DMU Space Analysis

Overview

Welcome to the *DMU Space Analysis User's Guide*. This guide is intended for users who need to become quickly familiar with the DMU Space Analysis Version 5 product.

This overview provides the following information:

DMU Space Analysis in a Nutshell

DMU Space Analysis  is a CAD-independent product dedicated to interference analysis, sectioning, measurement and 3D comparison. It addresses the design review environment of digital mock-ups (DMU) and can handle a wide range of products from consumer goods to very large automotive or aerospace projects as well as plants, ships and heavy machinery.

DMU Space Analysis is a dedicated DMU Navigator workbench and is available on both UNIX and Windows environments.

Before Reading this Guide

Before reading this guide, the user should be familiar with basic Version 5 concepts such as document windows, standard and view toolbars. We therefore recommend that you read the *Infrastructure User's Guide* that describes generic capabilities common to all Version 5 products. We also recommend that you read the *DMU Navigator User's Guide*.

You may also like to read the following complementary product guides, for which the appropriate license is required:

- *DMU Fitting Simulator User's Guide*
- *DMU Kinematics Simulator User's Guide*
- *DMU Optimizer User's Guide*.

Getting the Most Out of this Guide

To get the most out of this guide:

- If you are a first time user, we suggest that you start with the getting started tutorial. Once you have finished, you should move on to the user task section of this guide. This steps you through basic procedures and includes useful tips for getting the most out of the product.
- If you have used DMU Space Analysis before, go to your DMU Version 5 session and start reviewing your own documents. If you need some help in understanding tools and commands, use the on-line help. You can also take a look at the user task section of this guide to locate information with which you are not already familiar.

The workbench description, which describes DMU Space Analysis-dedicated menu bar and workbench toolbar, and the Customizing section, which explains how to customize your personal environment, will also certainly prove useful. A glossary of terms specific to DMU Space Analysis has also been included in this guide.

For information on automation capabilities, see *Space Analysis* on the Automation Documentation Home Page.

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, you will be using sample documents contained in either the online/spaug/samples folder or in the online/cfyug/samples folder. For more information about this, 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?

Enhanced Functionalities

Distance & Band Analysis

Export and publish distance results
: DLNames are supported when choosing the directory in which results will be stored

Sectioning

Integrated into DMU Navigator enhanced scenes
: Sectioning is available in the enhanced scene context

Export section results
: DLNames are supported when choosing the directory in which results will be stored

3D section cut
: Can now manage the section cut display when the sectioning tool is a plane and make measures on the generated section

Clash

Export clash results
: DLNames are supported when choosing the directory in which results will be stored

Batch processing existing CATProducts
: Better product management reduces memory needed for batch processing

Continued performance improvement in clash storage
: When you save a document containing interference results which include a triangular representation for contacts and clearances, there are significant reductions in the size of saved contact and clearance results

Measure Tools

Editing measures
: Using the Replace mechanism, you can replace the selections on which the measure is based in a part

Complementary angle in Measure Between & Measure Item
: Can now obtain the complementary angle

Integrated into DMU Navigator enhanced scenes
: Measures are available in the enhanced scene context

Integrated into the Advanced Meshing Tools workbench
: Measure Between and Measure Item commands are now available in the Advanced Meshing Tools workbench

Picking point selection mode in Measure Between
: You can now measure curve length in the section viewer

Customizing Settings

DMU Sectioning

New setting lets you manage 3D section cut display when sectioning tool is a plane

DMU Distance

DLNames are supported when choosing the directory in which the XML file will be stored and when selecting the stylesheet

DMU Clash - Process

DLNames are supported when choosing the directory in which the XML file will be stored and when selecting the stylesheet

Getting Started

This tutorial will guide you step-by-step through your first Space Analysis session, allowing you to get acquainted with the product.

You will need a DMU Navigator Version 5 session and should be familiar with basic concepts such as document windows, standard and view toolbars.

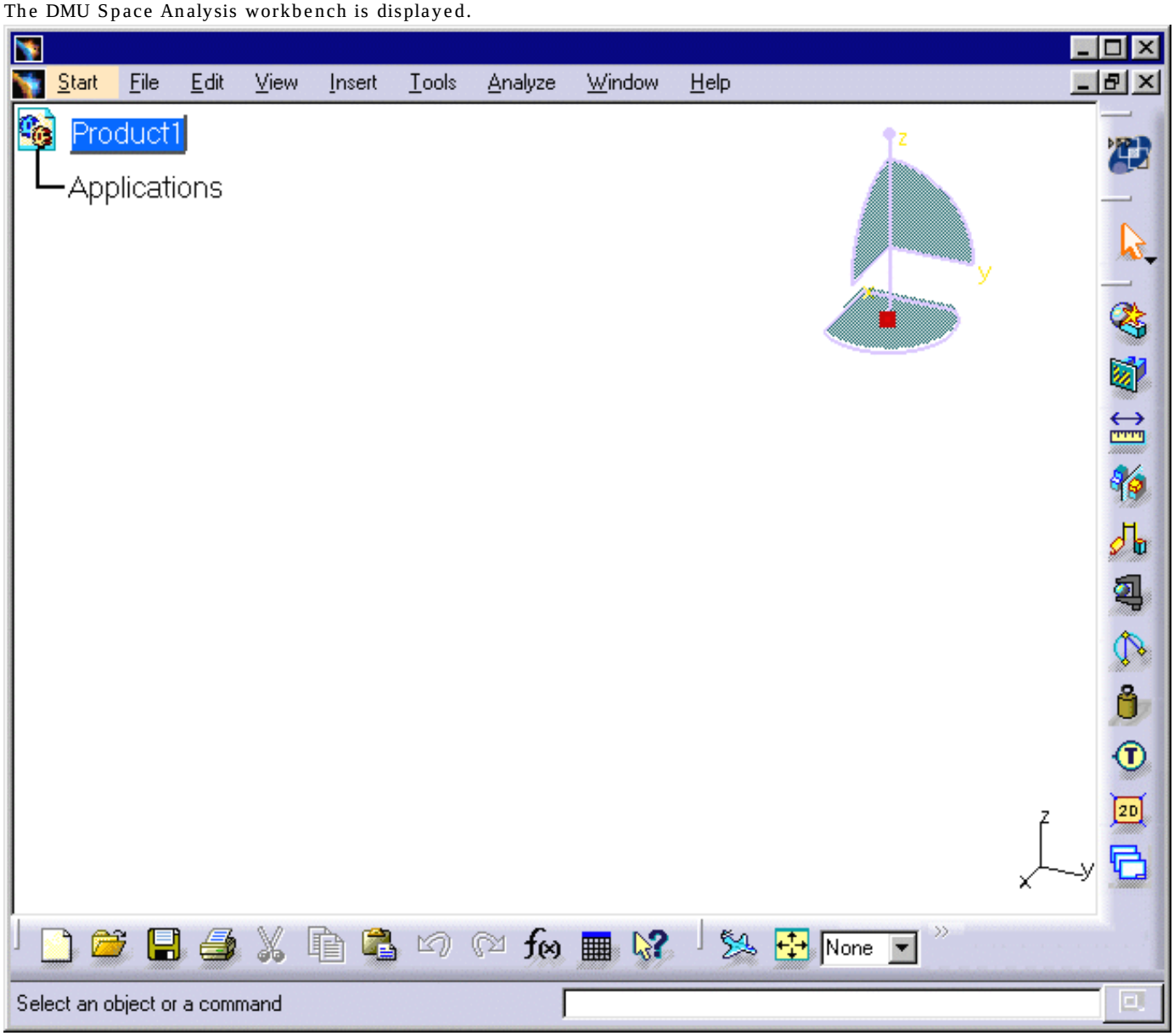
You should be able to complete this tutorial in about 15 minutes.

Setting Up Your DMU Space Analysis Session

This task shows you how to enter the DMU Space Analysis workbench and insert desired cgr files.

1. Select **Digital Mockup ->DMU Space Analysis** from the **Start** menu.

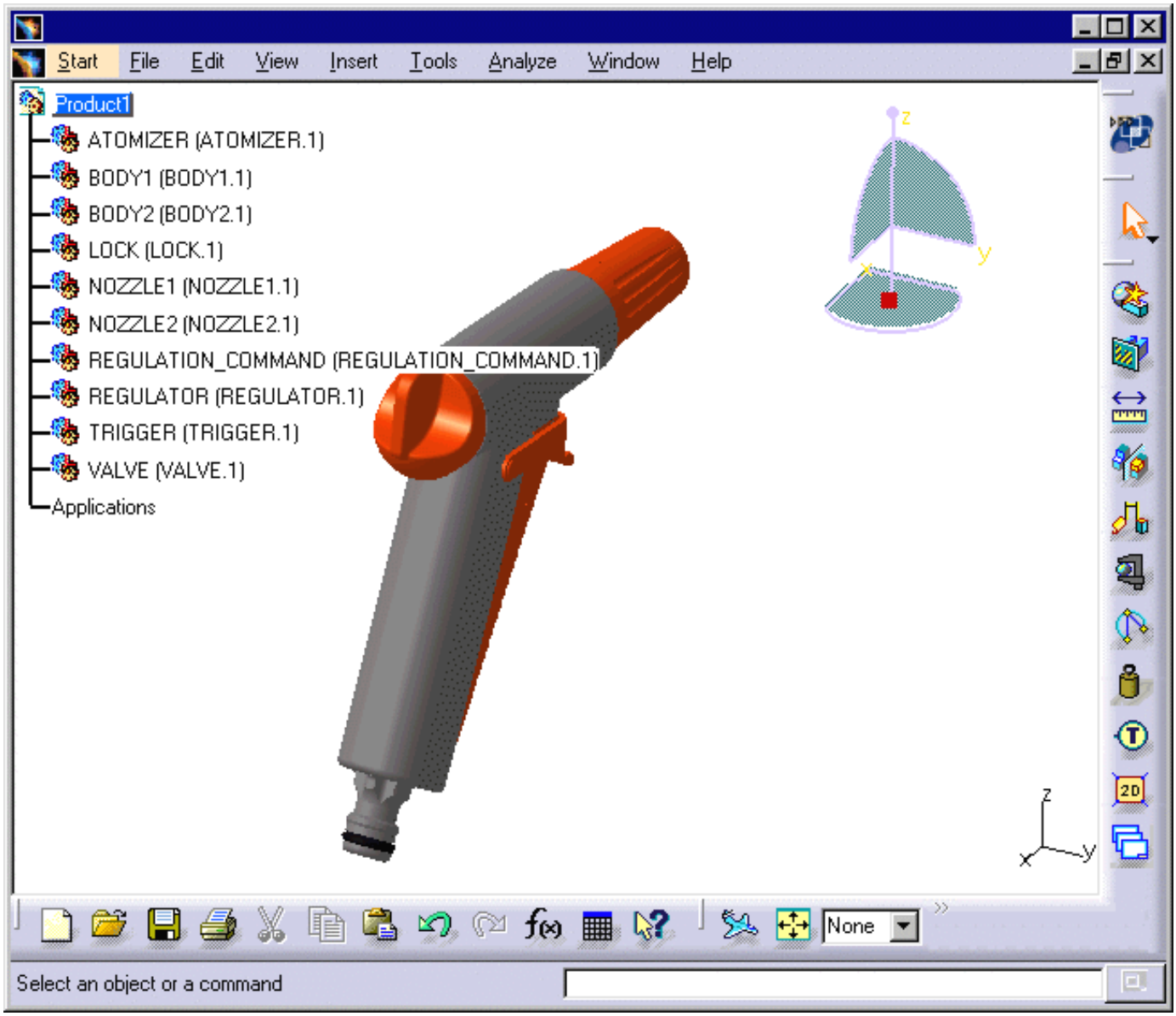
   The DMU Space Analysis workbench is displayed.



2. If not already active, select Product1 in the specification tree.
3. Select **Insert -> Existing Component...** from the menu bar to insert desired cgr files.

   The Insert an Existing Component dialog box appears.
4. Specify the file location for the cgr files of interest: cgr files are to be found in the common functionalities sample folder cfysm/\samples.

   For more information on where sample documents are installed by default, see Accessing Sample Documents in the *Infrastructure User's Guide*.
5. Set Files of type to cgr (*.cgr).
6. Select the following sample cgr files: ATOMIZER.cgr, BODY1.cgr, BODY2.cgr, LOCK.cgr, NOZZLE1.cgr, NOZZLE2.cgr, REGULATION_COMMAND.cgr, REGULATOR.cgr, TRIGGER.cgr and VALVE.cgr.
7. Click Open in the dialog box to insert selected cgr files into the active document.



8. Set dimension display:

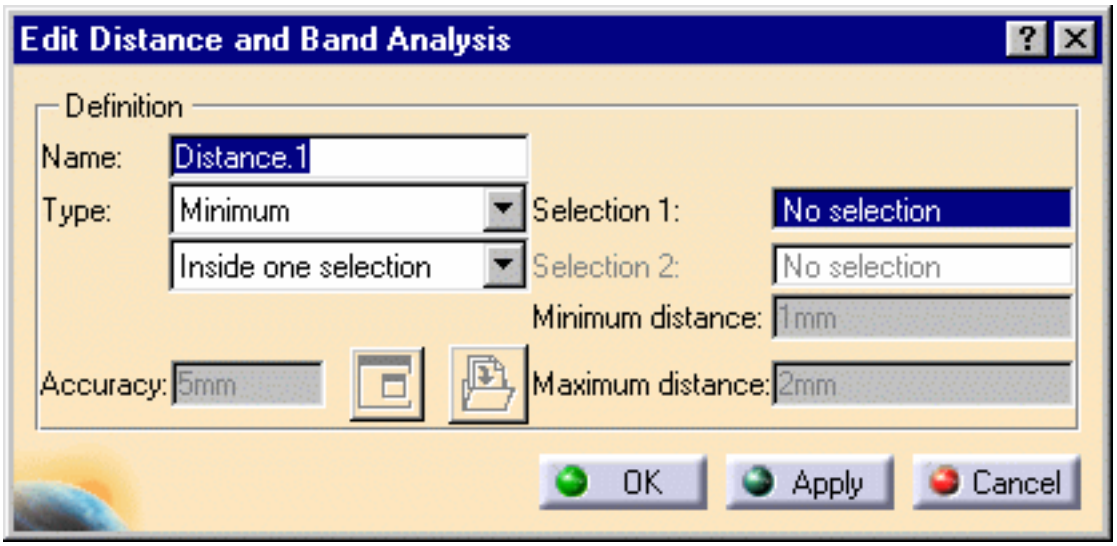
- Select the **Tools -> Options...** command
- In the **General** category, click the **Parameters and Measure** sub-category, then the **Unit** tab.
- In the lower part of the dialog box, set Decimal places for read/write numbers to 1 to display dimensions to one decimal place.
- Click **OK** when done.

Measuring Minimum Distances between Products

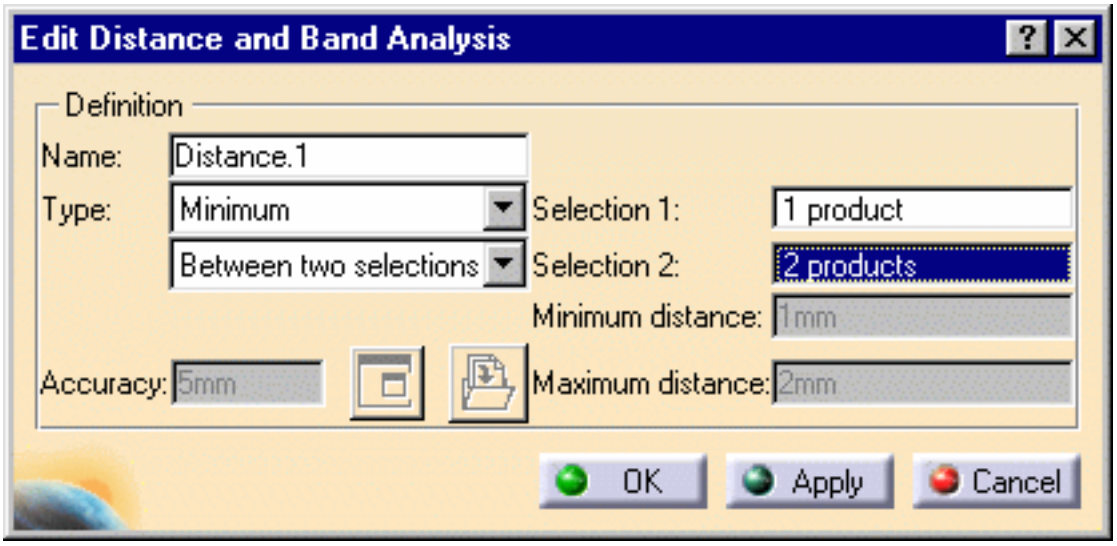
This task shows you how to measure minimum distances between products.

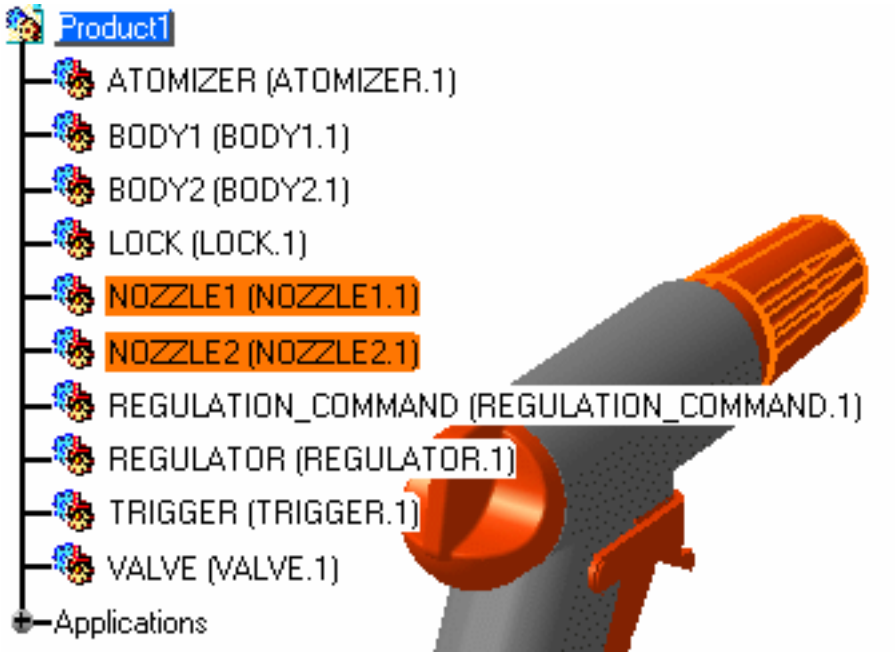
1. Click the Distance and Band Analysis icon in the DMU Space Analysis toolbar:

The Edit Distance and Band Analysis dialog box appears.



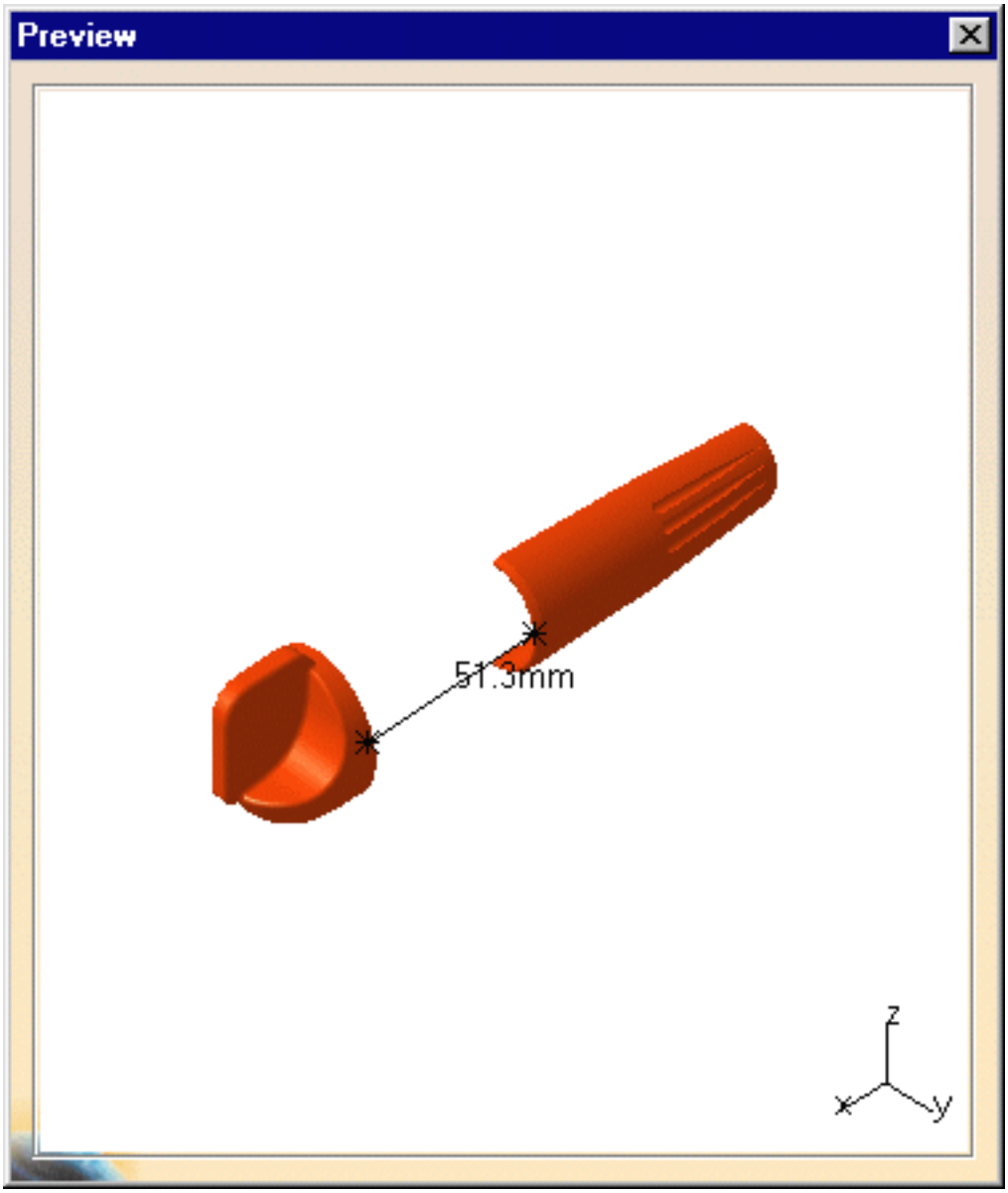
2. Select a product, for example the Regulation_Command.
3. Click the second Type drop-down list box and select Between two selections.
4. Select two other products, for example both nozzles.



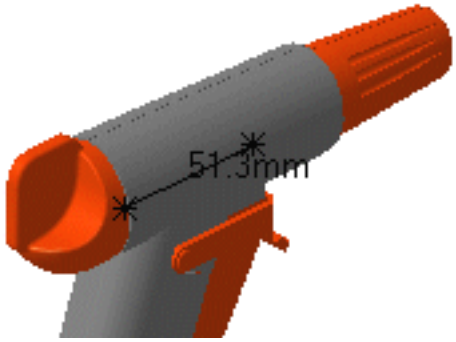


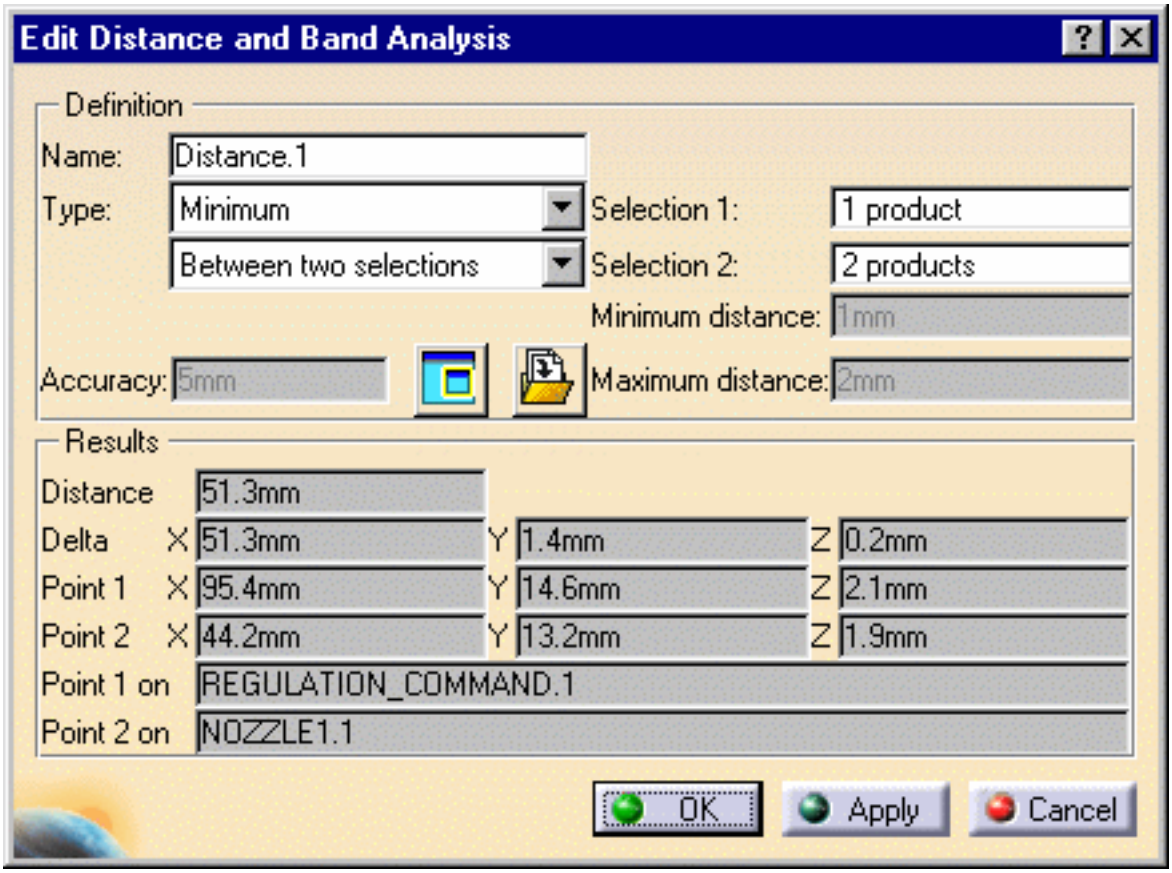
5. Click **Apply** to calculate the distance between selected products:

A Preview window appears visualizing selected products and the minimum distance (represented by a line, two crosses and a value).



The Edit Distance and Band Analysis dialog box expands to show the results and the minimum distance is also visualized in the geometry area.





6. Click **OK** in the Edit Distance and Band Analysis dialog box.

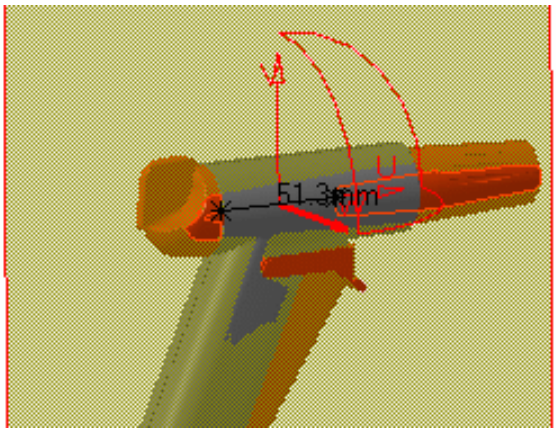
Sectioning

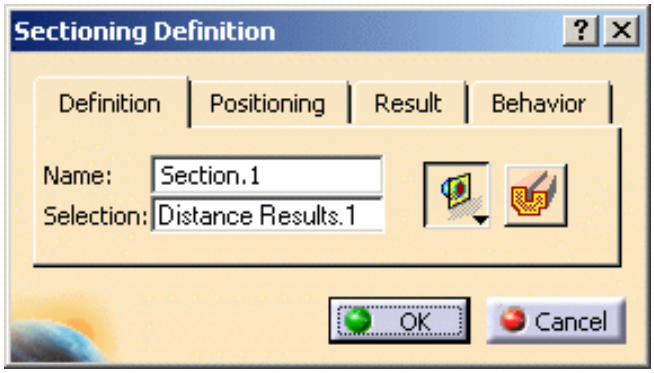
This task shows you how to create a section plane on the minimum distance.

1. Select the minimum distance in the geometry area.

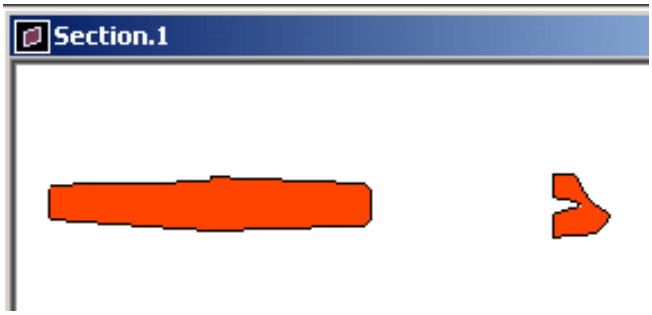
2. Click the Sectioning icon in the DMU Space Analysis toolbar:

   The section plane is created on the minimum distance. The Sectioning Definition dialog box appears.





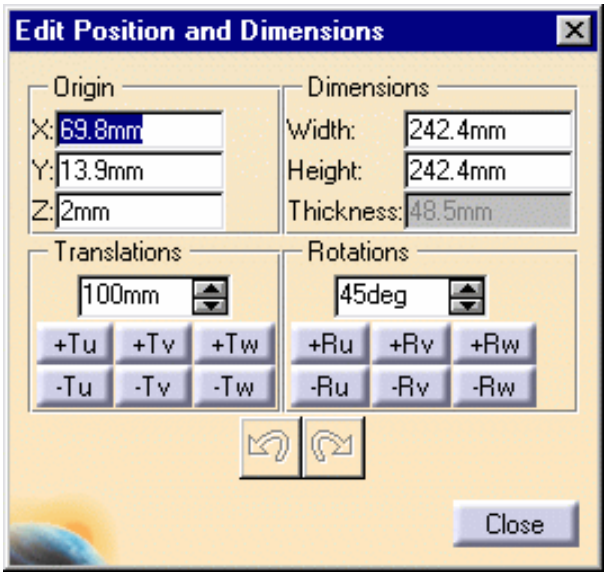
   The Section viewer, showing the generated section, is automatically tiled vertically alongside the document window. The section view is a filled view.



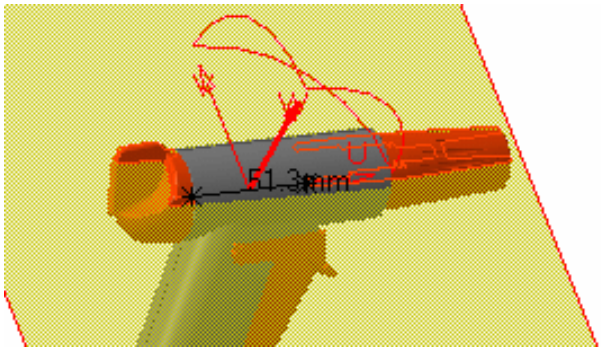
3. Click the Positioning tab, then the Edit Position and Dimensions icon to change parameters defining the current plane position.

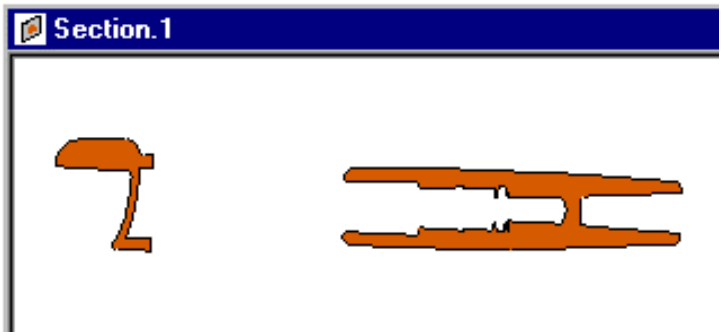
   The Edit Position and Dimensions dialog box appears.

   The U-axis of the section plane is positioned along the minimum distance.

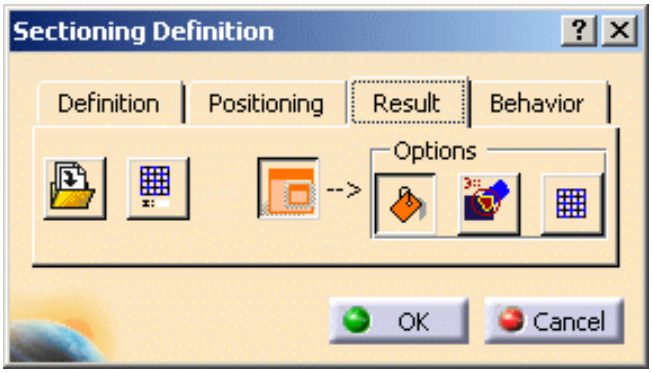


4. Click the +Ru and -Ru buttons (Rotations box) to rotate the plane around the minimum distance.
5. Click Close in the Edit Position and Dimensions dialog box when done.

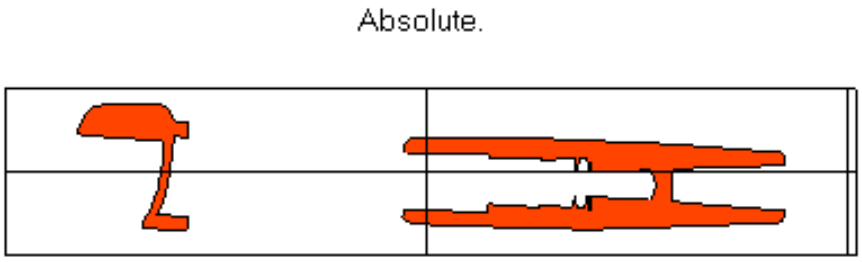




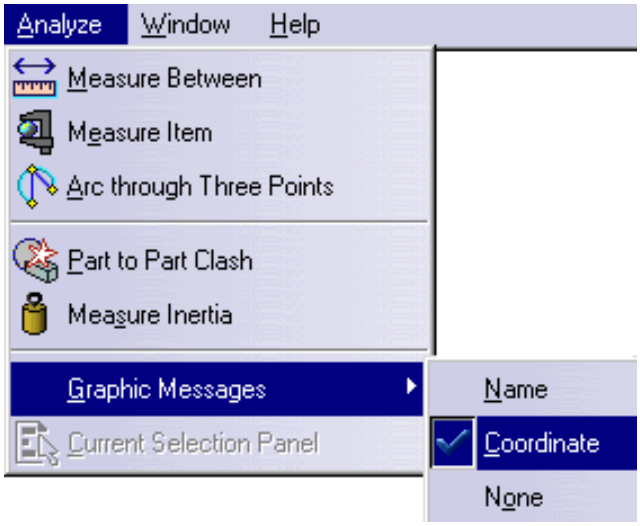
6. Click the Result tab in the Sectioning Definition dialog box to access commands specific to the Section viewer.



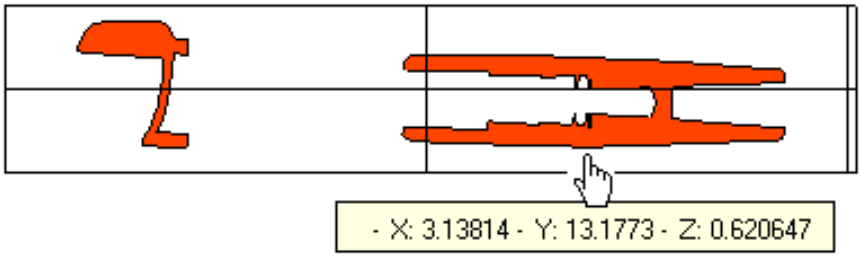
7. Select the Grid  icon to display a 2D grid.



8. Select **Analyze -> Graphic Messages -> Coordinate** from the menu bar to activate the coordinates option.

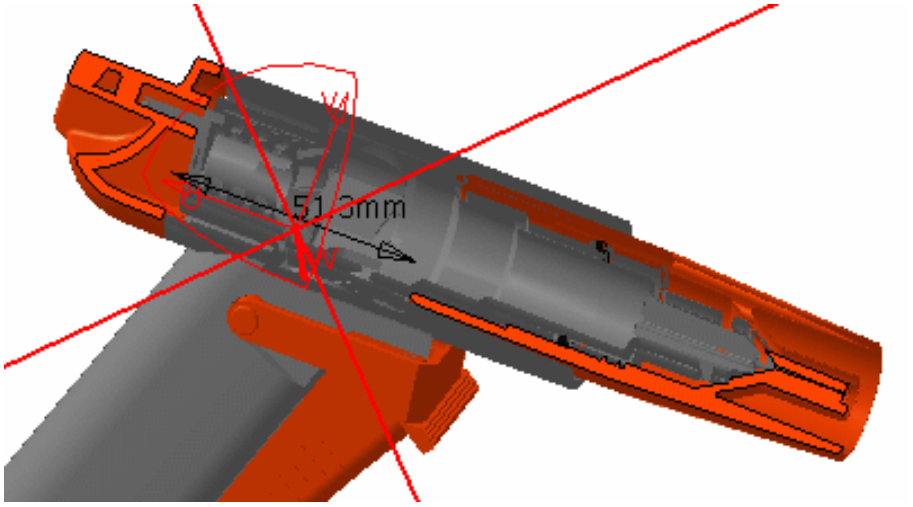


9. Move the mouse over the geometry in the results window to display the coordinates of the point selected.



10. Deselect the coordinates option.
11. In the Definition tab, click the Volume Cut icon to obtain a 3D section cut:

The material in the negative direction along the normal vector of the plane (W-axis) is cut away. The cavity within the product is exposed:



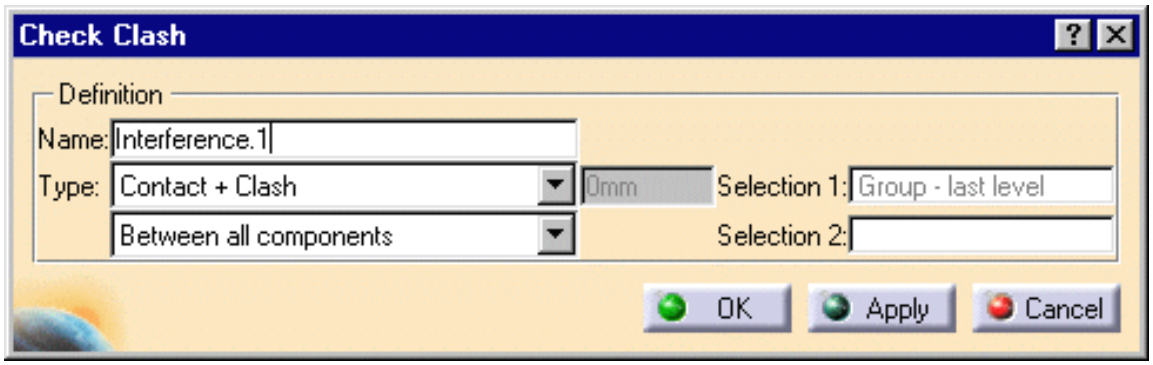
12. Re-click the Volume Cut icon to restore the representation.
13. Click **OK** in the Sectioning Definition dialog box when done to exit the sectioning command.

Detecting Clashes

This task shows you how to detect contacts and clashes between all the components in your document.

1. Click the Clash icon in the DMU Space Analysis toolbar:

The Check Clash dialog box appears.

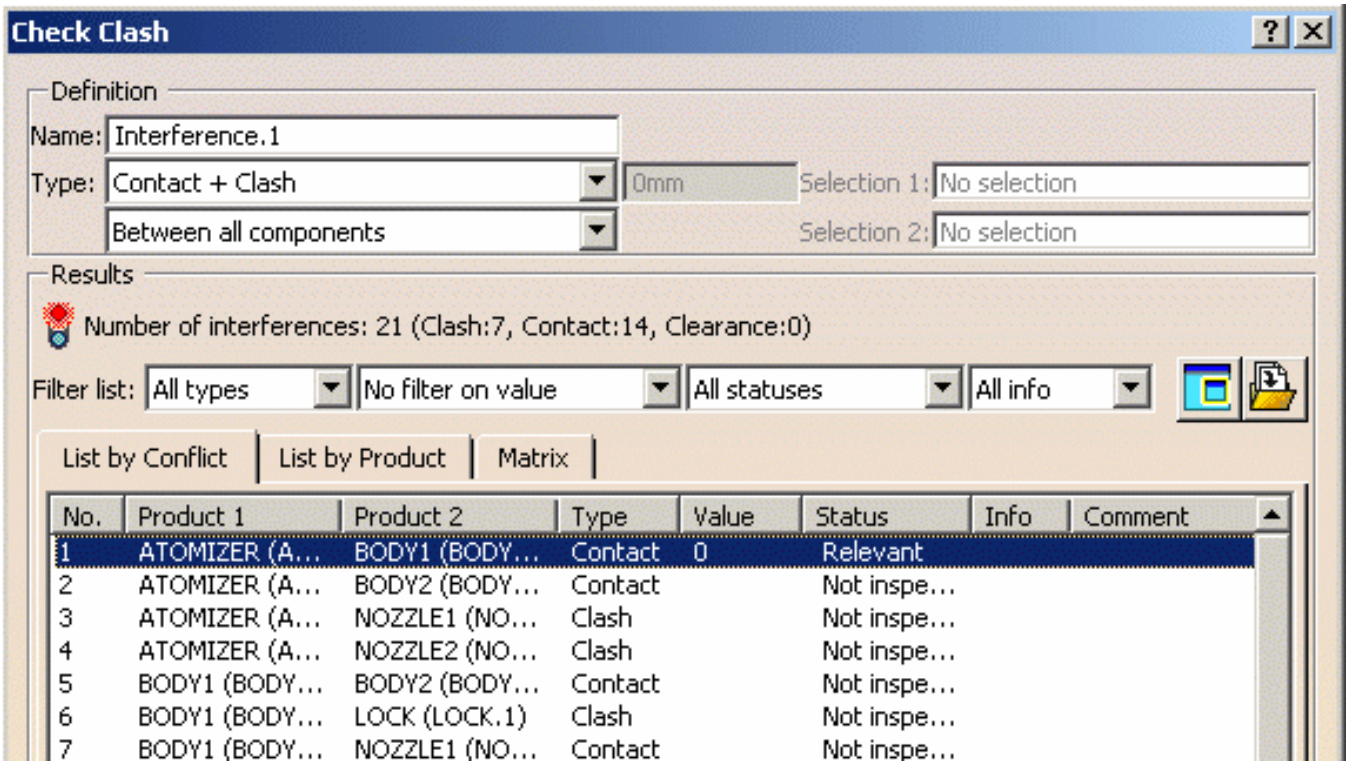


Contact + Clash checks whether two products occupy the same space zone as well as whether they are in contact. Between all components is the default value for the second Type drop-down list.

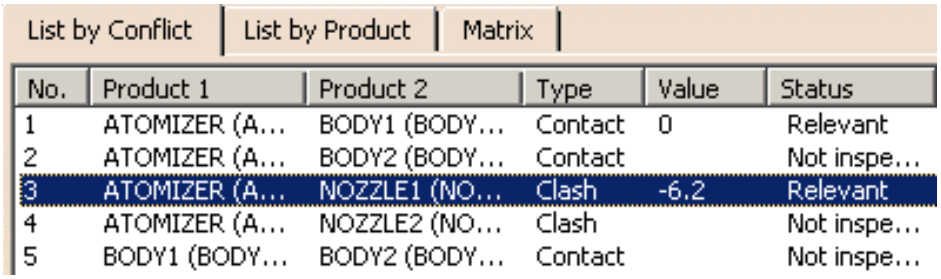
2. Click **Apply** to run the analysis:

The Check Clash dialog box expands to show the global results. 21 interferences have been detected.

The first conflict is selected by default: a contact.



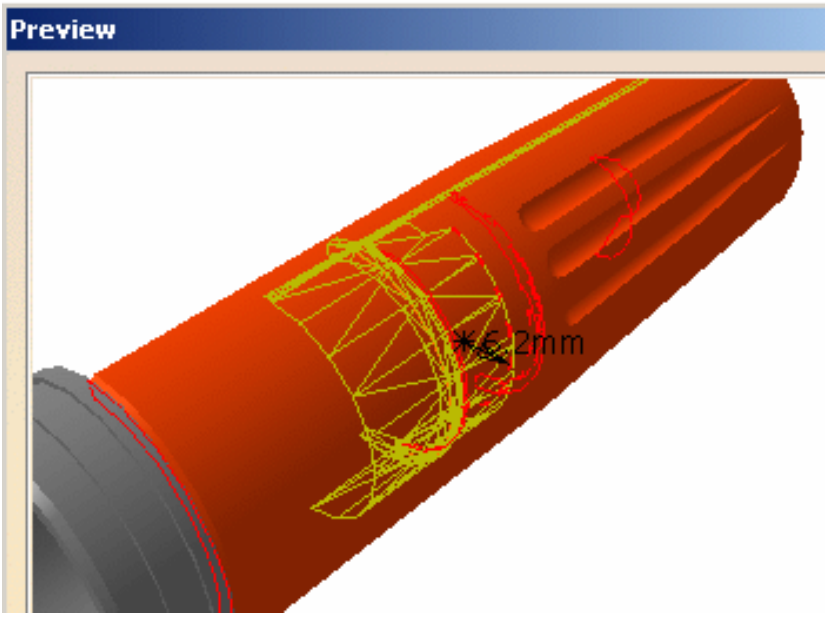
3. Select the first clash conflict in the list: the penetration depth is given.



A Preview window also appears showing the products in the selected conflict.

The clash is identified by red intersection curves, the value of the penetration depth is given and the direction of extraction indicated.

Note: In our example, the graphics representation reveals that the products are also in contact (identified by yellow triangles). The results given in the List by Conflict tab in the dialog box correspond to the worst case.



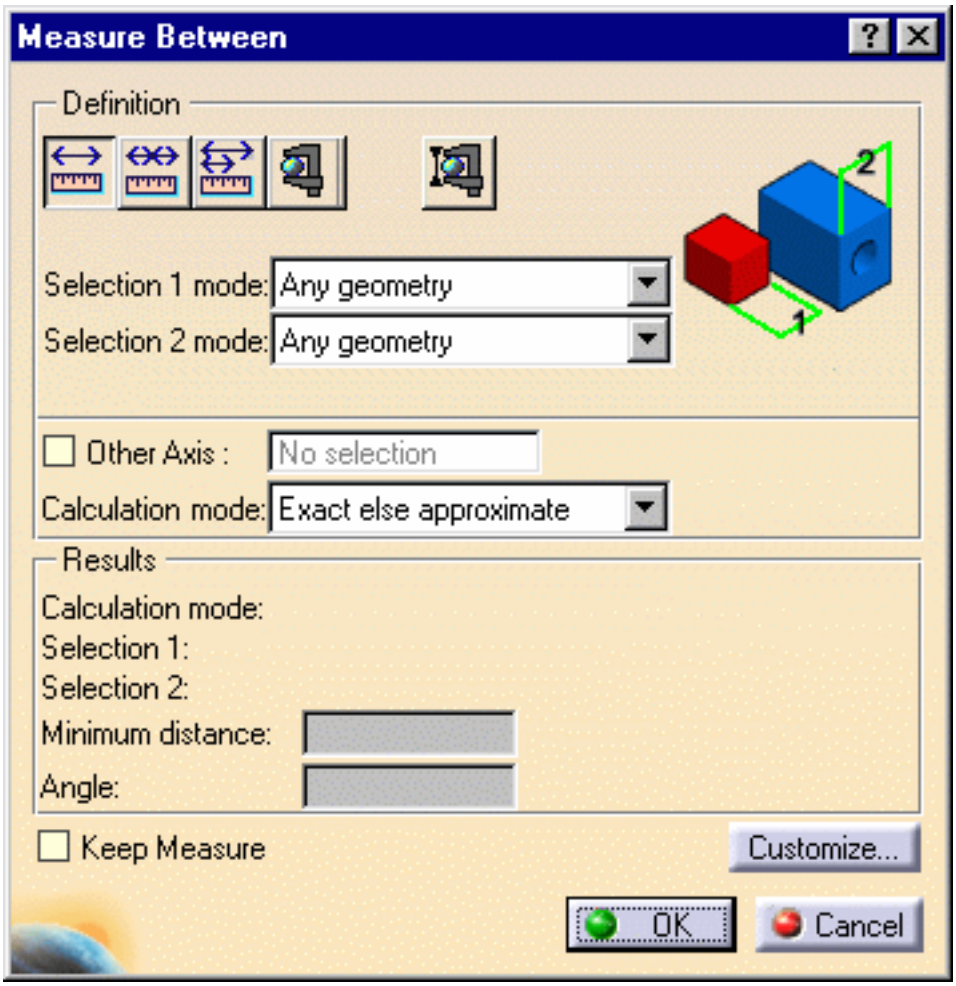
3. Click **OK** when done.

Measuring Distances between Geometrical Entities & Points

This task shows you how to measure distances between geometrical entities and points.

1. Click the Measure Between icon in the DMU Space Analysis toolbar.

   The Measure Between dialog box appears.

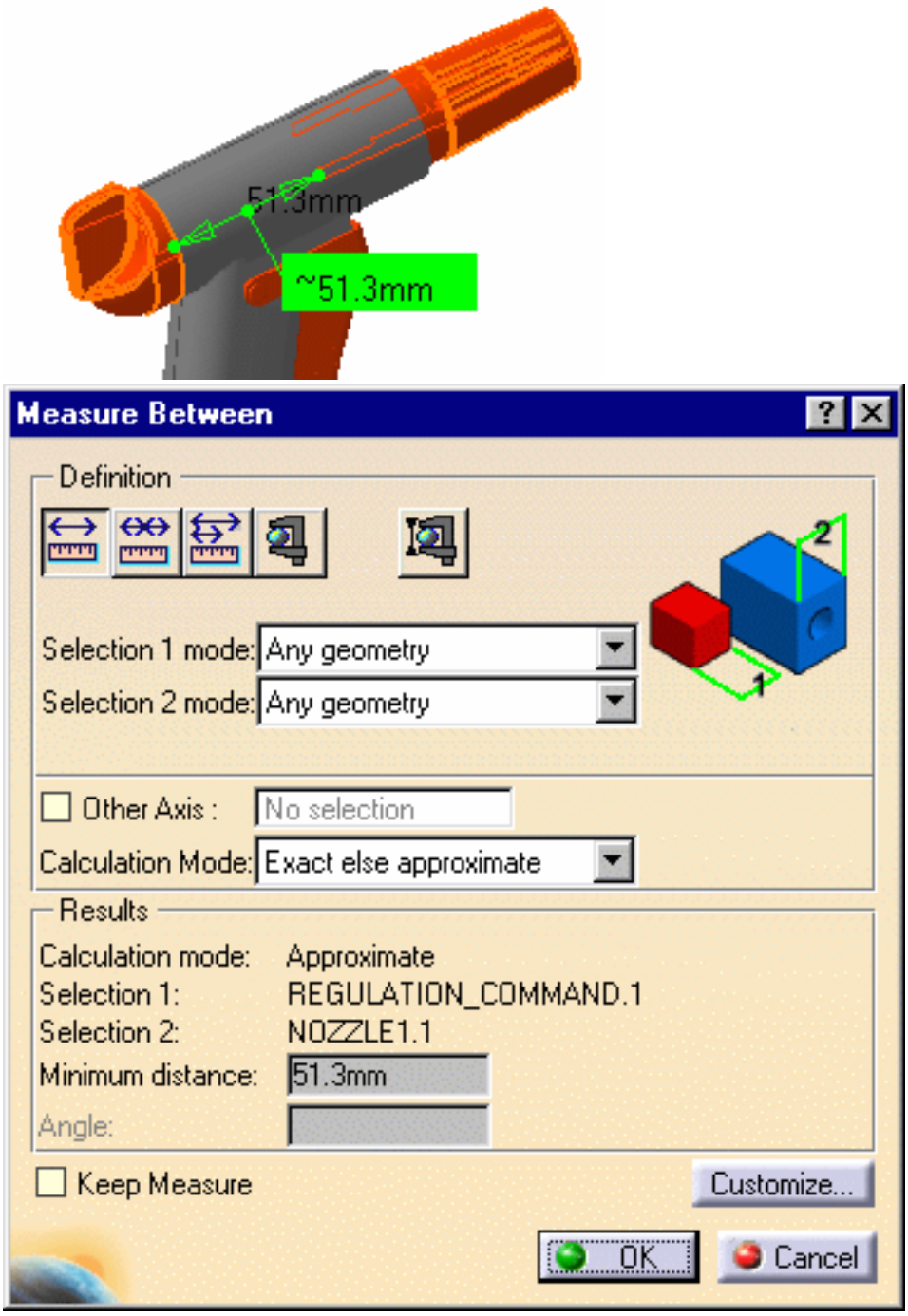


2. Select the Regulation_Command:

   The entity is highlighted in the geometry area.

3. Select one of the nozzles:

   A line representing the minimum distance vector is drawn between the selected items in the geometry area. Approximate distance values are displayed in the dialog box.



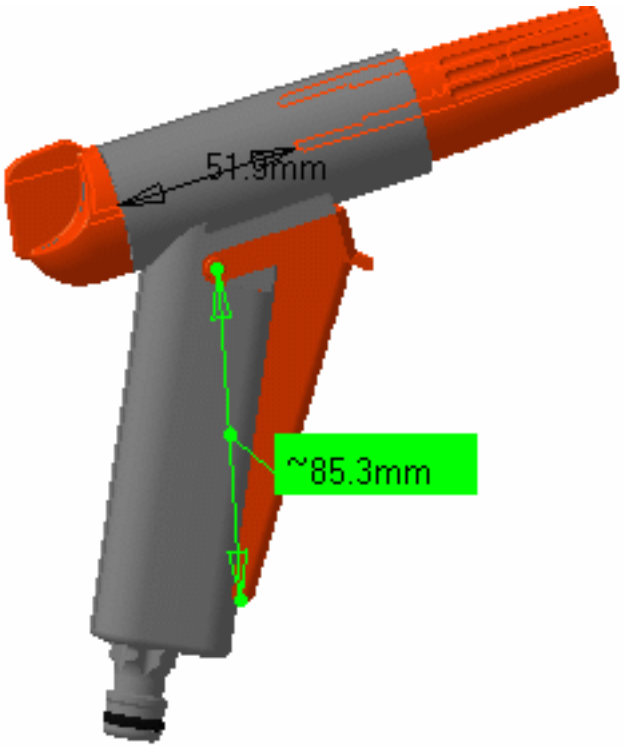
You will now measure distance between two points on geometry.

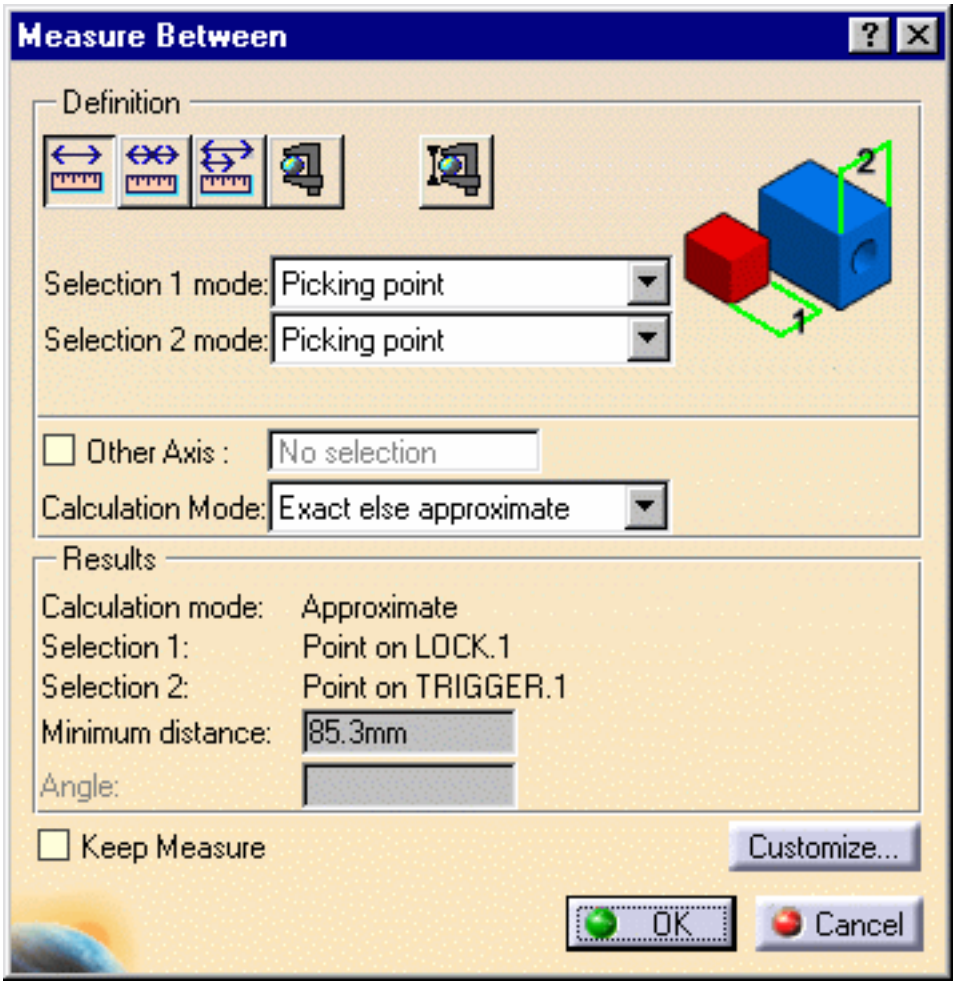
4. Select Picking point in both Selection1 mode and Selection 2 mode drop-down list boxes.
5. Select a first point.

6. Select a second point.

Dynamic highlighting as you move your cursor over geometrical entities helps you locate points.

The results are given in the dialog box.





7. Click **OK** in the Measure Between dialog box when done.

User Tasks

The tasks you will perform in the DMU Space Analysis workbench involve interference analysis, sectioning, distance and band analysis, 3D geometric comparison, exact measurement, and mass and inertia computation.

More advanced tasks cover combining commands, batch processing and knowledgeware capabilities.

Distance & Band Analysis

About distance & band analysis: Gives general information on the Edit Distance and Band Analysis command.

Measure minimum distances and distances along X,Y,and Z: Click the Distance and Band Analysis icon, define the type in the Edit Distance and Band Analysis dialog box, then select products to measure and click Apply.

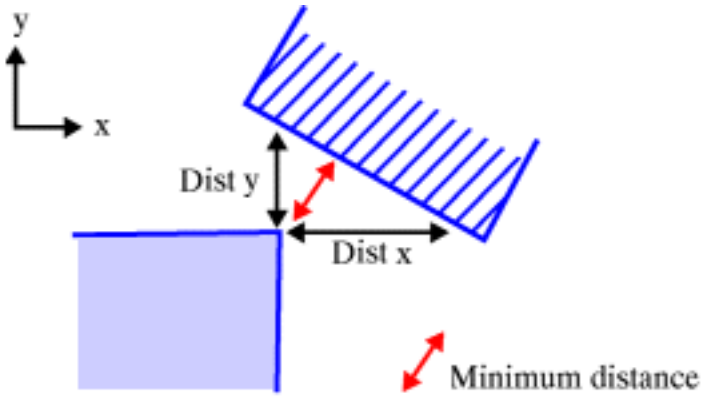
Run band analysis: Click the Distance and Band Analysis icon, measure the minimum distance then select Band Analysis in the Type box. Define the band and set the accuracy then click Apply.

Export & publish distance results: Click the Export As icon and select the desired format.

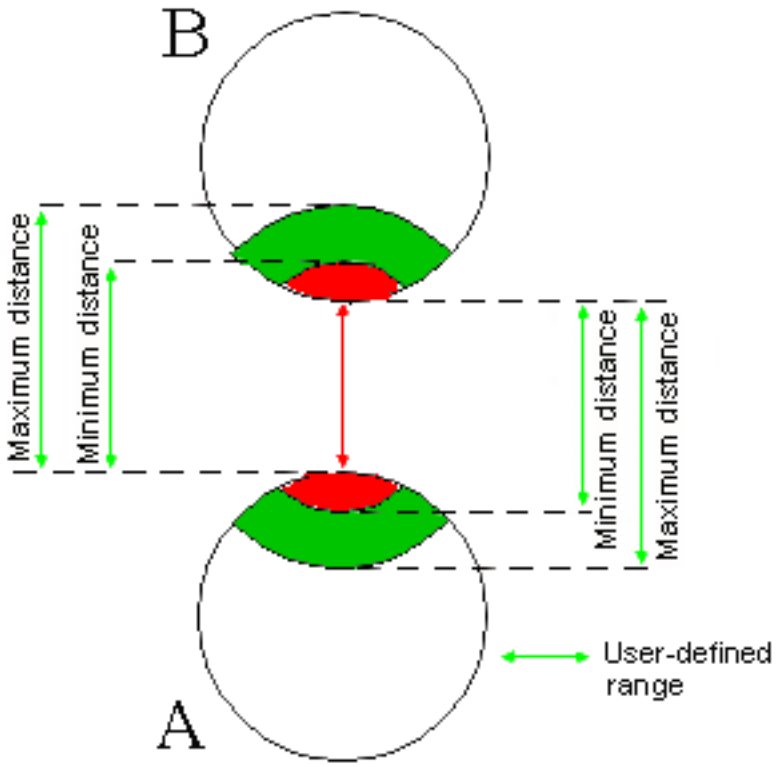
About Distance & Band Analysis

P1 In DMU-P1, you can measure the minimum distance only between products in the same selection, between products in the selection and other products in the document, or between products in two different selections, and view the results in a separate viewer.

P2 In DMU-P2, you can, in addition, measure distance along x, along y and along z.



You can also run a band analysis to compute and visualize the areas on products corresponding to a minimum distance within a user-defined range. For example, you want to know whether there is enough space for hands around a steering wheel (no red area) as well as determine what objects can be reached while keeping hands on the wheel (green area).



The green area on A corresponds to all the points on A for which the minimum distance to B is within the user-defined minimum and maximum distance.

The red area on A corresponds to all the points on A for which the minimum distance to B is less than the user-defined minimum distance.

Distance and Band Analysis Results

Results differ depending on the sag value used.
Sag corresponds to the fixed sag value for calculating tessellation on objects (3D fixed accuracy) set in the Performances tab of **Tools -> Options -> General -> Display.** By default, this value is set to 0.2 mm.
In Visualization mode, you can dynamically change the sag value for selected objects using the **Tools -> Modify SAG** command
Note: sag is offset from the skin inwards.

Associativity

Distance measurements are associative in both DMU-P1 and P2. If you modify one of the products (for example, move it or change the contents of a group), just run the measurement again to obtain the updated results.

Creating Groups of Products

In DMU-P2, prior to running your distance analysis, you can create groups containing the product(s) you want to analyze using the Group  icon in the DMU Space Analysis toolbar or **Insert -> Group...** in the menu bar.

Groups created are identified in the specification tree and can be selected from there for the analysis. Only one group per selection can be defined.

Measuring Minimum Distances & Distances Along X,Y,Z

This task explains how to measure minimum distance and distance along X, along Y and along Z between products.

In this example, you will measure the minimum distance and the distance along z between products in two different selections.

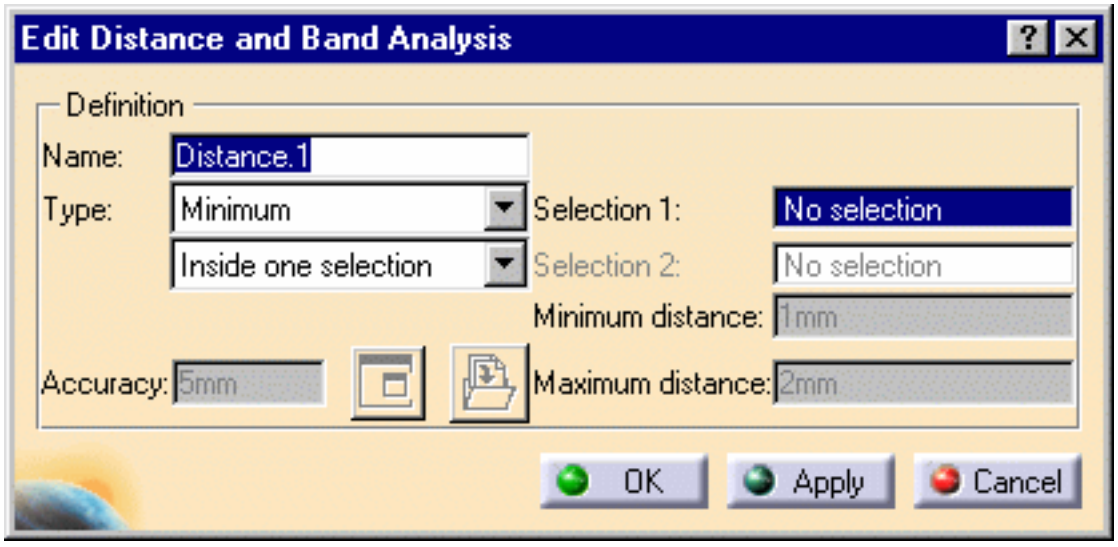
Measuring distances along a direction (X, Y or Z) is a P2 functionality.

Insert the following sample cgr files: ATOMIZER.cgr, BODY1.cgr, BODY2.cgr, LOCK.cgr, NOZZLE1.cgr, NOZZLE2.cgr, REGULATION_COMMAND.cgr, REGULATOR.cgr, TRIGGER.cgr and VALVE.cgr.

They are to be found in the online documentation filetree in the common functionalities sample folder cfysm/samples.

1. Click the Distance and Band Analysis icon in the DMU Space Analysis toolbar, or select **Insert -> Distance and Band Analysis** from the menu bar to calculate distances:

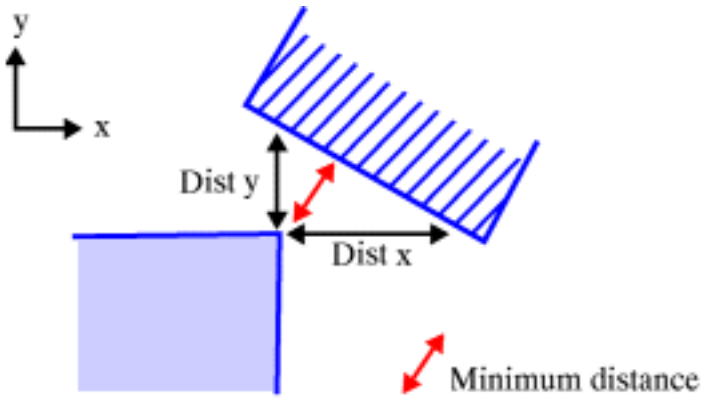
   The Edit Distance and Band Analysis dialog box appears.



   The default distance analysis is measuring the minimum distance inside one selection.

2. Ensure that the first Type drop-down list box is set to Minimum.

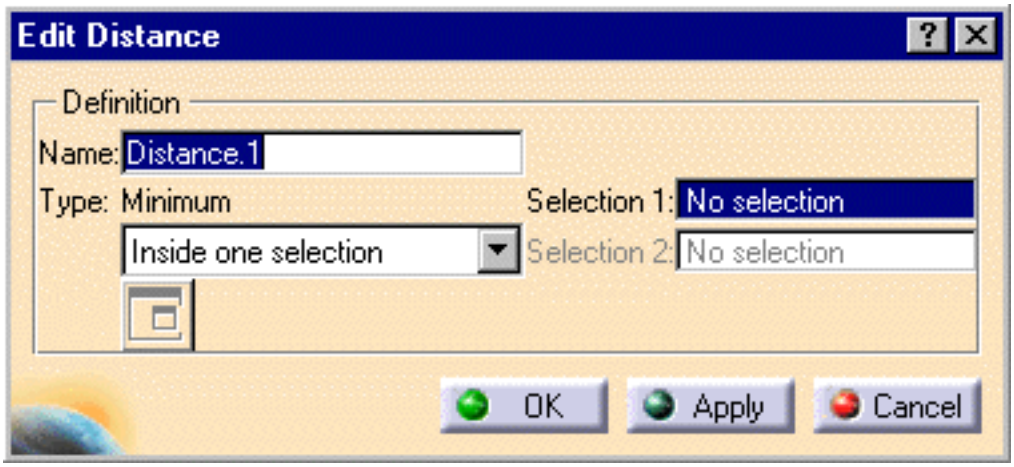
Defining Measurement Types



- Minimum
- Along X (P2 only)
- Along Y (P2 only)
- Along Z (P2 only)
- Band analysis (P2 only)

P1 Functionalities

In DMU-P1, you can measure the minimum distance only between products in the same selection, between products in the selection and other products in the document, or between products in two different selections and view the results in a separate viewer.



3. Select a product, for example the Trigger.
4. Click the second Type drop-down list box and select Between two selections.

Defining Computation Types

- Inside one selection (default type): within any one selection, tests each product of the selection against all other products in the same selection.
- Between two selections: tests each product in the first selection against all products in the second selection.
- Selection against all: tests each product in the defined selection against all other products in the document.

5. Select another product, for example the Regulation_Command.

Notes:

- Any sub-assembly in the specification tree is considered a valid selection.
- Click in selection fields (fields turn dark blue) to view your selections and be certain that you have selected the products you intended: selected products are highlighted.
- Continue clicking to select as many products as you want. Products will be placed in the active selection. To de-select products, reselect them in the specification tree or the geometry area.

6. Click **Apply** to calculate the distance:
A Preview window appears visualizing selected products and the minimum distance (represented by a line, two crosses and a value). The Edit Distance and Band Analysis dialog box expands to show the results.

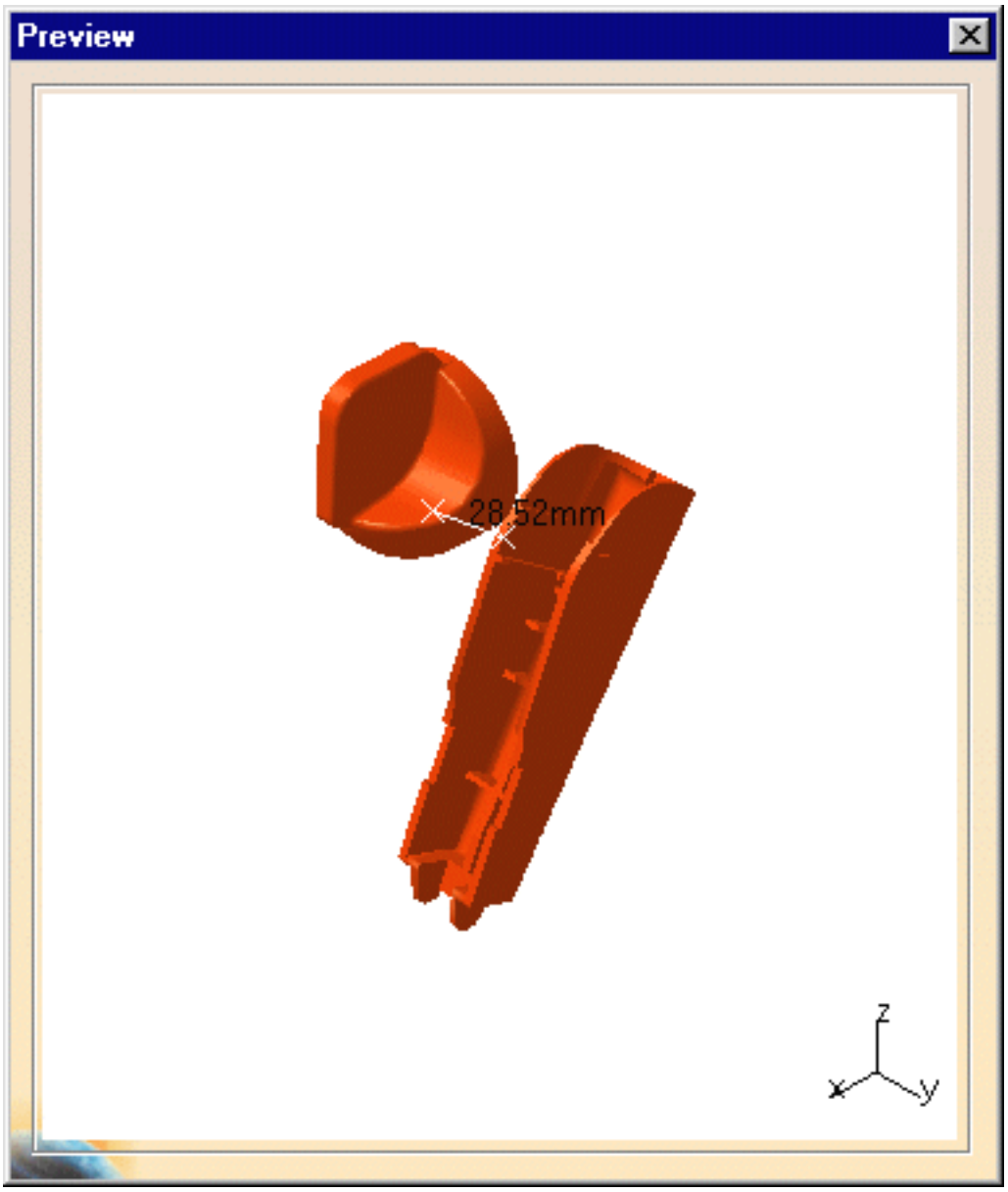
You can change the default display setting for the Preview window using the **Tools ->Options..., Digital Mockup** command (DMU Navigator tab),

If you close the Preview window, to open it again:

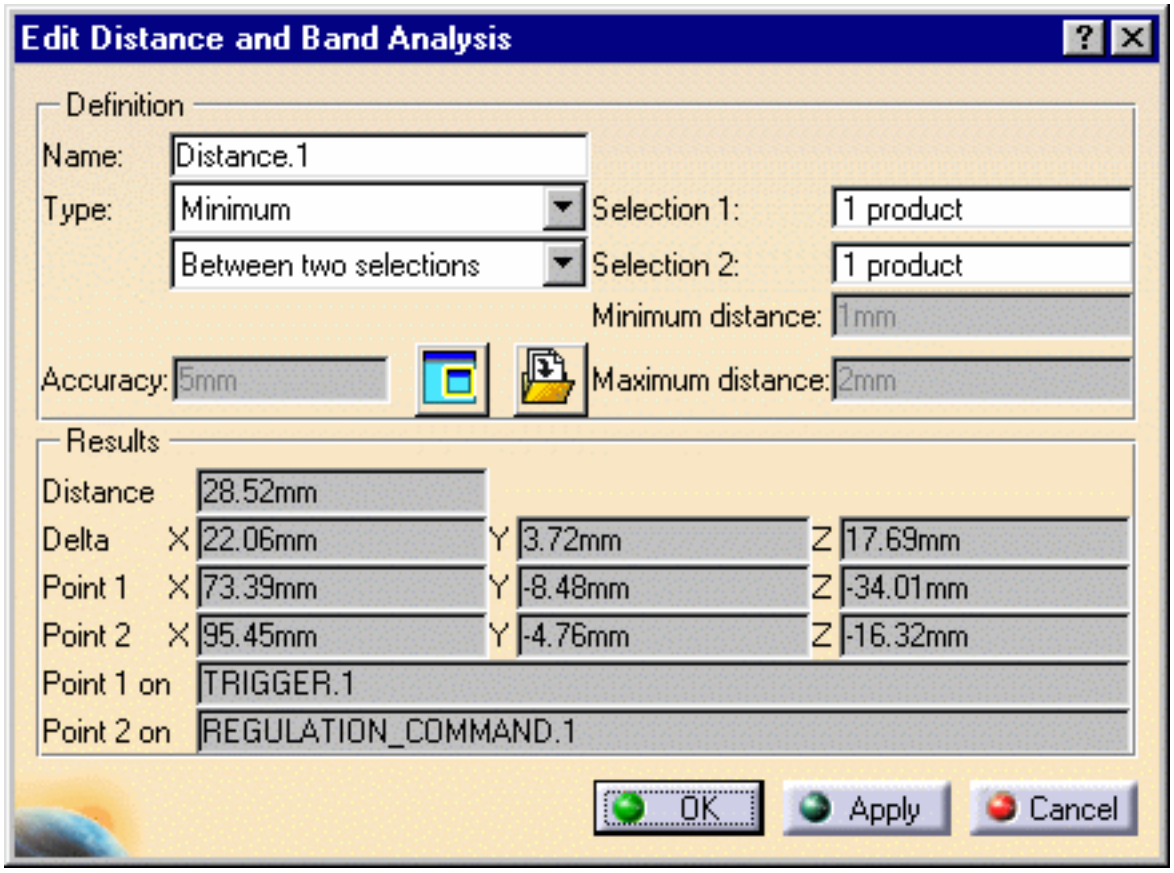
- Click the Results window  icon in the Edit Distance and Band Analysis dialog box
  This opens the dedicated results window.
- Click the Results window  icon a second time
  The results window is closed and the Preview window is restored.

If however, you close the Results window (using the Close button in the title bar), the Preview window will not be restored.

7. If necessary, pan, zoom and/or rotate in the Preview window to visualize the results better.



Minimum distance and other information identifying all distance components is given in the expanded dialog box. X,Y,Z coordinates of start and end points on products selected for the distance calculation as well as products themselves are identified.

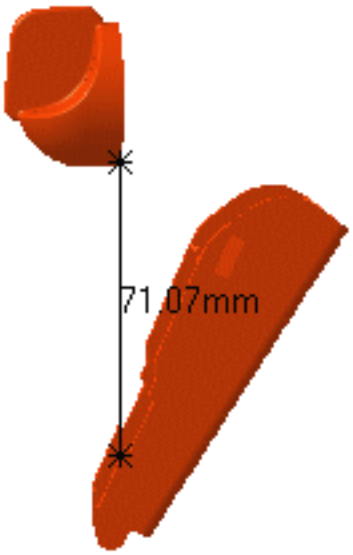


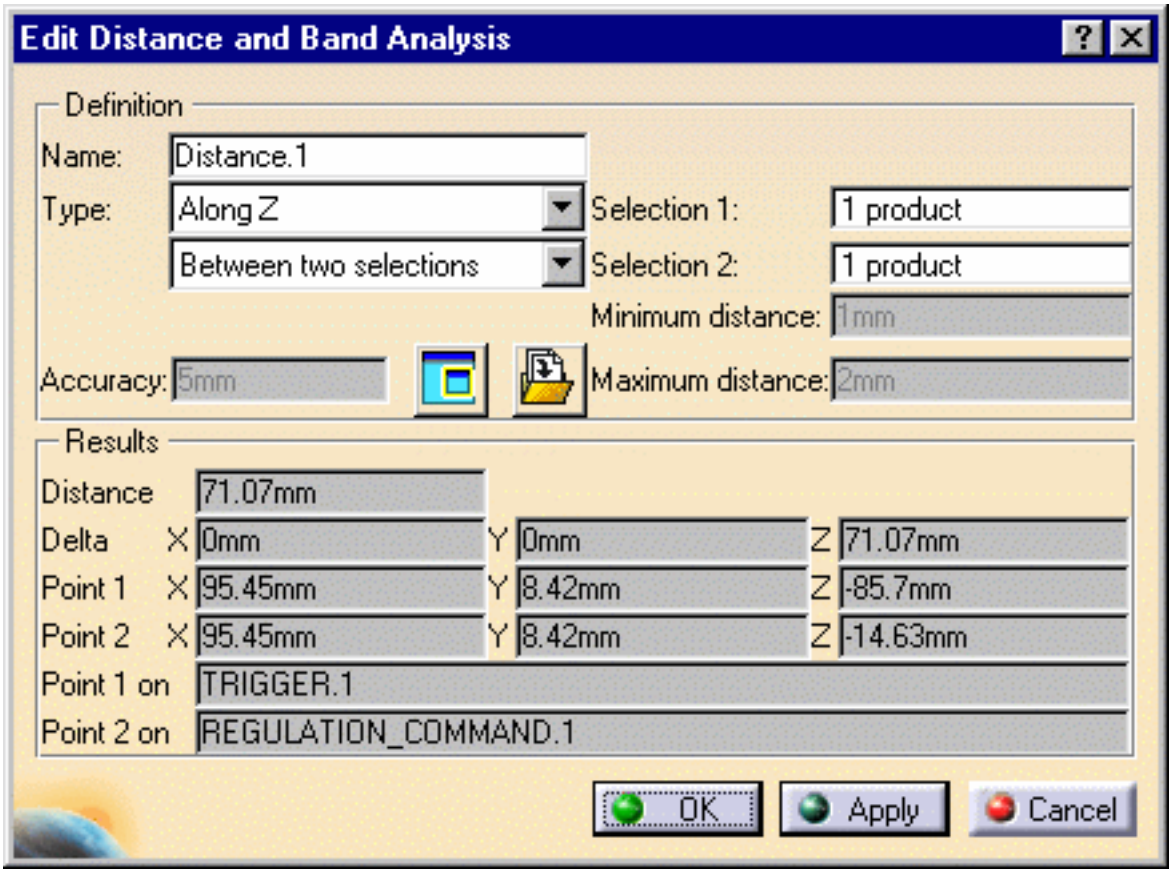
You can also view the results in a separate viewer. To do so, click the Results window  icon in the Edit Distance and Band Analysis dialog box. Object viewing commands and commands in the Window menu are available in this window. For example, using the Windows menu, you can tile the results window and the original document window vertically or horizontally.

You can use other DMU Space Analysis Toolbar commands in the Results window to, for example, measure surfaces.

8. Click the Type drop-down list box and select Along Z.
9. Click **Apply**.

Results are calculated and displayed in the dialog box and Preview window.





10. Click **OK** in the Edit Distance and Band Analysis dialog box when done.

The distance definition and results are kept as a specification tree feature. This means you can run the measurement again after, for example, moving one of the products or modifying the contents of a group: distance results will be updated to reflect changes made.

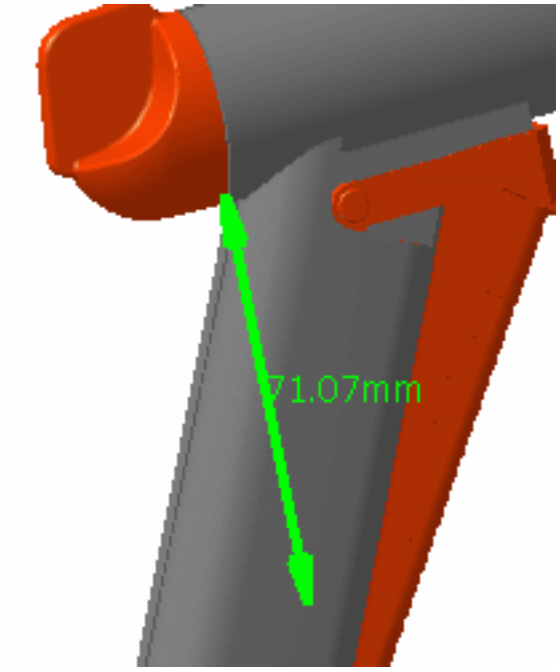
Changing the Color, Linetype and Thickness of the Distance Result

To make it easier to read your result, you can specify different properties for distance result. This is done via the Properties command or via the Graphic Properties toolbar.

The Properties command lets you change the color, linetype and thickness of the current distance result (under Lines and Curves in Graphics tab).

To access properties:

- Right-click the specification tree feature and select Properties.
- In the dialog box, click the Graphic tab.



You can also change these properties via the Graphic Properties toolbar.

To do so:

- Select the feature in the specification tree.
- Set the linetype, thickness and/or plane color in the Graphic Properties toolbar.

To return to the initial colors, select No color.

This command also lets you run a band analysis to compute and visualize the areas on products corresponding to a minimum distance within a user-defined range.

Running Band Analysis

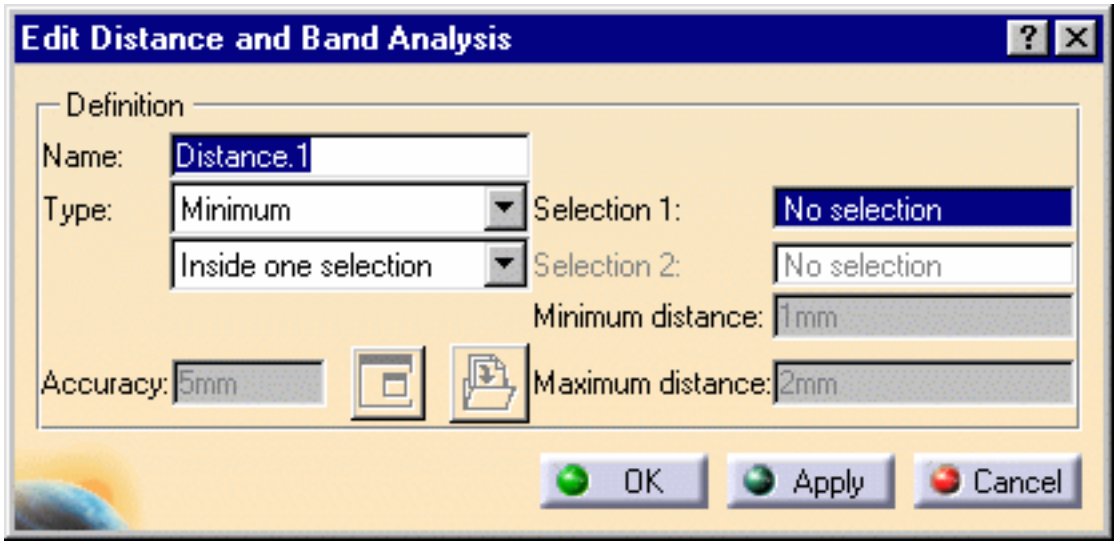
This task explains how to run a band analysis to compute and visualize areas on products corresponding to a minimum distance within a user-defined range.

Insert the following sample cgr files: ATOMIZER.cgr, BODY1.cgr, BODY2.cgr, LOCK.cgr, NOZZLE1.cgr, NOZZLE2.cgr, REGULATION_COMMAND.cgr, REGULATOR.cgr, TRIGGER.cgr and VALVE.cgr.

They are to be found in the online documentation filetree in the common functionalities sample folder cfysm/samples.

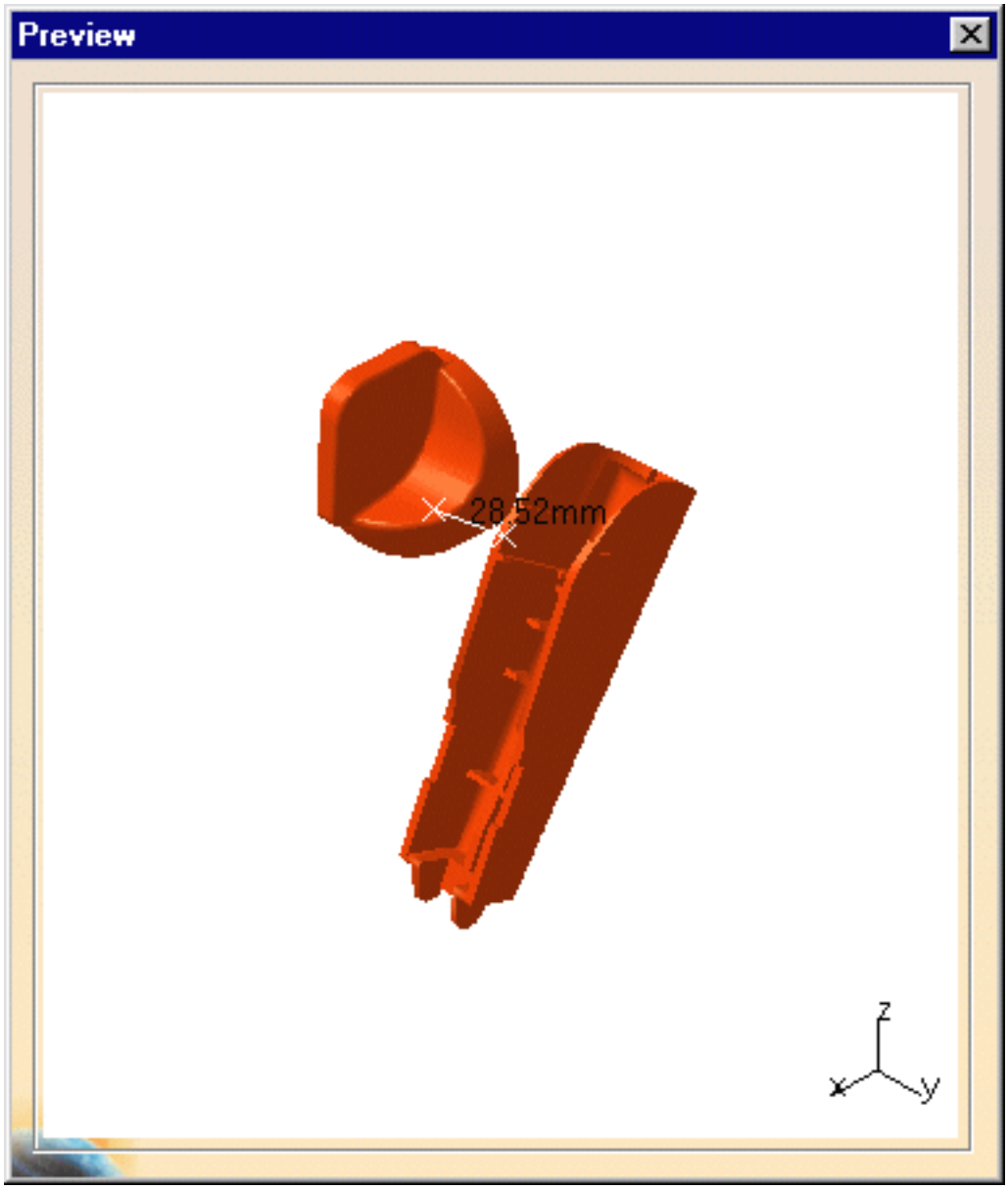
1. Click the Distance and Band Analysis icon in the DMU Space Analysis toolbar, or select **Insert -> Distance and Band Analysis** from the menu bar to calculate distances:

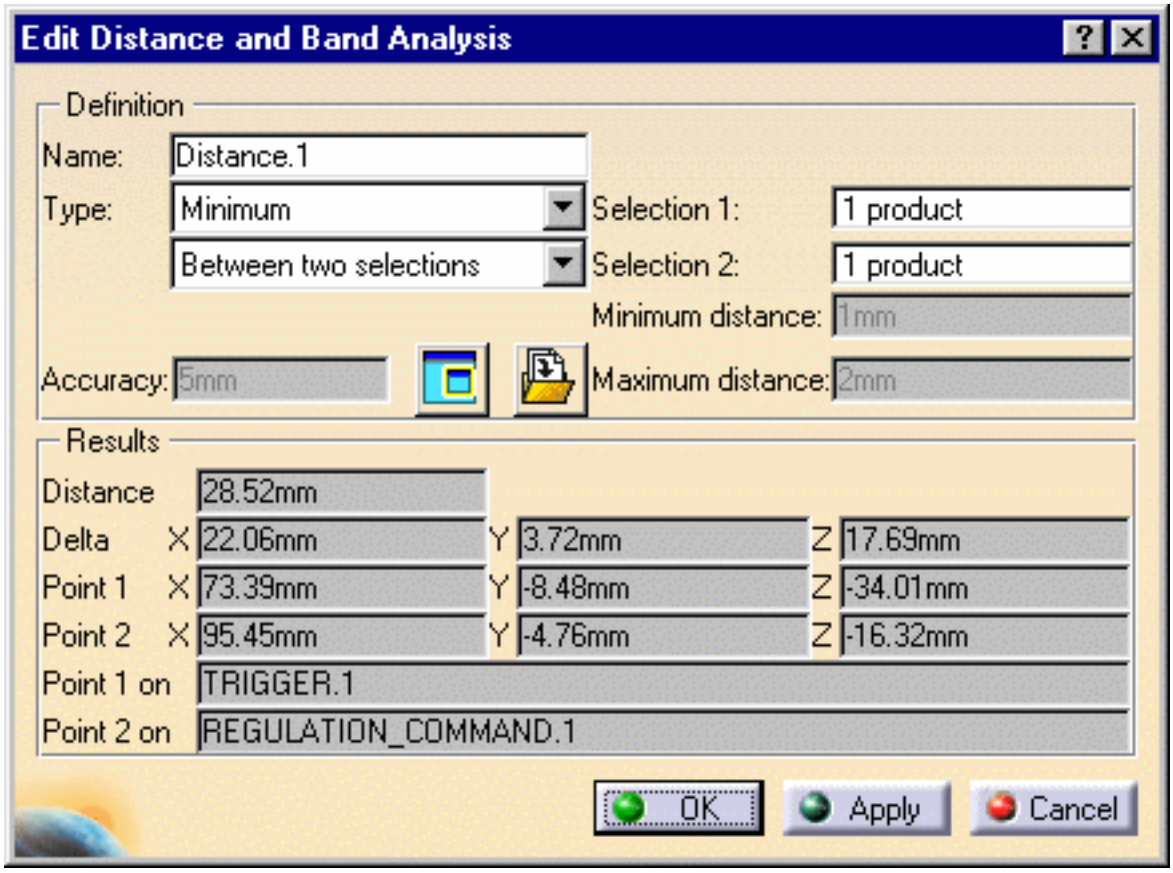
   The Edit Distance and Band Analysis dialog box appears.



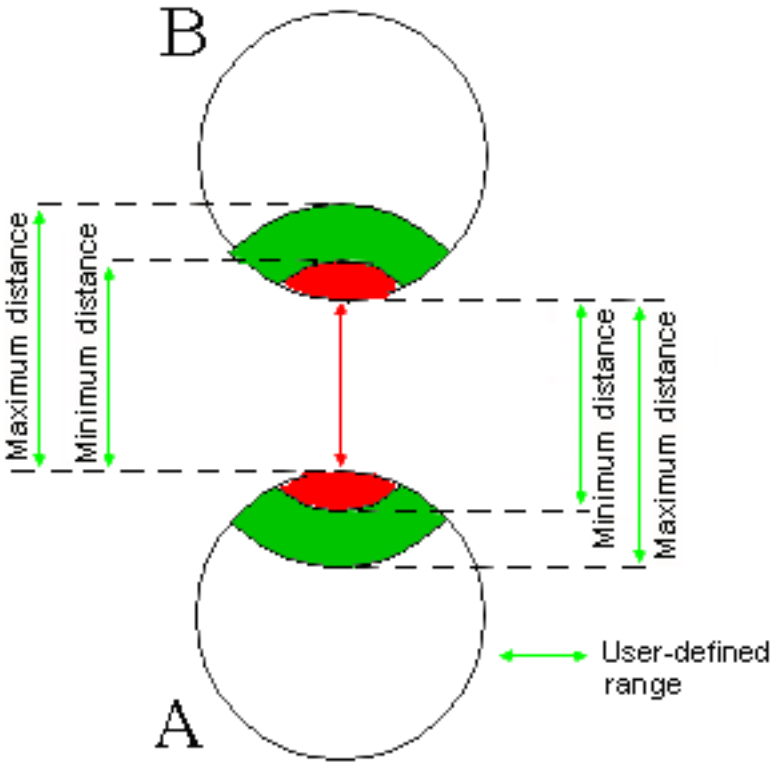
   The default distance analysis is measuring the minimum distance inside one selection.
2. Measure the minimum distance between the Trigger and the Regulation_Command.
3. Click **Apply**.

   A minimum distance of 28.52mm has been found.


Preview
28.52mm
Z
X
Y



4. Click the Type drop-down list box and select Band analysis.



The green area on A corresponds to all the points on A for which the minimum distance to B is within the user-defined minimum and maximum distance.

The red area on A corresponds to all the points on A for which the minimum distance to B is less than the user-defined minimum distance.

5. Set the Minimum and Maximum distance to define the band width, to 32 and 36mm respectively for example.

   Default values are 1 and 2mm respectively.

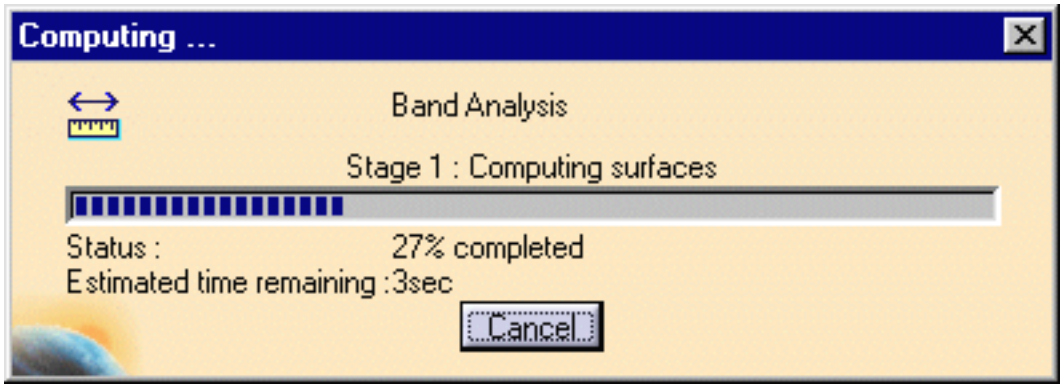
6. If necessary, set accuracy.

