2D Layout for 3D Design

User's Guide

Version 5 Release 16

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Glossary

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Overview

Welcome to the *2D Layout for 3D Design User's Guide*. This guide is intended for users who need to become quickly familiar with the 2D Layout for 3D Design Version 5 product.

This overview provides the following information:

- 2D Layout for 3D Design in a Nutshell
- Before Reading this Guide
- Getting the Most Out of this Guide
- Accessing Sample Documents
- Conventions Used in this Guide

2D Layout for 3D Design in a Nutshell

2D Layout for 3D Design is a new generation product that lets you design 3D models in an advanced 2D drafting-like production environment, enabling you to create layout views based on 2D geometry, while getting the most of other key capabilities such as dress-up, dimensions, annotations, 2D components, use-edges, constraints, and outputs of profiles and planes, for example. Once your design is laid out in 2D sheets and views, you will be able to print it directly or to generate a drawing sheet from it, to add views that will illustrate it.

The *2D Layout for 3D Design User's Guide* has been designed to show you how to create layout views of varying levels of complexity.

Before Reading this Guide

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

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

- *Interactive Drafting User's Guide*: explains how to create drawings of varying levels of complexity.
- *Sketcher User's Guide*: explains how to sketch 2D elements.

Getting the Most Out of this Guide

To get the most out of this guide, we suggest that you start reading and performing the step-by-step Getting Started tutorial. This tutorial will show you how to create a basic layout from scratch, while introducing a few more advanced functionalities such as formulas.

Once you have finished, you should move on to the User Tasks section, which deals with handling layout views and sheets, then creating and modifying the various types of 2D features you need to design your layout.

If you are an administrator, the Administration Tasks section is specifically aimed at you. You will see how to manage and customize standards.

The Workbench Description section, which describes the 2D Layout for 3D Design workbench, and the Customizing section, which explains how to customize the 2D Layout for 3D Design workbench, will also certainly prove useful.

We also suggest that you refer to the Glossary for information on the terms and concepts used throughout this documentation.

Accessing Sample Documents

To perform the scenarios, you will be using sample documents contained in the online\lo1ug_C2\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 Advanced Tasks
	Interoperability
	Workbench Description
	Customizing
	Administration Tasks
	Reference

Methodology

Frequently Asked Questions

Glossary

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Text Conventions

The following text conventions are used:

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

How to Use the Mouse

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

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

What's New?

New Functionalities

Clipping box for view background

You can now define a 3D box that clips the representation of a view background using:

- a clipping frame
- a back-clipping plane.

Enhanced Functionalities

View filter enhancements

A number of enhancements have been added to view filters. You can now:

- create display or mask filters.
- filter additional 3D elements.
- filter objects from direct selection.
- customize view creation to add filter capabilities. See Customizing Settings below.

Customizing Settings

2D Layout for 3D Design settings

2D Layout for 3D Design now has its own category of settings, available through **Tools -> Options -> Mechanical Design -> 2D Layout for 3D Design -> View Creation** tab. Drafting settings (available through **Tools -> Options -> Mechanical Design -> Drafting**) continue to be used for 2D Layout for 3D Design as well.

Getting Started

Before getting into the detailed instructions for using the 2D Layout for 3D Design workbench, the following tutorial aims at giving you a feel of what you can do with the product. It provides a step-by-step scenario showing you how to use key functionalities. You just need to follow the instructions as you progress along.

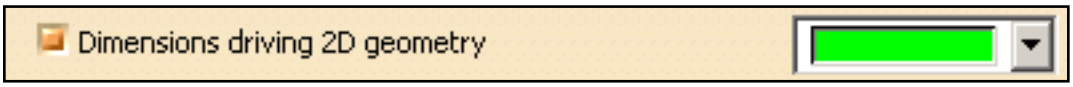
The main tasks described in this section are the following:

Entering the 2D Layout for 3D Design Workbench
Starting the Preliminary Design of a Part
Completing the Preliminary Design in Another View
Creating the 3D Part

Before starting this scenario, you should be familiar with the basic commands common to all workbenches. These are described in the *Infrastructure User's Guide*.

All together, the tasks should take about 45 minutes to complete.

