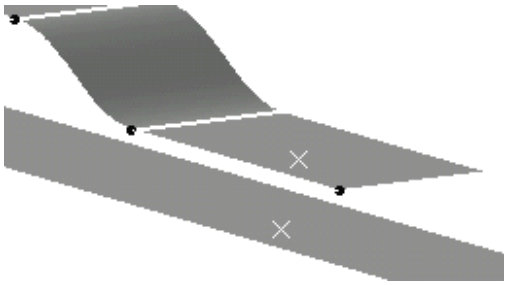
By default, this option is unchecked.

Contact Zones



Show Projected Points

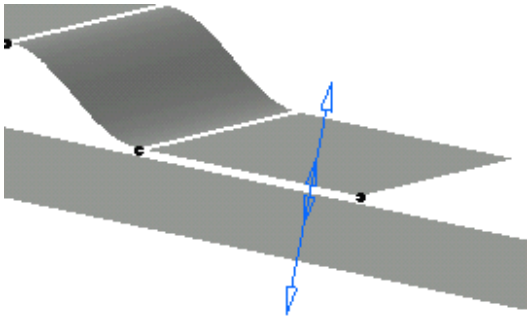
Select this option to visualize the fastener's projected points.



By default, this option is unchecked.

Show Material Orientation

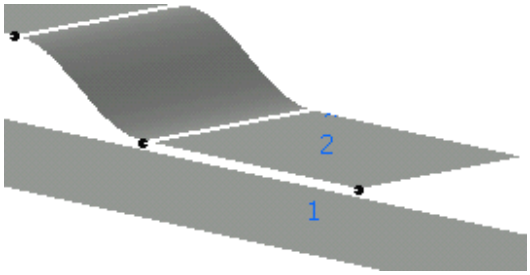
Select this option to visualize the fastener's material orientation.



By default, this option is unchecked.

Show Thickness Stack-Up Order

Select this option to visualize the fastener's thickness stack-up order.



By default, this option is unchecked.

- You can define the thickness and the material orientation for a GSD feature using the *Thin Parts Attribute* command in the Generative Shape Design workbench.
  Refer to the *Applying a Thickness* chapter in the Generative Shape Design documentation.
- The options listed above can also be accessed using the fasteners' properties via the contextual menu.
  Refer to the Editing the Fasteners' Properties chapter.

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