   The default value is 5mm. A lower value will give you a more accurate result.

   The value entered defines the maximum value for the length of the longest side of a triangular representation of the results. This representation is used to obtain the red and green surfaces.

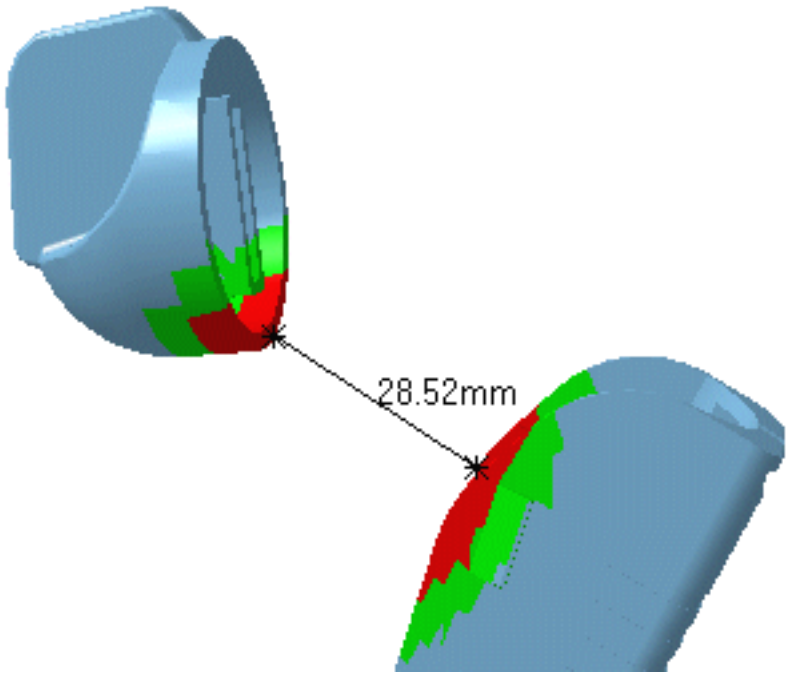
7. Click **Apply**.

   A progress bar is displayed letting you monitor and, if necessary, interrupt (**Cancel** option) the calculation.



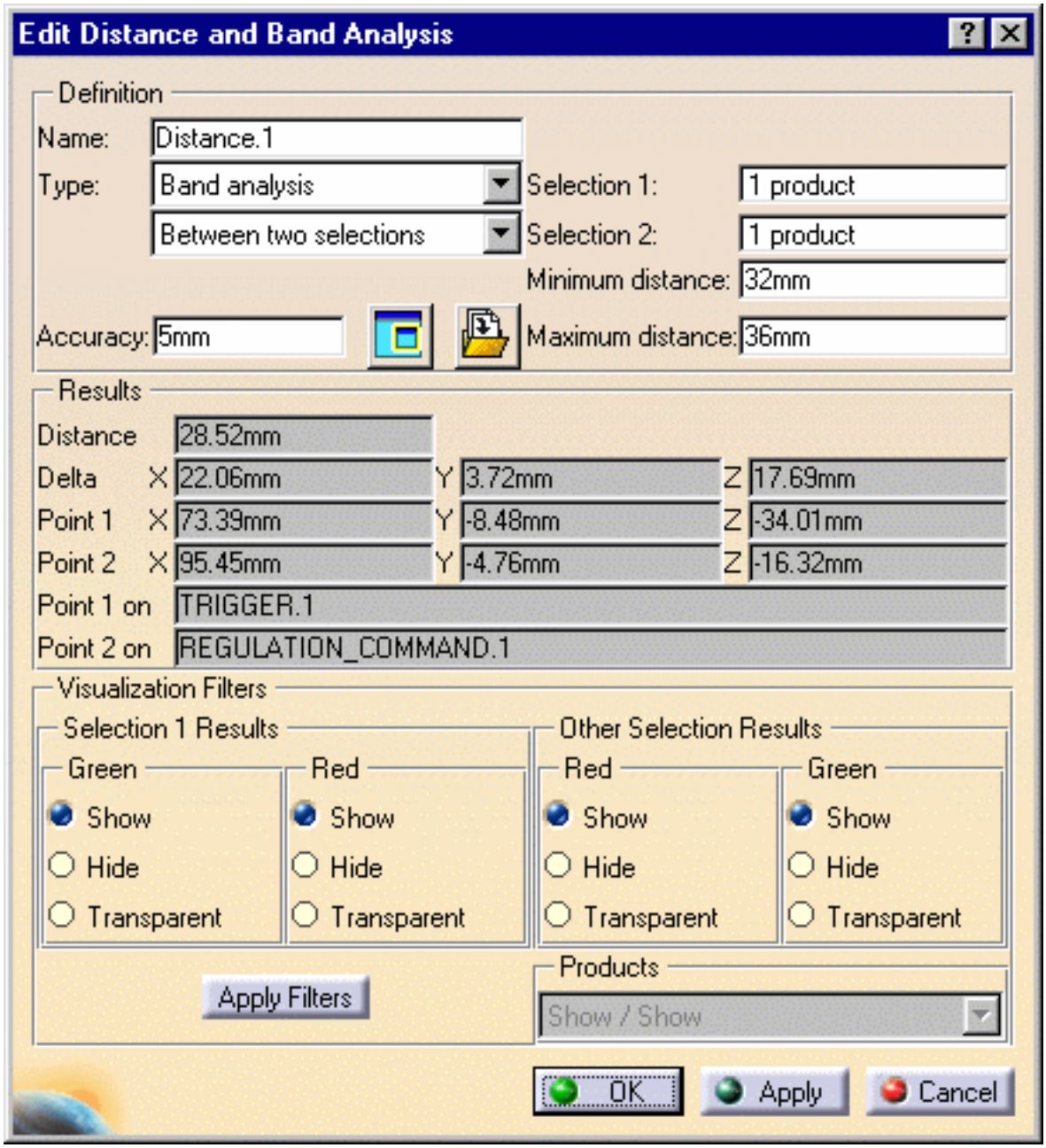
Green surfaces identify the areas where the minimum distance between the products is within the specified range.

Red surfaces identify the areas where the minimum distance between the products is less than the specified minimum distance.



The Edit Distance and Band Analysis dialog box expands to include filters letting you better visualize the green and red surfaces.

The Export As  icon letting you save band analysis results in a variety of different formats becomes available.



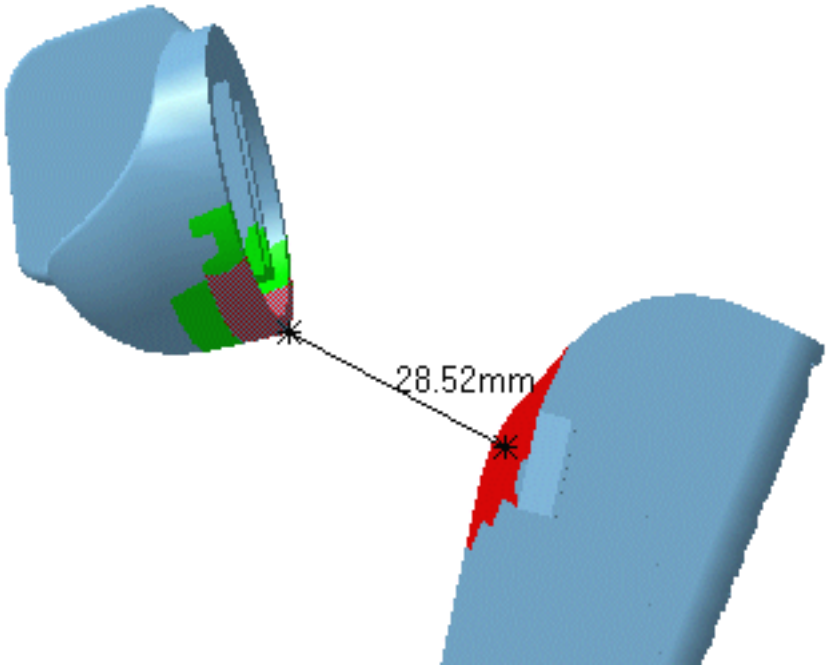
8. Set the appropriate options to show, hide or make transparent the green and red surfaces corresponding to band analysis results for components in selection 1.

9. Repeat for other components, i.e. those measured against selection 1.

10. Click Apply Filters to visualize the results.

In the Inside one selection computation type, visualization filters are valid where two products only are selected.

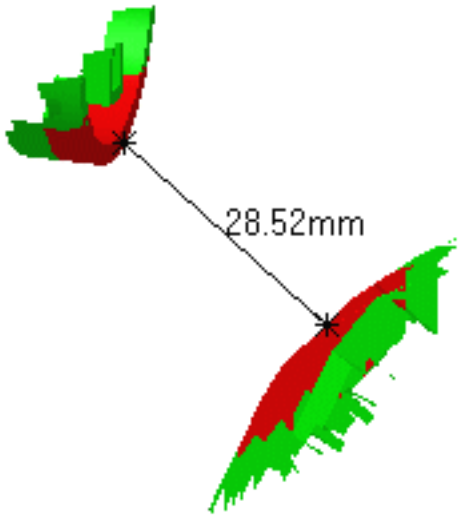
In our example, we set Selection 1 (the Trigger) results in green to Hide and Other Selection (the Regulation_Command) results in red to Transparent.



A Products drop-down box lets you choose whether or not to display the products in either or both selections, or make them transparent. This option is only available in the results window.

11. Close the Preview window and open a dedicated results window.

12. Set the drop-down box to Hide/Hide to visualize the results of the band analysis only.



13. Click **OK** in the Edit Distance and Band Analysis dialog box when done.

The band analysis definition and results are kept as a specification tree feature. This means you can run the measurement again after, for example, moving one of the products or modifying the contents of a group: distance results will be updated to reflect changes made.

Exporting & Publishing Distance Results

Using the Export As command, you can save band analysis results in a variety of different formats.

This task explains how to:

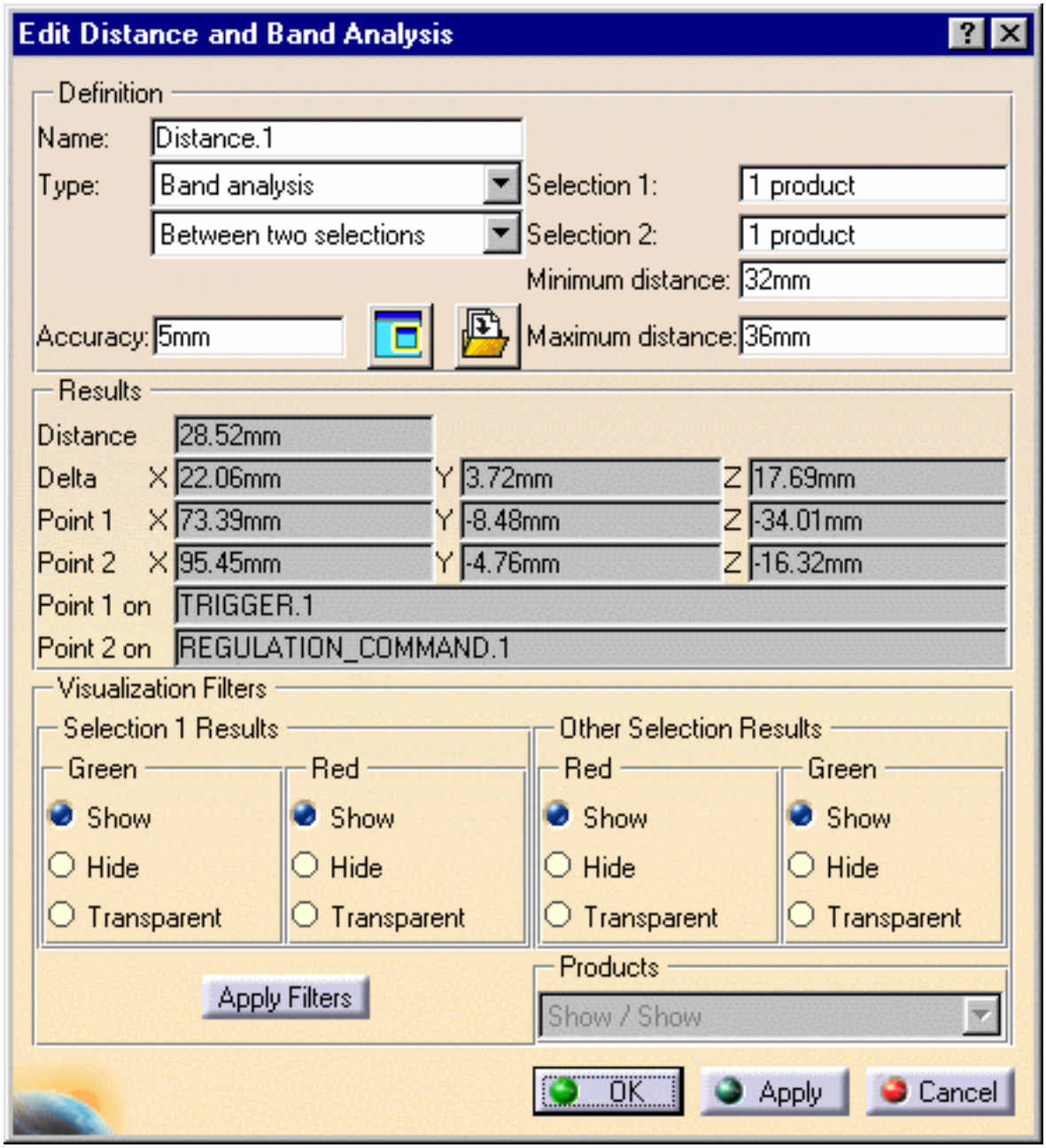
- Save surfaces as a model document for re-use in a CATIA V4 design context
- Export results in CGR and VRML (Virtual Reality Modeling Language) format
- Publish results to an XML file.

Insert the following sample cgr files: ATOMIZER.cgr, BODY1.cgr, BODY2.cgr, LOCK.cgr, NOZZLE1.cgr, NOZZLE2.cgr, REGULATION_COMMAND.cgr, REGULATOR.cgr, TRIGGER.cgr and VALVE.cgr.

They are to be found in the online documentation filetree in the common functionalities sample folder cfysm/samples.

1. Click the Distance and Band Analysis  icon in the DMU Space Analysis toolbar, or select **Insert -> Distance and Band Analysis** from the menu bar to run a band analysis. The Edit Distance and Band Analysis dialog box expands.

   The Export As  icon letting you export and publish band analysis results becomes available.



2. Click the Export As icon:

The Save As dialog box appears.

3. Specify the location of the document to be saved and, if necessary, enter a file name.

Note: If you set the DLName document environment (**Tools -> Options -> General -> Document**) as your current environment, clicking Export As will open the DLName dialog box instead of the usual Save As dialog box.

The DLName document environment lets you restrict the access to specific folders referenced by logical names referred to as 'DLNames'. Each folder is assigned a logical name. In this mode, you can only access documents in folders referenced by DLNames.

For more information on the DLName document environment, see the *Infrastructure User's Guide*.

4. Click the Save as type drop-down list and select the desired format:
   - model: to export to a V4 model.
     Surfaces exported can be re-used in a CATIA V4 design context.
     Each of the four surfaces is saved as a polyhedral surface (*POL) in the model document.

     Note: The green and red color coding of exported surfaces is kept.

   - cgr: to save results in a CGR file.
   - wrl: to save results as a VRML document.
   - xml: to publish results to an XML file.
     Each time you export results to an XML file, a folder containing all the necessary files and images is created.
5. Click Save to save the results in a file in the desired format.

Browsing the XML file

Using the **Tools ->Options..., Digital Mockup** command (DMU Space Analysis ->DMU Distance tab), you can:

- Specify the default location of the XML file
- Indicate the style sheet used, and
- Have the browser open automatically.

To have your browser open automatically showing the exported results, ensure your browser is already open on your desktop.

Sectioning

About sectioning: Gives general information about the **Sectioning** command.

Create section planes: Click the **Sectioning** icon.

Change section graphic properties: Gives information about changing line segment color, linetype, and thickness, as well as plane color.

Create section slices: Create a section plane then click the **Section Slice** icon in the Sectioning Definition dialog box.

Create section boxes: Create a section plane then click the **Section Box** icon in the Sectioning Definition dialog box.

More about the Section viewer: Create a section plane.

Create 3D section cuts: Create a section plane then click the **Volume Cut** icon.

Manipulate planes directly: Create a section plane, drag plane edges to re-dimension, drag plane to move it along the normal vector, press and hold left and middle mouse buttons down to move plane in U,V plane of local axis system or drag plane axis to rotate plane.

Position planes using the Edit Position and Dimensions command: Create a section plane, click the **Edit Position** icon and enter parameters defining the plane position in the dialog box.

Position planes on a geometric target: Create a section plane, click the **Geometrical Target** icon and point to the target of interest.

Snap boxes to planes: Create a section box, click the **Geometrical Target** icon and select two or three planes.

Snap planes to points and/or lines: Create a section plane, click the **Positioning by 2/3 Selections** icon and make your selections.

Export section results: Generate section results then click the **Export As** icon to export to a V4 model, V5 CATPart, IGES or VRML document.

Capture section results: Generate section results then select **Tools ->Image ->Capture**

Annote generated sections: Gives information about annotating using generic measure tools and 3D and 2D annotation tools.

Manage the update of section results: Generate section results, then select appropriate option in Behavior tab and exit command.

More about the contextual menu: Right-click the section feature or section in geometry area and select the command from the menu.

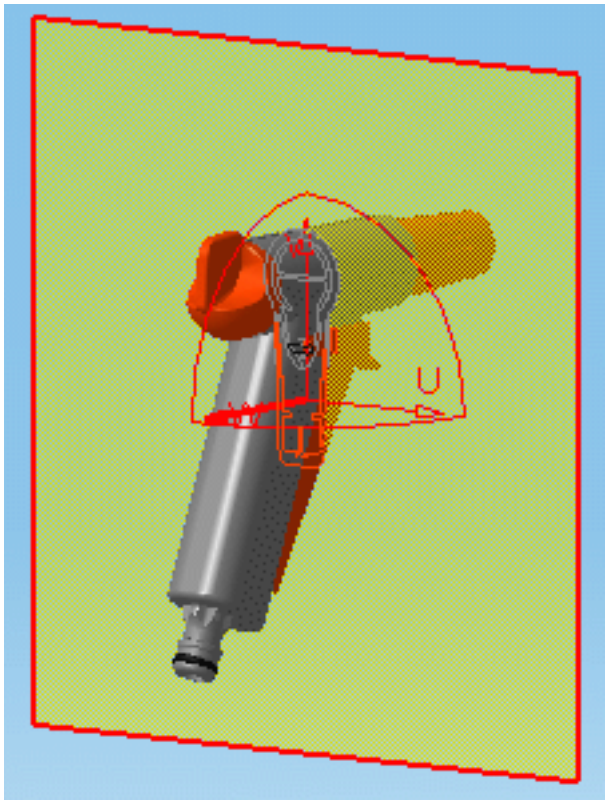
About Sectioning

Using cutting planes, you can create sections, section slices, section boxes as well as 3D section cuts of your products automatically.

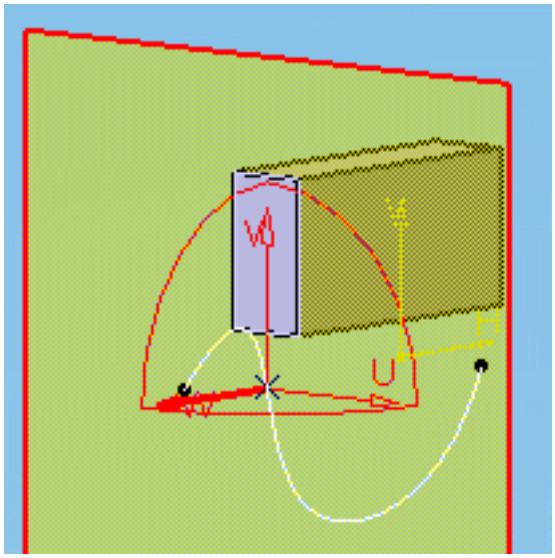
Creating section slices and section boxes are DMU-P2 functionalities.

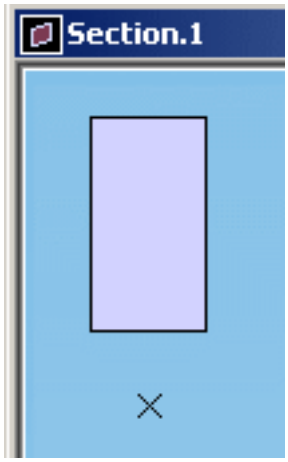
The Section Plane

The section plane is created parallel to absolute coordinates Y, Z. The center of the plane is located at the center of the bounding sphere around the products in the selection you defined.



- Line segments represent the intersection of the plane with all surfaces and volumes in the selection. By default, line segments are the same color as the products sectioned.
- Points represent the intersection of the plane with any wireframe elements in the selection, and are visible in both the document window and the Section viewer.

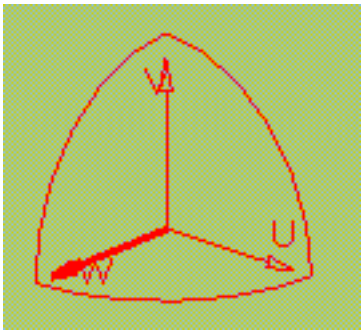




Notes:

- Any surfaces or wireframe elements in the same plane as the section plane are not visible.
- If no selection is made before entering the command, the plane sections all products.

In DMU-P1, you cannot select products to be sectioned: the plane sections all products.



A plane has limits and its own local axis system. The letters U, V and W represent the axes. The W-axis is the normal vector of the plane.

You can customize settings to locate the center and orient the normal vector of the plane as well as de-activate the default setting taking wireframe elements into account. This is done using the **Tools ->Options..., Digital Mockup ->DMU Space Analysis** command (DMU Sectioning tab).

Manipulating the Plane

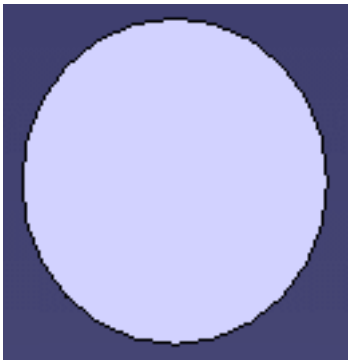
Sectioning is dynamic (moving the plane gives immediate results). You can manipulate the cutting plane in a variety of ways:

- Directly
- Position it with respect to a geometrical target, by selecting points and/or lines
- Change its current position, move and rotate it using the Edit Position and Dimensions command.

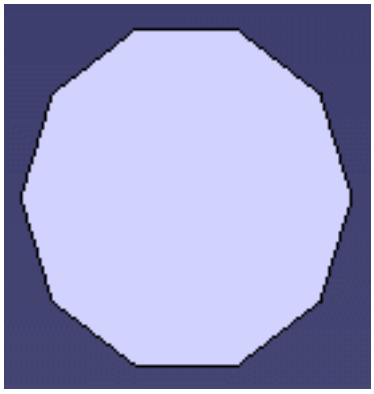
Section Results

Results differ depending on the sag value used.

Using default value (0.2mm):



Using a higher value:



Sag corresponds to the fixed sag value for calculating tessellation on objects (3D fixed accuracy) set in the Performance tab of **Tools -> Options -> General -> Display**. By default, this value is set to 0.2 mm.
In Visualization mode, you can dynamically change the sag value for selected objects using the **Tools -> Modify SAG** command.

The 3D Section Cut

3D section cuts cut away the material from the cutting plane to expose the cavity within the product, beyond the slice or outside the box.

Creating Groups of Products

In DMU-P2, prior to creating your section plane, you can create a group containing the product(s) of interest using the Group  icon in the DMU Space Analysis toolbar or **Insert -> Group...** in the menu bar.

Groups created are identified in the specification tree and can be selected from there for sectioning. Only one group per selection can be defined.

Creating Section Planes

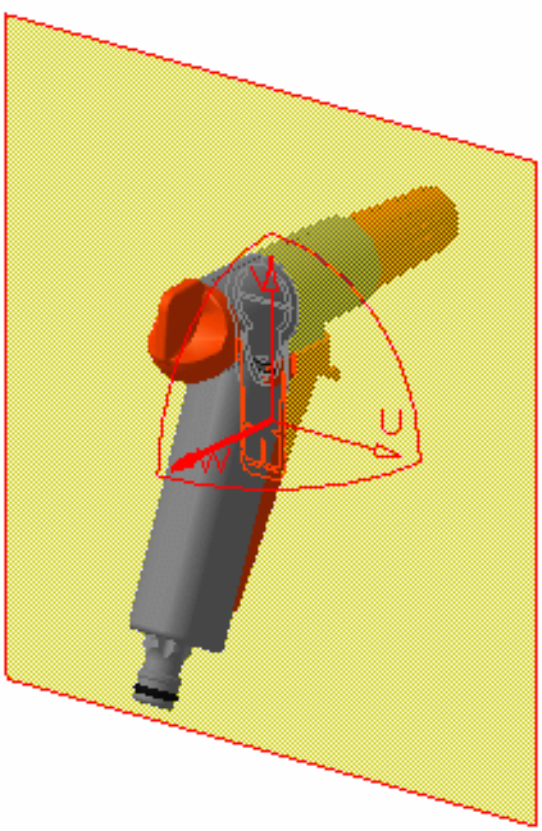
This task shows how to create section planes and orient the normal vector of the plane.

Insert the following cgr files: ATOMIZER.cgr, BODY1.cgr, BODY2.cgr, LOCK.cgr, NOZZLE1.cgr, NOZZLE2.cgr, REGULATION_COMMAND.cgr, REGULATOR.cgr, TRIGGER.cgr and VALVE.cgr.

They are to be found in the online documentation filetree in the common functionalities sample folder **cfysm/samples**.

1. Select **Insert -> Sectioning** from the menu bar, or click the Sectioning icon in the DMU Space Analysis toolbar to generate a section plane.

The section plane is automatically created. If no selection is made before entering the command, the plane sections all products. If products are selected, the plane sections selected products.



P1 Functionality

In DMU-P1, you cannot select products to be sectioned: the plane sections all products.

The Section Plane

The plane is created parallel to absolute coordinates Y,Z. The center of the plane is located at the center of the bounding sphere around the products in the selection you defined.

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

A section plane has limits and its own local axis system. U, V and W represent the axes. The W-axis is the normal vector of the plane. The contour of the plane is red.

You can dynamically re-dimension and reposition the section plane. For more information, see Manipulating Section Planes Directly.

Using the **Tools ->Options...** command (DMU Sectioning tab under **Digital Mockup ->DMU**

Space Analysis, you can change the following default settings:

- Location of the center of the plane
- Orientation of the normal vector of the plane
- Sectioning of wireframe elements.

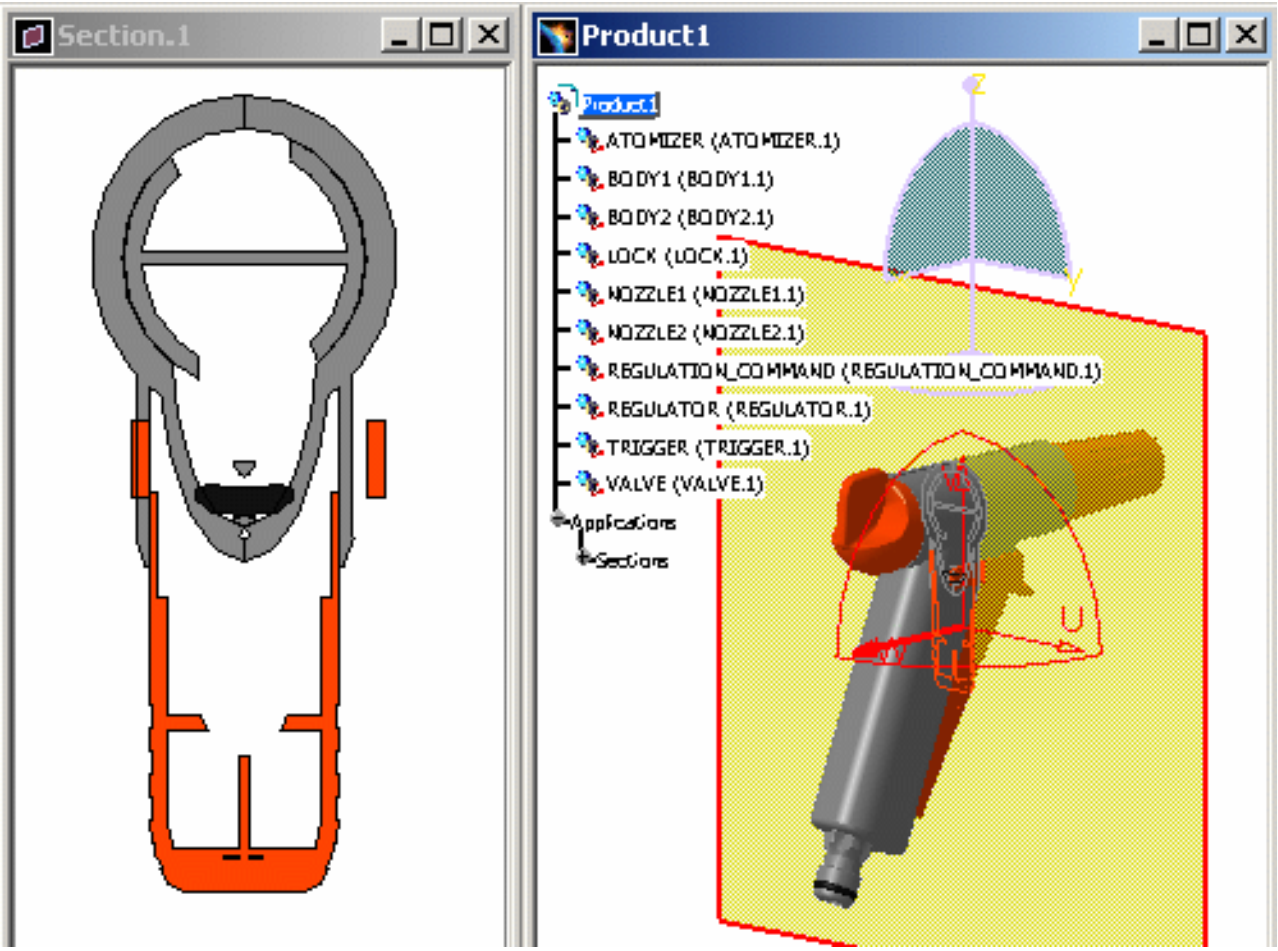
P2 Functionalities

In DMU-P2, you can create as many independent section planes as you like.

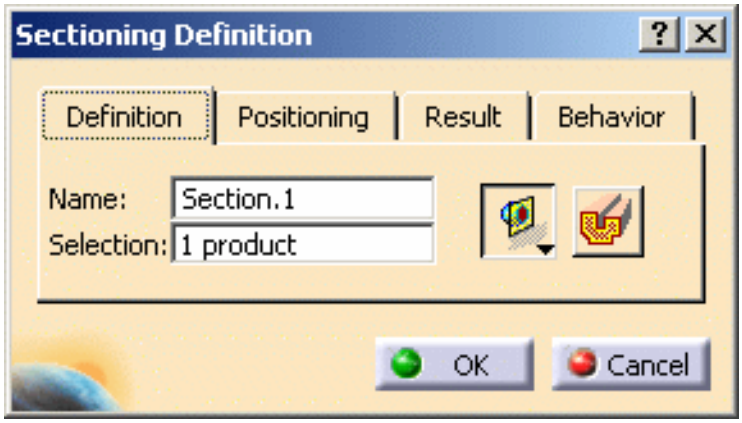
Results Window

A Section viewer is automatically tiled vertically alongside the document window. It displays a front view of the generated section and is by default, locked in a 2D view.

Notice that the section view is a filled view. This is the default option. The fill capability generates surfaces for display and measurement purposes (area, center of gravity, etc.).



Sectioning Definition Dialog Box



The Sectioning Definition dialog box appears.

This dialog box contains a wide variety of tools letting you position, move and rotate the section plane as well as create slices, boxes and section cuts. For more information, see Positioning Planes with respect to a Geometrical Target, Positioning Planes Using the Edit Position Command, Creating Section Slices, Creating Section Boxes and Creating 3D Section Cuts.

P2 Functionalities

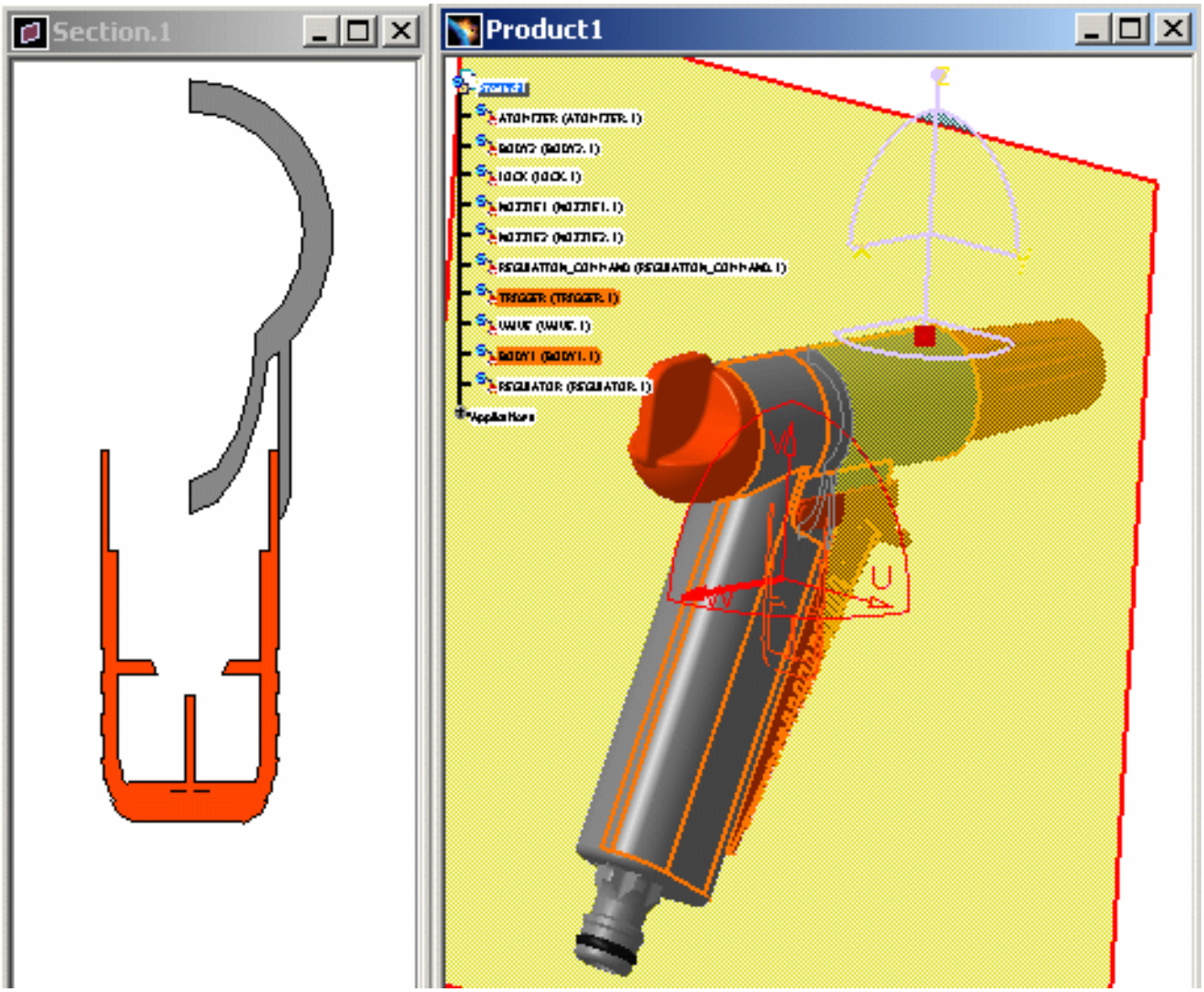
Creating section slices and section boxes are DMU-P2 functionalities.

2. Click the Selection box to activate it.
3. Click products of interest to make your selection, for example the TRIGGER and BODY1.

Products selected are highlighted in the specification tree and geometry area.

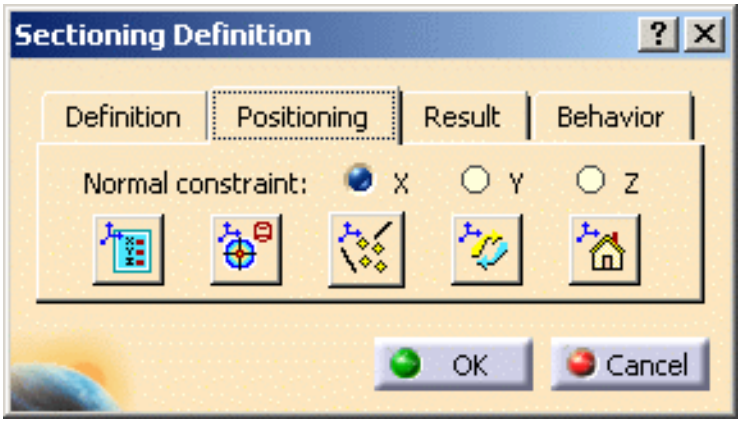
Note: Simply continue clicking to select as many products as you want. Products will be placed in the active selection. To de-select products, reselect them in the specification tree or in the geometry area.

The plane now sections only selected products.



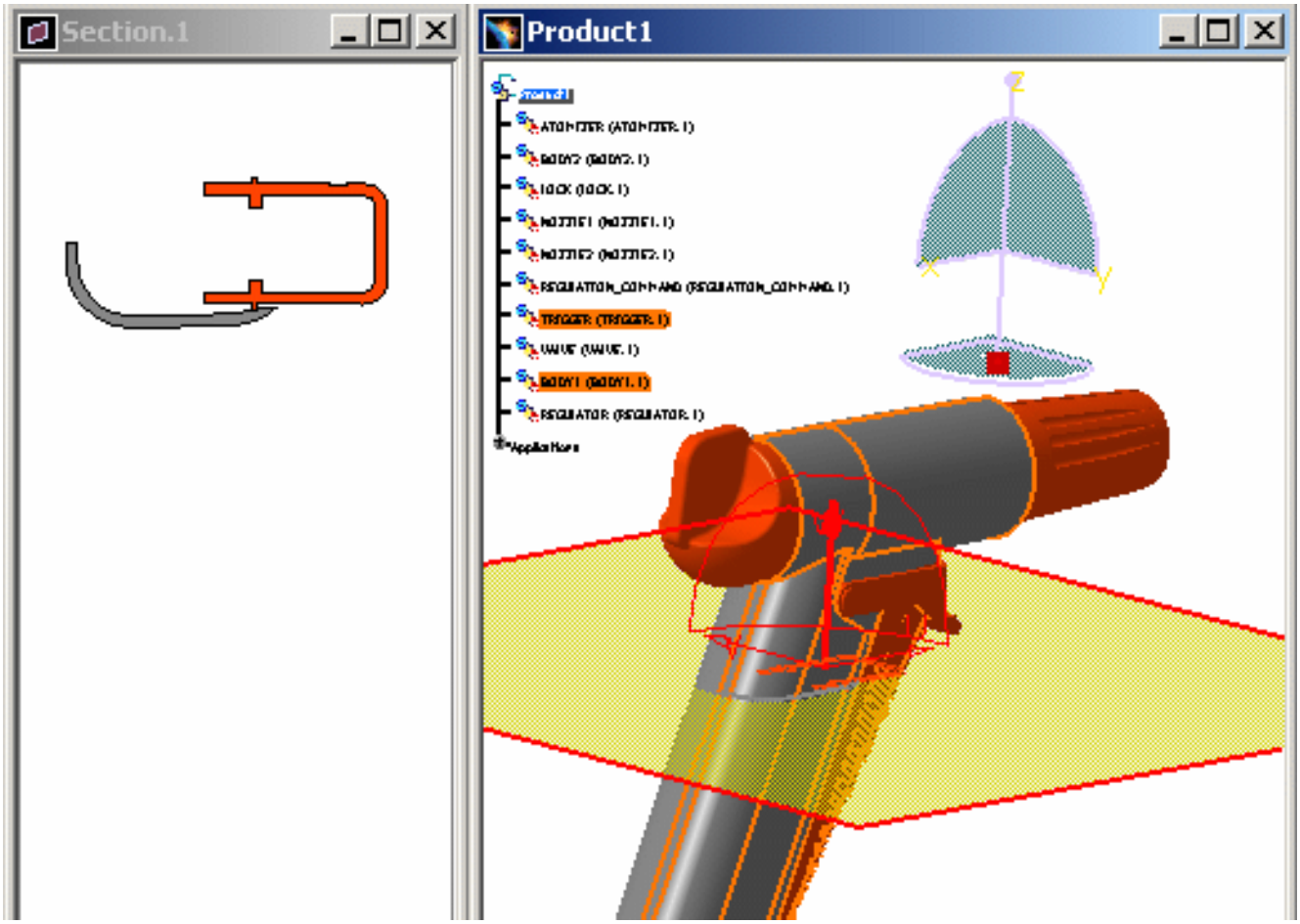
You can change the current position of the section plane with respect to the absolute axis system of the document:

4. Click the Positioning tab in the Sectioning Definition dialog box.

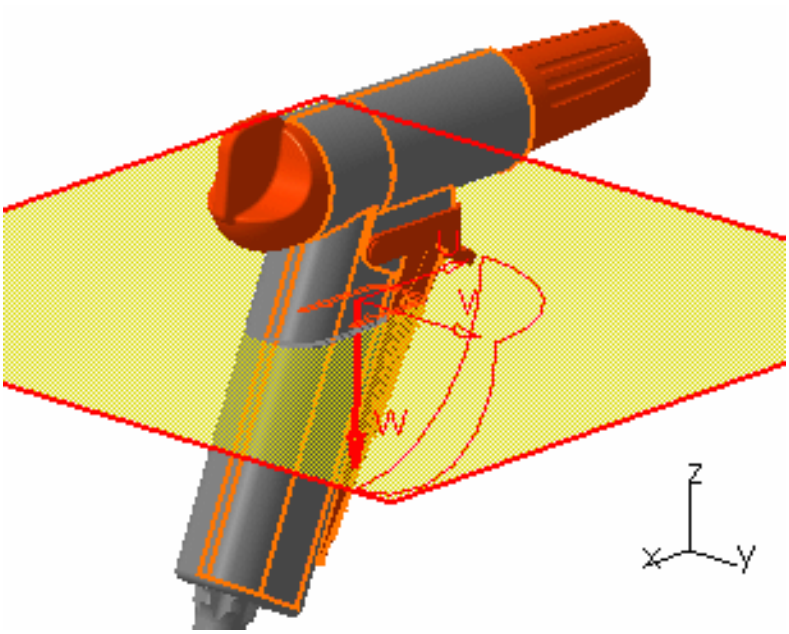


5. Select X, Y or Z radio buttons to position the normal vector (W-axis) of the plane along the selected absolute system axis.

Select Z for example. The plane is positioned perpendicular to the Z-axis.



6. Double-click the normal vector of the plane (W-axis) or click the Invert Normal  icon to invert it.



7. Click **OK** when done.

The section plane definition and results are kept as a specification tree feature.

By default, the plane is hidden when exiting the command. Use the **Tools->Options, Digital Mockup-> DMU Space Analysis** command (DMU Sectioning tab) to change this setting.

To show and edit the plane again, double-click the specification tree feature or select Hide/Show the plane representation in the contextual menu.

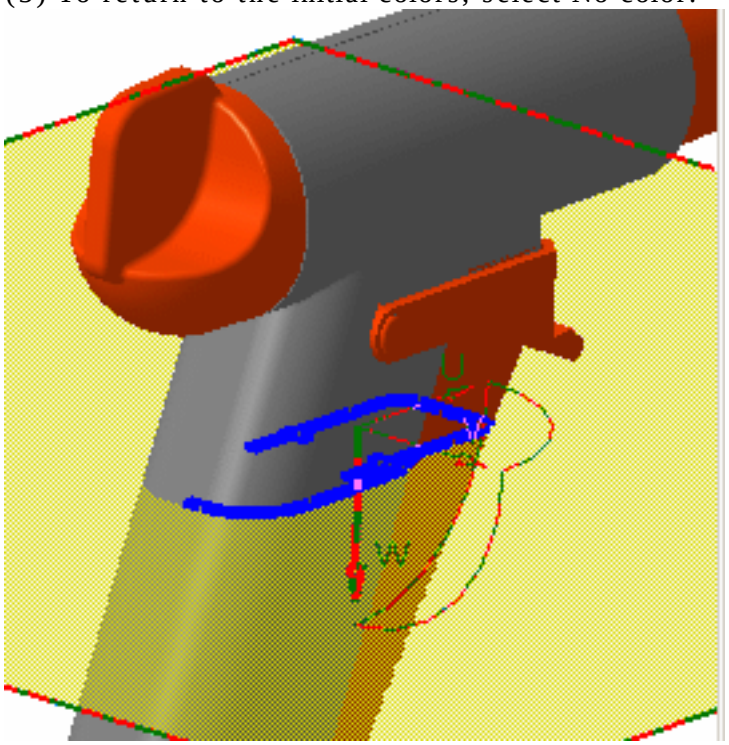
Changing Section Graphic Properties

To make it easier to read your result, you can:

- Specify different properties (color, linetype and thickness) for section line segments
  By default line segments are the same color as the products sectioned.
- Change the plane color of the current feature.

You can change...	Via Properties command - Graphic tab	Via Graphic Properties toolbar[3]
Line segment:		
• Color	• Yes[1] (Lines and Curves)	• No
• Linetype	• Yes[1] (Lines and Curves)	• Yes
• Thickness	• Yes[1] (Lines and Curves)	• Yes
Plane:		
• Color	• Yes[2] (Fill Color)	• Yes

Legend:
(1) To change the line segment color, linetype and thickness, right-click the section in the geometry area and select Properties
(2) To change the plane color, right-click the specification tree feature and select Properties
(3) To return to the initial colors, select No color.



Creating Section Slices

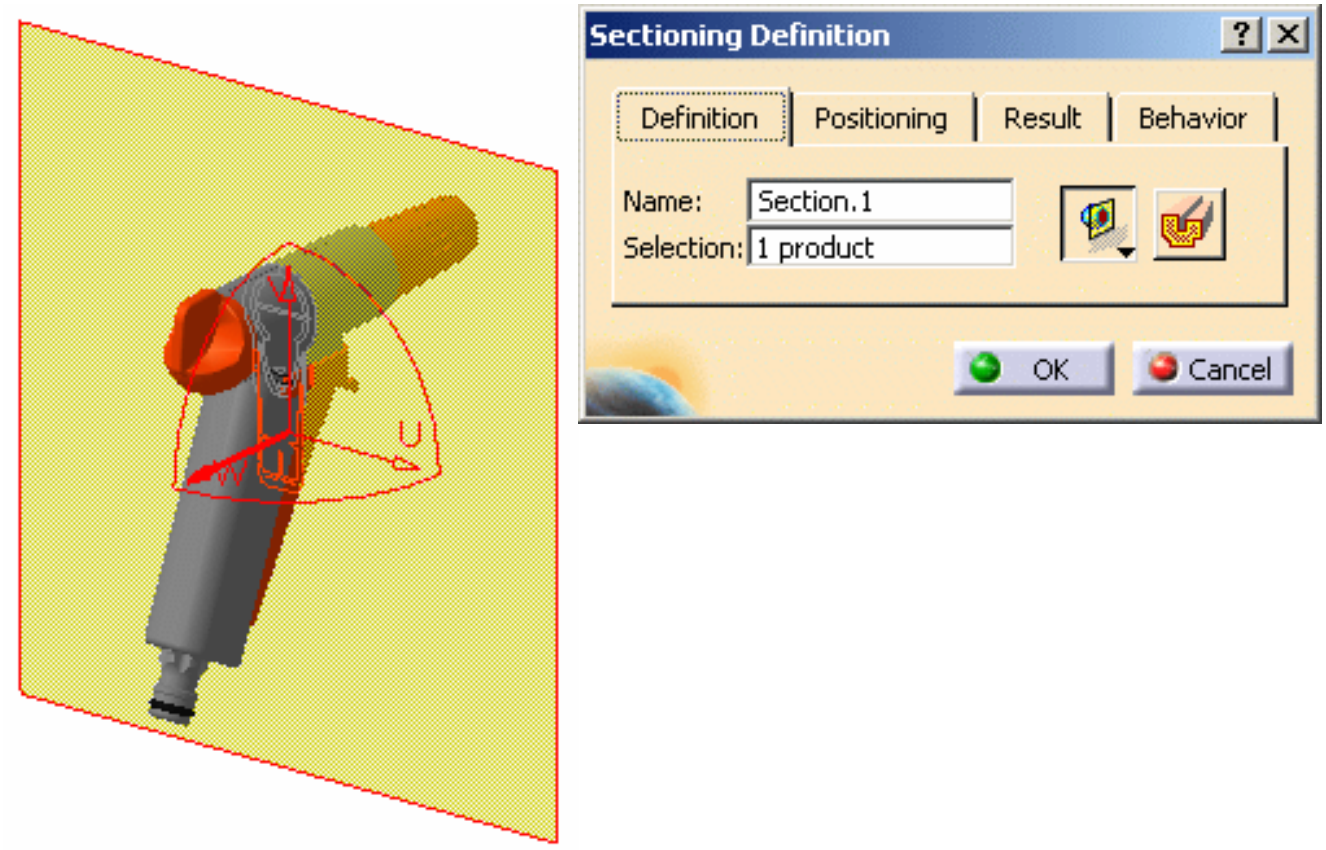
This task explains how to create section slices. To do so, you must first create the master section plane.

Insert the following cgr files: ATOMIZER.cgr, BODY1.cgr, BODY2.cgr, LOCK.cgr, NOZZLE1.cgr, NOZZLE2.cgr, REGULATION_COMMAND.cgr, REGULATOR.cgr, TRIGGER.cgr and VALVE.cgr.

They are to be found in the online documentation filetree in the common functionalities sample folder **cfysm/samples**.

1. Select **Insert -> Sectioning** from the menu bar, or click the Sectioning icon in the DMU Space Analysis toolbar to create a section plane.

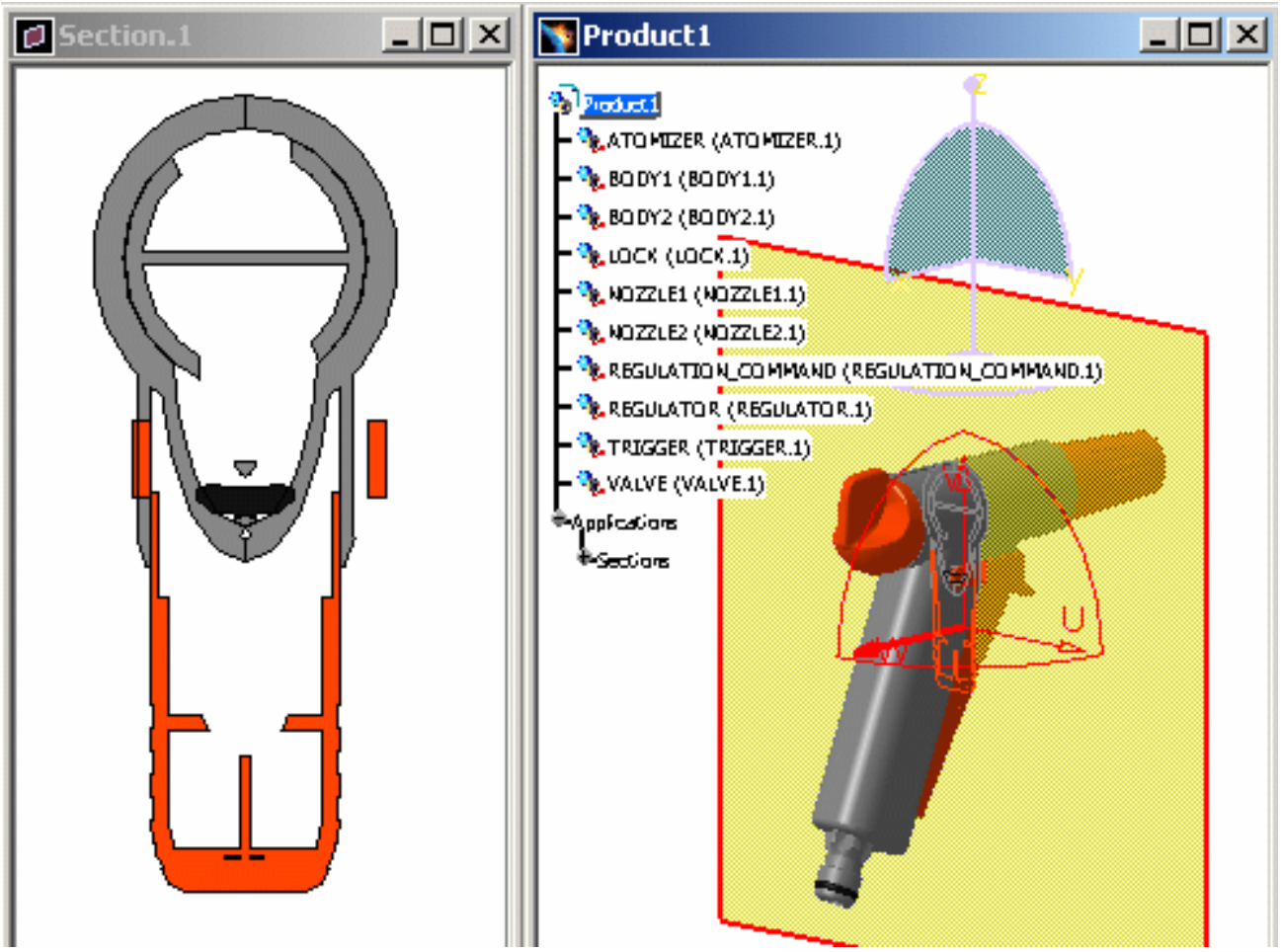
   The section plane is automatically created. If no selection is made, the plane sections all products. If products are selected, the plane sections selected products.



This plane is the master plane and controls all operations on the section slice.

The Sectioning Definition dialog box is displayed. This dialog box contains a wide variety of tools letting you position, move and rotate the master plane. For more information, see Positioning Planes with respect to a Geometrical Target, and Positioning Planes Using the Edit Position Command.

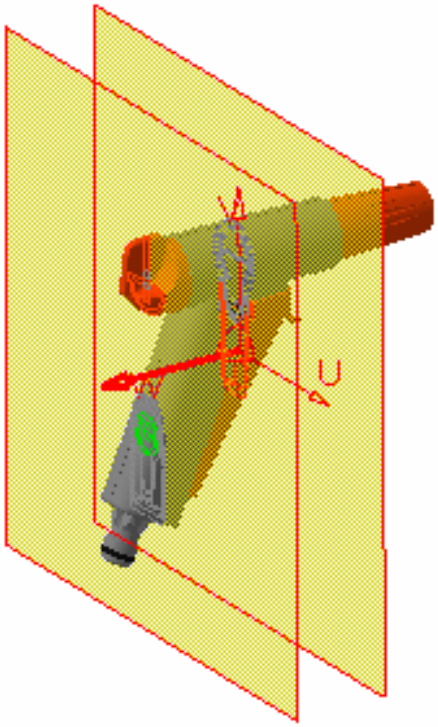
A Section viewer is automatically tiled vertically alongside the document window. It displays a front view of the generated section and is by default, locked in a 2D view.



2. In the Definition tab of the Sectioning Definition dialog box, click the Section Slice drop-down icon to create a section slice:

   A second plane, parallel to the first, is created. Together both planes define a section slice through your products. The contours of both planes are red.

   The Section viewer is automatically updated.



3. Adjust the thickness of the section slice: position the cursor over one of the slave plane edges, click then drag to translate the plane in the desired direction.

Note: As you move the cursor over plane edges, the cursor changes appearance and arrows identifying directions along which slice thickness can be defined appear. The thickness of the slice is also indicated as you drag.

4. Click **OK** when done.

Creating Section Boxes

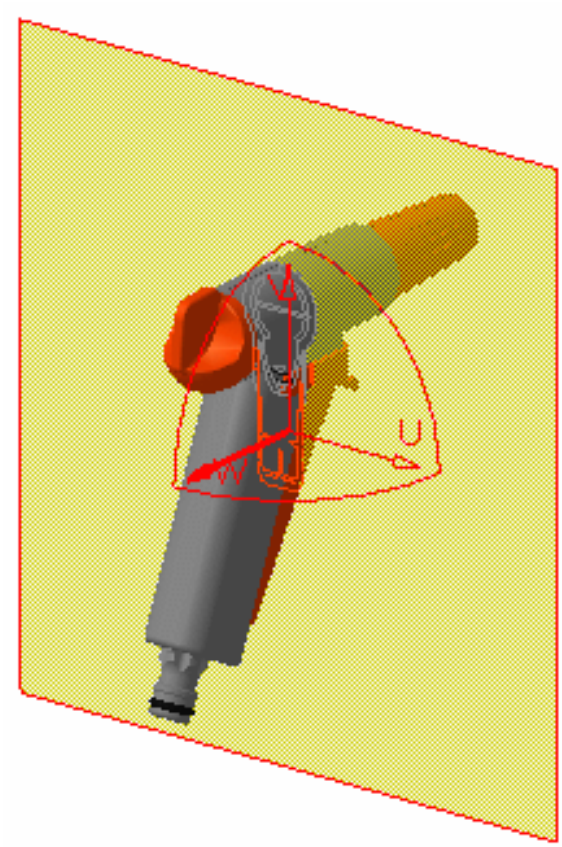
This task explains how to create section boxes. To do so, you must first create the master section plane.

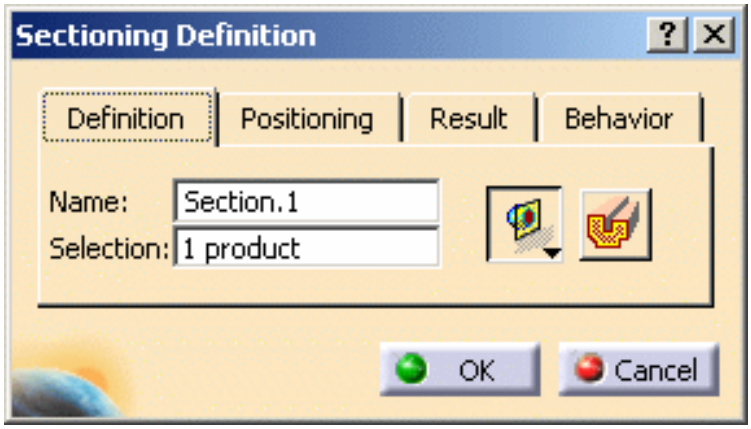
Insert the following cgr files: ATOMIZER.cgr, BODY1.cgr, BODY2.cgr, LOCK.cgr, NOZZLE1.cgr, NOZZLE2.cgr, REGULATION_COMMAND.cgr, REGULATOR.cgr, TRIGGER.cgr and VALVE.cgr.

They are to be found in the online documentation filetree in the common functionalities sample folder **cfysm/samples**.

1. Select **Insert -> Sectioning** from the menu bar, or click the Sectioning icon in the DMU Space Analysis toolbar to generate a section plane.

   The section plane is automatically created. If no selection is made before entering the command, the plane sections all products. If products are selected, the plane sections selected products.

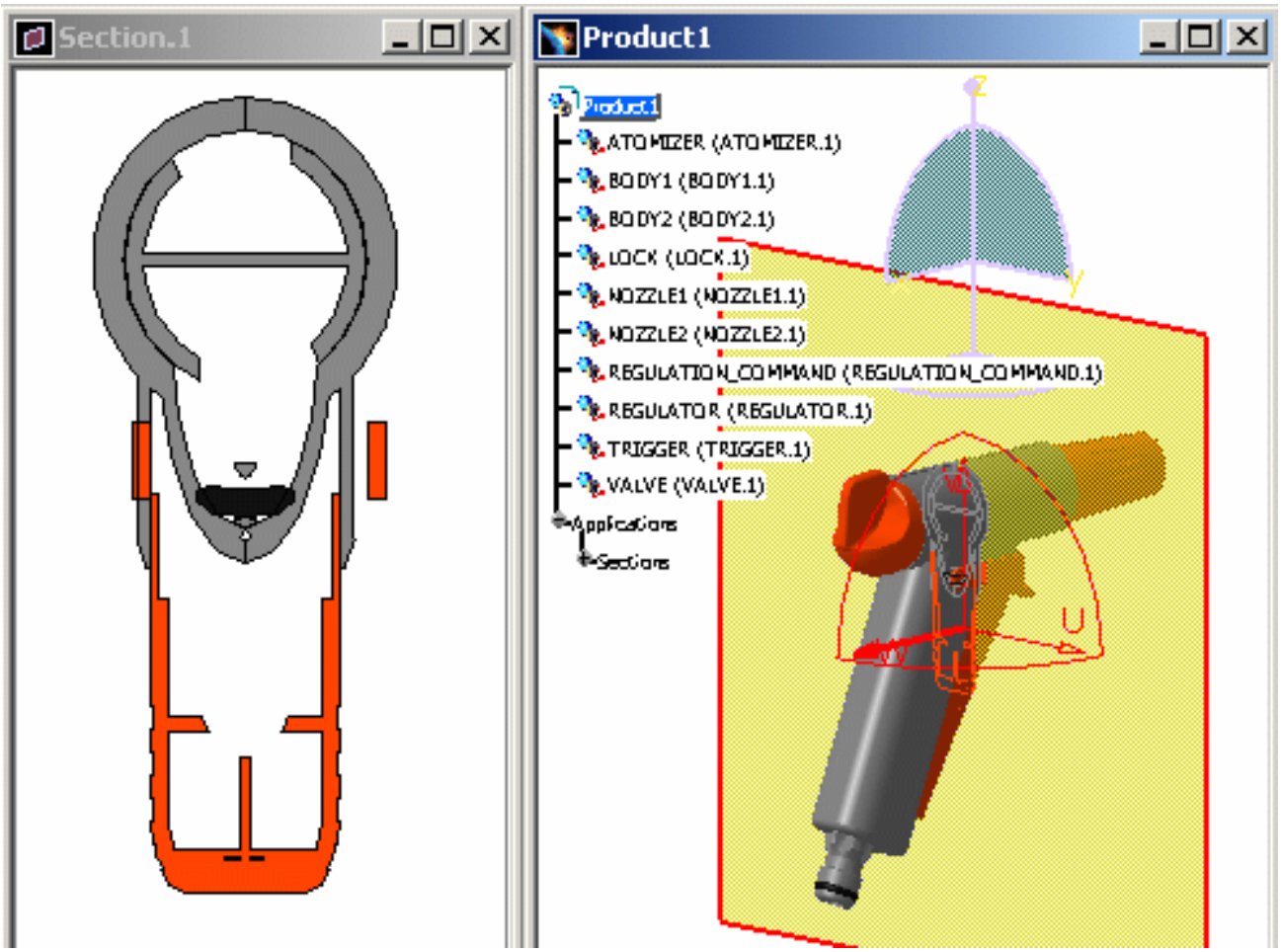




This plane is the master plane and controls all operations on the section box.

The Sectioning Definition dialog box is displayed. This dialog box contains a wide variety of tools letting you position, move and rotate the master plane. For more information, see Positioning Planes with respect to a Geometrical Target, and Positioning Planes Using the Edit Position Command.

A Section viewer is automatically tiled vertically alongside the document window. It displays a front view of the generated section and is by default, locked in a 2D view.



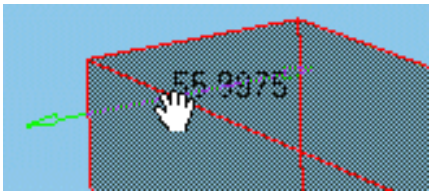
2. In the Definition tab, click the Section Box drop-down icon to create a section box:

A sectioning box is created. The contours of box planes are red. The Section viewer is automatically updated.

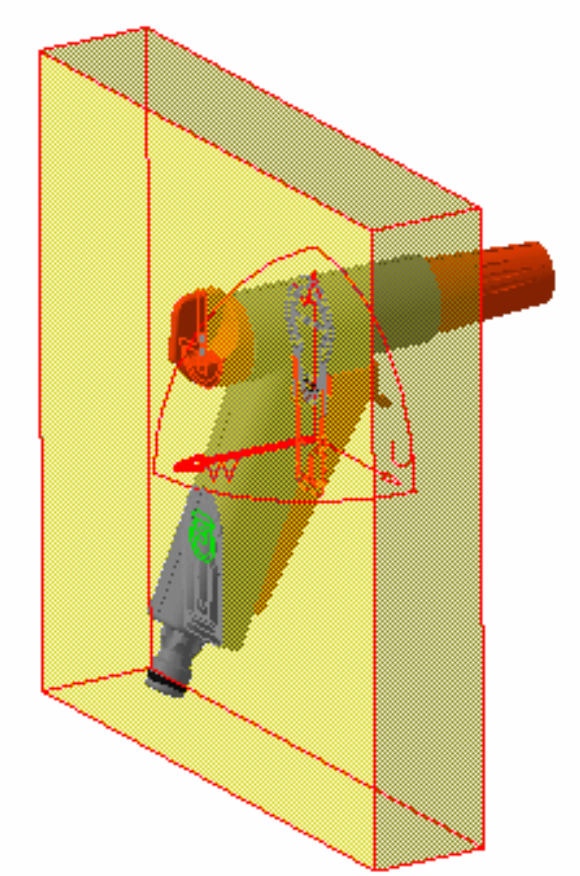
3. Adjust the thickness of the section box: position the cursor over one of the slave box plane edges, click then drag to translate the plane in the desired direction.

Notes:

- As you move the cursor over box edges, the cursor changes appearance and arrows identifying the directions along which box thickness is defined appear. Box thickness is also indicated as you drag.



- You can also re-size the box by clicking and dragging one of the box sides. Arrows likewise appear to help you.
- Use the Geometrical Target icon in the Positioning tab to snap boxes to planes.



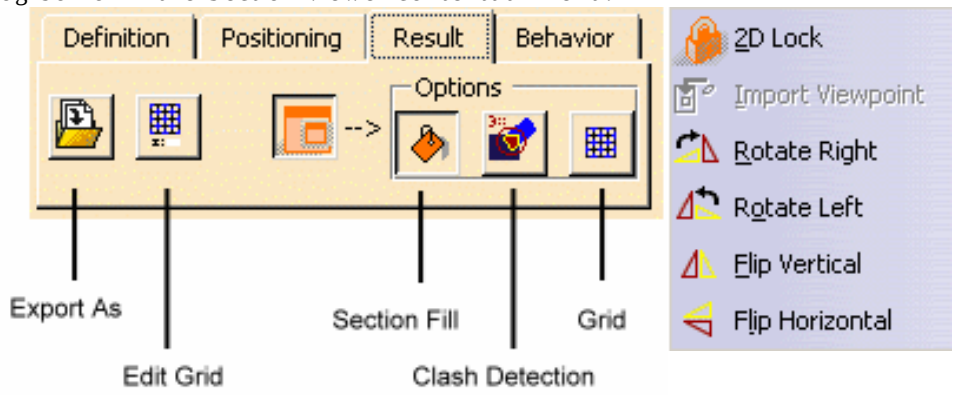
4. Click **OK** when done.

You can create boxes around the various areas of your product and then, using the Volume Cut command isolate the area on which you want to work.

More About the Section Viewer

This task illustrates how to make the most of section viewer capabilities.

Most of the commands described in this task are to be found in the Result tab of the Sectioning Definition dialog box or in the Section viewer contextual menu.



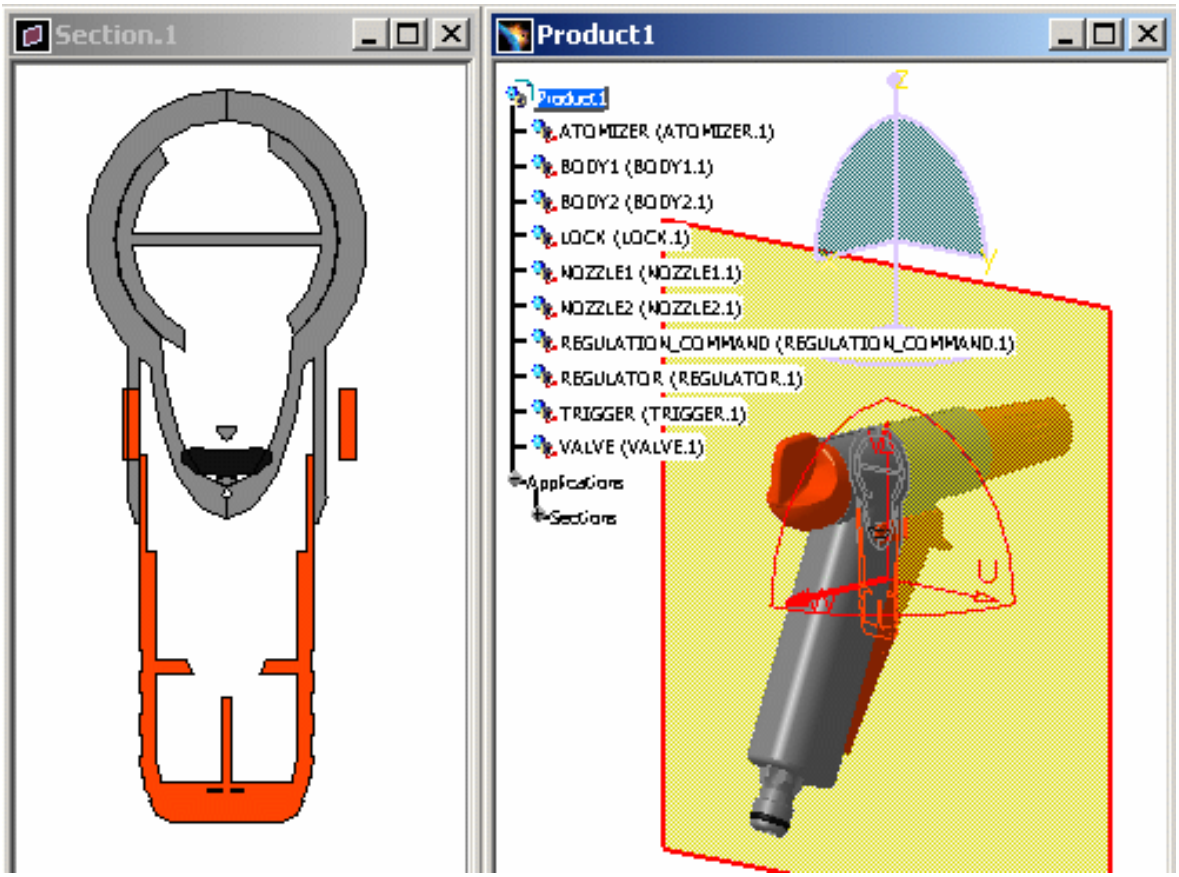
Insert the following cgr files: ATOMIZER.cgr, BODY1.cgr, BODY2.cgr, LOCK.cgr, NOZZLE1.cgr, NOZZLE2.cgr, REGULATION_COMMAND.cgr, REGULATOR.cgr, TRIGGER.cgr and VALVE.cgr.

They are to be found in the online documentation filetree in the common functionalities sample folder **cfysm/samples**.

1. Select **Insert -> Sectioning** from the menu bar, or click the Sectioning icon in the DMU Space Analysis toolbar and create the desired section plane, slice or box and corresponding section.
   The Section viewer is automatically tiled vertically alongside the document window. It displays a front view of the section, and is by default, locked in a 2D view. Points representing the intersection of the section plane with any wireframe elements are also visible in the Section viewer.

   Notice that the section view is a filled view. This is the default option. The fill capability generates surfaces for display and measurement purposes (area, center of gravity, etc.). To obtain a correct filled view, the section plane must completely envelop the product.

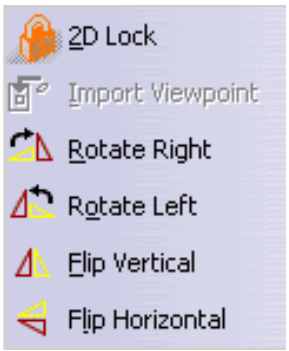
   To obtain an unfilled view, de-activate the Section Fill icon in the Result tab of the Sectioning Definition dialog box.



- In the Section viewer, the appearance of the cursor changes to attract your attention to the existence of the contextual menu.
- You can change the default settings for this window using **Tools ->Options...** command (DMU Sectioning tab under **Digital Mockup ->DMU Space Analysis**).

Orienting the Section

2. Orient the generated section.



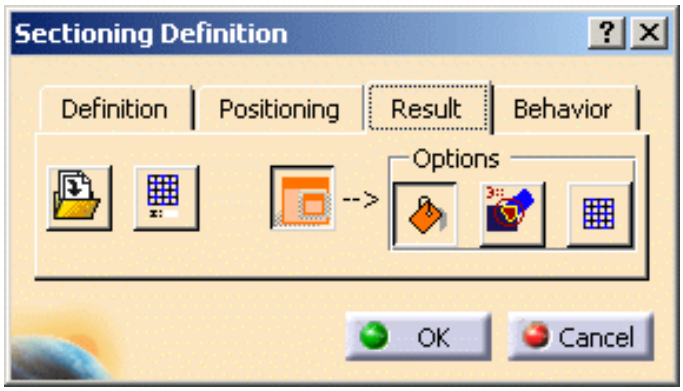
Flip and Rotate commands are to be found in the contextual menu. Right-click in the Section viewer and:

- Select Flip Vertical or Flip Horizontal to flip the section vertically or horizontally 180 degrees.
- Select Rotate Right or Rotate Left to rotate the section right or left 90 degrees.

Orienting the section using Flip and Rotate commands is not persistent. If you exit the section viewer, any flip and rotate settings are lost.

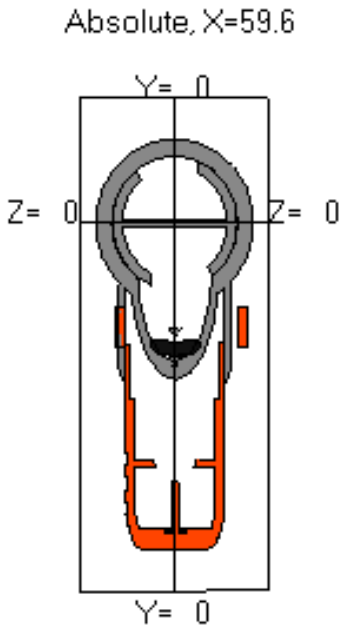
Working with the 2D Grid

3. Click the Result tab in the Sectioning Definition dialog box, then select the Grid icon under Options to display a 2D grid.



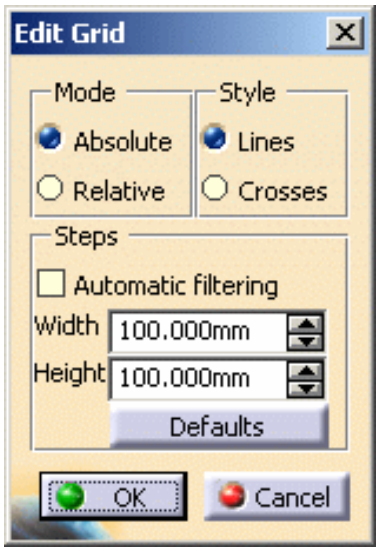
By default, grid dimensions are those of the generated section. Moving the section plane re-sizes the grid to results.

To size the grid to the section plane, clear the Automatic grid re-sizing check box in the DMU Sectioning tab (**Tools -> Options..., Digital Mockup -> DMU Space Analysis**).
You can edit the grid step, style and mode using the Edit Grid command.



4. Select the Edit Grid icon to adjust grid parameters:

The Edit Grid dialog box appears:



In the above example, the grid mode is absolute and the style is set to lines.

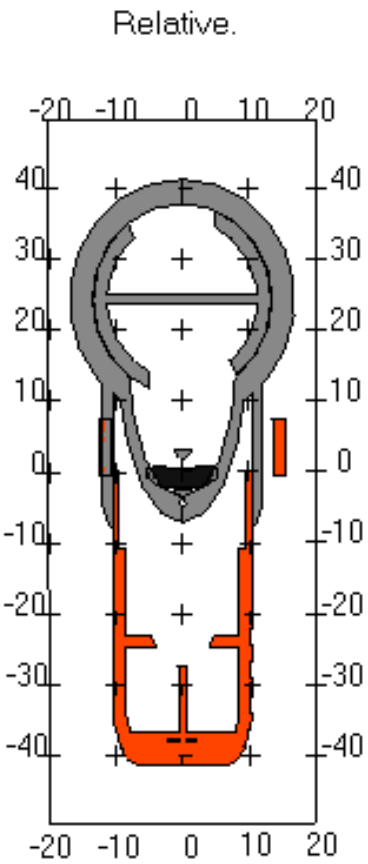
In the absolute mode, grid coordinates are set with respect to the absolute axis system of the document.

The grid step is set to the default value of 100. The arrows let you scroll through a discrete set of logarithmically calculated values. You can also enter a grid step manually.

Units are current units set using **Tools-> Options** (Units tab under **General-> Parameters and Measure**).

5. Scroll through grid width and height and set the grid step to 10 x 10.

6. Click the Relative mode option button:

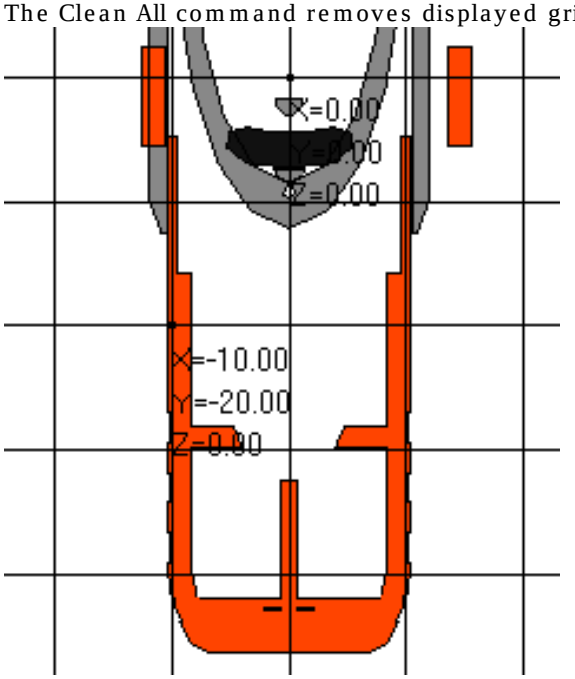


In the relative mode, the center of the grid is placed on the center of section plane.

7. Click the Crosses style option button.

Grid parameters are persistent: any changes to default parameters are kept and applied next time you open the viewer or re-edit the section.

8. Click the Automatic filtering checkbox to adjust the level of detail of grid display when you zoom in and out.
9. Right-click the grid then select Coordinates to display the coordinates at selected intersections of grid lines. The Clean All command removes displayed grid coordinates.



Note: You can customize both grid and Section viewer settings using the **Tools -> Options...** command (DMU Sectioning tab under **Digital Mockup ->DMU Space Analysis**).
Alternatively, select **Analyze ->Graphic Messages ->Coordinate** to display the coordinates of points, and/or **Name to** identify products as your cursor moves over them.

Clicking turns the temporary markers into 3D annotations.

10. Click **OK** in the Edit Grid dialog box when done.

Working with a 3D View

By default, the Section viewer is locked in a 2D view. De-activating the 2D view lets you:

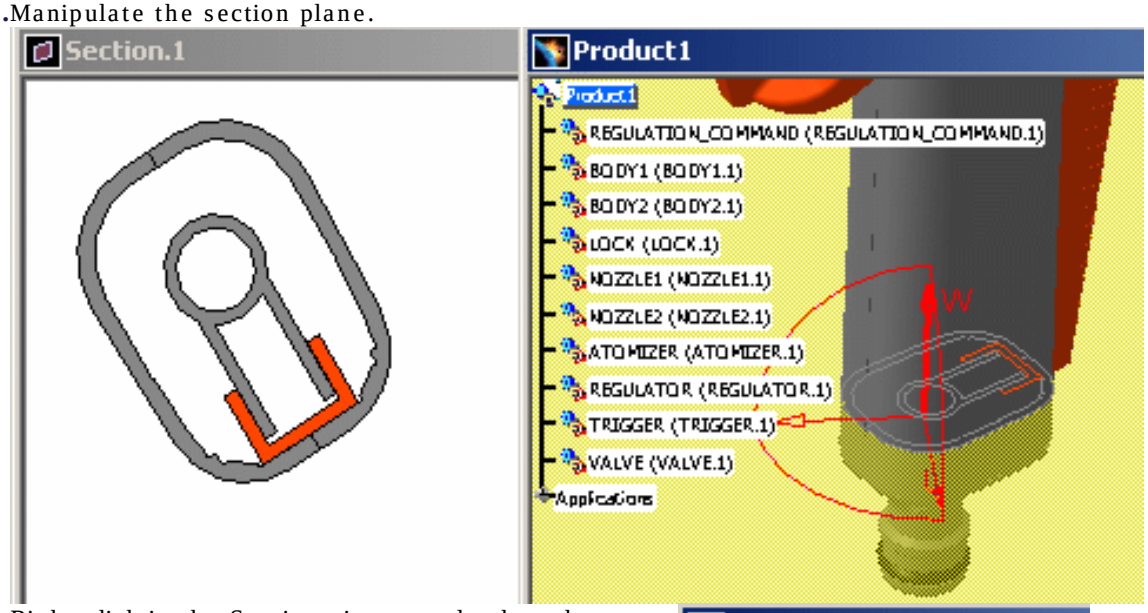
- Work in a 3D view and gives you access to 3D viewing tools
- Set the same viewpoint in the Section viewer as in the document window.

Returning to a 2D view snaps the viewpoint to the nearest orthogonal view defined in the Section viewer.

11. Right-click in the Section viewer and select the 2D Lock command from the contextual menu.

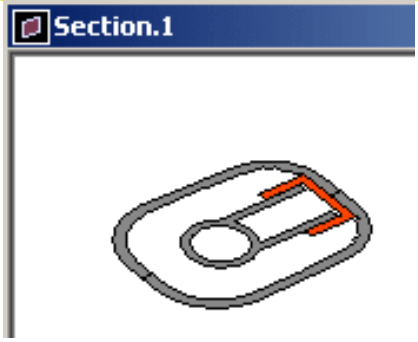
The Import Viewpoint command becomes available.

12. Manipulate the section plane.

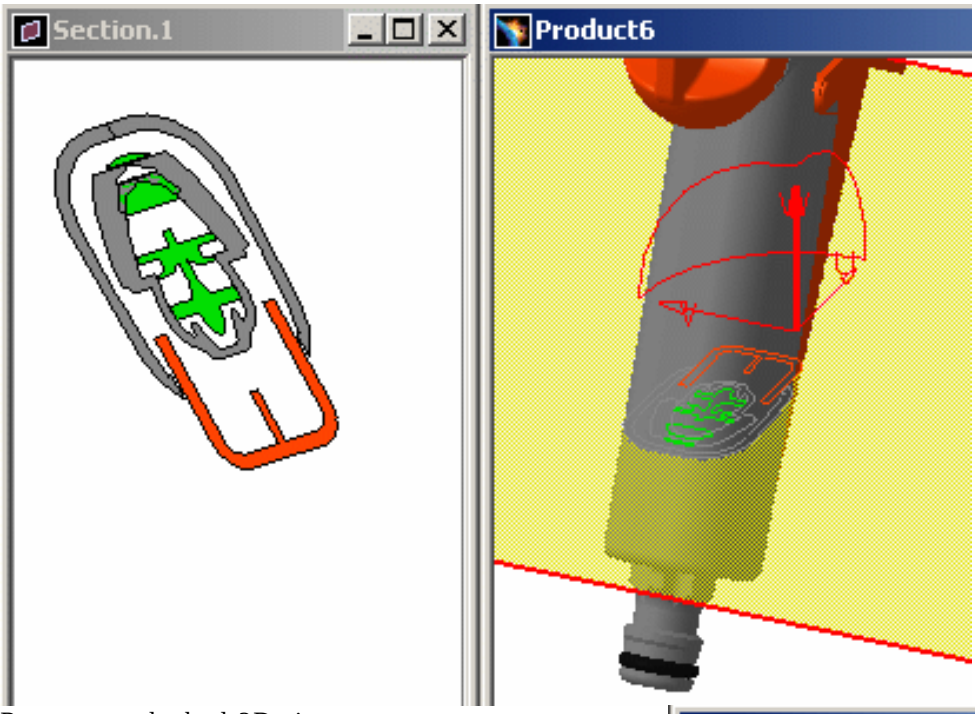


13. Right-click in the Section viewer and select the Import Viewpoint command from the contextual menu.

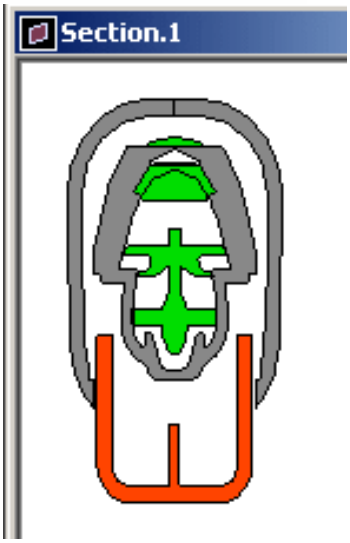
The viewpoint in the Section viewer is set to that of the document window.



14. Continue manipulating the section plane.



15. Return to a locked 2D view.



The viewpoint in the Section viewer snaps to the nearest orthogonal viewpoint in this viewer and not to the viewpoint defined by the local axis system of the plane in the document window.

You can also save sectioning results in a variety of different formats using the **Export As** command in the Result tab of the Sectioning Definition dialog box or the **Capture** command (**Tools ->Image ->Capture**).

16. Click **OK** in the Sectioning Definition dialog box when done.

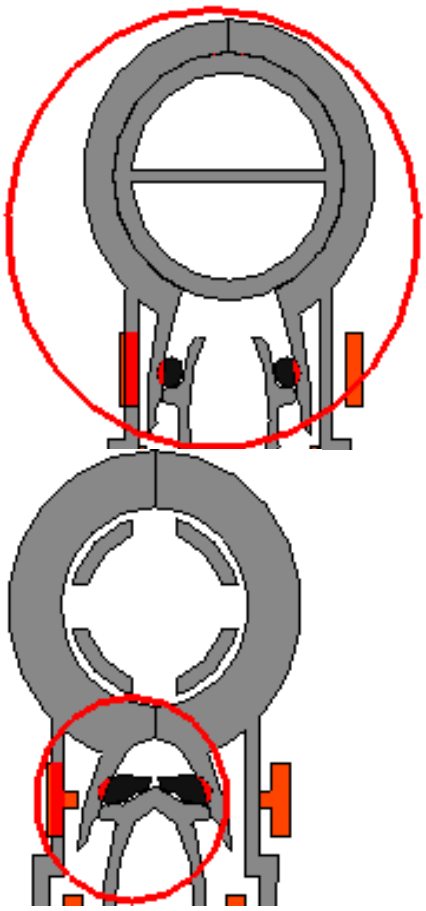
If you exit the Sectioning command with the Section viewer still active, this window is not closed and filled sections remain visible.

P2 Functionality - Detecting Collisions

In DMU-P2, You can detect collisions between 2D sections. To do so, click the Clash Detection icon in the Result tab of the Sectioning Definition dialog box.

Clashes detected are highlighted in the Section viewer.

Collision detection is dynamic: move the section plane and watch the Section viewer display being updated.



Creating 3D Section Cuts

3D section cuts cut away the material from the plane, beyond the slice or outside the box to expose the cavity within the product.

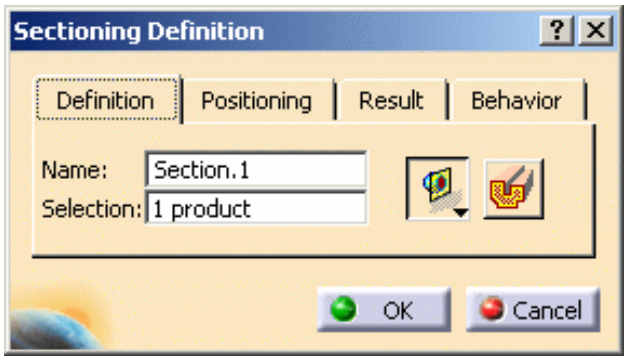
This task explains how to create 3D section cuts.

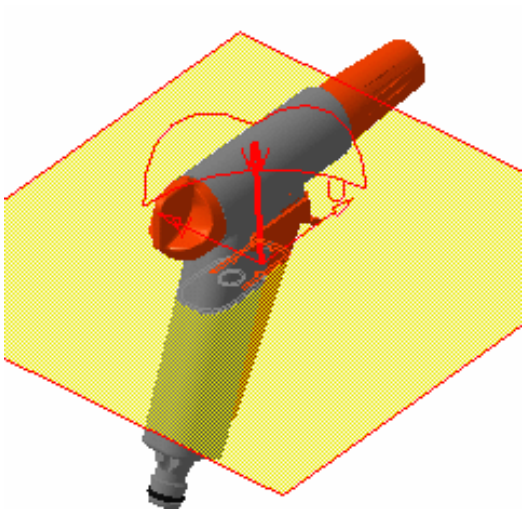
Insert the following cgr files: ATOMIZER.cgr, BODY1.cgr, BODY2.cgr, LOCK.cgr, NOZZLE1.cgr, NOZZLE2.cgr, REGULATION_COMMAND.cgr, REGULATOR.cgr, TRIGGER.cgr and VALVE.cgr.

They are to be found in the online documentation filetree in the common functionalities sample folder **cfysm/samples**.

1. Select **Insert -> Sectioning** from the menu bar, or click the Sectioning icon in the DMU Space Analysis toolbar and create a section plane.

   The Sectioning Definition dialog box appears.

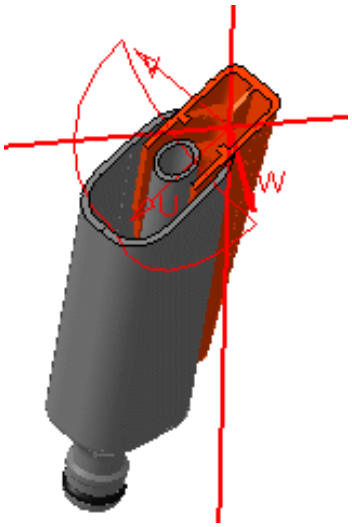




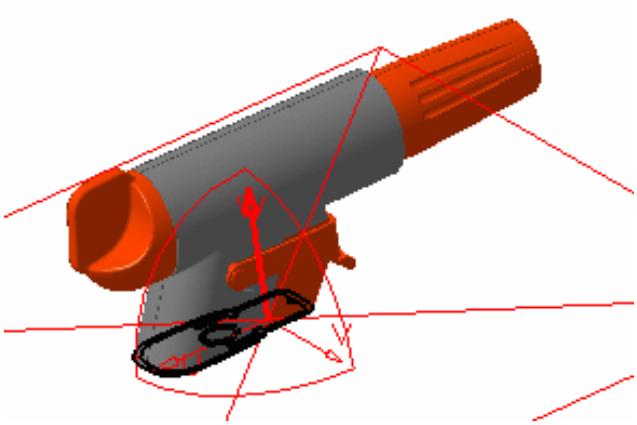
2. In the Definition tab, click the Volume Cut icon to obtain a section cut:

   The material in the negative direction along the normal vector of the plane (W-axis) is cut away exposing the cavity within the product.

   Note: In some cases, the normal vector of the plane is inverted to give you the best view of the cut.



Double-click the normal vector of the plane to invert it, or click the Invert Normal icon in the Positioning tab of the Sectioning Definition dialog box.

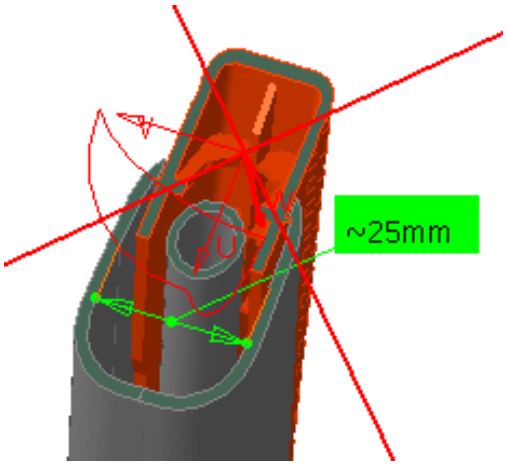


3. Re-click the icon to restore the material cut away.
4. Click **OK** when done.

3D Section Cut Display

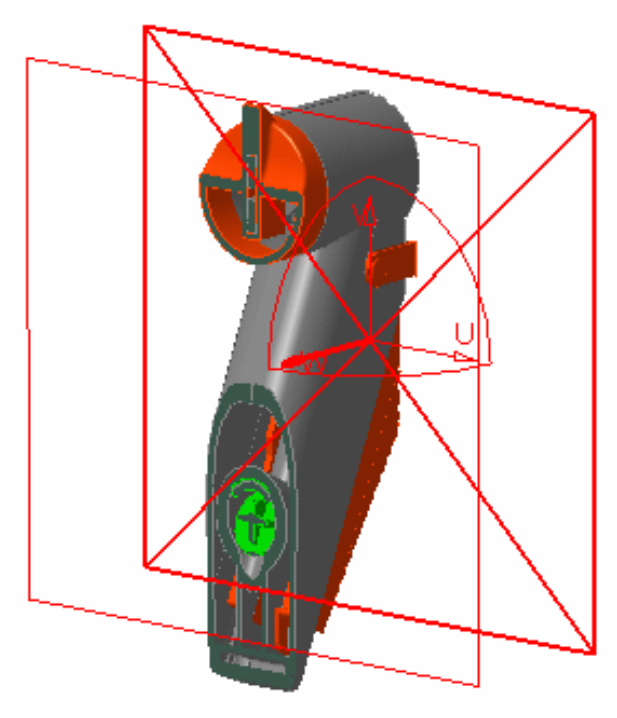
The 3D section cut display is different when the sectioning tool is a plane. To obtain the same display as for slices and boxes (see illustrations below) and make measures on the generated wireframe cut:

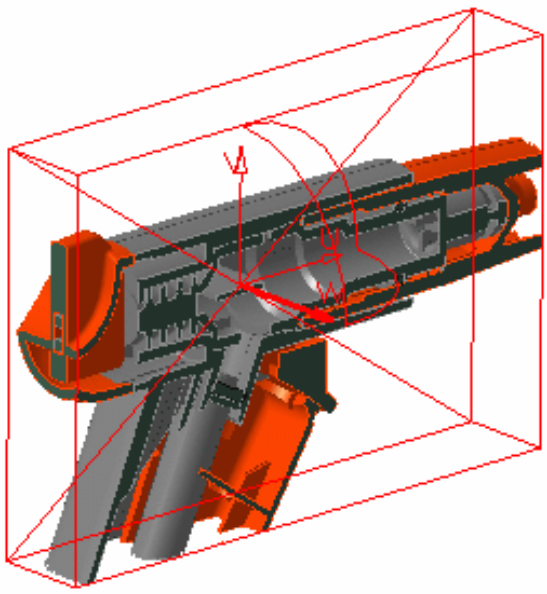
- Select the **Allow measures on a section created with a simple plane** option in the **DMU Sectioning** tab (**Tools -> Options, Digitial Mockup -> DMU Space Analysis**)
- Then, create your section cut based on a plane.



When the Sectioning Tool is a Slice:

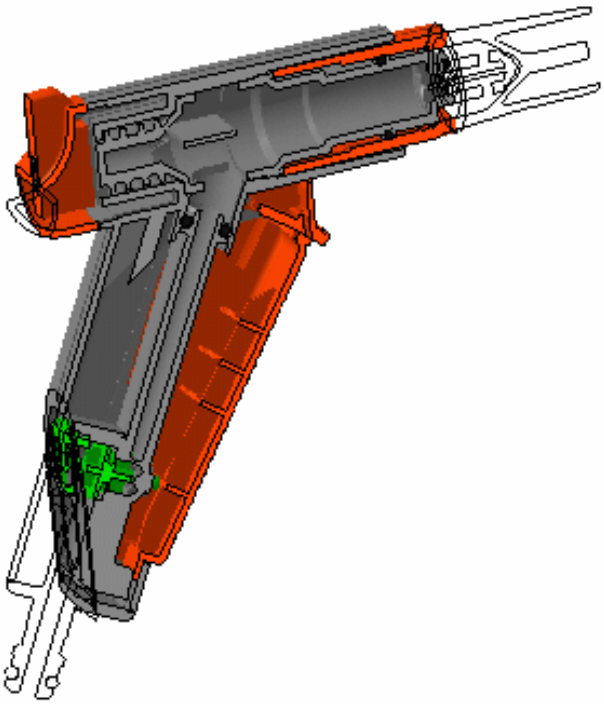
When the Sectioning Tool is a Box:





P2 Functionality

In DMU-P2, you can turn up to six independent section planes into clipping planes using the Volume Cut command to focus on the part of the product that interests you most.