- Setting the options in **Tools -> Options -> Mechanical Design -> Drafting** is recommended to improve the software performance. For more information, refer to the Customizing Settings section.
- For the purpose of this Getting Started, the color for the **Dimensions driving 2D geometry** option was set to green (instead of the default blue). You may leave the default color as is. However, if you want to customize it to replay this Getting Started in the same conditions, go to **Tools -> Options -> Mechanical Design -> Drafting -> Dimension** tab, and click the **Types and colors...** button in the **Analysis Display Mode** area. In the **Types and colors** dialog box, choose green for the **Dimensions driving 2D geometry** option, and then click **Close**. The driving dimensions you will subsequently create will then be displayed in green instead of blue.

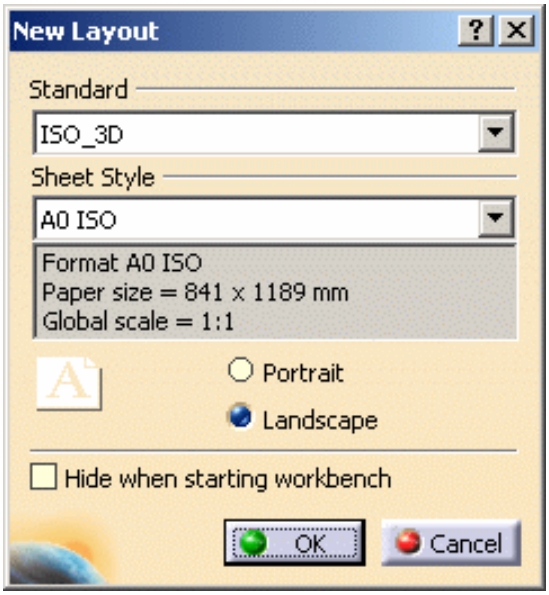


Entering the 2D Layout for 3D Design Workbench

This first task shows you how to enter the 2D Layout for 3D Design workbench and start a new layout.

1. Select **Start -> Mechanical Design** from the menu bar.
2. Select the **2D Layout for 3D Design** workbench.

   The **New Layout** dialog box is displayed, allowing you to choose a standard, a sheet style and an orientation for your new layout. Among other things, the sheet style defines the sheet format, paper size, scale and default orientation.