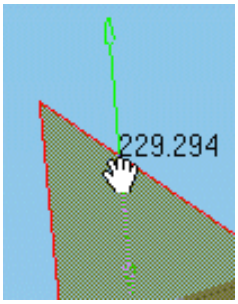
DMU Review

Section cuts created during DMU Reviews are not persistent and are only valid for the duration of the review. If you exit the DMU Review, the section cut is lost.

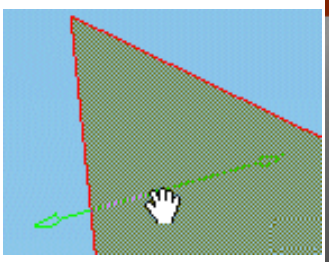
Manipulating Planes Directly

You can re-dimension, move and rotate section planes, or the master plane in the case of section slices and boxes, directly. As you move the cursor over the plane, the plane edge or the local axis system, its appearance changes and arrows appear to help you.

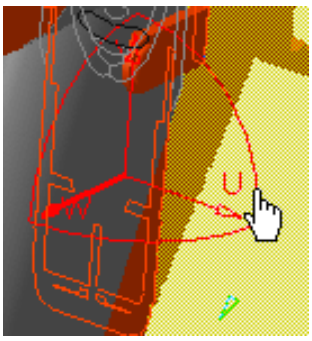
Re-dimensioning:



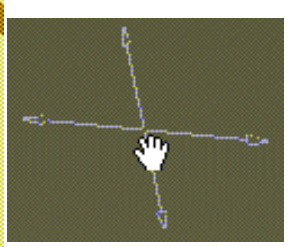
Moving along the normal vector of the plane:



Rotating:



Moving in the x,y plane of the local axis system:



Sectioning results are updated in the Section viewer as you manipulate the plane.

To change this setting and have results updated when you release the mouse button only, de-activate the appropriate setting in the DMU Sectioning tab (**Tools ->Options..., Digital Mockup ->DMU Space Analysis**).

This task illustrates how to manipulate section planes directly.

Insert the following cgr files: ATOMIZER.cgr, BODY1.cgr, BODY2.cgr, LOCK.cgr, NOZZLE1.cgr, NOZZLE2.cgr, REGULATION_COMMAND.cgr, REGULATOR.cgr, TRIGGER.cgr and VALVE.cgr.

They are to be found in the online documentation filetree in the common functionalities sample folder **cfysm/samples**.

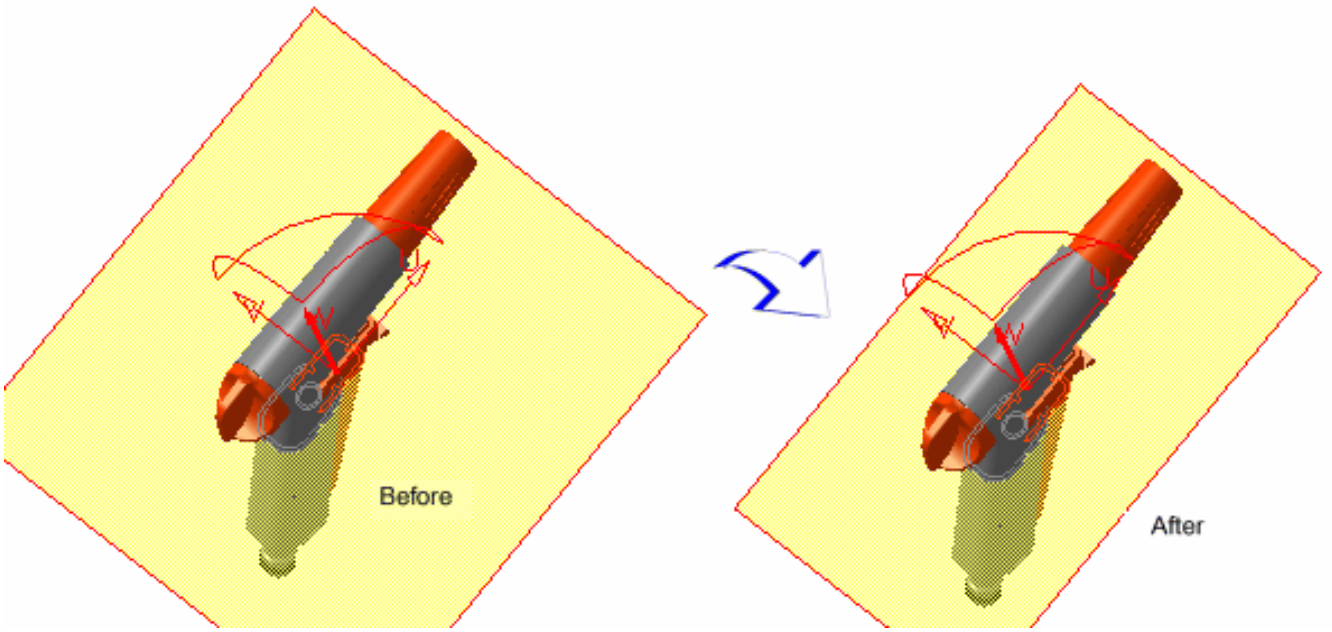
1. Select **Insert -> Sectioning** from the menu bar, or click the Sectioning icon in the DMU Space Analysis toolbar and create a section plane.

   A Section viewer showing the generated section is automatically tiled vertically alongside the document window.

   The generated section is automatically updated to reflect any changes made to the section plane.

   You can re-dimension the section plane:
2. Click and drag plane edges to re-dimension plane:

   Note: A dynamic plane dimension is indicated as you drag the plane edge.



You can view and edit plane dimensions in the Edit Position and Dimensions command. The plane height corresponds to its dimension along the local U-axis and the width to its dimension along the local V-axis.

You can move the section plane along the normal vector of the plane:

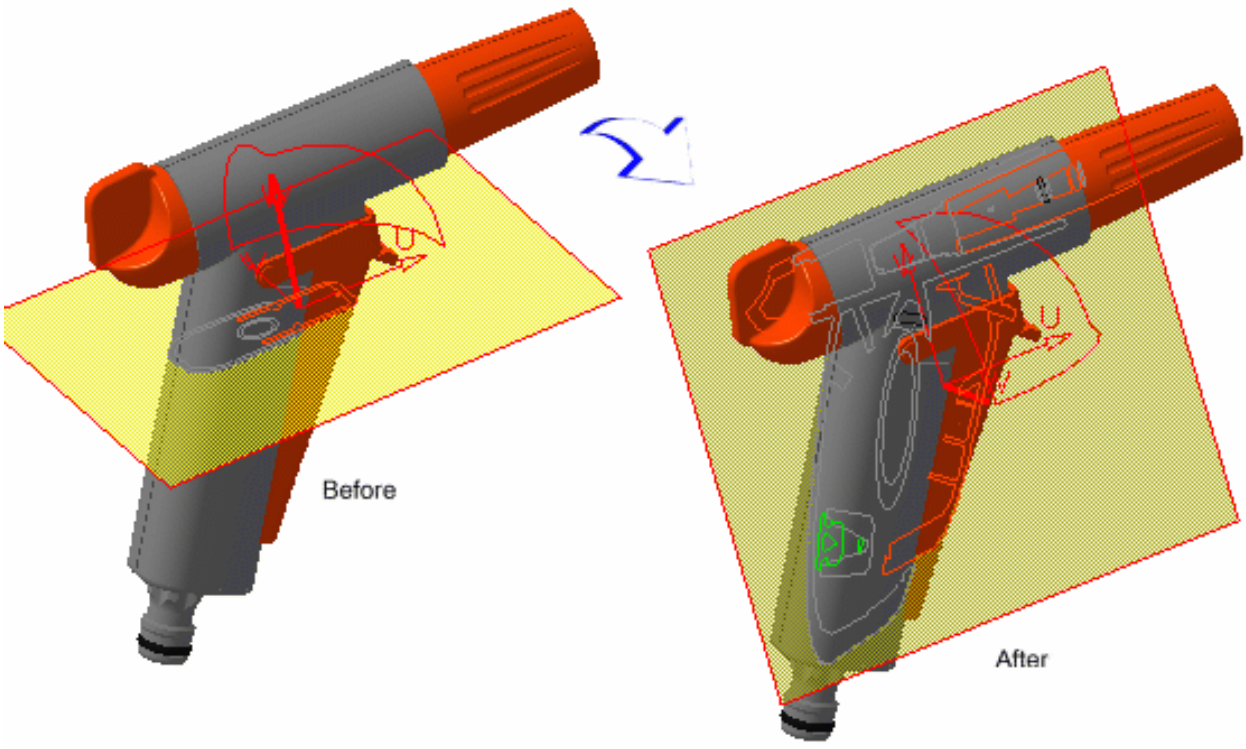
3. Move the cursor over the plane, click and drag to move the plane to the desired location.

You can move the section plane in the U,V plane of the local axis system:

4. Press and hold down the left mouse button, then the middle mouse button and drag (still holding both buttons down) to move the plane to the desired location.

You can rotate the section plane around its axes:

5. Move the cursor over the desired plane axis system axis, click and drag to rotate the plane around the selected axis.



6. (Optional) Click the Reset Position icon in the Positioning tab of the Sectioning Definition dialog box to restore the center of the plane to its original position.
7. Click **OK** in the Sectioning Definition dialog box when done.

Positioning Planes Using the Edit Position and Dimensions Command

In addition to manipulating the plane directly in the geometry area, you can position the section plane more precisely using the Edit Position and Dimensions command. You can move the plane to a new location as well as rotate the plane. You can also re-dimension the section plane.

In the case of section slices and boxes, it is the master plane that controls how the slice or box will be positioned.

This task illustrates how to position and re-dimension the section plane using the Edit Position and Dimensions command.

Insert the following cgr files: ATOMIZER.cgr, BODY1.cgr, BODY2.cgr, LOCK.cgr, NOZZLE1.cgr, NOZZLE2.cgr, REGULATION_COMMAND.cgr, REGULATOR.cgr, TRIGGER.cgr and VALVE.cgr.

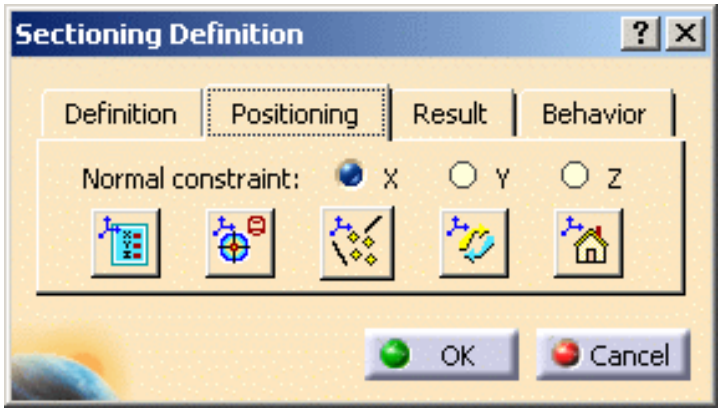
They are to be found in the online documentation filetree in the common functionalities sample folder **cfysm/samples**.

1. Select **Insert -> Sectioning** from the menu bar, or click the Sectioning icon in the DMU Space Analysis toolbar and create a section plane.

   A Section viewer showing the generated section is automatically tiled vertically alongside the document window. The generated section is automatically updated to reflect any changes made to the section plane.

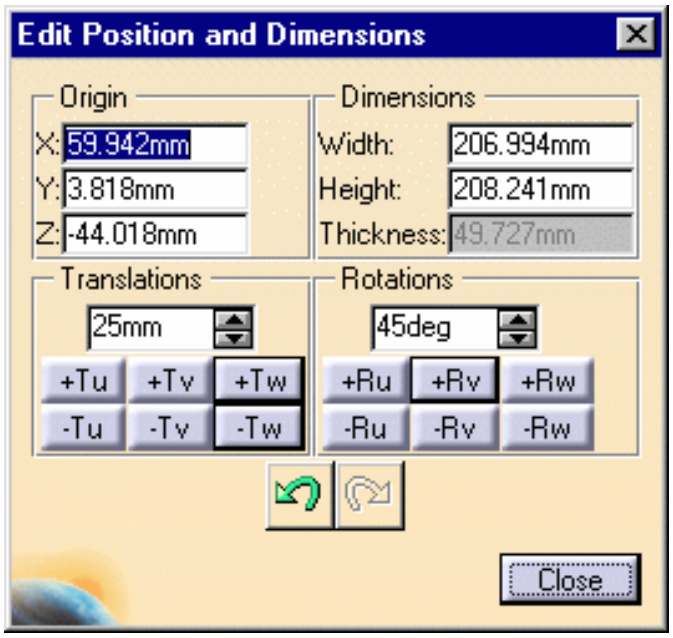
   The Sectioning Definition dialog box is also displayed.

2. Click the Positioning tab in the Sectioning Definition dialog box.



3. Click the Edit Position and Dimensions icon to enter parameters defining the position of the plane:

   The Edit Position and Dimensions dialog box appears.



4. Enter values in Origin X, Y or Z boxes to position the center of the plane with respect to the absolute system coordinates entered.

By default, the center of the plane coincides with the center of the bounding sphere around the products in the current selection.

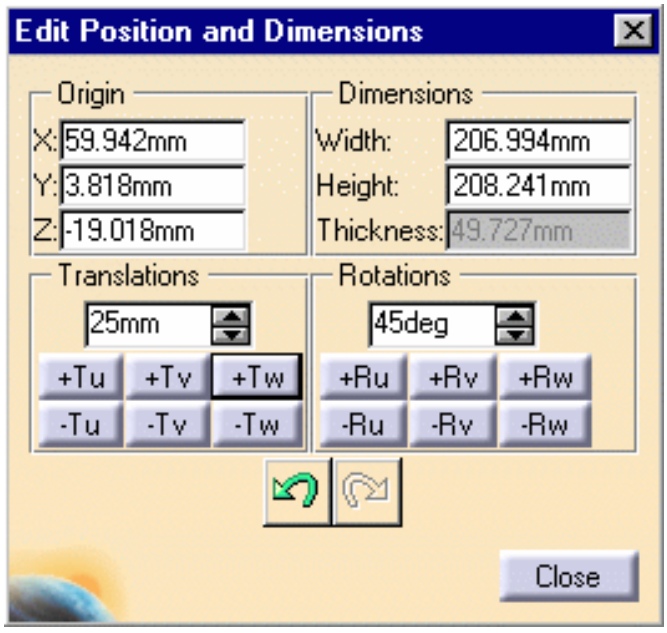
Using the **Tools -> Options...** command (DMU Sectioning tab under **Digital Mockup ->DMU Space Analysis**), you can customize settings for both the normal vector and the origin of the plane.

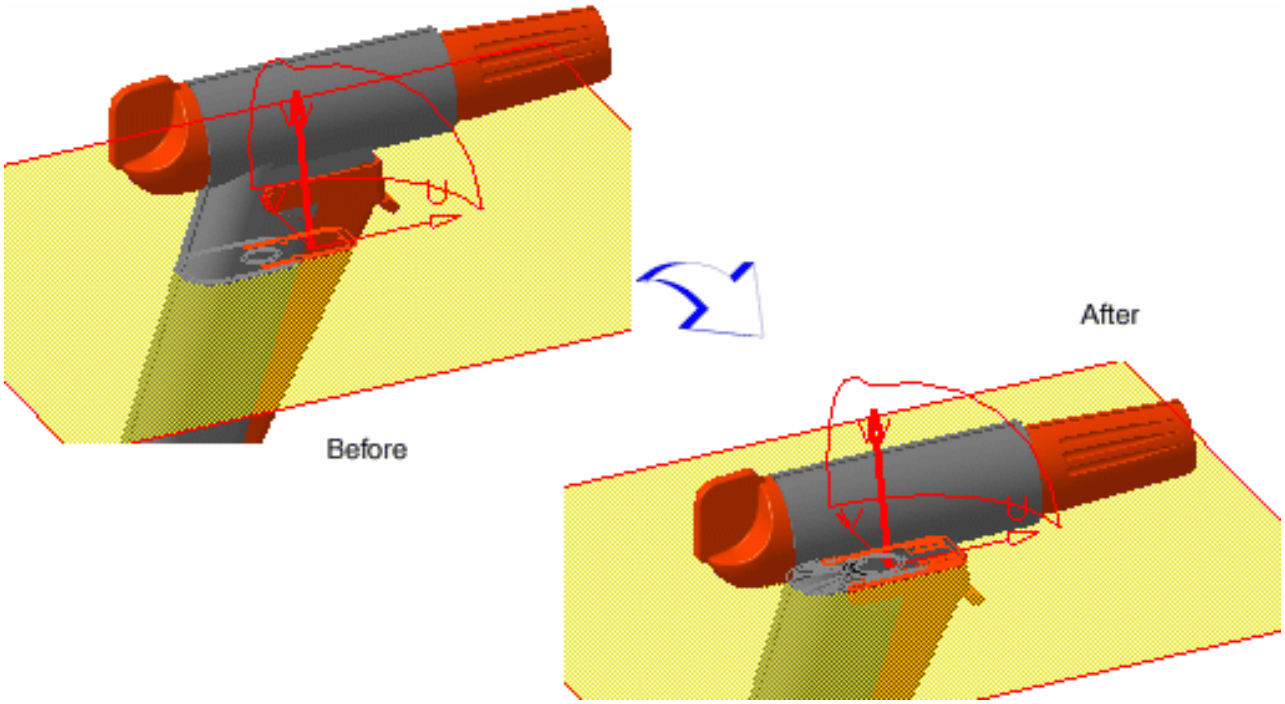
You can move the section plane to a new location. Translations are made with respect to the local plane axis system.

5. Enter the translation step directly in the Translation spin box or use spin box arrows to scroll to a new value, then click -Tu, +Tu, -Tv, +Tv, -Tw, +Tw, to move the plane along the selected axis by the defined step.

Note: Units are current units set using **Tools-> Options** (Units tab under **General-> Parameters and Measure**).

Change the translation step to 25mm and click +Tw for example. The plane is translated 25 mm in the positive direction along the local W-axis.



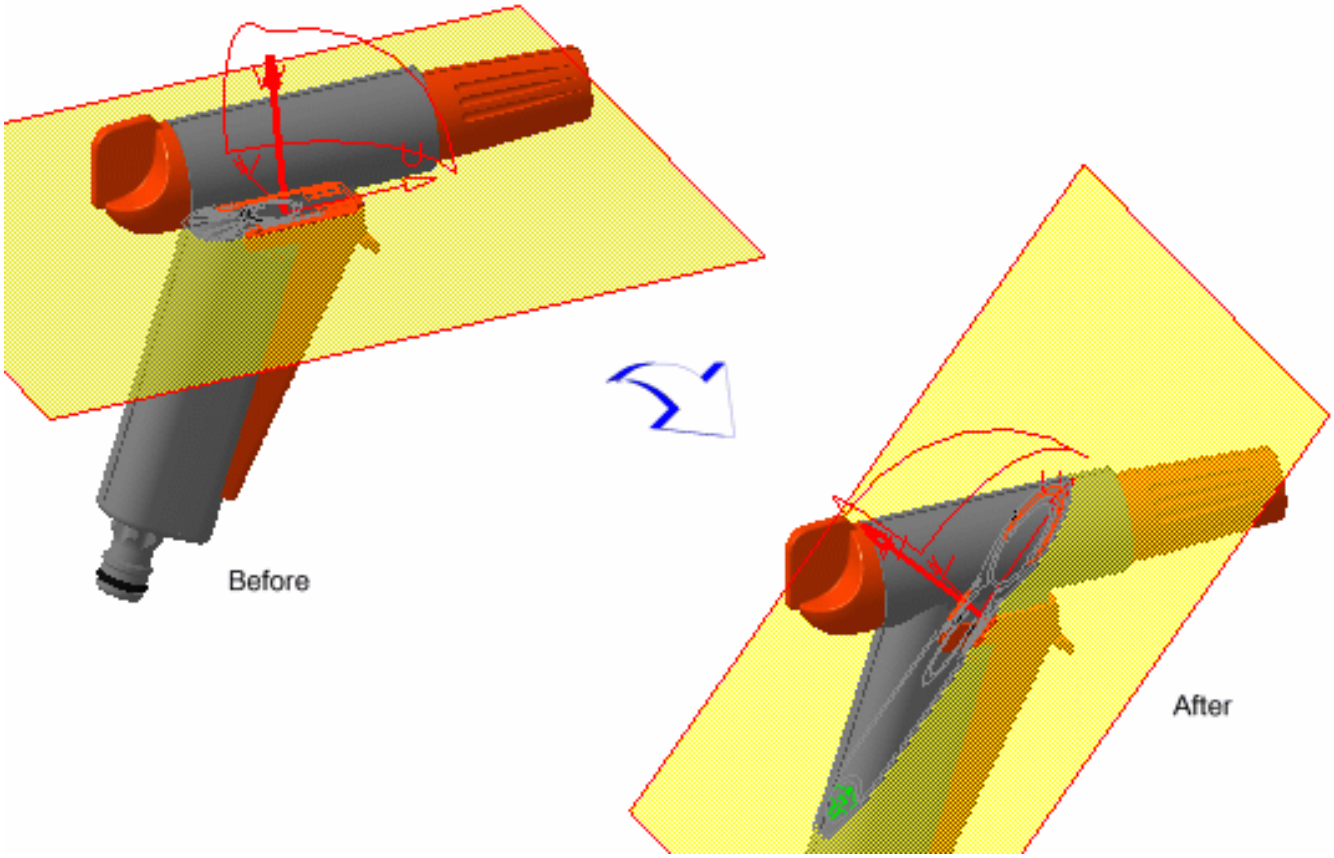


You can rotate the section plane. Rotations are made with respect to the local plane axis system.

6. Enter the rotation step directly in the Rotation spin box or use spin box arrows to scroll to a new value, then click -Ru, +Ru, -Rv, +Rv, -Rw, +Rw, to rotate the plane around the selected axis by the defined step.

Note: Units are current units set using **Tools -> Options**.

With a rotation step of 45 degrees, click +Rv for example to rotate the plane by the specified amount in the positive direction around the local V-axis.

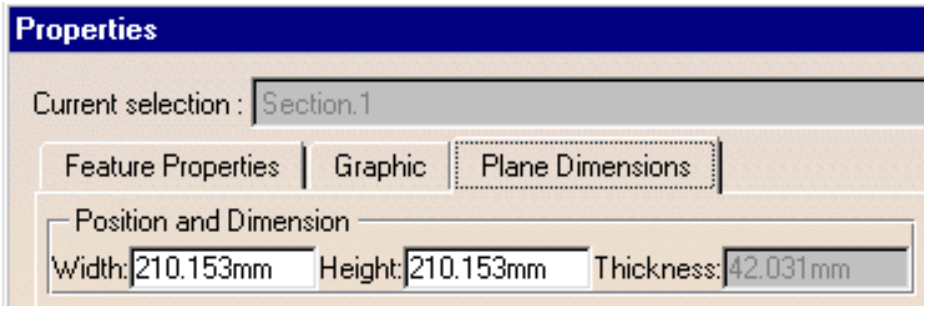


You can edit plane dimensions. The plane height corresponds to its dimension along the local U-axis and the width to its dimension along the local V-axis. You can also edit slice or box thickness.

7. Enter new width, height and/or thickness values in the Dimensions box to re-dimension the plane.

The plane is re-sized accordingly.

- Use Undo and Redo icons in the Edit Position and Dimensions dialog box to cancel the last action or recover the last action undone respectively.
- Use the Reset Position  icon in the Positioning tab of the Sectioning Definition dialog box to restore the section plane to its original position.
- You can also view and edit plane dimensions in the Properties dialog box (**Edit -> Properties** or via the contextual menu).
  This command is not available when using the sectioning command.



8. Click Close in the Edit Position and Dimensions dialog box when satisfied.
9. Click OK in the Sectioning Definition dialog box when done.

Positioning Planes On a Geometric Target

You can position section planes, section slices and section boxes with respect to a geometrical target (a face, edge, reference plane or cylinder axis). In the case of section slices and boxes, it is the master plane that controls how the slice or box will be positioned.

This task illustrates how to position a section plane with respect to a geometrical target.

Insert the following cgr files: ATOMIZER.cgr, BODY1.cgr, BODY2.cgr, LOCK.cgr, NOZZLE1.cgr, NOZZLE2.cgr, REGULATION_COMMAND.cgr, REGULATOR.cgr, TRIGGER.cgr and VALVE.cgr.

They are to be found in the online documentation filetree in the common functionalities sample folder **cfysm/samples**.

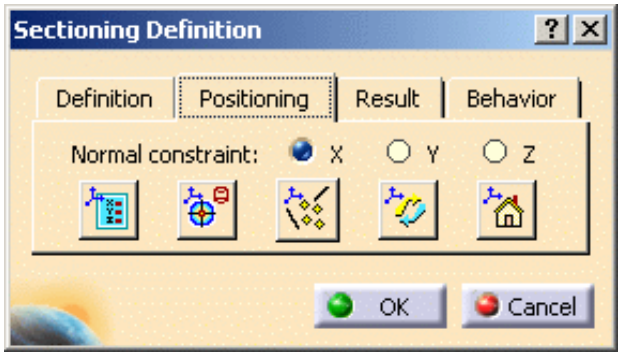
1. Select **Insert -> Sectioning** from the menu bar, or click the Sectioning icon in the DMU Space Analysis toolbar and create a section plane.

   The Sectioning Definition dialog box appears.

   A Section viewer showing the generated section is automatically tiled vertically alongside the document window.

   The generated section is automatically updated to reflect any changes made to the section plane.

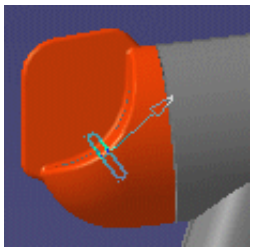
2. Click the Positioning tab in the Sectioning Definition dialog box.



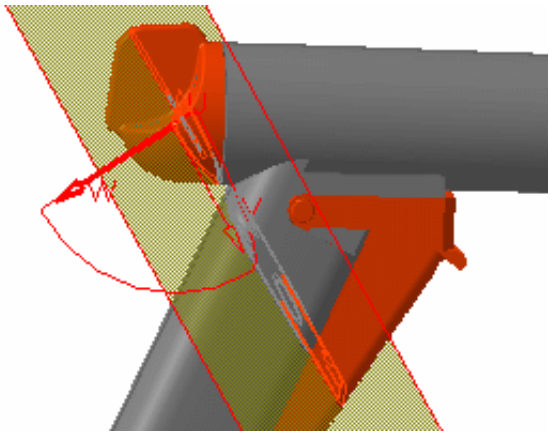
3. Click the Geometrical Target icon to position the plane with respect to a geometrical target.

4. Point to the target of interest:

   A rectangle and vector representing the plane and the normal vector of the plane appear in the geometry area to assist you position the section plane. It moves as you move the cursor.

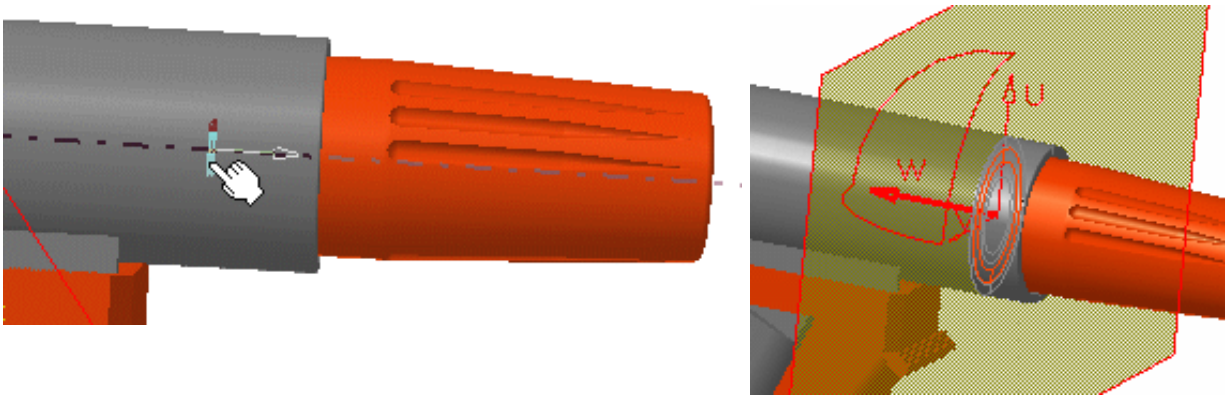


5. When satisfied, click to position the section plane on the target.



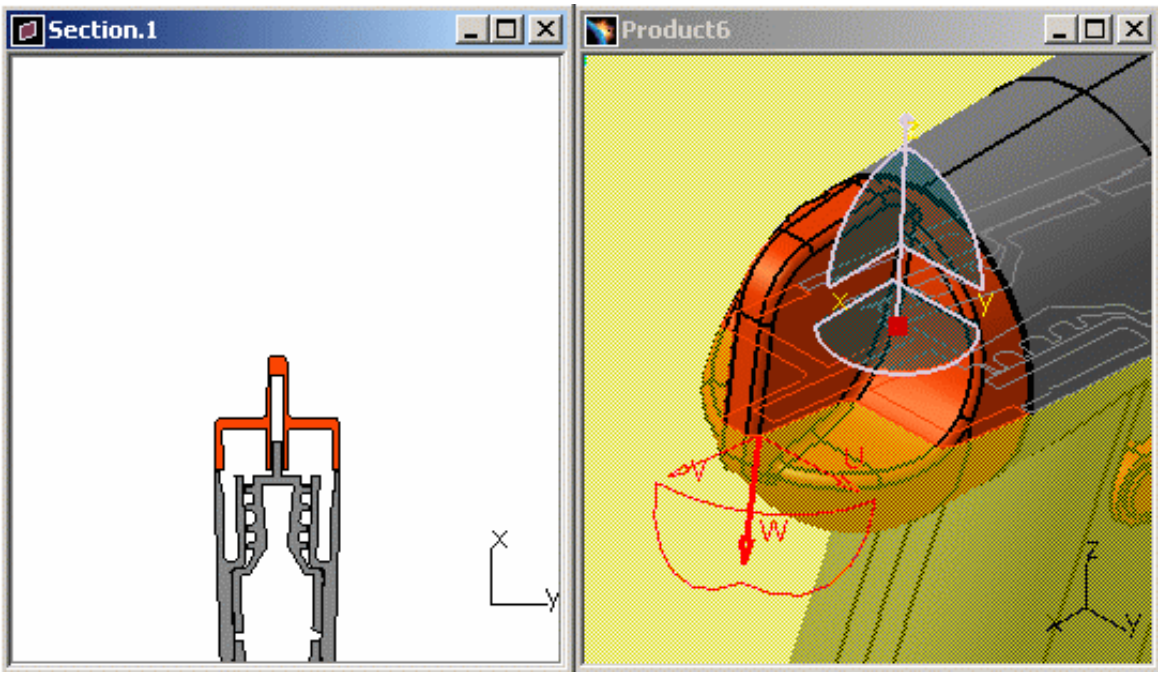
Notes:

- To position planes orthogonal to edges, simply click the desired edge.
- A smart mode recognizes cylinders and snaps the plane directly to the cylinder axis. This lets you, for example, make a section cut normal to a hole centerline. To de-activate this mode, use the Ctrl key.



- Selecting the Automatically reframe option in the DMU Sectioning tab (**Tools -> Options -> Digital Mockup -> DMU Space Analysis**), reframes the Section viewer and locates the point at the center of the target at the center of the Section viewer.

  Zooming in lets youpinpointthe selected point.
  This is particularly useful when using snap capabilities in a complex DMU session containing a large number of objects.



P2 Functionality

In DMU-P2, you can move the plane along a curve, edge or surface:

- Point to the target of interest
- Press and hold down the Ctrl key
- Still holding down the Ctrl key, move the cursor along the target. The plane is positioned tangent to the small target plane. As you move the cursor, the plane moves along the curve or edge.

6. (Optional) Click the Reset Position icon to restore the center of the plane to its original position.
7. Click OK in the Sectioning Definition dialog box when done.

Snapping Section Boxes to Planes

You can snap section boxes to two planes. The first target positions the master plane, the second defines a rotation (if needed) and adjusts box dimensions.

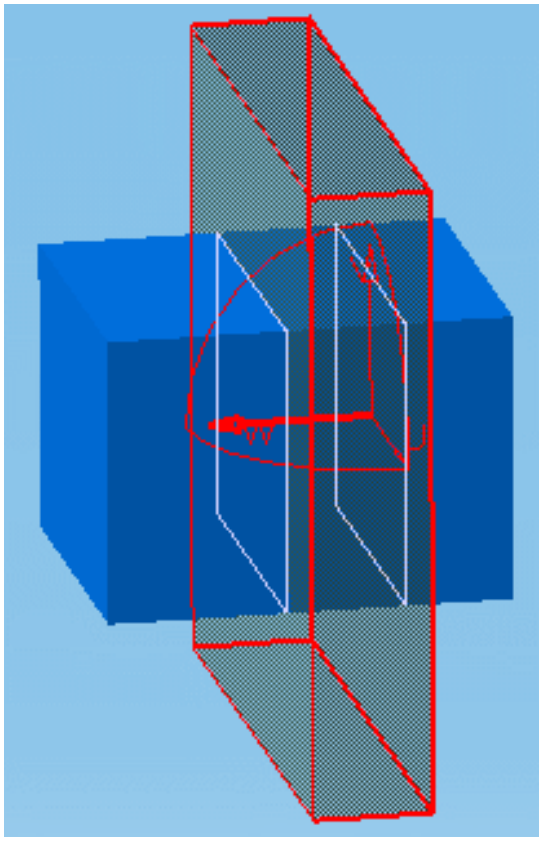
This task illustrates how to snap a section box to two planes.

No sample document is provided.

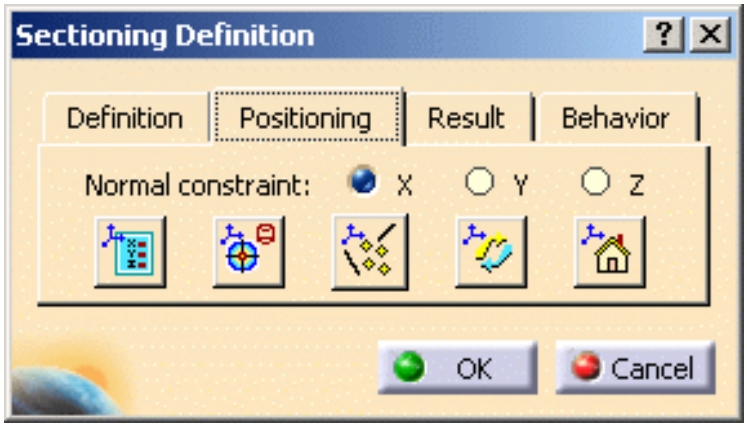
1. Select **Insert -> Sectioning** from the menu bar, or click the Sectioning icon in the DMU Space Analysis toolbar and create a section box.

   The Sectioning Definition dialog box appears.

   A Section viewer showing the generated section is automatically tiled vertically alongside the document window.



2. Click the Positioning tab, then the Geometrical Target icon to snap the box to planes.



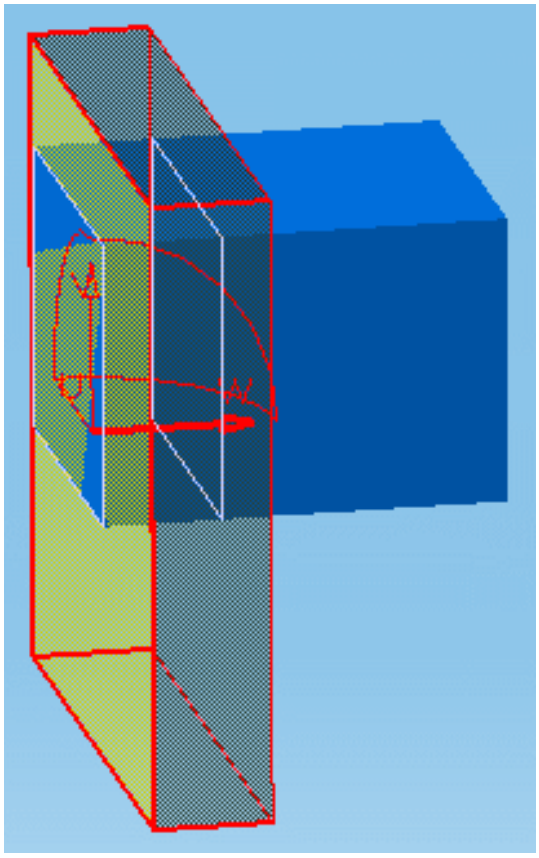
3. Point to the first plane of interest.

   The Geometrical Target command recognizes that it is a section box.

   A rectangle and vector representing a plane and the normal vector of the plane appear in the geometry area as well as the figure 1 to assist you. It moves as you move the cursor.

4. When satisfied, click to position the master plane of the section box on the first target.

   Note that the visual aid now displays the figure 2.



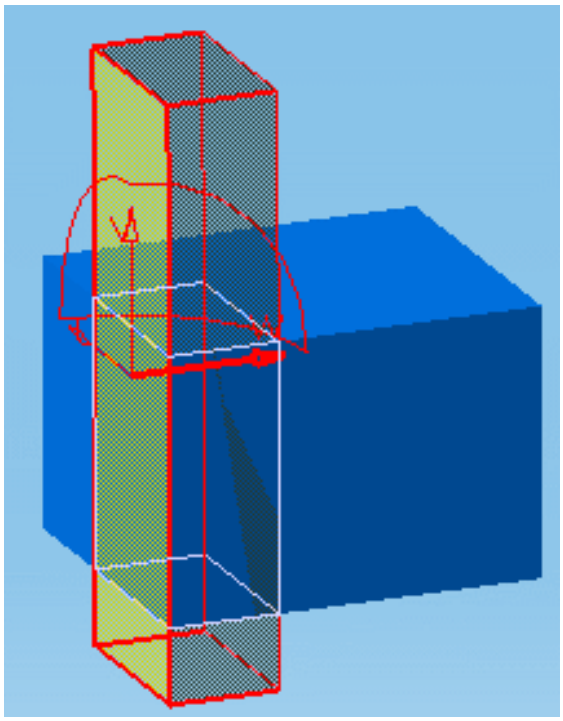
5. Select a second plane.

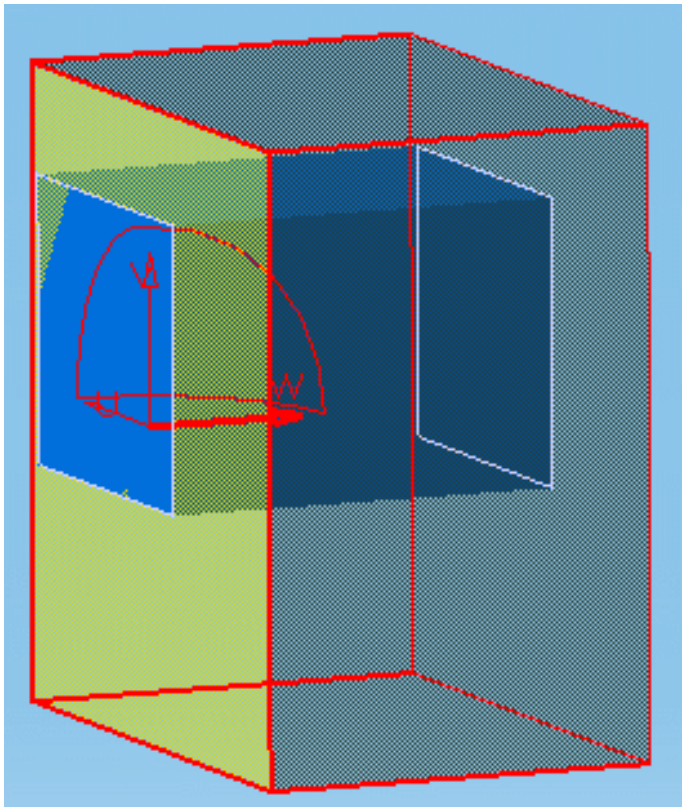
   This plane adjusts box dimensions, and if required, rotates the box.

   The section box is totally constrained to selected planes.

   The two selected planes are parallel: box thickness is modified

   The two selected planes are perpendicular: box height is modified





6. Click OK in the Sectioning Definition dialog box when done.

Snapping Planes to Points and/or Lines

You can position section planes by selecting three points, two lines, or combination of the two.

This task illustrates how to snap a section plane to a selection consisting of lines and/or points.

No sample document is provided.

1. Select **Insert -> Sectioning** from the menu bar, or click the Sectioning icon in the DMU Space Analysis toolbar and create a section plane.

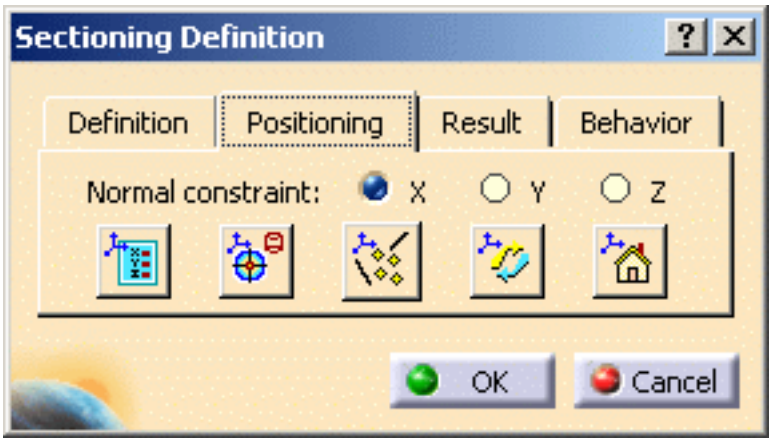
   The Sectioning Definition dialog box appears.

   A Section viewer showing the generated section is automatically tiled vertically alongside the document window.

2. Click the Positioning tab of the Sectioning Definition dialog box.

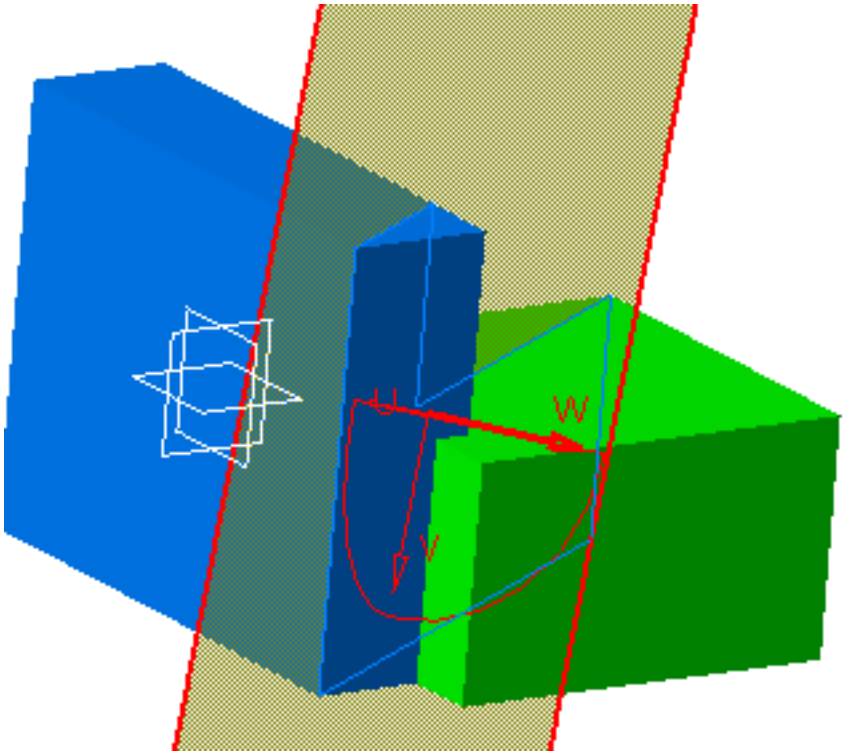
3. Click the Positioning by 2/3 Selections icon.

   The section plane is hidden.



4. Make your selection of lines and/or points.
   - The current selection is highlighted in red.
   - The cursor changes to assist you make your selection. It identifies the type of item (point, line, cylinder, cone, etc.) beneath it.

   A plane passing through the selection is computed and the section plane automatically snapped to this plane.



5. Click OK in the Sectioning Definition dialog box when done.

Exporting Section Results

You can save section results in a variety of different formats using:

- The **Export As** command. This command is particularly useful for exporting results to CATIA V4.
- The Capture command.

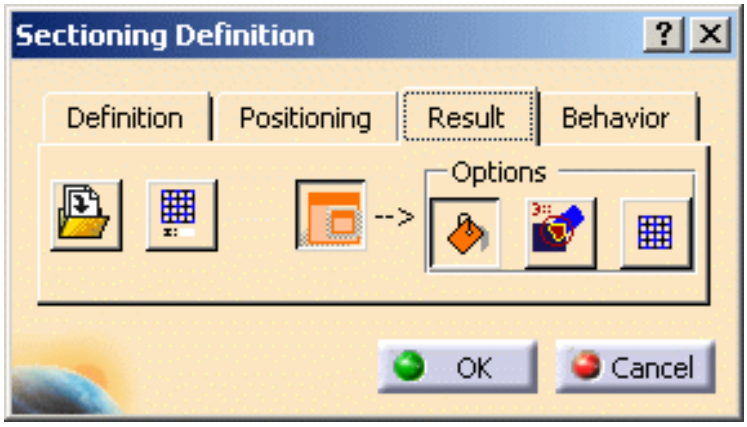
This task illustrates how to export sectioning results in a number of different formats using the **Export As** command.

Insert the following cgr files: ATOMIZER.cgr, BODY1.cgr, BODY2.cgr, LOCK.cgr, NOZZLE1.cgr, NOZZLE2.cgr, REGULATION_COMMAND.cgr, REGULATOR.cgr, TRIGGER.cgr and VALVE.cgr.

They are to be found in the online documentation filetree in the common functionalities sample folder **cfysm/samples**.

1. Select **Insert -> Sectioning** from the menu bar, or click the Sectioning icon in the DMU Space Analysis toolbar and create the desired section plane, slice or box and corresponding section.
2. Click the Result tab in the Sectioning Definition dialog box.
3. Click the Export As icon:

   The Save As dialog box appears.



4. Specify the location of the document to be saved and, if necessary, enter a file name.
5. Click the Save as type drop-down list and select the desired format.

   You can save sectioning results as:

   - a V4 model (.model)
   - a V5 CATPart (.CATPart)
   - a V5 CATDrawing (.CATDrawing)
     In this case, polylines are generated.
   - DXF and DWG formats (.dxf/.dwg)
   - a STEP document (.stp)
   - an IGES document (.igs)
   - a Virtual Reality Modeling Language (VRML) document (.wrl).

6. Click **Save** to save the results in a file in the desired format.

Note: If you set the DLName document environment (**Tools -> Options -> General -> Document**) as your current environment, clicking Export As will open the DLName dialog box instead of the usual Save As dialog box.

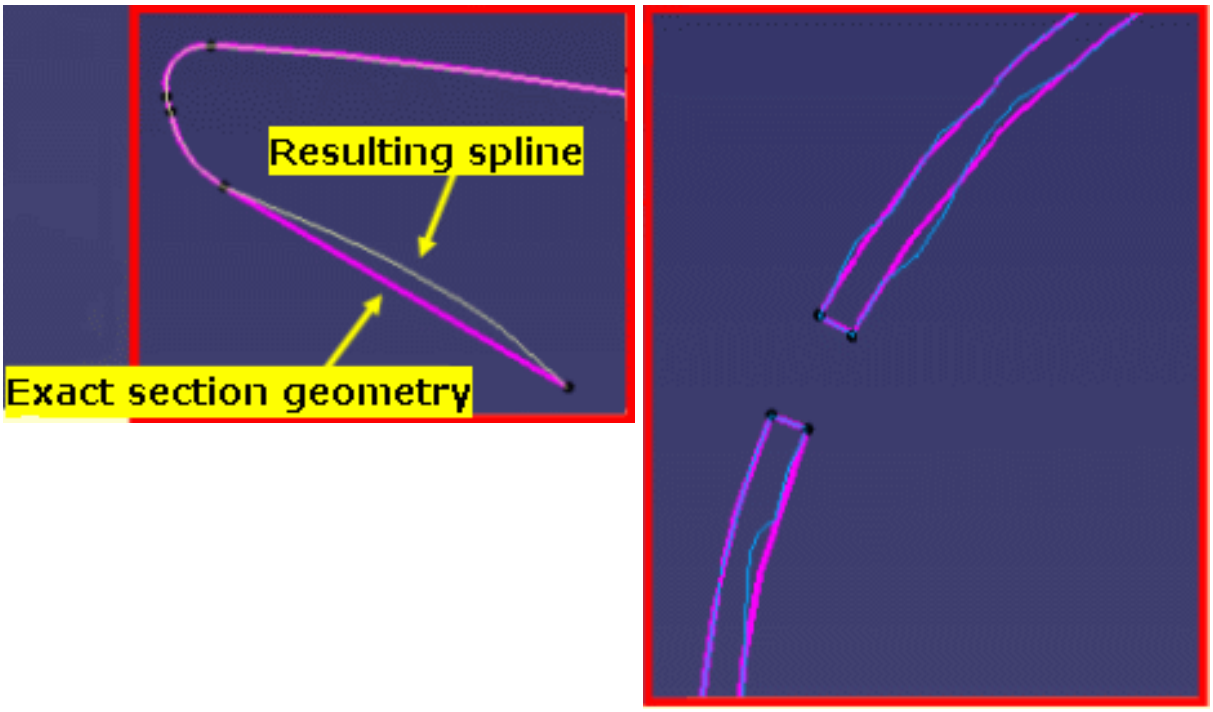
The DLName document environment lets you restrict the access to specific folders referenced by logical names referred to as 'DLNames'. Each folder is assigned a logical name. In this mode, you can only access documents in folders referenced by DLNames.

For more information on the DLName document environment, see the *Infrastructure User's Guide*.

7. Exit the Sectioning command when done.

More About the CATPart

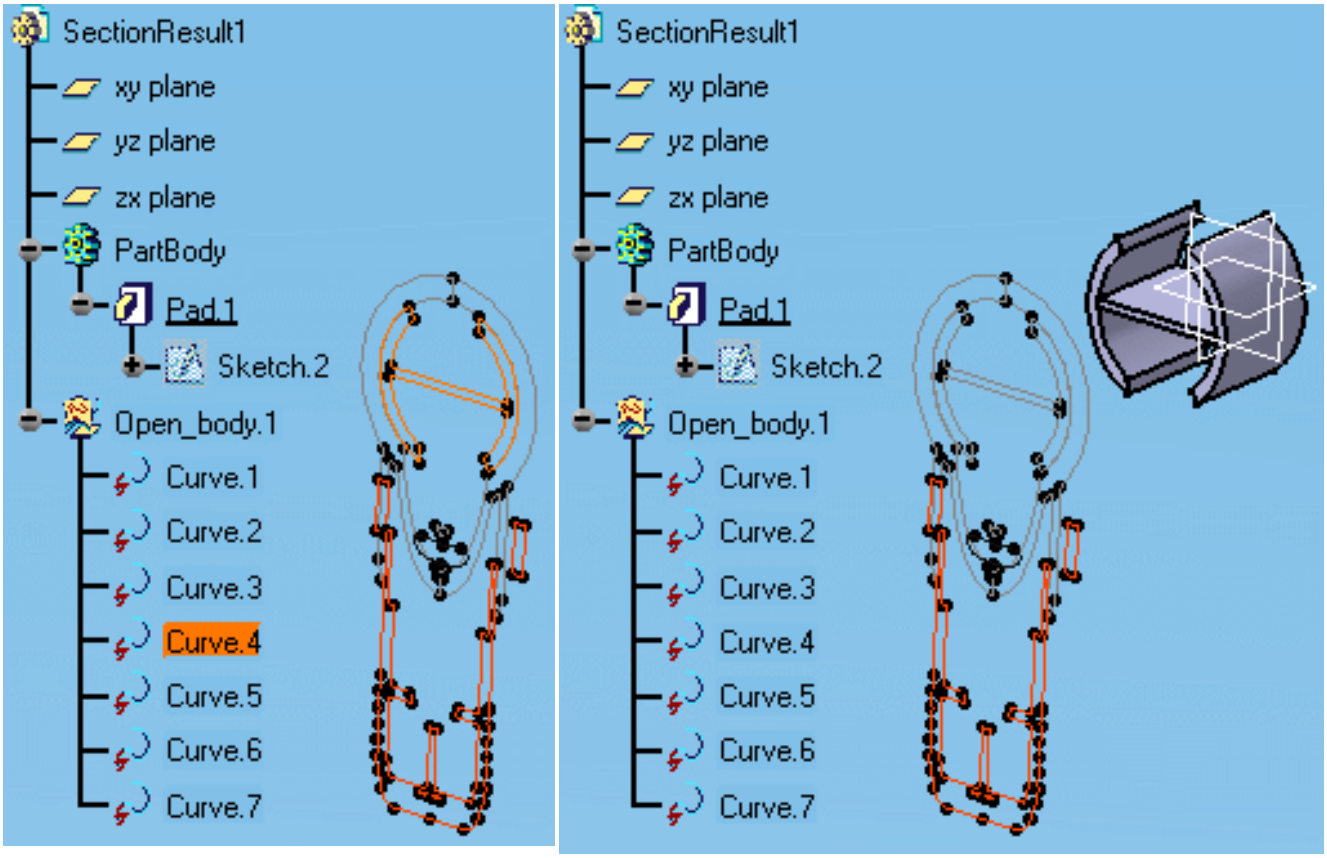
For each element sectioned, a topologically correct spline is generated under the Open_body. Each spline is obtained by interpolating all the points making up the element sectioned. The resulting spline may not be smooth.



These curves can then be used, for example, to create features:

- Select the Update icon to update your CATPart document
- Create a sketch from your curve using the Project 3D Elements command in the Sketcher workbench
- Create your feature, for example a pad.

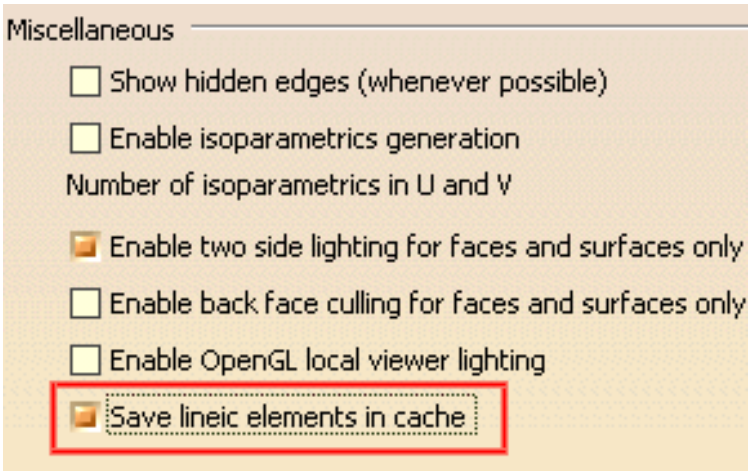
Note: Section result colors are exported when section results are saved as CATPart documents.



Working with a Cache System

If you are working with a cache system, you must select the Save lineic elements in cache check box to be able to properly save your section result in V4 model, V5 CATPart, STEP, IGES, and VRML formats. This will save wireframe section results. If you do not do so, the document will be empty.

- Save lineic elements in cache is located in the Miscellaneous box of the Performance tab, **Tools->Options->General->Display**



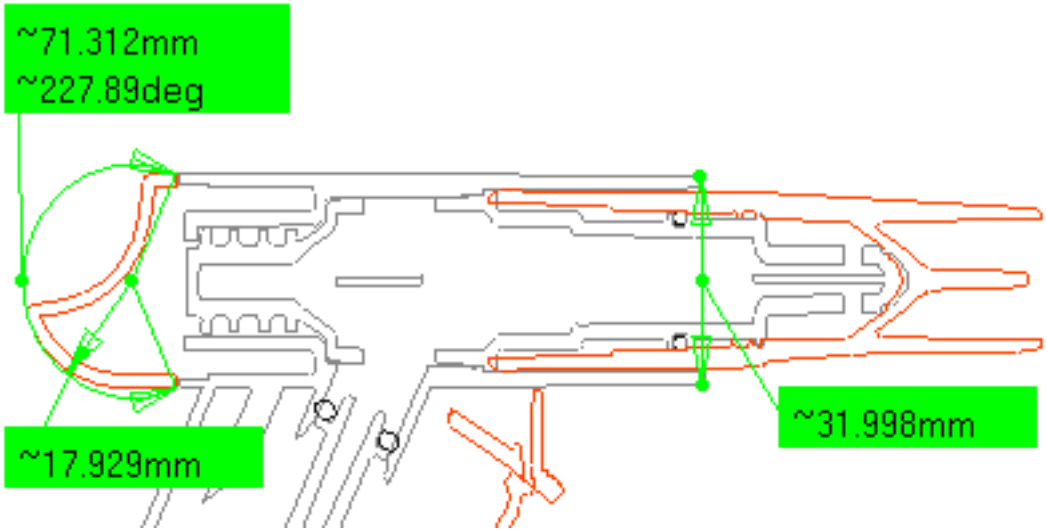
Capturing Section Results

This task illustrates how to export sectioning results to a CGM document using the **Capture** command.
For more information on this command, see the *Infrastructure User's Guide*.

1. Generate your section.
2. In the active viewer, select **Tools ->Image ->Capture.**
3. Click the Vector Mode  icon in the Capture toolbar.
4. Click the Save As  icon. Different CGM types are proposed in the Save As dialog box.
   The CGM file can then be opened (**File ->Open**) in the DMU Mockup 2D workbench and used for comparison purposes. For more information on 2D comparison functionalities, see the *DMU Navigator's User Guide*.

Annotating Generated Sections

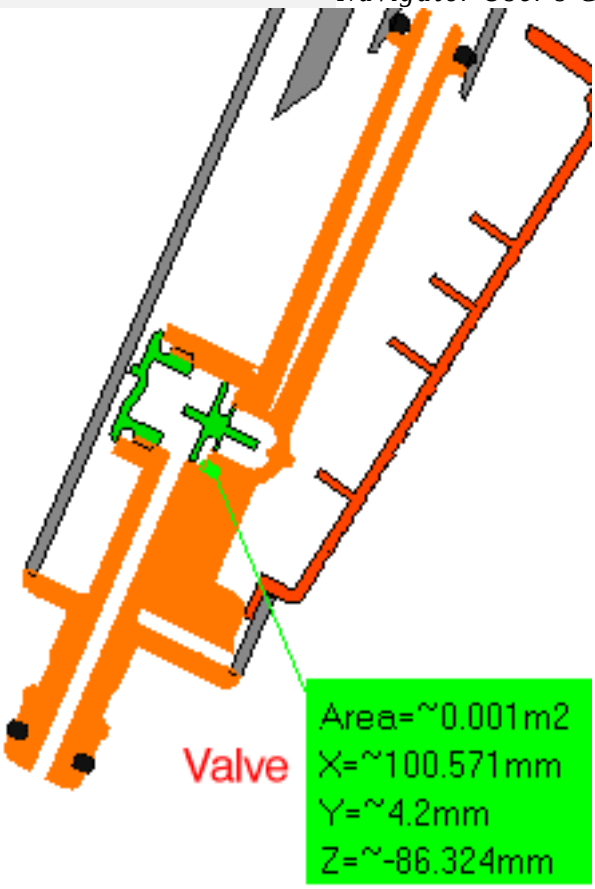
You can use generic measure tools, keeping measures as annotations, as well as 2D and 3D annotation tools to annotate generated sections in the Section viewer.



Create the desired section plane and corresponding section.

To...	Select...
Measure distances at item level	Measure Between  You can measure minimum distances between edge primitives or between sections of elements (solid of a model, body of a CATPart). You can also measure distances between points representing the intersection of the plane with any wireframe elements present and other sectioned items, however, you cannot make measures with respect to grid elements. **Note**: The Measure Between result is different depending on whether or not the section view is a filled view. If the section view is filled, the result obtained is the minimum distance between sections of selected elements. If the section view is unfilled, the result obtained is the minimum distance between edge primitives. For more information, see Measuring distances and angles between geometrical elements, and Measuring minimum distance in the Section viewer.
Measure properties	Measure Item  You can measure properties on generated sections. For more information, see Measuring properties.

Measure distances at product level	Distance and Band Analysis  in DMU Space Analysis You can measure minimum distance between the sections of selected products (model, CATPart, etc.). For more information, see Measuring minimum distances and distances along X, Y, Z, and Measuring minimum distance in the Section viewer.
Add 3D text	3D Annotation  in DMU Space Analysis For more information on 3D annotation, see the *DMU Navigator User's Guide*.
Create and annote 2D views	Creating Annotated Views  in DMU Space Analysis The Creating Annotated Views command lets you create and annotate a 2D view of your section using the arrows, lines, text, etc. provided in the DMU 2D Marker toolbar. 2D views are identified in the specification tree and can be recovered using the Managing Annotated Views  icon. For more information on 2D annotation, see the *DMU Navigator User's Guide*.



Managing the Update of Section Results

A number of options are provided to let you manage section update once you have exited the Sectioning command. This is particularly useful, for example, if you run a fitting simulation or kinematics operation that moves products affecting the section result.

These options are to be found in the Behaviour tab of the Sectioning Definition dialog box.

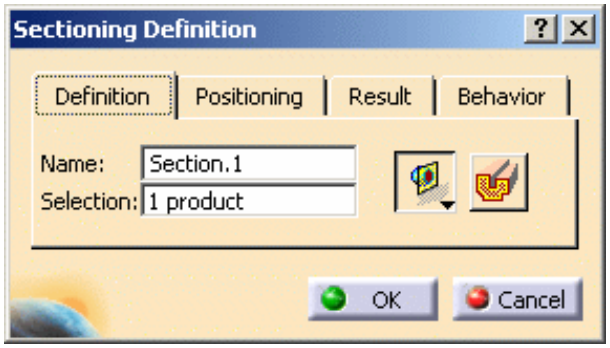
This task shows how to manage the update of section results.

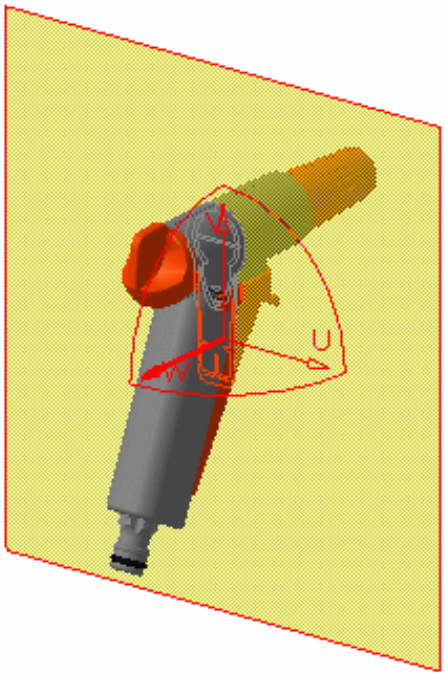
Insert the following cgr files: ATOMIZER.cgr, BODY1.cgr, BODY2.cgr, LOCK.cgr, NOZZLE1.cgr, NOZZLE2.cgr, REGULATION_COMMAND.cgr, REGULATOR.cgr, TRIGGER.cgr and VALVE.cgr.

They are to be found in the online documentation filetree in the common functionalities sample folder **cfysm/samples**.

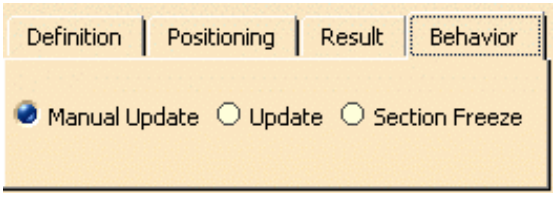
1. Select **Insert -> Sectioning** from the menu bar, or click the Sectioning icon in the DMU Space Analysis toolbar and create a section plane.

   The Sectioning Definition dialog box appears.





2. Click the Behavior tab in the Sectioning Definition dialog box.



   Three options are available in this tab:

   - Manual update (default value)
   - Automatic update
   - Section freeze

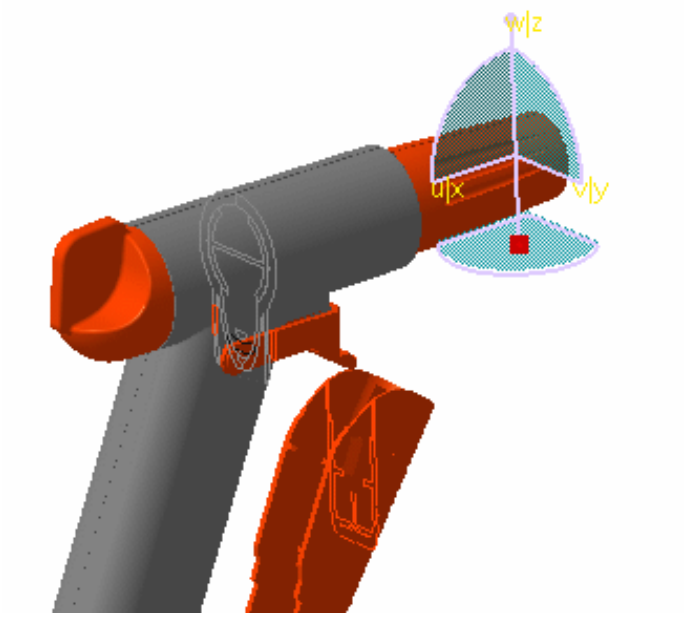
   By default, after exiting the command, the generated section is not updated when you move products affecting the section result (manual update). This, for example, will improve performance of fitting simulation and kinematics operations.

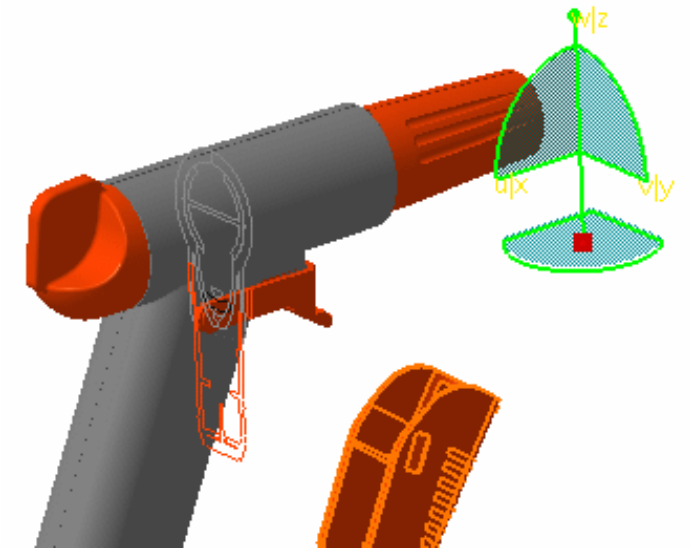
   Section results that are not up-to-date are identified by the section icon and the update symbol in the specification tree.

3. Click Automatic update to update the section automatically, after exiting the command, when you move products for example.

   In the example below, after exiting the Sectioning command, we moved a product using the 3D compass. The product was moved along the Y-axis such that it continued to intersect the section plane.

   Automatic update turned on: Automatic update turned off:

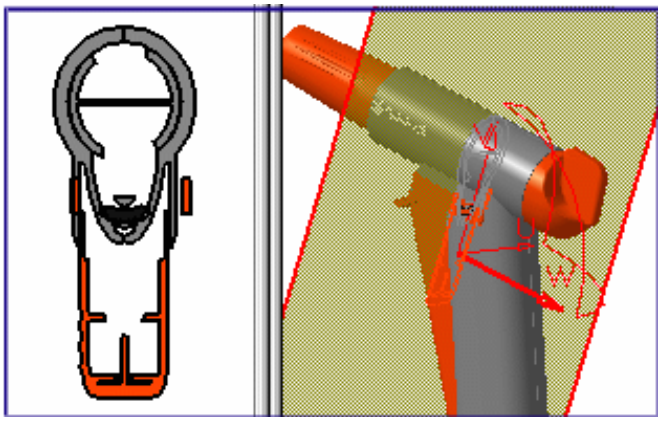




4. Click Section freeze to freeze section results.

Notes:

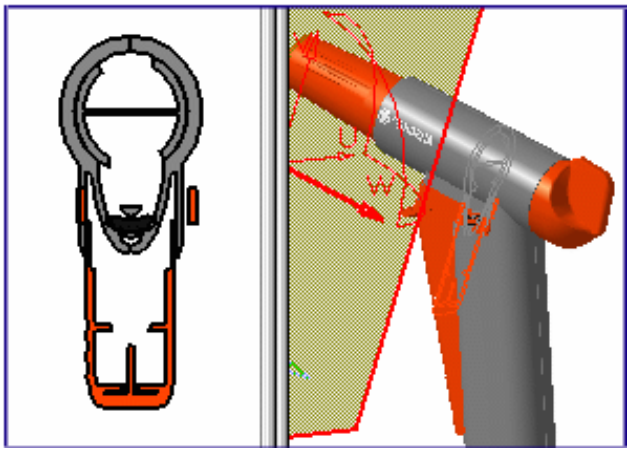
- This command takes effect immediately: section results will not be updated if you resize or move the plane, or move products affecting the result.
- Frozen section results are identified in the specification tree by the section icon plus a lock.



5. Move the section plane:

Note that the section result in neither the document window nor the Section viewer is updated.

You can in this way create a history of sectioning operations.



Frozen section results are identified in the specification tree by the section icon plus a lock .

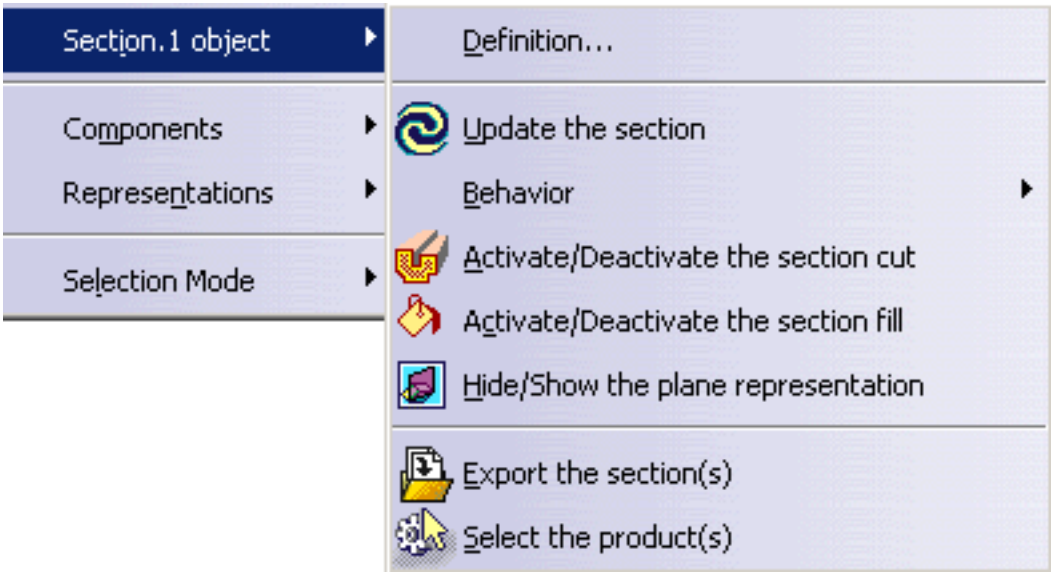
6. Reset the default option in the Behavior tab, and click **OK** in the Sectioning Definition dialog box when done.

Toggling on and off these commands can also to be done via the contextual menu.

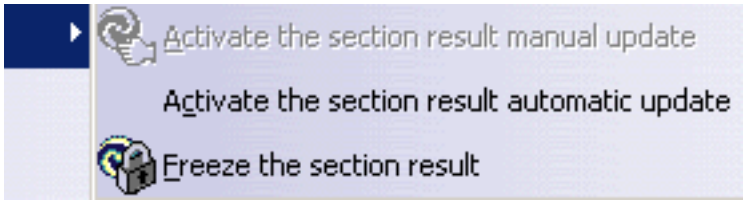
More About the Contextual Menu

The following commands are available in the contextual menu when you have exited the command.

1. Unless specified otherwise, simply right-click the specification tree feature or the section in the geometry area, select Section.1 object and then the command of interest from the menu.



- Definition...: lets you modify the selected section object.
- Update the section: locally updates the selected section.
  Note: In scene contexts, this command is labelled 'Force update the section' and updates both the scene and the geometry area to reflect modifications made to the scene.
- Behavior: lets you manage section update. These are the same options as those found in the Behavior tab of the Sectioning Definition dialog box. The greyed out option is the current option and by default, is the one set in the dialog box before exiting the command.



- Activate the section result manual update:
the generated section is not updated when you move products affecting the section result.

- Activate the section result automatic update:
the generated section is automatically updated when you move products affecting the result.

- Freeze the section result:
the generated section is not updated if you resize or move the plane, or move products affecting the result.

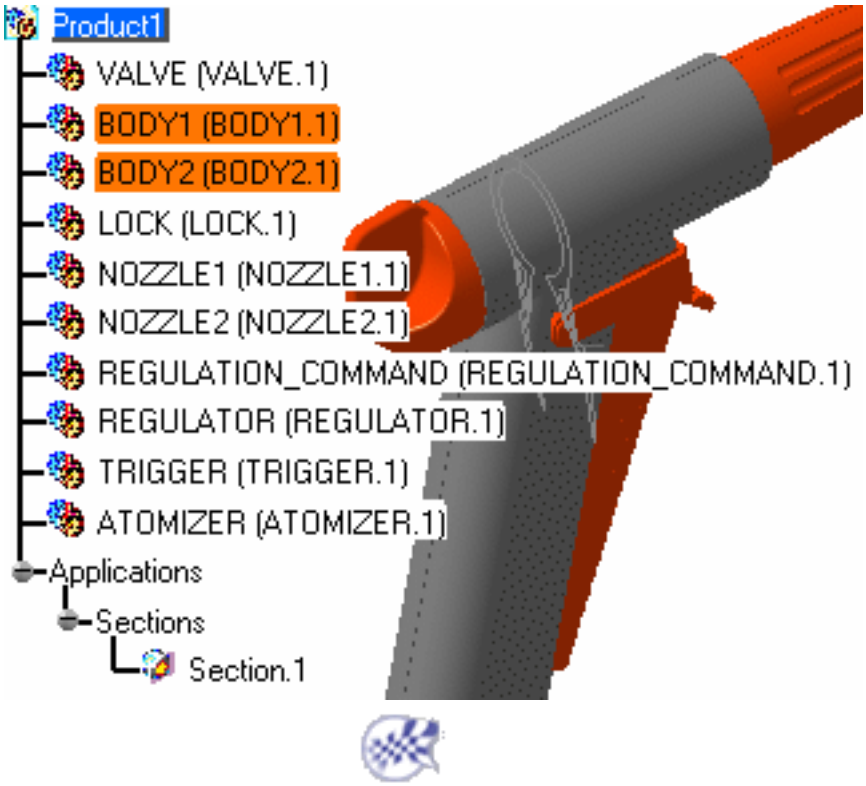
- Activate/Deactivate the section cut: turns the Volume Cut command on or off.
- Activate/Deactivate the section fill: turns the fill capability on or off.
- Hide/Show the plane representation: turns the section plane on or off.
- Export the section(s): lets you save section results in CATPart, IGES, model, STEP, VRML formats.
Note: If you want to save results as a CATDrawing, use the Export As command in the Sectioning Definition dialog box.

Note: Multiple selection tools are available for all these contextual menu commands. You can, for example, export a multiple selection of section results to a CATPart document.

- Select the product(s): highlights products in the specification tree associated with selected sections:

1. Select a section or Ctrl-click to select sections in the geometry area of the document window or in the Results window.
2. Right-click to access the contextual menu and choose Select the product(s).

Associated product(s) are highlighted in the specification tree.


Product1
VALVE (VALVE.1)
BODY1 (BODY1.1)
BODY2 (BODY2.1)
LOCK (LOCK.1)
NOZZLE1 (NOZZLE1.1)
NOZZLE2 (NOZZLE2.1)
REGULATION_COMMAND (REGULATION_COMMAND.1)
REGULATOR (REGULATOR.1)
TRIGGER (TRIGGER.1)
ATOMIZER (ATOMIZER.1)
Applications
Sections
Section.1

Interference Checking & Analysis

About interference checking & analysis: Gives general information.

Part-to-Part Clash Command

Analyze part to part clashes: Select **Analyze -> Part to Part Clash**, define the type in the Clash Detection dialog box, then select parts and click Apply.

Clash Command

Detect interferences using the clash command: Click the Clash icon, define the type in the Check Clash dialog box, then select the product(s) and click Apply.

Read clash command results: Run a check for interferences and read the global results in the Check Clash dialog box and Preview window.

Compare clash command results: gives information on how to interpret Check Clash dialog box results when comparing results.

Make a finer analysis of results of clash command results: Run a check for interferences then click More in the Check Clash dialog box and apply visualization filters to make a finer analysis of results.

View clash command results in a dedicated window: Run a check for interferences then click the Results window icon.

Export clash command results in text or model format: Run a check for interferences, then click the Export As icon.

Publish clash command results: Switch to the DMU Navigator workbench and click the Feature Publish icon.

Export clash results and geometry in XML format: Set the appropriate settings in the DMU Clash Process tab, run a check for interferences, then click the Export As icon.

Import clash results: Run a dedicated macro.

About Interference Checking & Analysis

You can check your document for clash, contact and clearance conflicts to determine whether document components interfere with each other.

Depending on your needs, you can choose between different levels of analysis, ranging from a simple detection of interferences between parts (**Analyze -> Part to Part Clash**) to a detailed clash, contact and clearance analysis of products and/or elements (**Clash** command). Note that only visible parts and products are taken into account; interference checking is not done on parts and products in the No Show space. Results differ depending on the interference type selected for the analysis.

You can analyze Clash command results using a variety of tools and visually browse interferences in a preview window or separate viewer. You can also export and publish results in text, XML and HTML formats.
As part of a standalone clash process, you can export both clash results and geometry to a dedicated file-based database. Results and geometry are managed in XML files. You can then import these results back into your session.

Creating Groups of Products

In DMU-P2, prior to running your interference analysis, you can create groups containing the product(s) you want to analyze using the Group  icon in the DMU Space Analysis toolbar or **Insert -> Group...** in the menu bar.

Groups created are identified in the specification tree and can be selected from there for the analysis. Only one group per selection can be defined.

ENOVIA Integration

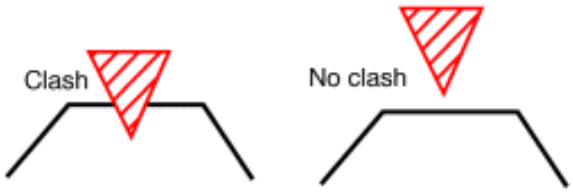
When working with ENOVIA LCA, the user can send the clash object directly from the ENOVIA query panel to CATIA. The clash result is re-created in the CATIA session. To import the products involved and visualize the clash, simply click the appropriate interference line in the Check Clash dialog box.
You can also compare clash results with those stored in ENOVIA VPM or ENOVIA LCA.

Interference Analysis Results

Interference checking and analysis is done on polyhedric models represented by triangular meshes. Results differ depending on the interference type selected for the analysis. The following illustrates expected results for the different analysis combinations.

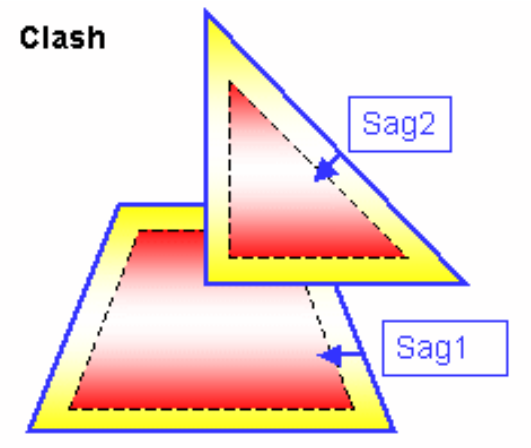
Clash

Given for information only. This option is not available.

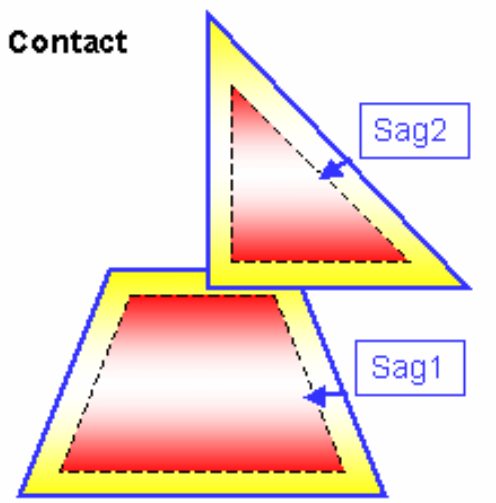


Contact + Clash

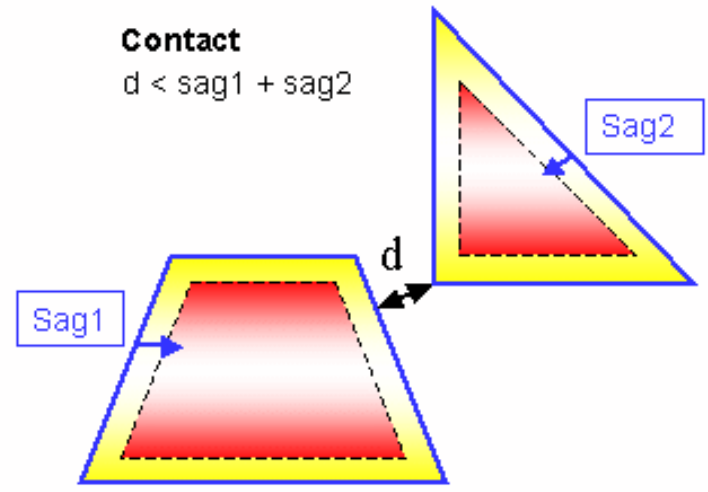
- If red zones overlap, a clash is detected.



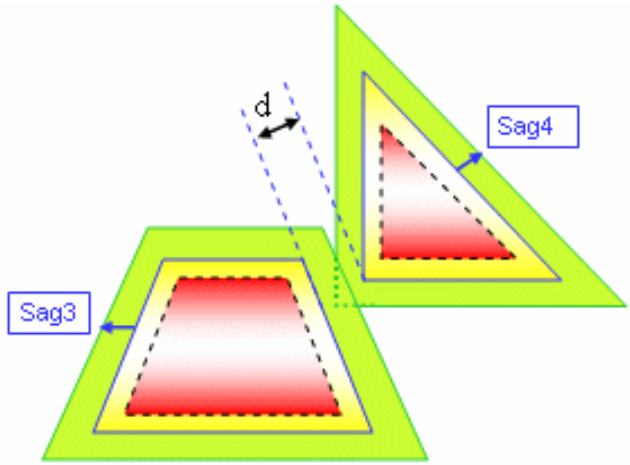
- If yellow zones only overlap, a contact is detected.



- If the shortest distance (d) between the yellow zones is less than the total sag (sag1 + sag2), a contact is detected.



Same illustration as left showing outer sag (green zone):



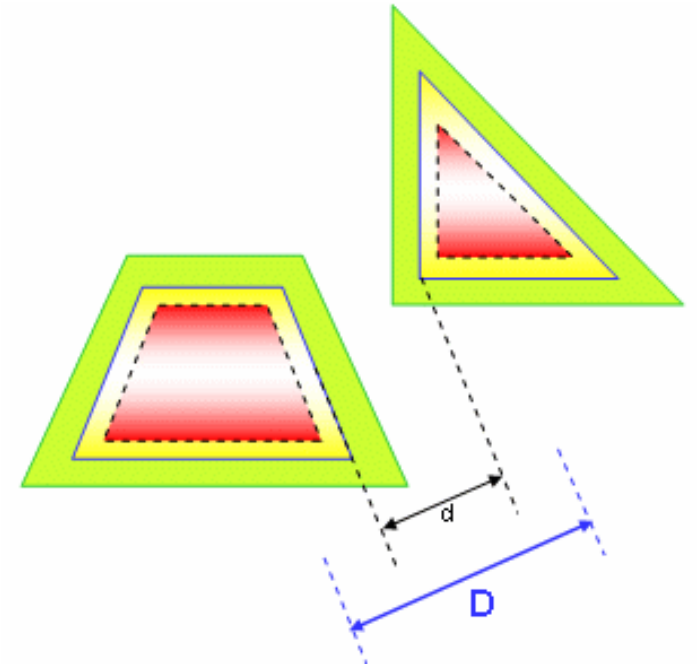
Legend for Contact + Clash illustrations:

- Blue contour: triangular mesh.
  Note: the topology is not represented.
- Dotted line: inner sag tolerance.
- Red zone: clash zone
- Yellow zone: contact zone based on inner sag
- Green zone: contact zone based on outer sag
- Yellow + Green zones: total contact zone

Clearance + Contact + Clash

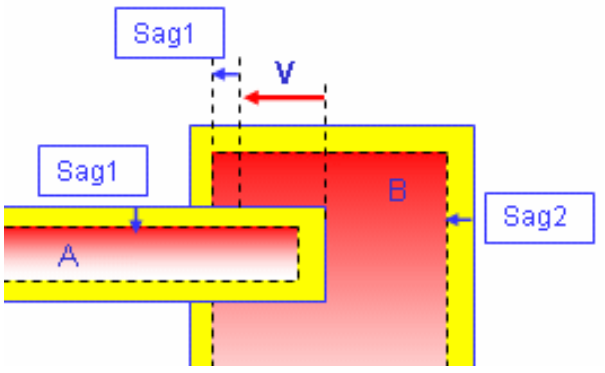
- If the total sag (sag1 + sag2) is less than the shortest distance (d), and the shortest distance (d) is less than the specified clearance distance (D), a clearance is detected.

Clearance
sag1 + sag2 < d < D

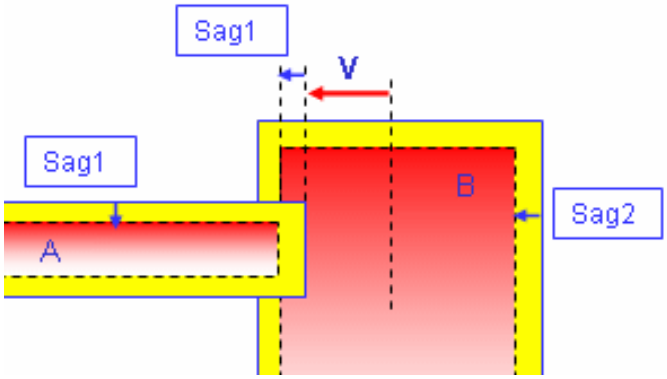


Penetration Depth

Clash



Contact

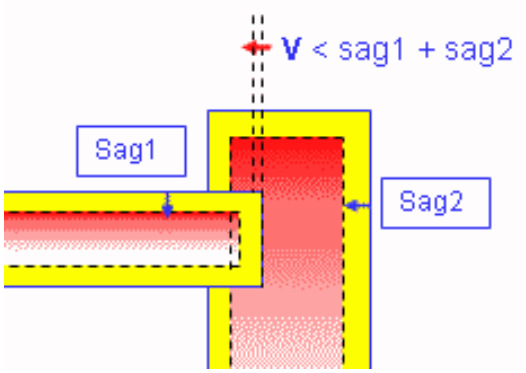


V is the penetration vector and visualizes the penetration depth.

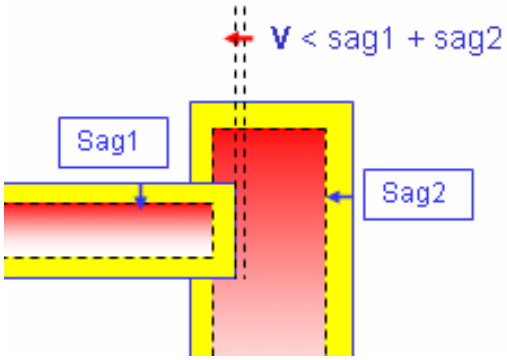
In the second illustration above, if you translate product A by V, both products are in contact.

Important: If V < sag1 + sag2, a clash may be detected (see below).

Clash

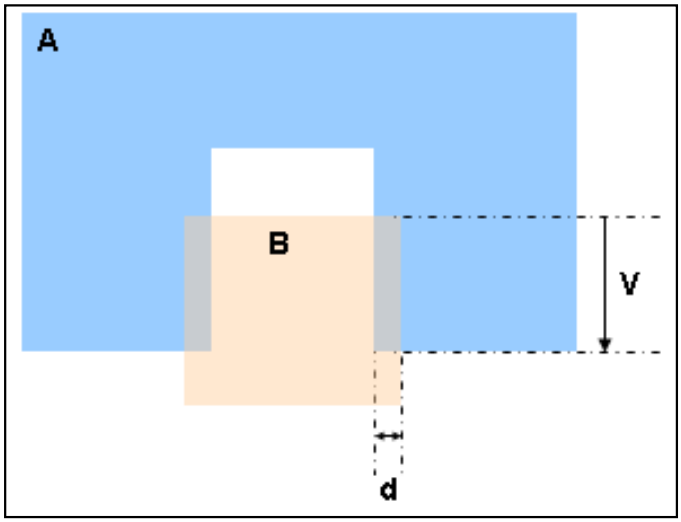


Contact



Penetration Depth versus Authorized Penetration

The penetration depth (an interference result) is not to be confused with authorized penetration (an input).



A clash is detected between products A and B.

V is the penetration vector and visualizes the penetration depth.

Note that V is greater than the authorized penetration in our illustration.

If the authorized penetration is greater than distance d, no clash is detected.

Sag

The sag corresponds to the fixed sag value for calculating tessellation on objects (3D fixed accuracy) set in the Performances tab of **Tools -> Options -> General -> Display.** By default, this value is set to 0.2 mm. The sag value set in this tab is offset from the skin inwards (blue contour) on both selection 1 and selection 2.

This value is valid for both the Part to Part Clash and the Clash commands.

In Visualization mode, you can dynamically change the sag value for selected objects using the **Tools -> Modify SAG** command.

Analyzing Part to Part Clashes

This task explains how to check for clashes and clearances between parts in your document.

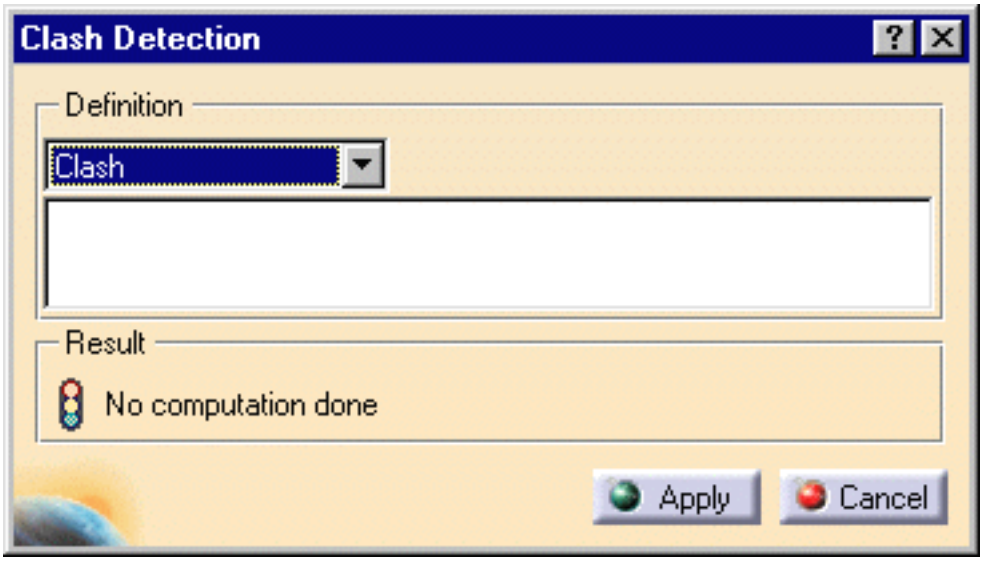
Insert the following sample cgr files: ATOMIZER.cgr, BODY1.cgr, BODY2.cgr, LOCK.cgr, NOZZLE1.cgr, NOZZLE2.cgr, REGULATION_COMMAND.cgr, REGULATOR.cgr, TRIGGER.cgr and VALVE.cgr.

They are to be found in the online documentation filetree in the common functionalities sample folder cfysm/samples.

1. Select **Analyze -> Part to Part Clash** in the menu bar.

The Clash Detection dialog box is displayed.

The default option is **Clash.**

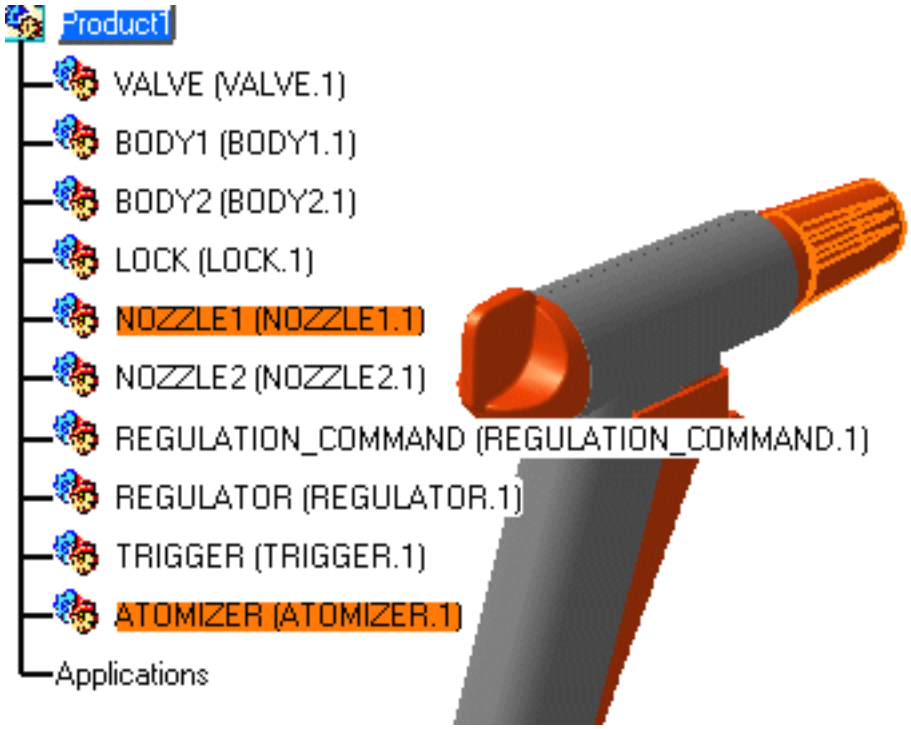


The Clash option (i.e. Contact + Clash) detects parts occupying the same space zone and parts in contact.

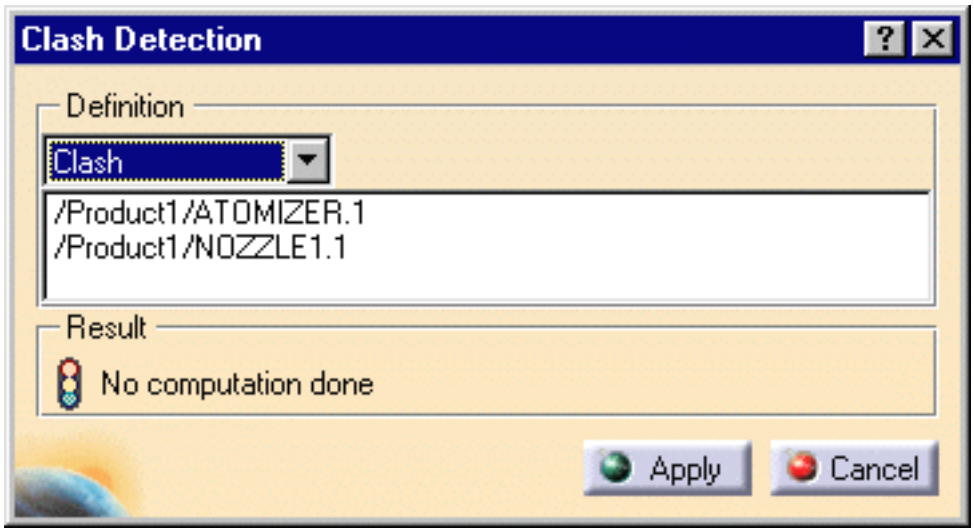
The Clearance option (i.e. Clearance + Contact + Clash) detects parts occupying the same space zone and parts in contact, and parts separated by less than the defined clearance distance.

Results differ depending on the option selected for the analysis. For more information, see figure.

2. Click the first part: Atomizer.
3. Control-click the second part: Nozzle1.

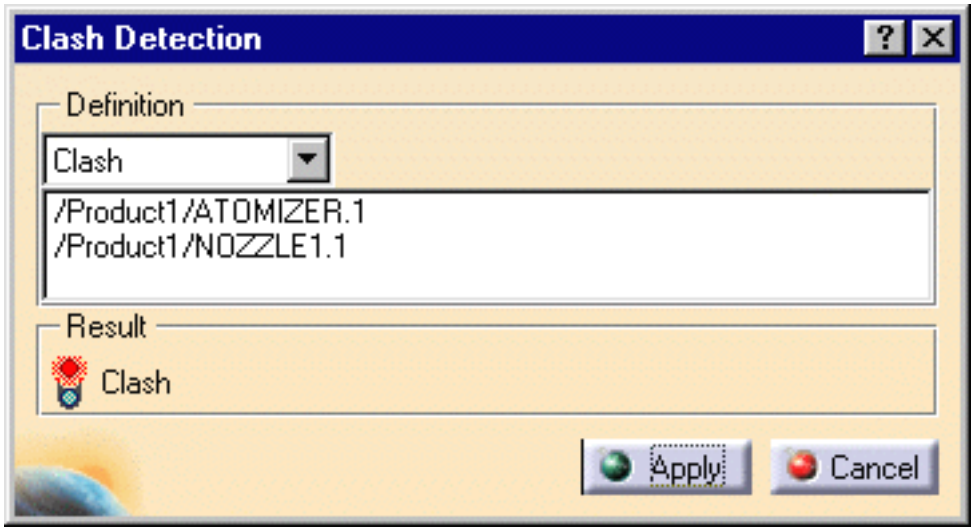


The parts are displayed in the Clash Detection dialog box.

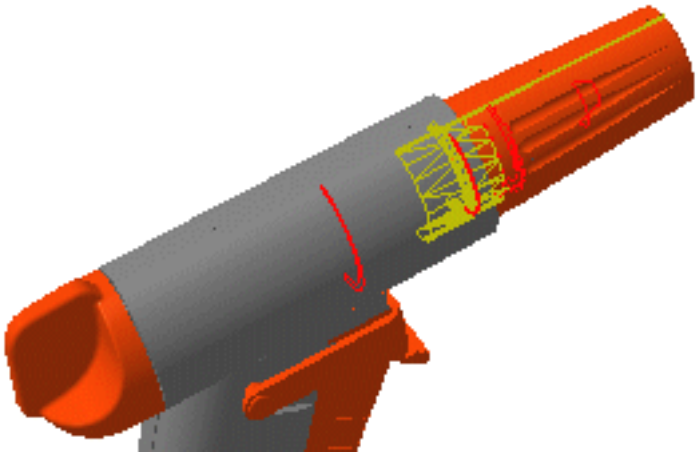


4. Click **Apply**.

The detection status icon is red and a clash has been detected.



5. Take a closer look at the geometry area:



Points at which parts clash are identified in red and those at which contact is made in yellow.

6. If desired, repeat to detect clashes and contacts between other parts.

To run clearance analyses, click the drop-down list and select Clearance (i.e. Clearance + Contact + Clash), then enter a clearance distance in the field that appears. Select parts for the analysis and click Apply.

Clearance conflicts are identified in green.

7. When done, click **Cancel**.

Detecting Interferences Using the Clash Command

Detecting Interferences

Interference checking and analysis is done on polyhedric models represented by triangular meshes. By default, checking for interferences is done in two steps and is well suited for reviews in which all conflicts do not require analysis every time.

A. Initial computation: detects and identifies the different types of interference.

In DMU-P2, you can change the default settings (DMU Clash tab) to have:

- The penetration depth and minimum distance calculated for all interferences detected during this step. This will let you sort conflicts in increasing value of penetration depth, for example, and you can then begin your analysis with the most serious conflicts.
- All numeric and graphic results defined in the Detailed Computation tab computed for all interferences detected during this step. This means that the initial computation takes longer, however, access to all results is immediate.

B. Detailed computation: computes the graphics representation of interferences (triangles for contacts and clearances, curves for clashes) as well as the penetration depth or minimum distance.

By default, penetration depth is calculated at element level. You can change this setting on the DMU Clash - Penetration tab of your settings.

In DMU-P2, you can change the detailed computation default settings (DMU Clash - Detailed Computation tab),

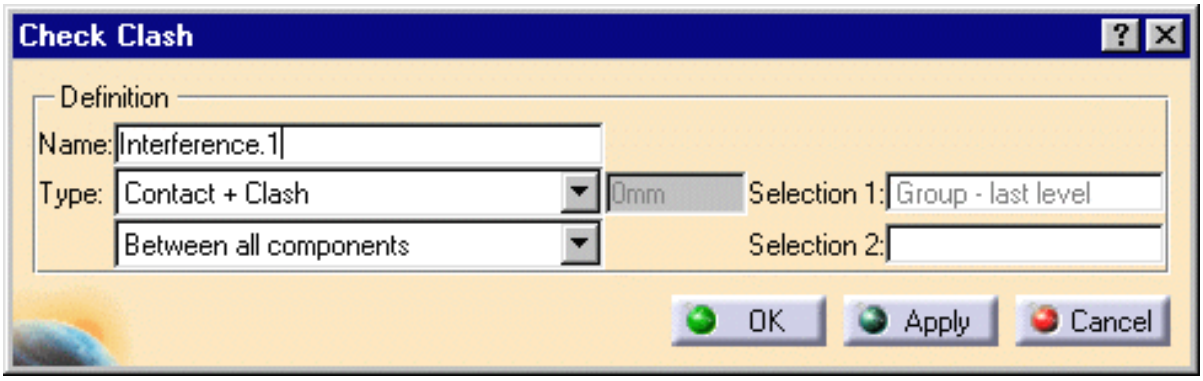
This task explains how to use the Clash command to check for interferences in your document.

Insert the following sample cgr files: ATOMIZER.cgr, BODY1.cgr, BODY2.cgr, LOCK.cgr, NOZZLE1.cgr, NOZZLE2.cgr, REGULATION_COMMAND.cgr, REGULATOR.cgr, TRIGGER.cgr and VALVE.cgr.

They are to be found in the online documentation filetree in the common functionalities sample folder cfysm/samples.

1. Select **Insert -> Clash** from the menu bar, or click the Clash icon in the DMU Space Analysis toolbar:

The Check Clash dialog box appears. An entry for the interference appears in the specification tree.



The default interference analysis is detecting clashes and contacts between all components in the document.

In DMU-P2, you can customize the default type and clearance value settings using the **Tools ->Options...** command (DMU Clash tab under **Digital Mockup ->DMU Space Analysis**).

2. Activate the first Type drop-down list box and select the interference type.

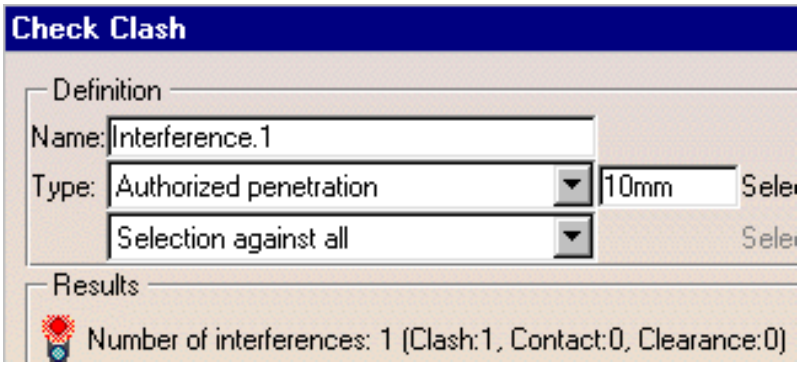
Defining Interference Types

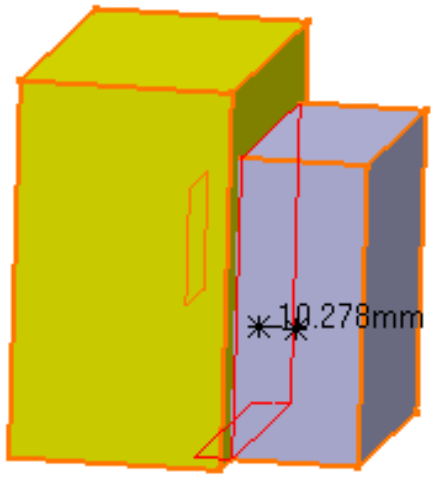
- Contact + Clash: checks whether two products occupy the same space zone as well as whether two products are in contact (the minimum distance is less than the total sag).
- Clearance + Contact + Clash: In addition to the above, checks whether two products are separated by less than the pre-defined clearance distance.

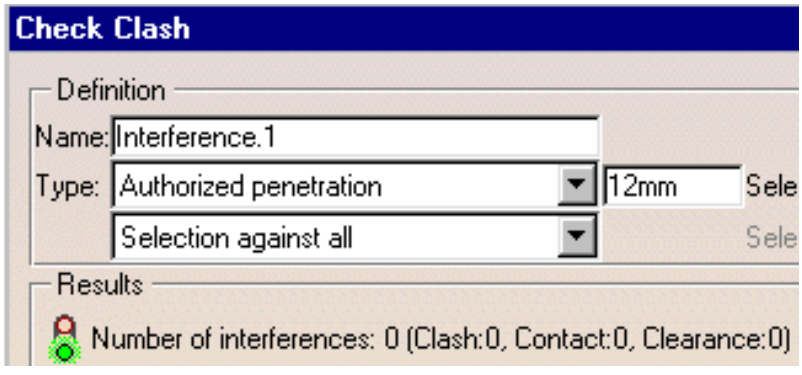
Results differ depending on the interference type selected for the analysis. See figure.

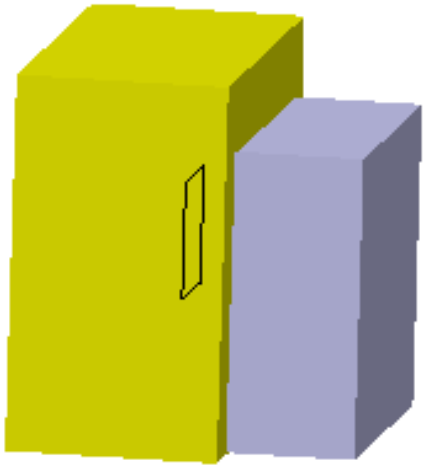
- Authorized penetration: lets you define a margin within which two products can occupy the same space zone without generating a clash. Sag is taken into account.
  Enter the value in the field that becomes active next to the interference type.
  This is particularly useful when dealing with cables, joints and other flexible objects, for example.

  Note: Do not combine this interference type with the Retrieve Information from PDM setting (**Tools -> Options, Digital Mockup -> DMU Space Analysis, DMU Clash tab**).









- Clash rule: lets you use knowledgeware capabilities in the Clash command.

3. If you set the interference type to Clearance + Contact + Clash, enter the desired clearance value in the field that becomes active next to the interference type.
4. Activate the second Type drop-down list box and select the computation type.

Defining Computation Types

- Between all components (default option): tests each product in the document against all other products.
- Inside one selection: within any one selection, tests each product of the selection against all other products in the same selection.
- Selection against all: tests each product in the defined selection against all other products in the document.
- Between two selections: tests each product in the first selection against all products in the second selection.

5. Select the products to check for interference:

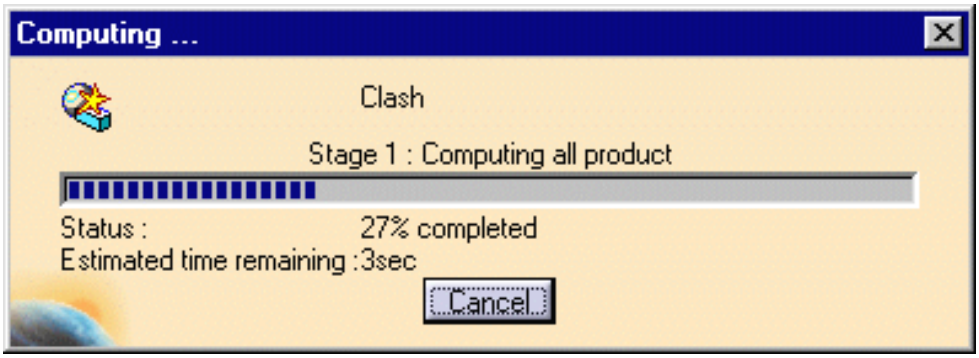
If you set the computation type to Between two selections, define the first selection then click to activate the second selection field (Selection 2) and select desired products.

Notes:

- Any sub-assembly in the specification tree is considered a valid selection.
- Continue clicking to select as many products as you want. Products are placed in the active selection. To de-select products, reselect them in the specification tree or in the geometry area.
- Click in selection fields (fields turn black) to view your selections and be certain that you have selected the products you intended: selected products are highlighted.

6. Click **Apply** to check for interferences.

A progress bar is displayed letting you monitor and, if necessary, interrupt (**Cancel** option) the calculation.



The Check Clash dialog box expands to show the results and a Preview window appears showing the products in conflict.

Note: If the calculation is canceled, no results are displayed.

Simply change the calculation parameters (interference type and selection) and click **Apply** to run another interference analysis.

Multi-CPU Computers

Advantage is taken of multi-CPU computers to distribute the clash calculation among processors, thus reducing computation time. This is only the case if the computation involves a matrix of products.

Note that in this context the progress bar does not display.

Reading Clash Command Results

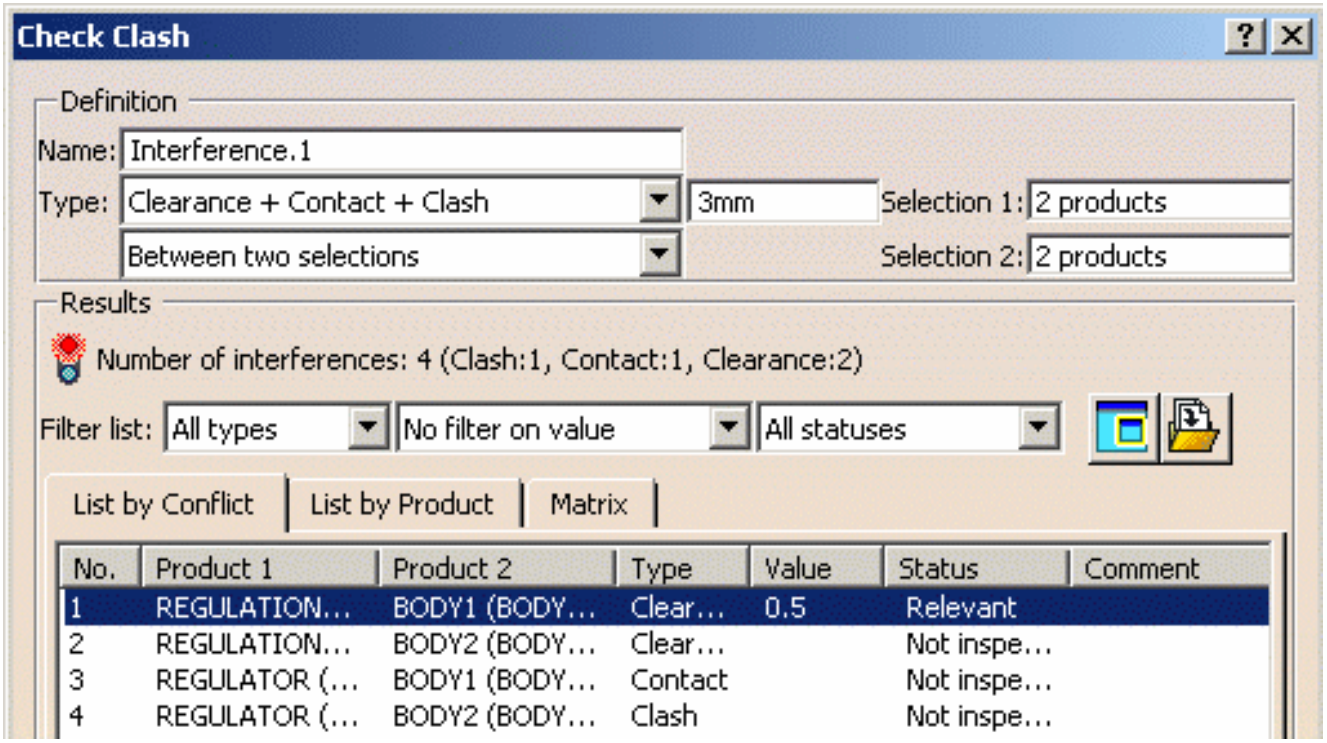
This task explains how to read the global results in the Check Clash dialog box and browse through them in the Preview window.

Insert the following sample cgr files: ATOMIZER.cgr, BODY1.cgr, BODY2.cgr, LOCK.cgr, NOZZLE1.cgr, NOZZLE2.cgr, REGULATION_COMMAND.cgr, REGULATOR.cgr, TRIGGER.cgr and VALVE.cgr.

They are to be found in the online documentation filetree in the common functionalities sample folder cfysm/samples.

1. Select **Insert -> Clash** from the menu bar or click the Clash icon in the DMU Space Analysis toolbar and run a check of type Clearance (3mm) + Contact + Clash between two selections: the Regulation_ Command and the Regulator (Selection 1), and the two body products (Selection 2).

   The Check Clash dialog box expands to show the results.



   The dialog box identifies the number of interferences detected along with the type: 4 interferences have been detected.

   Status lights are color-coded as follows:

   - red: at least one conflict is relevant
   - orange: no relevant conflicts, at least one conflict is Not inspected
   - green: all conflicts are Irrelevant.

Interference results are presented in three different ways in the dialog box:

- List by Conflict tab: lists results by conflict with one conflict per line.
- List by Product tab: lists results by product. There may be more than one conflict per product.
- Matrix tab (P2 only): presents conflicts in the form of a matrix.

By default, results are organized by conflict in the List by Conflict tab.

The first conflict in the list, in our example a clearance, is selected by default and a detailed computation has been run. The minimum distance is specified in the dialog box and both the minimum distance and green triangles identifying the products separated by less than the specified clearance distance of 3mm are displayed in the geometry area.
Naming conventions for products in conflict are the same as those used in the specification tree.

Using the **Tools -> Options -> Infrastructure -> Product Structure** command, you can customize the name, reference or description of a product or component (**Nodes Customization** tab).

P2 In DMU-P2, you can customize the display in the Results box as well as change the default setting to have the penetration depth and minimum distance or all numeric and graphic results calculated for all interferences detected during the initial computation (**DMU Space Analysis ->DMU Clash** tab).

Preview

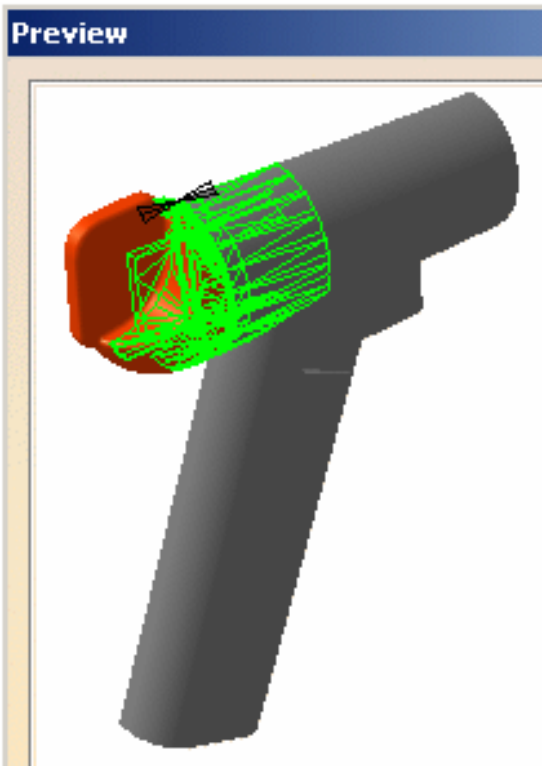
A Preview window also appears showing the products in conflict only.

Using the **Tools ->Options..., Digital Mockup** command, you can change the default display setting for the Preview window (**DMU Navigator** tab).

If you close the Preview window, to open it again:

- Click the Results window  icon in the Check Clash dialog box
  This opens the interference results window.
- Click the Results window  icon a second time
  The results window is closed and the Preview window is restored.

If however, you close the Results window (using the Close button in the title bar), the Preview window will not be restored.



Color Coding for Conflicts

- Clash: red intersection curves identify clashing products.
- Contact: yellow triangles identify products in contact.
- Clearance: green triangles identify products separated by less than the specified clearance distance.

2. If necessary, pan, zoom and/or rotate in the Preview window to visualize the interference better.
3. Select interferences in turn in the List by Conflict tab to run a detailed computation.

As you select them, the Value and Status columns in the Check Clash dialog box, and the Preview window are updated.

Detailed Computation Mode

By default, the detailed computation mode is set to Element level. Triangles identifying contacts and clearances, intersection curves identifying clashes and the penetration depth or minimum distance are computed and displayed.

Note: The graphics display and storage of the triangular representation is costly and may adversely affect performance.

In DMU-P2, you can change this setting using the **Tools ->Options...** command (DMU Clash - Detailed Computation tab under **Digital Mockup ->DMU Space Analysis**).

Computing and Displaying Contacts & Clearance Results as Surfaces:

To have contact and clearance results computed and displayed as surfaces (rather than triangles):

- Select **Tools ->Options...**, **Digital Mockup ->DMU Space Analysis,** DMU Clash - Detailed Computation tab
- Click Surface under Contact & Clearance Results
- Set the accuracy, then click **OK** in the Options dialog box.

  The accuracy entered defines the maximum value for the length of the longest side of a triangular representation of the results. A lower value will give you a more accurate result but will also require a longer computation time. The triangular representation is then used to obtain the yellow (contact) and green (clearance) surfaces.

You can also view the selected interference in a dedicated viewer. To do so, click the Results window  icon in the Check Clash dialog box.

4. Experiment with filter list capabilities and view the results.

Filtering the Display in the Dialog Box

You can filter the display of results in tabs by:

- Type: clash, contact or clearance.
- Value: no filter, increasing value or decreasing value. The penetration depth is reported in the Value field for a clash, the minimal distance for a clearance, and a zero value for a contact.
- Status: all, not inspected, relevant, irrelevant. Until selected, all interferences are reported not inspected. Inspected interferences can be relevant or irrelevant.

Note: Other columns are available when comparing results.

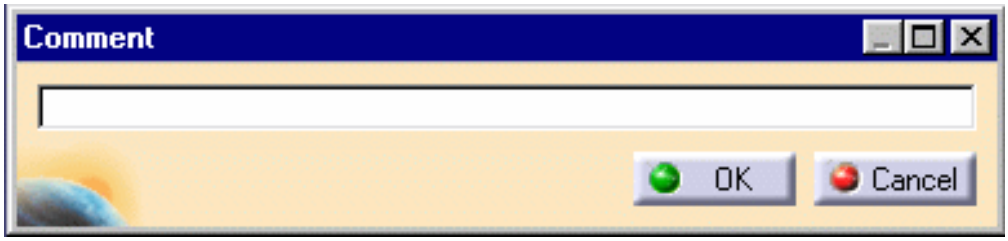
5. To change the status of an inspected conflict, click the status field of the appropriate conflict:

   The conflict status changes from relevant to irrelevant and vice-versa depending on the initial value.

   You can also add comments to selected conflicts:

6. Add a comment to the selected conflict:
   - Click the Comment field
   - Enter your comment in the dialog box that appears and click OK.

   Note: Do not use the following characters: <, >, &, /. These characters are not supported when exporting results to XML files.

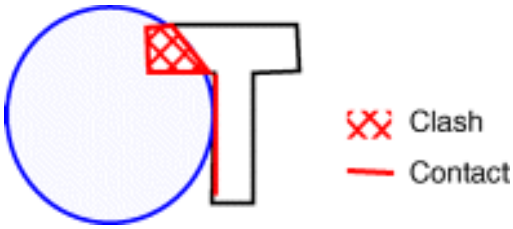


7. Click the List by Product tab to display conflicts associated with products.

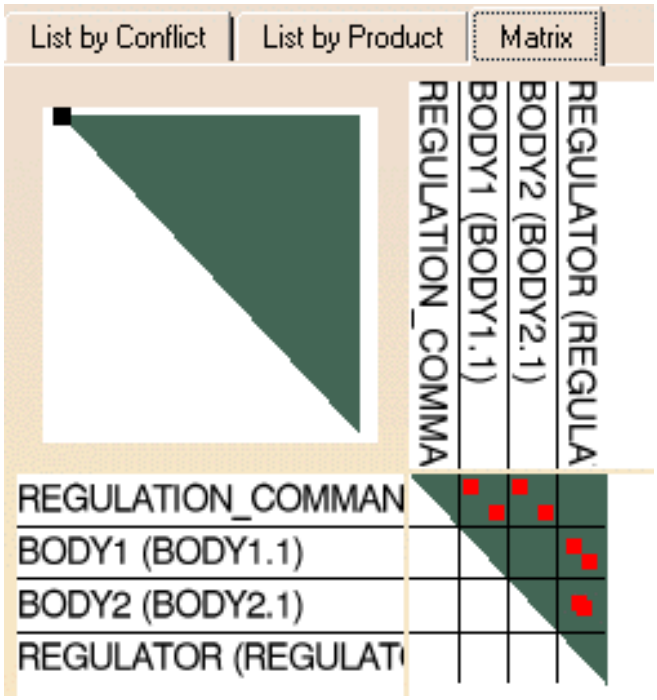
List by Conflict | List by Product | Matrix

No.	Product 1	Product 2	Type	Value	Status
1	REGULATION...	BODY1 (BODY...	Clear...	0.5	Relevant
2		BODY2 (BODY...	Clear...	0.5	Relevant
1	BODY1 (BODY...	REGULATION...	Clear...	0.5	Relevant
3		REGULATOR (...	Contact	0	Relevant
2	BODY2 (BODY...	REGULATION...	Clear...	0.5	Relevant
4		REGULATOR (...	Clash	-0.95	Relevant
3	REGULATOR (...	BODY1 (BODY...	Contact	0	Relevant
4		BODY2 (BODY...	Clash	-0.95	Relevant

   Results are organized by product in the List by Product tab. There may be more than one conflict per product.



8. Click the Matrix tab to display conflicts in the form of a matrix.



Color Coding and Symbols Used in Matrix tab

- Clash:
- Contact:
- Clearance:
- Current conflict: highlighted box around current conflict
- Not inspected: yellow
- Relevant: red
- Irrelevant: green.

To change the status of a conflict, click a current conflict again or right-click and select Change status from the contextual menu.

To add a comment, right-click a current conflict and select Comment from the contextual menu.

The area in the top left-hand corner is a preview area representing the matrix.

- Zoom (press and hold down the middle mouse button, then click the left mouse button and drag, still holding the middle mouse button down) in the matrix itself to get a better view of the results.
- Drag the square symbol in the preview area to home in on the result of interest.

See Finer analysis to find out how visualization filters work and how to read detailed results.

You can also export and publish clash command results in a variety of formats.

9. Click **OK** to exit when done.

Both the interference results and the definition are kept as specification tree features.

The numeric value (minimum distance or penetration depth) of the current result is kept when you exit the command. If you subsequently edit the results, the system re-displays the last current result analyzed.

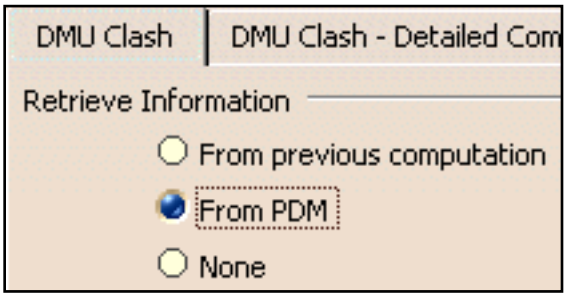
Note: Clicking Deselect in the Check Clash dialog box removes the current results from the selection.

Comparing Clash Command Results

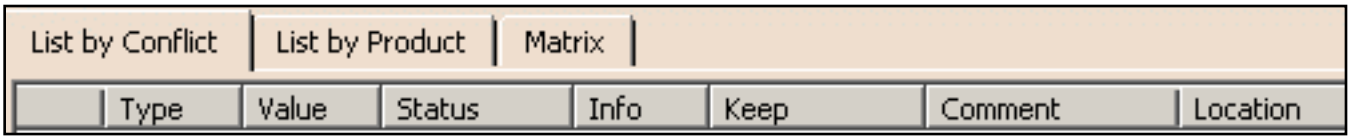
You can compare clash command results with a previous computation, or with results stored in a PDM system (ENOVIA VPM or ENOVIA LCA).

For interoperability to work correctly, ensure that you have set up both your CATIA V5 and ENOVIA LCA or ENOVIA VPM sessions properly.

Before running your check, set the appropriate Retrieve Information option in the **DMU Clash** tab (**Tools -> Options..., Digital Mockup -> DMU Space Analysis**).



Additional information is provided in the Check Clash dialog box to help you analyze results.



Info Column

New:
: The conflict is new.

Old:
: The conflict is identical to an existing conflict, either previously computed or in the PDM system.

Modified:
: The conflict already exists, but changes to product geometry were made in the PDM system after initial clash results were stored.
All modified results are given the not inspected status.

Modified_old:
: The conflict already exists, but changes to product geometry do not impact stored clash results.

Note: You can filter results in this column.

Keep Column (Retrieve Information from PDM only)

This column lets you define which results you want to store in the PDM database.

No value:
No decision to keep or otherwise results has been made.
Keep:
Clash results will be stored in the PDM database when you exit the command (**OK** in the Check Clash dialog box).
No Keep:
Clash results will not be stored in the PDM database.

Cannot Keep:
Clash results cannot be stored in the PDM database. The geometry or position of at least one of the two conflicting products has changed. You must first save products in the PDM system before saving clash results.

Note: Simply click the Keep field of the appropriate conflict to toggle to No keep.

Location Column (Retrieve Information from PDM only)

This column distinguishes between database and session conflicts.

Base:
The conflict in the session and the one stored in the PDM system are identical. There has been no change in product geometry or position of either of the two conflicting products. Nor has there been any change in conflict attributes (status, comment, etc.).
Session:
The conflict in the session is different from the one stored in the PDM system: the geometry or position of one of the two conflicting products, or one of the conflict attribues (status, comment, etc.) has changed.

Note: These two values are managed by the system. You cannot modify them directly.

Making a Finer Analysis of Clash Command Results

This task explains how to apply Clash command visualization filters and read detailed results.

Insert the following sample model files: ATOMIZER.model, BODY1.model, BODY2.model, LOCK.model, NOZZLE1.model, NOZZLE2.model, REGULATION_COMMAND.model, REGULATOR.model, TRIGGER.model and VALVE.model.

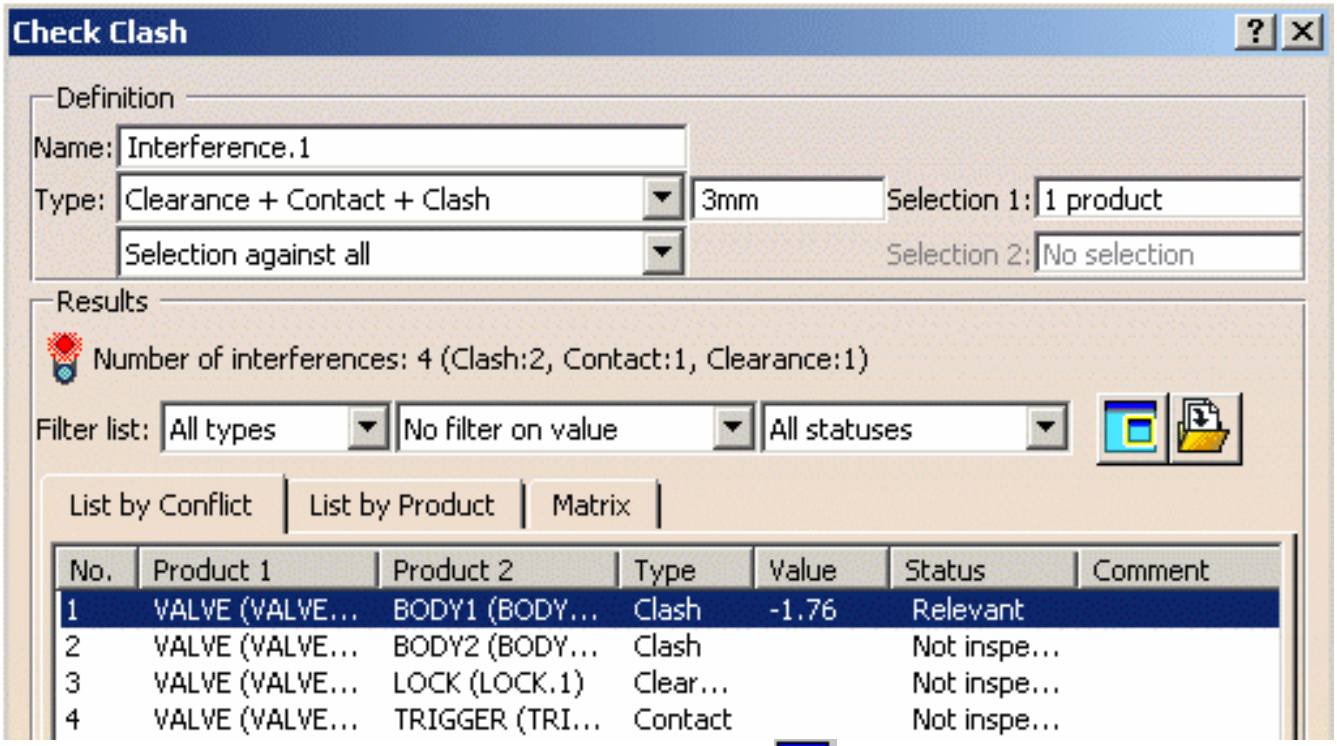
They are to be found in the online documentation filetree in the common functionalities sample folder cfysm/samples.

Switch to Design Mode (Edit -> Design Mode).

1. Select **Insert -> Clash** from the menu bar or click the Clash  icon in the DMU Space Analysis toolbar to run a check of type Clearance (3mm) + Contact + Clash between the valve and all the other products in the document (Selection against all).

To enable a finer analysis at element level, ensure that the detailed computation mode is set to Element level. To do so, select the **Tools ->Options..., Digital Mockup** command (DMU Space Analysis ->DMU Clash Detailed Computation tab).
Element level is the default detailed computation mode. Triangles identifying contacts and clearances, intersection curves identifying clashes and the penetration depth or minimum distance are computed and displayed.
Note: The graphics display and storage of the triangular representation is costly and may adversely affect performance.
The Check Clash dialog box expands to show the results of the initial computation. The number of interferences detected along with the type is indicated in the dialog box: 4 interferences have been detected.

Note: The results given in the various tabs of the dialog box correspond to the worst case.

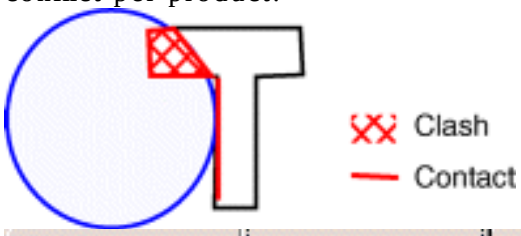


2. Close the Preview window and click the Results window icon in the Check Clash dialog box to view the selected interference in a dedicated viewer.
The Interference Results window appears displaying the products in conflict only.

To open the Preview window again, click the Results window icon a second time. The results window is closed and the Preview window is restored.

If however, you close the Results window (using the Close button in the title bar), the Preview window will not be restored.

3. Click the List by Product tab to display conflicts associated with products.
Results are organized by product in the List by Product tab. There may be more than one conflict per product.



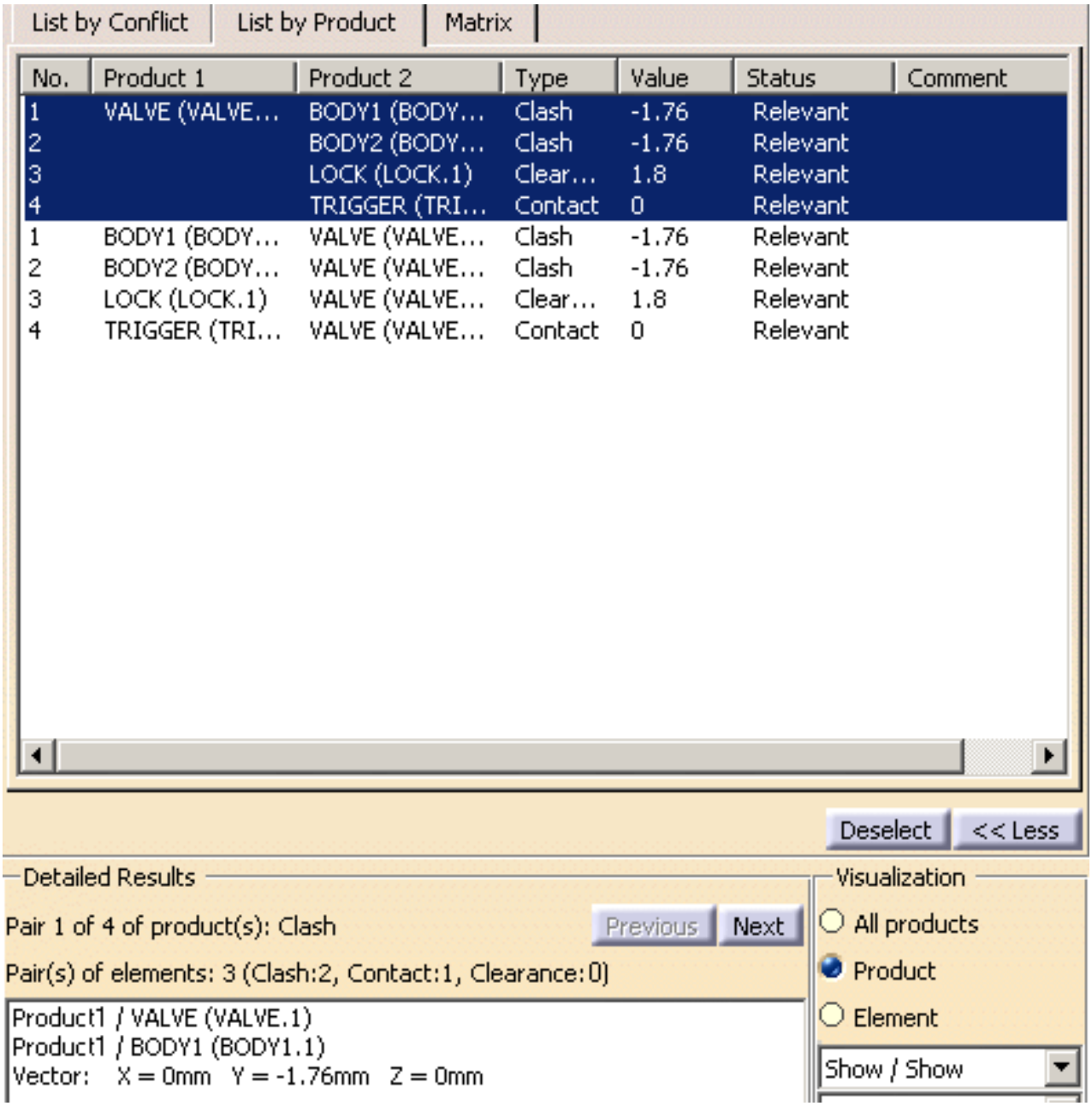
List by Conflict | List by Product | Matrix

No.	Product 1	Product 2	Type	Value	Status
1	VALVE (VALVE...	BODY1 (BODY...	Clash	-1.76	Relevant
2		BODY2 (BODY...	Clash		Not inspe...
3		LOCK (LOCK.1)	Clear...		Not inspe...
4		TRIGGER (TRI...	Contact		Not inspe...
1	BODY1 (BODY...	VALVE (VALVE...	Clash	-1.76	Relevant
2	BODY2 (BODY...	VALVE (VALVE...	Clash		Not inspe...
3	LOCK (LOCK.1)	VALVE (VALVE...	Clear...		Not inspe...
4	TRIGGER (TRI...	VALVE (VALVE...	Contact		Not inspe...

4. Select the product Valve.1, then More>> for a finer analysis.

The dialog box expands to include Detailed Results and Visualization boxes.

5. Click Product in the Visualization box to obtain an analysis by individual product.
The first conflict (a clash) between the valve.1 and body1.1 is identified in the Detailed Results box and is visible in the Results window.



X,Y,Z coordinates of the penetration vector are given.

The origin of the penetration vector is the center of inertia of the intersection curve.

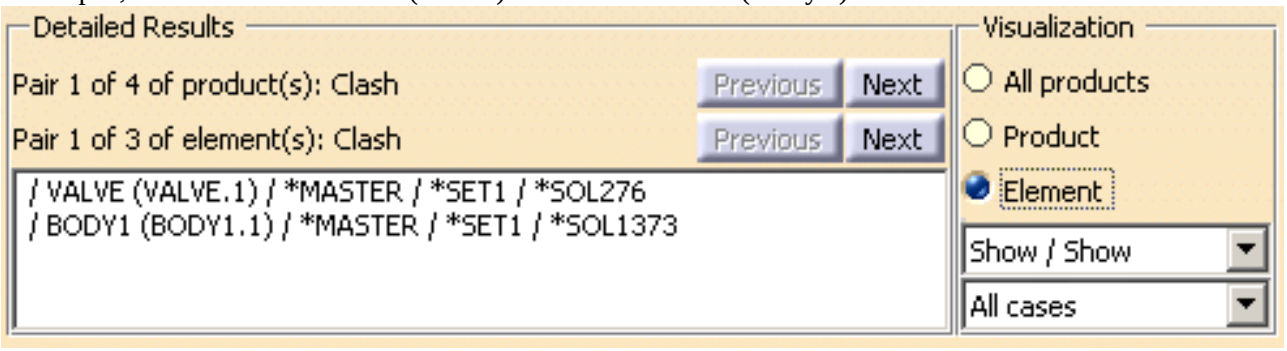
- Note that this vector may sometimes be located outside selected products.
- Special case: when one product is completely included in another, the origin of the penetration vector is the center of the local axis system of one of the products.

If an intersection volume is requested (**Tools ->Options...**, **Digital Mockup ->DMU Space Analysis,** DMU Clash - Detailed Computation tab), the volume is also given.
The valve product is in conflict with four other products (body1, body2, the lock and trigger). Previous and next icons in the Detailed Results box let you browse through them.

Note: The detailed computation has also identified three different pairs of elements within the first pair of products (valve and body1).

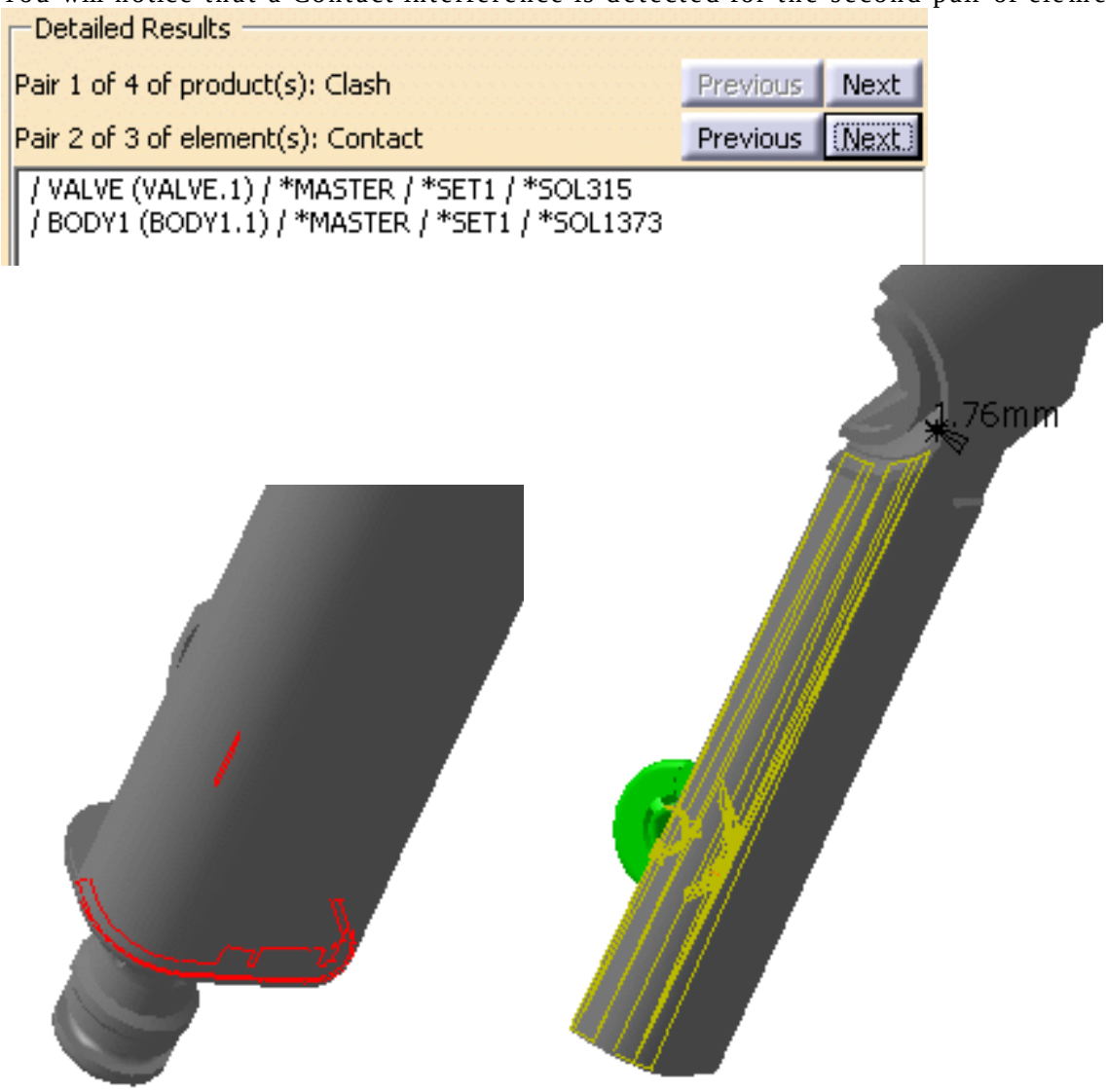
6. Click Element in the Visualization box to obtain an analysis at element level.

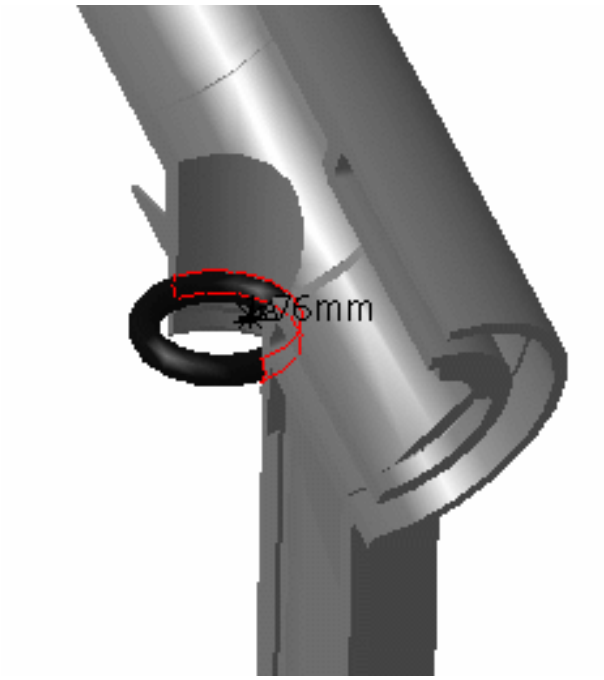
The Detailed Results box is updated and identifies the elements involved in the conflict. In our example, elements *SOL276 (Valve) and *SOL1373 (Body1).



7. Click Next to view detailed results for other pairs of elements in the same two products.

You will notice that a Contact interference is detected for the second pair of elements.





Using Visualization Filters

- All products: displays all products selected in the list.
- Product: displays products involved in the current conflict.
- Element: displays elements involved in the current conflict. Pairs of elements are listed in the Detailed Results box and can be browsed using the Next and Previous options.
  Note: The Element option is only available if the detailed computation mode is set to Element level. This option is particularly useful when in Design mode.
- Show / Show drop-down box: a variety of combinations lets you show, hide or place in low light the geometries of the two products involved in the conflict. This option has no effect in the Preview window.
- All cases drop-down box: lets you select the graphics representation you want to see displayed (All cases, None or Worst cases). If Element is selected, the Worst case option is not available.

Note: The effects of all options, except the Show / Show drop-down box, can be seen in the Preview window.

8. Browse through other pairs of elements in other pairs of products using Next and Previous icons.

You can export as well as publish clash results in the following formats:

- Text
- V4 model
- XML
- HTML.

9. Click **OK** to exit when done.

The interference definition and results are kept as specification tree features.

The numeric value (minimum distance or penetration depth) of the current result is kept when you exit the command. If you edit the results, the system re-displays the last current result analyzed.

Note: Clicking Deselect in the Check Clash dialog box removes the current results from the selection.

Viewing Clash Command Results in a Dedicated Window

This task explains how to view selected interferences in a separate viewer. It also explains how to use the Show / Show visualization filter (P2 only).

Insert the following sample cgr files: ATOMIZER.cgr, BODY1.cgr, BODY2.cgr, LOCK.cgr, NOZZLE1.cgr, NOZZLE2.cgr, REGULATION_COMMAND.cgr, REGULATOR.cgr, TRIGGER.cgr and VALVE.cgr.

They are to be found in the online documentation filetree in the common functionalities sample folder cfysm/samples.

1. Select **Insert -> Clash** from the menu bar or click the Clash  icon in the DMU Space Analysis toolbar to run a check of type Contact + Clash inside one selection comprising Valve and Body1.

   The Check Clash dialog box expands to show the results of the initial computation. The first interference is selected by default and a detailed computation has been run.

   Note: The results given in the various tabs in the dialog box correspond to the worst case.
2. Close the Preview window.
3. Click the Results window  icon in the Check Clash dialog box to view the selected interference in a dedicated viewer.

   The Interference Results window opens. By default, the detailed computation mode is set to Element level. Triangles identifying contacts and clearances, intersection curves identifying clashes and the penetration depth or minimum distance are computed and displayed.

   Object viewing commands and commands in the Window menu are available in this window. You can, for example, tile the Interference results window and the original document window vertically or horizontally.

Color Coding for Conflicts

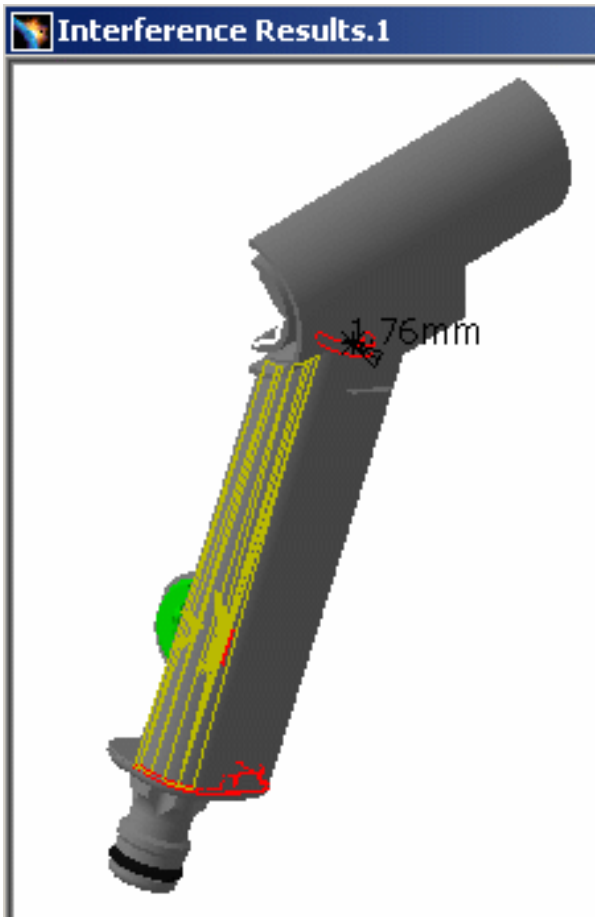
- Clash: red intersection curves identify clashing products.
- Contact: yellow triangles identify products in contact.
- Clearance: green triangles identify products separated by less than the specified clearance distance.

Computing and Displaying Contact and Clearance Results as Surfaces

In DMU-P2, you can compute and display contact and clearance results as surfaces (rather than triangles). To do so:

- Select **Tools ->Options...**, **Digital Mockup ->DMU Space Analysis,** DMU Clash - Detailed Computation tab
- Click Surface under Contact & Clearance Results
- Set the accuracy, then click **OK** in the Options dialog box.

  The accuracy entered defines the maximum value for the length of the longest side of a triangular representation of the results. This representation is used to obtain the yellow and green surfaces.



Voxel volume:

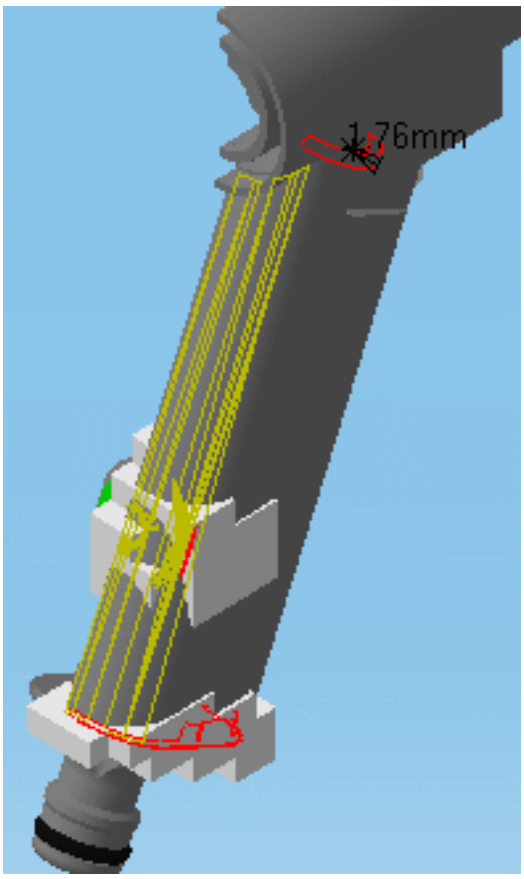
Computing and Displaying Clash Results as an Intersection Volume

In DMU-P2, you can compute and display clash results as a voxel volume or a boolean solid. To do so:

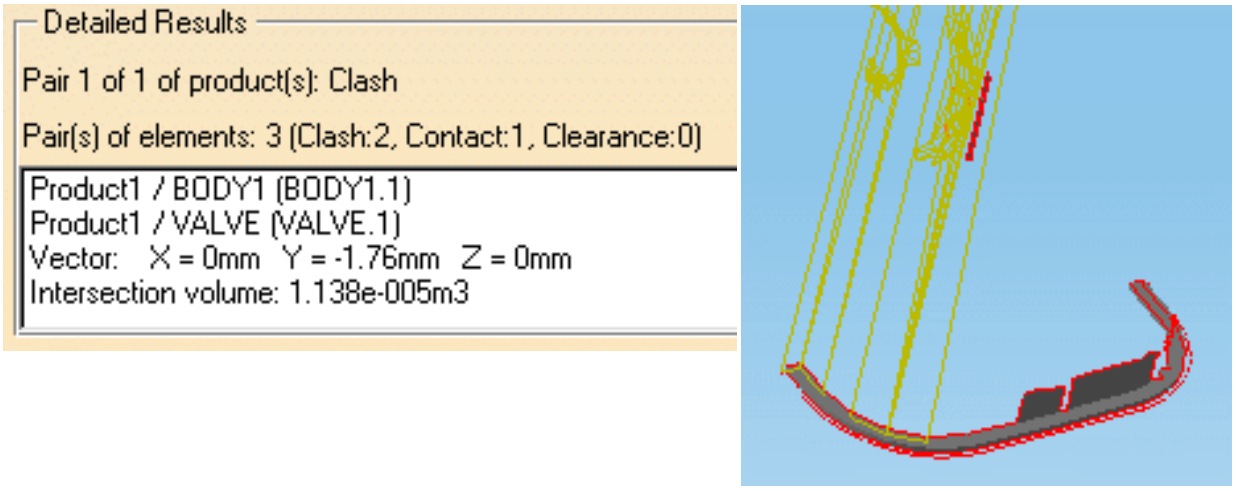
- Select **Tools ->Options...**, **Digital Mockup ->DMU Space Analysis,** DMU Clash - Detailed Computation tab
- Select the type of volume of interest under Intersection Volume
- If you selected Voxel, set the voxel size.

  The voxel size is the edge-length of the voxel or cube. A lower value will give you a more accurate result but will also require a longer computation time and more memory.
- Click **OK** in the Options dialog box when done.

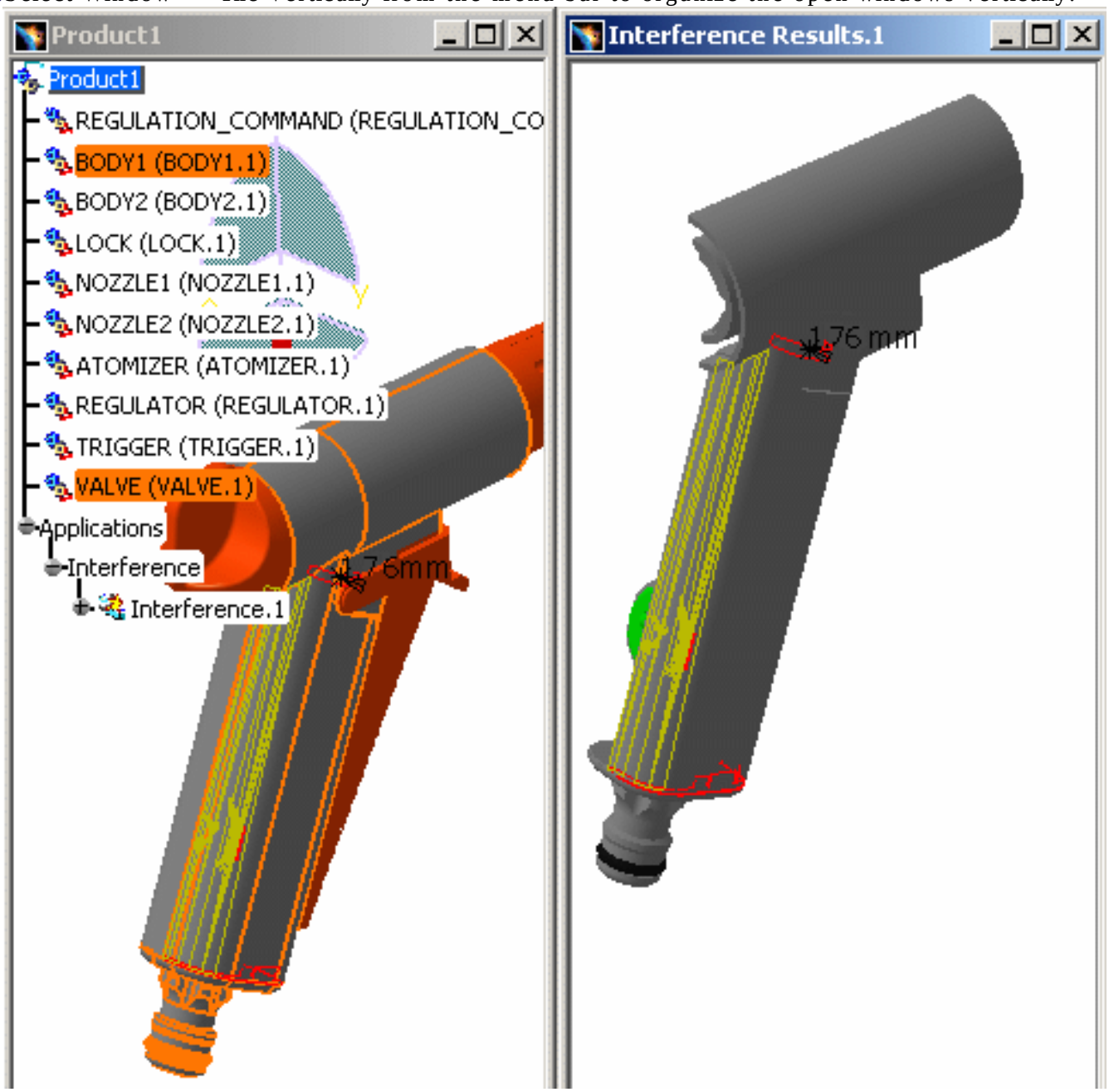
  The intersection volume is also given in the Detailed Results box of the Check Clash dialog box.



Boolean solid:



4. Select Window -> Tile Vertically from the menu bar to organize the open windows vertically.



5. Click More in the Check Clash dialog box to access visualization filters.

Using Visualization Filters

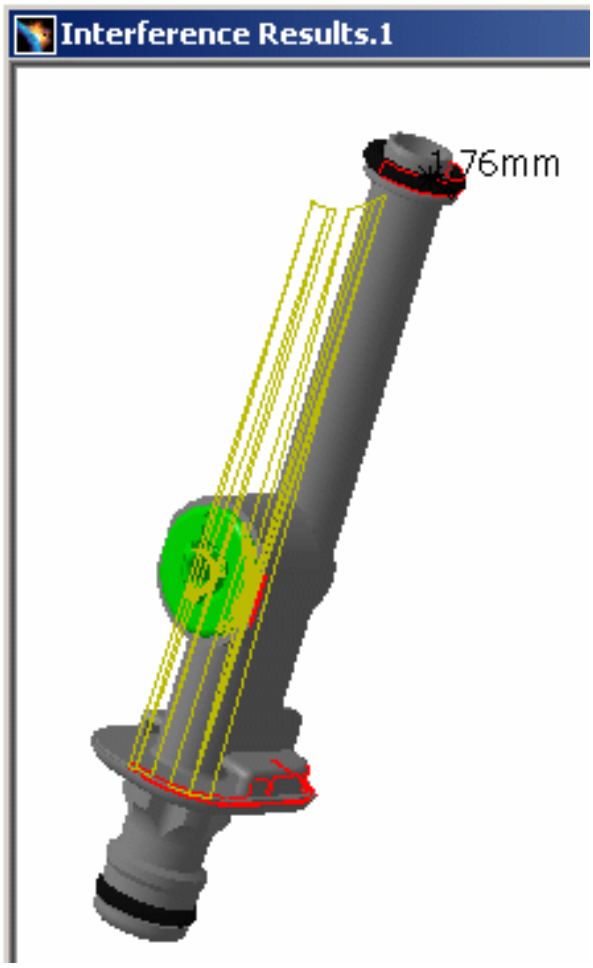
Visualization filters offer a more precise display in the Interference Results window.

- All products: displays all products selected in the list.
- Product: displays products involved in the current conflict.
- Element: displays elements involved in the current conflict. Pairs of elements are listed in the Detailed Results box and can be browsed using the Next and Previous options.
  Note: The Element option is only available if the detailed computation mode is set to Element level. This option is particularly useful when in Design mode.
- Show / Show drop-down box: a variety of combinations lets you better visualize the geometries of the two products involved in the conflict.
- All cases drop-down box: lets you select the graphics representation you want to see displayed (All cases, None or Worst cases). If Element is selected, the Worst case option is not available.

Note: The effects of all options, except the Show / Show drop-down box, can be seen in the Preview window.
The Show / Show visualization filter offers a variety of combinations letting you show, hide or place in low light the geometries of the two products involved in the conflict.

6. Select Show / Noshow in the drop-down box:

Only the valve (product 1) is visible. The body (product 2) is hidden.



7. Select Noshow / Noshow in the drop-down box:

Geometries of both products are hidden. The graphics representations of interferences only are displayed.

See Finer Analysis to find out how other visualization filters work and how to read detailed results. You can also export and publish clash command results.

8. Click **OK** in the Check Clash dialog box to exit when done.

Interference results are kept as specification tree features.
In DMU-P2, the interference definition is also kept.

The graphic display (minimum distance or penetration depth) of the current result is kept when you exit the command. If you subsequently edit the results, the system re-displays the last current result analyzed.

Note: Clicking Deselect in the Check Clash dialog box removes the current results from the selection.
The different viewpoints set using the mouse (zoom, rotate, etc.) in the Interference Results window are also saved when you exit the command. If you subsequently edit the results, the system re-displays pairs of products from the last viewpoint set. Note, however, that:

- This applies when interferences are viewed via the List by Conflict tab
- Viewpoints are not saved in the document and are therefore only valid for the duration of your session.

Exporting Clash Command Results in Text or Model Format

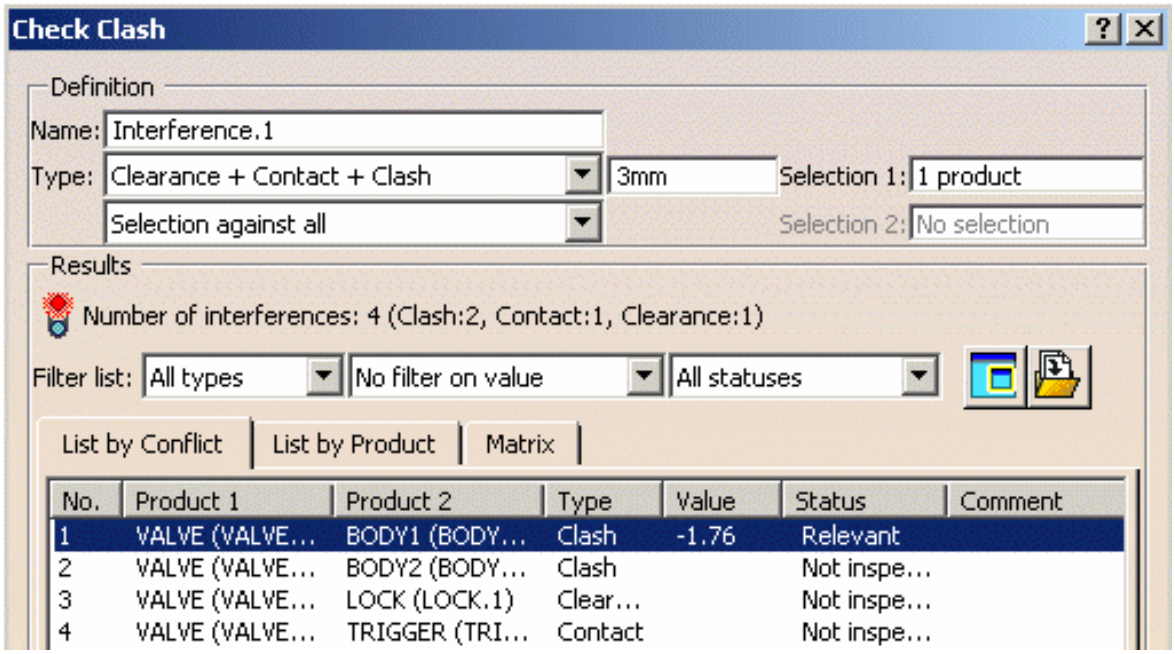
This task illustrates how to export clash results in text and V4 model formats using the **Export As** command.

Insert the following sample model files: ATOMIZER.model, BODY1.model, BODY2.model, LOCK.model, NOZZLE1.model, NOZZLE2.model, REGULATION_COMMAND.model, REGULATOR.model, TRIGGER.model and VALVE.model.

They are to be found in the online documentation filetree in the common functionalities sample folder cfysm/samples.

Switch to Design Mode (Edit -> Design Mode).

1. Select **Insert -> Clash** from the menu bar or click the Clash icon in the DMU Space Analysis toolbar to run a check of type Clearance (3mm) + Contact + Clash between the valve and all the other products in the document (Selection against all).
   The Check Clash dialog box expands to show the results of the initial computation.



2. Browse through results using dialog box commands and/or the Results viewer.
3. To write results to a text file, click the Export As icon in the Check Clash dialog box:

   The Export As dialog box is displayed.

   - Set Type to *.txt
   - Identify the folder in which you want to save the file.
   - Enter a file name.
   - Click Save to save the results in a text file.

4. To save the selected interference results as a V4 model:
   - Select the conflict of interest in the Check Clash dialog box.
   - Click the Export As  icon.
     The Export As dialog box is displayed.
   - Set Type to *.model
   - Identify the folder in which you want to save the file.
   - Enter a file name.
   - Click Save to save the results as a V4 model.
   - Click OK to confirm in the Confirm Export as V4 model dialog box that appears.

   Intersection curves (clash) or the minimum distance line (clearance) are exported as V4 lines. You can also save intersection volumes (clash) as a V4 model.

   Important: Results are exported in V4 model format conflict by conflict.

   Note: If you set the DLName document environment (**Tools -> Options -> General -> Document**) as your current environment, clicking Export As will open the DLName dialog box instead of the usual Export As dialog box.

   The DLName document environment lets you restrict the access to specific folders referenced by logical names referred to as 'DLNames'. Each folder is assigned a logical name. In this mode, you can only access documents in folders referenced by DLNames.
5. Click **OK** in the Check Clash dialog box to exit when done.

   You can also:
   - Publish clash results in HTML format
   - Export clash results and geometry in XML format.

Publishing

A number of tools let you prepare html reports that you can publish on the enterprise intranet or the Internet for collaborative communication. You can for example take a snapshot of a clash conflict detected during your review, add appropriate comments or annotations and even select parts to insert as a VRML link.

This task explains how to publish reports.

Insert all the cgr files from the samples folder.

1. In the menu bar, select **Tools -> Publish -> Start Publish**.

   The Select Publish File dialog box appears.

2. Identify the path where you want to save the report as well as the report name then click Save.

Note: Reports are published in html format. The date created and the person who created the report are indicated.

The Publishing Tools toolbar appears.

3. Click the **Feature Publish** icon , then select the feature you want to publish in the specification tree.

The DMU features you can publish are the following:

- simulations
- interferences
- URL

4. When done, click the **Stop Publish** icon in the Publishing Tools toolbar or select Tools -> Publish -> Stop Publish.

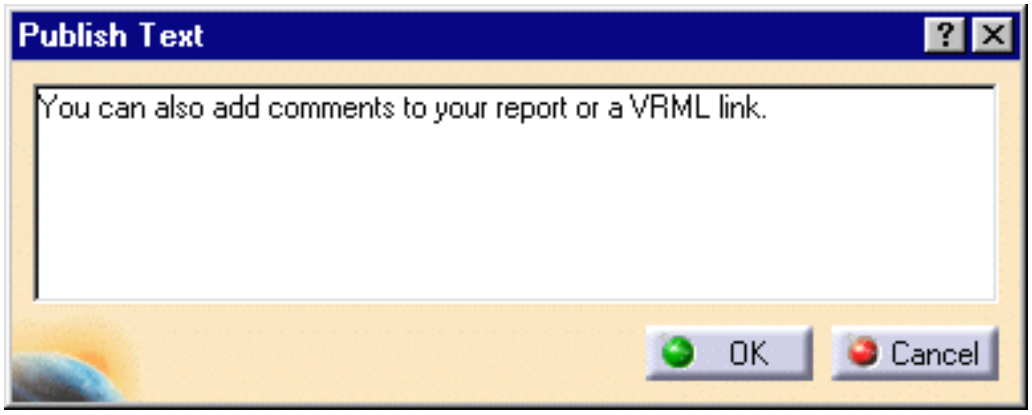
You can use other Publishing Tools commands to add a comment or take snapshots, for example.

5. Click the **Snapshot** icon to insert a screen capture into your report.

6. Click the **Text** icon to add a comment.

   The Publish Text dialog box appears.

7. Enter your comment then click **OK**.



8. Take other snapshots.

9. Select one or more products then click the VRML icon Publishing Tools toolbar to insert selected components as a VRML link.

Note: In the DMU Mock-up 2D Workshop, the VRML command is not available.

10. When done, click the **Stop Publish** icon in the Publishing Tools toolbar or select Tools -> Publish -> Stop Publish.

The published report is automatically displayed if you set the **Browser automatically opened** option in the DMU Navigator Settings (Tools -> Options -> Digital Mockup -> DMU Navigator).

Clicking the images in the html page enlarges them.

Exporting Clash Results & Geometry in XML Format

As part of a standalone clash process, you can export clash results and geometry to a dedicated file-based database. Results and geometry are managed in XML files.

Recommendations:

- Use Internet Explorer 6.0 or higher on Windows, and Netscape 7.0 (or higher) or an equivalent browser complying with the W3C standard for XSLT 1.0 transformations on UNIX.
- When renaming products or adding comments to clash results, do not use the following characters <, >, &, /. These characters are not supported when exporting results to XML files.

You can then import these results back into your session.

This task illustrates how to export clash results and geometry to a dedicated database.

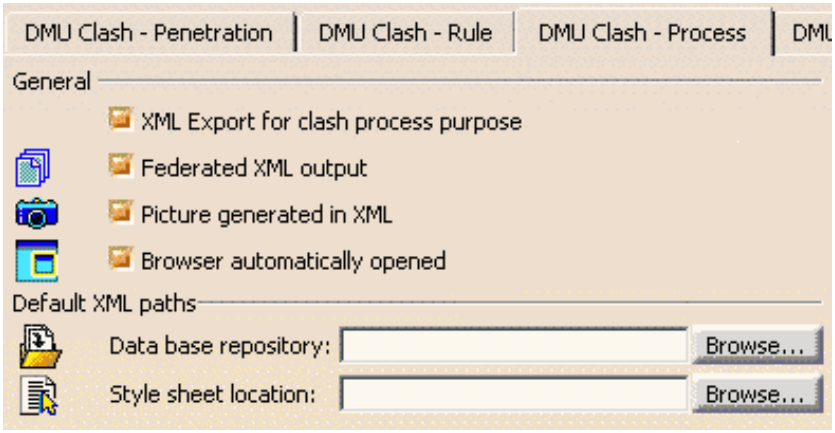
Insert the following sample cgr files: ATOMIZER.cgr, BODY1.cgr, BODY2.cgr, LOCK.cgr, NOZZLE1.cgr, NOZZLE2.cgr, REGULATION_COMMAND.cgr, REGULATOR.cgr, TRIGGER.cgr and VALVE.cgr.

They are to be found in the online documentation filetree in the common functionalities sample folder cfysm/samples.

1. Set XML export options in the DMU Clash - Process tab page (**Tools -> Options -> Digital Mockup -> DMU Space Analysis**).

   You can choose to export clash results and geometry to:

   - One XML file per conflict plus one header file summarizing results (Federated XML output checkbox selected. This is the default option),
     Or,
   - A single large XML file containing all conflicts generated.

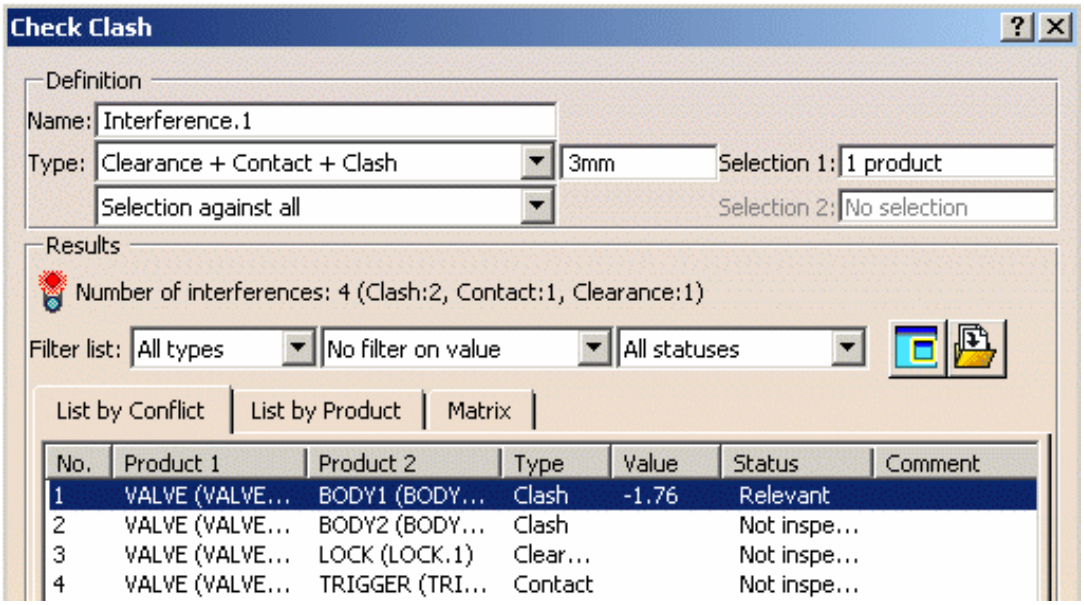


   Note: Exporting clash results and geometry as part of a standalone clash process is the default setting.

   To have your browser open automatically showing the exported results, ensure your browser is already open on your desktop.

2. Select **Insert -> Clash** from the menu bar or click the Clash icon in the DMU Space Analysis toolbar to run a check of type Clearance (3mm) + Contact + Clash between the valve and all the other products in the document (Selection against all).

   The Check Clash dialog box expands to show the results of the initial computation.



3. Browse through results using dialog box commands and/or the Results viewer.

4. Click the Export As icon in the Check Clash dialog box.

   The Save As dialog box appears. It is set to export results and geometry in XML format to the database repository identified in your settings.

   Why XML ?

   Note: If you set the DLName document environment (**Tools -> Options -> General -> Document**) as your current environment, clicking Export As will open the DLName dialog box instead of the usual Export As dialog box.

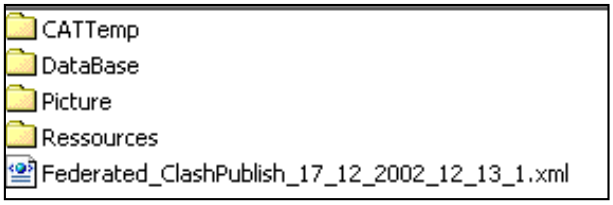
   The DLName document environment lets you restrict the access to specific folders referenced by logical names referred to as 'DLNames'. Each folder is assigned a logical name. In this mode, you can only access documents in folders referenced by DLNames.

5. Click **OK** in the Save As dialog box when done.

   A dialog box informs you of the success of the operation.

   All the necessary files and images are created in corresponding folders.

   By default (Federated XML output), one header XML file summarizing clash results and as many XML files as conflicts are generated in the database folder or repository. One PNG (Portable Network Graphics) image per conflict and two small images, each showing one of the products in conflict, are also generated and stored in a dedicated picture folder.



6. Open your browser and read the summary in the header XML file.

7. Select an interference (computation result table) to jump to the file giving information about the conflict of interest.

Notes:

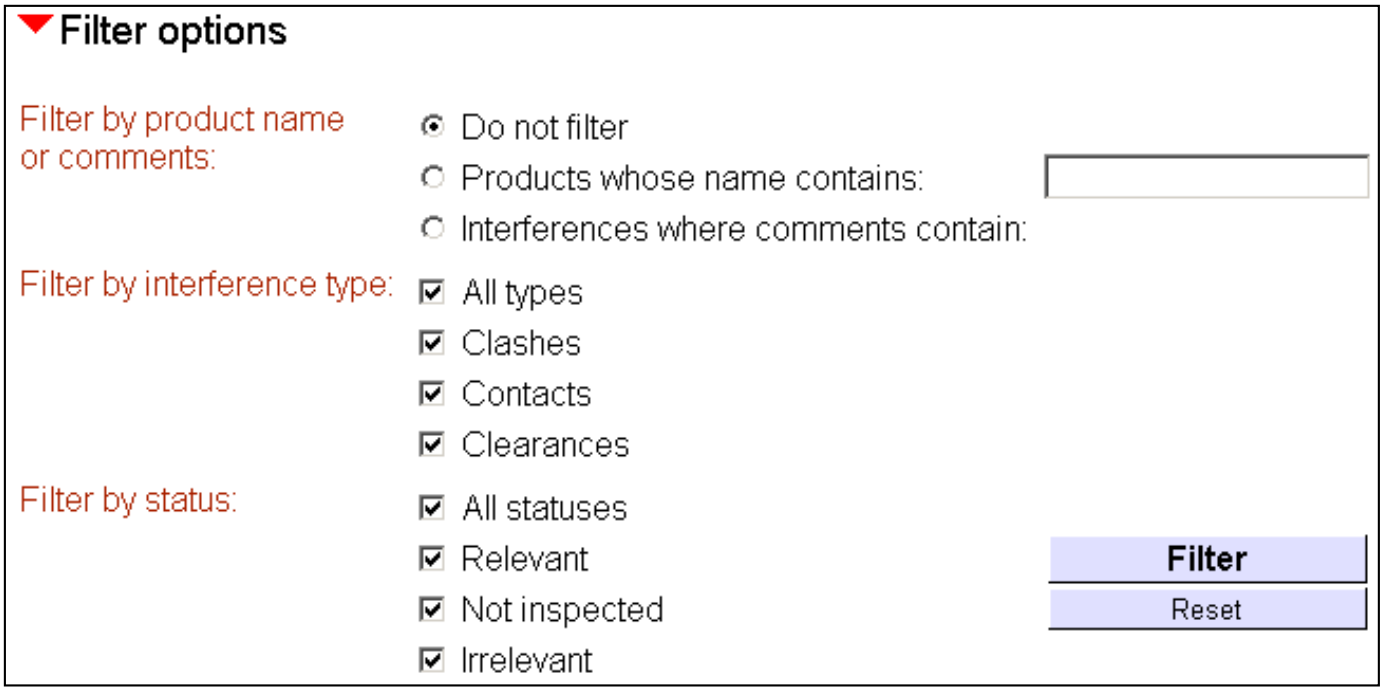
- The viewpoints set when visualizing the interferences in the Results window (not the Preview) are saved and exported along with the clash results. If no viewpoint is set, there is an automatic zoom on the interference.
- Any filters applied to the results in the dialog box are taken into account so you can use the filters to export only pertinent results.

8. Click **OK** in the Check Clash dialog box to exit when done.

Filtering Clash Results

If you choose to export clash results to a single large XML file (Federated XML output checkbox cleared), you can sort the display of computation results in the file. A Filter options entry under Computation Result lets you do so.

- Click Filter options to expand and show the form

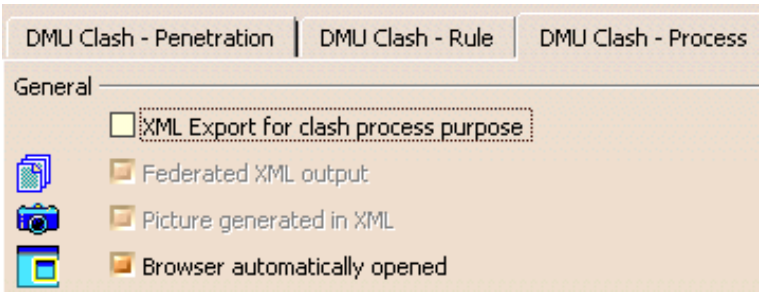


- Set filter options
- Click Filter.

Exporting Clash Results when Not in a Standalone Clash Process

When not working in a standalone clash process, clash results only can be exported in XML format. Clash geometry is not exported. This, however, is not the recommended method.

1. De-activate the XML Export for clash process purpose option in the DMU Clash - Process tab page (**Tools -> Options -> Digital Mockup -> DMU Space Analysis**).



2. Run your check and browse the results.

3. Click the Export As icon in the Check Clash dialog box.

   The Export As dialog box is displayed.

   - Set Type to XML (default type)
   - Identify the folder in which you want to save the file.
   - Enter a file name.
   - Click Save to publish the results.

   Each time you export results, a folder containing all necessary files and images is created.

4. Open your browser and read the results.

   Note: No suitable browser is currently available on UNIX to read results in this mode.

   - Click the interference (computation result table) in the browser to jump to the part of the page displaying the selected interference and appropriate results.

Notes:

- The viewpoints set when visualizing the interferences in the Results window are saved and exported along with the clash results. Viewpoints are not those in the Preview window.

- Any filters applied to the results in the dialog box are taken into account so you can use the filters to export only pertinent results.

Using the **Tools ->Options..., Digital Mockup** command (DMU Space Analysis ->DMU Clash Process tab), you can:

- Specify the default location of the XML file
- Indicate the style sheet used, and
- Have the browser open automatically.
  To have your browser open automatically showing the exported results, ensure your browser is already open on your desktop.

Reading XML Clash Files

XML has its own vocabulary. The following explains XML grammar as used in XML clash files. Explanations are given in blue.

Elements and attributes required to be able to import clash results back into DMU session are identified in red.

<?xml version="1.0" encoding="ISO-8859-1"?>
<!ELEMENT Comment EMPTY> -> a Comment node does not contain any children
<!ELEMENT Picture EMPTY> -> a Picture node does not contain any children
<!ELEMENT Vector EMPTY> -> a Vector node does not contain any children
<!ELEMENT Point EMPTY> -> a Point node does not contain any children
<!ELEMENT Date EMPTY> -> a Date node does not contain any children
<!ELEMENT ViolatedRule EMPTY> -> a ViolatedRule node does not contain any children
<!ELEMENT Viewpoint (Vector+)> -> a Viewpoint node contains 1 or n Vector nodes
<!ELEMENT Matrix (Vector+ , Point)> -> a Matrix node contains 1 or n Vector nodes and a Point node
<!ELEMENT Product (Matrix*)> -> a Product node contains 0 or n Matrix nodes
<!ELEMENT Specification (Product+)> -> a Specification node contains 1 or n Product nodes
<!ELEMENT GeometricAspect (Point*, Viewpoint*)> -> a GeometricAspect node contains 0 or n Point nodes and 0 or n Viewpoint nodes
<!ELEMENT Interference (Product+, ViolatedRule*, GeometricAspect, Comment, Picture*)> -> an Interference node contains 1 or n Product nodes, 0 or n ViolatedRule nodes, a GeometricAspect node, a Comment node and 0 or n Picture nodes
<!ELEMENT ClashResult (Interference*)> -> a ClashResult node contains 0 or n Interference nodes
<!ELEMENT ClashSpec (Specification*)> -> a ClashSpec node contains 0 or n Specification nodes
<!ELEMENT ClashElement (Date, ClashSpec, ClashResult)> -> a ClashElement node contains a Date node, a ClashSpec node and a ClashResult node

<!ATTLIST ClashElement
Responsible CDATA #REQUIRED -> name of the person who owns the clash
CATDocument CDATA #REQUIRED -> useful in standalone mode; URL path to the CATProduct
Version (1.0|1.1|1.2) "1.2"> -> the XML version; 1.2 refers to the current version

<!ATTLIST ClashSpec -> describes the clash specification
Name CDATA #REQUIRED -> name of the clash node in the DMU tree
DistanceClearance CDATA #REQUIRED -> distance clearance distance specified in the clash specification
TabTitle CDATA #REQUIRED -> useful for the XSL style sheet; only the name of a title
TabListTitle CDATA #REQUIRED -> useful for the XSL style sheet; only the name of a title
SelectMode (INSIDE|OUTSIDE|ALL|TWO_LIST) "ALL" -> corresponds to the second Type drop-down list box in interactive mode
CaseOfCalc
(INTERFERE_CONT|INTERFERE_CLEAR|INTERFERE_PENETR_CLASH|INTERFERE_RULE)
"INTERFERE_CONT"> -> corresponds to the first Type drop-down list box (interference type) in interactive mode

<!ATTLIST ClashResult -> describes the clash result
Name CDATA #REQUIRED -> the name of the clash result in the DMU tree
TabTitle CDATA #REQUIRED -> just here for the XSL style sheet; name of the title
NbInterf CDATA #REQUIRED -> number of interferences contained in the clash result
ComputationCase

(INTERFERE_CONT|INTERFERE_CLEAR|INTERFERE_PENETR_CLASH|INTERFERE_RULE)
"INTERFERE_CONT"> -> the interference type used to check for interferences

<!ATTLIST Date -> attributes corresponding to the date of the export
Year CDATA #REQUIRED
Month CDATA #REQUIRED
Day CDATA #REQUIRED
Hour CDATA #REQUIRED
Minute CDATA #REQUIRED
Second CDATA #REQUIRED>

<!ATTLIST Product
Alias CDATA #REQUIRED -> the name of the clashing product
ProcessMode (ENOVIAV5|ENOVIAVPM|STANDALONE) "ENOVIAV5" -> describes the type of connection we need to retrieve the product
DescriptionID CDATA #IMPLIED -> depending on the type of connection, the descriptionID will contain:

- ENOVIAVPM: coID compID caEnv catab Clinkable
- ENOVIA LCA: UUID/DocumentVersion
- STANDALONE: the path in the tree from the terminal node to the root product, excluding the root product itself

ShapeSource CDATA #IMPLIED -> the path to the shape when working in standalone
ShapeName CDATA #REQUIRED -> the name of the shape
Alien (YES|NO) "YES"> -> useful only in a Space Engineering Assistant context (SPE product)

<!ATTLIST Point
Ux CDATA #REQUIRED -> the X-coordinate of the point
Uy CDATA #REQUIRED -> the Y-coordinate of the point
Uz CDATA #REQUIRED> -> the Z-coordinate of the point

<!ATTLIST Vector
CorrespondingData (Rx|Ry|Rz) "Rx" -> the corresponding rotation vector in the matrix
Vx CDATA #REQUIRED -> the X-coordinate of the vector
Vy CDATA #REQUIRED -> the Y-coordinate of the vector
Vz CDATA #REQUIRED> -> the Z-coordinate of the vector

<!ATTLIST Viewpoint -> the focus, angle and projection for the corresponding viewpoint
Focus CDATA #REQUIRED
Angle CDATA #REQUIRED
Projection (CYLINDRIC|CONIC) "CYLINDRIC">

<!ATTLIST Picture
HRef CDATA #REQUIRED
Preview CDATA #IMPLIED> -> the URL corresponding to the picture and its preview (relative path)

<!ATTLIST Comment
Value CDATA #REQUIRED> -> if a comment has been defined on the corresponding interference, the value will contain it

<!ATTLIST GeometricAspect
IntersectCurveLength CDATA #REQUIRED -> contains the length of the curve describing the clash
ExtractOrDistValue CDATA #REQUIRED> -> contains the length of the vector corresponding to:

- If the result is a clearance, it contains the minimum distance between the two products
- If the result is a clash, it contains the length of the extraction vector

<!ATTLIST Interference
NumInterf CDATA #REQUIRED -> the position of the interference in the clash feature
ResultType (Clash|Contact|Clearance) "Clash" -> the interference type, either clash, contact or clearance
DistanceClearance CDATA #IMPLIED -> just in case of clearance computation; contains the clearance value defined in the dialog box
Status (NotInspected|Relevant|Irrelevant|Solved) "NotInspected" -> the status of the interference
RetrieveInf (Old|New|Modified) "New"> -> the information about clash history

The following node is needed only in case of clash computation with knowledge rules:

<!ATTLIST ViolatedRule
Name CDATA #REQUIRED -> the name of the rule
TypeCalc CDATA #REQUIRED -> the type of computation defined by the rule
Clearance CDATA #REQUIRED -> the clearance value defined in the rule
Priority CDATA #REQUIRED -> the priority of the rule
PenetrationCandidate CDATA #IMPLIED -> Yes if it is candidate for penetration, No if not
Severity CDATA #IMPLIED> -> the severity of the rule

Comparing Products

This task explains how to compare two parts or two products to detect differences between them and identify where material has been added and/or removed.

This is useful when comparing assemblies or products at different stages in the design process or when considering internal and external (client) changes to the same product.

Two comparison modes are available:

- Visual (default value): the comparison is entirely visual. A single view shows the results.
- Geometric: differences between assemblies or products are represented by cubes with separate views showing added and removed material.

Visual comparison offers faster and finer comparison:

- Computation time is proportional to the size of the Visual Comparison viewer.
- Visual comparison is purely in terms of pixels; zooming in gives a better view.

In both modes, you can compare assemblies or products with respect to the:

- Absolute axis system in the document (default value), or
- Local axis systems.

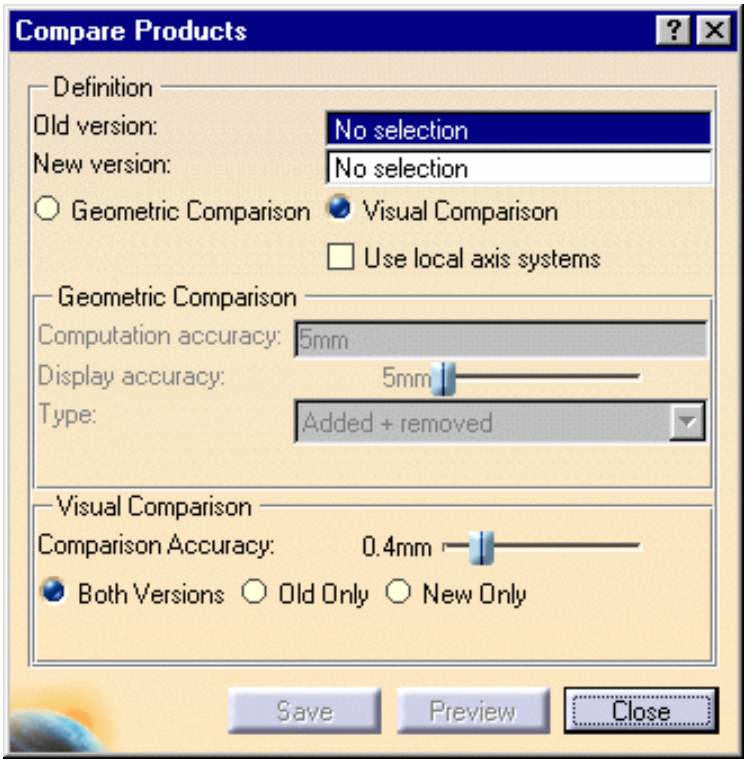
Insert the PEDALV1.model and PEDALV2.model documents in the DMU Space Analysis samples folder **spaug/samples**.

Important: Products or parts you want to compare must be in the same CATproduct document.

1. Click the Compare Products icon  in the DMU Space Analysis toolbar:

   The Compare Products dialog box appears.

   Visual comparison is the default comparison mode.



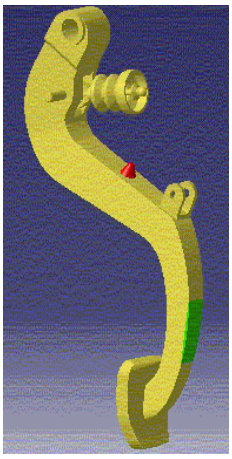
2. Select one of the products you want to compare (old version), PEDALV1 for example.
3. Select the other product (new version), PEDALV2 for example.

   The spatial coordinates of PEDALV1 and PEDALV2 are defined with respect to the absolute axis system of the document and are the same.

   Note: Multi-selection capabilities are not available in this command.

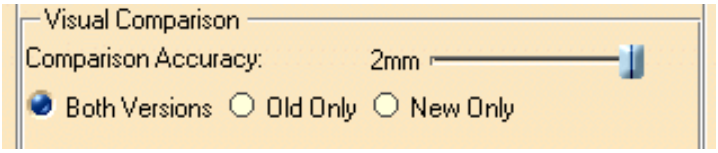
4. Click **Preview** to run the visual comparison.

   A Visual Comparison viewer opens showing the results:
   - Yellow: common material
   - Red: added material
   - Green: removed material.

   You can re-size the viewer if desired.

5. Move the Comparison accuracy slider to the far right and click **Preview** again.



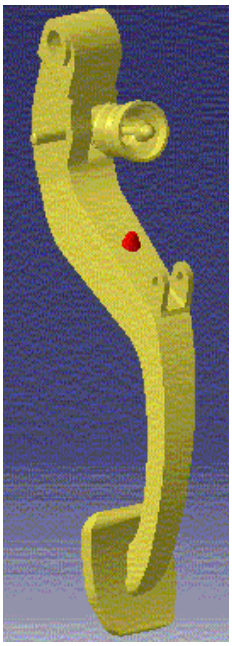
Comparison Accuracy

Comparison accuracy corresponds to the minimum distance between two products beyond which products are considered different. A higher value gives a cleaner image.

As you can see, the green area is no longer detected at the higher setting: it is no longer considered different.

The default value (0.4 mm) is twice the default sag value for calculating tessellation on objects. Sag (3D fixed accuracy) is set in the Performances tab of **Tools -> Options -> General -> Display.**

The default comparison accuracy is the recommended value for visual comparison



Visual Comparison Options

Options let you view:

- Both versions: common material and both versions of the product.
- Old only: common material and the old version of the product.
- New only: common material and the new version of the product.