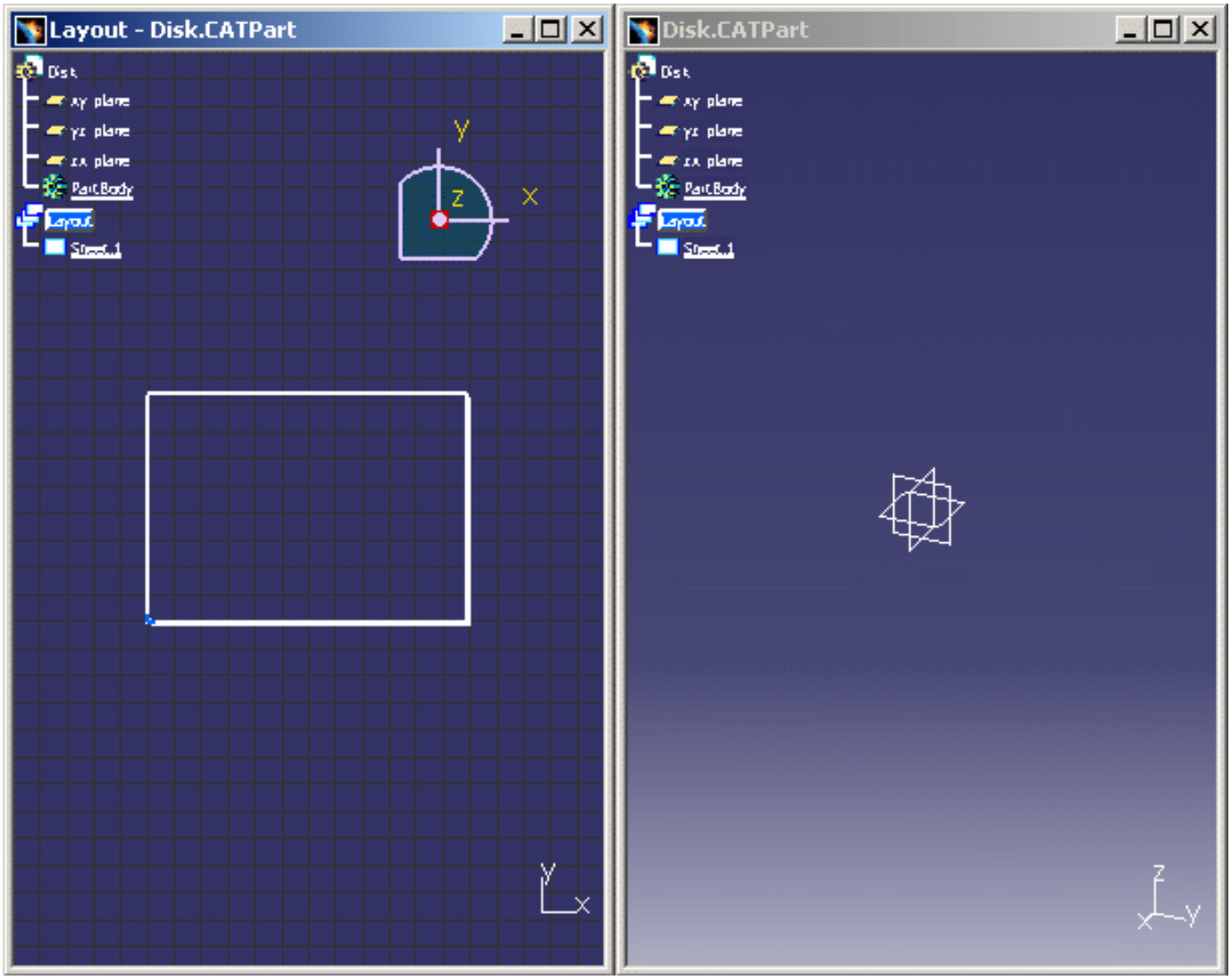


3. Make sure the **ISO_3D** standard is selected. Leave the other options with their default values.
4. Click **OK**.

   The **New Part** dialog box is displayed.

5. Enter a name for the part that will be associated to your layout (Disk, for example), and click **OK**.

   An empty sheet is created in a specific 2D window, and the associated part document is created and opened in a 3D window.



The commands for creating and editing features are available in the 2D Layout for 3D Design workbench toolbars. Now, to fully discover the 2D Layout for 3D Design workbench, let's perform the next tasks. You will begin by starting the preliminary design of a part.

Starting the Preliminary Design of a Part

In this task, you will learn how to create the preliminary design of a part in the empty sheet you created in the previous task. This involves the following steps:

- configuring your options
- creating a new design view
- creating 2D geometry
- creating a center line with reference
- creating dimensions

Your new layout should still be open from the previous task. If not, open the Disk.CATPart document.

At this stage, you may want to maximize the 2D window. You will not be working in the 3D window for the moment, so you do not need to have it displayed all the time.

Configuring your options

For more information on the various options available in the **Visualization** and in the **Tools** toolbar, refer to Layout Tools. For more information on settings, refer to Customizing Settings.

1. In the **Visualization** toolbar, make sure that:
   - the **Sketcher Grid** and **Cutting Plane** icons are inactive,
   - the **Display Backgrounds as Specified for Each View**, **Show Constraints** and **Analysis Display Mode** icons are active.
2. In the **Tools** toolbar, make sure that the **Create Detected Constraints** icon is active. You can configure the other icons as desired.
3. Go to **Tools -> Options -> Mechanical Design -> Drafting -> Dimension** tab, and select the **Create driving dimension** option. You will use this option to create driving radius dimensions in the next steps.
4. Click **OK** to validate your settings and exit the **Options** dialog box.

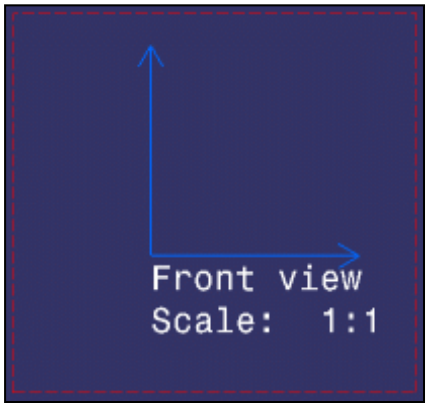
Creating a new design view

1. Click the **New View** icon in the **Layout** toolbar.
2. Click on the sheet to position the new view.

   You may find it interesting to note how the view is previewed in the part window (you need to zoom out, as the view box defined in the ISO_3D standard has sides of 1000mm - for more information on the standards, see Administration Tasks).