You can use the Measure Between command to make measures, for example between two points, in the Visual Comparison viewer.

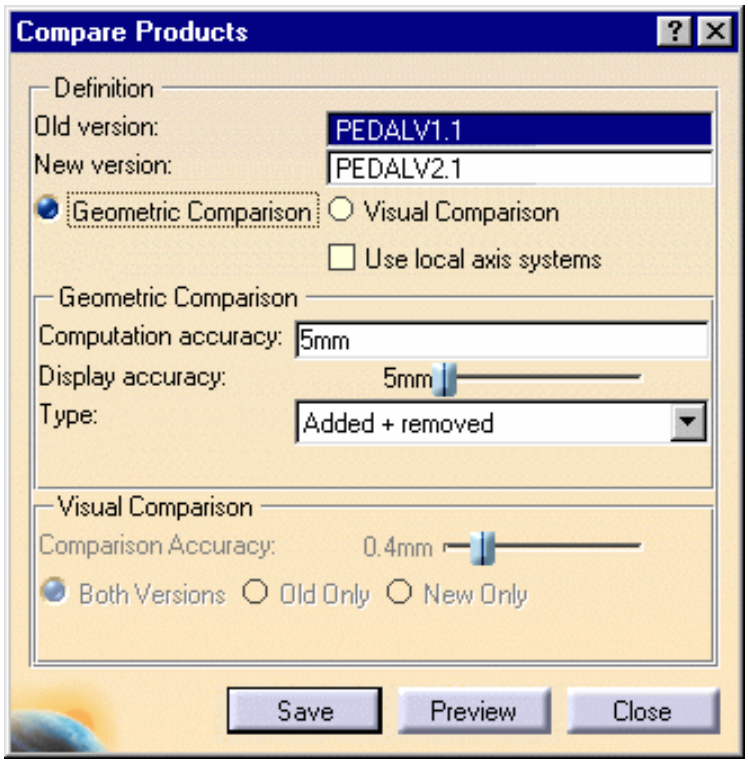
Non-selected products in the main document window are placed in low light.

Note: You cannot save the results in this comparison mode.

Making a Geometric Comparison

You will run a geometric comparison on the same two products.

6. Select the Geometric comparison check box.



7. Set the computation accuracy by entering a value. In our example, we will keep the default value of 5mm.

Setting Computation Accuracy

The computation accuracy determines the size of the cubes used to represent the material added and/or removed. A lower setting results in slower computation time, but a more precise calculation of differences.

8. Move the slider to the right to set the display accuracy to 20mm for example.

Setting Display Accuracy

Independently of the computation accuracy, you can set the display accuracy to a coarser display of the computation results to give a better graphics display performance.

By default, the display accuracy is set to the same value as the computation accuracy.

9. Select the type of comparison you want to run from the Type drop-down list, Added + removed for example.

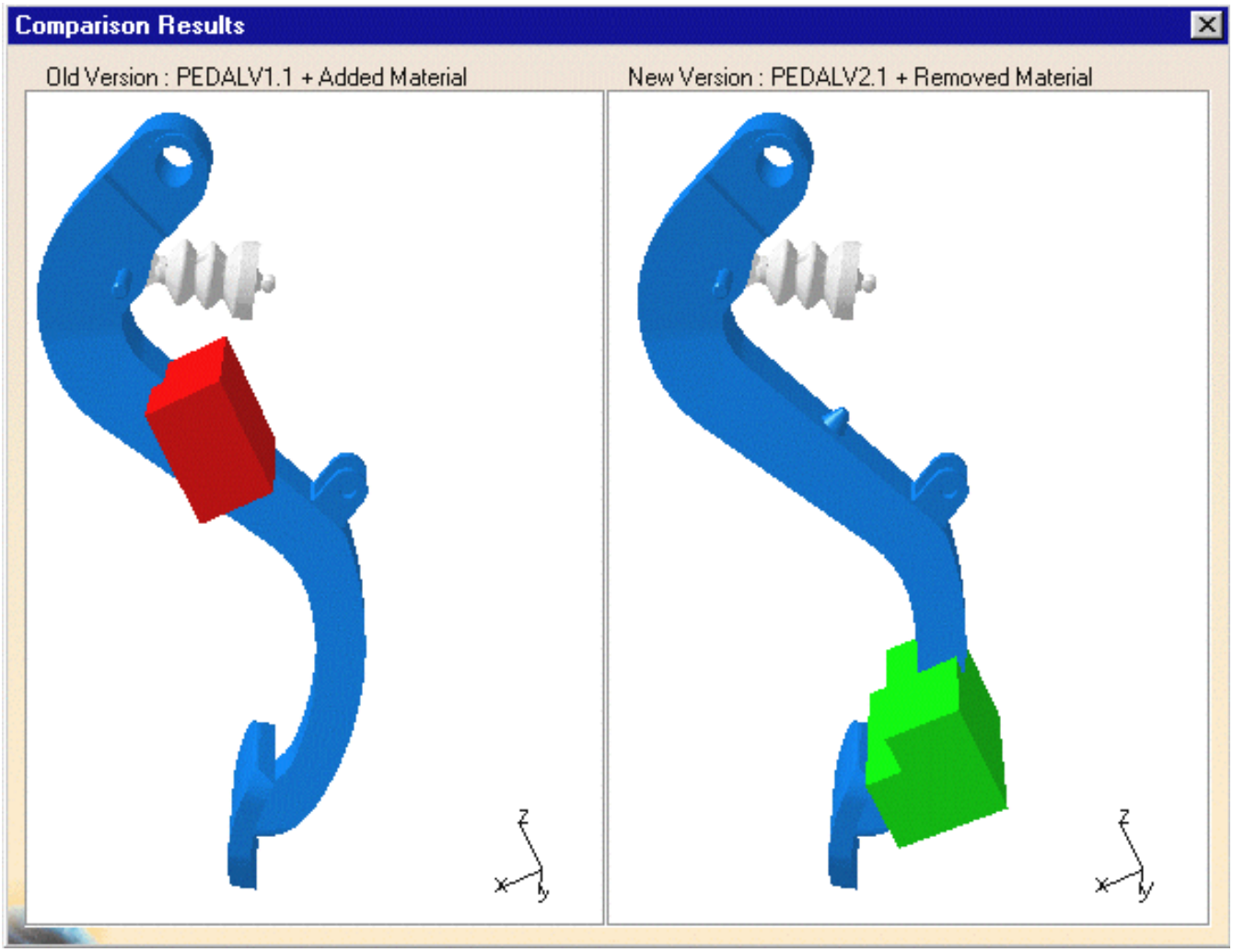
Defining Type

- Added: Computes differences where material has been added only.
- Removed: Computes differences where material has been removed only.
- Added + removed: Computes differences where material has been both added and removed. Differences are displayed in separate views and saved in different files.
- Changed: Computes differences where material has been both added and removed, displaying all changes in both views and letting you save changes in the same file.

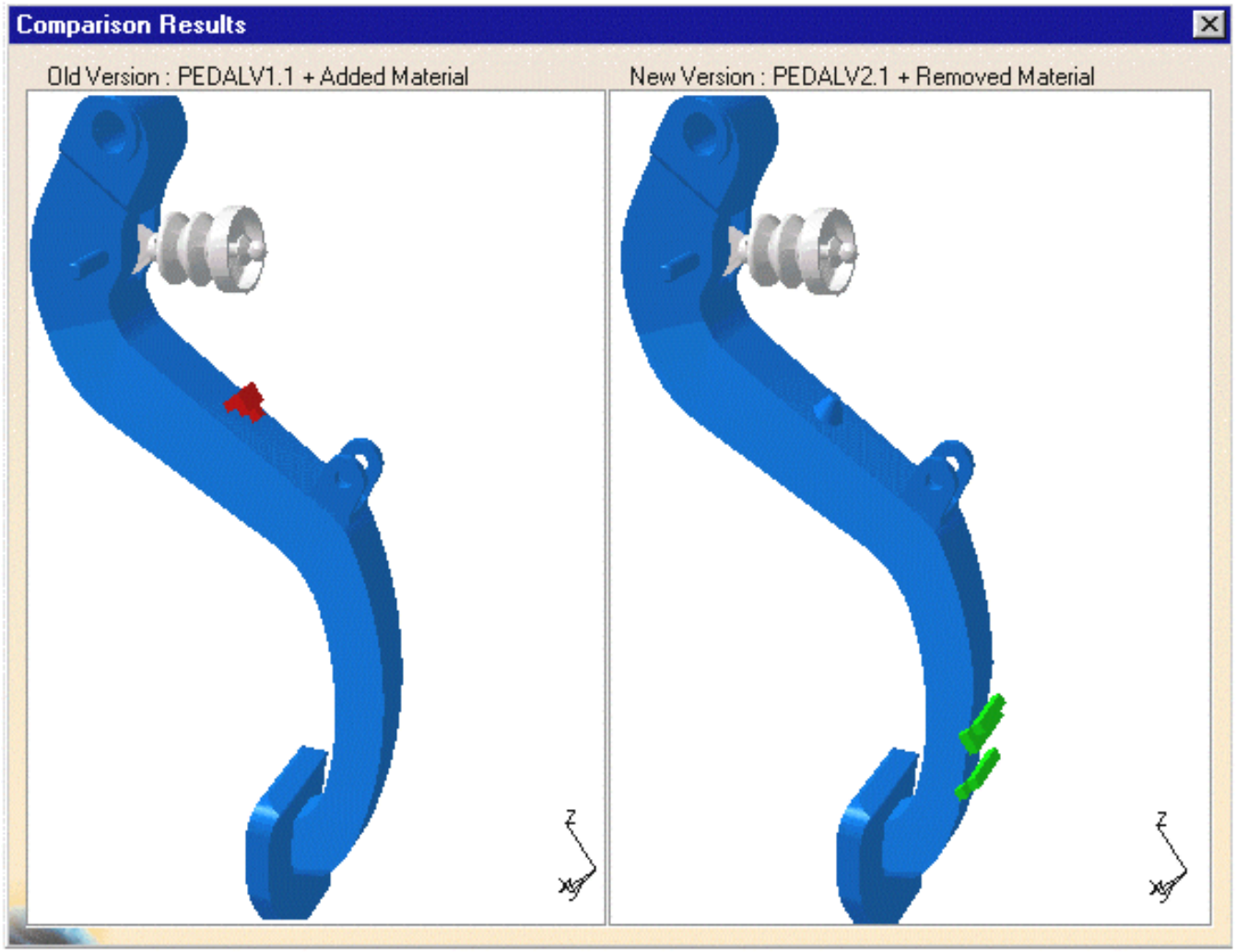
10. Click **Preview** to run the geometric comparison:

A progress bar is displayed letting you monitor and, if necessary, interrupt (**Cancel** option) the calculation.

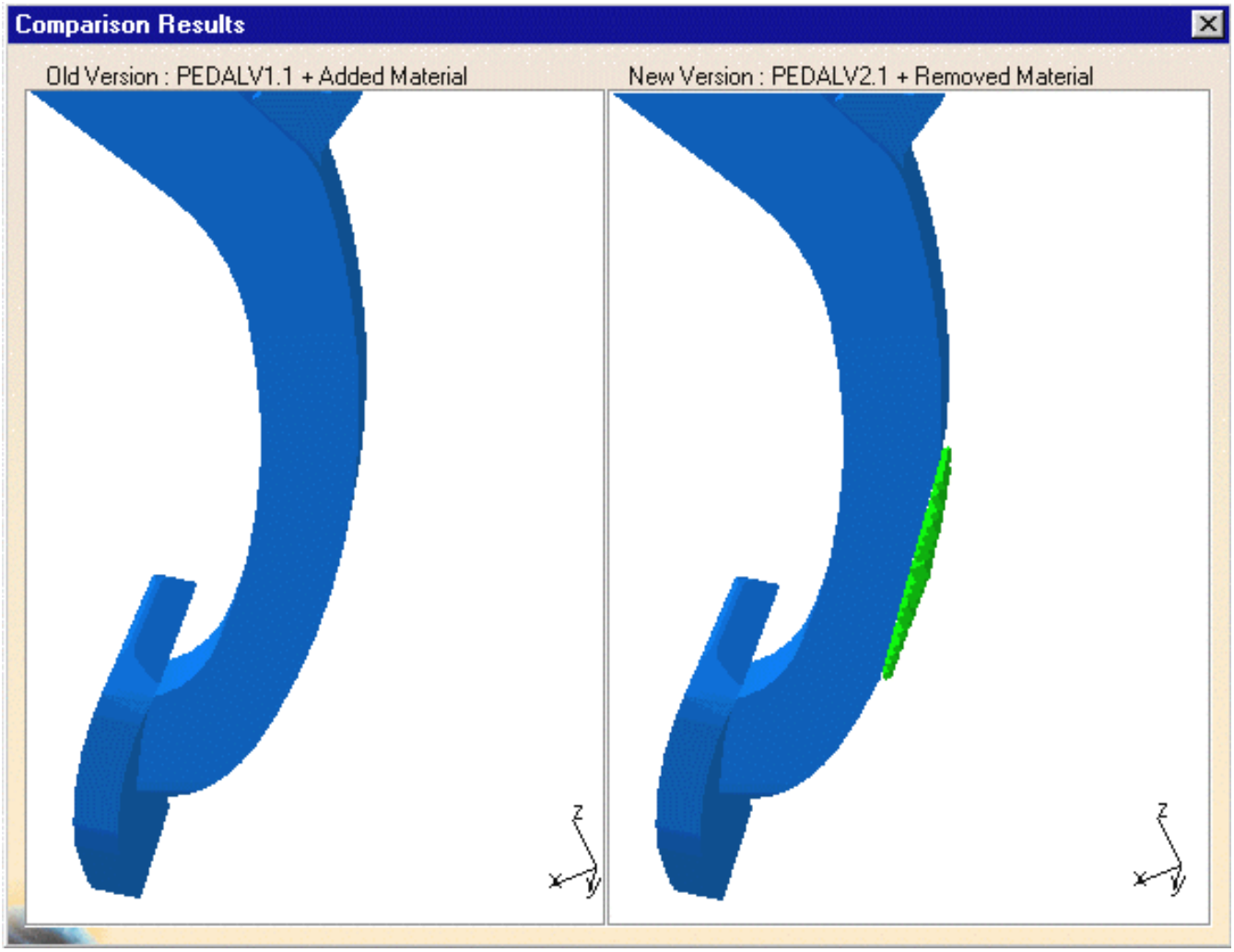
A dedicated viewer appears showing the results. Differences are represented by cubes. Added material is shown in red; removed material in green.



11. Repeat the comparison adjusting the display accuracy to the same value as the computation accuracy (5mm):



12. Repeat the comparison adjusting the computation accuracy to 2mm.



P2 Functionality

In DMU-P2, you can save the displayed results (cubes) in 3dmap format (.3DMap), as a cgr file (.cgr), a Virtual Reality Modeling Language (VRML) document (.wrl) or a V4 model (.model).

The 3dmap format can be inserted into a product and other DMU Space Analysis (**Clash** or **Sectioning**) or DMU Navigator (**Proximity Query**) commands run to evaluate the impact of modifications.

Colors assigned to added (red) and removed (green) material will also be saved making changes more visible when re-inserted into a document.

The Save As dialog box is proposed when you click **Save** in the Compare Products dialog box:

- Specify the location of the document to be saved and, if necessary, enter a file name.
- Click the Save as type drop-down list and select the desired format.
- Click **Save** to save the results in a file in the desired format.

Using Local Axis Systems

You will now run a visual comparison using local axis systems.

This option, available in both visual and geometric comparison modes, lets you compare two products defined with respect to local axis systems, irrespective of the position of products in the document.

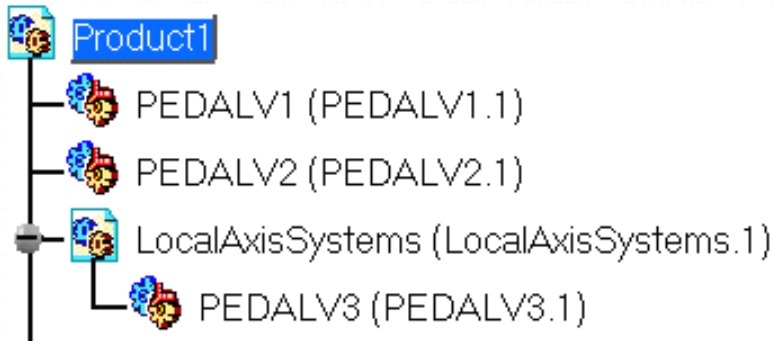
13. Insert the PEDAL.CATProduct document in the DMU Space Analysis samples folder and click the Compare Products icon again.

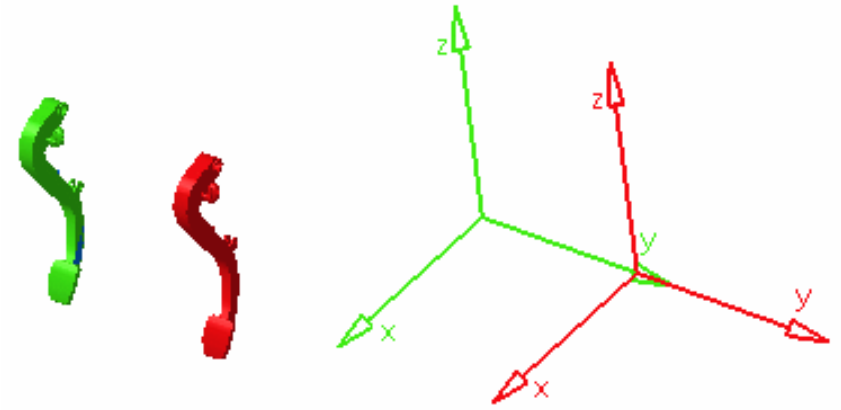
14. Select the old version: select PEDALV1 again.

    The product and its axis system are highlighted in green.

15. Select the New version: PEDALV3.

    The product and its axis system are highlighted in red.

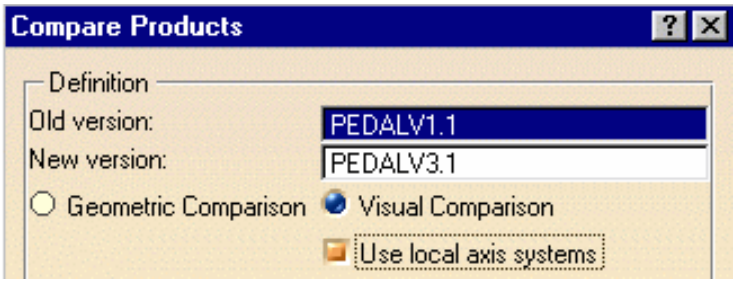




Spatial coordinates of PEDALV1 and PEDALV3 are different when defined with respect to the absolute axis system in the document but are the same when defined with respect to local axis systems.

16. Set the comparison accuracy as desired.

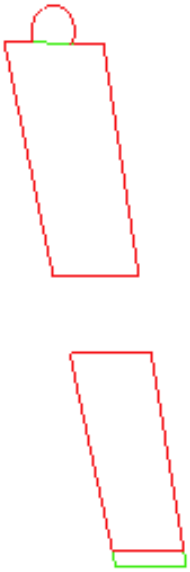
17. Select the Use local axis systems check box:



Local axes of the two products are superimposed in the main document window. The old version axis system is the reference axis system.

You can generate a section in the main document window: added and removed material is visible in the generated section.

You can also make measures using the Measure Between command.



18. Click **Preview** to view results in the Visual Comparison viewer.

A progress bar is displayed letting you monitor and, if necessary, interrupt (**Cancel** option) the calculation.

Notice that you get the same results as you did when comparing PEDALV1 and PEDALV2: for the purposes of this task, PEDALV3 is a copy of PEDALV2 that has been positioned differently in the document.

19. Click **Close** when done.

Combining the Compare Products command with other DMU Space Analysis and DMU Navigator toolbar commands

You can, for example, run a query for products immediately surrounding the added material (**Proximity Query** command in the DMU Data Navigation toolbar) and then analyze for clashes (**Clash** command). This offers the advantage of letting you, for example, focus on a part of an engine rather than analyzing the whole engine and then having to sift through the results to find those relevant.

Measure Tools

About measure tools: Gives general information on the various commands available.

Arc through Three Points

Measure arc sections: Click the Arc through Three Points icon, then select three points along a curve or an arc.

Measure Between

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

Measure Item

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

Measure Thickness

Measure thickness: Click the Measure Item icon, then the Measure Thickness icon and select an item.

Measure Inertia

Measure inertia: Click the Measure Inertia icon, then select an item.

2D Measure

Measure distance, angle and radius on 2D documents: Click the 2D Measure icon, calibrate, then make your measure.

About Measure Tools

A certain number of generic tools are supplied with DMU Space Analysis. These tools let you measure the minimum distance and angle between geometrical entities or points, properties associated to selected items, and the moments of inertia, density and mass of items.

You can also measure the length, radius and angle of an arc drawn through three points as well as measure distances, angles and the radius on 2D documents of both pixel and vector type.

Note: In the No Show space, measure commands are not accessible.

Shading with Edges

To get the most out of these tools, set the Render Style to Shading with Edges. The various command options work differently depending on the selected render style.

Measure Between, Item and Inertia commands

Measure Between, Measure Item and Measure Inertia offer a complete set of exact measurements on and between products, parts and features, as well as on all types of 3D formats, including in Visualization mode and for CGR files.

Exact measures are also available on V4 skins.

These commands also offer an approximate mode for faster calculation.

Measure Between and Measure Item are useful when performing sectioning operations to, for example, annotate the generated section. To do so, select the Keep Measure check box.

Measuring Arc Sections

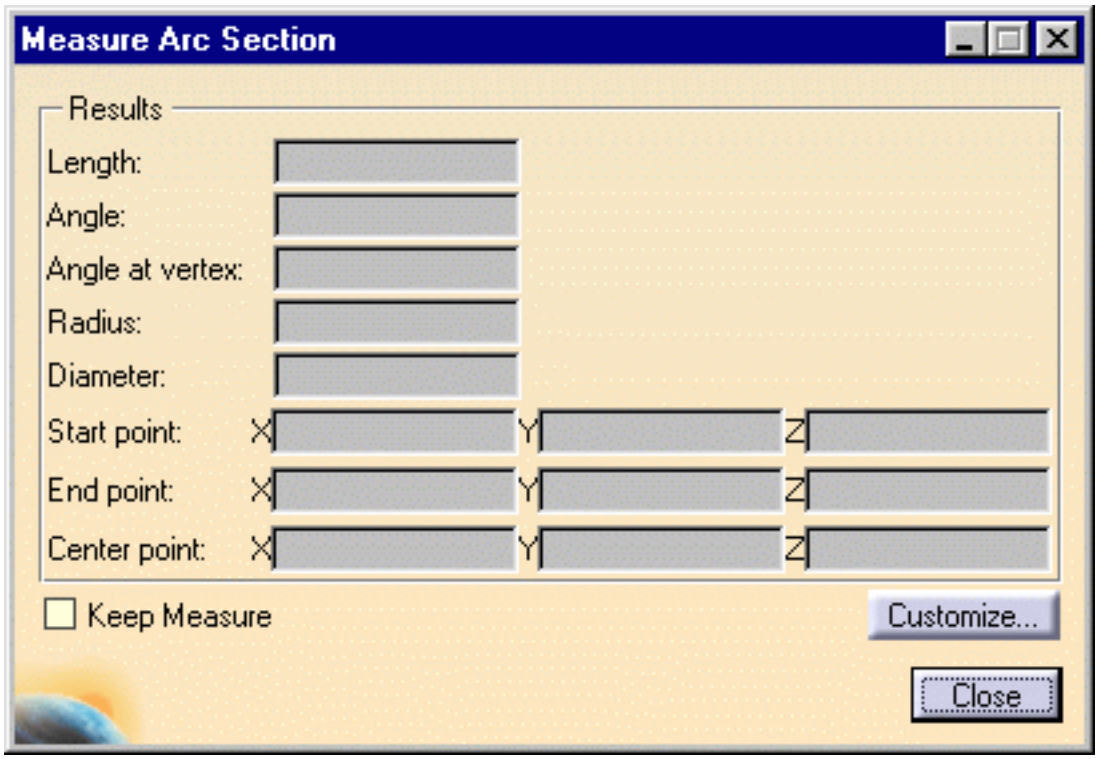
This task explains how to measure the length, radius and angles of an arc drawn through three points.

Insert the following sample model files: ATOMIZER.model, BODY1.model, BODY2.model, LOCK.model, NOZZLE1.model, NOZZLE2.model, REGULATION_COMMAND.model, REGULATOR.model, TRIGGER.model and VALVE.model.

They are to be found in the online documentation filetree in the common functionalities samples folder cfyug/samples.

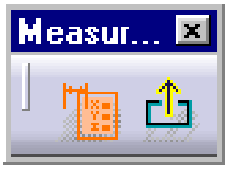
1. Click the Arc through Three Points icon in the DMU Space Analysis toolbar or select **Analyze -> Arc through Three Points** from the menu bar:

The Measure Arc Section dialog box appears.



- A Keep Measure option in the dialog box lets you keep the current and subsequent measures as features. This is useful if you want to keep the measures as annotations for example.
- Double-clicking an existing measure lets you re-edit the presentation of the measure, review information in the dialog box or delete measures.

P1 Functionalities

In DMU-P1, the Measure Tools toolbar also appears. 

This toolbar has two icons:

- Measure Dialogs : lets you show or hide the associated dialog box.
- Exit Measure : lets you exit the measure. This is useful when the dialog box is hidden.

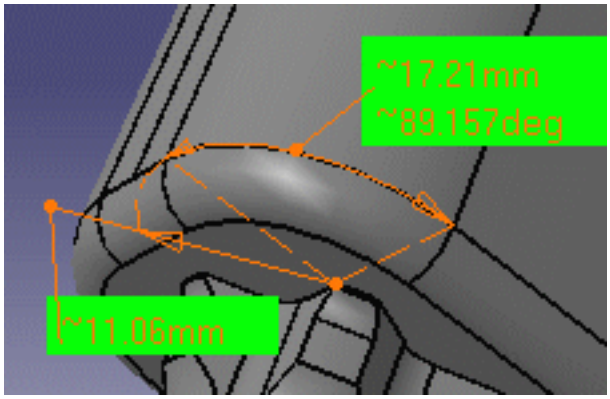
2. Select three points along a curve or an arc.

Notes:

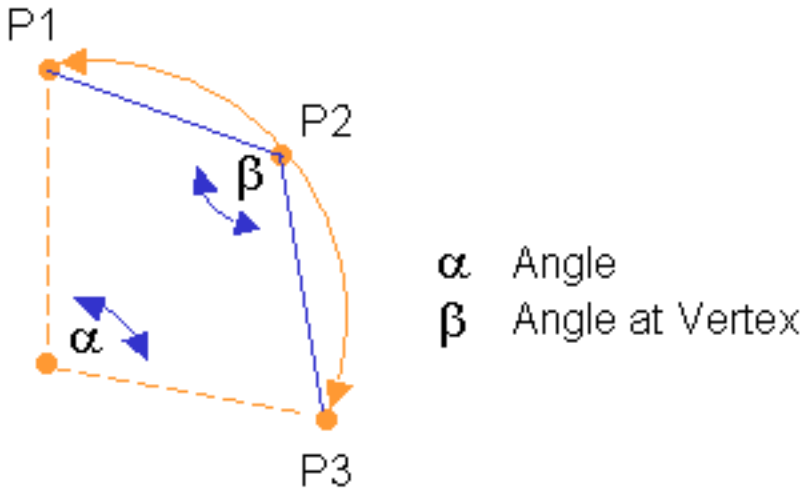
- The appearance of the cursor  changes to reflect the measure command that you are in. A number (1, 2 then 3) also helps you identify where you are in your measure.
- Dynamic highlighting  as you move the cursor over geometrical entities helps you locate points. As you move over edges, the edge (and not the surface) is highlighted.

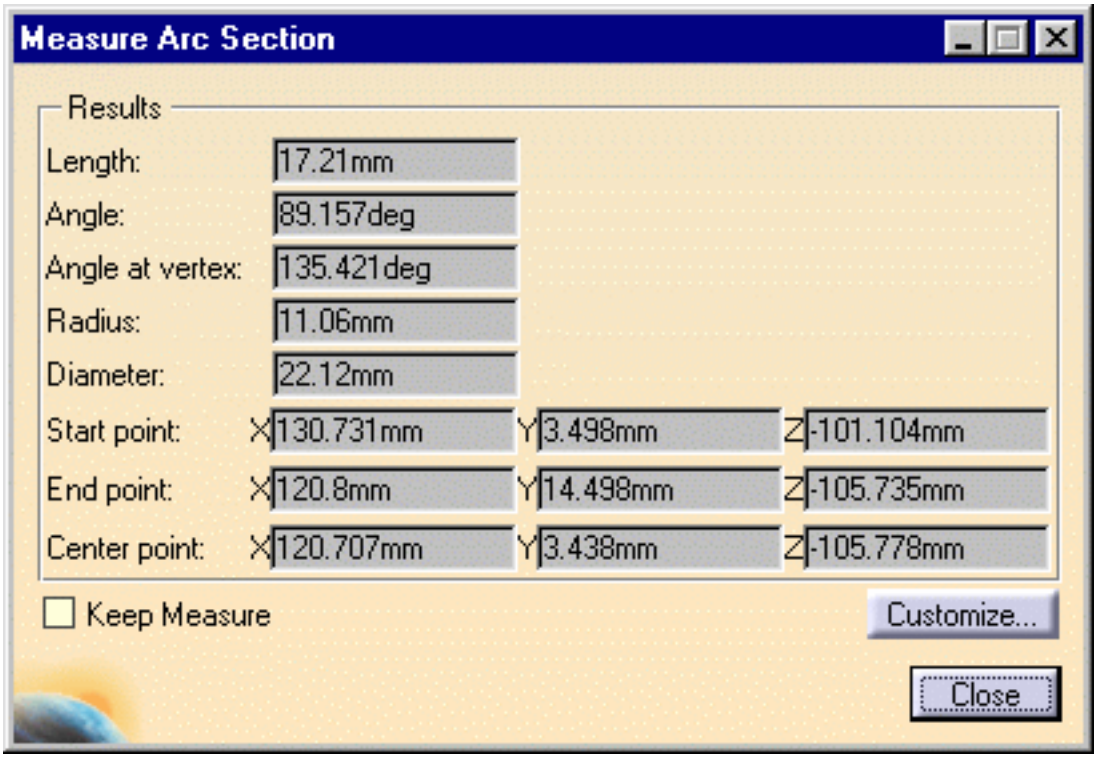
An arc is fitted through the three selected points and is displayed along with its center point.

The arc length, angle and radius are also visualized.



The dialog box is updated and now gives the length, angle, angle at vertex and radius or diameter of the arc as well as start, end and center point coordinates.





The number of decimal places, the display of trailing zeros and limits for exponential notation is controlled by the Units tab in the Options dialog box (**Tools-> Options**, **General-> Parameters and Measure**). For more information, see the Infrastructure User's Guide.

3. If necessary, adjust the presentation of the measure:

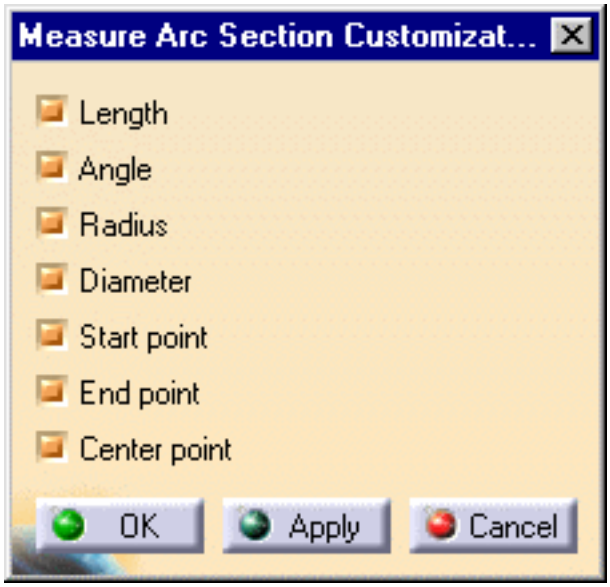
You can move the radius line and text anchor point as well as the texts of the measure.

The Properties command (Graphics tab) lets you change the fill color and transparency as well as the color, linetype and thickness of measure lines.

Note: You cannot vary transparency properties, the current object is either the selected color or transparent.

Customizing Your Measure

You can, at any time, customize the display of the results in both the geometry area and the dialog box. To do so, click Customize... in the Measure Arc Section dialog box and set your display in the Measure Arc Section Customization dialog box. By default, all results are displayed.



4. Click **Close** when done.

Measuring Distances between Geometrical Entities

The Measure Between command lets you measure distance between geometrical entities. You can measure:

- Minimum distance and, if applicable angles, between points, surfaces, edges, vertices and entire products
  Or,
- Maximum distance between two surfaces, two volumes or a surface and a volume.

This section deals with the following topics:

Insert the following sample model files: ATOMIZER.model, BODY1.model, BODY2.model, LOCK.model, NOZZLE1.model, NOZZLE2.model, REGULATION_COMMAND.model, REGULATOR.model, TRIGGER.model and VALVE.model.

They are to be found in the online documentation filetree in the common functionalities sample folder cfysm/samples.

Restriction: Neither Visualization Mode nor cgr files permit selection of individual vertices.

Note: In the No Show space, the Measure Between command is not accessible.

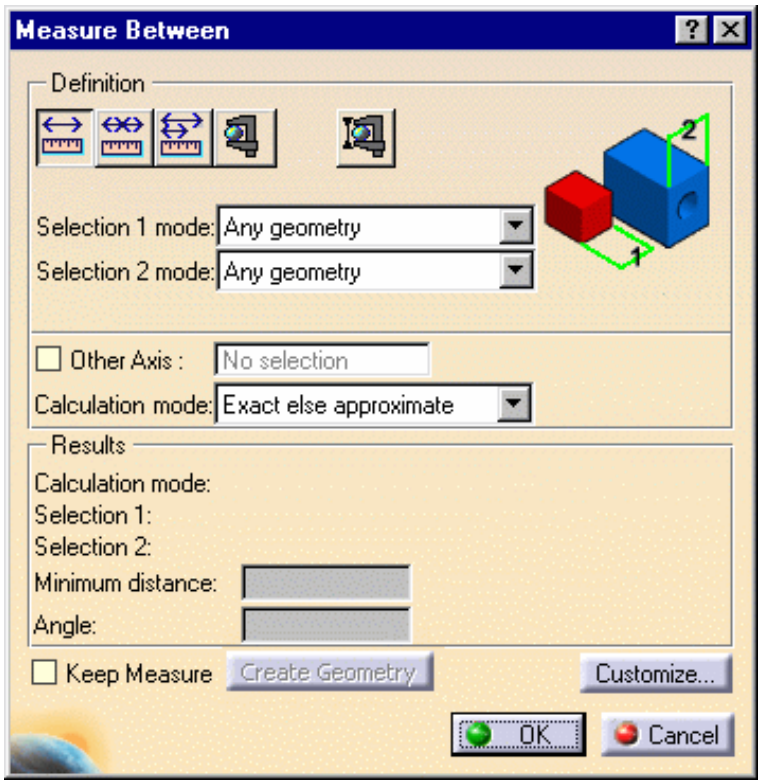
Measuring Minimum Distance and Angles

This task explains how to measure minimum and, if applicable, angles between geometrical entities (points, surfaces, edges, vertices and entire products).

1. Click the Measure Between  icon.

   In DMU, you can also select **Analyze-> Measure Between** from the menu bar.

   The Measure Between dialog box appears.



By default, minimum distances and if applicable, angles are measured.

By default, measures made on active products are done with respect to the product axis system. Measures made on active parts are done with respect to the part axis system.
Note: This distinction is not valid for measures made prior to Version 5 Release 8 Service Pack 1 where all measures are made with respect to the absolute axis system.

Dialog box options

- You can also measure distances and angles with respect to a local V5 axis system.
- A Keep Measure option in the dialog box lets you keep the current and subsequent measures as features. This is useful if you want to keep the measures as annotations for example.

  Some measures kept as features are associativeand can be used to valuate parameters or in formulas.

In the Drafting and Advanced Meshing Tools workbenches, measures are done on-the-fly. They are not persistent. This means that they are not associative and cannot be used as parameters.

- A Create Geometry option in the dialog box lets you create the points and line corresponding to the minimum distance result.
- A Customize... option opens the Measure Between Customization dialog box and lets you set the display of measure results.

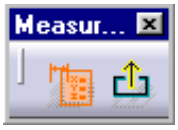
Accessing other measure commands

- The Measure Item command  is accessible from the Measure Between dialog box.
- In DMU, the Measure Thickness command is also accessible from the Measure Between dialog box. For more information, see the *DMU Space Analysis User's Guide*.

P1-Only Functionality

In P1, the Measure Tools toolbar appears.

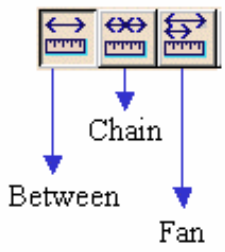
This toolbar has two icons:

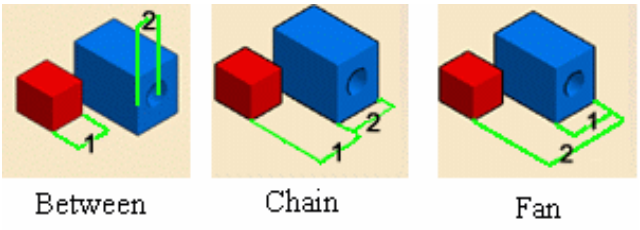


- Measure Dialogs : lets you show or hide the associated dialog box.
- Exit Measure : lets you exit the measure. This is useful when the dialog box is hidden.

2. Select the desired measure type.

Notice that the image in the dialog box changes depending on the measure type selected.





Defining Measure Types

- Between (default type): measures distance and, if applicable, angle between selected items.
- Chain: lets you chain measures with the last selected item becoming the first selection in the next measure.
- Fan: fixes the first selection as the reference so that you always measure from this item.

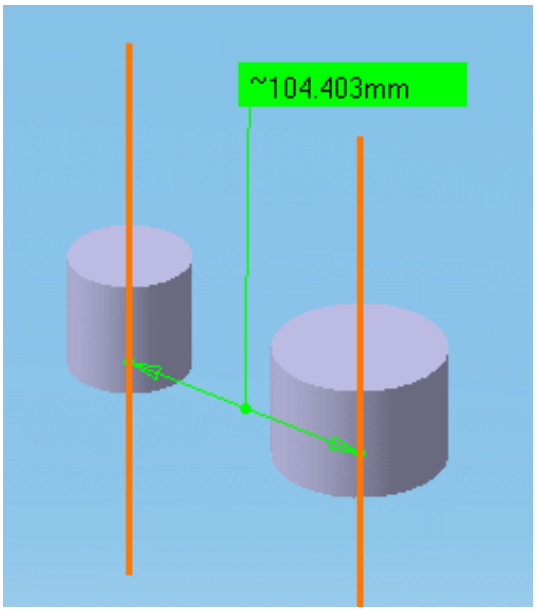
3. Set the desired mode in the Selection 1 and Selection 2 mode drop-down list boxes.

Defining Selection 1 & Selection 2 Modes

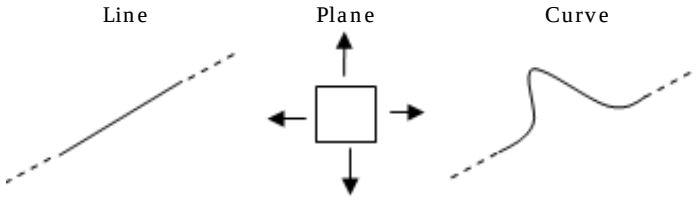
- Any geometry (default mode): measures distances and, if applicable, angles between defined geometrical entities (points, edges, surfaces, etc.).
  Note: The Arc center mode is activated in this selection mode.

This mode recognizes the axis of cylinders and lets you measure the distance between two cylinder axes for example.

Selecting an axis system in the specification tree makes the distance measure from the axis system origin.
You can select sub-entities of V5 axis systems in the geometry area only. For V4 axis systems, distances are always measured from the origin.

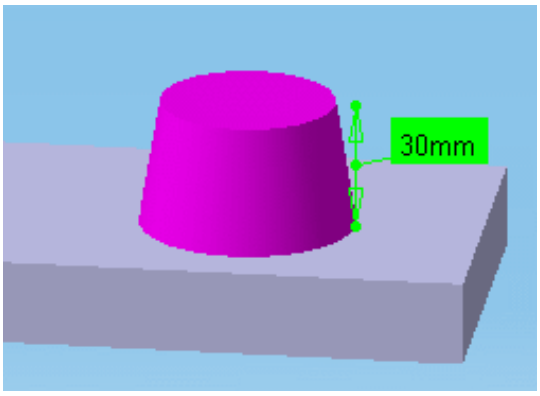


- Any geometry, infinite: measures distances and, if applicable, angles between the infinite geometry (plane, line or curve) on which the selected geometrical entities lie. Curves are extended by tangency at curve ends.

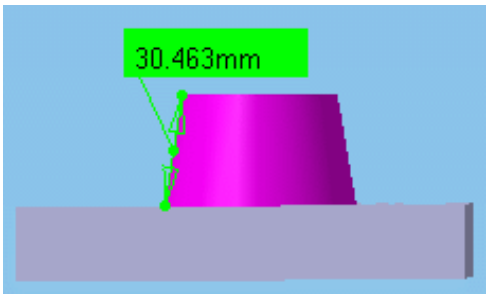


The Arc center mode is activated and this mode also recognizes cylinder axes. For all other selections, the measure mode is the same as any geometry.

Any geometry, infinite



Any geometry

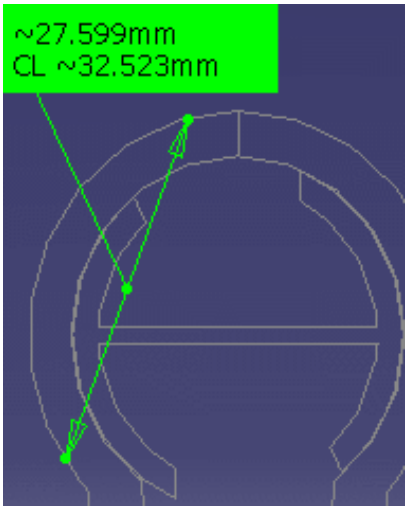


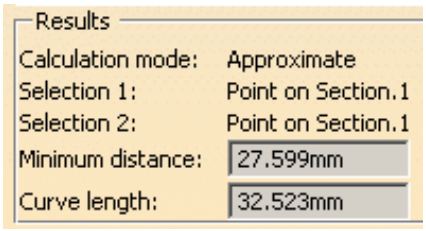
- Picking point: measures distances between points selected on defined geometrical entities. Always gives an approximate measure.

In the DMU section viewer, selecting two picking points on a curve gives the distance along the curve between points (curve length or CL) as well as the minimum distance between points.

Notes:

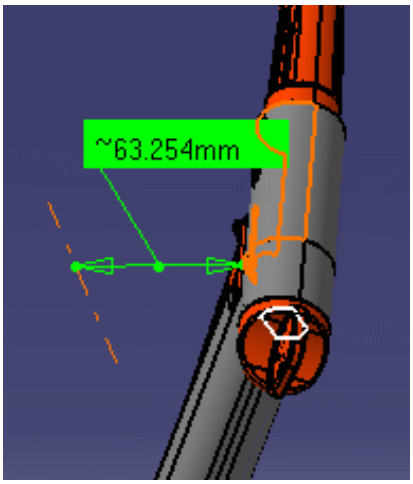
- Both points must be located on the same curve element.
- The minimum distance option must be set in the Measure Between Customization dialog box.





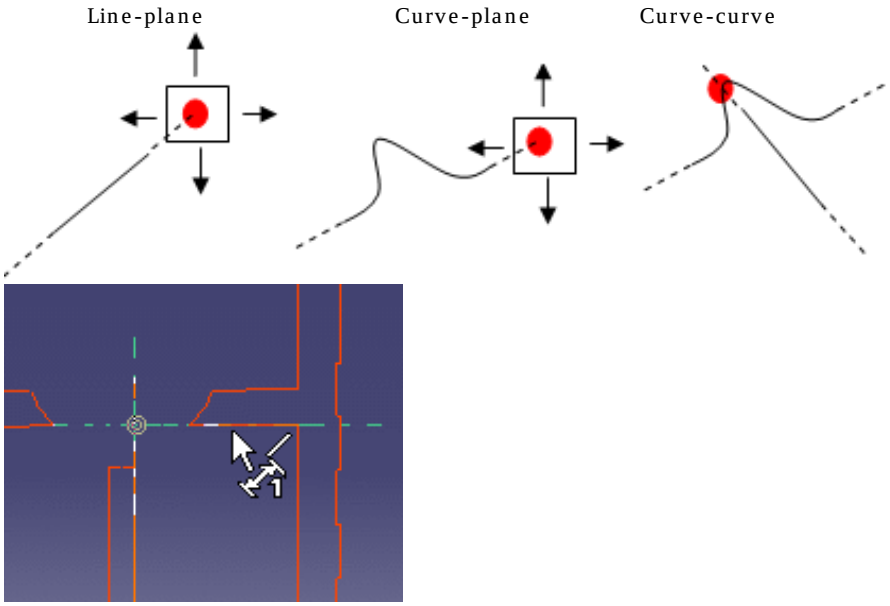
- Point only: measures distances between points. Dynamic highlighting is limited to points.
- Edge only, Surface only: measures distances and, if applicable, angles between edges and surfaces respectively. Dynamic highlighting is limited to edges or surfaces and is thus simplified compared to the Any geometry mode. All types of edge are supported.
- Product only: measures distances between products.
  Products can be specified by selecting product geometry, for example an edge or surface, in the geometry area or the specification tree.
- Picking axis: measures distances and, if applicable, angles between an entity and an infinite line perpendicular to the screen.

  Simply click to create infinite line perpendicular to the screen.



- Intersection: measures distances between points of intersection between two lines/curves/edges or a line/curve/edge and a surface. In this case, two selections are necessary to define selection 1 and selection 2 items.

  Geometrical entities (planar surfaces, lines and curves) are extended to infinity to determine the point of intersection. Curves are extended by tangency at curve ends.

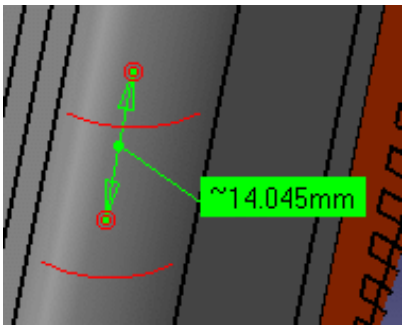


Note: Only intersections which result in points of intersection are managed.

- Edge limits: measures distances between endpoints or midpoints of edges. Endpoints only are proposed on curved surfaces.
- Arc center: measures distances between the centers of arcs.
- Center of 3 points arc: measures distances between the centers of arcs defined by 3 points.

To define arc center, click three points on the geometry.

Note: The resulting measure will always be approximate and non associative.



- Coordinate: measures distances between coordinates entered for selection 1 and/or selection 2 items.

4. Set the desired calculation mode in the Calculation mode drop-down list box.

Defining the Calculation Mode

- Exact else approximate (default mode): measures access exact data and wherever possible true values are given. If exact values cannot be measured, approximate values are given (identified by a ~ sign).
- Exact: measures access exact data and true values are given. Note that you can only select exact items in the geometry area or specification tree.
  In certain cases, in particular if products are selected, a warning dialog box informs you that the exact measure could not be made.
  After some geometric operations, vertices (and corresponding macropoints) may combine several representations on different supports (curves or surfaces). These representations are not all located in the same position in space which means that the exact position of the vertex cannot be determined. Only one vertex representation is visualized. Measure Between measurements are made with respect to the visualized representation. Measuring distance between two points therefore depends on the chosen representation. Any calculation errors are due to the fact that the exact position of the vertex cannot be determined.
- Approximate: measures are made on tessellated objects and approximate values are given (identified by a ~ sign).

Note: You can hide the display of the ~ sign using the **Tools -> Options** command (**General -> Parameters and Measure -> Measure Tools**).

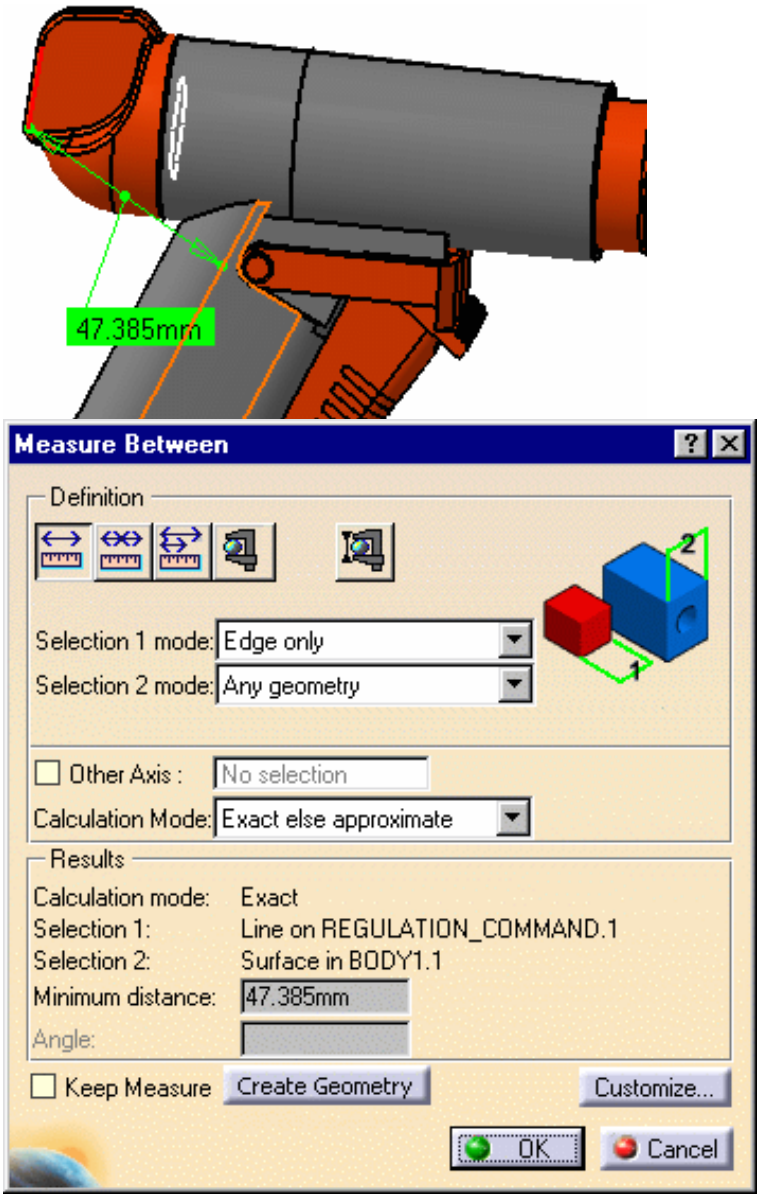
5. Click to select a surface, edge or vertex, or an entire product (selection 1).

Notes:

- The appearance of the cursor has changed to assist you.
- Dynamic highlighting of geometrical entities helps you locate items to click on.

6. Click to select another surface, edge or vertex, or an entire product (selection 2).

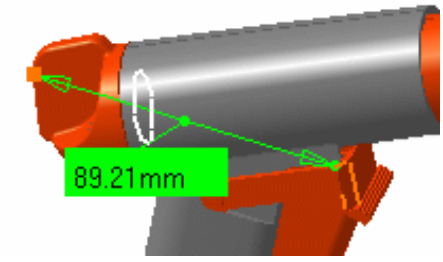
A line representing the minimum distance vector is drawn between the selected items in the geometry area. Appropriate distance values are displayed in the dialog box.



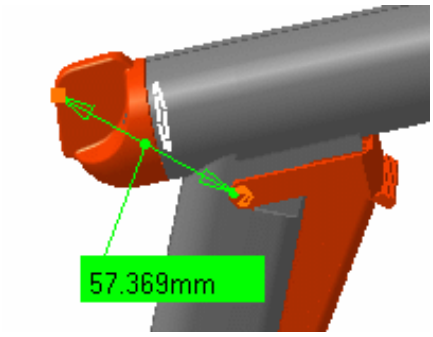
By default, the overall minimum distance and angle, if any, between the selected items are given in the Measure Between dialog box.

The number of decimal places, the display of trailing zeros and limits for exponential notation is controlled by the Units tab in the Options dialog box (**Tools ->Options**, **General ->Parameters and Measure**). For more information, see the *Infrastructure User's Guide*.

7. Select another selection and, if desired, selection mode.
8. Set the Measure type to Fan to fix the first selection so that you can always measure from this item.
9. Select the second item.



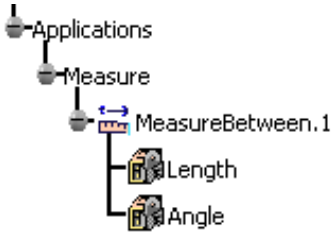
10. Select another item.



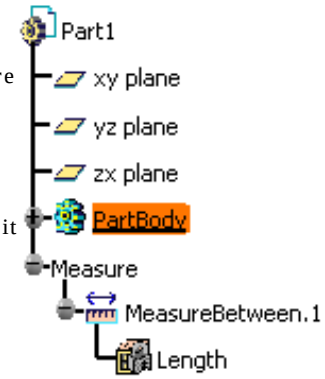
Using the Other Selection... command in the contextual menu, you can access the center of spheres.

11. Click **OK** when done.

If you checked the Keep Measure option in the Measure Between dialog box, your measures are kept as features and your specification tree will look something like this if measures were made on the active product.



Or like this, if measures were made on the active part.



Note: If the product is active, any measures on parts are placed in No Show.

Some measures kept as features are associative. In Design Mode, if you modify a part or move a part in a product structure context and the measure is impacted, it will be identified as not up-to-date in the specification tree. You can then update it locally have it updated automatically.

When measures are used to valuate parameters, an associative link between the measure and parameter is created. Measures can also be used in formulas.

Sectioning measure results

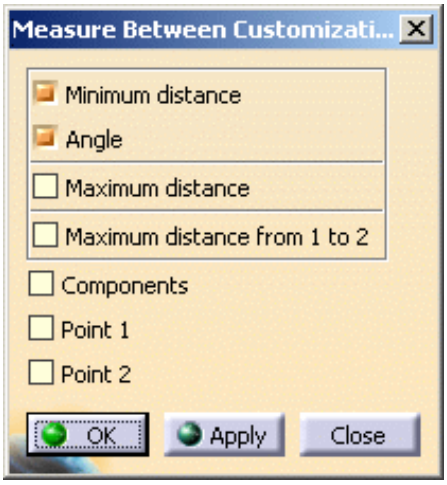
Having made and kept your measure, select it then click the Sectioning icon to section measure results. The plane is created parallel to the direction defined by the measure and sections entities selected for the measure only. All section plane manipulations are available.

Note: You may need an appropriate license to access the Sectioning command.

Customizing Measure Between

Customizing lets you choose what distance you want to measure:

- Minimum distance (and angle if applicable)
- Maximum distance
- Maximum distance from 1 to 2.



Note: These options are mutually exclusive. Each time you change option, you must make your measure again.

By default, minimum distances and if applicable, angles are measured.

You can also choose to display components and the coordinates of the two points (point 1 and point 2) between which the distance is measured.

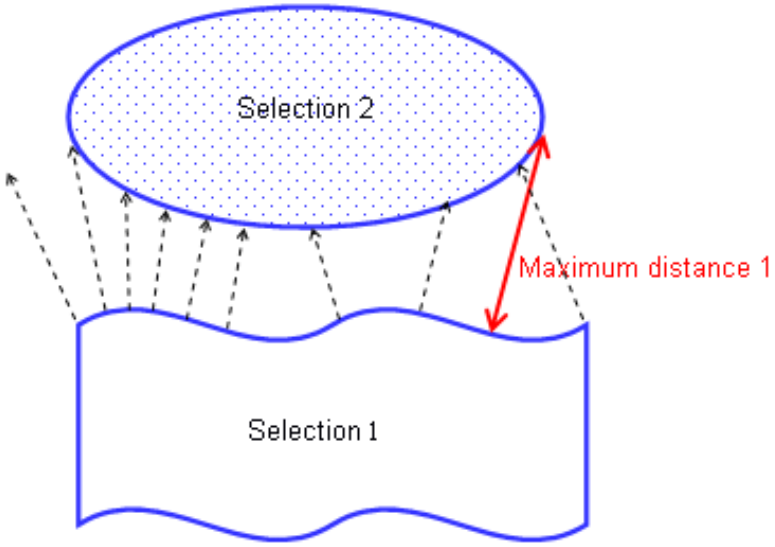
What you set in the dialog box determines the display of the results in both the geometry area and the dialog box.

Measuring Maximum Distance

You can measure the maximum distance between two surfaces, two volumes or a surface and a volume.

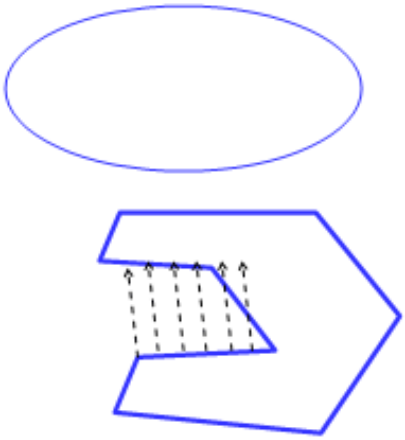
Distance is measured normal to the selection and is always approximate. Two choices are available:

- Maximum distance from 1 to 2: gives the maximum distance of all distances measured from selection 1.
  Note: This distance is, in general, not symmetrical.

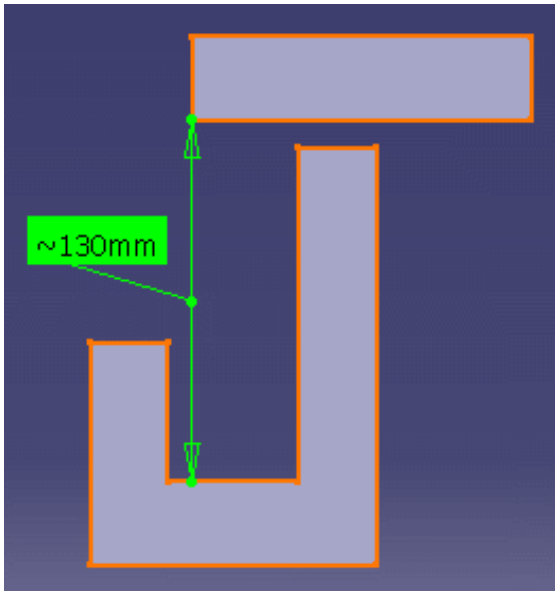


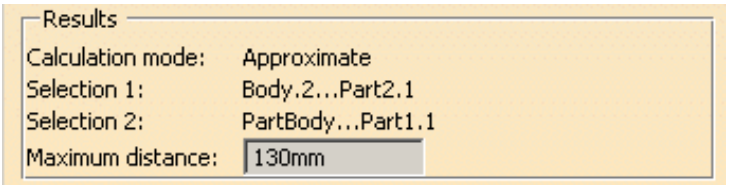
- Maximum distance: gives the highest maximum distance between the maximum distance measured from selection 1 and the maximum distance measured from selection 2.

Note: All selection 1 (or 2) normals intersecting selection 1 (or 2) are ignored.



1. Click Customize... and check the appropriate maximum distance option in the Measure Between Customization dialog box, then click **OK**.
2. Make your measure:
   - Select the desired measure type
   - Set the desired selection modes
   - Set the desired calculation mode
   - Click to select two surfaces, two volumes or a surface and a volume.





3. Click **OK** when done.

Measuring Distances in a Local Axis System

An Other Axis option in the dialog box lets you measure distance in a local axis system.

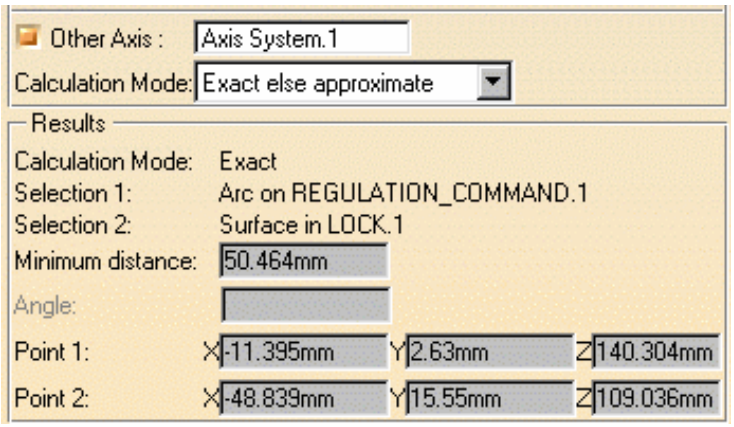
This type of measure is associative: if you move the axis system, the measure is impacted and can be updated.

You will need a V5 axis system.

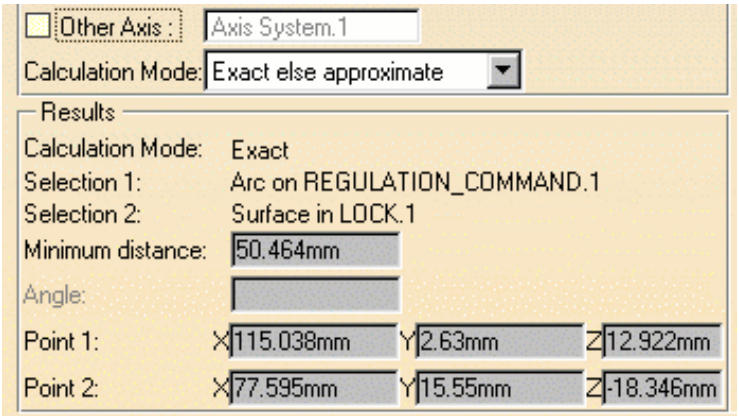
1. Select the Other Axis checkbox in the dialog box.
2. Select a V5 axis system in the specification tree or geometry area.

3. Make your measure.

   In the examples below, the measure is a minimum distance measure and the coordinates of the two points between which the distance is measured are shown.



   Same measure made with respect to absolute axis system:



   Note: All subsequent measures are made with respect to the selected axis system.

4. To change the axis system, click the Other Axis field and select another axis system.
5. To return to the absolute axis system, click to clear the Other Axis checkbox.
6. Click **OK** when done.

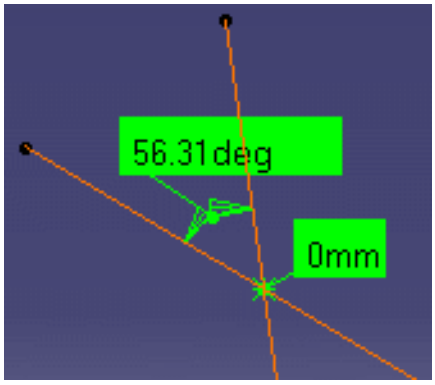
Measuring Angles

- Exact angles
- Complementary angles

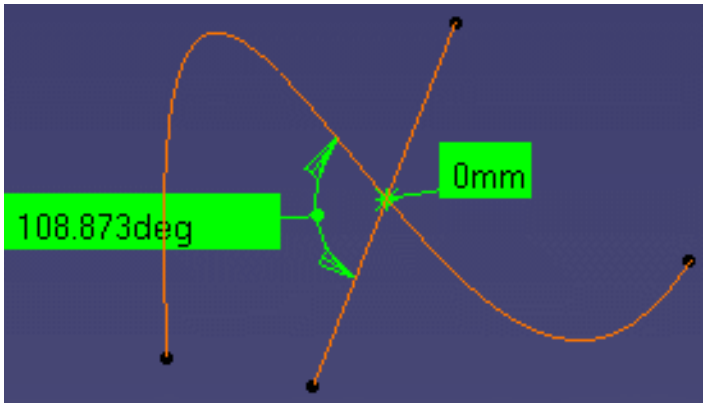
Exact Angles

The Measure Between command lets you measure exact angles between the following geometrical entities that have (at least) one common point.

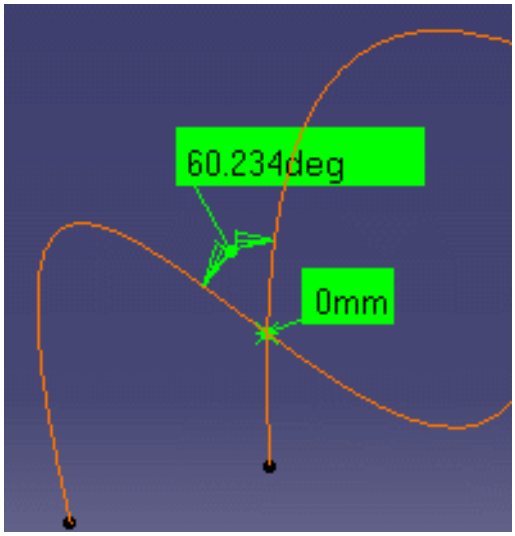
Two lines (even if not in the same plane):



A line and a curve:

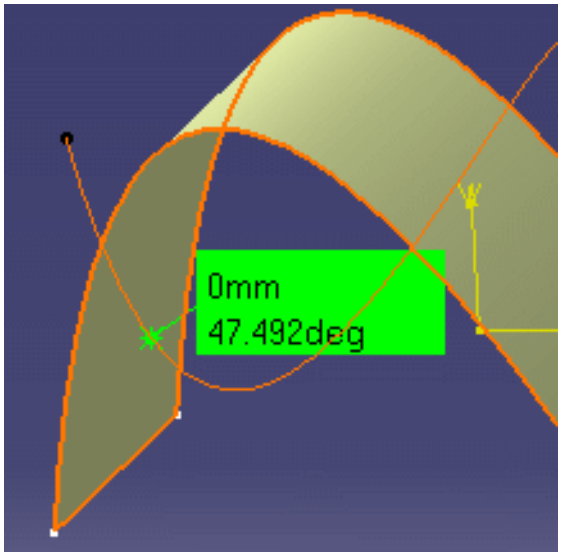


Two curves:

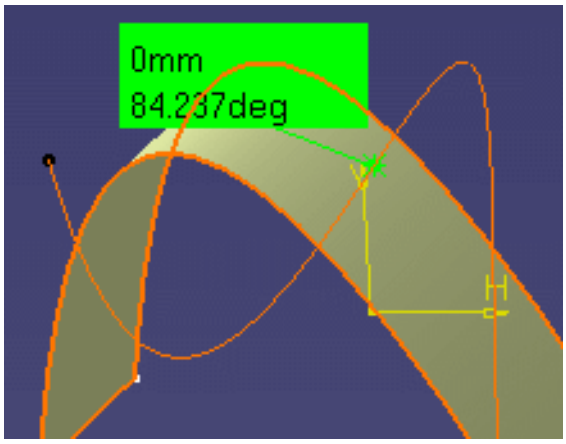


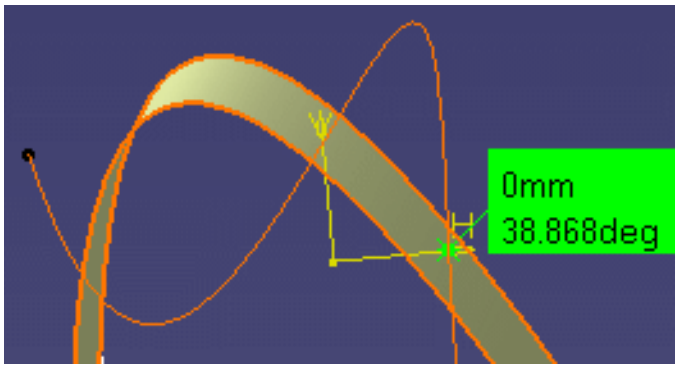
Note: In the above three cases, if entities intersect more than once, the measure is made at the point of intersection nearest the point at which selection 1 is made.

A curve and a surface:

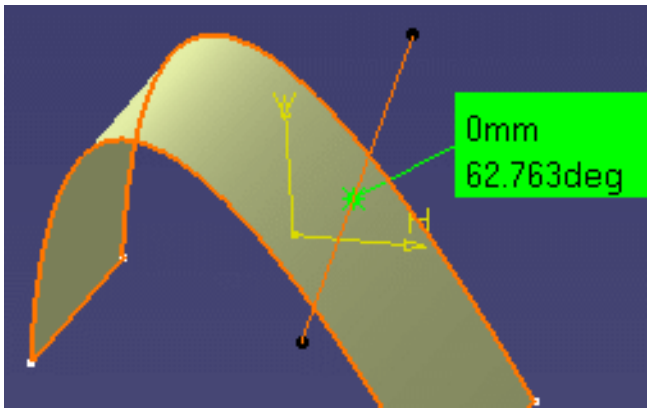


Note: If the curve and surface intersect more than once, the measure is made at the point of intersection nearest the point of the selection on the curve.

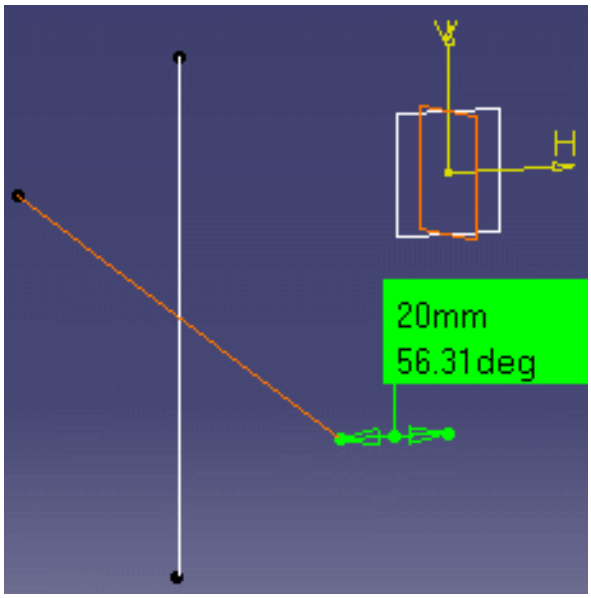




A line and a surface:



A line and a plane:

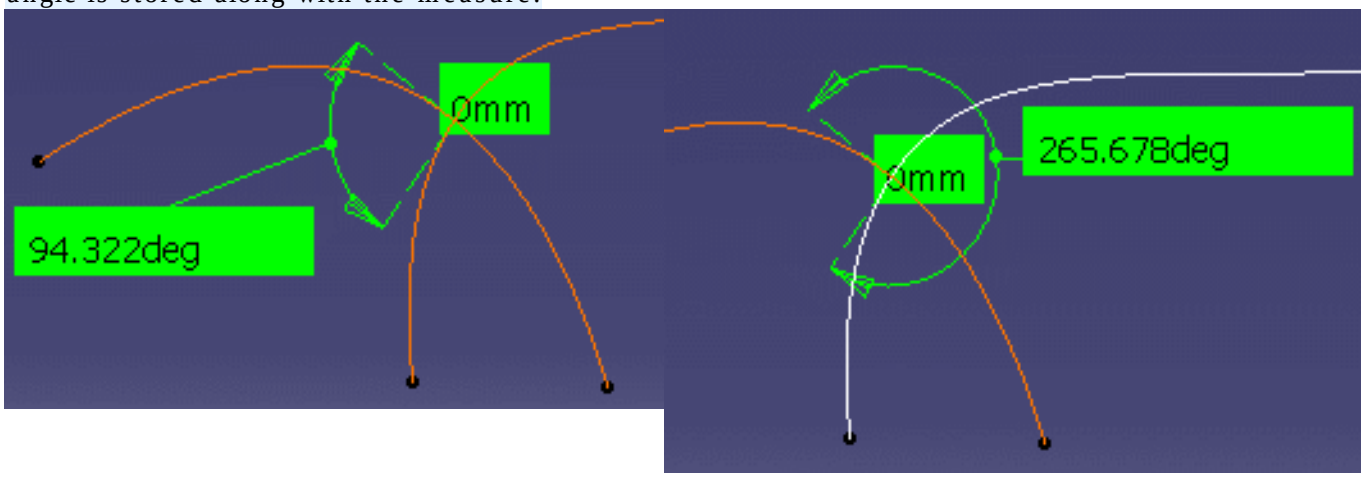


Two surfaces: You can also measure the angle between two surfaces provided both surfaces are planar.

Complementary Angles

You can obtain the complementary angle (360° - the initial angle measured) when measuring between two curves: drag the angle line to show the complementary angle.

Note: The dialog box and knowledge parameters are refreshed. The value of the complementary angle is stored along with the measure.



Measure Cursors

The appearance of the Measure Between and Measure Item cursor changes as you move it over items to reflect the measure command you are in and to help you identify the selection. Dynamic highlighting of surfaces, points, and vertices, etc. also helps you locate items to click on.

Measure Between	Measure Item	Geometry
		Surface
		Planar surface
		Line
		Curve
		Point
		Circle
		Sphere
		Cylinder
		Volume

In Measure Between, a number (1 for selection 1 and 2 for selection 2) identifies where you are in your measure.

Measuring Properties

The Measure Item command lets you measure the properties associated to a selected item (points, edges, surfaces and entire products).

This section deals with the following topics:

Insert the following sample model files: ATOMIZER.model, BODY1.model, BODY2.model, LOCK.model, NOZZLE1.model, NOZZLE2.model, REGULATION_COMMAND.model, REGULATOR.model, TRIGGER.model and VALVE.model.

They are to be found in the online documentation filetree in the common functionalities sample folder cfysm/samples.
Restriction: Neither Visualization Mode nor cgr files permit selection of individual vertices.

Note: In the No Show space, this command is not accessible.

Measuring Properties

This task explains how to measure the properties associated to a selected item.

1. Switch to Design Mode (**Edit ->Representations ->Design Mode**).

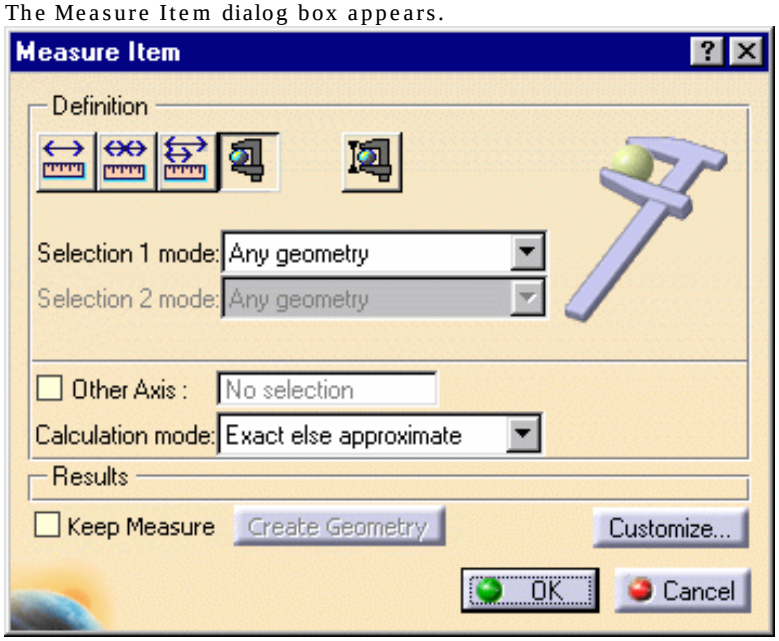
2. Set **View -> Render Style** to Shading with Edges.

   Note: You cannot use this command, if Shading only is selected.

3. Click the Measure Item  icon.

   In DMU, you can also select **Analyze -> Measure Item** from the menu bar.

   The Measure Item dialog box appears.



By default, properties of active products are measured with respect to the product axis system. Properties of active parts are measured with respect to the part axis system.
Note: This distinction is not valid for measures made prior to Version 5 Release 8 Service Pack 1 where all measures are made with respect to the absolute axis system.

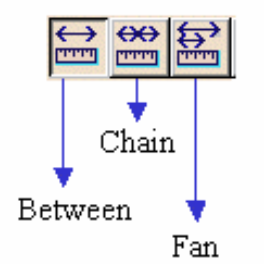
Dialog box options

- You can also measure properties with respect to a local V5 axis system.
- The Keep Measure option lets you keep current and subsequent measures as features. This is useful if you want to keep measures as annotations for example.

  Some measures kept as features are associative and can be used to valuate parameters or in formulas.
  In the Drafting and Advanced Meshing Tools workbenches, measures are done on-the-fly. They are not persistent. This means that they are not associative and cannot be used as parameters.

- A Create Geometry option in the dialog box lets you create the center of gravity from measure results.
- A Customize... option lets you customize the display of measure results.

Accessing other measure commands



- The Measure Between command is accessible from the Measure Item dialog box. Simply click one of the Measure Between icons in the Definition box to switch commands.
- In DMU, the Measure Thickness command is also accessible from the Measure Item dialog box. For more information, see the appropriate task in the *DMU Space Analysis User's Guide*.

P1-Only Functionality

In P1, the Measure Tools toolbar appears.
This toolbar has two icons:

- Measure Dialogs : lets you show or hide the associated dialog box.
- Exit Measure : lets you exit the measure. This is useful when the dialog box is hidden.

4. Set the desired measure mode in the Selection 1 mode drop-down list box.

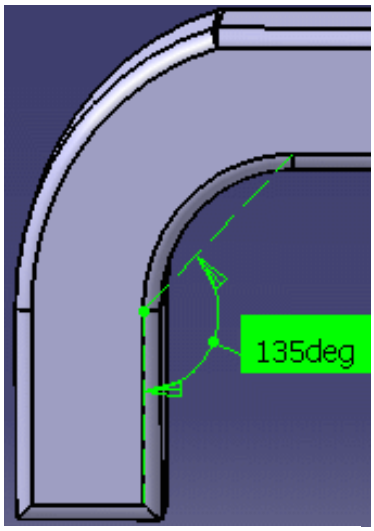
Defining the Selection 1 Mode

- Any geometry (default mode): measures the properties of the selected item (point, edge, surface or entire product).
- Point only: measures the properties of points. Dynamic highlighting is limited to points.
- Edge only: measures the properties of edges. All types of edge are supported.
- Surface only: measures the properties of surfaces.

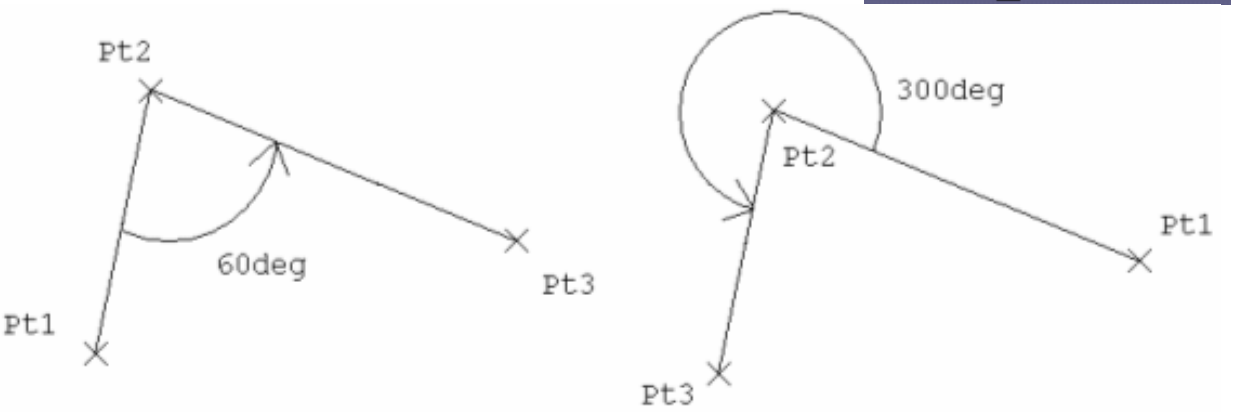
  In the last three modes, dynamic highlighting is limited to points, edges or surfaces depending on the mode selected, and is thus simplified compared to the Any geometry mode.

- Product only: measures distances between products.
  Products can be specified by selecting product geometry, for example an edge or surface, in the geometry area or the specification tree.
- Angle by 3 points: measures the angle between two lines themselves defined by three points.

To define lines, select three existing points in the geometry area or in the specification tree.
Note: You cannot select picking points.
Smart selection is offered. This means that a sphere or circle, for example, are seen as points.



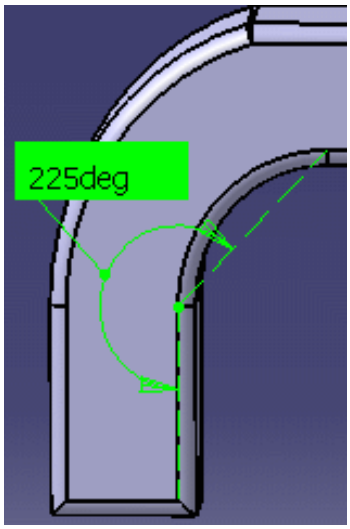
The resulting angle is always positive. It is measured in a counterclockwise direction and depends on the order in which points were selected as well as your viewpoint (the normal to the plane is oriented towards you).



You can drag the angle line to show the complementary angle (360° - the initial angle measured).

You can also obtain the complementary angle when measuring the angle on arcs.

Note: The dialog box and knowledge parameters are refreshed. The value of the complementary angle is stored along with the measure.



- Thickness (DMU only): measures the thickness of an item. For more information, see the appropriate task in the *DMU Space Analysis User's Guide*.
- The Measure Item command lets you access the radius of an exact cylinder or sphere.
- The Measure Item command also recognizes ellipse-type conic sections. Description: Ellipse in Part1.1
- Using the Other Selection... command in the contextual menu, you can access the axis of a cylinder as well as the center of a sphere to, for example, measure between two cylinder axes.

5. Set the desired calculation mode in the Calculation mode drop-down list box.

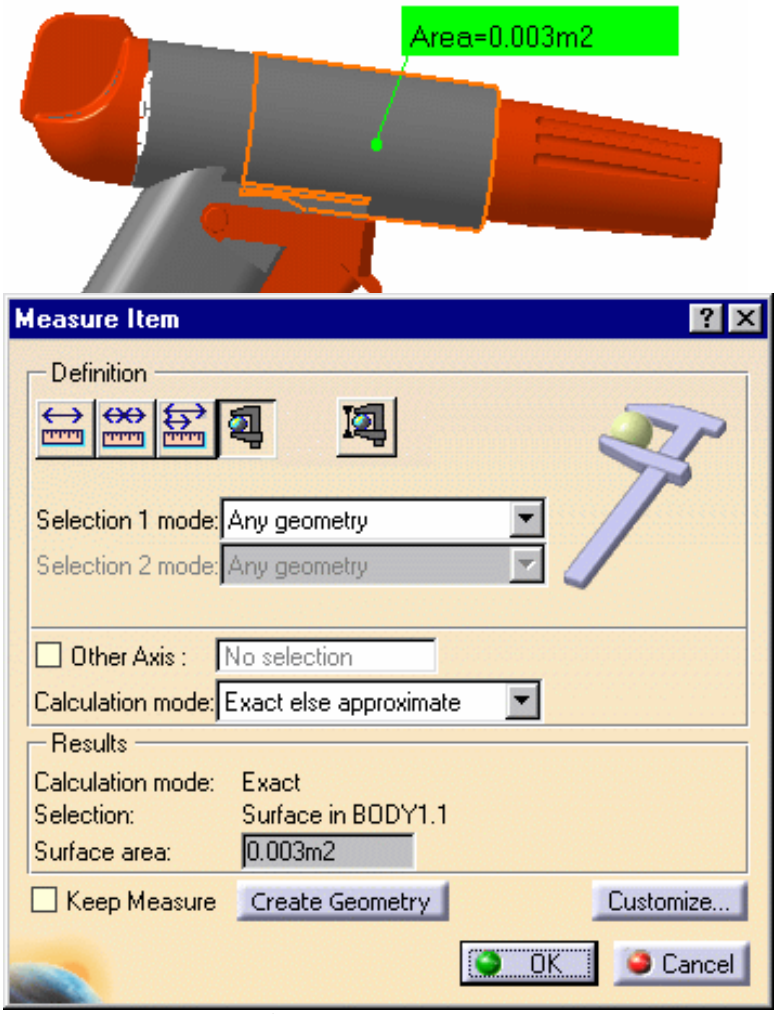
Defining the Calculation Mode

- Exact else approximate (default mode): measures access exact data and wherever possible true values are given. If exact values cannot be measured, approximate values are given (identified by a ~ sign).
- Exact: measures access exact data and true values are given. Note that you can only select exact items in the geometry area or specification tree.
  In certain cases, in particular if products are selected, a warning dialog box informs you that the exact measure could not be made.
- Approximate: measures are made on tessellated objects and approximate values are given (identified by a ~ sign).

Note: You can hide the ~ sign using the **Tools -> Options** command (**General ->Parameters and Measure ->Measure Tools**).

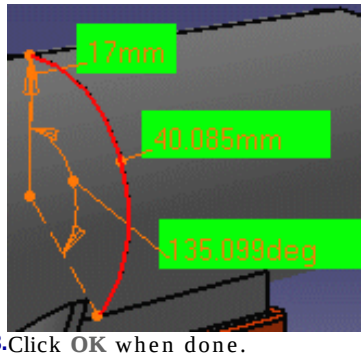
6. Click to select the desired item.

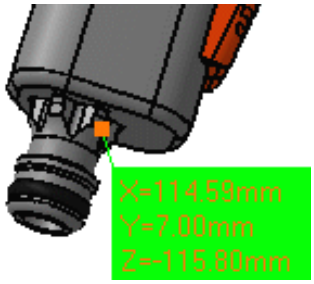
Note: The appearance of the cursor has changed to assist you.



The dialog box gives information about the selected item, in our case a surface and indicates whether the result is an exact or approximate value. The surface area is also displayed in the geometry area.
The number of decimal places, the display of trailing zeros and limits for exponential notation is controlled by the Units tab in the Options dialog box (**Tools-> Options**, **General-> Parameters and Measure**). For more information, see the Infrastructure User's Guide.

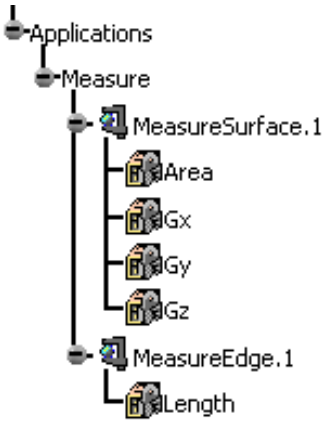
7. Try selecting other items to measure associated properties.



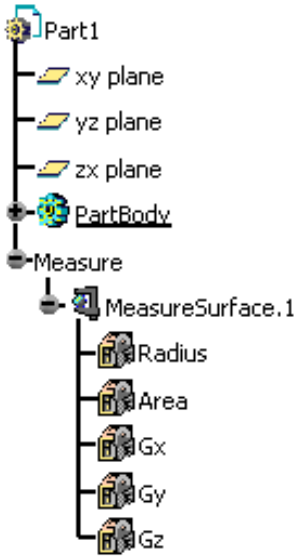


8. Click **OK** when done.

If you checked the Keep Measure option in the Measure Item dialog box, your measures are kept as features and your specification tree will look something like this if properties of the active product were measured.



Or like this, if properties were those of the active part.



Note: If the product is active, any measures made on the active part are placed in No Show.

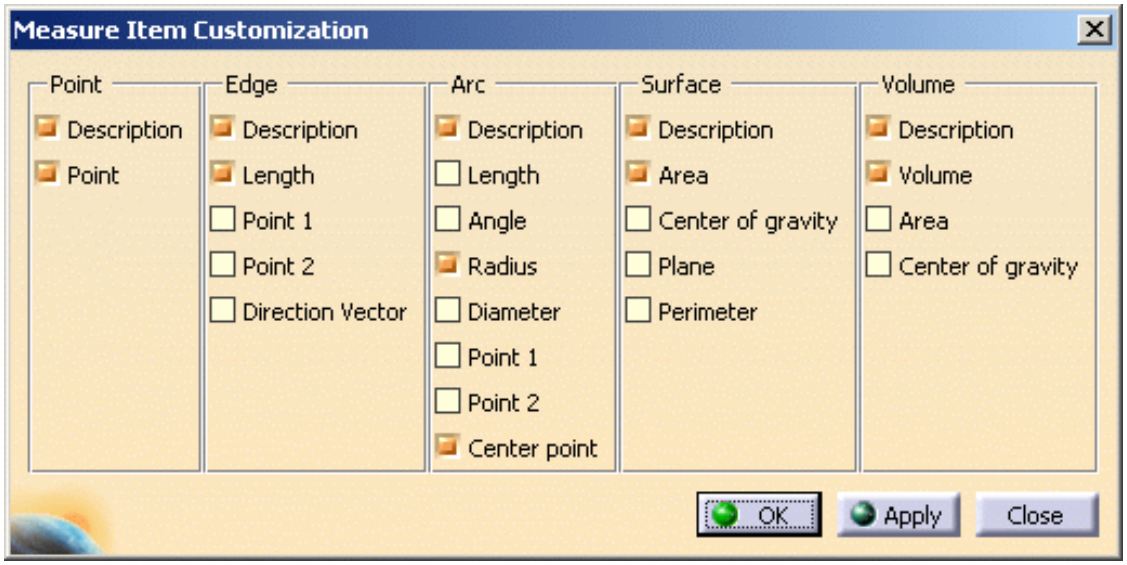
Some measures kept as features are associative. In Design Mode, if you modify a part or move a part in a product structure context and the measure is impacted, it will be identified as not up-to-date in the specification tree. You can then update it locally have it updated automatically.

When measures are used to valuate parameters, an associative link between the measure and parameter is created. Measures can also be used in formulas.

Customizing the Display

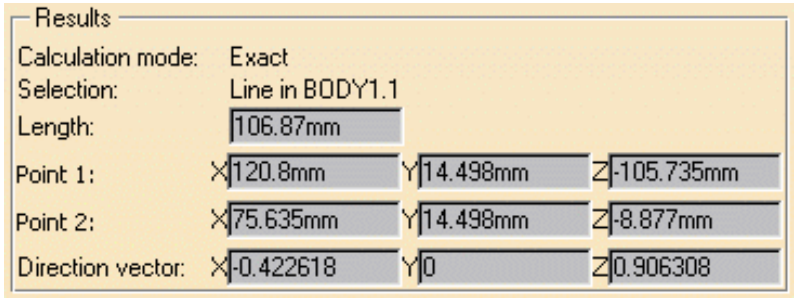
Customizing lets you choose the properties you want to see displayed in both the geometry area and the dialog box.

1. Click Customize... in the Measure Item dialog box to see the properties the system can detect for the various types of item you can select. By default, you obtain:



Edges

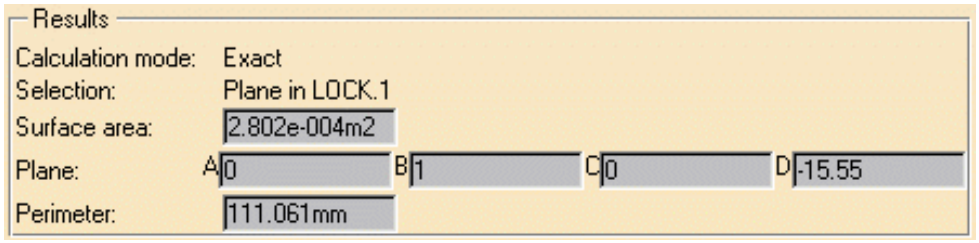
The system detects whether the edge is a line, curve or arc, taking model accuracy into account and displays the properties as set in the Measure Item Customization dialog box.



Note: If the angle of an arc is less than 0.125 degrees, only the arc length is displayed in the geometry area. The angle and radius are not displayed.

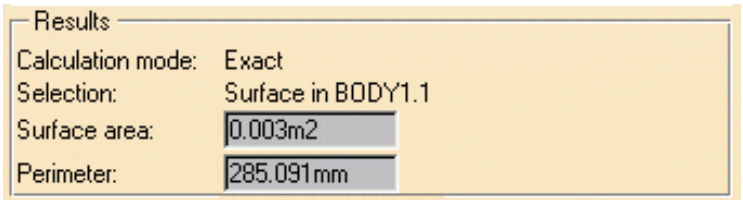
Surfaces

- **Center of gravity**: The center of gravity of surfaces is visualized by a point. In the case of non planar surfaces, the center of gravity is attached to the surface over the minimum distance.
- **Plane**: gives the equation of a planar face. The equation of a plane is: **Ax + By + Cz + D = 0**.



Note that there is an infinite number of equations possible (and an infinite number of solutions for values ABC and D). The result given by Measure Item does not necessarily correspond to that in the feature specification. This is because the measure is based on topology and does not know the feature specification associated with the measured item.

- **Perimeter**: Visualization mode does not permit the measure of surface perimeter.



2. Set the properties you want the system to detect, then click **Apply** or **Close**.
   The Measure Item dialog box is updated if you request more properties of the item you have just selected.
3. Select other items to measure associated properties.

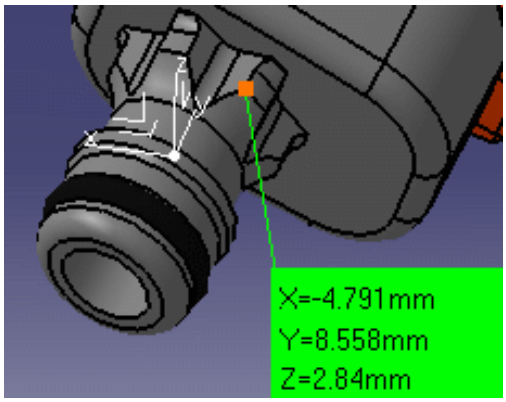
Measuring Properties in a Local Axis System

An Other Axis option in the dialog box lets you measure properties in a local axis system.

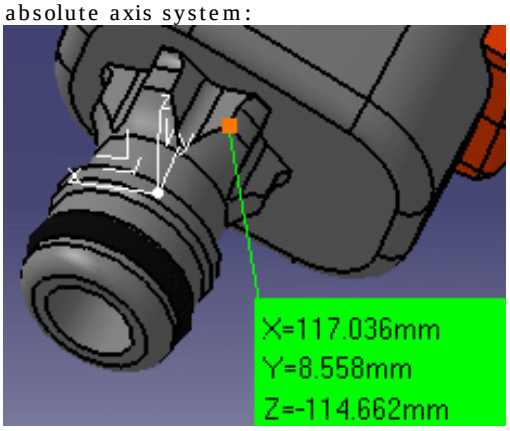
This type of measure is associative: if you move the axis system, the measure is impacted and can be updated.
You will need a V5 axis system.

1. Select the Other Axis checkbox in the Measure Item dialog box.
2. Select a V5 axis system in the specification tree or geometry area.
3. Make your measure.

Measure made with respect to local axis system:



Same measure made with respect to absolute axis system:



Note: All subsequent measures are made with respect to the selected axis system.

4. To change the axis system, click the Other Axis field and select another axis system.
5. To return to the main axis system, click to clear the Other Axis checkbox.
6. Click **OK** when done.

Measuring Thickness

This task explains how to measure the thickness of a selected item along the normal to the surface at the picking point.
Important: This measure is approximate. It is not asssociative and therefore cannot be updated.

This command can be accessed from the Measure Item and Measure Between commands.

Insert the following sample model files: ATOMIZER.model, BODY1.model, BODY2.model, LOCK.model, NOZZLE1.model, NOZZLE2.model, REGULATION_COMMAND.model, REGULATOR.model, TRIGGER.model and VALVE.model.

They are to be found in the online documentation filetree in the common functionalities samples folder **cfysm/samples**.

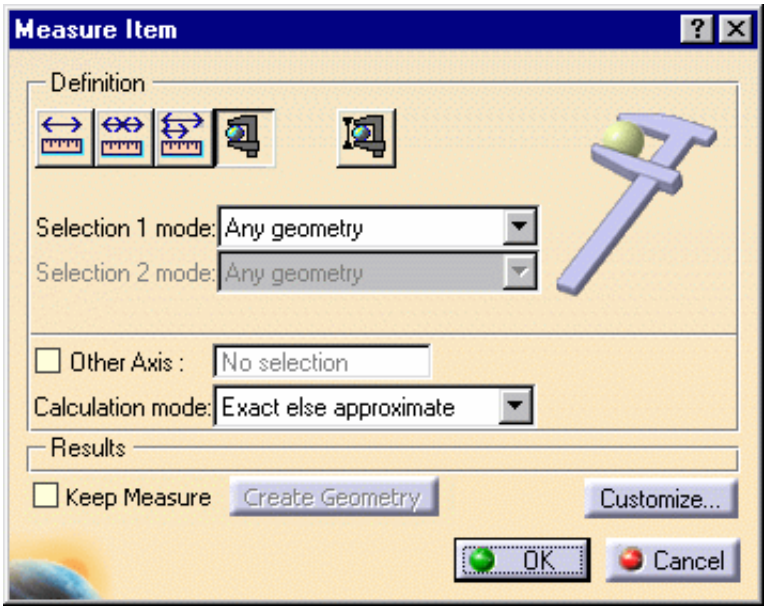
1. Switch to Design Mode (**Edit -> Representations -> Design Mode**).