   An empty primary view is created, displaying a blue axis in a red frame, as well as the view name and scale. Additionally, the Front View item is added to the specification tree.

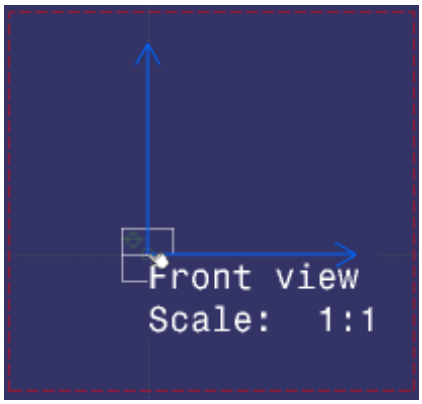
   In our scenario, the primary view is a front view. The view type for the primary view is defined in the current standard, i.e. ISO_3D in our scenario.



Creating 2D geometry

1. Click the **Circle** icon in the **Geometry Creation** toolbar. The **Tools Palette** is automatically displayed.

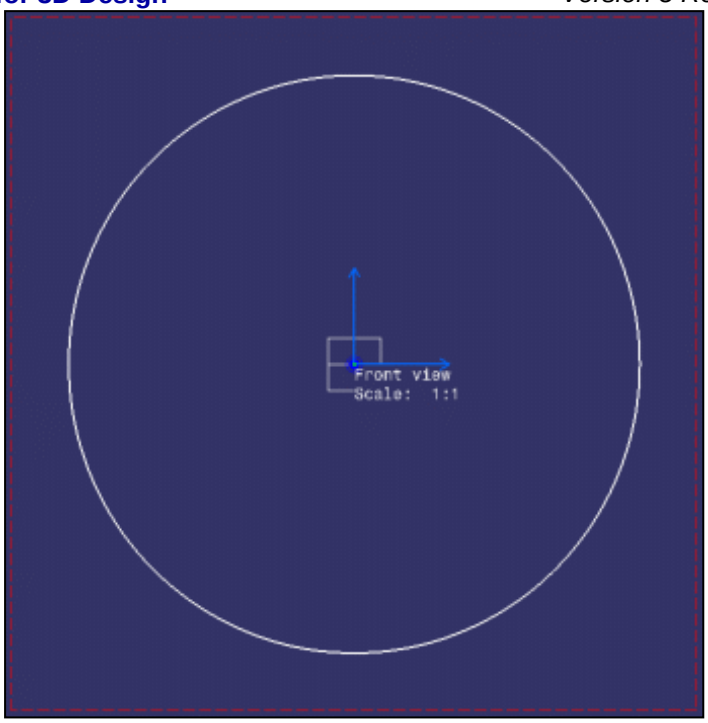
2. Click to select the front view origin as the circle center.



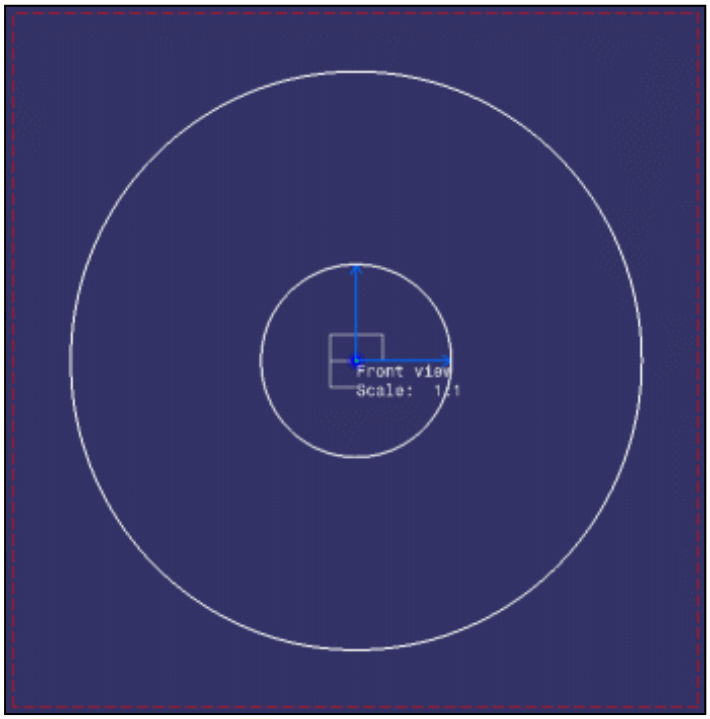
3. In the **Tools Palette**, type 90 as the radius value and press Enter.

You do not need to position the cursor in the **Tools Palette**, as already it has the focus. Simply start typing on your keyboard.

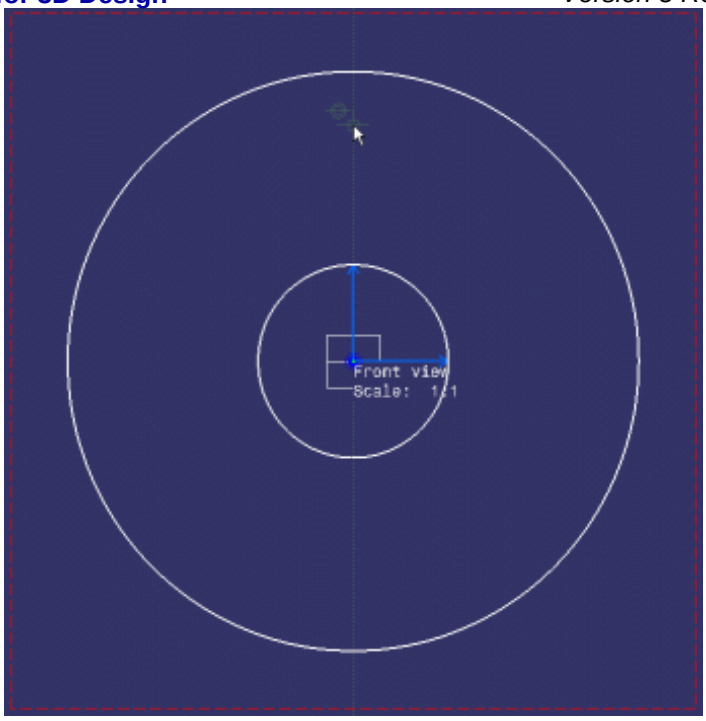
The circle is created.

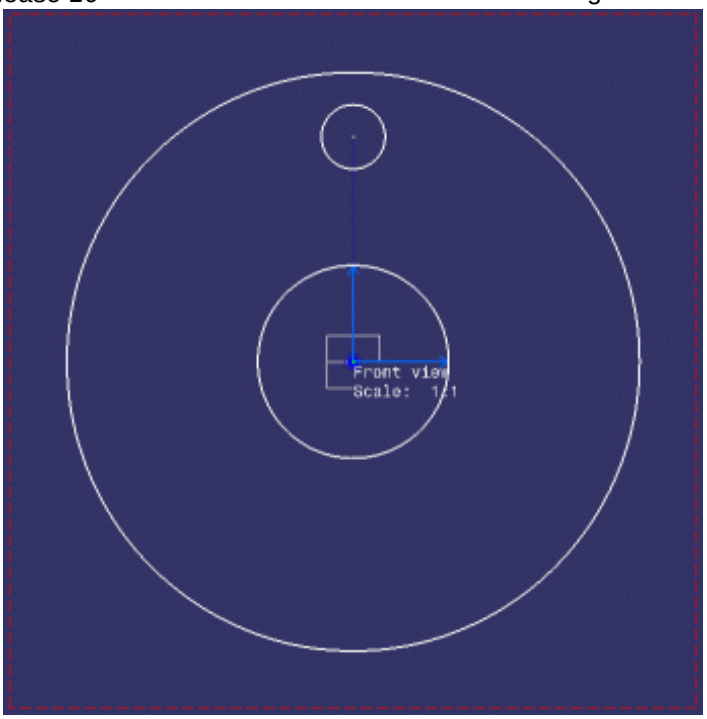


4. Repeat steps 1 to 3 to create a second circle, this time entering 30 as the radius value.



5. Repeat steps 1 to 3 to create a third circle, this time pointing to the absolute axis V direction so as to use it as the reference for the circle center, and entering 10 as the radius value.