2. Set **View -> Render Style** to Shading with Edges.

   Note: You cannot use this command, if Shading only is selected.

3. Click the Measure Item icon.

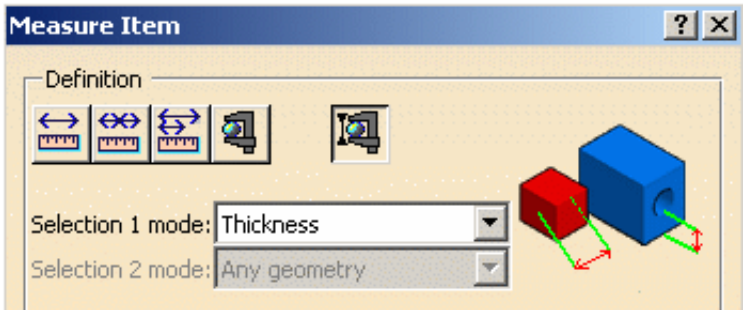
   The Measure Item dialog box appears.



For more information on measuring other properties of selected items (points, edges, surfaces and entire products), see Measuring Properties.

The Keep Measure option lets you keep current and subsequent measures as features.

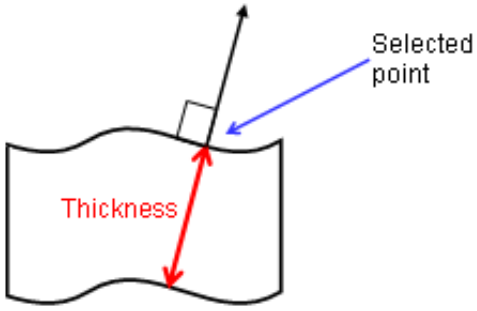
4. Click the Thickness icon in the dialog box or set the measure mode in the Selection 1 mode drop-down list box to Thickness.

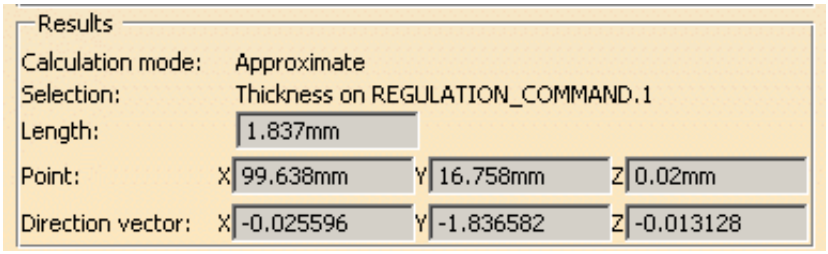


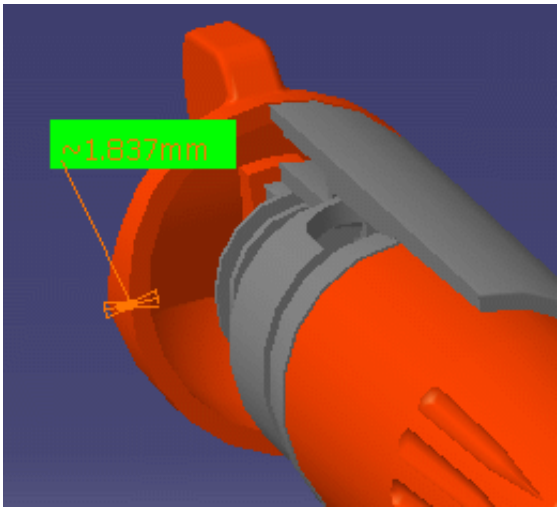
5. Make your measure.

Note: The dynamic feedback as you move your cursor over the item.

Thickness is measured along the normal to the surface at the picking point. An approximate value is given.







6. Click **OK** when done.

If you selected the Keep Measure check box in the dialog box, your measure is kept as a feature.
Important: This measure is not asssociative and therefore cannot be updated. This is identified in the specification tree by the measure icon plus a lock.

The Properties command (Graphics tab) lets you change the fill color and transparency as well as the color, linetype and thickness of measure lines.

Note: You cannot vary transparency properties, the current object is either the selected color or transparent.

Measuring Inertia

The Measure Inertia command lets you measure:

- 3D inertia properties of surfaces and volumes (explained below)
- 2D inertia properties of plane surfaces.

Note: In the No Show space, this command is not accessible.

This section deals with the following topics:

Measuring 3D Inertia

This task explains how to measure the 3D inertia properties of an object.

You can measure the 3D inertia properties of both surfaces and volumes, as well as retrieve the density or surface density if valuated from V4 model type documents. You can also retrieve inertia equivalents set in Knowledgeware formulas.

The area, density, mass and volume (volumes only) of the object are also calculated.

Note: You cannot measure inertia properties of either wireframe or infinite elements.

For examples showing 3D inertia properties measured on surfaces. To find out more about notations used.

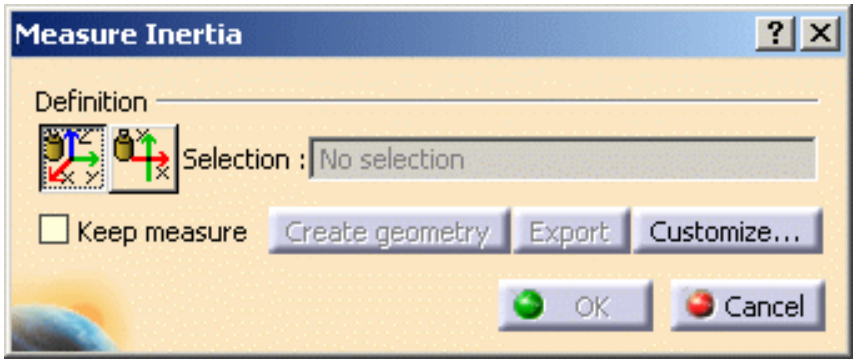
Insert the Valve.cgr document from the samples folder. It is to be found in the online documentation filetree in the common functionalities sample folder cfysa/samples.

1. Click the Measure Inertia  icon.

In DMU, you can also select **Analyze -> Measure Inertia** from the menu bar.

The Measure Inertia dialog box appears. By default 3D inertia properties are measured.

The Measure 2D Inertia icon lets you measure 2D inertia properties of plane surfaces.

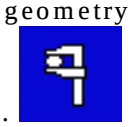


Dialog box options

- A Keep Measure option in the dialog box lets you keep current and subsequent measures as features in the specification tree. Some measures kept as features are associative and can be used as parameters.

In the Drafting workbench, the Keep Measure option is not available. Measures are done on-the-fly. They are not persistent. This means that they are not associative and cannot be used as parameters.

- A Create Geometry option lets you create the center of gravity and the axis system for principal axes in a part from inertia results.
- An Export option lets you write results to a text file.
- A Customize... option lets you define what will be computed and displayed in the dialog box.

Note: When you move the cursor over the geometry or specification tree, its appearance changes to reflect the measure command you are in. 

2. Click to select the desired item in the specification tree, for example Valve.

Selecting Items

- In the geometry area, you can select individual faces and edges on cgr files and in Visualization mode.

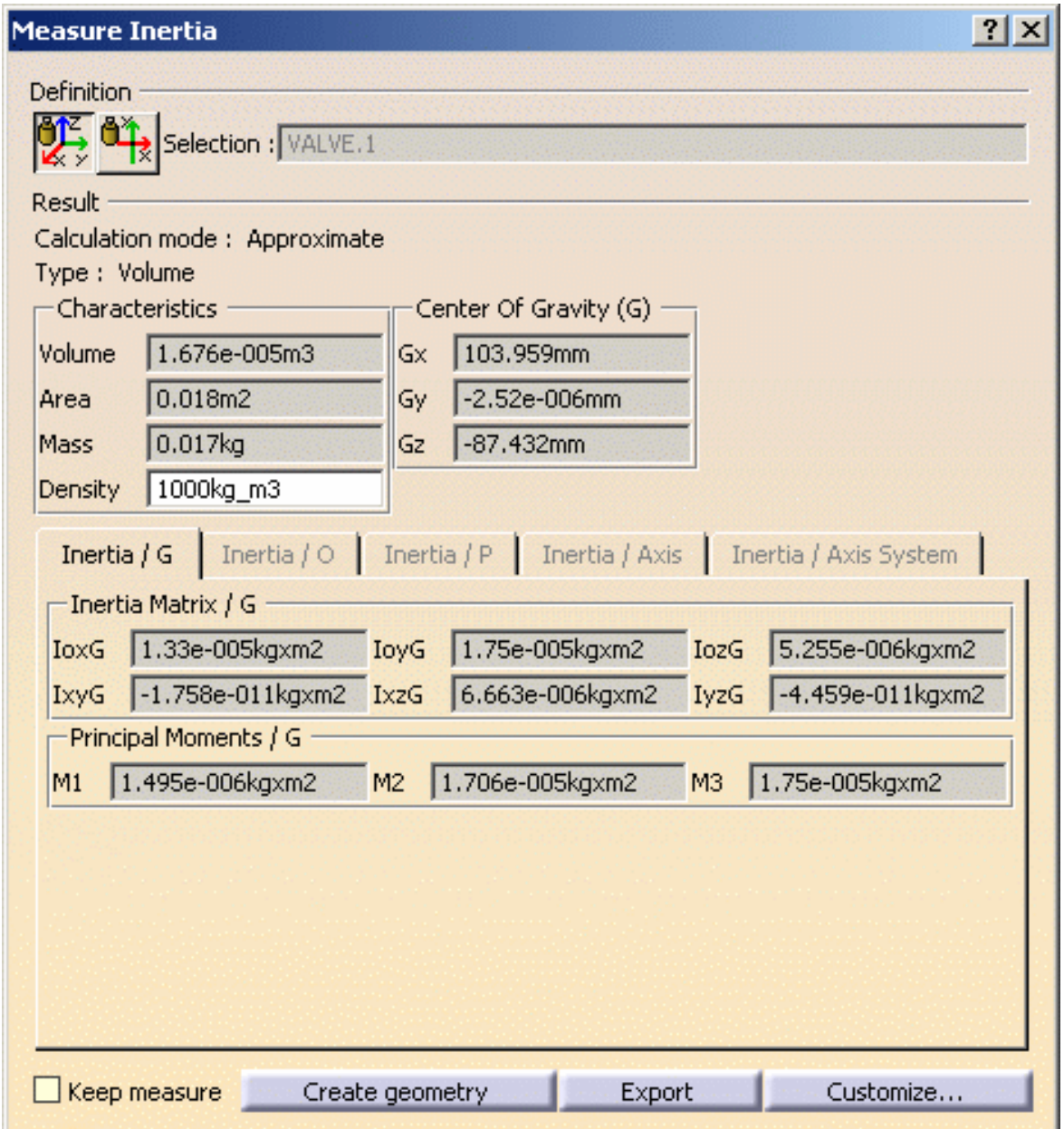
- Ctrl-click in the geometry area or the specification tree to add other items to the initial selection.
- Shift-click in the specification tree to make a multiple selection.
- Drag (using the left mouse button) to select items using the bounding outline.
- (P2 only) Use the Group command to make your multiple selection.

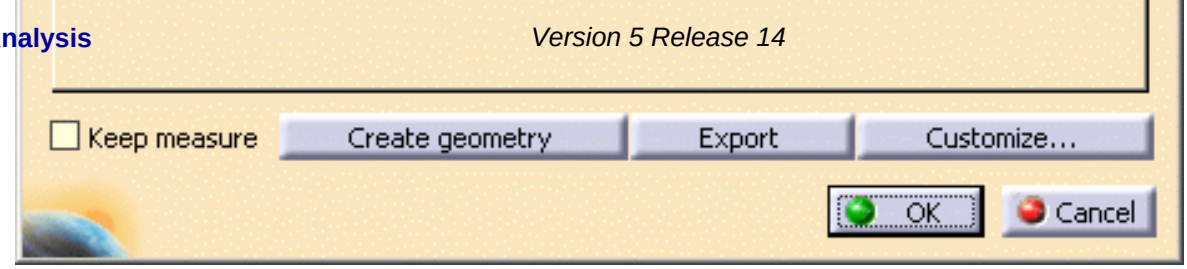
Notes:

- Only items of the same type can be included in a multiple selection or a bounding outline; you cannot mix volumes and surfaces.
- Inertia measures made on a multiple selection of items are not associative.

The **Dialog Box** expands to display the results for the selected item.

The measure is made on the selection, geometry, assembly or part. To measure the inertia of individual sub-products making up an assembly and see the results in the document window, you must select the desired sub-product.
In our example, the item selected has no sub-products.





The dialog box identifies the selected item and indicates whether the calculation is exact or approximate:

- In Design mode, measures access exact data and wherever possible true values are given. Note that it is possible to obtain an exact measure for most items in design mode.
- In Visualization mode, measures are made on tessellated items and approximate values are given.

In addition to the center of gravity G, the principal moments of inertia M and the matrix of inertia calculated with respect to the center of gravity, the dialog box also gives the area, volume (volumes only), density and mass of the selected item.

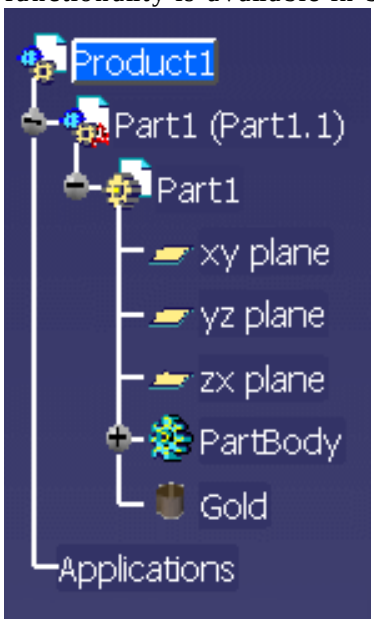
You can also compute and display the principal axes A. To do so, you must first activate the appropriate option in the Measure Inertia Customization dialog box.

The density is that of the material, if any, applied to a product, part or part body:

- If no density is found, a default value is displayed. You can, if desired, edit this value. If you do so, all the other inertia values are re-calculated. The default value is 1000 kg/m3 for volumes and 10 kg/m2 for surfaces.
- If sub-products or part bodies have different densities, the wording Not uniform is displayed.

Notes:

- You can access the density of parts saved as CGR files and opened in visualization mode. This functionality is available in both a part and a product context.



To do so:

- Select the Save density in cgr option in the Meaure Tools tab (**Tools ->Options ->General ->Parameters and Measure**).
- Open a part to which material has been applied and save as CGR type.
  The density is stored in the CGR file.

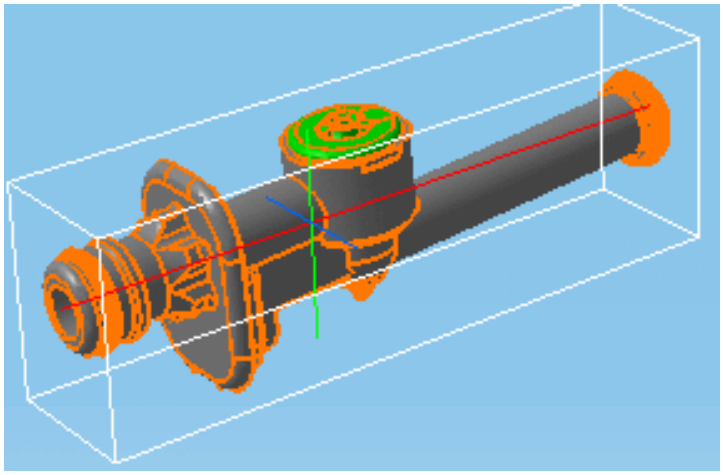
Important: The material must be applied to the part node. If materials are applied to part bodies, no density is saved.

- Close the Part document.
- Open the CGR file or switch to DMU Space Analysis and insert the part saved as CGR, then measure the inertia.

- You must be in design mode to access the density of part bodies to which materials have been applied.
- Unless specified otherwise, material inheritance is taken into account.
- Density is a measure of an item's mass per unit volume expressed in kg/m3; surface density is a measure of an item's mass per unit area expressed in kg/m2.

The number of decimal places, the display of trailing zeros and limits for exponential notation is controlled by the Units tab in the Options dialog box (**Tools ->Options**, **General ->Parameters and Measure**).

In the Geometry Area, axes of inertia are highlighted and a bounding box parallel to the axes and bounding the selected item also appears.

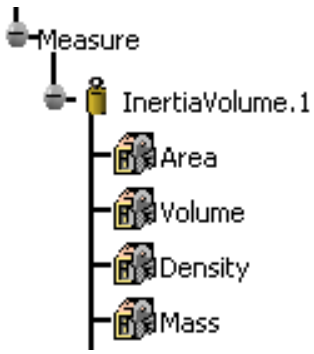


Color coding of axes:

- Red: axis corresponding to the first moment M1
- Green: axis corresponding to the second moment M2
- Blue: axis corresponding to third moment M3.

3. Click Customize... to customize the inertia computation and define what will be exported to the text file.
4. Click **OK** when done.

If you checked the Keep Measure option in the Measure Inertia dialog box, your measures are kept as features and your specification tree will look something like this.

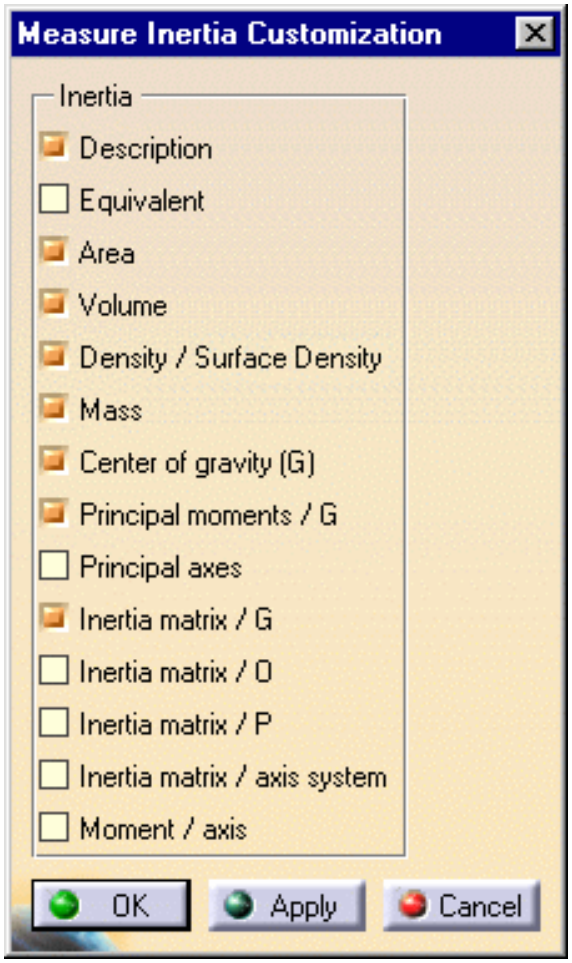


Some measures kept as features are associative and can be used as parameters.

You can write a macro script to automate your task. See *Space Analysis* on the Automation Documentation Home Page.

Customizing Your Measure

You can, at any time, define what will be computed and displayed in the Measure Inertia dialog box.



1. Click Customize... in the Measure Inertia dialog box.

   The Measure Inertia Customization dialog box opens.

   Note: The inertia properties checked here are also the properties exported to a text file.
2. Click the appropriate options to compute and display in appropriate tabs of the Measure Inertia dialog box the:
   - Inertia equivalents
   - Principal axes
   - Inertia matrix with respect to the origin O
   - Inertia matrix with respect to a point P
   - Inertia matrix with respect to an axis system
   - Moment of inertia about an axis
3. Click **Apply** or **OK** in the Measure Inertia Customization dialog box when done.

Measuring 2D Inertia

This task explains how to measure the inertia properties of plane 2D surfaces.

You can measure the area, center of gravity, principal moments, inertia matrix as well as the principal axes.
You can measure the inertia properties of plane surfaces including DMU sections. The area of the surface is also calculated.

Note: You cannot measure inertia properties of either wireframe or infinite elements.
To find out more about notations used.

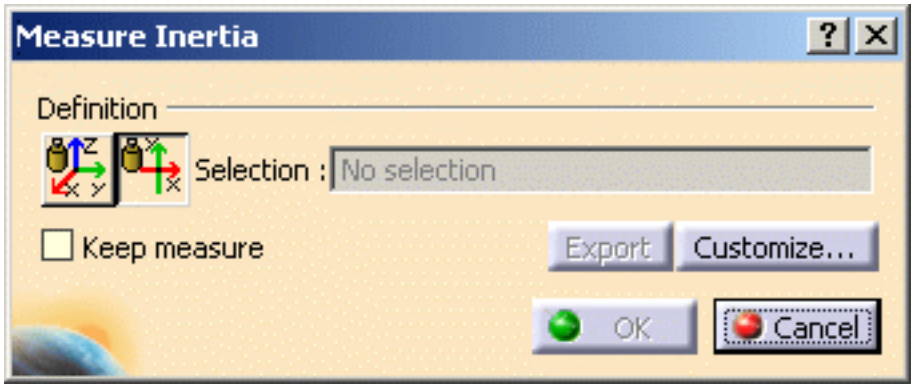
No sample document provided.

1. Click the Measure Inertia icon.

In DMU, you can also select **Analyze -> Measure Inertia** from the menu bar.

The Measure Inertia dialog box appears.

2. Click the Measure 2D Inertia icon.



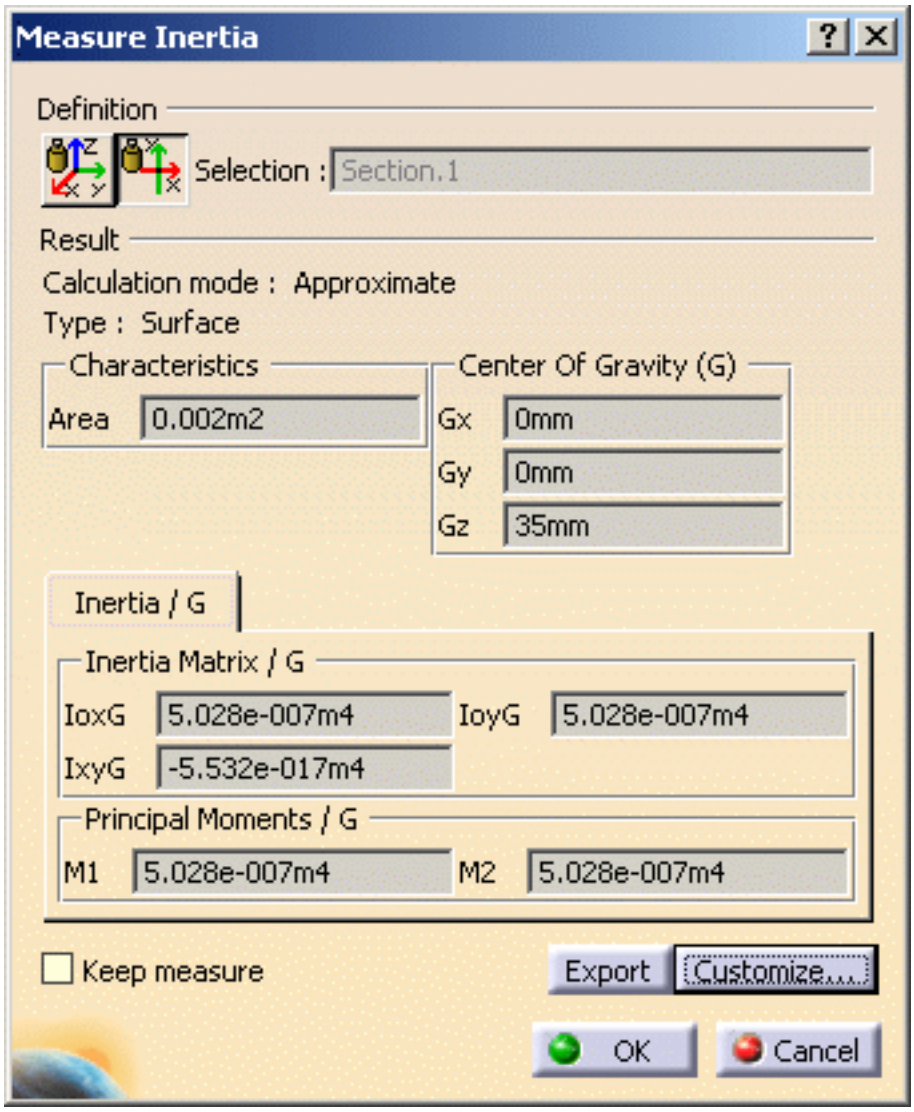
Dialog box options

- A Keep Measure option in the dialog box lets you keep current and subsequent measures as features. Some measures kept as features are associative and can be used as parameters.
  Note: This option is not available in the Drafting workbench.
- An Export option lets you write results to a text file.
- A Customize... option lets you define what will be computed and displayed in the dialog box.

When you move the cursor over the geometry or specification tree, its appearance changes to reflect the measure command you are in. 

3. Click to select a plane 2D surface in the geometry area or the specification tree.

The **Dialog Box** expands to display the results for the selected item.



The dialog box identifies the selected item, in our case a DMU section, and indicates whether the calculation is exact or approximate:

- In Design mode, measures access exact data and wherever possible true values are given. Note that it is possible to obtain an exact measure for most items in design mode.
- In Visualization mode, measures are made on tessellated items and approximate values are given.

In addition to the center of gravity G, the principal moments of inertia M and the matrix of inertia, the dialog box also gives the area of the selected item.

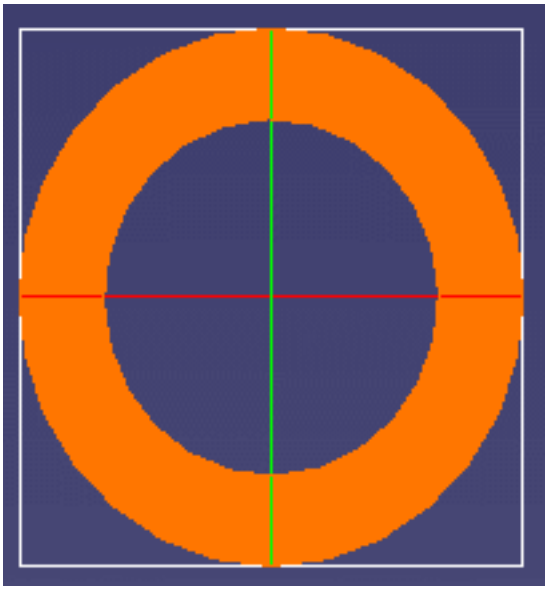
The center of gravity G is computed with respect to the document axis system. The matrix of inertia is expressed in an axis system whose origin is the center of gravity and whose vectors are the axes of inertia.

Note: The matrix of inertia and the principal moments do not take density into account.

You can also compute and display the principal axes A. To do so, you must first activate the appropriate option in the Measure Inertia Customization dialog box.

The number of decimal places, the display of trailing zeros and limits for exponential notation is controlled by the Units tab in the Options dialog box (**Tools ->Options**, **General ->Parameters and Measure)**.

In the Geometry Area, the axes of inertia are highlighted and a bounding box parallel to the axes and bounding the selected item also appears.



Color coding of axes:

- Red: axis corresponding to the first moment M1
- Green: axis corresponding to the second moment M2

4. Click **OK** in the Measure Inertia dialog box.

If you checked the Keep Measure option in the Measure Inertia dialog box, your measures are kept as features.

Customizing Your Measure

You can, at any time, define what will be computed and displayed in the tabs of the Measure Inertia dialog box.

When measuring 2D plane surfaces, in addition to the properties computed by default, you can compute and display the principal axes.

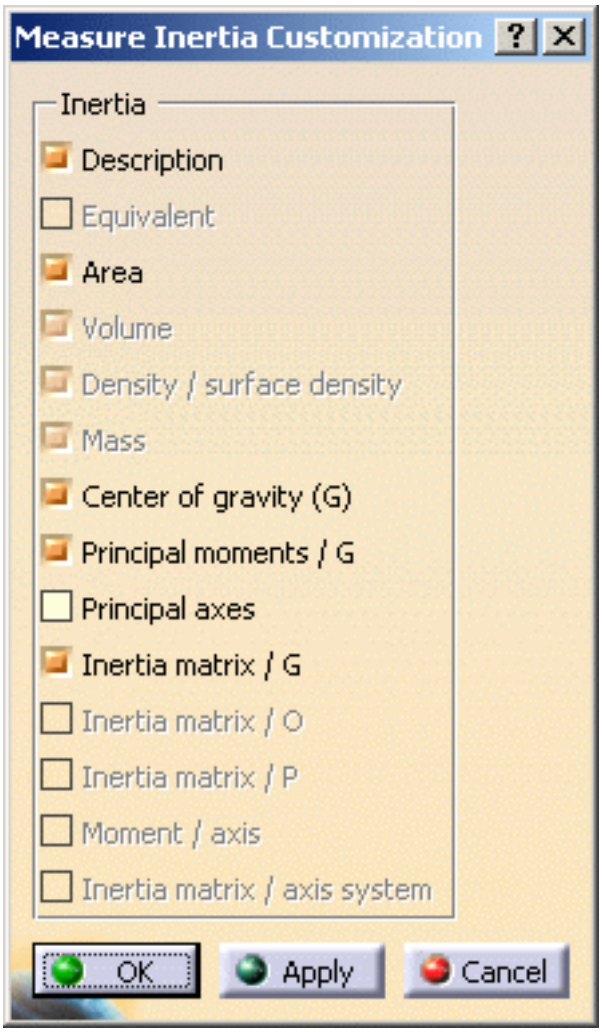
1. Click Customize... in the Measure Inertia dialog box.

   The Measure Inertia Customization dialog box opens.

   Note: The inertia properties checked here are also the properties exported to a text file.

2. Click the appropriate options:
   - Principal axes

3. Click **Apply** or **OK** in the Measure Inertia Customization dialog box when done.



Exporting Measure Inertia Results

This task shows you how to export both 3D and 2D inertia results to a text file.

Insert the Body1.cgr and the Body2.cgr documents from the common functionalities samples folder.

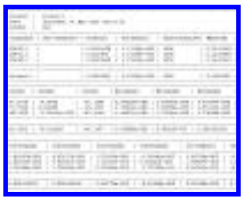
1. Select the root product and click the Measure Inertia icon.

   The dialog box expands to display the results for the selected item.
2. Click Export to write the results to a text (*.txt) file.

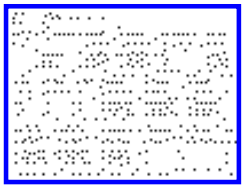
   Important: Results shown in the Measure Inertia dialog box only are exported. Exported results are given in current units.

3. Identify the file name and location in the Export Results dialog box that appears, then click **Save**.

Note: The examples given below concern 3D inertia results.



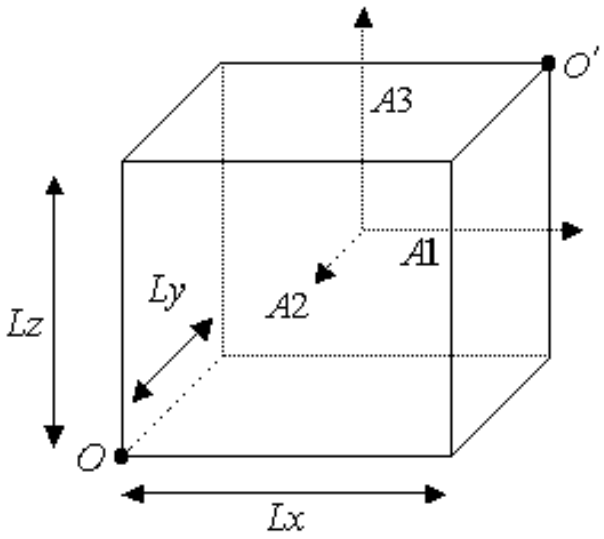
Note: If an assembly comprises sub-products or a part comprises part bodies, individual results for all sub-products or part bodies are also exported and written to the text file.



If the principal axes A are exported, bounding box values are also exported.

BBOx[mm]	BBOy[mm]	BBOz[mm]
0.143784	-0.0144969	-0.123277
0.16293	-0.0170551	-0.0783764
0.161982	-0.00353007	-0.0793372
0.169575	-0.0167384	-0.107493

BBLx[mm]	BBLy[mm]	BBLz[mm]
0.130313	0.0383596	0.0289959
0.166176	0.089425	0.0207404
0.166088	0.0894922	0.0206928
0.188821	0.0908866	0.0337512



$$O' = O + \vec{A1}.Lx + \vec{A2}.Ly + \vec{A3}.Lz$$

where BBOx,y,z defines the origin and BBLx,y,z the length

along the corresponding axis.

Note: When importing the text file into an Excel spreadsheet, do not forget to identify the pipe character (|) used as separator in the Text Import Wizard dialog box.

Notations Used for Inertia Matrices

This section will help you read the information given in the Measure Inertia dialog box for Inertia Matrix / G, Inertia Matrix / O, Inertia Matrix / P and Inertia Matrix / Axis System A.

Moments and Products of 3D Inertia

Iox	Moment of inertia of the object about the ox axis:	$Iox = \int_M (y^2 + z^2) dM$
Ioy	Moment of inertia of the object about the oy axis:	$Ioy = \int_M (x^2 + z^2) dM$
Ioz	Moment of inertia of the object about the oz axis:	$Ioz = \int_M (x^2 + y^2) dM$
Pxy	Product of inertia of the object about axes ox and oy:	$Pxy = \int_M (x.y) dM$
Pxz	Product of inertia of the object about axes ox and oz:	$Pxz = \int_M (x.z) dM$
Pyz	Product of inertia of the object about axes oy and oz:	$Pyz = \int_M (y.z) dM$

(where M is the mass of the object; units: $kg.m^2$)

Moments and Products of 2D Inertia

Iox	Moment of inertia of the surface about the ox axis:	$IoX = \int_A (y^2) dA$
Ioy	Moment of inertia of the surface about the oy axis:	$IoY = \int_A (x^2) dA$
Pxy	Product of inertia of the surface about axes ox and oy:	$Pxy = \int_A (x.y) dA$

(where A is the surface; units: m^4)

Matrix of Inertia

3D Inertia: 2D Inertia:

$$I = \begin{bmatrix} Iox & -Pxy & -Pxz \\ -Pxy & Ioy & -Pyz \\ -Pxz & -Pyz & Ioz \end{bmatrix} \quad I = \begin{bmatrix} IoX & -Pxy \\ -Pxy & IoY \end{bmatrix}$$

where I is the matrix of inertia of the object with respect to orthonormal basis Oxyz

Expression in Any Axis System:

I is the matrix of inertia with respect to orthonormal basis Oxyz.

Huygen's theorem is used to transform the matrix of inertia: Oxyz → Pxyz (parallel axis theorem).

Let I' be the matrix of inertia with respect to orthonormal basis Pxyz

where $V = \overrightarrow{PO}$

$$I' = I + m.\begin{bmatrix} V_y^2 + V_z^2 & -V_x.V_y & -V_x.V_z \\ -V_x.V_y & V_x^2 + V_z^2 & -V_y.V_z \\ -V_x.V_z & -V_y.V_z & V_x^2 + V_y^2 \end{bmatrix}$$

M = {u,v,w}: transformation matrix from basis (Pxyz) to basis (Puvw)
TM is the transposed matrix of matrix M.

J is the matrix of inertia with respect to an orthonormal basis Puvw:

J = TM.I'.M

Additional Notation used in Measure Inertia command

Ixy = (-Pxy)

Ixz = (-Pxz)

Iyz = (-Pyz)

Note: Since entries for the opposite of the product are symmetrical, they are given only once in the dialog box.

IoxG Moment of inertia of the object about the ox axis with respect to the system Gxyz, where G is the center of gravity.

IoxO Moment of inertia of the object about the ox axis with respect to the system Oxyz, where O is the origin of the document.

IoxP Moment of inertia of the object about the ox axis with respect to the system Pxyz, where P is a selected point.

IoxA Moment of inertia of the object about the ox axis with respect to the system Axyz, where A is a selected axis system.

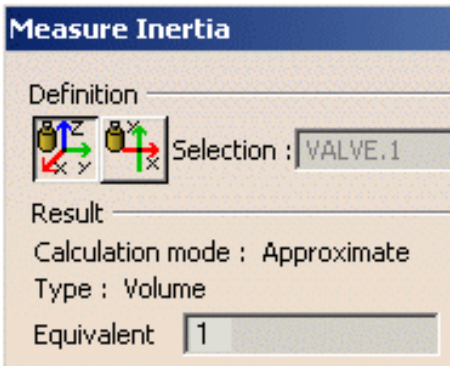
etc.

Inertia Equivalents

If your document contains inertia equivalents set using Knowledgeware capabilities, then the Inertia command will not calculate the inertia properties of the selected geometry but return the equivalent values.

The Equivalent box of the Measure Inertia dialog box indicates whether or not equivalents have been used:

- 0: the measure is made on the selection, geometry or assembly
- 1 or more: One or more inertia equivalents are taken into account.



To display inertia equivalents in the Measure Inertia dialog box:

1. Click Customize... in the Measure Inertia dialog box.
   The Measure Inertia Customization dialog box appears.
2. Check Equivalent in the Measure Inertia Customization dialog box.
3. Click **Apply**.

Equivalents are user parameters set using the Knowledgeware formula command f(x) under parts or products and imported from text (*txt) or Excel (*xls) files. Sets of equivalent parameters must be valid to be taken into account. To be valid, all the properties shown in the example below must be listed.

An example of a text file follows. In text files, the name of the property and the value are separated by a tab stop.

```
Equivalent_IsSurface   false
Equivalent_IsVolume    true
Equivalent_Area        6m2
Equivalent_Volume      1m3
Equivalent_Mass        1000kg
Equivalent_COGx        75mm
Equivalent_COGy        -10mm
Equivalent_COGz        -25mm

Equivalent_MatGxx      50000gxmm2
Equivalent_MatGyy      50000gxmm2
Equivalent_MatGzz      50000gxmm2
Equivalent_MatGxy      0gxmm2
Equivalent_MatGxz      0gxmm2
Equivalent_MatGyz      0gxmm2
```

In Excel files, simply list property names and values in two separate columns.

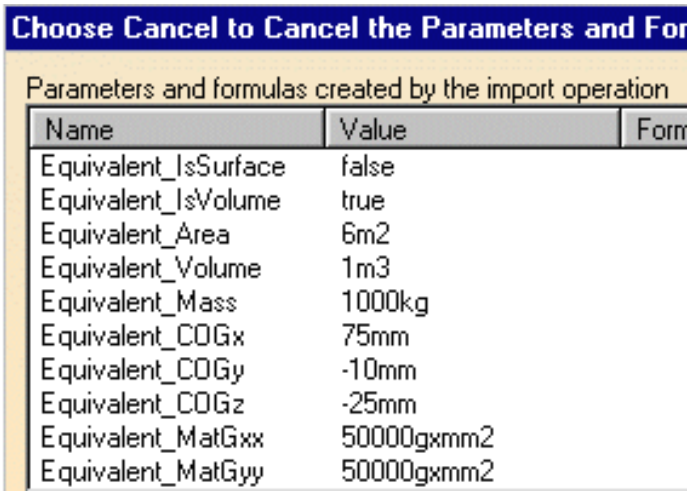
Importing Inertia Equivalents

1. Select the product to which you want to associate inertia equivalents.
2. Click the formula f(x) icon.

3. Click Import... in the Formulas dialog box.

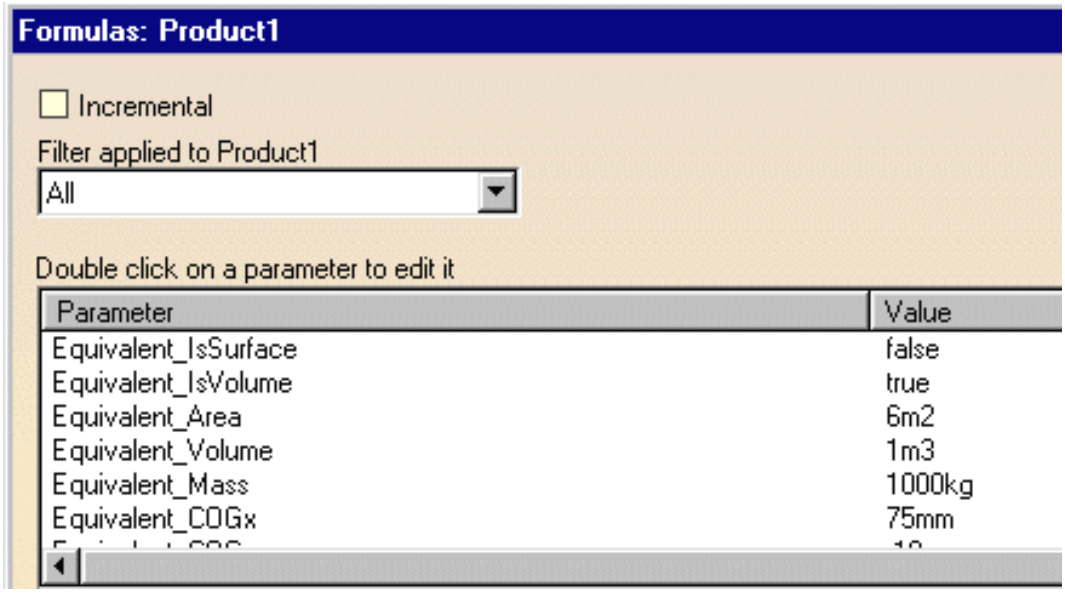
4. Select the text or Excel file containing the inertia equivalents in the file selection dialog box, then click Open.

   Parameters to be imported are listed.



5. Click **OK** to import all the parameters listed into the document.

   Imported parameters are now displayed in the Formulas dialog box.



6. Click **OK** in the Formulas dialog box.

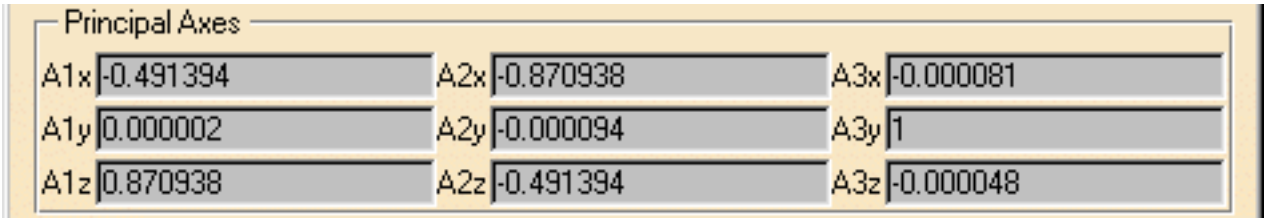
You are now ready to run your inertia calculation.

- Having imported inertia equivalents, you no longer need the representations of the product or sub-products and you can de-activate them (**Edit ->Representations**). De-activated representations are unloaded. This frees the geometry area and improves system response time.
- To display parameters in the specification tree, select the Parameters checkbox below Display in Specification Tree in the Display tab of the Options dialog box (**Tools-> Options-> Infrastructure-> Part Infrastructure**).

Measuring the Principal Axes A about which Inertia is Calculated

1. In the Measure Inertia Customization dialog box, click Principal axes.
2. Click **Apply**.

   The Inertia / G tab in the Measure Inertia dialog box becomes available.
3. Click the Inertia / G tab to display the principal axes about which inertia is calculated.



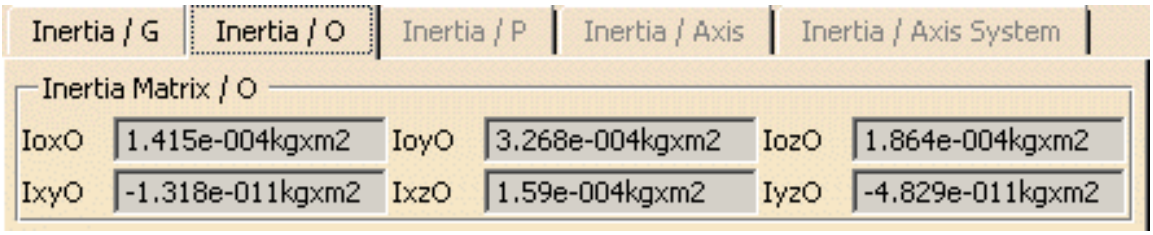
Note: If you checked the Keep Measure option, bounding box values are also displayed in the specification tree.

You can create the axis system corresponding to the principal axes.

Measuring the Inertia Matrix with respect to the Origin O of the Document

1. In the Measure Inertia Customization dialog box, click Inertia matrix / O.
2. Click **Apply**.

   The Inertia / O tab in the Measure Inertia dialog box becomes available. Entries for the inertia matrix appear in the specification tree.
3. Click the Inertia / O tab to display the inertia matrix of selected items with respect to the origin O of the document.



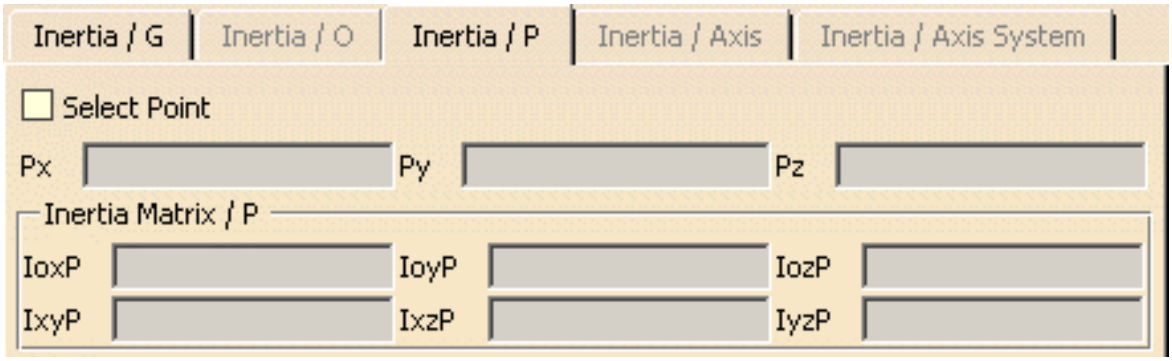
Measuring the Inertia Matrix with respect to a Point P

Insert or open the InertiaVolume.CATPart from the common functionalities sample folder **cfysm/samples**.

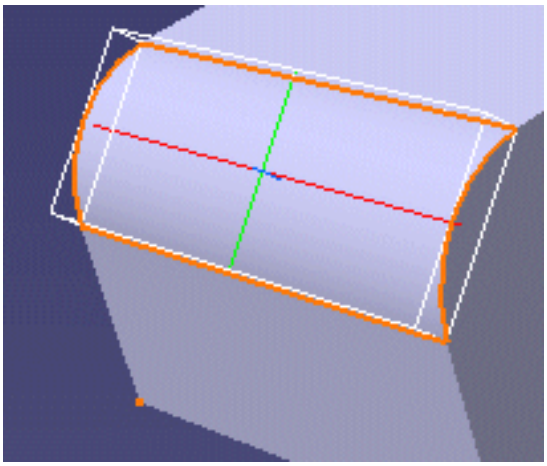
1. In the Measure Inertia Customization dialog box, click Inertia matrix / P.

   Note: Only points created in the Part Design workbench are valid.
2. Click **Apply**.

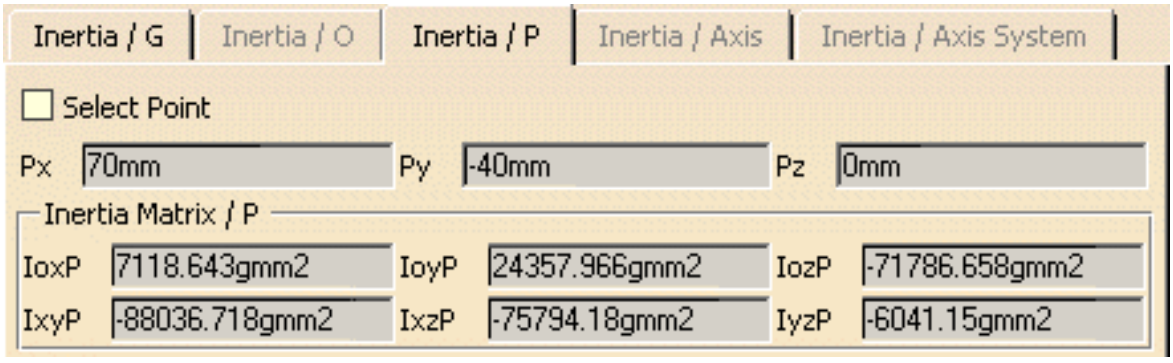
   The Inertia / P tab in the Measure Inertia dialog box becomes available.
3. Click the Inertia / P tab.



4. Select the Select point checkbox.
5. Select a point in the geometry area:



The coordinates of the point and the inertia matrix are given in the dialog box.



Selecting another item calculates the inertia matrix of the selected item with respect to the same point. To change point, click the Select point checkbox again, then select another point.

Measuring the Matrix of Inertia with respect to an Axis System

Insert or open the InertiaVolume.CATPart from the common functionalities sample folder **cfysm/samples**.

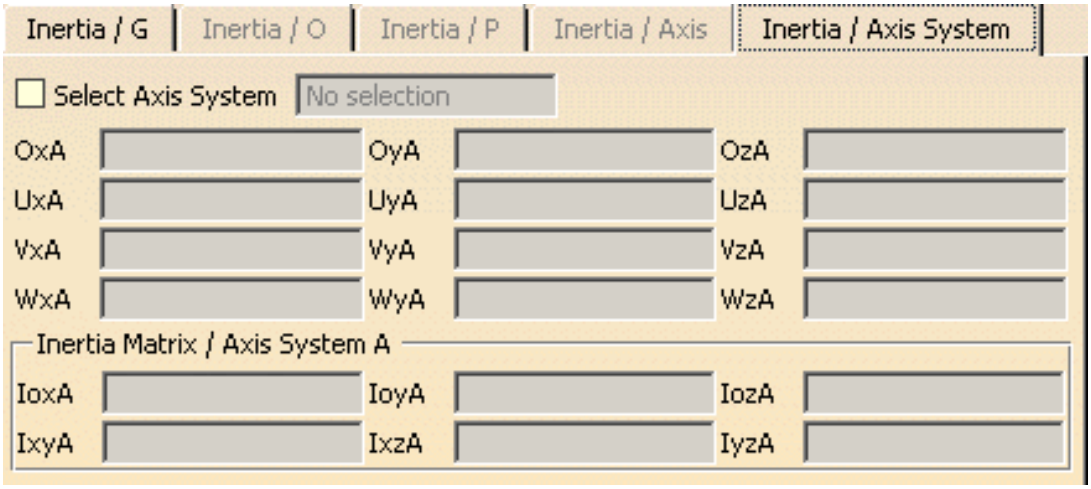
1. In the Measure Inertia Customization dialog box, click Inertia matrix / axis system.

   Note: Only axis systems created in the Part Design workbench (Axis System command) are valid.

2. Click **Apply**.

   The Inertia / Axis System tab in the Measure Inertia dialog box becomes available.

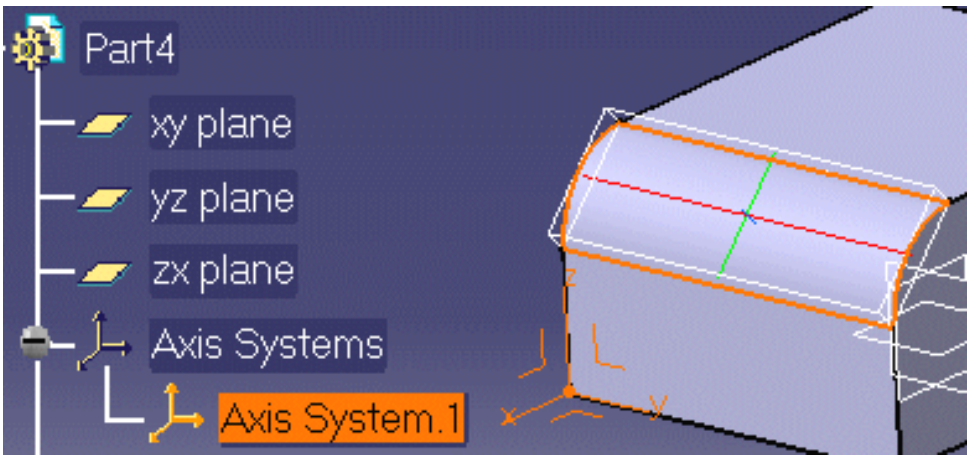
3. Click the Inertia / Axis System tab.
4. Select the Select axis system checkbox.

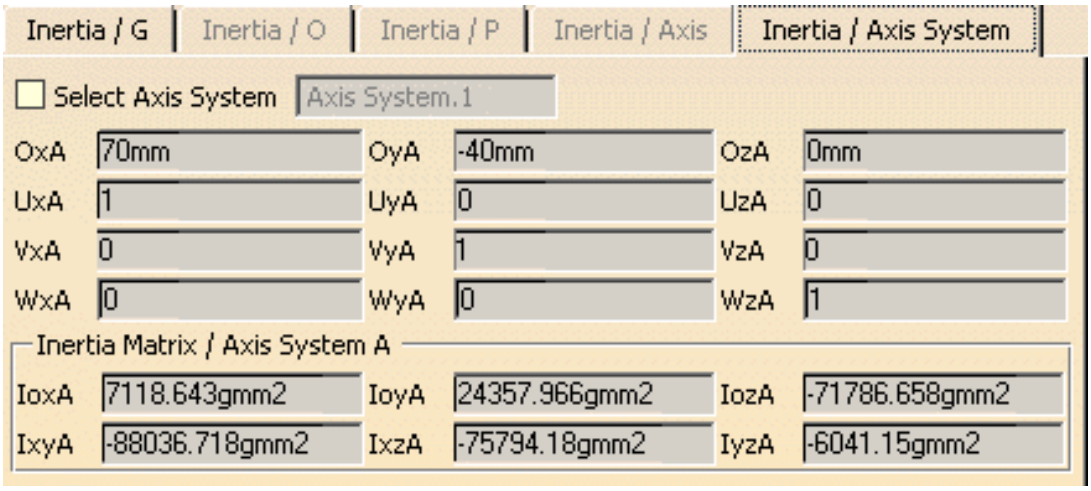


5. Select an axis system in the specification tree:

   Note: You must select the axis system in the specification tree.

   The name of the axis system as well as the origin O, (U, V, W) -vectors and the matrix of inertia with respect to the axis system are given in the dialog box. Entries for the matrix of inertia appear in the specification tree.





Selecting another item measure inertia properties of the selected item with respect to the same axis system. To change axis system, click the Select axis system checkbox again, then select another axis system.

If you checked the Keep Measure option in the Measure Inertia dialog box, your matrix of inertia measures are kept as features and, if made with respect to a V5 axis system, are associative.

Measuring the Moment of Inertia about an Axis

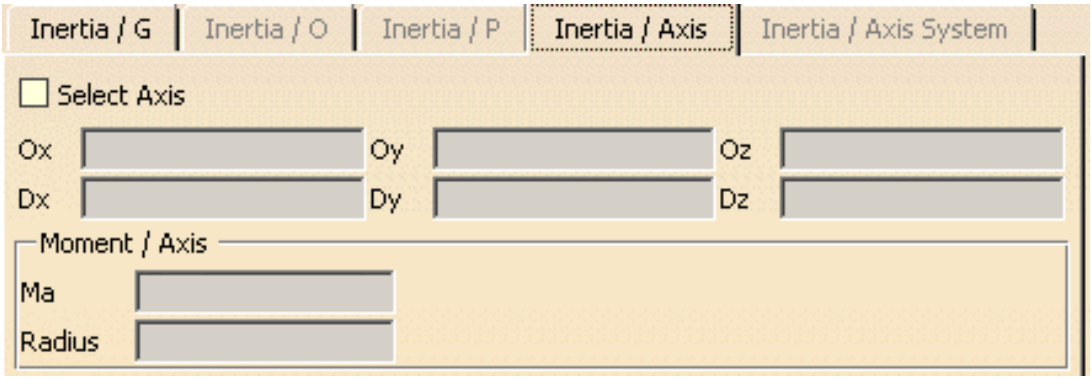
Insert or open the InertiaVolume.CATPart from the common functionalities sample folder **cfysm/samples**.

1. In the Measure Inertia Customization dialog box, click Moment / axis to measure inertia with respect to an axis.

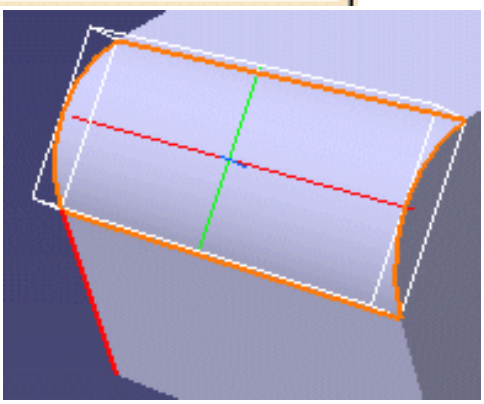
   Note: Only axes created in the Part Design workbench are valid.

2. Click **Apply**.

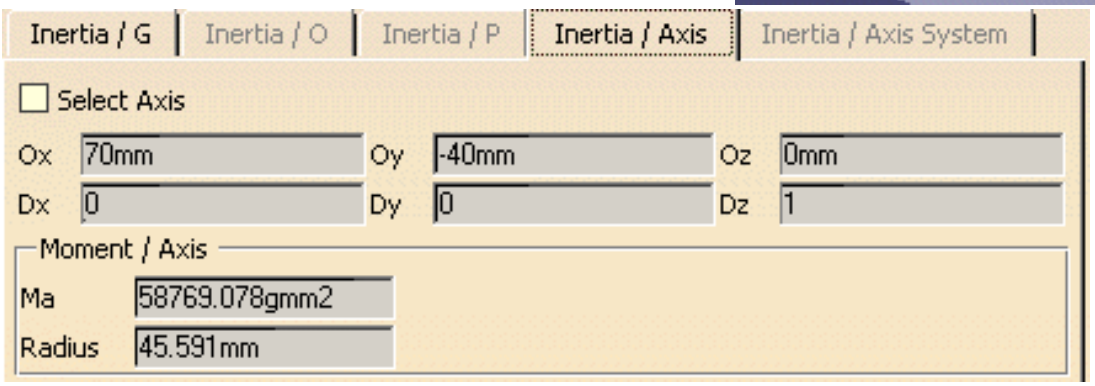
   The Inertia / Axis tab in the Measure Inertia dialog box becomes available.
3. Click the Inertia / Axis tab.
4. Select the Select axis checkbox.



5. Select an axis in the geometry area:



   The equation and direction vector of the axis as well as the moment of inertia Ma about the axis and the radius of gyration are given in the dialog box.

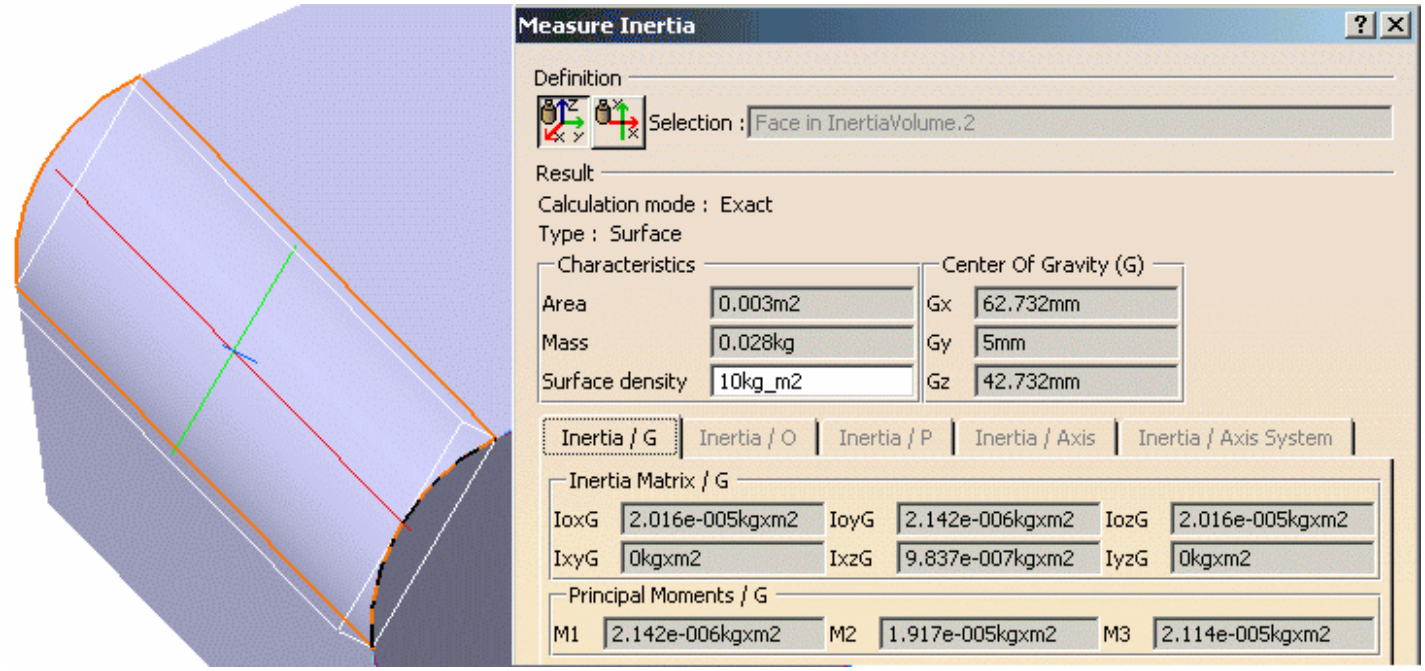


   Selecting another item measures the inertia of the selected item about the same axis. To change axis, click the Select axis checkbox again, then select another axis.

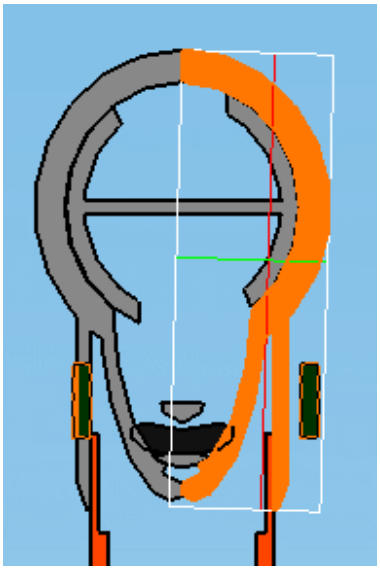
3D Inertia Properties of a Surface

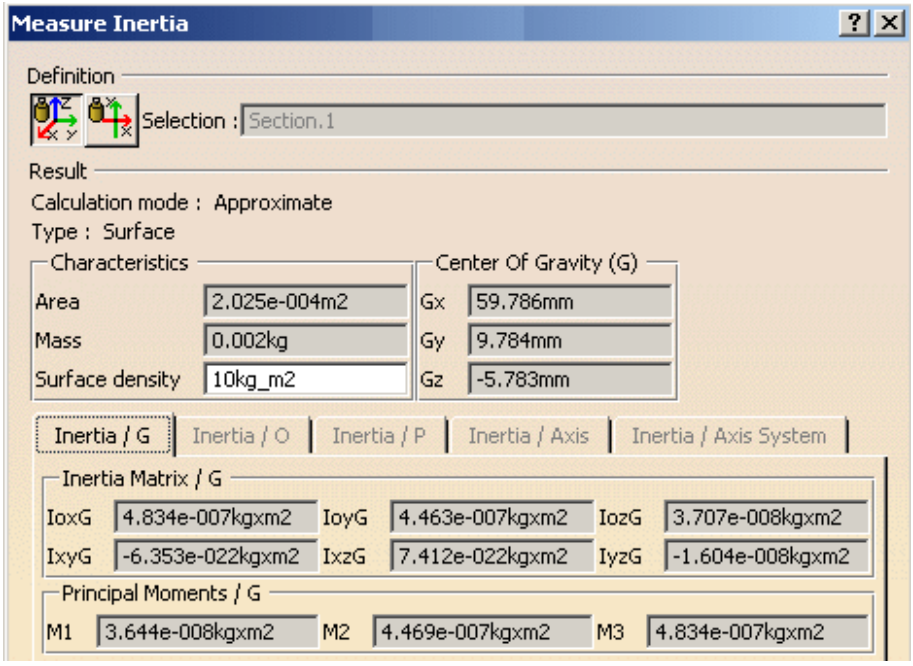
You can measure 3D inertia properties on exact and tessellated surfaces. Examples showing a surface and a DMU section are given below.

Insert or open the InertiaVolume.CATPart from the common functionalities sample folder **cfysm/samples**.



The DMU section is a tessellated surface.





Exact Measures on CGRs and in Visualization Mode

Measure Between and Measure Item commands permit exact measures on inserted CGR files (created from CATIA V4 models, CATParts) as well as on Visualization mode geometry. This lets you make measures in Visualization mode without having to load the part. For more information, see the table below: the red cells indicate where an exact measure is not possible.

Measure Item

	Point	Line	Arc except length	Curve	Plane except area, center of gravity	Cylinder/Cone except area, center of gravity	Sphere except area, center of gravity	Surface revolution	Surface	Volume	Assembly
Exact measure											

Measure Between

	Point	Line	Center of arc	Curve	Infinite plane	Axis cylinder/ Cone	Center of sphere	Surface (plane, cylinder, sphere, any)
Point								
Line								
Center of arc								
Curve								
Infinite plane								
Axis cylinder								
Center of sphere								
Surface								

Don't forget to set the selection mode correctly. To make a measure with respect to an infinite plane means you must set the selection mode to Any geometry, infinite.

Creating Geometry from Measure Results

This task explains how to create geometry from the results of measures made using:

- Measure Between
- Measure Item
- Measure Inertia

All geometry is created under the Open_body of a new or existing part.

The part containing the measure geometry must remain in the same position with respect to the document root and must not be re-ordered.

Associativity

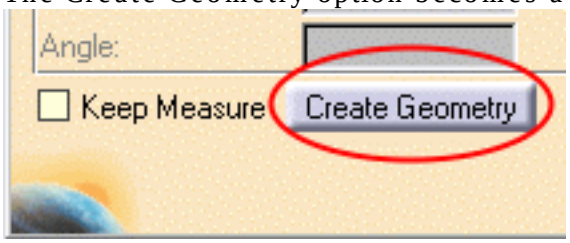
In a product, the geometry you create can be either associative or non associative with the measure. If you want to create associative geometry, check the Keep link with selected object option in **Tools -> Options -> Infrastructure -> Part Infrastructure, General tab**

In a part, the geometry you create is associative.

Note: In both cases, associative geometry can only be created if your measure is associative.

1. Make your measure using the appropriate measure command.

   The Create Geometry option becomes available in the measure dialog box.

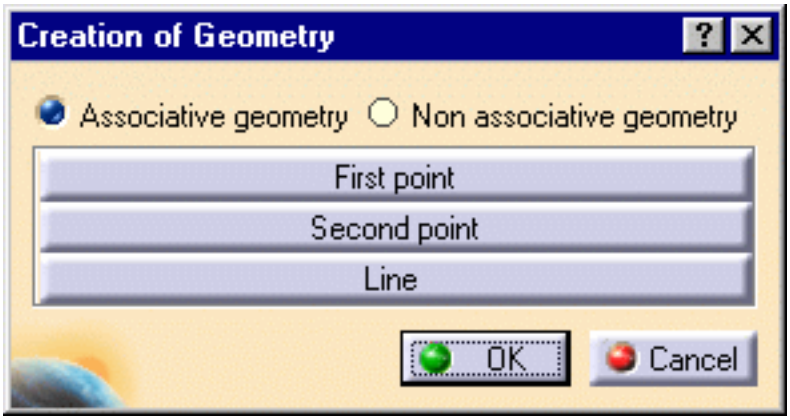


2. Click Create Geometry and follow instructions depending on whether you are in a product or part:

If you are in a...	Then...

Product	the Geometry Creation dialog box appears letting you choose where you want the geometry created.  Select one of the two options below, then click **OK**: ○ A new CATPart In which case a CATPart is inserted under the active product. ○ An existing CATPart In which case, click the option and select the CATPart in the specification tree.
Part	the geometry is automatically created in an existing open_body or a new open_body if one does not exist.

The Creation of Geometry dialog box appears.
The example below shows the dialog box for a measure made using the Measure Between command.



3. Are you in a product?

   ○ If yes, select the appropriate associativity option depending on whether or not you want to link the geometry to the measure.

   ○ If no, read on.

Notes:

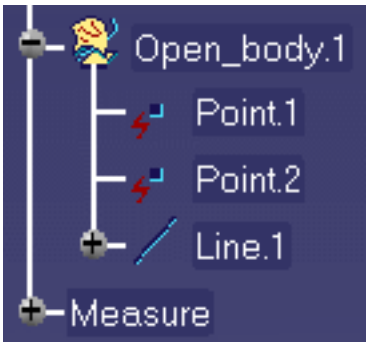
- In a part, the geometry you create is associative.
- In both a product and a part, associative geometry can only be created if your measure is associative.
- If Associative geometry is selected, the Keep Measure option is checked to ensure that the geometry created is based on measure results.

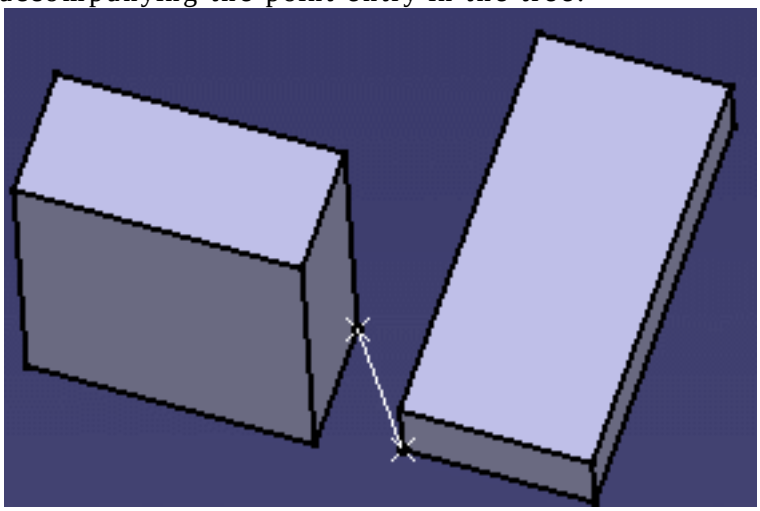
4. Select options in the Creation of Geometry dialog box to create geometry desired.

If you made your measure using...	Then you can create...
Measure Between	○ First point ○ Second point These are the two points between which the minimum distance is measured ○ Line: the line representing the minimum distance result.
Measure Item	○ Center of gravity
Measure Inertia	○ Center of gravity ○ Axis system (for the principal axes).

5. Click **OK** in the Creation of Geometry dialog box when done.

Geometry is created in the geometry area and is added to the specification tree under the Open_body of a new or an existing part.
In the Measure Between example below, created geometry is non-associative. This is identified by the red symbol accompanying the point entry in the tree.





6. Click **OK** in the measure dialog box when done.

Editing Measures

In addition to adjusting the presentation of the measure, you can also edit the measure itself and:

- Change one of the selections on which it was based. This is particularly useful in design mode where you no longer have to redo your measure.
  You can also change selections that no longer exist because they were deleted.
- In a part, replace selections using the replace mechanism.

Changing Selections

You can change selections on which your measure is based in Measure Between and Measure Item commands.

1. Double-click the measure in the specification tree or geometry area.

2. Make new selections.

Notes:
In Measure Between, you can change selection modes when making new selections.
For invalid measures where one selection has been deleted, you only have to replace the deleted selection.
For all other measures, repeat all selections.

In Measure Item, you cannot change the selection 1 mode. If you selected a curve, you must make a selection of the same type, i.e. another curve.

3. Click **OK** when done.

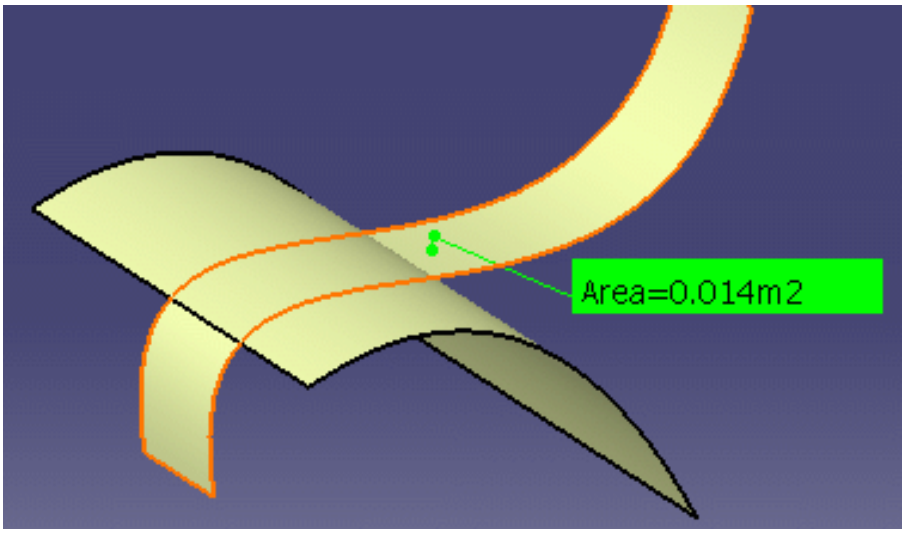
Replacing Selections in a Part

In a part, you can change selections using the Replace mechanism in Measure Between, Measure Item and Measure Inertia commands.

Note: In a product, measures are not integrated into the Replace mechanism. When you replace a product on which a measure is made, the measure is no longer valid and links to the geometry are deleted.
Special case: If your measure was made on a product in the specification tree, the measure is identified as not up-to-date after you replace the product.

1. Make your measure using the appropriate measure command.

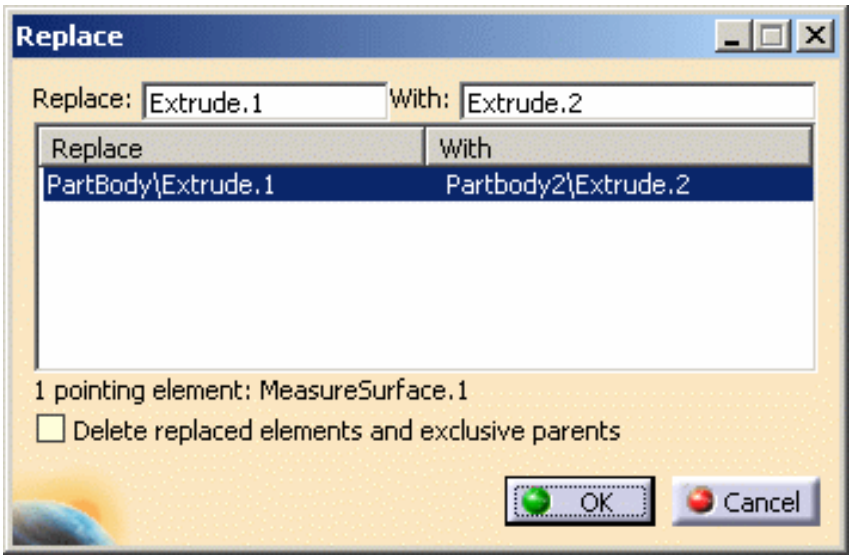
Important: You must select items or entities in the specification tree.



2. Right-click the selection you want to replace and select Replace... from the contextual menu.

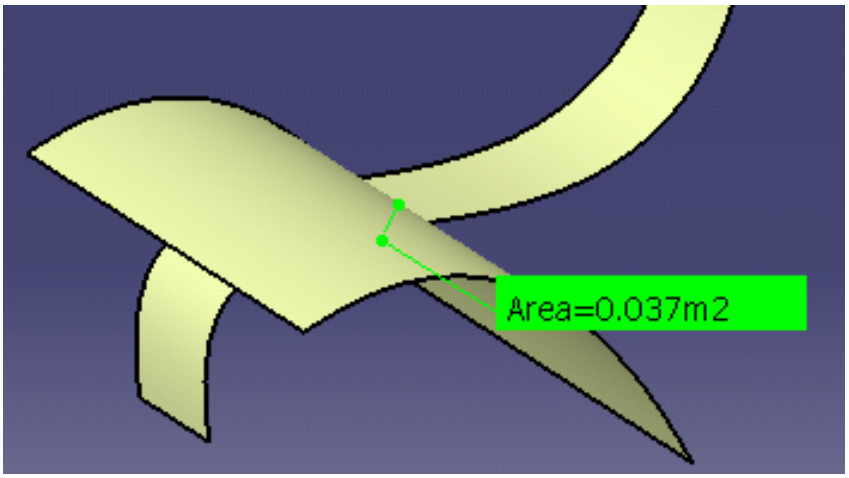
   Note: Replacing a selection impacts all items or entities linked to the selection.

   The Replace dialog box appears.



3. Make a new selection.
4. Click **OK** in the Replace dialog box.

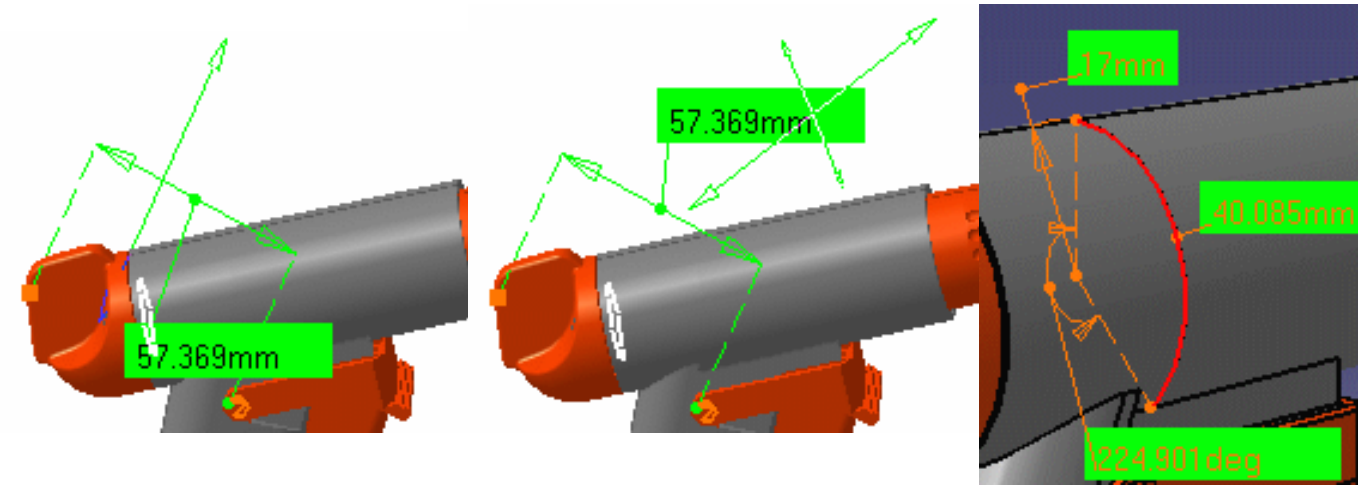
   The measure is identified as not up-to-date.
5. Right-click the measure in the specification tree and select **Local update** from the contextual menu to update your measure.



Editing the Presentation of your Measure

You can adjust the presentation of Measure Between and Measure Item measures.

You can move the lines and text of the measure.



The Properties command (Graphics tab) lets you change the fill color and transparency as well as the color, linetype and thickness of measure lines.

Note: You cannot vary transparency properties, the current object is either the selected color or transparent.

Updating Measures

Design Mode measures and, in Visualization Mode, measures on products selected in the specification tree only are associative.

If you modify a part in a part document, or modify, move, delete, etc. a part in a product document and the measure is impacted, it will be identified as not up-to-date in the specification tree. You can then update it locally have it updated automatically.

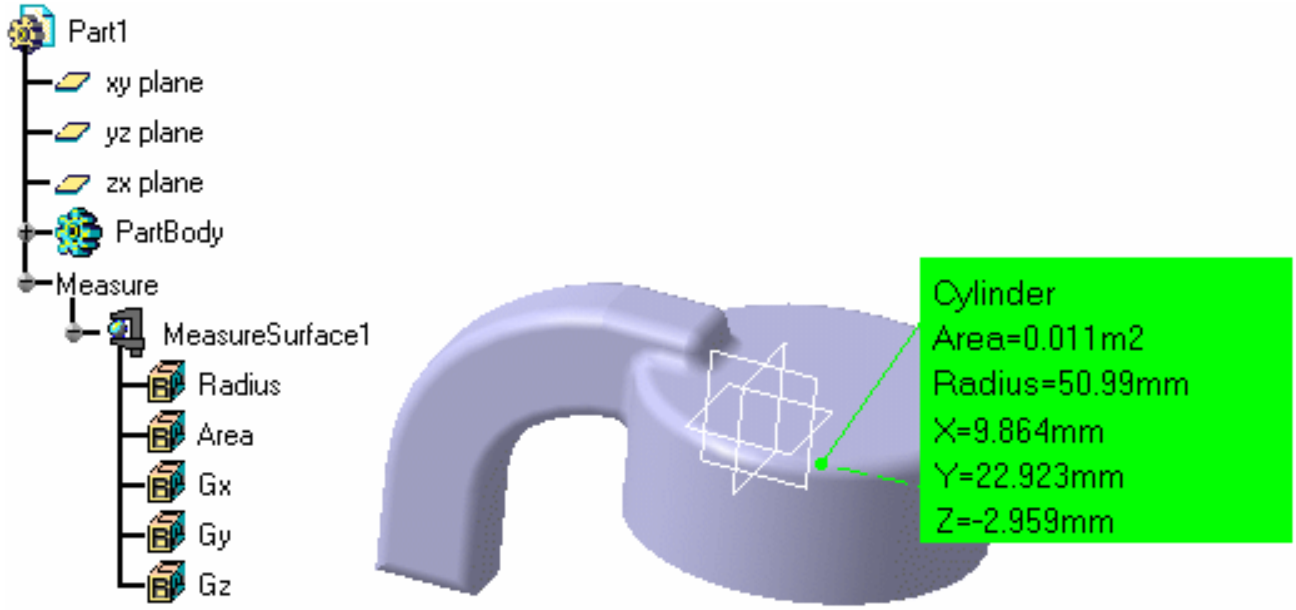
This task explains how to update a Measure Item measure following a change in part radius.

Open the AssociativeMeasures.CATPart from the **cfysm/samples** folder.

Note: In ENOVIA DMU, insert the CATPart.

1. Measure the properties of a part.

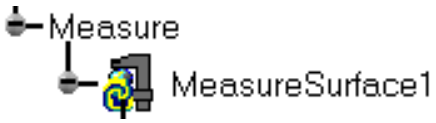
Important: Do not forget to check the Keep Measure option in the Measure dialog box to keep measures as features in the specification tree. The Keep Measure option is available in the Measure Between, Measure Item and Measure Inertia commands.



2. Modify the part, for example decrease the value of the radius.

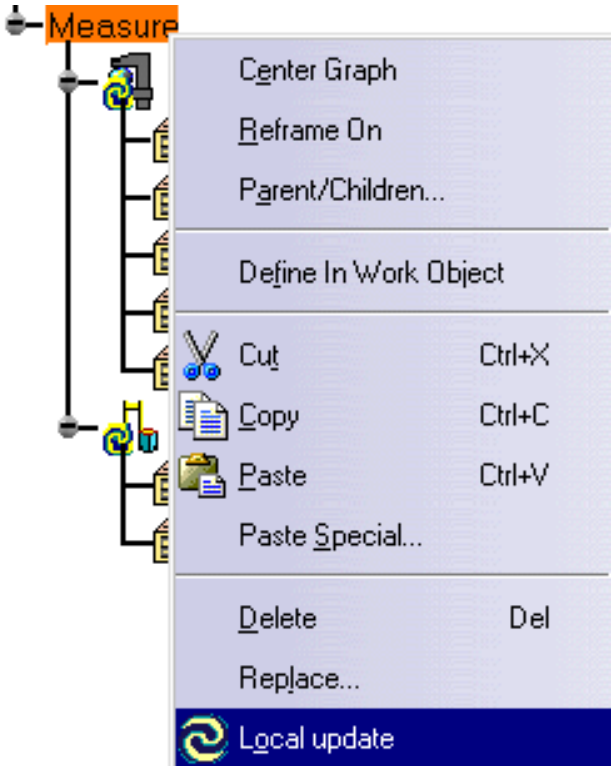
Note: You cannot modify parts in ENOVIA DMU, move the part instead.

The measure icon in the specification tree changes to indicate that the measure is not up-to-date and requires updating.



3. Update the measure.

In a part document, you can update either an individual measure or the Measure entry. To do so, right-click in the specification tree and select **Local update** from the contextual menu.



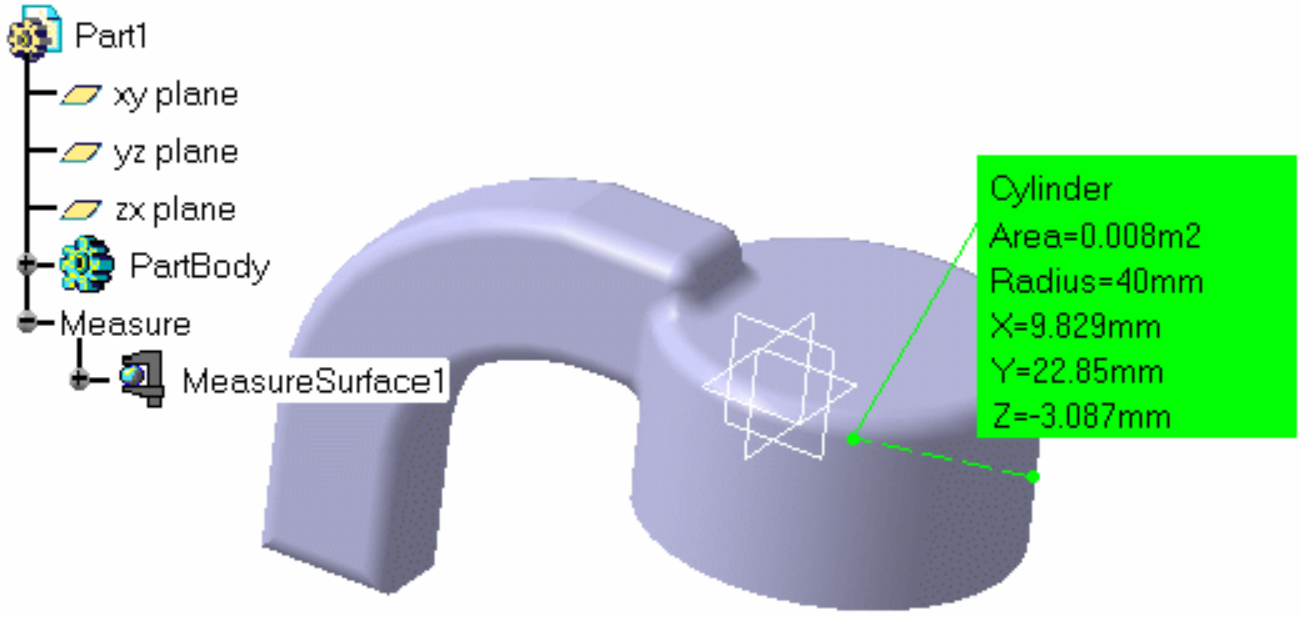
Selecting the Measure entry in the specification tree lets you update all measures needing updating in one go.

In a product document, right-click the measure and select **Measure object -> Measure Update** from the contextual menu.

Note: If the measure is considered up-to-date, a **Force Measure Update** entry appears instead in the contextual menu. The **Force Measure Update** command is also available in the DMU scene context.

In Measure Between and Measure Item commands, moving the cursor over the measure in the geometry area or the specification tree dynamically highlights all related items in both the geometry area and specification tree.

The measure is updated to reflect modifications to the part.



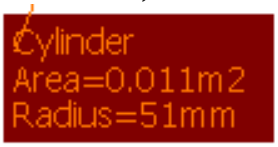
Automatic Update

To have your measures updated automatically in a part or a product document, check the appropriate option in **Tools -> Options -> General -> Parameters and Measure, Measure Tools tab**.

Notes:

- In a part document, you must also check the Automatic Update option in **Tools -> Options -> Infrastructure -> Part Infrastructure -> General.**
- In a product document, if you edit a part, any measures will be automatically updated when you activate the product.
- When you open a document, measures are not updated. For example, if you save a document in which measures are not up-to-date and then re-open the document, measures remain in the not up-to-date status even if you checked the automatic update option.

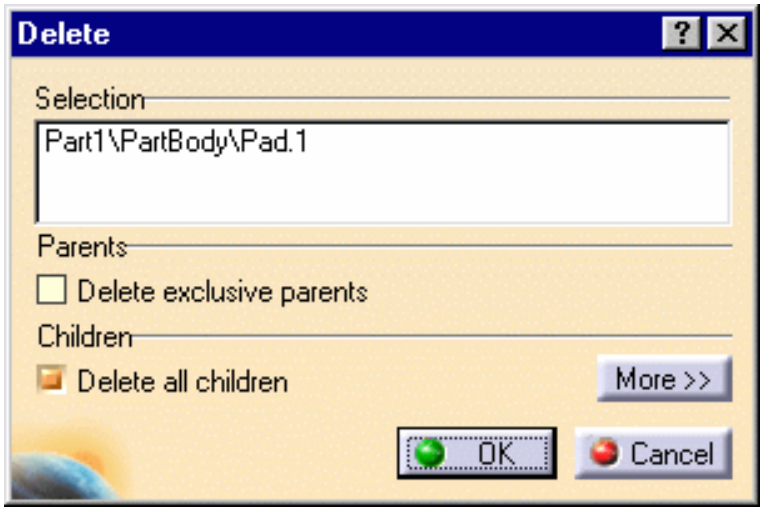
Invalid Measures

Measures that are no longer valid are identified in the specification tree by the measure icon plus an exclamation mark, for example  and, for Measure Between and Measure Item measures, the measure itself changes color. 

Measures are no longer valid if links are not resolved. This happens if:

- You delete an item on which a measure is made.

  In a part document, Measure Between, Measure Item and Measure Inertia measures are integrated into the parent-child mechanism. If you delete items on which measures are made, the Delete dialog box appears letting you delete measures at the same time. Associated measures are highlighted in the specification tree.



Similarly, if you delete measures, the Delete dialog box lets you delete the measured items provided they are exclusively used for the measures (Delete exclusive parents option).

Note: In a product document, measures are not integrated into the product Delete mechanism. When you delete a product on which a measure is made, the measure is no longer valid and links to the geometry are also deleted.

- You switch to visualization mode or open a document in visualization mode. For example, your measure is made on a face in design mode, you save the document and then re-open it in visualization mode.

Restrictions

- Measures made prior to Version 5 Release 6 are not associative and therefore cannot be updated. These measures are identified in the specification tree by the measure icon plus a lock, for example  .
- Visualization Mode measures and measures on cgr files made in the geometry area are not associative and therefore cannot be updated. These measures are identified as above:  .
  Only products selected in the specification tree in Visualization Mode can be updated.
- Measures made in Picking point, Picking axis, Intersection and Center of 3 points selection modes are not associative.
- Inertia measures made on a multiple selection of items are not associative.
- In the Drafting and Advanced Meshing Tools workbenches, measures are done on-the-fly and are not persistent. This means that they are not associative.

Using Measures in Knowledgeware

Measures can be used in formulas. A set of results (length, angle, etc.) is associated to each measure feature in the specification tree. Each piece of information can be used to create parameters in formulas or to create geometry. To read more, see specifying measures in formulas.

Measures can also be used to valuate parameters. When doing so, an associative link between the measure and parameter is created.

This task explains how to create a point on the measured center of gravity of a part.

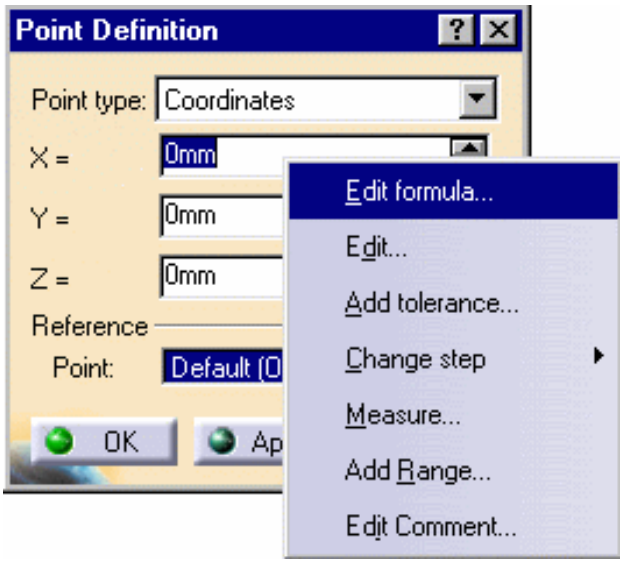
Have completed the task on associative measures.

In ENOVIA DMU, you need a Part Design license to complete this task.

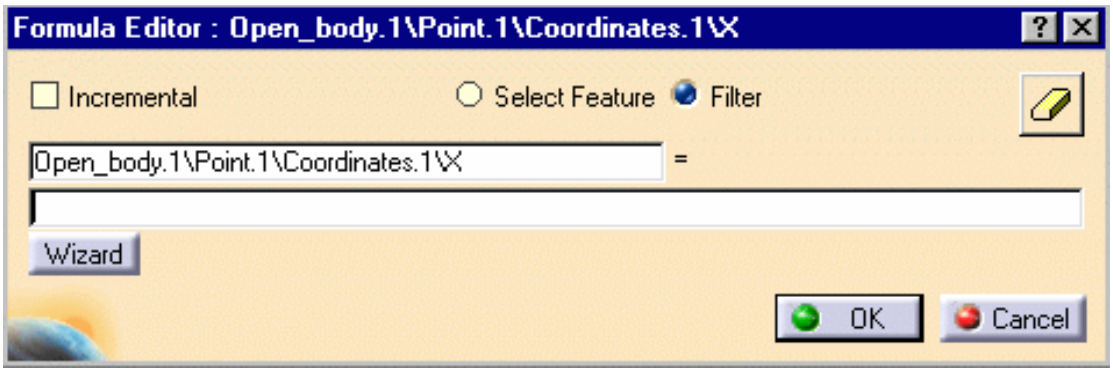
1. Create a point using the Point command.

The Point Definition dialog box appears for you to enter point coordinates.

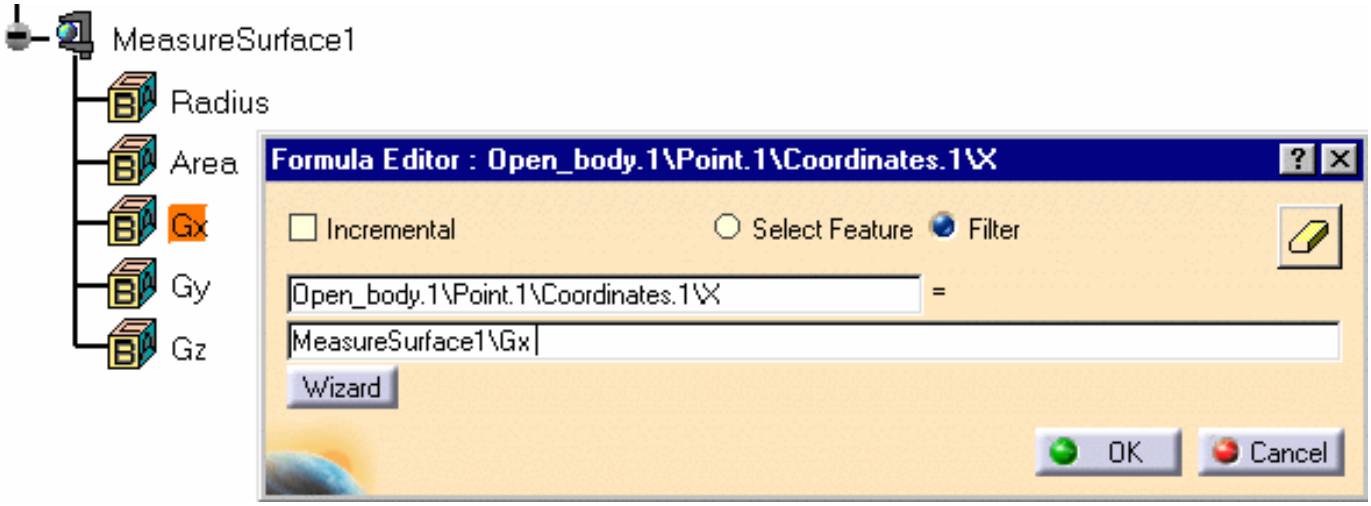
2. Right-click the X= field and select Edit formula... from the contextual menu.