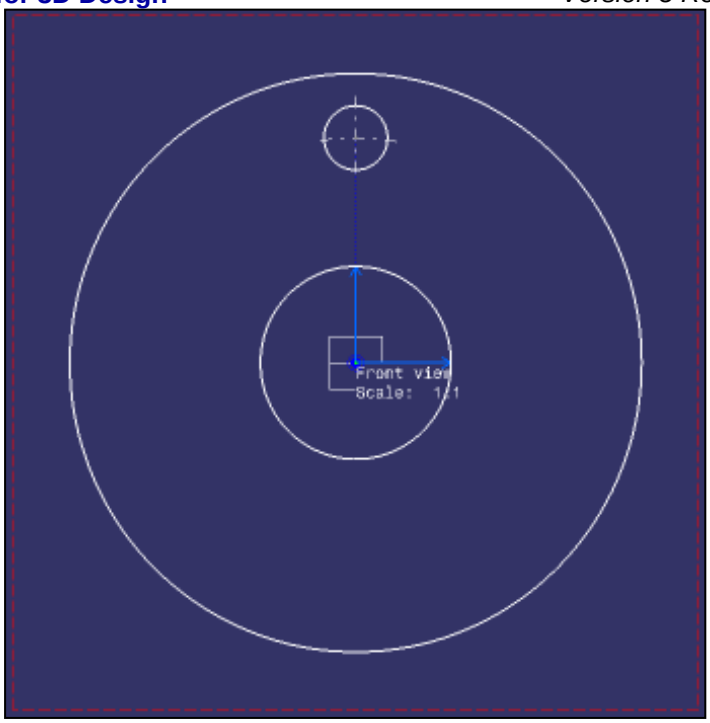
Creating a center line with reference

At this stage, you will be creating a center line with reference so as to show that there will be a hole pattern along it.

1. Click the **Center Line with Reference** icon in the **Dress-up** toolbar (**Axis and Threads** sub-toolbar).

2. Select the circle to which the center line will be applied, that is the smallest circle (the last-created one).
3. Select the circle that will serve as the center line reference, that is the biggest circle (the first-created one).

   The center lines are created and are associative with the reference circle.

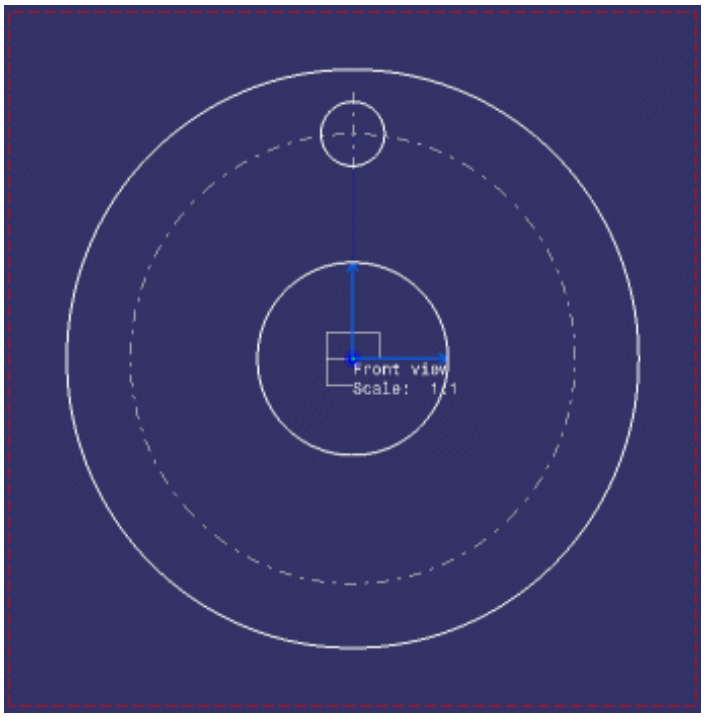


4. Select the center lines. Manipulators appear.

5. Press the Ctrl key and drag the horizontal center line along the reference circle.

6. Click in the free space to validate.

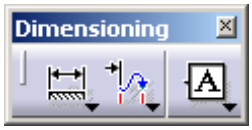
The horizontal center line is extended along its reference circle.



Creating dimensions

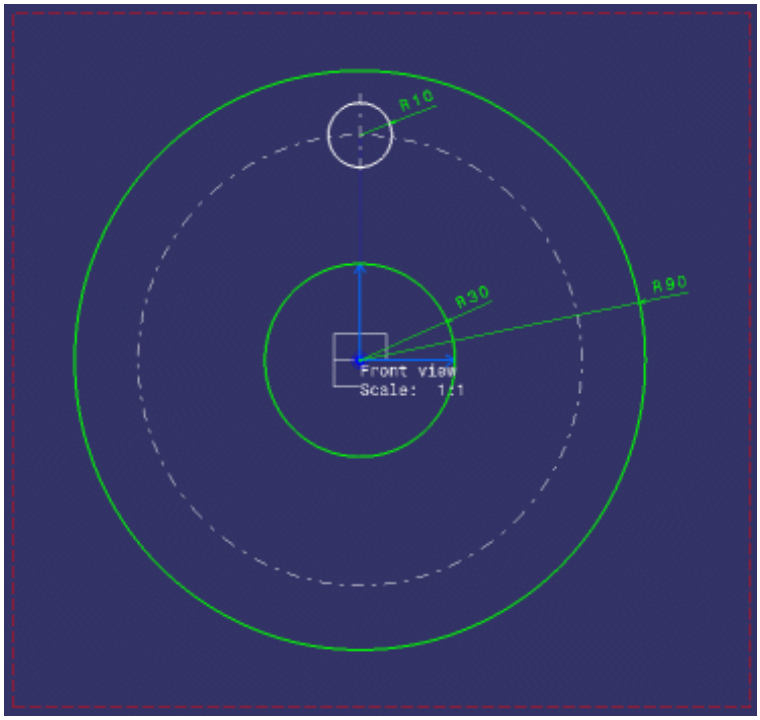
The dimensions that you will be creating in this task will be driving dimensions, as previously defined when configuring your options.

1. Click the **Radius Dimensions** icon in the **Dimensioning** toolbar (**Dimensions** sub-toolbar).



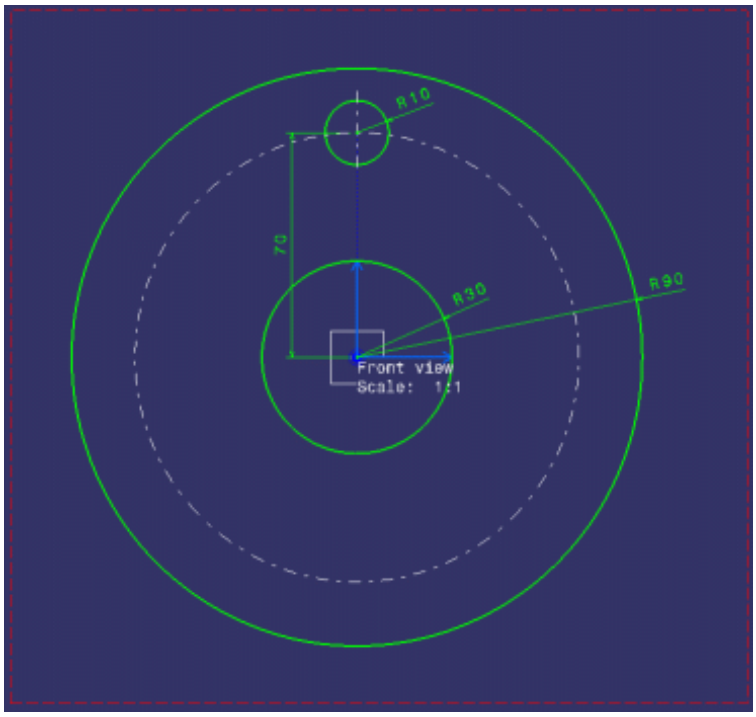
The **Tools Palette** is automatically displayed,

2. Make sure the **Force dimension on element** icon is active.
3. Select a circle.
4. Click at the location where you want to position the dimension. The dimension is created.
5. Repeat steps 1 to 4 to create dimensions for the two other circles (the **Force dimension on element** icon remains active).
6. Re-position your dimensions if necessary.



7. Click the **Dimensions** icon in the **Dimensioning** toolbar.
8. Select the small and then the medium-size circles (or their center points) to create a distance dimension between their center points. The dimension is previewed.
9. If the previewed dimension value is not 70, type 70 as the distance value in the **Tools Palette** and press Enter.