The Formula Editor dialog box appears letting you define the x coordinate by a formula.

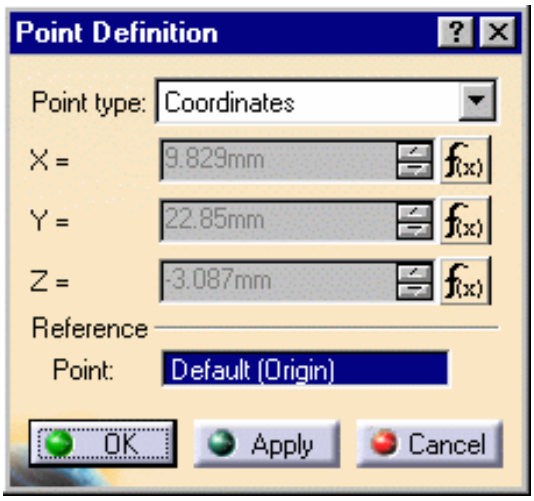


3. Select the measured X coordinate in the specification tree, then click OK.

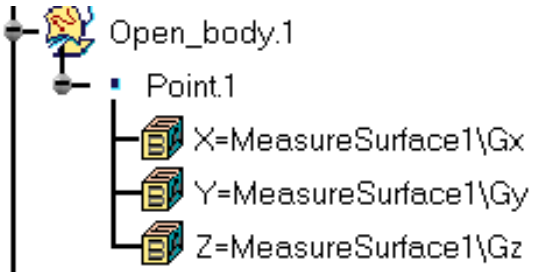


The Point Definition dialog box is updated.

4. Repeat for Y and Z coordinates.



5. Click **OK** in the Point Definition dialog box to create the point at the measured center of gravity of the part.



Create a line on the point using the Line command, then move the object using the 3D compass and see everything update automatically.

Measuring Distance, Angle and Radius on 2D Documents

This task explains how to measure distances, angles and radii on 2D documents of both vector and pixel type.

Note: In the No Show space, this command is not accessible.

No sample document is provided.

1. Select **File -> Open** and open a 2D document.

   The Digital Mock-Up 2D workshop is opened and displays the selected document.
   You can measure distance, angle and radius on documents in vector formats such as cgm, hpgl as well as in raster formats such as jpeg, bmp.

   For more information on:

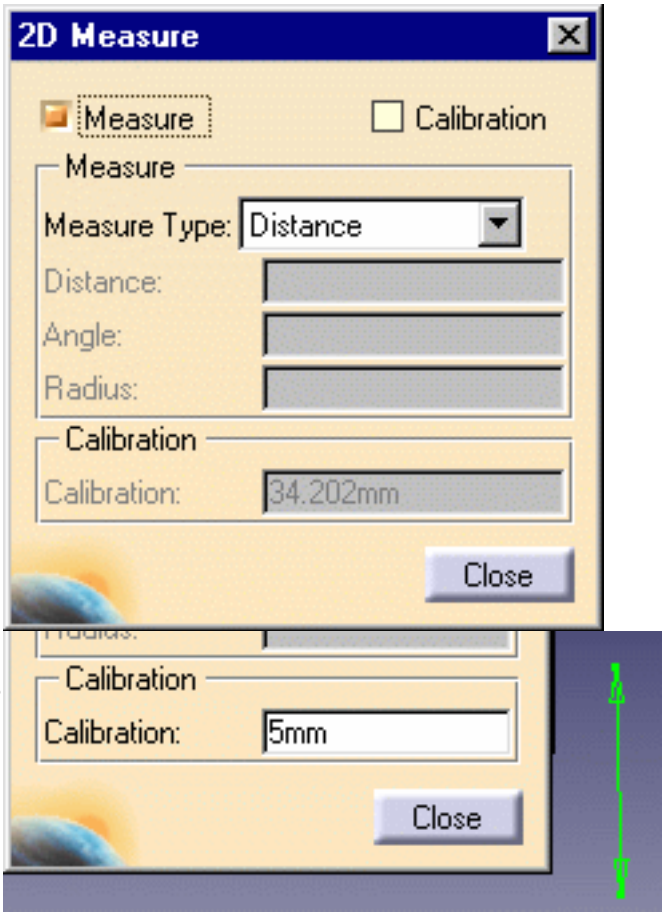
   - 2D documents you can open, see Inserting Components in the *DMU Navigator User's Guide*.
   - DMU 2D workshop, see the *DMU Navigator User's Guide*.

2. Click the 2D Measure icon in the DMU 2D Tools toolbar.
   The 2D Measure dialog box appears.

   An automatic calibration, based on the width of the drawing, is proposed for **vector type documents**. The dialog box opens directly in the Measure mode.

   Click Calibration to visualize the reference distance (green arrow) and adjust the calibration if necessary.

   All measures will be made with respect to this reference.

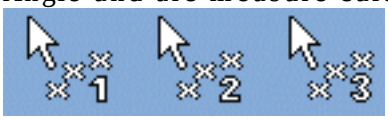
   For **pixel type documents**, calibrating is necessary to make measures and the dialog box opens in the Calibration mode.

   - To calibrate, click two points to define the reference, then enter a distance in the Calibration field.
   - Click the Measure check box to make your measure.

Note: The appearance of the cursor has changed to assist you. A number also helps you identify where you are in your measure or calibration.

Calibration and distance measure cursor:

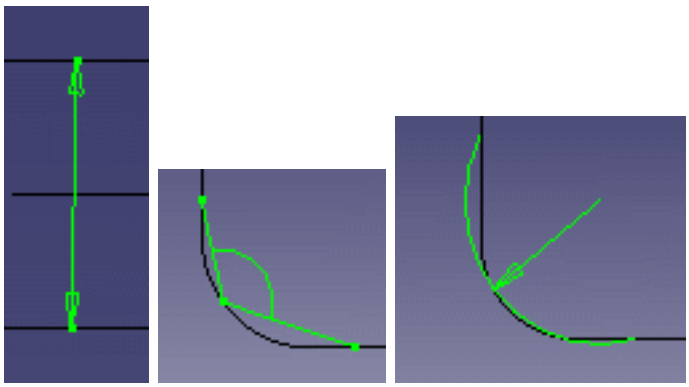
Angle and arc measure cursor:



3. Set the desired Measure type in the Measure type drop-down list box.

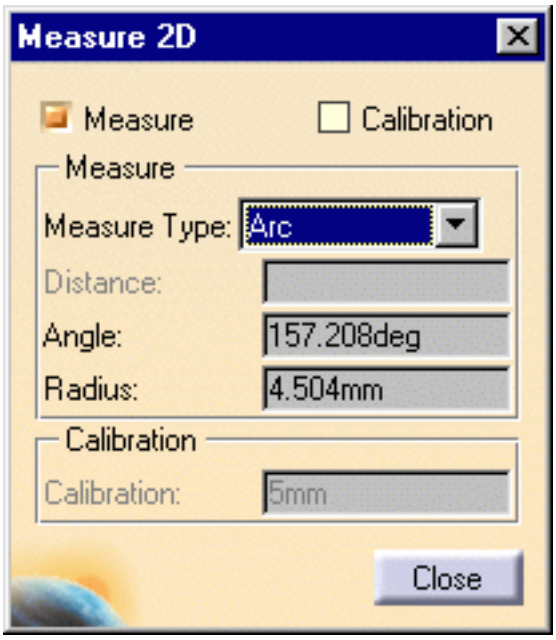
Defining Measure Types

- Distance: measures the distance between two points.
- Angle: measures the angle defined by three points.
- Arc: measures the angle and radius of an arc fitted through three points.



4. Click to define the points between which the measure is made.
The cursor snaps to vector elements in vector-type documents.

The dialog box is updated and gives the appropriate information depending on the type of measure made.



5. Click **Close** when done.

The calibration value and reference distance are stored in memory and are re-proposed if you enter the command again whilst in the same document.

Annotating

These tasks are documented in the *DMU Navigator User's Guide* where more information on annotating can be found.

Add 3D annotations: Click the 3D Annotation icon, then click where you want to place the text, enter the text in the Annotation Text dialog box and click **OK**.

Create annotated views: Click the Create an Annotated View icon, then annotate the active view using commands in the DMU 2D Marker toolbar.

Manage annotated views: Click the Manage Annotated Views icon, then double-click the desired 2D view in the dialog box to recover it.

Defining Groups

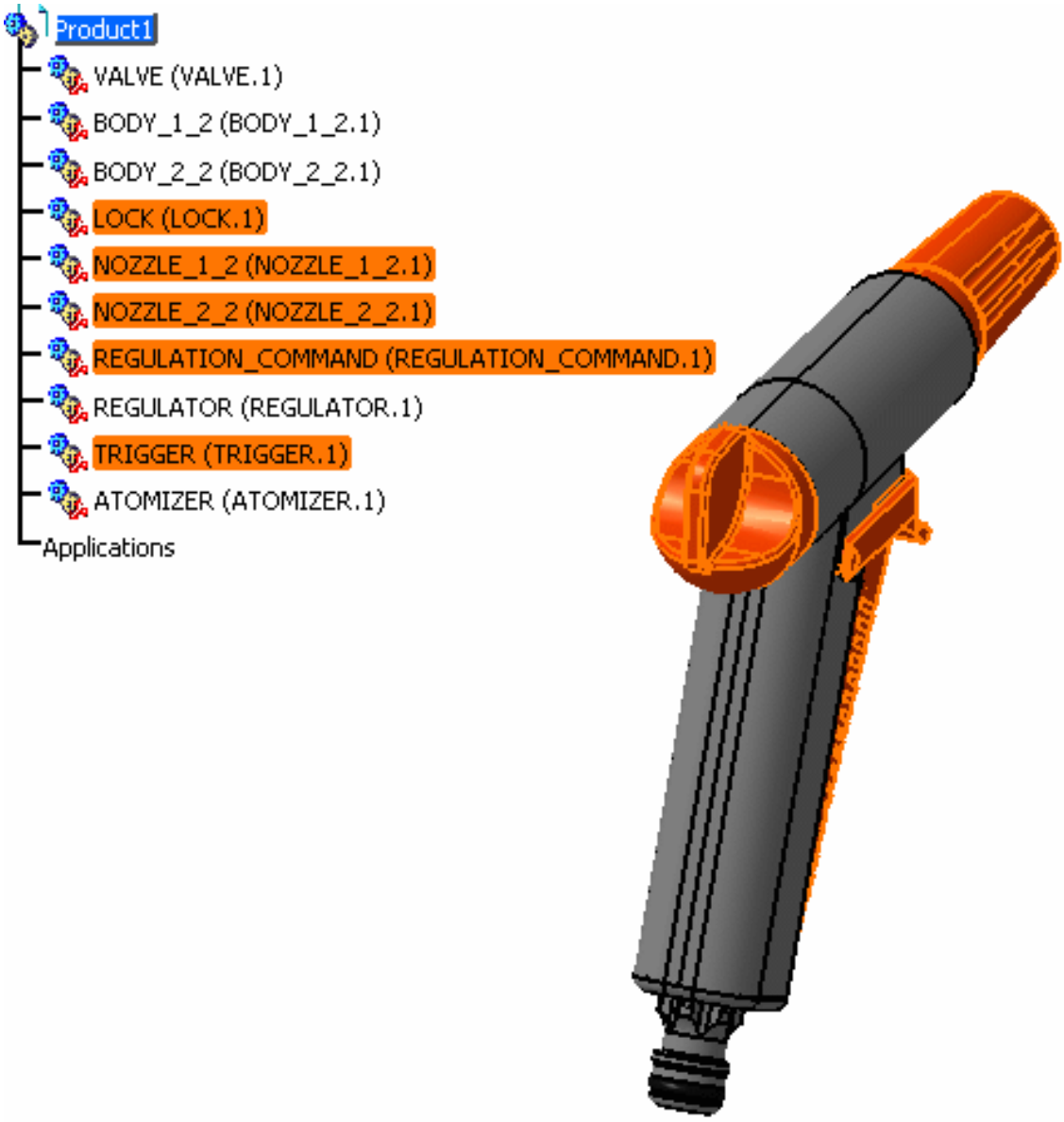
This task explains how to define groups of products. A group is a set of products explicitly defined by selecting products individually. Groups are persistent and can be stored in the document.

- Cross-highlighting is now established between the Preview display and the Edit Group dialog box containing the list of group content.
- In the Edit Group dialog box and in the Preview window, you can now remove components.
- In the Edit Group dialog box, you can now multi-select components.
- It is now possible to show objects in the Preview window that are in Hide visibility status in the Main window
- It is now possible to define groups of groups.
- In order to improve performance, it is possible to replace a set of brother products by their common father if all the brothers belong to the group
- Replace terminal nodes by children

Insert the following cgr files from the cfysm samples folder:

ATOMIZER.cgr
BODY1.cgr
BODY2.cgr
LOCK.cgr
NOZZLE1.cgr
NOZZLE2.cgr
REGULATOR.cgr
REGULATION_COMMAND.cgr
TRIGGER.cgr
VALVE.cgr

1. In the specification tree or in the geometry area, select the products you wish to constitute the initial group content (you can use ctrl-click to multi-select products).

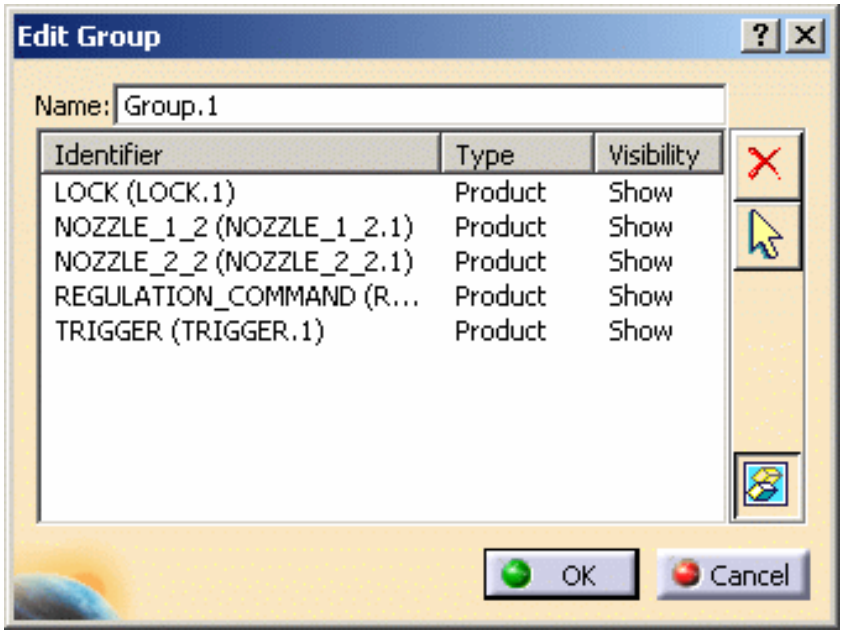


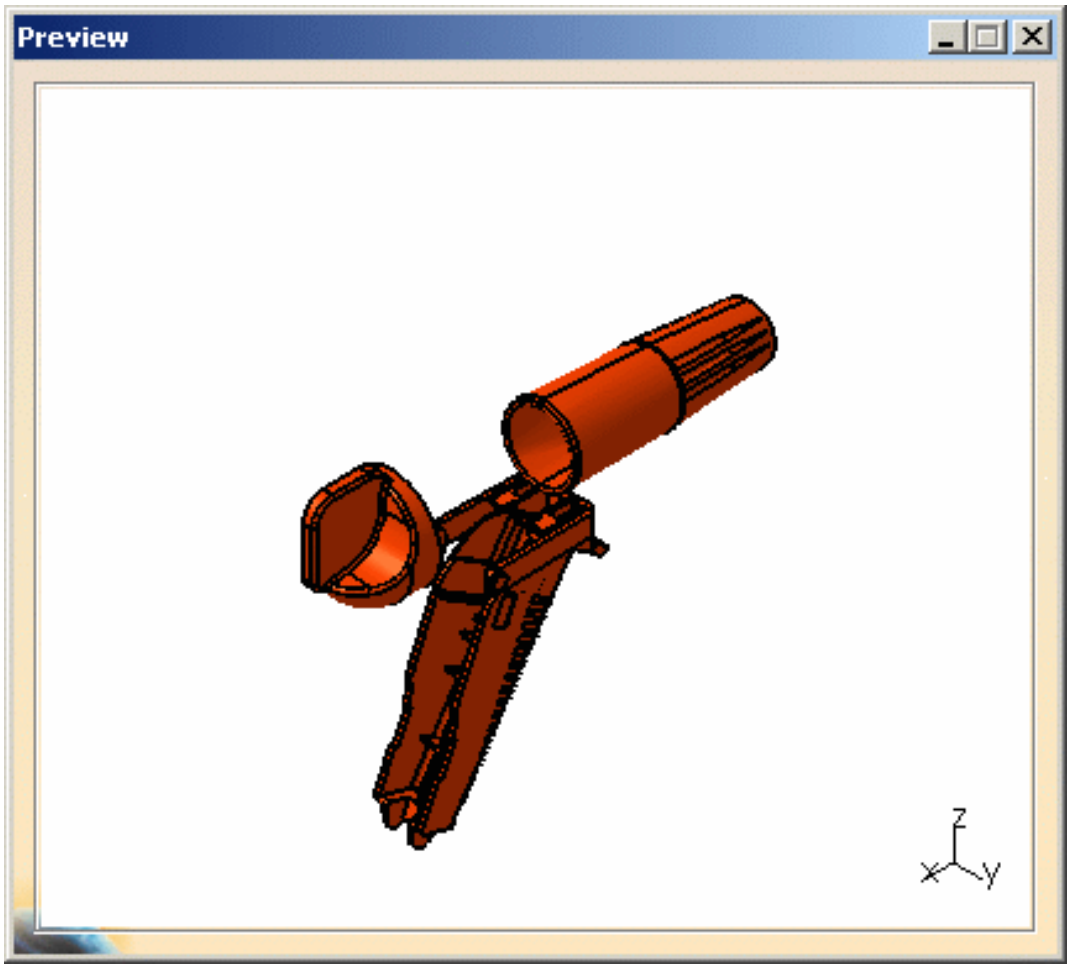
Creating a Group

2. To create a Group, in the DMU Review Creation toolbar, click the **Group** icon or, in the menu bar, select **Insert -> Group** .

   The **Edit Group** dialog box and the **Preview** window appear.

   The Preview window displays the selected products.





Product representations visualized in the Group Preview window do not take sticker representations into account.

To customize the default display setting for the Preview window, see *Customizing General DMU Settings* in the DMU Navigator user guide.

3. To add a product to the Group content, select the product in the specification tree or the geometry area.

   The product is added to the Group content listed in the Edit Group dialog box.

4. To remove a product from the Group content, you can either:

   - de-select the product in the Specification Tree or in the Main window
   - select the product in the Edit Group dialog box and click the **Remove from Group** icon
   - select the product in the Preview window, right-click and select **Remove from Group** in the contextual menu

   Note: The multi-selection is now available in the Identifier list of the Edit Group dialog box.

5. In the **Name** text-entry field, enter a name for the group you wish to create.

6. Click **OK** to create the group.

   The group is identified in the specification tree. Groups created in this manner are persistent and can be stored in the document.

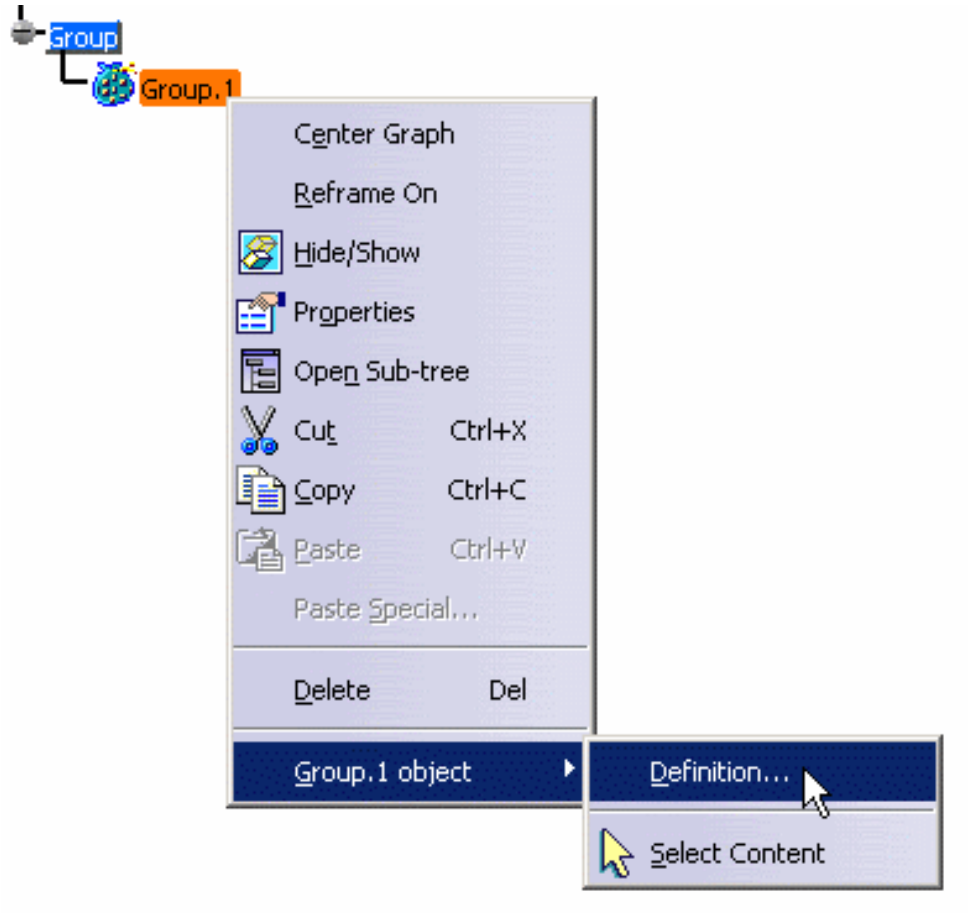
Editing a Group

7. To edit a group, in the specification tree, double-click the group

   or

   right-click the group and select **Group.1 object -> Definition** from the contextual menu.

   The **Edit Group** dialog box appears and lists the content of the group you just created. Products in the group are highlighted in the specification tree and in the geometry area.



Note that although the group is selected and its content is highlighted in both the specification tree and in the geometry area, the group content is not considered selected.

8. Modify the Group content as desired.

9. Click **OK** to confirm.

Selecting Group Content in the Main Viewer

10. To select the Group content in the main viewer, right-click the Group in the specification tree and select **Group.1 object -> Select Content** from the contextual menu.

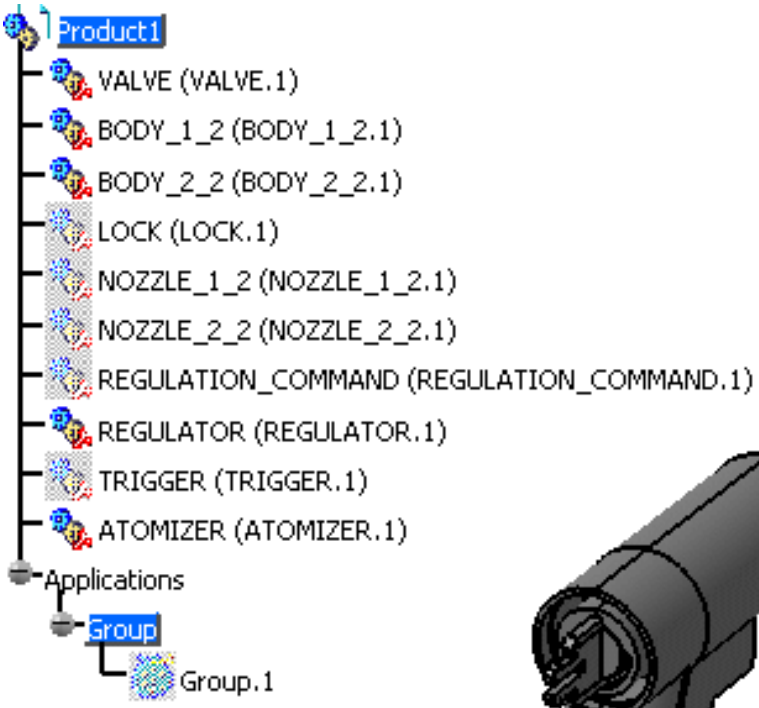
    The group components are highlighted in both the specification tree and in the geometry area and the group content is considered selected.

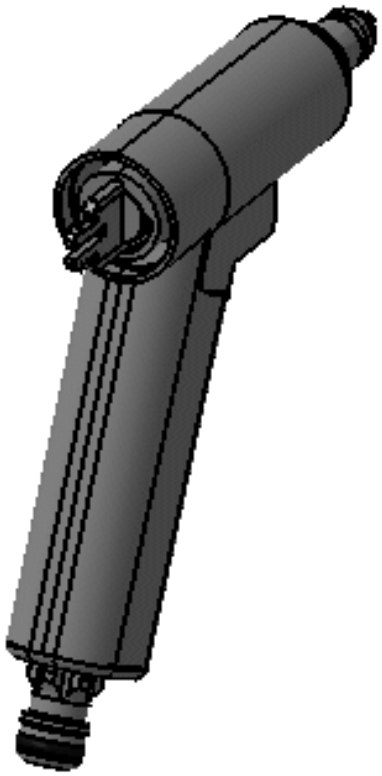
Hiding Group Content

11. To hide the Group content, click the **Hide / Show** icon  .

    The group components are hidden and the icons are grayed out in the specification tree.

Note: If you then move individual components back into the show space, the group icon in the specification tree remains grayed out.





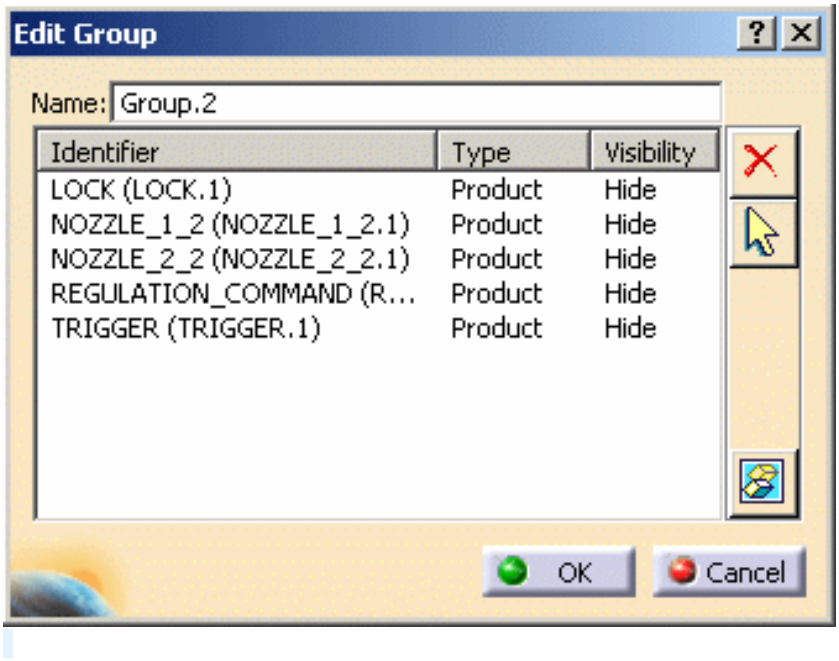
Showing Hidden Components in the Preview Window

12. After having hidden the group content in the previous step, double-click the Group in the specification tree.

    The Edit Group dialog box and the Preview window appear.

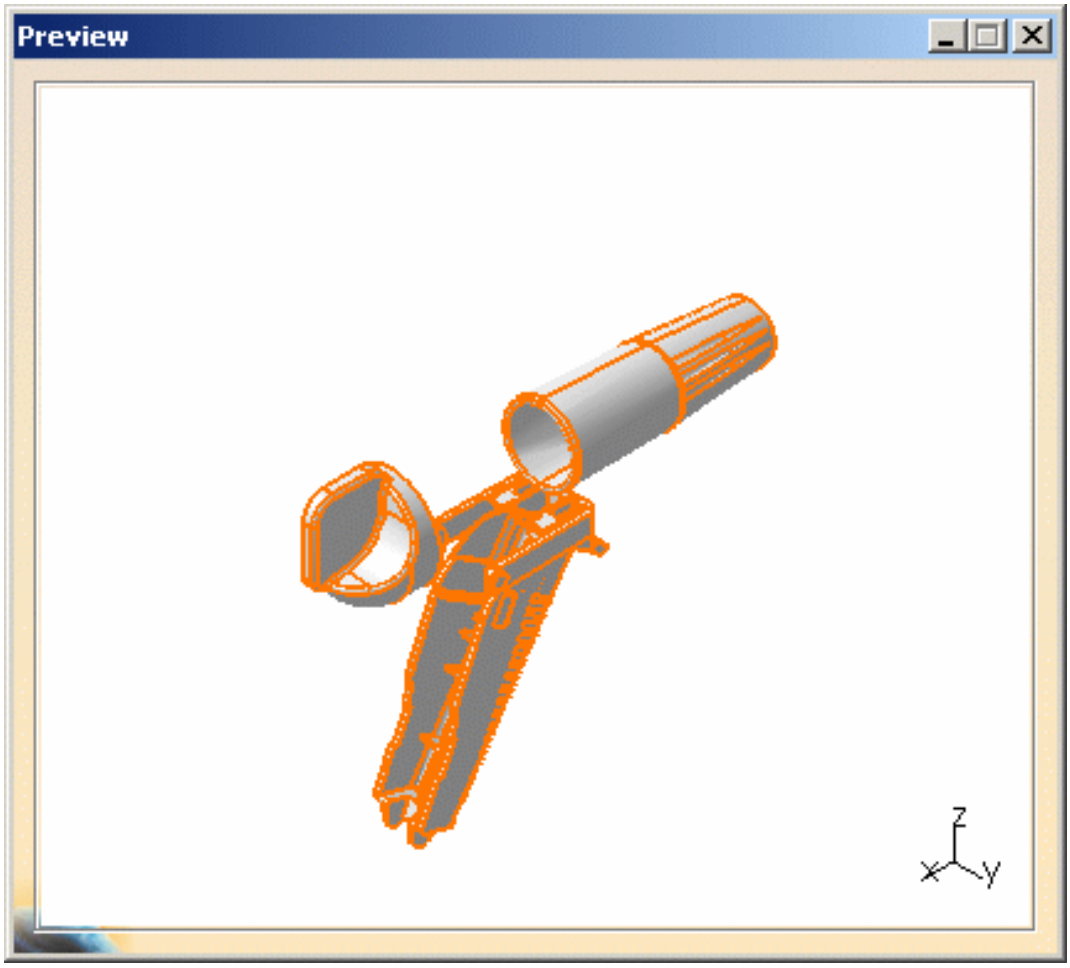
    The group content is indicated in the specification tree, but the icons are still grayed, indicating that the products are hidden.

    The Edit Group dialog box will look as follows:



13. In the Edit Group dialog box, select all of the components (the multi-selection is now available) and click the **Show Hidden Objects** icon  .

    The **Visibility** attribute of each component will still be **Hide**, corresponding to the visibility status in the Main window. but the components will now be visible in the Preview window.



In the Edit Group dialog box, the value in the Visibility column corresponds to the value of the graphic property of the instance and is consistent with the visibility state of the icon in the specification tree. However, it is possible, due to inheritance, that it not be consistent with the product as displayed in the Main window.

Cross-Highlighting between the Edit Group dialog box and the Preview window

14. In the Edit Group dialog box, de-select LOCK and NOZZLE_1_2.

    The components are also de-selected in the Preview window.

15. In the Preview window, re-select LOCK and NOZZLE_1_2.

    The components are now selected in the Edit Group dialog box.

Note: The multi-selection is now available in the Identifier list of the Edit Group dialog box.

Defining Groups of Groups

You can now define groups of groups. A group that contains sub-groups will be automatically updated upon any modification to the content of its sub-groups.

Note: When you expand a group in the specification tree, you will not see products belonging to any sub-groups in order to avoid possible confusion with the product structure tree content.

16. Create Group.2 consisting of the components BODY_1_2 and BODY_2_2.
17. Create Group.3 consisting of components VALVE and ATOMIZER.
18. Click the title bar of the **Edit Group** dialog box to activate it.
19. In the specification tree, click Group.2.

    Group.2 is added to the content of Group.1.
20. In the specification tree, click Group.3.

    Group.3 is added to the content of Group.1.
21. In the **Edit Group** dialog box, click **OK** to confirm.

    The content of Group.1 in the specification tree now includes Group.2 and Group.3 as sub-groups.
22. In the specification tree, select Group.1.

    The content of Group.1 is highlighted in the main window. Its content now includes its original content as defined above in the section Creating a Group plus the content of its sub-groups, Group.2 and Group.3.

Replacing by highest common Father

The objective of this command is to optimize group content by replacing a set of brother products by their common father when all of the brothers belong to the same group. This optimization is especially interesting when selecting a group and highlighting its content.

Note: The optimization is not applied recursively on sub-groups.

1. In **Tools -> Customize**, add the **Factorize Content** command using a customized toolbar or access the command directly in **Views -> Commands List**.
2. If you added the command using a customized toolbar, then click the **Replace content by higher father** icon  .

   Brother products are accordingly replaced by their highest common father.

Replacing Content by Terminal Nodes

This command explodes groups: products are replaced by their children.

Note: This command is not recursive on sub-groups, sub-groups will be not exploded.

1. In **Tools -> Customize**, add the **Explode Content** command using a customized toolbar or access the command directly in **Views -> Commands List**.
2. If you added the command using a customized toolbar, then click the **Replace content by terminal nodes** icon  .
   Content is accordingly replaced by its terminal nodes.

You can also modify group properties (color, line type and weight).

Combining Space Analysis Commands

Sectioning in Interference & Distance Viewers
Measuring Minimum Distance in the Section Viewer
Sectioning & Visual Comparison

Sectioning in Interference & Distance Viewers

P2

You can section both your clash, distance and band analysis results directly in the results window of the appropriate command. All section plane manipulations, geometrical target and volume cut commands are available.

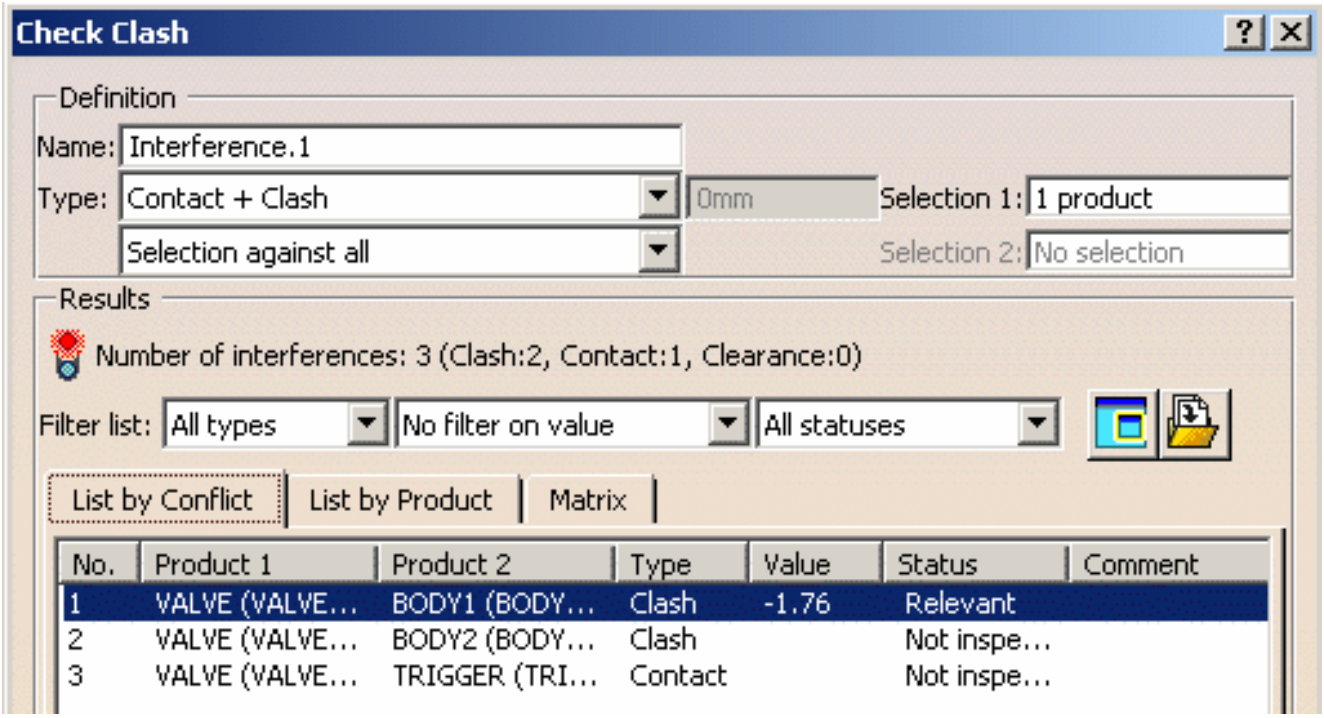
If you then browse your section in the Section viewer, the penetration depth or minimum distance and clash, distance or band analysis results are also visible.

This task explains how to section clash results in the Interference results window.

Insert the following sample cgr files: ATOMIZER.cgr, BODY1.cgr, BODY2.cgr, LOCK.cgr, NOZZLE1.cgr, NOZZLE2.cgr, REGULATION_COMMAND.cgr, REGULATOR.cgr, TRIGGER.cgr and VALVE.cgr.

They are to be found in the online documentation filetree in the common functionalities sample folder cfysm/samples.

1. Select **Insert -> Clash** from the menu bar or click the Clash icon in the DMU Space Analysis toolbar to run a check of type Contact + Clash between the valve and all the other products in the document (Selection against all).
   The Check Clash dialog box expands to show the results of the initial computation. The number of interferences detected along with the type is indicated in the dialog box: 3 interferences have been detected.



2. Click the Results window icon in the Check Clash dialog box to view the selected interference in a dedicated viewer.

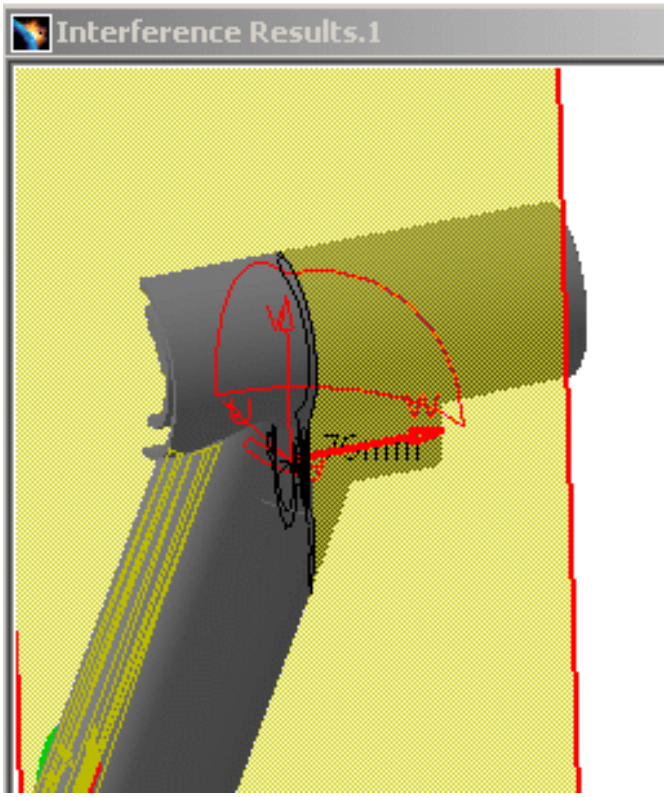
   The Interference Results window appears displaying the products in conflict only.

3. Tile the Interference Results and document windows vertically (**Window -> Tile Vertically**).
4. Click the Sectioning icon.

   The section plane is generated on the current clash result.

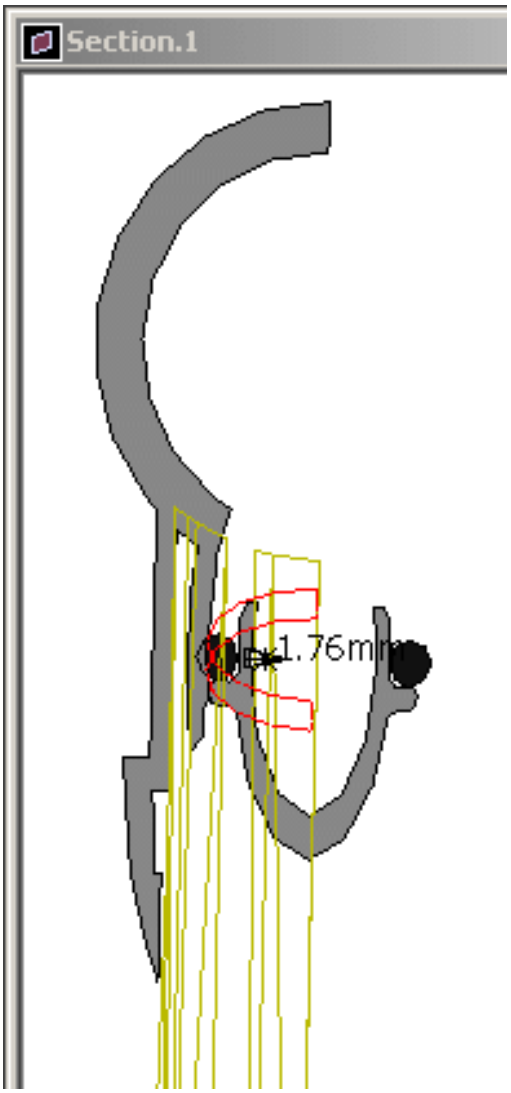
   The plane is created parallel to the direction of the penetration depth (clash) or minimum distance (clearance). In the case of a contact result, the center of the plane is placed on the center of the bounding sphere around the products selected.

   Sectioning tools are available in the Interference Results window: you can manipulate the plane directly, create a 3D section cut and position the plane on a target.

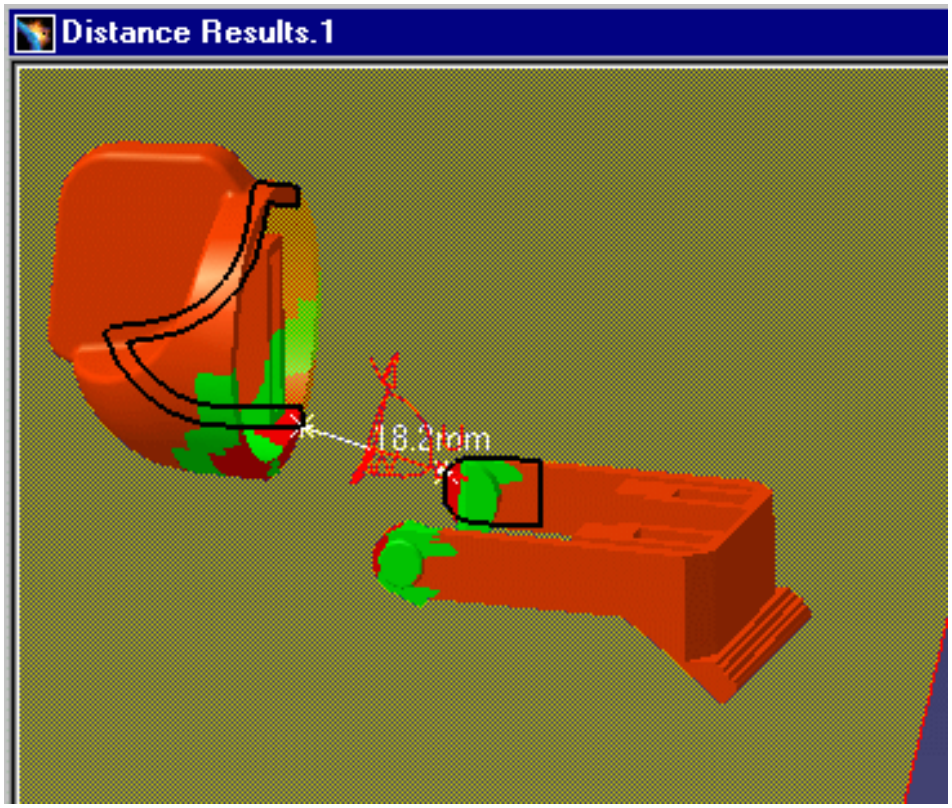


The Section viewer is automatically tiled vertically alongside other document windows.

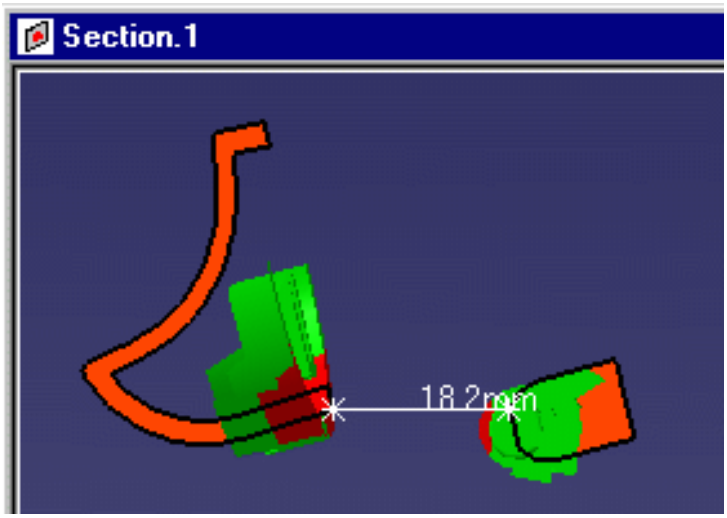
The penetration depth and clash results are viewed in the Section viewer.



Example showing sectioning in the Distance Results window and Section viewer. The section plane is created parallel to the direction of the minimum distance and the center of the plane is placed on the center of the measured distance.



Note: The minimum distance and band analysis results are shown in the Section viewer.



You can use the Clash Detection icon (Result tab of Sectioning Definition dialog box) in the Section viewer to detect collisions between 2D sections.

Measuring Minimum Distance in the Section Viewer

Distance & Band Analysis lets you measure minimum distance at product level, irrespective of whether or not the section view is a filled view. The result you obtain is the minimum distance between the sections of selected products (model, CATPart, etc.).

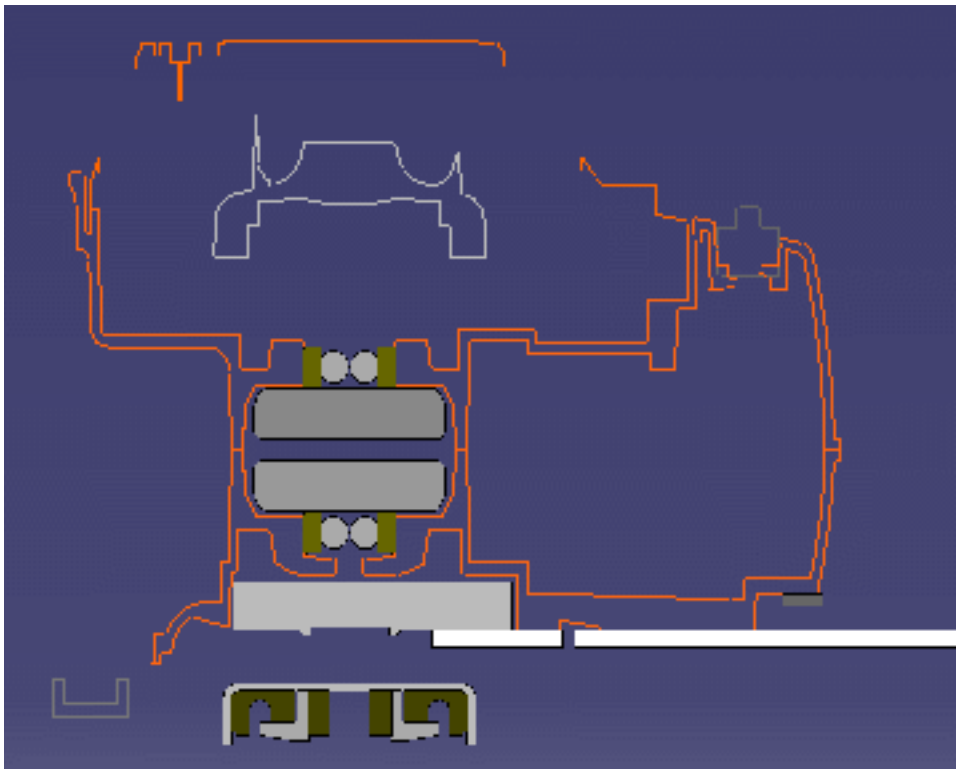
Measure Between lets you measure minimum distance between edge primitives or at element level depending on whether the section view is a filled view or not:

- Filled section view: the result is the minimum distance between the sections of selected elements (solid of a model, body of a CATPart).
- Unfilled section view: the result is the minimum distance between edge primitives.

This task explains how to measure minimum distance in the Section viewer.

No sample document is provided.

1. Select **Insert -> Sectioning** from the menu bar, or click the Sectioning icon in the DMU Space Analysis toolbar and create a section plane and the corresponding section.
   The Section viewer is automatically tiled vertically alongside the document window. By default, the section view is a filled view.



To measure minimum distance between edge primitives, the section view must be unfilled.

2. Click **OK** in the Sectioning Definition dialog box to exit the sectioning command.

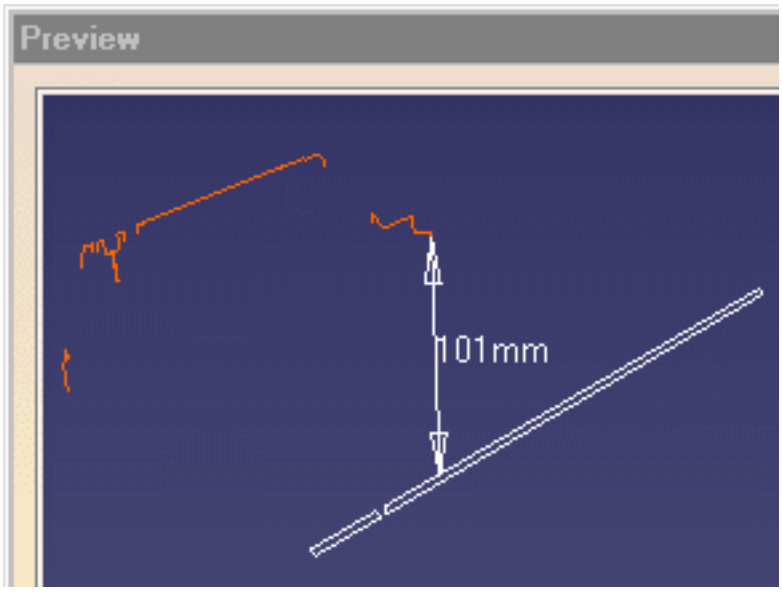
Note: The Section viewer remains tiled alongside the document window.

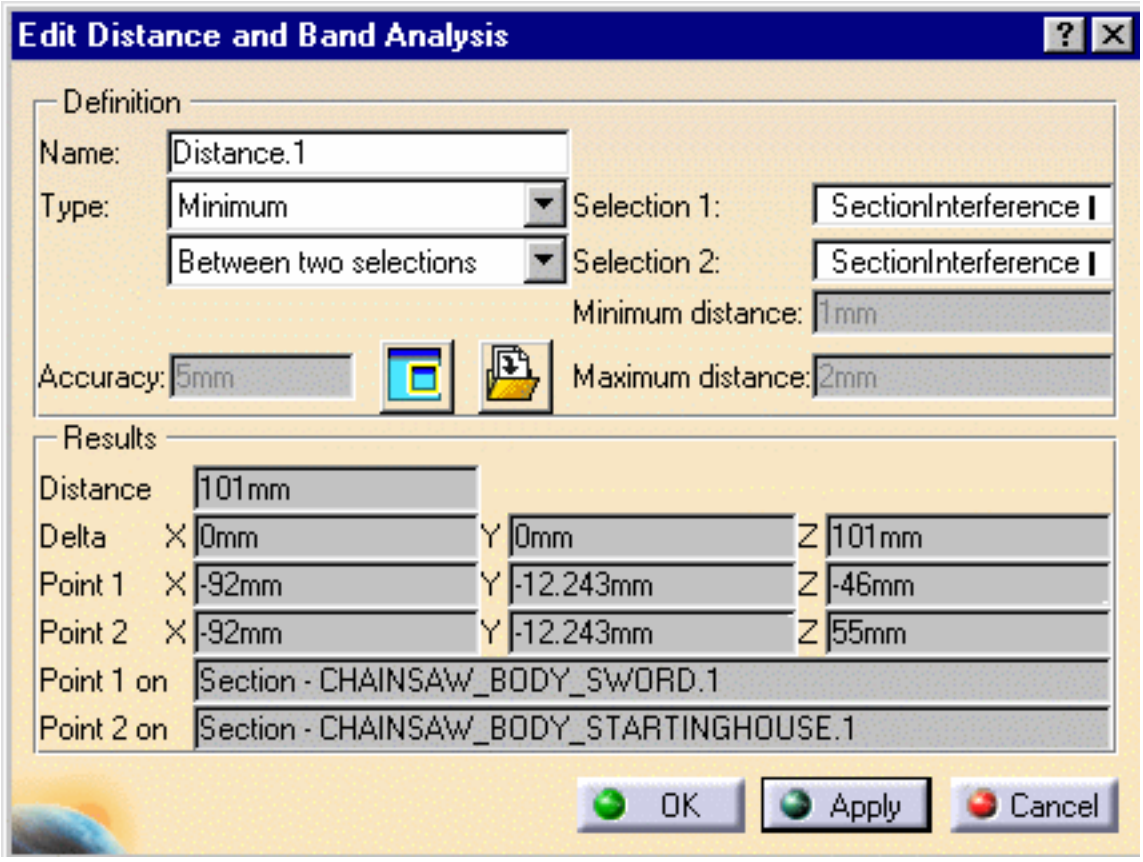
3. Select your measure command:

Measure at Product level

- Click the Distance and Band Analysis icon and make your measure between two selections.

The result you obtain is the minimum distance between the sections of selected products (model, CATPart, etc.).

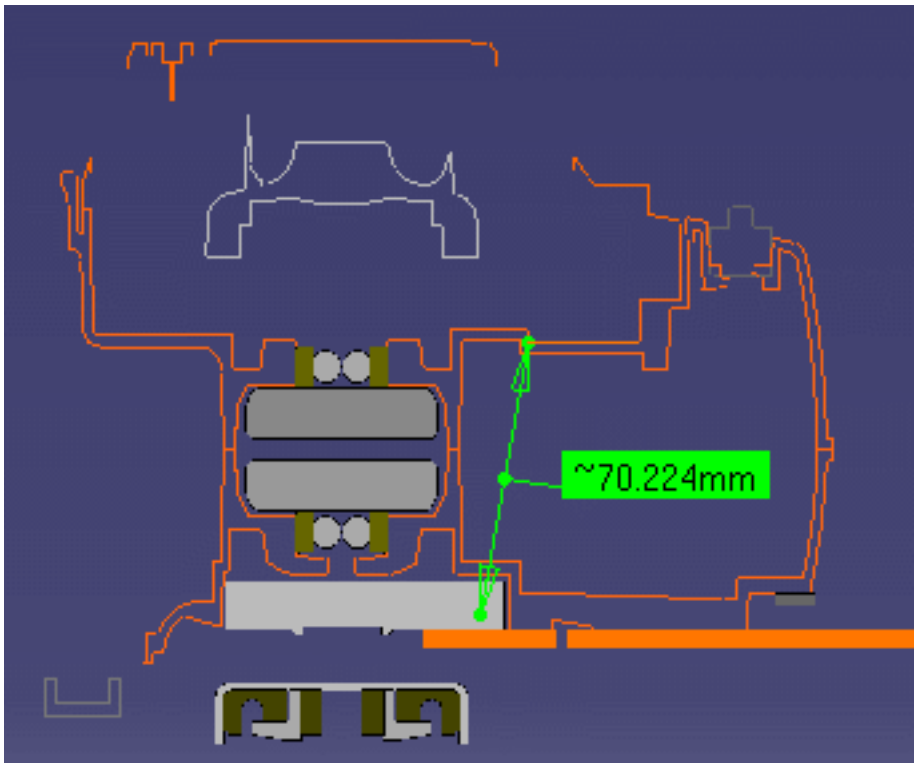


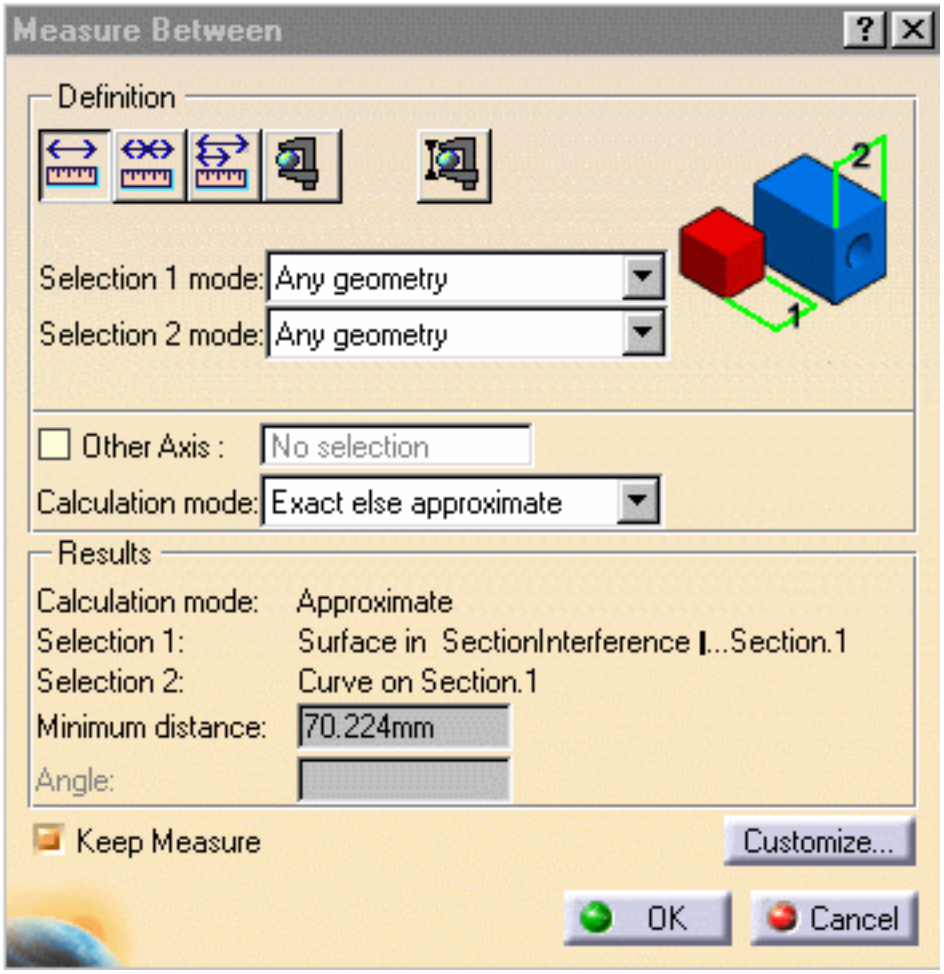


Measure at Element level

- Click the Measure Between icon and make your measure.

The result is the minimum distance between the sections of elements (solid of a model, body of a CATPart).



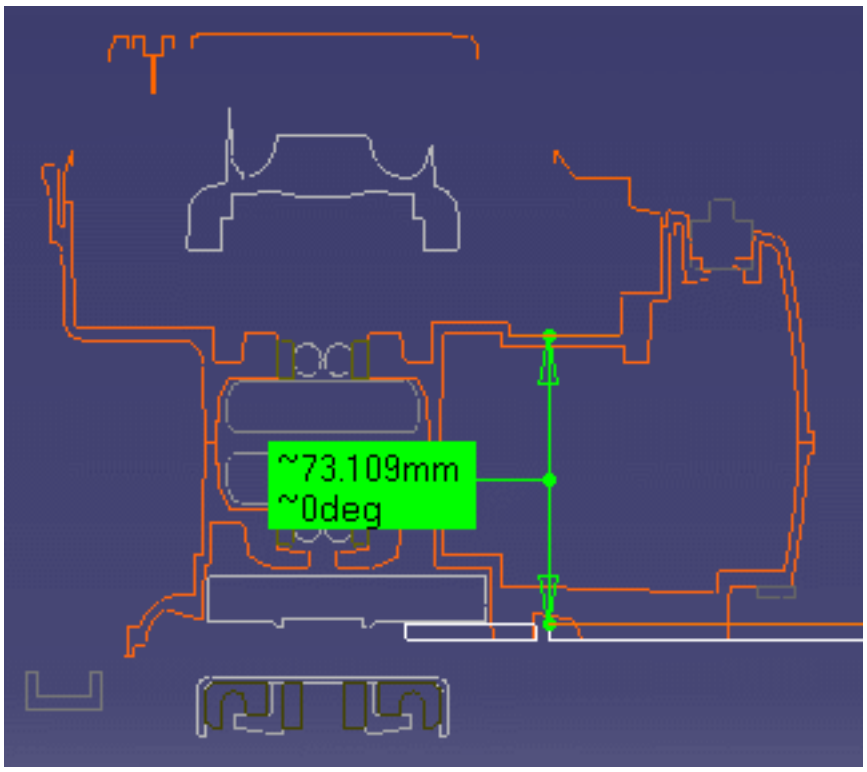


Between Edge Primitives

Important: To be able to measure the minimum distance between edge primitives, you must de-activate the Section Fill icon in the Result tab of the Sectioning Definition dialog box before you exit the Sectioning command.

- Click the Measure Between icon and make your measure.

The result is the minimum distance between edge primitives.



Results

Calculation mode:	Approximate
Selection 1:	Curve on Section.1
Selection 2:	Curve on Section.1
Minimum distance:	73.109mm
Angle:	0deg

If the Keep Measure option is selected, measures made using measure tools in the Section viewer are added to the specification tree under the Section entry. These measures can only be seen in the Section viewer and are not visible in the 3D document.

Sectioning & Visual Comparison

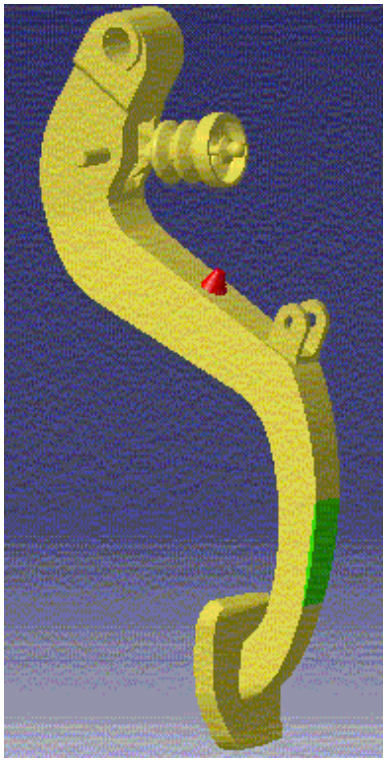
You can section your visual comparison results and browse the section in the Section viewer. Comparison colors identifying common, added and/or removed material are kept in the Section viewer.

This task illustrates the integration between Compare Products and Sectioning commands.

No sample document is provided.

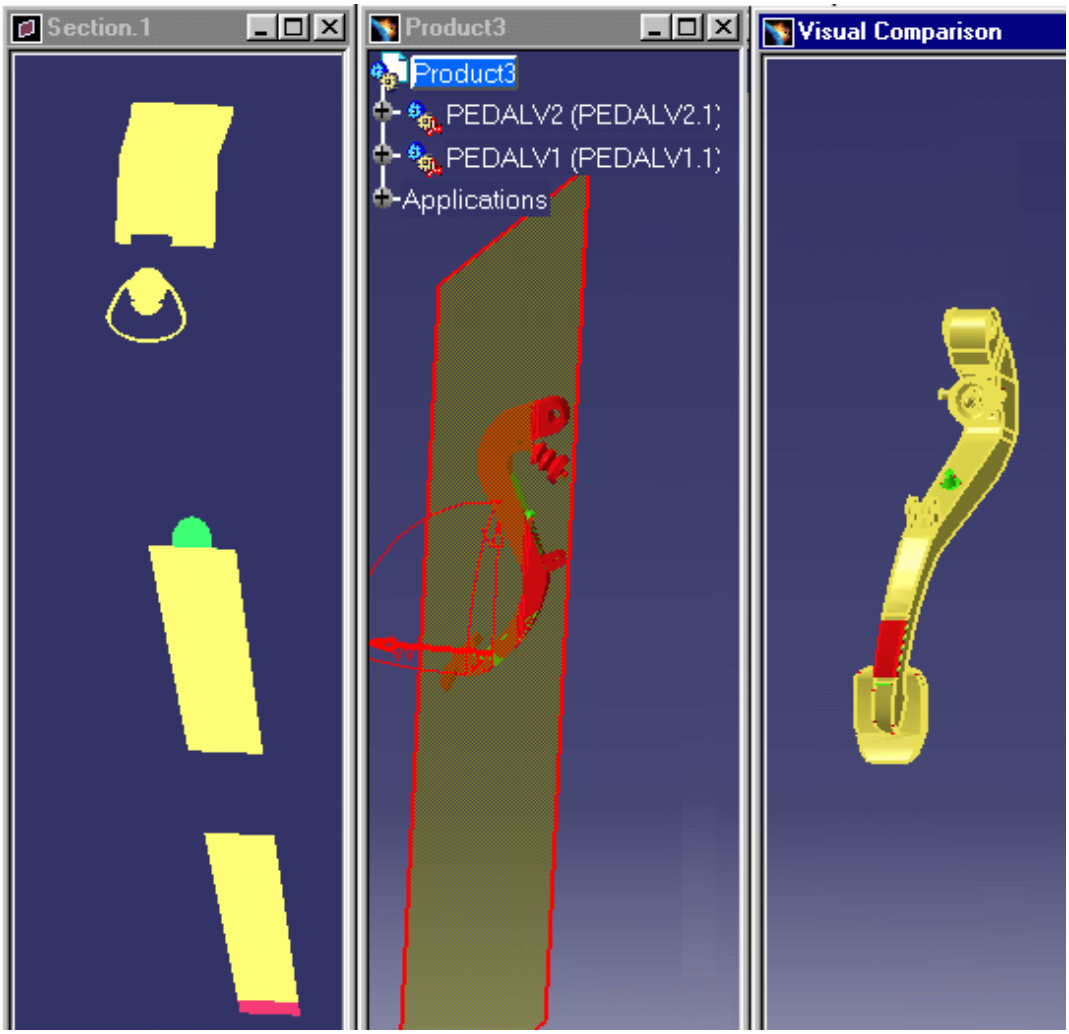
1. Click the Compare Products icon and run a visual comparison.

   The Visual Comparison viewer opens showing the results.



2. Click the Sectioning icon.

   Sectioning tools are available in the main document window: you can manipulate the plane directly, create a 3D section cut and position the plane on a target.
   The Section viewer is automatically tiled vertically alongside the other windows and is locked in a 2D view.



3. Manipulate the section plane in the document window and browse results in the Section viewer.
Note: Comparison colors identifying common, added and/or removed material are kept in the Section viewer:
   - Yellow: common material
   - Red: added material
   - Green: removed material.

Batches and Macros for Interference Analysis

Batch Processing CGR Files & Model Documents

This task explains how to run interference analyses as a batch process on cgr files.

Note: The batch process can also be run on model documents.

The batch process takes into account penetration mode settings.

To batch process interference analyses, you need to prepare two input files, one defining the computation parameters and the other listing the cgr files or model documents you want to analyze.

On UNIX, the variable DISPLAY must be correctly valuated.

1. Prepare the input file defining computation parameters.

   A typical computation parameters file looks like this:

Computation type	= clearance + contact + clash
Clearance value	= 20.0
Input file	= OK_YLE
Output file	= OK_YLE.out
Result file	= OK_YLE.result

   Computation type:
   You can run your analysis on one of two interference types:
   - Contact + Clash
   - Clearance + Contact + Clash

   Clearance value:
   Is required if you specified Clearance + Contact + Clash as computation type and must be specified in millimeters. The clearance value must be a positive value.

   Input file:
   Lists the files you want to analyze. You can batch process cgr files.

   Output file:
   Gives the overall results of the computation and lists any files posing a problem during the computation.

   Result file:
   Gives the detailed results, listing the pairs of files in conflict and indicating the interference type and value.

2. Prepare the input file listing the files you want to analyze:

A typical model input file looks like this:

```
group           = 1
#####################################
path            = /u/users/yle/mdlv5/WHITE.cgr
matrix           = 1.000000 0.000000 0.000000
                0.000000 1.000000 0.000000
                0.000000 0.000000 1.000000
                100.0000 0.000000 0.000000
//-------------------------------------------------
path            = /u/users/yle/mdlv5/BLUE.cgr
matrix           = 1.000000 0.000000 0.000000
                0.000000 1.000000 0.000000
                0.000000 0.000000 1.000000
                0.000000 -100.000 0.000000
//-------------------------------------------------
group            = 2
#####################################
path            = /u/users/yle/mdlv5/YELLOW.cgr
matrix           = 1.000000 0.000000 0.000000
                0.000000 1.000000 0.000000
                0.000000 0.000000 1.000000
                -300.000 0.000000 -200.000
//-------------------------------------------------
path            = /u/users/yle/mdlv5/RED.cgr
matrix           = 1.000000 0.000000 0.000000
                0.000000 1.000000 0.000000
                0.000000 0.000000 1.000000
                0.000000 0.000000 100.0000
//-------------------------------------------------
path            = /u/users/yle/mdlv5/file_not_present_in_directory.cgr
```

Note: Special characters # and // identify comments.

Group (optional) Used to group files you want to analyze.
Groups must be described in increasing order: group 1, group 2, etc.
The batch process tests each file in the group against all files in all other groups. No test is run on files within a group.
Note: If no group is specified, the batch process tests each file listed against all other files.

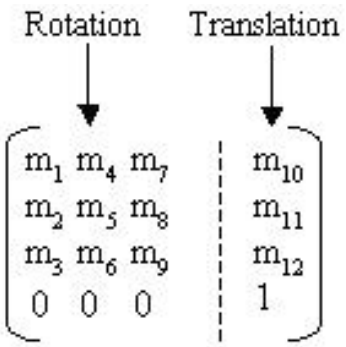
Path Tells the batch where to find the files you want to analyze.

Matrix (optional) Positions the file just identified with respect to the absolute axis system of the document. Enter data in mm in text format. For example:

```
1.000000 0.000000 0.000000
0.000000 1.000000 0.000000
0.000000 0.000000 1.000000
100.0000 0.000000 0.000000
```

The matrix is read in order as follows:
m_1, m_2, m_3
m_4, m_5, m_6
m_7, m_8, m_9
m_{10}, m_{11}, m_{12}

For information, this corresponds to the transposed matrix:



3. Run the following shell to start the batch process:

Under UNIX:

- Place yourself in the following directory:
  cd /install_folder/code/command
- Run the command:
  ./catstart -run"ITFCHECK input_file_defining_computation_parameters."

Under Windows:

- Write a shell script containing the following lines:
  cd \install_folder\code\bin
  ITFCHECK "input_file_defining_computation_parameters"
- Run the shell.

Note: The "install_folder" is the name of the installation directory or folder. For more information on installing DMU, see the *Infrastructure User's Guide*.

The results of the batch process are written to the two files identified in the calculation parameters input file.

Output file: indicates the number of interferences detected along with the type, and lists any files posing a problem during the computation.
If no output file is indicated, the results are written to a default output file ITFCHECK.out in the directory from which the batch was run.

Typical output file:

```
======== Global Result of Computation =============



Number of models to study:                5
Number of problematic models:             1
Number of computations:                   4
Number of interferences:                  4
Number of clashes detected:               1
Number of contacts detected:              0
Number of clearances detected:            3



=====================================================

======== List of Problematic Models =================
Inexisting file or file not accessible:
-------------------------------------------------
/u/users/yle/mdlv5/file_not_present_in_directory.cgr
-------------------------------------------------
=====================================================
```

Result file: gives the detailed results, listing the pairs of files in conflict and indicating the interference type and value (penetration depth or minimum distance depending on whether a clash or clearance conflict was detected).
If no result file is indicated, detailed results are written to the output file.

Typical result file:

```
-----------------------------------------------
/u/users/yle/mdlv5/WHITE.cgr
matrix          1.000000 0.000000 0.000000
              0.000000 1.000000 0.000000
              0.000000 0.000000 1.000000
              100.0000 0.000000 0.000000
/u/users/yle/mdlv5/YELLOW.cgr
matrix          1.000000 0.000000 0.000000
              0.000000 1.000000 0.000000
              0.000000 0.000000 1.000000
              -300.000 0.000000 -200.000
Inteference type = Clearance  Value = 233.452
-----------------------------------------------
/u/users/yle/mdlv5/WHITE.cgr
matrix          1.000000 0.000000 0.000000
              0.000000 1.000000 0.000000
              0.000000 0.000000 1.000000
              100.0000 0.000000 0.000000
/u/users/yle/mdlv5/RED.cgr
matrix          1.000000 0.000000 0.000000
              0.000000 1.000000 0.000000
              0.000000 0.000000 1.000000
              0.000000 0.000000 100.0000
Inteference type = Clash  Value = -15.3265
-----------------------------------------------
```

```
/u/users/yle/mdlv5/BLUE.cgr
matrix          1.000000 0.000000 0.000000
              0.000000 1.000000 0.000000
              0.000000 0.000000 1.000000
              0.000000 -100.000 0.000000
/u/users/yle/mdlv5/YELLOW.cgr
matrix          1.000000 0.000000 0.000000
              0.000000 1.000000 0.000000
              0.000000 0.000000 1.000000
              -300.000 0.000000 -200.000
Inteference type = Clearance  Value = 270
-----------------------------------------------
/u/users/yle/mdlv5/BLUE.cgr
matrix          1.000000 0.000000 0.000000
              0.000000 1.000000 0.000000
              0.000000 0.000000 1.000000
              0.000000 -100.000 0.000000
/u/users/yle/mdlv5/RED.cgr
matrix          1.000000 0.000000 0.000000
              0.000000 1.000000 0.000000
              0.000000 0.000000 1.000000
              0.000000 0.000000 100.0000
Inteference type = Clearance  Value = 70
-----------------------------------------------
```

Note: The penetration mode for the examples given in this task is product.

You can batch process interference analyses on ENOVIA VPM and ENOVIA LCA data, however, in this case the batch must be run from ENOVIA VPM or ENOVIA LCA respectively. For more information, see ENOVIA VPM or ENOVIA LCA documentation.

Batch Processing Existing CATProducts

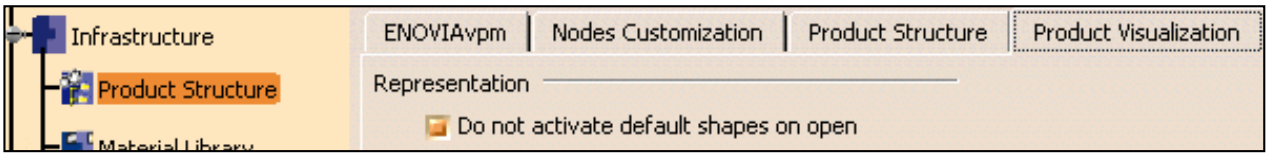
This task explains how to run interference analyses as a batch process on existing CATProduct documents.

Results are written to as many XML files as clash specifications, as well as in the output CATProduct.
The CATProduct document must contain clash specifications. One or more specifications can be defined.

1. Start an interactive CATIA session.

2. Set appropriate CATIA settings in the Options dialog box:
   - Select the **Tools -> Options** command
   - To batch process existing CATProducts correctly, select the Work with the cache system option in the **Cache Management** tab (**Infrastructure -> Product Structure**).

   - To reduce the memory needed during batch processing, select the Do not activate default shapes on open option in the **Product Visualization** tab (**Infrastructure -> Product Structure**)



   When this option is selected, the batch process manages the number of products loaded and unloaded at any one time, thereby reducing the memory needed.

3. Write clash specifications using the Clash command.

   Note: A simple **OK** is sufficient in the Check Clash dialog box.

4. Prepare the input file:

   Only one input file is necessary.

   A typical input file looks like this:

```
Input CATProduct = /u/users/yle/Test.CATProduct
Output CATProduct = /u/users/yle/Test_Result.CATProduct
Output file = Test_Result.txt
Directory for XML output = /u/users/yle/Test_XML
String for Save = 29_feb_2002
```

   Input CATProduct — Tells the batch where to find the CATProduct you want to analyze.

   Output CATProduct — Tells the batch where to write the CATProduct with the results.
   - If Output CATProduct = Input CATProduct, equivalent to a Save operation.
   - If Output CATProduct is different from Input CATProduct, equivalent to a Save As operation.
   - If not valuated, no CATProduct is written.

   Output file — Summaries the input information.

   Directory for XML output — Tells the batch where to write the XML files. If this keyword is not valuated, no XML files are generated.

String for Save	String used to generate the XML filename. Rule used to generate the XML filename: XML filename = specification_identifier + String_for_Save + extension .xml

5. Run the following shell to start the batch process:

Under UNIX:

- Place yourself in the following directory:
  cd /install_folder/code/command
- Run the command:
  ./catstart -run"ITFCHECK input_file_defining_computation_parameters."

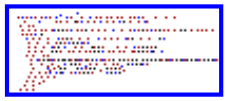
Under Windows:

- Write a shell script containing the following lines:
  cd \install_folder\code\bin
  ITFCHECK "input_file_defining_computation_parameters"
- Run the shell.

Note: The "install_folder" is the name of the installation directory or folder. For more information on installing DMU, see the *Infrastructure User's Guide*.

The results of the batch process are written to as many XML files as clash specifications. They are also saved in the output CATProduct document.

Typical XML file:



You can import and analyze your results in an interactive DMU session using the Clash command.

Using a Macro to Import Clash Results Saved in XML Format

This task illustrates how to import clash results saved in XML files generated:

- From batch processing existing CATProducts
- In a standalone database.

This is done by running a macro which opens a CATProduct and imports the clash results stored in an XML file.

Have an XML file containing clash results and generated from batch processing an existing CATProduct.

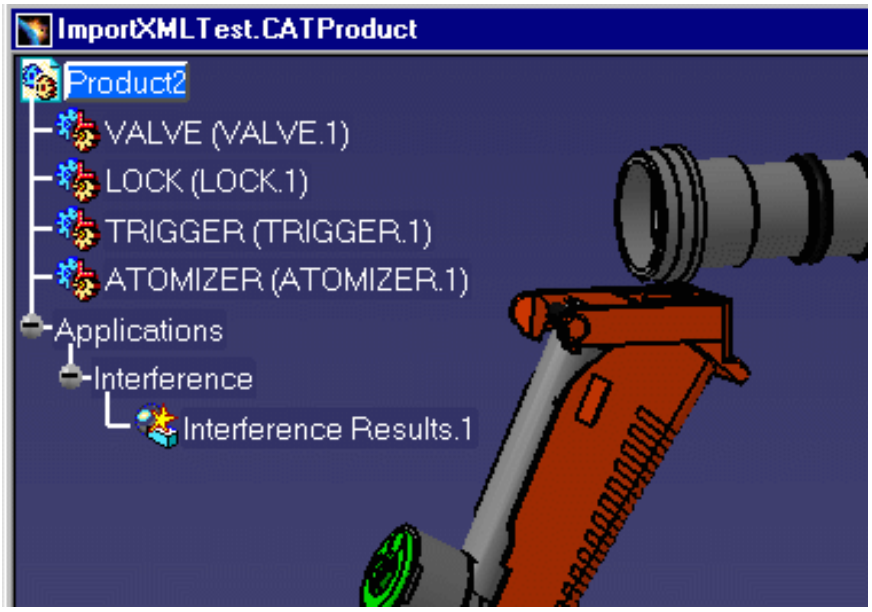
A sample macro, ImportClashResultsfromXML.CATScript, is supplied.

1. Open and edit the macro to:
   - Select **Tools -> Macro -> Macros**
   - Identify the folder containing the macro, select the CATScript then click **Edit...**
   - Add the name and path of the CATProduct that will be opened and that will contain clash results. Typically, this CATProduct contains the products on which the interference analysis was run.
   - Add the name and path of the XML file containing the clash results you want to import.
   - Save the macro.

Sample Macro

```
Sub CATMain()

' 1 - Load the document
sProductPath = CATIA.SystemService.Environ("E:\tmp\ImportXMLTest")
Dim documents1 As Documents
Set documents1 = CATIA.Documents
Set productDocument1 = documents1.Open("E:\tmp\ImportXMLTest.CATProduct")
CATIA.ActiveWindow.ActiveViewer.Viewpoint3D.ProjectionMode = 0

' 2 - Retrieve the Clashes collection
Dim rClashes As ClashResults
Set rClashes = CATIA.ActiveDocument.Product.GetTechnologicalObject("ClashResults")

' 3 - Read a clash stored on the disk, and import it in the active document
Dim rClash As ClashResult
Set rClash = rClashes.AddFromXML("E:\tmp\XML_file_to_Import.xml", CatClashImportTypeClashOnly)

End Sub
```

2. Click **Run** to run the macro.
   A CATProduct is opened in your current DMU session.



Double-click the results to analyze them using the Clash command.

Selecting the Compute all numeric and graphic results check box in the DMU Clash tab page (**Tools -> Options -> Digital Mockup -> DMU Space Analysis**) means that all numeric and graphic results are computed and displayed directly when you import clash results via the macro. This is useful when presenting results in a meeting for example. Cleared, the detailed computation will only be run as you select conflicts in the Check Clash dialog box.

You can also add your macro to a toolbar if desired using the **Tools -> Customize...** command. Select the Commands tab, then the Macros category: all macros will be detected and listed. You can then drag and drop onto toolbars for convenient access.

Importing XML Clash Results & VPM

You can also import clash results saved in XML files that concern products stored in VPM. This is done using the CATDMUBuilder batch process.

This batch reads the XML file, queries VPM for the products impacted, then re-creates the corresponding CATProduct and clash results.

For more information on CATDMUBuilder, see the *DMU Navigator User's Guide*.

Using a Macro to Export Clash Results to an XML File

This task illustrates how to export, in XML format, a clash result saved in a CATProduct.

This is done by running a macro which opens the CATProduct containing the clash results and exports the first result to an XML file.

- Have a CATProduct containing clash results.
- Have cleared the Federated XML output option in the DMU Clash - Process tab of the Options dialog box (**Tools -> Options -> Digital Mockup -> DMU Space Analysis**).

1. Open and edit the macro to:
   - Select **Tools -> Macro -> Macros**
   - Identify the folder containing the macro, select the CATScript then click **Edit...**
   - Add the name and path of the CATProduct that will be opened and that contains the clash results you want to export.
   - Add the name (without the file extension) and path of the XML file that will contain exported clash results.
   - Save the macro.

   ## Sample Macro

```
Sub CATMain()

' 1 - Load the document
sProductPath = CATIA.SystemService.Environ("C:\data\XML")
Dim documents1 As Documents
Set documents1 = CATIA.Documents
Set productDocument1 = documents1.Open("C:\data\XML\product1.CATProduct")
CATIA.ActiveWindow.ActiveViewer.Viewpoint3D.ProjectionMode = 0

' 2 - Retrieve the Clashes collection
Dim rClashes As Clashes
Set rClashes = CATIA.ActiveDocument.Product.GetTechnologicalObject("Clashes")

' 3 - Retrieve the first clash result
Dim rClash As Clash
Set rClash = rClashes.Item(1)

' 4 - Export it as a XML document
rClash.Export CatClashExportTypeXMLResultOnly, "C:\data\XML\MacroExport\product1"

End Sub
```

   - If the CATProduct containing clash results has been loaded into your CATIA session, delete the first step from the macro.
   - You can also add a loop to your macro to export more than one clash result.

2. Click **Run** to run the macro.
   - A CATProduct is opened in your current DMU session.
   - A single XML file containing the first clash result is generated. This file is located in the folder identified in the macro.

For more information on XML grammar, see reading XML clash files.

You can also add your macro to a toolbar if desired using the **Tools -> Customize...** command. Select the Commands tab, then the Macros category: all macros will be detected and listed. You can then drag and drop onto toolbars for convenient access.

Using a Macro to Batch Process Product Comparison

If you perform a task repeatedly, you can take advantage of a macro to automate it. A macro is a series of functions, written in a scripting language, that you group in a single command to perform the requested task automatically.

You can in this way run a macro automating the geometric comparison of hundred of products to detect differences between them. You can compare .model, .CATPart and .cgr documents.

A sample macro, ComparisonMacro.CATScript, is supplied. It can be found in the DMU Space Analysis samples folder **spaug/samples.**

This task explains how to automate geometric comparison.

1. Open and edit the sample macro, ComparisonMacro.CATScript, from the sample folder with a Text editor.

   Appropriate comments have been added to the sample to help you edit it.

```
Language="VBSCRIPT"

''==========================================
'' MinDiff = Difference limit,
'' if (difference percent > MinDiff)
'' then products are considered different
''==================================
Dim MinDiff As Double
MinDiff = 0.3

''============================
'' Difference percentages :
''============================
'' added : Added Material /Material in Version1
'' removed : Removed Material /Material in Version1
Dim added As Double
Dim removed As Double

''================
'' Program Start
''================
Sub CATMain()

Dim documents1 As Documents
Set documents1 = CATIA.Documents

''======================
'' New Product Creation
''======================
Dim productDocument1 As Document
```

```
Set productDocument1 = documents1.Add("Product")

Dim product1 As Product
Set product1 = productDocument1.Product

Dim products1 As Products
Set products1 = product1.Products

''=========================================
'' Names of the two products to compare
''=========================================

Dim arrayOfVariantOfBSTR1(1)

''arrayOfVariantOfBSTR1(0) = "path\name_of_document"
' ' arrayOfVariantOfBSTR1(1) = "path\name_of_document"

''=======================
'' Insertion of products
''=======================
products1.AddComponentsFromFiles arrayOfVariantOfBSTR1, "*"

Dim optimizerWorkBench1 As Workbench
Set optimizerWorkBench1 = productDocument1.GetWorkbench("OptimizerWorkBench")

''=========================================
'' Products to compare
''=========================================

Dim product2 As Product
Set product2 = products1.Item(1)

Dim product3 As Product
Set product3 = products1.Item(2)

''=========================================
'' Comparison
''=========================================
Dim partComps1 As PartComps
Set partComps1 = optimizerWorkBench1.PartComps
Dim partComp1 As PartComp
''Set partComp1 = partComps1.GeometricComparison(product2, product3, 2.000000,
2.000000, 2, added, removed)

''=========================================
'' Start Comparison
'' Parameters :
'' product2 : first product to compare (Old Version)
'' product3 : second product to compare (New Version)
'' 2.000000 : computation accuracy (mm)
'' 2.000000 : display accuracy (mm)
```

```
'' 2 : computation type : 0=Added, 1=Removed, 2=Added+Removed
''=========================================
Set partComp1 = partComps1.Add(product2, product3, 2.000000, 2.000000, 2)

''=======================================
'' Read computation results
''=======================================

'' Retrieve the percent of added material (value is between 0.0 and 1.0)
Dim PercentAdded As Double
PercentAdded = partComps1.AddedMaterialPercentage

'' Retrieve the percent of removed material (value is between 0.0 and 1.0)
Dim PercentRemoved As Double
PercentRemoved = partComps1.RemovedMaterialPercentage

'' Retrieve the volume of added material (mm3)
Dim VolumeAdded As Double
VolumeAdded = partComps1.AddedMaterialVolume

'' Retrieve the volume of removed material (mm3)
Dim VolumeRemoved As Double
VolumeRemoved = partComps1.RemovedMaterialVolume

''=======================================
'' Typical comparison result management
''=======================================
If PercentAdded > MinDifference Then
msgbox "Difference detected : Added = " & Cstr(PercentAdded) & " , Removed = " &
Cstr(PercentRemoved) & " VolumeAdded = " & Cstr (VolumeAdded) & " VolumeRemoved
= " & Cstr(VolumeRemoved)

''=========================================
'' Save of added and removed Material
''=========================================
Dim document1 As Document
Set document1 = documents1.Item("AddedMaterial.3dmap")
document1.Activate
document1.SaveAs "E:\users\sbc\DemoSMT\Comparison\AddedMaterial.3dmap"

Dim document2 As Document
Set document2 = documents1.Item("RemovedMaterial.3dmap")
document2.Activate
document2.SaveAs "E:\users\sbc\DemoSMT\Comparison\RemovedMaterial.3dmap"

document2.Close
document1.Close

''
=========================================================
'' Import AddedMaterial Only
''
```

```
'============================================================
Dim var11 ( 0 )
var11 ( 0 ) = "E:\users\sbc\DemoSMT\Comparison\AddedMaterial.3dmap"
products1.AddComponentsFromFiles var11, "*"

''
'============================================================
'' Definition du view point
''
'============================================================

CATIA.ActiveWindow.ActiveViewer.Viewpoint3D.PutSightDirection Array(1., 1, 0)
CATIA.ActiveWindow.ActiveViewer.Viewpoint3D.PutUpDirection Array(0, 0, 1)

CATIA.ActiveWindow.ActiveViewer.Reframe
CATIA.ActiveWindow.ActiveViewer.ZoomIn()
CATIA.ActiveWindow.ActiveViewer.ZoomIn()

''
'============================================================
'' Save image As .jpg
''
'============================================================
CATIA.ActiveWindow.ActiveViewer.CaptureToFile catCaptureFormatJPEG ,
"E:\users\sbc\DemoSMT\Comparison\MyImage.jpg"

Else
msgbox "No difference detected between products"
End If

productDocument1.Activate

End Sub
```

2. Run the macro in batch mode from Windows or your UNIX workstation.

For information on editing and running macros, see the appropriate task in the *Infrastructure User's Guide*.

Knowledgeware Capabilities

Knowledgeware Rule-based Clash

Clash rules written using knowledgeware capabilities can be used in a standalone clash process or in an ENOVIA LCA clash process, ensuring clash analyses take corporate practices into account.

Multiple shape representations are supported letting you define context-specific shapes, for example a disassembly shape or an operator shape for a manually-operated valve.
This task shows you how to apply clash rules in interactive interference analysis. This involves two steps:

1. Writing clash rules using Knowledge Expert
2. Running interference analysis using the Clash command.

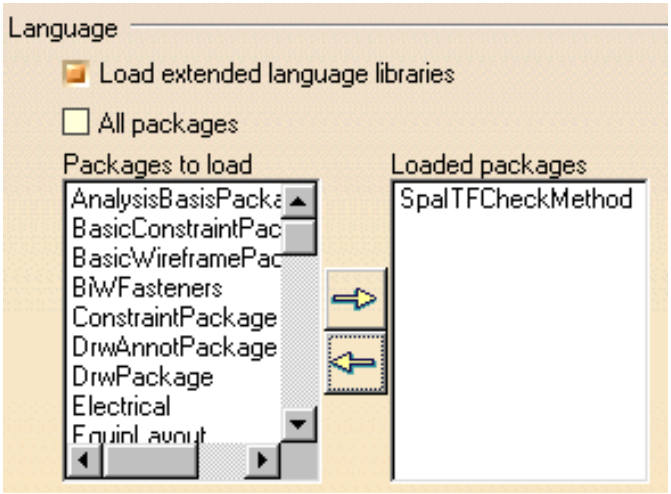
Two samples are provided:

- Knowledge_Rule-based_Clash.CATProduct (contains alternate shapes)
- Rule_for_Interference.CATProduct (contains rule).

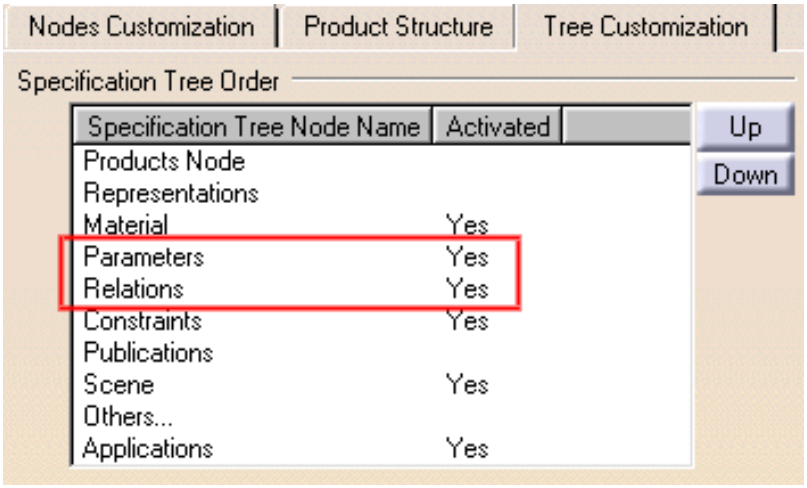
Writing Your Clash Rule

Before you begin, make sure you have selected the required packages.

1. To customize Knowledgeware settings:
   - Select **Tools -> Options, General -> Parameters and Measure**
   - Click the Knowledge tab
   - Select the Load extended language libraries check box
   - Load the SpaITFCheckMethod package

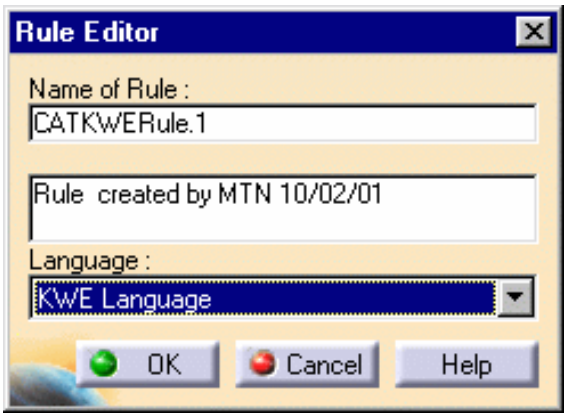


2. To customize Product Structure settings:
   - Select the **Infrastructure** category, then the **Product Structure** sub-category, and click the **Tree Customization** tab
   - Activate both Parameters and Relations options (the Constraints option is set to Yes by default)



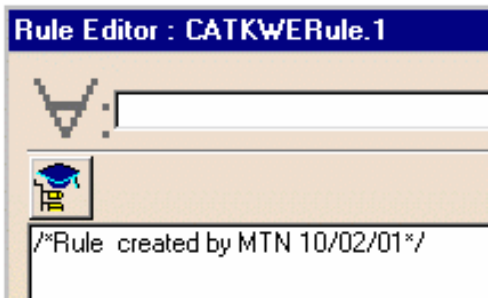
You are now ready to write your rule.

3. Select **Knowledgeware -> Knowledge Expert** from the **Start** menu to switch to the Knowledge Expert workbench.
4. Click the Expert Rule icon.



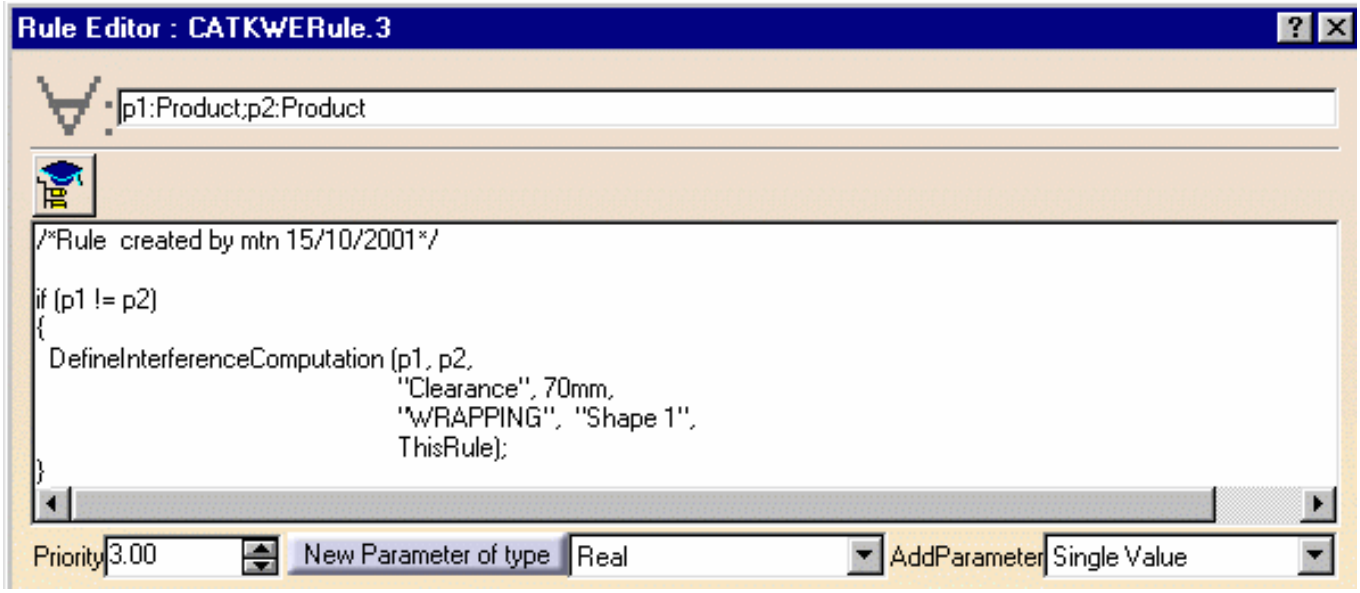
The Rule Editor dialog box appears.

5. (Optional) Modify the default rule name and comments.
6. Click **OK** when done.



The Rule Editor appears.

7. Write your rule.
   Here is a clash rule example below:



For more information about expert rules, see the *Knowledge Expert User's Guide*.

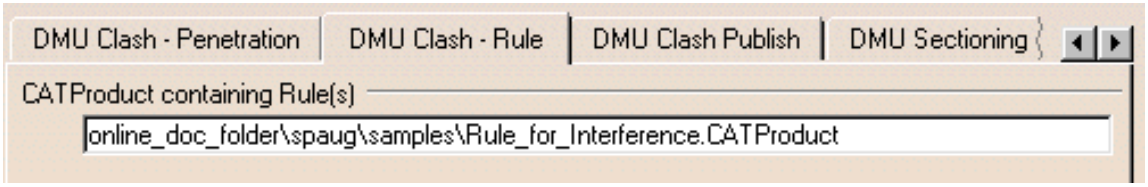
8. Click **Apply** to check rule syntax.
9. Click **OK** to add the expert rule to the document.
10. Click the Update icon to solve the rule base. If needed, update the document.
11. Save your clash rule in the CATProduct document.

Clash rules must be saved in a separate CATProduct document from the one in which you will run your interference analysis.

You can save more than one rule in the CATProduct document. If you do so, do not forget to give rules a priority in the Rule Editor dialog box. When running your interference analysis, rule priority is taken into account and if pairs of products and shapes satisfy more than one rule, the results of the rule with the highest priority only are output.

Running Your Interference Analysis

12. Identify the location of the document containing the clash rules:
    - **Tools -> Options -> Digital Mockup -> DMU Space Analysis**.
    - Click the DMU Clash - Rule tab.
    - Enter the full path for the CATProduct containing the clash rules.
    - Click **OK** when done.

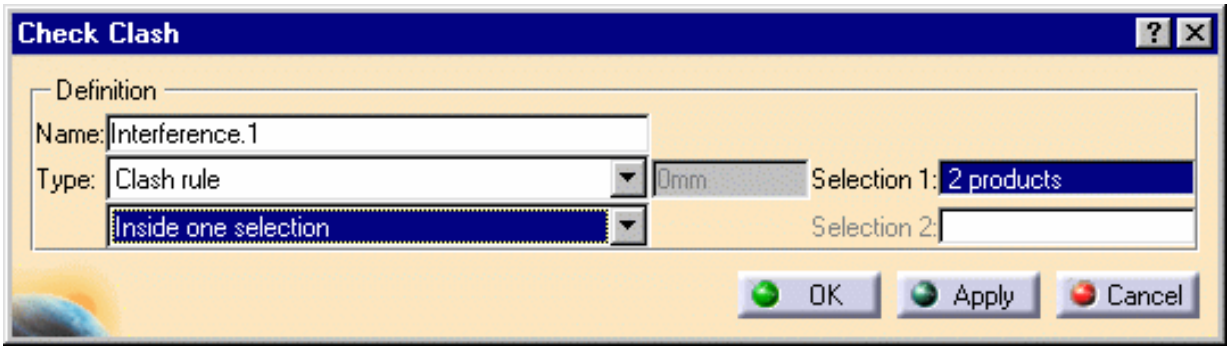


A sample document, Rule_for_Interference.CATProduct, containing rules is provided in the DMU Space Analysis samples folder **spaug/samples**.

13. Open the sample document containing alternate shapes: Knowledge_Rule-based_Clash.CATProduct.
14. Select **Insert -> Clash** from the menu bar or click the Clash icon in DMU Space Analysis toolbar.

The Clash dialog box appears.

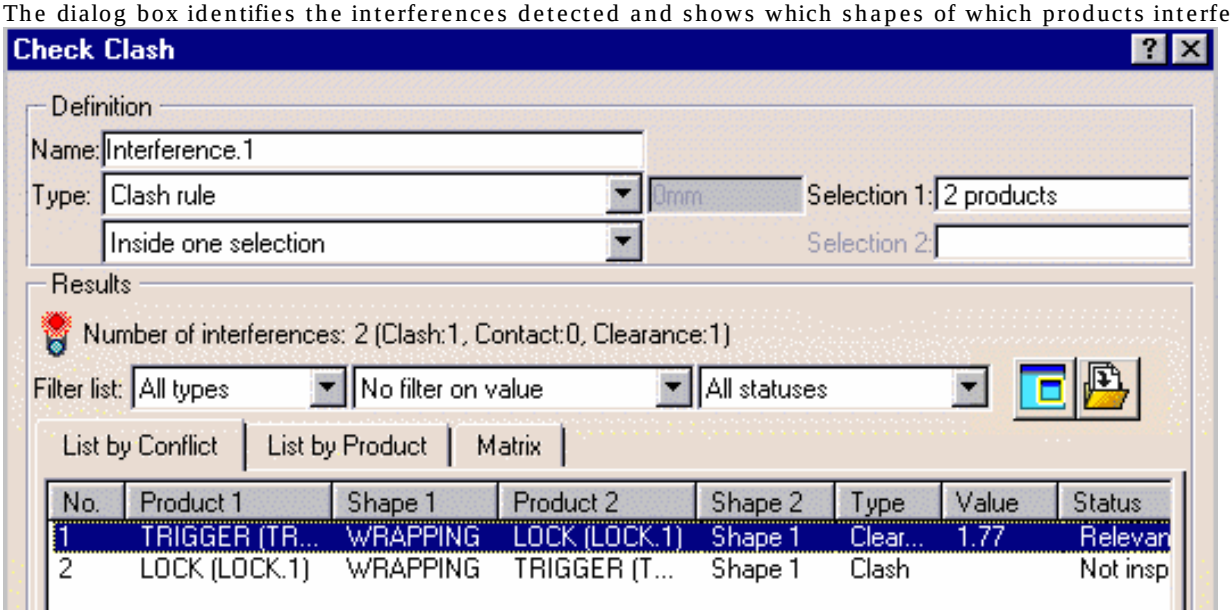
15. Activate the first Type drop-down list box and select Clash rule.
16. Activate the second Type drop-down list box and select the computation type.
17. Select the products to check for interference, for example the trigger and the lock.



18. Click **Apply**.

The Check Clash dialog box expands to show the results.

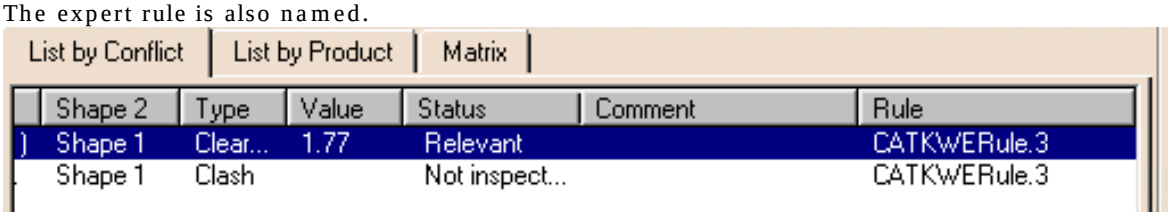
The dialog box identifies the interferences detected and shows which shapes of which products interfere.



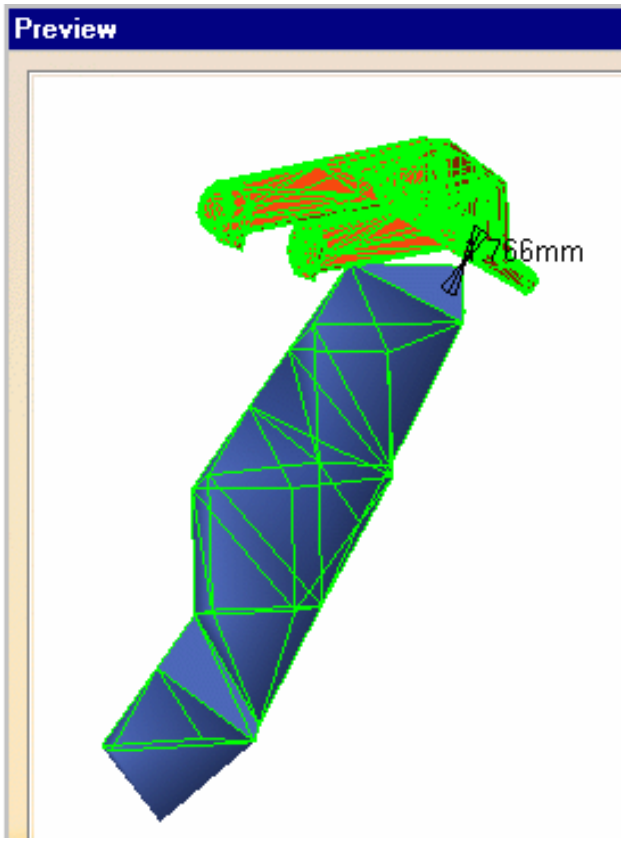
Note: Since you can only view the results of one product-shape combination at any one time, List by Product and Matrix tabs are not available.

19. Scroll to the right in the List by Conflict tab.

The expert rule is also named.



A Preview window showing shapes in conflict also appears.



For more information on:

- Managing representations, see the Product Structure User's Guide.
- the Clash command, see Interference Checking & Analysis.

Knowledgeware Rule-based Clash & ENOVIA LCA

The results of the rule-based clash can be compared to results already stored in ENOVIA LCA and new results can be saved back into the ENOVIA LCA database. Storing results in ENOVIA LCA saves both the shape and rule names.

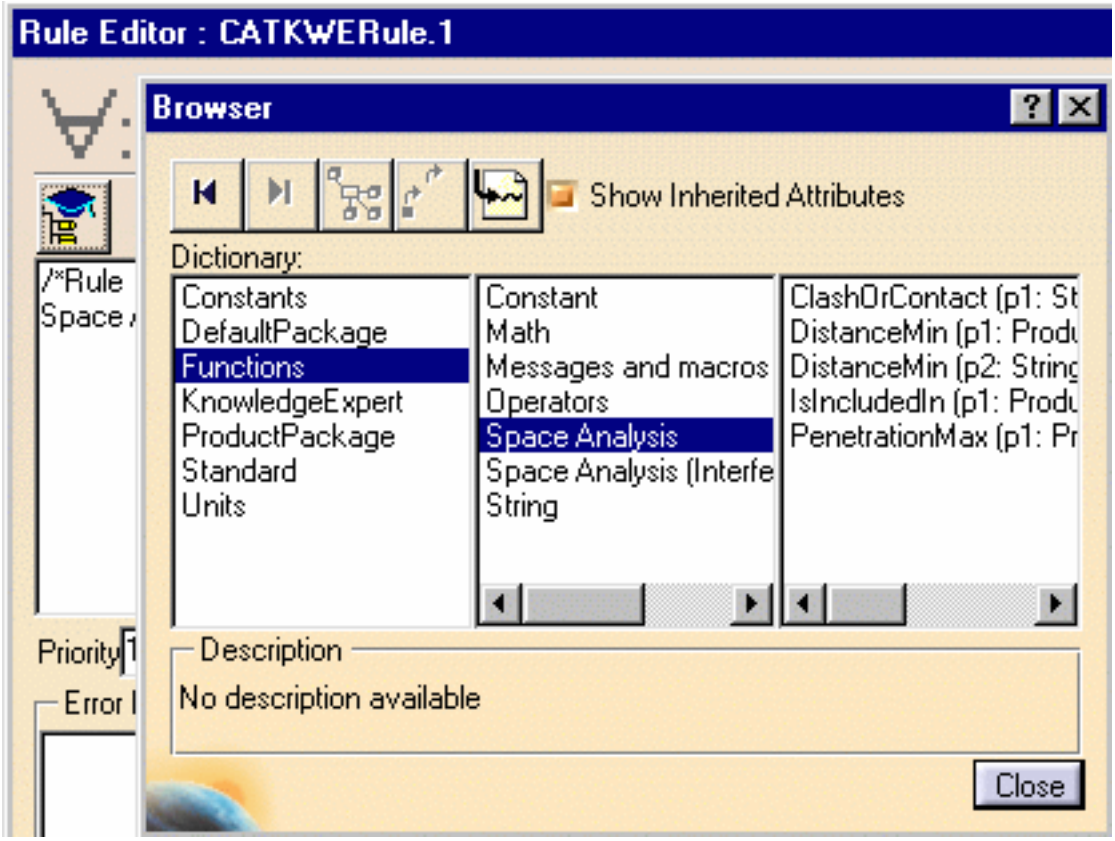
To do so, do not forget to select the Retrieve Information From PDM check box in the DMU Clash tab (**Tools -> Options..., Digital Mockup -> DMU Space Analysis**).

Space Analysis Functions in Knowledge Expert

Minimum distance, distances along X, Y and Z as well as clash, contact and clearance results can be used in checks and rules.

Five functions have been added to the dictionary:

- ClashOrContact (p1: String, Product, Product,...): Boolean
- DistanceMin (p1: Product, p2: Product): Length
- DistanceMin (p2:String,p1:Product,P2:Product,...): Length
- IsIncludedIn (p1: Product, p2: Product): Boolean
- PenetrationMax (p1: Product, p2: Product): Length



For more information, see Space Analysis under Functions in the Reference Information section of the *Knowledge Expert User's Guide*.

Working with CGRs in DMU

This section provides information on working with CGRs in a DMU context. This is the default document format used for DMU sessions when working in Visualization mode and with the cache system. This format is not suitable for design.

CGR stands for CATIA Graphical Representation. CGR files always have a .cgr extension.
All MULTICAx Plug-in products convert native part data into CGR format.

When using CGRs, a representation of the geometry only is available; the geometry is not available.

Advantages:

- Requires less memory
- Documents are lighter
- Considerably reduces the time required to load your data

For example, with the following CATProduct document, you can easily check the difference between the .Model document and the corresponding CGR file:

With **.Model:** 6,708 MB

With **CGR:** 670KB

This may be useful when dealing with complex products or assemblies involving large amounts of data.

Drawbacks:

- No design possible
- Relations between objects cannot be established because they are based on design features. For example, in DMU Kinematics, you cannot create joints because joints are built on design constraints.

In DMU:

- CGR documents can be inserted directly into DMU sessions
- They are generated automatically from exact geometry (V4 Models, CATParts...) when working with the cache system
- You can also generate CGR representations in certain DMU applications and save them as CGR files, for example DMU Optimizer

Definitions:

Cache system concept: Two different modes are available when a component (V4 model, V5 CATPart, V5 CATProduct, etc.) is inserted into a DMU Navigator CATProduct document

- Design mode
- Visualization mode

Using a cache system considerably reduces the time required to load your data. The cache system is organized into two parts:

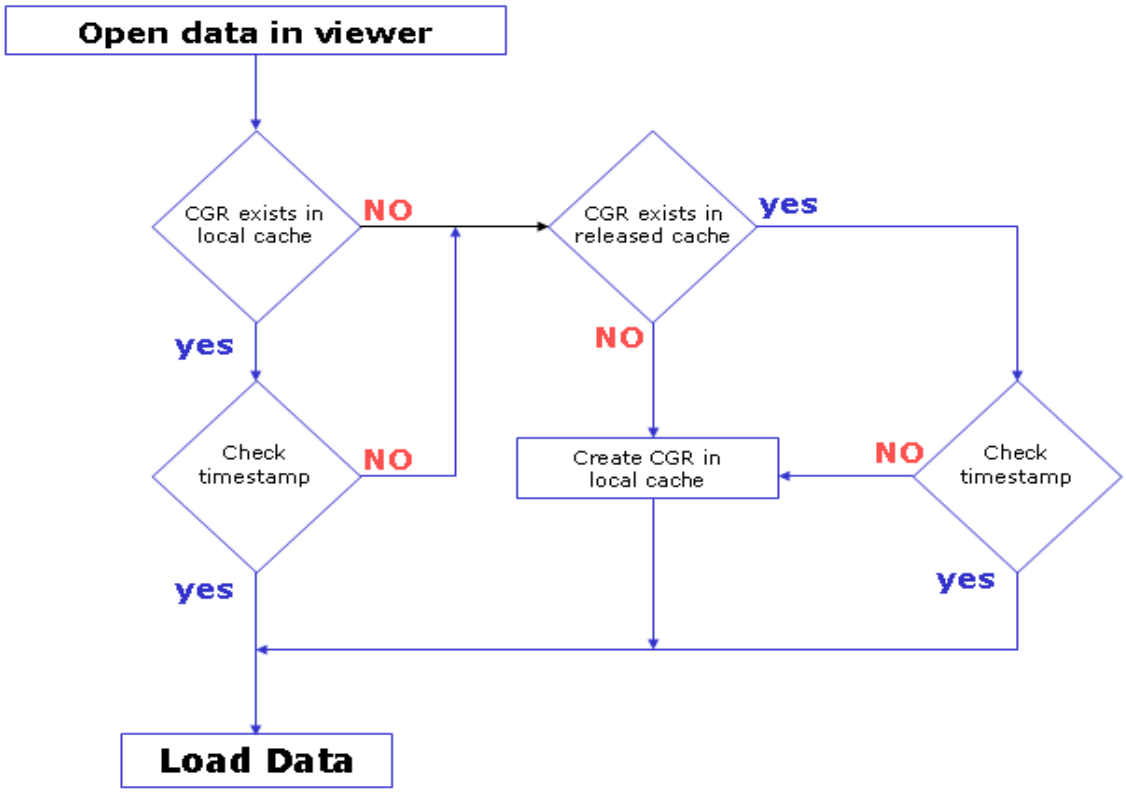
Local cache: a read/write directory located locally on your machine and used to store cgr files. The first time a component is inserted, it is tesselated. This means that the corresponding cgr file is computed and saved in the local cache as well as displayed in the document window. The next time this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

Released cache: a read-only cache that can be located anywhere on your network.
Several directories can be defined for the released cache. If a cgr file cannot be found in the local cache, the software browses the released cache directories in their listed order to see if the cgr file is located in one of them. If the cgr file is still not found, the component is tesselated and the resulting cgr file is saved in the local cache. The site administrator is normally responsible for the released cache.

A timestamp enables the verification that no modifications have been made to a document since the generation of the corresponding cgr file found in either of the above caches:

Timestamp: the date and hour at which the origin document was last modified. If you activate the **Check timestamp** button, then before a cgr file is loaded into a viewer, its timestamp will be checked to verify that no modifications have been made to the document since the generation of the cgr file. If you do not activate the **Check timestamp** button, then a cgr file of a document will be loaded without any verification of its time-wise coherence with the document.

The process used for loading data using the caches is as follows:



Please refer to Customizing Cache Settings in Customizing for DMU Navigator section (*DMU Navigator User's Guide*)

Design mode: A working mode in which the exact geometry is available and documents (V4 models, V5 CATParts, V5 CATProducts, etc.) are inserted as is.

Tessellation: The generation of a triangular mesh representation of an object from a solid or surface.

Visualization mode: A working mode in which a representation of the geometry only is available and the corresponding cgr file, if it exists, is inserted from the data cache.

Cache System & multi-processors:
You can choose the creation of cache files in multi-process for this, select the **Multi-Process Visualization Mode with local cache** checkbox in **Tools->Options->Infrastructure->Product Structure->Product visualization.**
You also need to customize multi-process settings

For more detailed information please read Customizing Product Visualization Settings and Customizing Multi-Process settings

DMU Applications

The following table lets you see at a glance what you can do with CGRs in the various DMU applications, and lists in particular any restrictions to working with CGRs in a DMU context.

Note: in DMU kinematics context, you cannot work on cgrs.

DMU Space Analysis		
Commands	**Insert CGRs and Run...**	**Save Results as CGR...**
Clash	Yes	No
Sectioning	Yes	No
Distance and band analysis	Yes	Band Analysis Results only
Compare Products	Yes	No
2D Measure	No	No
Arc through 3 points	Yes	No
Measure Between	Yes, except for measure between points (1) (2)	No
Measure Item	Yes, except for measure properties of points (1) (2)	No
Measure Thickness	Yes	No
Measure Inertia	Yes	No

(1) **Points** in edge limits, arc center and picking point selection modes are taken into account

(2) **Exact / approximate measures on CGRs:** exact measures can only be obtained on canonical elements in Measure Between and Measure Item commands; all other measures are approximate.

DMU Optimizer		
Commands	**Insert CGRs and Run...**	**Save Results as CGR...**
Silhouette	Yes	Yes
Wrapping	Yes	Yes
Thickness	Yes	Yes
Offset	Yes	Yes
Swept Volume	Yes	Yes
Free Space	Yes	Yes
Simplification	Yes	Yes
Result of a merging operation	Yes	Yes
3D Cut	Yes	Yes

DMU Navigator		
Commands	**Insert CGRs and Run...**	**Save Results as CGR...**
Annotated View	Yes	No
Camera	Yes	Not Applicable
3D Annotation	Yes	No
Hyperlink	Yes	Not Applicable
Group	Yes	Not Applicable
Scene	Yes	Yes

Symmetry	Yes	Yes
Translation or Rotation	Yes	Yes
Axis System Creation	Yes, except selecting a sub-element	Yes
Point Creation	Yes, except selecting a sub-element	Yes
Line Creation	Yes, except selecting a sub-element	Yes
Plane Creation	Yes, except selecting a sub-element	Yes
Cumulative Snap	Yes	Yes
Modify Sag	No, requires the part document	No
Spatial Query	Yes	Not Applicable
Current Selection	Yes	Not Applicable
Publish	Yes	Not Applicable
Reset Position	Yes	Yes
Init Position	Yes	Yes
Track	Yes	No
Color Action	Yes	No
Visibility Action	Yes	No
Sequence	Yes	No
Clash Detection	Yes	Not Applicable
Record Viewpoint Animation	Yes	Not Applicable

DMU Space Analysis Interoperability

In this section, you will find information on working with PDM systems and DMU Space Analysis.

Working with ENOVIA LCA

Optimal DMU PLM Usability for Space Analysis
Interference Analysis / ENOVIA LCA Interoperability

Optimal PLM Usability for DMU Space Analysis

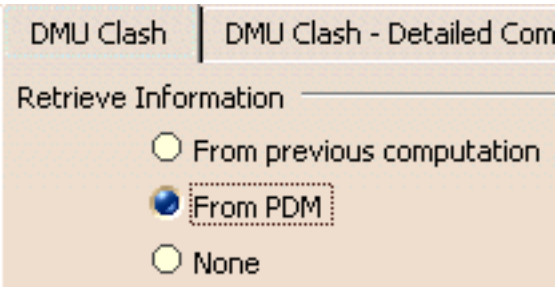
When working with ENOVIA LCA (i.e. when you open a PRC saved in Explode mode), the safe save mode warns users that DMU data, with the exception of clash results when retrieved from PDM systems, cannot be correctly saved in ENOVIA.

Since all DMU Space Analysis commands focus on analysis or review rather than design, all commands are available at all times. A warning dialog box, however, informs the user that the various items created in DMU Space Analysis (distances, sections, measures, etc.) cannot be saved in ENOVIA LCA. This dialog box appears on entering DMU Space Analysis commands. The one exception is the Clash command.

Clash Command

Clash results cannot be saved in ENOVIA LCA when the Retrieve Information option in the **DMU Clash** tab page is set to either From previous computation or None (**Tools -> Options -> Digital Mockup -> DMU Space Analysis**). A warning dialog box displays.

Only clash results created using the option Retrieve Information from PDM can be saved in ENOVIA LCA. See Retrieving Information from ENOVIA LCA.



Interference Analysis / ENOVIA LCA Interoperability

DMU users running interference analyses can take advantage of ENOVIA LCA interoperability. The table below summarizes what can be done.

You can...
Retrieve information from ENOVIA LCA
Save clash results in ENOVIA LCA
Work with clash rules
Send the clash result and import products involved from ENOVIA LCA
Batch process interference analysis on ENOVIA LCA data

Retrieving Information from ENOVIA LCA

You can compare clash command results with those stored in ENOVIA LCA and new results can be saved back into the ENOVIA database.

- Retrieving information from ENOVIA LCA

For more information on ENOVIA LCA capabilities, see LCA Product Interference Management.

Saving Clash Results in ENOVIA LCA

- Saving clash results in ENOVIA LCA

Working with Clash Rules and ENOVIA LCA

- Knowledgeware rule-based clash in the user task section of this guide.

Sending Clash Result from ENOVIA LCA

The user can send the clash object directly from the ENOVIA query panel to CATIA. The clash result is re-created in the CATIA session. To import the products involved and visualize the clash, simply click the appropriate interference line in the Check Clash dialog box.

Batch Processing Interference Analysis

The batch ITFCHECK must be run from your ENOVIA LCA session. For more information, see the LCA Enterprise Architecture Installation Guide.

Retrieving Information from ENOVIA LCA

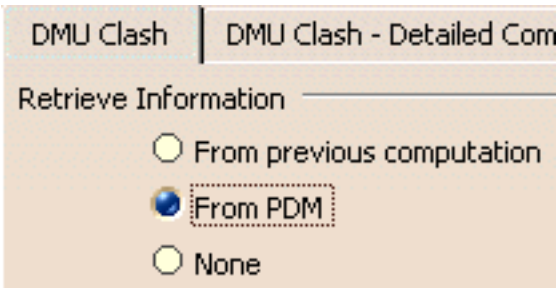
This task explains how to work with the Clash command and ENOVIA LCA. You can:

- Retrieve clash results from ENOVIA LCA
- Compare new and old clash results
- Save clash results in ENOVIA LCA.

Have a DMU V5 session and an ENOVIA LCA session running

Customizing Clash Settings in DMU V5

1. In DMU V5, set the clash settings in the **DMU Clash** tab to retrieve information from PDM (**Tools -> Options..., Digital Mockup -> DMU Space Analysis**).



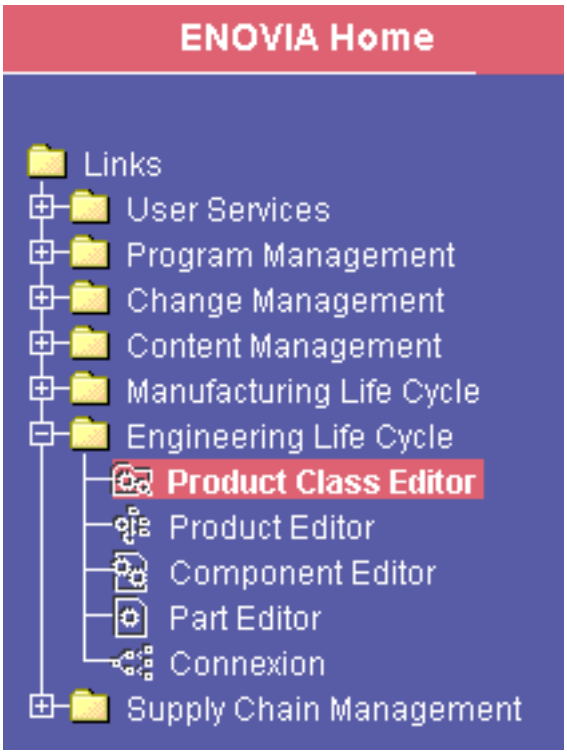
Establishing the DMU V5 / ENOVIA LCA Connection from DMU V5

2. Still in DMU V5, select **Infrastructure -> Product Structure** from the **Start** menu to launch the Product Structure workbench and establish the connection between DMU V5 and ENOVIA LCA.
3. Display the ENOVIA toolbar by clicking **Toolbars -> ENOVIA LCA** from the **View** menu.

4. Click the Connect to ENOVIA LCA icon.

Loading Models from ENOVIA LCA

5. In ENOVIA Home panel, select the Engineering Life Cycle folder and double-click the Product Class Editor bookmark.



The Product Class View panel is displayed.

6. Right-click a Product Class Root (PRC) and select **Open** from the contextual menu.

The PRC is opened in the Product Editor panel.

To create a PRC, please refer to the appropriate ENOVIA LCA documentation.

7. Right the PRC and select **Expand** from the contextual menu to expand the product.

All instances are displayed.

8. Right-click the instance of your choice and select **Send To -> CATIA V5** from the contextual menu.

The document is displayed in your DMU V5 session.

Detecting Interferences & Comparing Results

9. In DMU V5, switch to the DMU Space Analysis workbench (**Start -> Digital Mockup -> DMU Space Analysis**).

10. Select **Insert -> Clash** from the menu bar or click the Clash  icon in the DMU Space Analysis toolbar and run an interference check.

    The Check Clash dialog box expands to show the results.

    The dialog box identifies the number of interferences detected along with the type. By default, results are organized by conflict in the List by Conflict tab.

    Additional information is provided in the Check Clash dialog box to help you compare clash results with those already stored in ENOVIA LCA.

Saving Clash Results in ENOVIA LCA

11. Identify the results you want to store in ENOVIA LCA in the Keep column of the Check Clash dialog box.
12. Click **OK** when done in the Check Clash dialog box.

    A dialog box appears asking you to confirm that you want to save results in ENOVIA LCA.
13. Click **OK** in the dialog box to confirm your save.

    Clash results are written to ENOVIA LCA and a dialog box informs you whether or not the operation was successful.

- Using ENOVIA LCA search capabilities, check that clash results have indeed been written to the database.
- Don't forget to commit changes to the ENOVIA LCA database before exiting your ENOVIA LCA pplication.

ENOVIAVPM / Interference Analysis

DMU users running interference analyses can take advantage of ENOVIAVPM interoperability. The table below summarizes what can be done.

You can...
Retrieve information from ENOVIAVPM
Save clash results in ENOVIAVPM
Batch process interference analysis on ENOVIAVPM data

Retrieving Information from ENOVIAVPM

You can compare clash command results with those stored in ENOVIA VPM and new results can be saved back into the ENOVIA database.

- Retrieving information from ENOVIAVPM

For more information on ENOVIAVPM capabilities, see the VPM Installation & Administration Guide.

Saving Clash Results in ENOVIAVPM

- Saving clash results in ENOVIAVPM
- Saving DMU applicative data in ENOVIAVPM in the DMU Navigator User's Guide.

Batch Processing Interference Analysis

The batch ITFCHECK must be run from your ENOVIAVPM session. For more information, see the VPM Installation & Administration Guide.

Retrieving Information from ENOVIAVPM

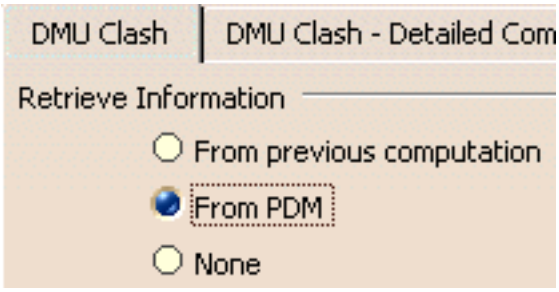
This task explains how to work with the Clash command and ENOVIAVPM. You can:

- Retrieve clash results from ENOVIAVPM
- Compare new and old clash results
- Save clash results in ENOVIAVPM.

Have a DMU V5 session and an ENOVIAVPM session running

Customizing Clash Settings in DMU V5

1. In DMU V5, set the clash settings in the **DMU Clash** tab to retrieve information from PDM (**Tools -> Options..., Digital Mockup -> DMU Space Analysis**).



2. Ensure that the Penetration depth or minimum distance option (Numeric Result) is selected in the **DMU Clash - Detailed Computation** tab.

   If this is not the case, a dialog box displays when running your interference check.

Establishing the DMU V5 / VPM Connection from DMU V5

3. Refer to Saving DMU Applicative Data in ENOVIAVPM in the DMU Navigator User's Guide for information on establishing the connection between ENOVIAVPM and DMU V5.

Loading Models from VPM

4. In ENOVIAVPM product structure graph, select a product and send it to your DMU V5 session.

Detecting Interferences & Comparing Results

5. In DMU V5, switch to the DMU Space Analysis workbench (**Start -> Digital Mockup -> DMU Space Analysis**).

6. Select **Insert -> Clash** from the menu bar or click the Clash  icon in the DMU Space Analysis toolbar and run an interference check.

   The Check Clash dialog box expands to show the results.

   The dialog box identifies the number of interferences detected along with the type. By default, results are organized by conflict in the List by Conflict tab.

   Additional information is provided in the Check Clash dialog box to help you compare clash results with those already stored in ENOVIAVPM.

Saving Clash Results in VPM

7. Identify the results you want to store in ENOVIAVPM in the Keep column of the Check Clash dialog box.
8. Click **OK** when done in the Check Clash dialog box.

   A dialog box appears asking you to confirm that you want to save results in VPM.

9. Click **OK** in the dialog box to confirm your save.

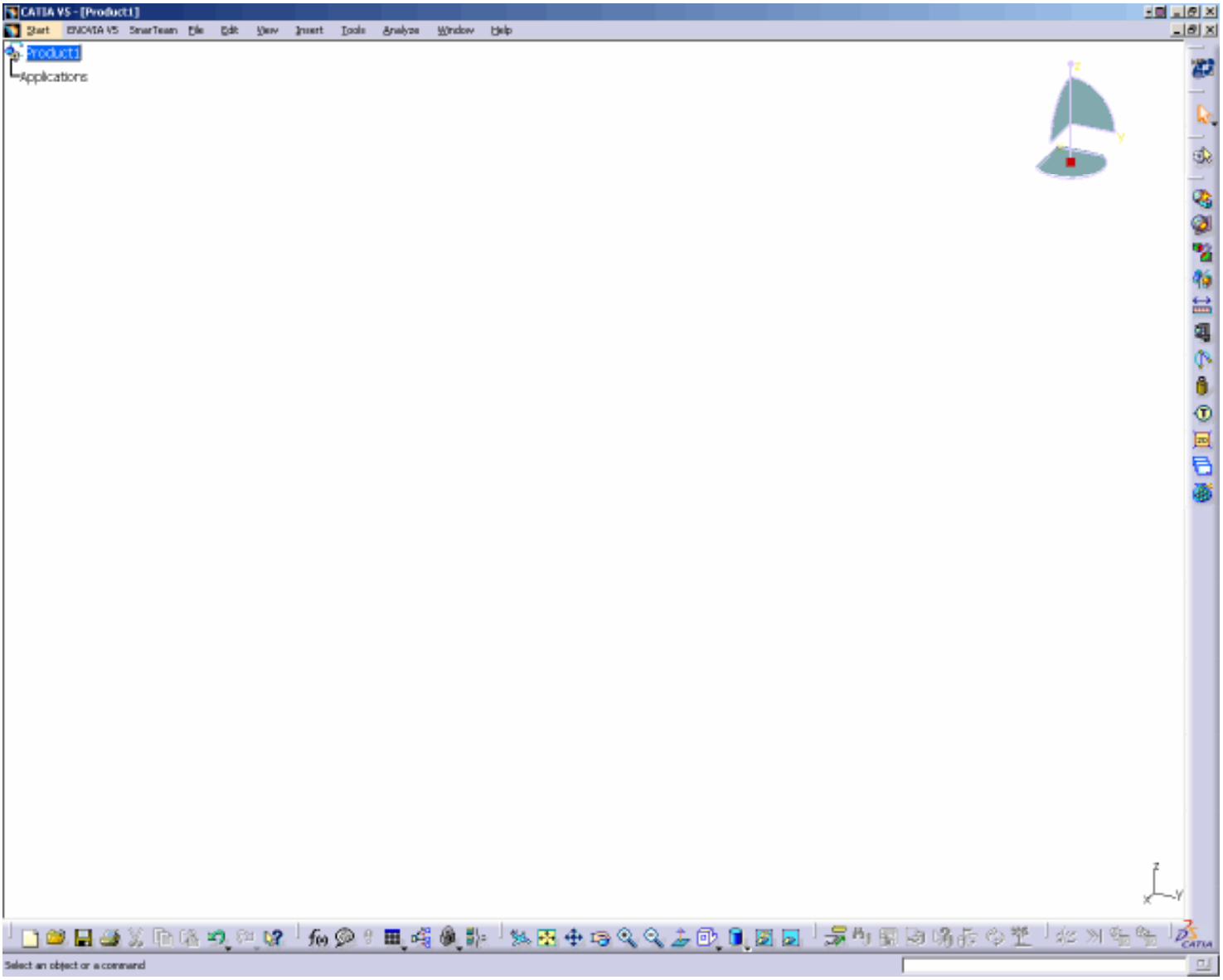
   Clash results are written to ENOVIAVPM and a dialog box informs you whether or not the operation was successful.

- Using ENOVIAVPM query capabilities (Interferences Management panel), check that clash results have indeed been written to the PSN graph.
- Don't forget to commit changes in the PSN graph to the ENOVIAVPM database before exiting your VPM application.

Workbench Description

The DMU Space Analysis Version 5 application window looks like this:

Click the hotspots to see related documentation.



Menu Bar
Toolbar
Specification Tree

DMU Space Analysis Menu Bar

This section presents the menu bar tools and commands dedicated to DMU Space Analysis.

Start File Edit View Insert Tools Analyze Window Help

Insert

For...	See...
Clash	Detecting Interferences Using the Clash Command
	Reading Clash Command Results
	Finer Analysis of Clash Command Results
Sectioning	Sectioning
Distance and Band Analysis	Distance and Band Analysis
3D Annotation	Adding 3D Annotations in the *DMU Navigator's User Guide*
Group	Defining Groups

Tools

For...	See...
Macro	See the *DMU Space Analysis Journaling Guide*
Options...	Customizing
Publish	Publishing (for clash command results)

Modify SAG

Modifying the Sag value in Visualization Mode in the *DMU Navigator User's Guide*

Analyze

For...	See...
Measure Between	Measuring Minimum Distances & Angles between Geometrical Entities or Points
Measure Item	Measuring Properties
Arc through Three Points	Measuring Arc Sections.
Part to Part Clash	Analyzing Part to Part Clashes
Measure Inertia	Measuring Inertia
Graphic Messages	Using the Section Viewer
Current Selection Panel	Viewing the Current Selection in the *DMU Navigator User's Guide*

DMU Space Analysis Toolbar

 See Analyzing Interferences

 See Sectioning

 See Distance and Band Analysis

 See Comparing Products

 See Measuring Minimum Distances & Angles between Geometrical Entities or Points.

 See Measuring Properties

 See Measuring Arc Sections .

 See Measuring Inertia

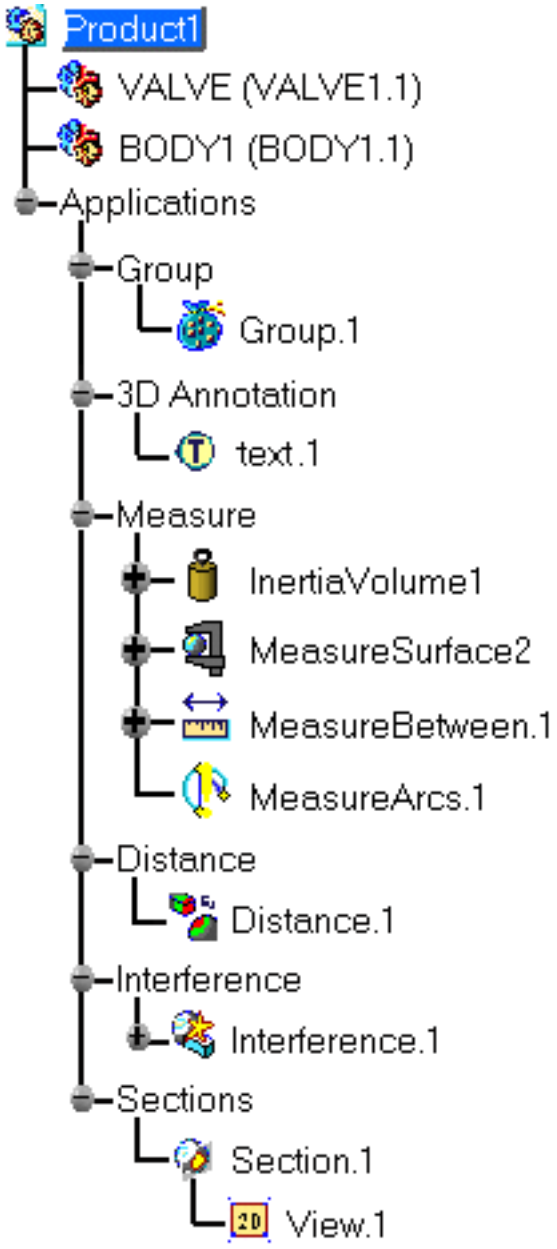
 See Adding 3D Annotations in the *DMU Navigator User's Guide*

 See Creating Annotated Views in the *DMU Navigator User's Guide*

 See Managing Annotated Views in the *DMU Navigator User's Guide*

 See Defining Groups

Specification Tree



Icons displayed in the specification tree and specific to the DMU Space Analysis workbench identify:

- Groups
- 3D annotations
- Measures made using the Measure Inertia command
- Measures made using the Measure Item command


Measure Item measures that require updating.


Measure Item measures that cannot be updated. This applies to measures made in Visualization mode or on cgr files as well as those made prior to Version 5 Release 6.


Measures made using the Measure Between command


Measure Between measures that require updating.


Measure Between measures that cannot be updated. This applies to measures made in Visualization mode or on cgr files as well as those made prior to Version 5 Release 6.


Measures made using the Arc Through Three Points command


Distance and band analysis entries


Clash entries


Sectioning entries


Section results that require updating.


Section results that have been frozen and cannot be updated unless the Freeze option is de-activated.


2D section views

For standard specification tree symbols, see *Specification Tree Symbols* in the *Infrastructure User's Guide*.

Customizing

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

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

   The Options dialog box appears.

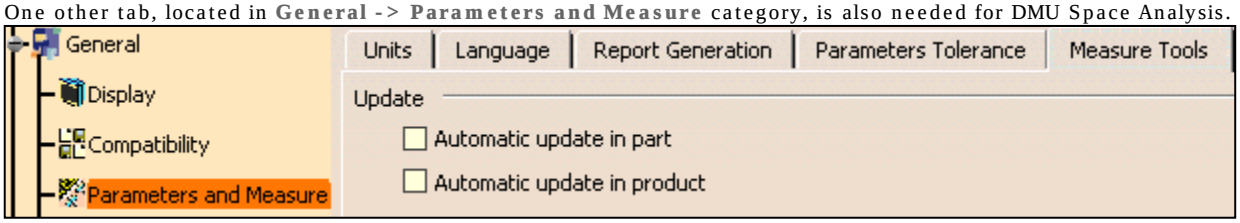
2. Select the **Digital Mockup** category in the left-hand box.

3. Select the **DMU Space Analysis** sub-category.

   Various DMU Clash tabs, the DMU Sectioning and the DMU Distance tabs appear.

   - DMU Clash lets you customize basic clash settings.
   - DMU Clash - Detailed Computation lets you define what is computed when you select a conflict or product in the Check Clash Results dialog box.
   - DMU Clash - Penetration lets you define how penetration depth is computed.
   - DMU Clash - Rule lets you identify the path to the CATProduct containing knowledgeware clash rules.
   - DMU Clash - Process lets you customize clash process XML settings.
   - DMU Sectioning lets you customize section plane, grid and section viewer settings.
   - DMU Distance lets you customize results window and publish settings of the **Distance and Band Analysis** command.

   One other tab, located in **General -> Parameters and Measure** category, is also needed for DMU Space Analysis.

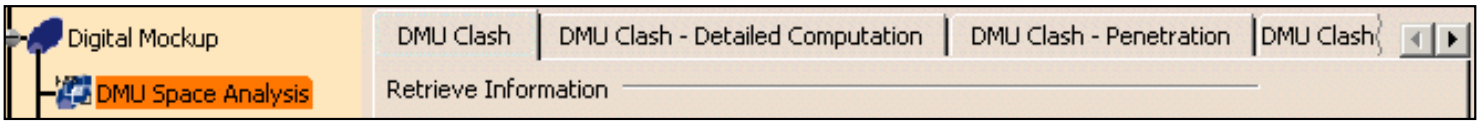


   - Measure tools lets you manage the update, graphic properties and save density settings of measure commands.

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

5. Click **OK** in the Options dialog box when done.

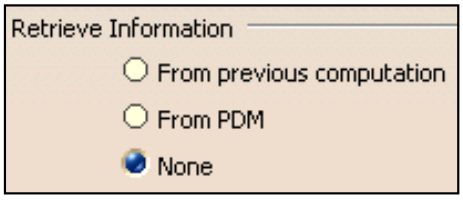
DMU Clash



The DMU Clash tab contains five categories of options:

- Retrieve information
- Results window
- Display in results box
- Type of computation
- During initial computation

Retrieve Information

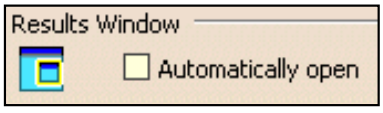


Select the option defining whether or not, and where, to retrieve clash results for comparison purposes:

- **From previous computation**
- **From PDM:** from either Enovia V5 or Enovia VPM.
  Note: This setting cannot be combined with the authorized penetration interference type.
- **None**: No comparison will be done.

By default, no information is retrieved.

Results Window

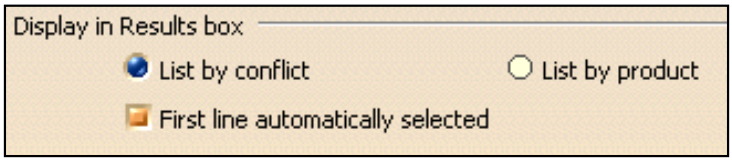


Automatically open

Selecting this checkbox automatically displays the results window when you run a clash command computation.

By default, this option is cleared.

Display in Results Box



List by conflict / List by product

Select the option specifying the default display in the results box of the Check Clash dialog box: **List by conflict** or **List by product**.

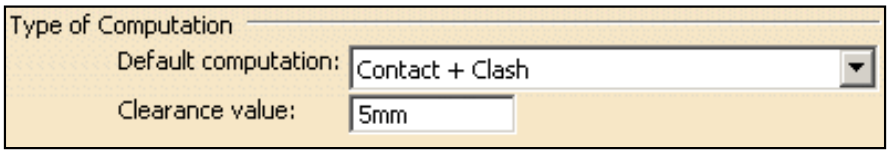
By default, the default display is set to **List by conflict**.

First line automatically selected

Selecting this checkbox automatically selects the first line in the List by Conflict or List by Product tab and computes the graphics representation of the selected interference as well as the penetration depth or minimum distance.

By default, this option is selected.

Type of Computation



Default computation

Select the default interference type that will display in the Check Clash dialog box from the drop-down list box. Two types are available:

- Contact + Clash: checks whether two products occupy the same space zone as well as whether two products are in contact.
- Clearance + Contact + Clash: In addition to the above, checks whether two products are separated by less than the pre-defined clearance distance.

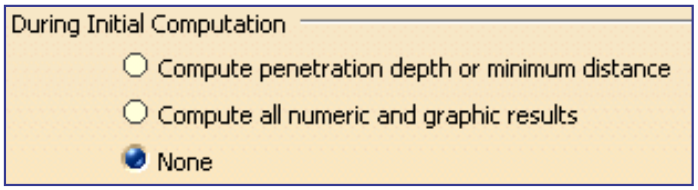
By default, default computation is set to Contact + Clash.

Clearance value

Enter the default clearance value that will display in the Check Clash dialog box when the Clearance + Contact + Clash interference type is selected.

By default, the clearance value is set to 5mm.

During Initial Computation

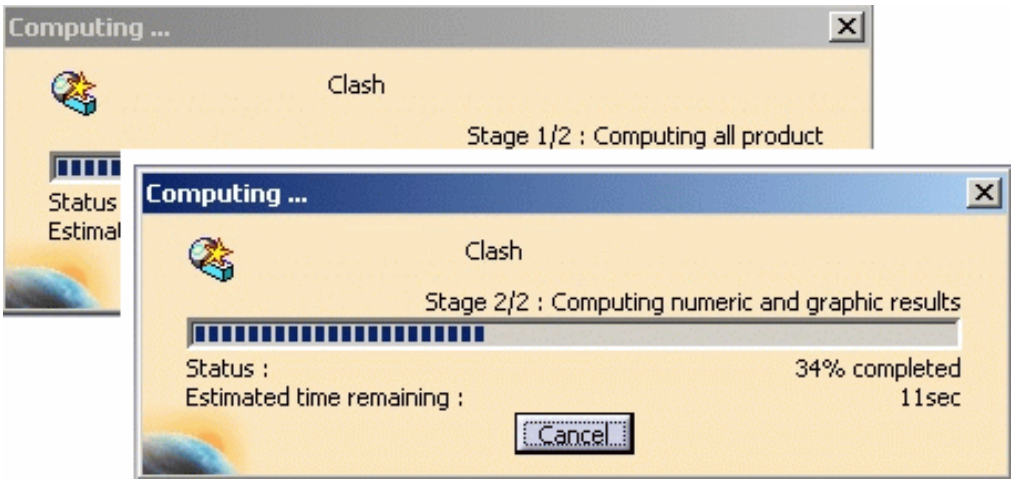


Select the option specifying what the system computes in addition to detecting and identifying the different types of interference. Three options are available:

- **Compute penetration depth or minimum distance**: if selected, automatically computes and displays the penetration depth and minimum distance for all interferences detected.
- **Compute all numeric and graphic results**: if selected, automatically computes and displays numeric results as well as graphic results as defined in the Detailed Computation tab.

Note: Computation time can be long.
In this case, the progress bar shows the status of the calculation in two different steps:

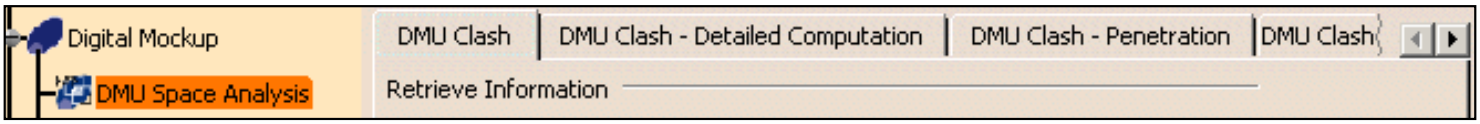
Stage 1/2: determines the number of interferences
Stage 2/2: computes appropriate numeric and graphic results.



- **None**: if selected, the initial computation detects and identifies the different types of interference only.

By default, this option is set to **None**.

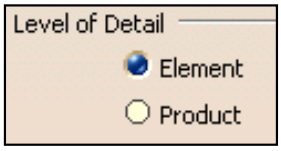
DMU Clash - Detailed Computation



The DMU Clash - Detailed Computation tab contains give categories of options:

- Level of detail
- Clash result
- Contact & clearance result
- Numeric result
- Intersection volume

Level of Detail



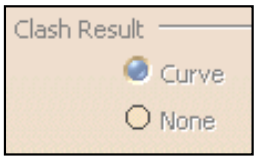
Select the option specifying at what level of detail the detailed computation is run:

- **Element**: lets you work globally at product level while allowing you to pinpoint the elements involved.
- **Product**: gives information at product level only.

Note: Element analysis in Detailed Results and Visualization boxes is only available if the level of detail is set to Element.

By default, the level of detail is set to **Element**.

Clash Result

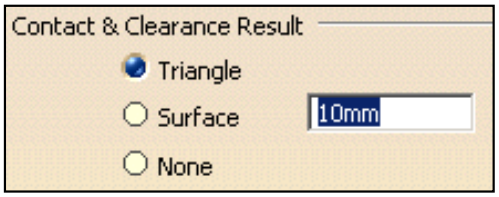


Select the option specifying the graphics representation for clashes.

- **Curve**: Red intersection curves identifying clashing products are computed and displayed.
- **None**: No intersection curves are computed.

By default, this option is set to **Curve**.

Contact & Clearance Result



Select the option specifying the graphics representation for contacts and clearances.

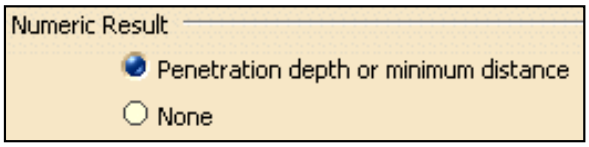
- **Triangle**: Yellow and green triangles identifying products in contact and products separated by less than the specified clearance distance respectively are computed and displayed.

  Note: The graphics display and storage of the triangular representation is costly and may adversely affect performance.
- **Surface**: Yellow and green surfaces identifying products in contact and products separated by less than the specified clearance distance respectively are computed and displayed.

  If you select this option, you must set an accuracy. The value entered defines the maximum value for the length of the longest side of a triangular representation of the results. This representation is used to obtain the yellow and green surfaces.
- **None**: No triangles or surfaces are computed.

By default, this option is set to **Triangle**.

Numeric Result

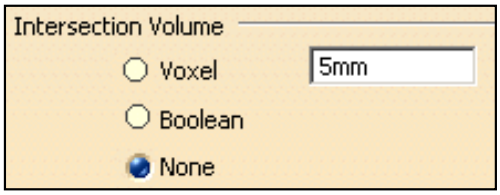


Select the option specifying whether or not a numeric result will be computed and displayed.

- **Penetration depth or minimum distance**: The penetration depth of clashes and the minimum distance for clearances is computed and displayed.
- **None**: No penetration depth or minimum distance is computed.

By default, this option is set to **Penetration depth or minimum distance**.

Intersection Volume



Select the option specifying the graphics representation for clashes. This representation can be requested in addition to that selected under Clash Result.

- **Voxel**: Voxels (cubes) identifying clashing products are computed and displayed.

If you check this option, you must set the voxel size. The value entered is the edge-length of the voxel or cube. A lower value will give a more accurate result but will also require a longer computation time and more memory. The minimum value is 1mm (0.03937 inch).

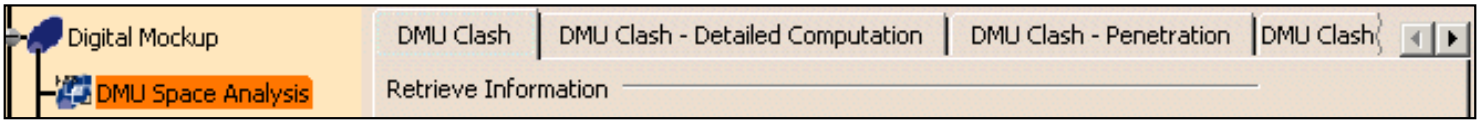
- **Boolean**: Boolean solids identifying clashing products are computed and displayed.
- **None**: No intersection volume is computed.

Notes:

- The intersection volume is only visible in the Results window.
- The value of the intersection volume is also given in the Detailed Results box of the Check Clash dialog box.
- The intersection volume is not stored in the model and cannot be saved in ENOVIA LCA or ENOVIA VPM.

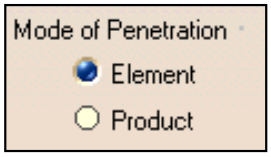
By default, this option is set to **None.**

DMU Clash - Penetration



The DMU Clash - Penetration tab contains only one category of options: Mode of Penetration.

Mode of Penetration



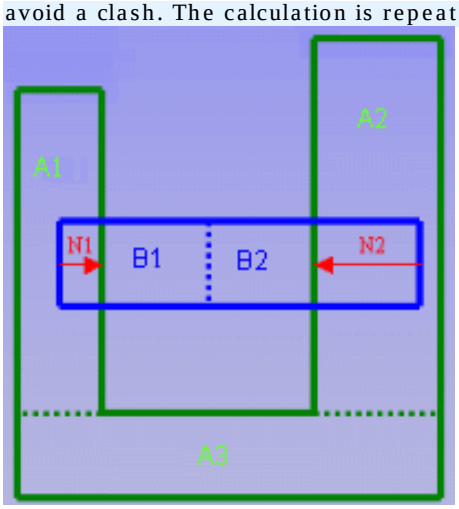
Select the option defining how penetration depth is computed. Two modes are available:

- Element
- Product

Element

This mode lets you assess the seriousness of a clash.

In this mode, penetration depth corresponds to the minimum distance by which it is necessary to translate an element to avoid a clash. The calculation is repeated for each pair of clashing elements and the maximum penetration depth output.

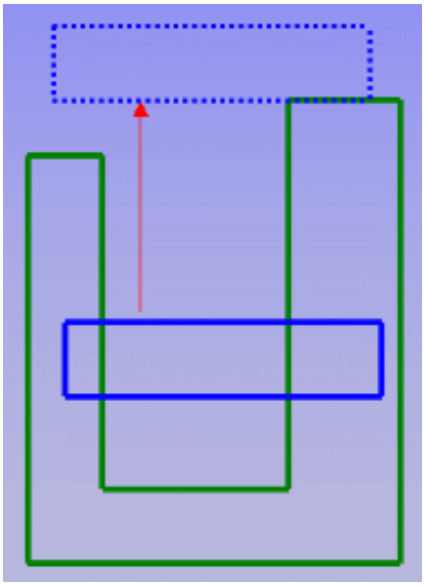


In our example, the green product (A) comprises three elements and the blue product (B) two elements.

Two different pairs of elements are clashing: A1 and B1; A2 and B2. The penetration depth for the first pair of elements is N1; for the second pair N2. The largest value is output, in our case, N2.

Note: You cannot use this calculation to translate products and avoid clashes.

Product

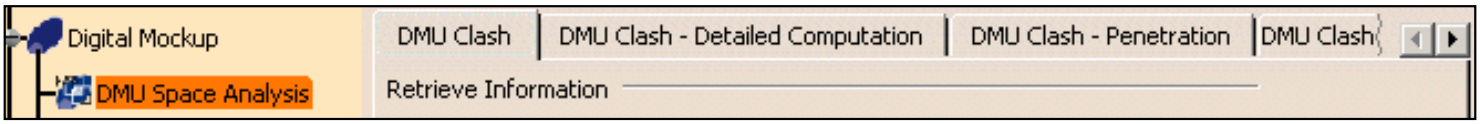


In the Product mode, penetration depth (visualized by a vector) is the minimum distance by which it is necessary to translate a product to avoid a clash.

This mode corresponds to the penetration depth computation prior to Version 5 Release 8.

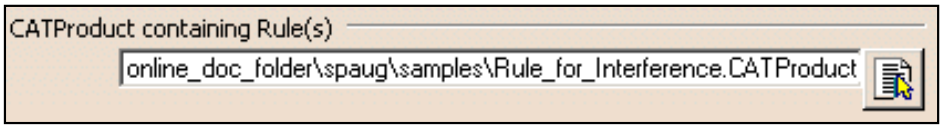
By default, the mode is set to **Element**.

DMU Clash - Rule



The DMU Clash - Rule tab contains only one category of options: CATProduct containing Rule(s).

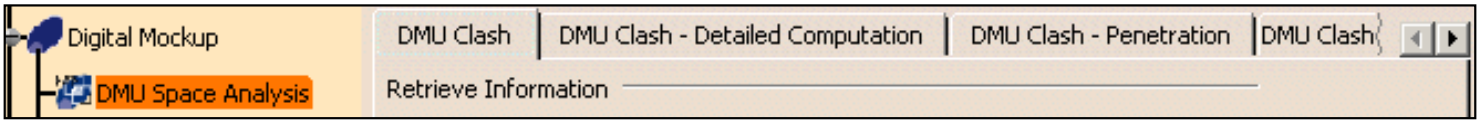
CATProduct containing Rule(s)



Enter the full path to the CATProduct containing knowledgeware clash rules, or select this path using the icon.

By default, this field is left blank.

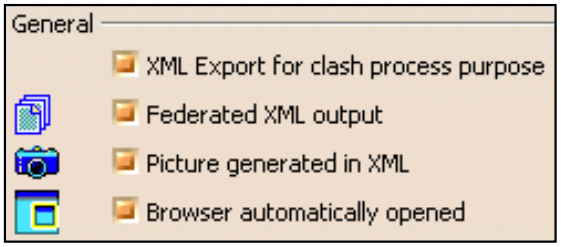
DMU Clash - Process



The DMU Clash - Process tab contains two categories:

- General
- Default XML paths

General



XML Export for clash process purpose

Selecting this checkbox exports clash results and geometry to XML files as part of a standalone clash process.

By default, this option is selected.

Federated XML output

Selecting this checkbox generates one file per conflict plus one header file summarizing results. If you de-activate this option, a single large XML file containing all conflicts is generated.

By default, this option is selected.

Picture generated in XML

Selecting this checkbox generates pictures in png format in a picture folder. This option has an obvious impact on performance.

By default, this option is selected.

Browser automatically opened

Selecting this checkbox automatically opens a browser displaying results when clash results are exported.

By default, this option is selected.

Default XML Paths

Data base repository

Enter the path giving the default location in which to save the XML files, or select this location using Browse...

Note: If you set the DLName document environment (**Tools -> Options -> General -> Document**) as your current environment, clicking Browse... will open the DLName dialog box instead of the usual Directory Browser dialog box.

The DLName dialog box displays the list of DLNames that have been defined when you set your document environment. Simply select the DLName, then click **OK** to validate.

By default, this field is left blank.

Style sheet location

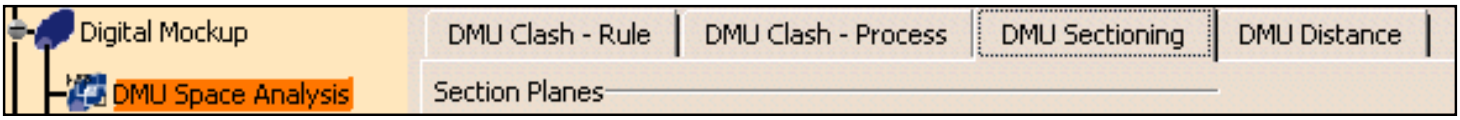
Enter the path identifying your customized style sheet, or select it using Browse...

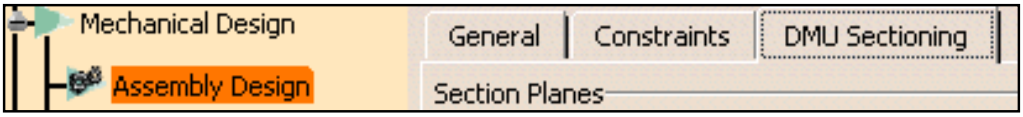
Note: If you set the DLName document environment (**Tools -> Options -> General -> Document**) as your current environment, clicking Browse... will open the DLName dialog box instead of the usual Style Sheet Selector dialog box.

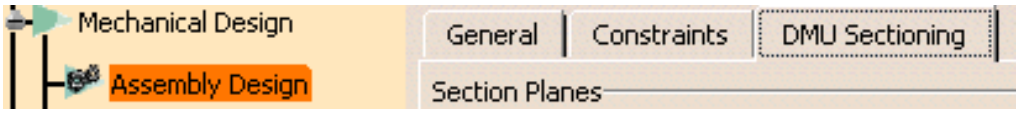
The DLName document environment lets you restrict the access to specific folders referenced by logical names referred to as 'DLNames'. Each folder is assigned a logical name. In this mode, you can only access documents in folders referenced by DLNames.

By default, this field is left blank.

DMU Sectioning



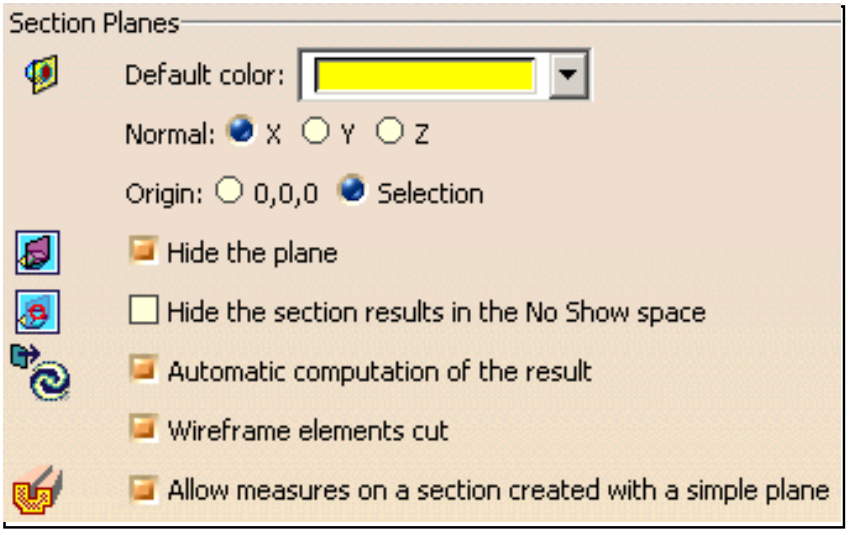




The DMU Sectioning tab contains three categories of options:

- Section planes
- Section grid
- Results window

Section Planes



Default color

Use the color chooser to define the default color of section planes.

By default, color is as shown above.

Normal X, Y, Z

Select the option specifying the absolute axis along which you want to orient the normal vector of the section plane (master plane in the case of slices and boxes).

By default, the normal vector is oriented along X.

Origin

Select the option locating the center of the plane:

- **0,0,0**: at absolute coordinates 0,0,0
- **Selection**: at the center of the bounding sphere around the products in the selection you defined.

By default, the origin option is set to **Selection**.

Hide the plane

Selecting this checkbox hides the plane on exiting the command. If cleared, the plane is kept in the Show space on exiting the command.

By default, this option is selected.

Hide the section results in the No Show space

Selecting this checkbox transfers the section results to the No Show place on exiting the command.

By default, this option is cleared.

Automatic computation of the result

Selecting this checkbox automatically updates sectioning results while manipulating the plane. If cleared, sectioning results are computed when you release the mouse button.

By default, this option is selected.

Wireframe elements cut

Selecting this checkbox takes wireframe elements into account and the section plane sections any wireframe elements present. Points represent the intersection of the plane with wireframe elements.
If cleared, wireframe elements are not taken into account.

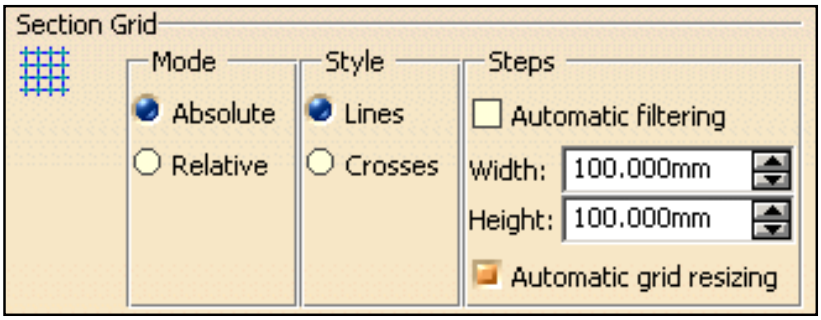
By default, this option is selected.

Allow measures on a section created with a simple plane

Selecting this checkbox gives the same 3D section cut display for a plane as in the case of a slice or box and lets you make measures on the wireframe section cut.

By default, this option is checked.

Section Grid



Mode

Select the option to locate the grid:

- **Absolute**: sets grid coordinates with respect to the absolute axis system of the document.
- **Relative**: places the center of the grid on the center of the section plane (master plane in the case of slices and boxes).

By default, the mode is set to **Absolute**.

Style

Select the option defining how the grid is represented: **Lines** or **Crosses**.

By default, the style is set to **Lines** .

Steps - Automatic filtering

Selecting this checkbox automatically adjusts the level of detail of the grid display when you zoom in and out.

By default, this option is cleared.

Steps - Width, Height

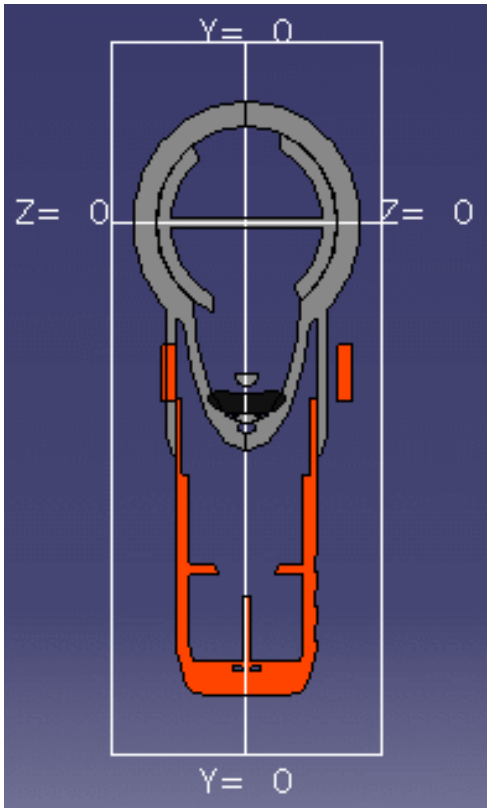
In the **Width** and **Height** boxes, type or select a new value to specify the spacing between grid lines. Units are current units set using **Tools -> Options.**

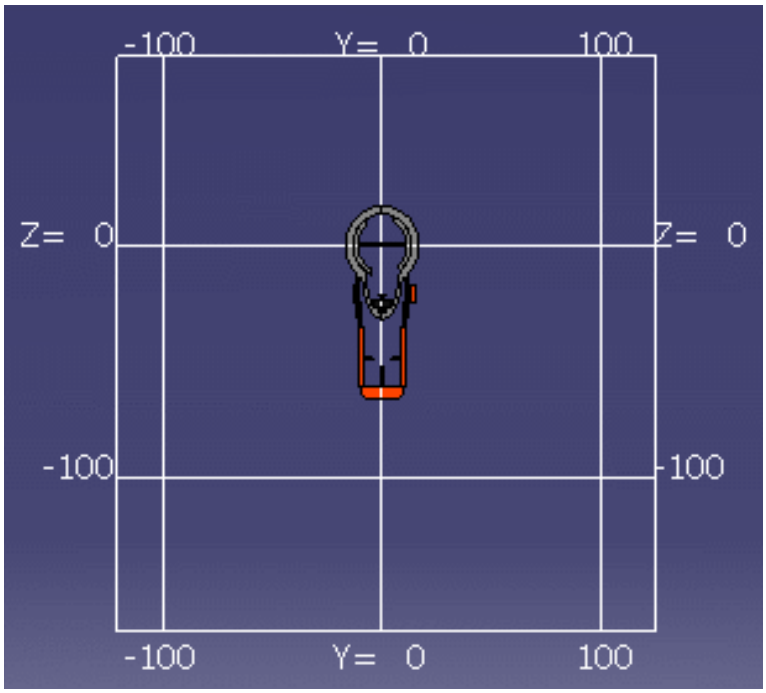
By default, width and height are set to 100.

Steps - Automatic grid resizing

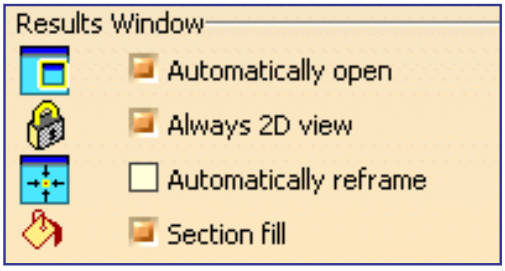
Selecting this checkbox automatically re-sizes the grid to section results when moving the section plane. If cleared, the grid has the same dimensions as the section plane.

By default, this option is selected.
Automatic grid resizing ON: Automatic grid resizing OFF:





Results Window



Automatically open

Selecting this checkbox always displays the Section viewer when in the Sectioning command.
Note: If cleared, the preview window is displayed.
If, in addition, you clear the Sectioning preview option in the Digital Mockup General tab (**Tools -> Options -> Digital Mockup -> General**), then no viewers are displayed.

By default, this option is selected.

Always 2D view

Selecting this checkbox locks the Section viewer in a 2D view. If cleared, you can work in a 3D view.

By default, this option is selected.

Automatically reframe

Selecting this checkbox automatically fits the results into the available space in both the Section viewer and preview window when manipulating the section plane in the document window.

By default, this option is cleared.

Section fill

Selecting this checkbox fills in the section to generate a surface for measurement and display purposes.

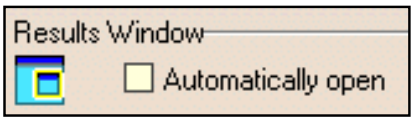
By default, this option is selected.

DMU Distance

The DMU Distance tab contains two categories of options:

- Results window
- XML publish default directory

Results Window

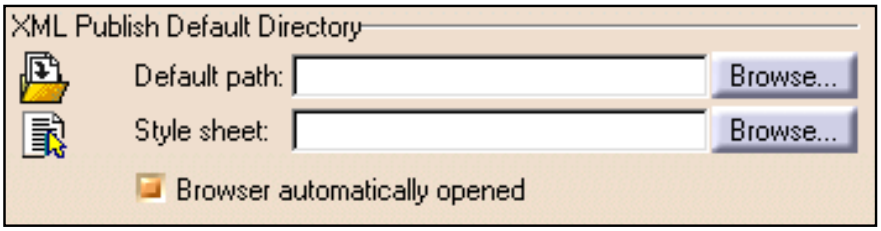


Automatically open

Selecting this checkbox automatically displays the results window when you run a distance command computation. If cleared, the Preview window displays.

By default, this option is cleared.

XML Publish Default Directory



Default path

Enter the path giving the default location in which to save the XML file, or select this location using Browse...

Note: If you set the DLName document environment (**Tools -> Options -> General -> Document**) as your current environment, clicking Browse... will open the DLName dialog box instead of the usual Directory Browser dialog box.

The DLName dialog box displays the list of DLNames that have been defined when you set your document environment. Simply select the DLName, then click **OK** to validate.

By default, this field is left blank.

Style sheet

Enter the path identifying your customized style sheet, or select it using Browse...

Note: If you set the DLName document environment (**Tools -> Options -> General -> Document**) as your current environment, clicking Browse... will open the DLName dialog box instead of the usual Style Sheet Selector dialog box.

The DLName document environment lets you restrict the access to specific folders referenced by logical names referred to as 'DLNames'. Each folder is assigned a logical name. In this mode, you can only access documents in folders referenced by DLNames.

By default, this field is left blank.

Browser automatically opened

Selecting this checkbox automatically opens a browser displaying results when distance results are exported.

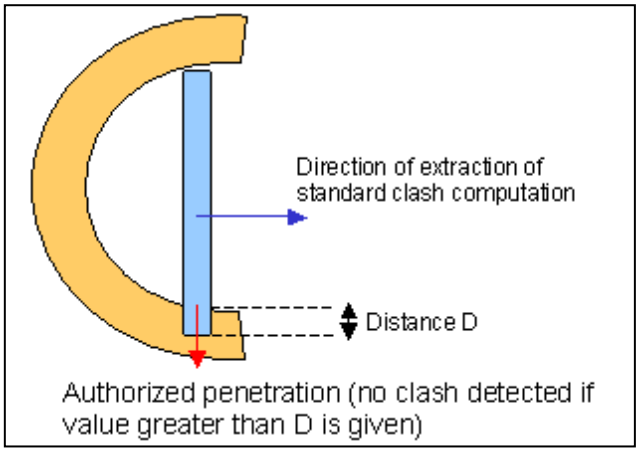
By default, this option is selected.

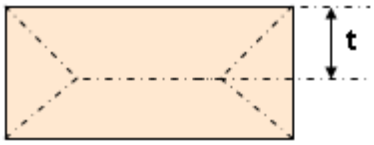
Glossary

A B C D E G I M N P S T V X

A

authorized penetration An interference type that lets you define a margin within which two products can occupy the same space zone without generating a clash. Sag is taken into account.





t is half the minimum thickness of the product.

Maximum authorized penetration: The value entered for authorized penetration must not be greater than half the minimum thickness (t).

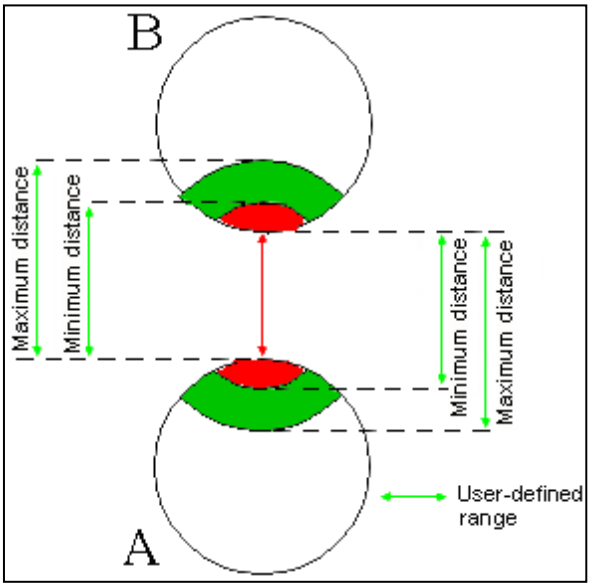
Minimum authorized penetration: The value entered must not be less than twice the sag.

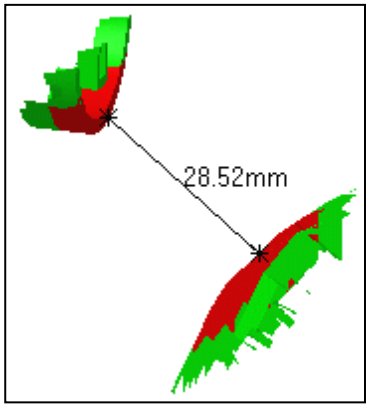
Notes: This interference type:

- Cannot be combined with the Retrieve Information from PDM setting (**Tools -> Options, Digital Mockup -> DMU Space Analysis, DMU Clash tab**).
- Must not be confused with the penetration depth (an interference result).

B

band analysis An option in the Edit Distance and Band Analysis command that computes and visualizes the areas on products corresponding to a minimum distance within a user-defined range.





The green area on A corresponds to all the points on A for which the minimum distance to B is within the user-defined minimum and maximum distance.

The red area on A corresponds to all the points on A for which the minimum distance to B is less than the user-defined minimum distance.

between all components A computation type whereby each product in the document is tested against all other products.

between two selections A computation type whereby each product in the first selection is tested against all products in the second selection.

C

CGR format CGR stands for CATIA Graphical Representation. This format is the common format used for all V5 data. CGR files always have a .cgr extension.

clash An interference type where a conflict is detected if two elements occupy the same space zone.

clearance interference An interference type where a conflict is detected if two elements are separated by less than a pre-defined distance. Also termed clearance violation or near miss.

comparison accuracy A visual comparison setting corresponding to the minimum distance between two products beyond which products are considered different.

component A last level entity in the product structure.

computation accuracy A geometric comparison setting that determines the size of the cubes used to represent the material added and/or removed.

contact interference An interference type where a conflict is detected if two elements are in contact.

D

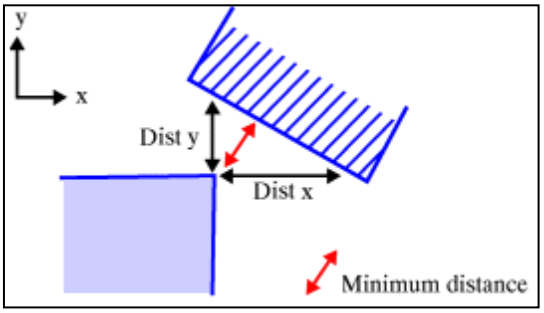
design mode A working mode in which the exact geometry is available and documents (V4 models, V5 CATParts, V5 CATProducts, etc.) are inserted as is. See also visualization mode.

detailed computation mode The second step when checking for interferences. The first step (initial computation) detects and identifies the different types of interference. Depending on your clash settings, the second step computes the graphics representation of interferences as well as the penetration depth or minimum distance.

direction of extraction The direction along which a product must be moved in order to avoid a clash. Visualized as a vector.

display accuracy A geometric comparison setting that defines the fineness or coarseness of the display of the computation results.

distance along x,y,z Options in the Edit Distance and Band Analysis command that measure distances between products along x, y or z.



DMU Digital Mockup

E

element A geometrical sub-set.

G

group of products A set of products defined explicitly using the Group icon in the DMU Space Analysis toolbar or **Insert -> Group...** in the menu bar. A persistent entity that can be stored in the document.

I

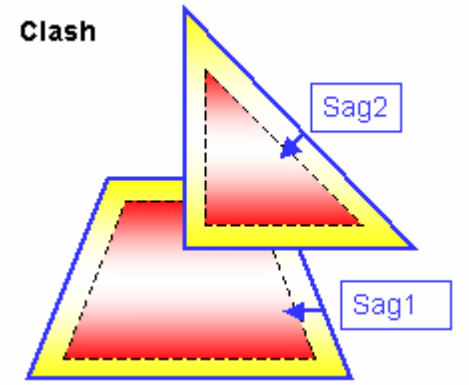
inside one selection A computation type whereby, within any one selection, each product of the selection is tested against all other products in the same selection.

interference A space conflict. Different types can be detected: clash, contact and clearance. See also authorized penetration.

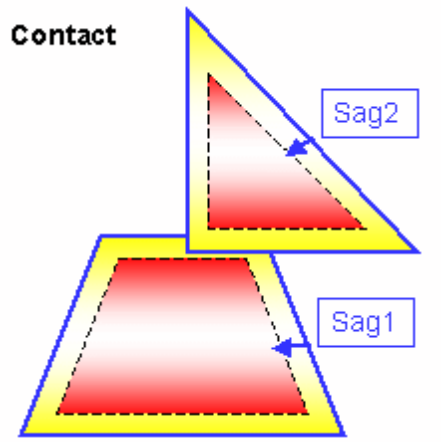
interference results Interference checking and analysis is done on polyhedric models represented by triangular meshes. Results differ depending on the interference type selected for the analysis. The following illustrates expected results for the different analysis combinations.

Contact + Clash

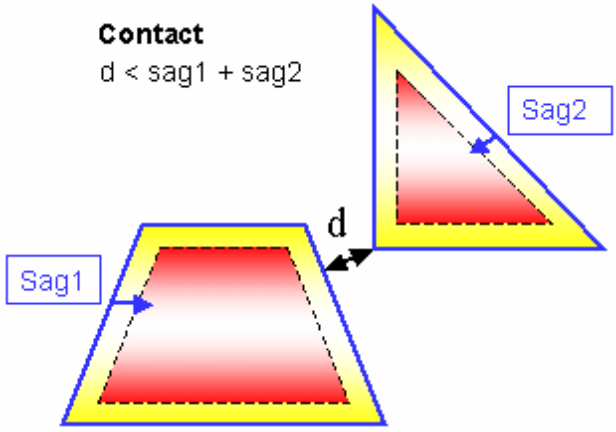
- If red zones overlap, a clash is detected.



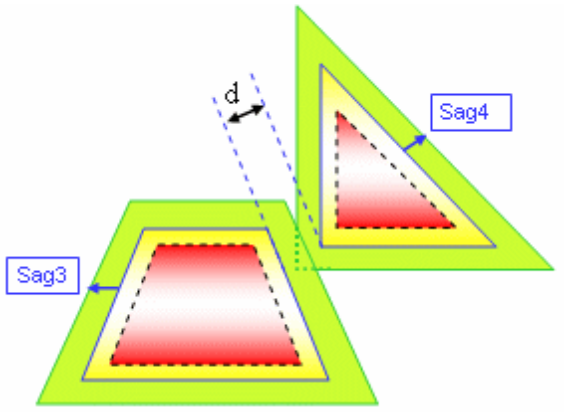
- If yellow zones only overlap, a contact is detected.



If the shortest distance (d) between the yellow zones is less than the total sag (sag1 + sag2), a contact is detected.



Same illustration as left showing outer sag (green zone):



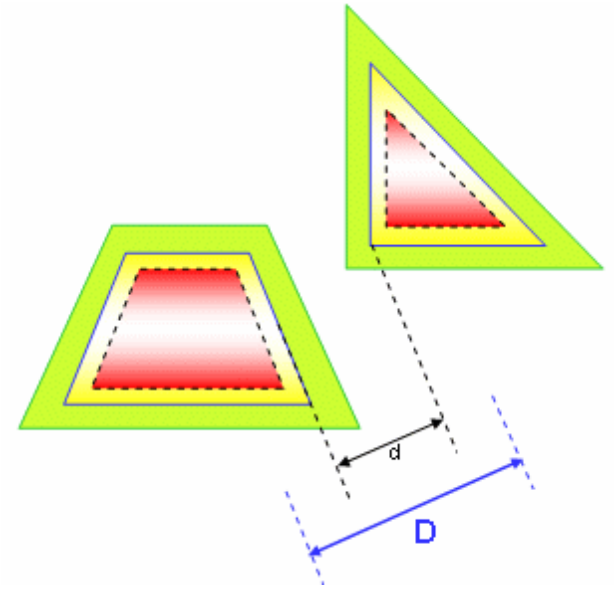
Legend for Contact + Clash illustrations:

- Blue contour: triangular mesh.
  Note: the topology is not represented.
- Dotted line: inner sag tolerance.
- Red zone: clash zone
- Yellow zone: contact zone based on inner sag
- Green zone: contact zone based on outer sag
- Yellow + Green zones: total contact zone

Clearance + Contact + Clash

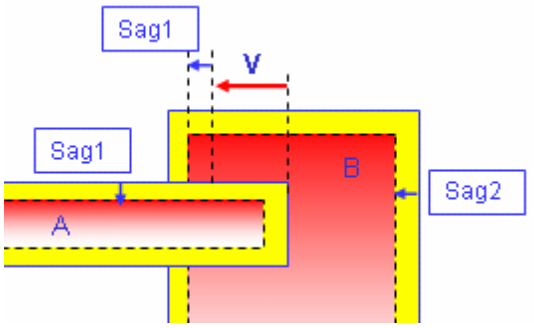
- If the total sag (sag1 + sag2) is less than the shortest distance (d), and the shortest distance (d) is less than the specified clearance distance (D), a clearance is detected.

Clearance
sag1 + sag2 < d < D

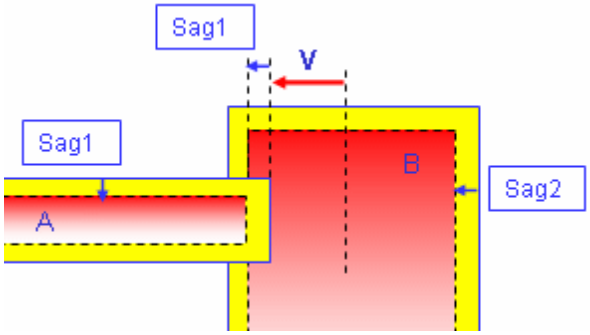


Penetration Depth

Clash



Contact

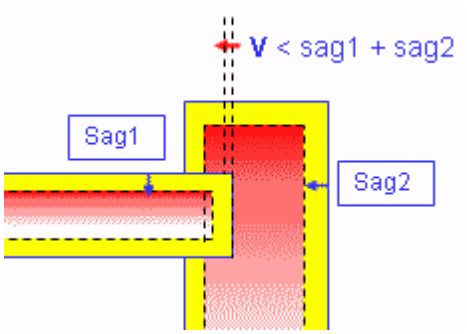


V is the penetration vector and visualizes the penetration depth.

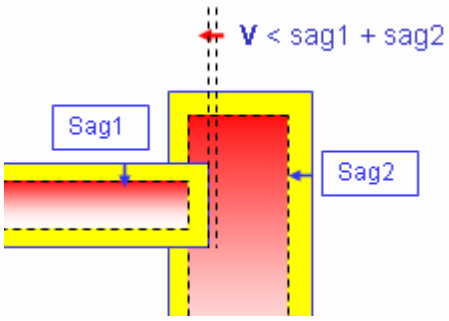
In the second illustration above, if you translate product A by V, both products are in contact.

Important: If V < sag1 + sag2, a clash may be detected (see below).

Clash

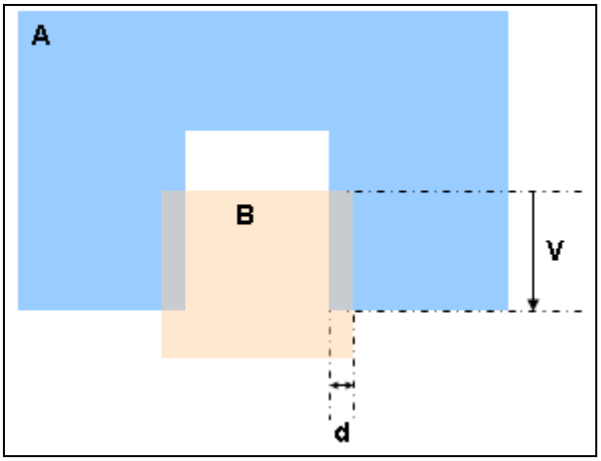


Contact



Penetration Depth versus Authorized Penetration

The penetration depth (an interference result) is not to be confused with authorized penetration (an input).



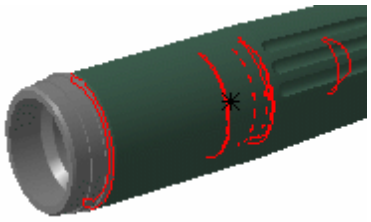
A clash is detected between products A and B.

V is the penetration vector and visualizes the penetration depth.

Note that V is greater than the authorized penetration in our illustration.

If the authorized penetration is greater than distance d, no clash is detected.

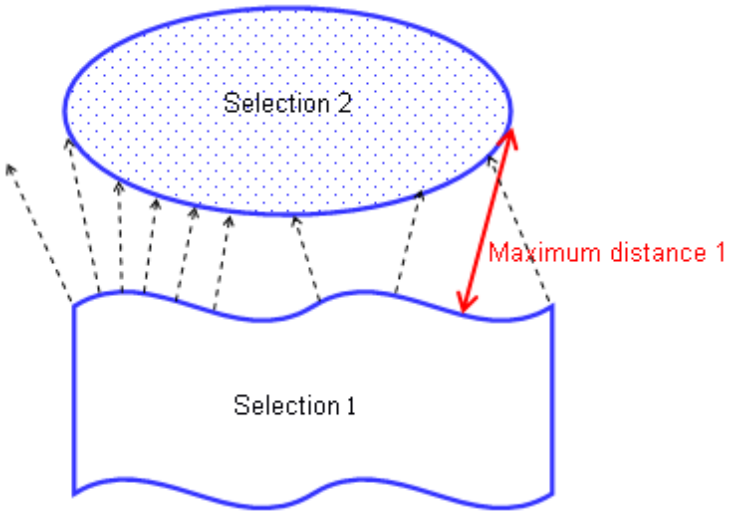
intersection curve Line segments identifying clash conflicts between products.



M

maximum distance An option in the Measure Between command that gives the approximate maximum distance normal to the selection. Two choices are available:

- Maximum distance from 1 to 2: maximum distance of all distances measured from selection 1.



- Maximum distance: the highest maximum distance between the maximum distance measured from selection 1 and the maximum distance measured from selection 2.

minimum distance An option in the Edit Distance and Band Analysis command that measures the distance between the two closest points of two product representations.

Also an option in the Measure Between command that measures the minimum distance, and if applicable the angle, between geometrical entities points, surfaces, edges, vertices and entire products).

N

notations For information on notations used for inertia.

P

penetration mode

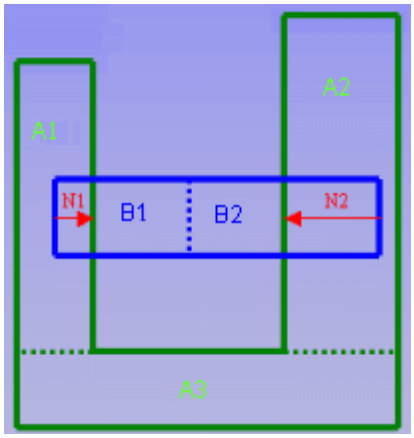
The penetration mode determines how the penetration depth is computed. Two modes are available:

- Element (default)
- Product.

This mode is set on the DMU Clash - Penetration tab of your settings (**Tools -> Options, Digital Mockup, DMU Space Analysis**).

Element Penetration Mode:

In this mode, penetration depth corresponds to the minimum distance by which it is necessary to translate an element to avoid a clash. The calculation is repeated for each pair of clashing elements and the maximum penetration depth output.



In our example, the green product (A) comprises three elements and the blue product (B) two elements.

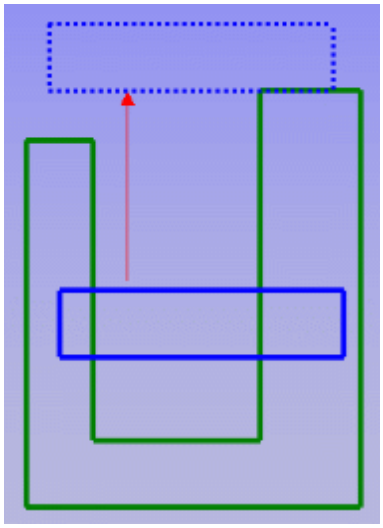
Two different pairs of elements are clashing: A1 and B1; A2 and B2. The penetration depth for the first pair of elements is N1; for the second pair N2. The largest value is output, in our case, N2.

Note: You cannot use this calculation to translate products and avoid clashes.

Product Penetration Mode:

In this mode, penetration depth corresponds to the minimum distance by which it is necessary to translate a product to avoid a clash.

This mode corresponds to the penetration depth computation prior to Version 5 Release 8.



penetration vector

Visualizes the penetration depth. The origin of the penetration vector is the center of inertia of the intersection curve. Note that this vector may sometimes be located outside selected products.
Special case: when one product is completely included in another, the origin of the penetration vector is the center of the local axis system of one of the products.

product

In DMU workbenches, a subset or constituent part of a CATProduct document, for example a cgr file or V4 model.

S

sag

Used in the Part to Part Clash, Clash, Distance and Band Analysis, and Sectioning commands.
Corresponds to the fixed sag value for calculating tessellation on objects (3D fixed accuracy) set in the Performances tab of **Tools -> Options -> General -> Display.** By default, this value is set to 0.2 mm. Note that results obtained differ depending on the sag value used.
In Visualization mode, you can dynamically change the sag value for selected objects using the **Tools -> Modify SAG** command.

section

A set of segments identifying the cross-sectional boundary produced by the intersection of a section plane and a selection of products. Sections are 2D views that help the user manage 3D documents.

section cut The result of sectioning a product and cutting away the material from the cutting plane, beyond the slice or outside the box.

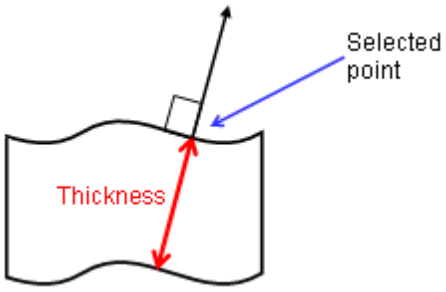
section plane A cutting plane used to create sections and section cuts. The plane has limits and can be manipulated. This is the master plane when creating section slices and boxes.

selection against all A computation type whereby each product is tested against all other products in the document.

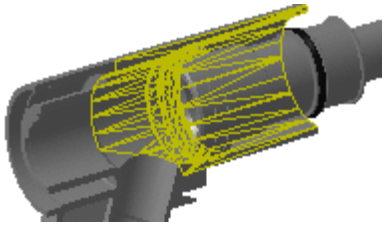
T

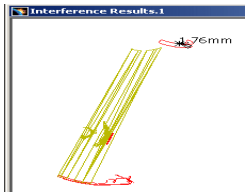
thickness A measure mode that gives the approximate item thickness along the normal to the surface at the selected picking point.

Can be accessed from Measure Item and Measure Between commands.



triangular representation Graphics representation identifying products in contact (yellow) or separated by less than the specified clearance (green) distance.





V

visualization mode A working mode in which a representation of the geometry only is available and the corresponding cgr file, if it exists, is inserted from the data cache. See also design mode.

X

XML

XML (Extensible Markup Language) is a standard format that can be used as a simple way to exchange data. The advantage of using this format to publish clash and distance results is two-fold:

- Results and how the results are presented are two separate entities, so you can customize your own style sheet to present the results the way you want without impacting the results themselves.
- XML makes documents smarter and more portable. You can for example publish the same results to a web browser as well as to your company database.

Note: XML processors letting you generate HTML pages corresponding to the XML file and style sheet are available on the market.

Index

Numerics

A

B

C

clearance

contact

interference(s)

command

2D Measure

Arc through Three Points

Capture

Clash

Clash Detection

Compare Products

Distance and Band Analysis

Edit Grid

Edit Position and Dimensions

Export As

Flip Horizontal

Flip Vertical

Geometrical Target

Grid

Invert Normal

Measure Between

Measure Inertia

Measure Item

Part to Part Clash

Positioning by 2 or 3 Selections

Reset Position

Results Window

Rotate Left

Rotate Right

Section Box

Section Fill

Section Slice

Sectioning

D

E

F

G

I

K

L

M

N

P

R

S

T

U

V

X