The small circle will be moved accordingly.

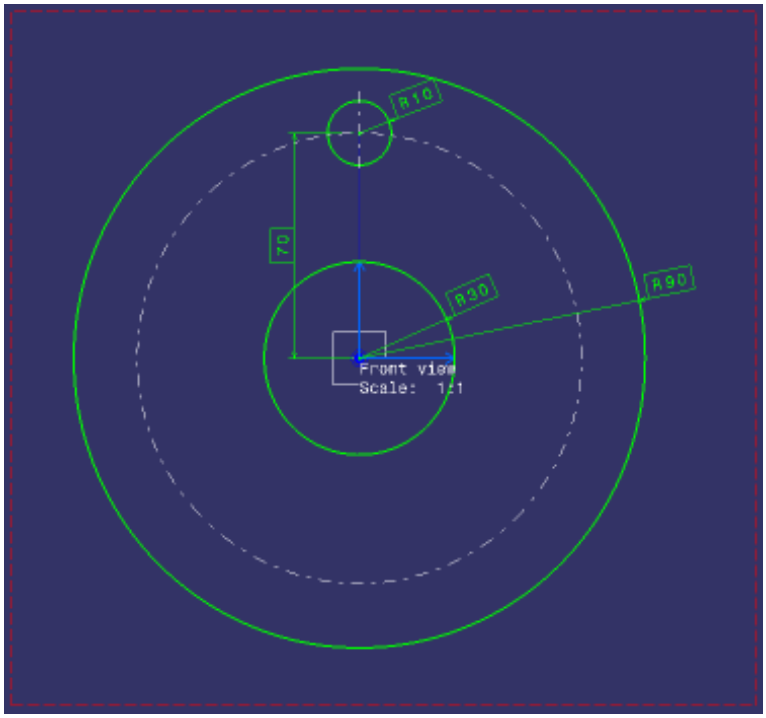


10. Click at the location where you want to position the dimension. The dimension is created.

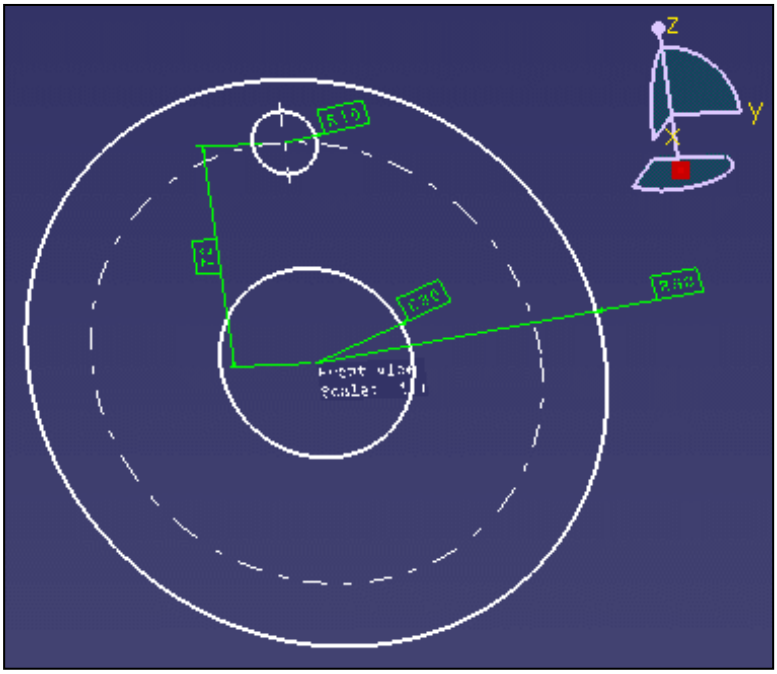
11. Multi-select all dimensions using the Ctrl key.

12. Click the **Frame** icon in the **Text Properties** toolbar. The **Frames** sub-menu is displayed.

13. Select the variable-size rectangle frame . Rectangle frames are added to all dimensions. This shows that they are reference dimensions.



You are now done creating your front view. Notice how the layout is previewed in the part window.



Now, let's complete the preliminary design of your part in another view.

Completing the Preliminary Design in Another View

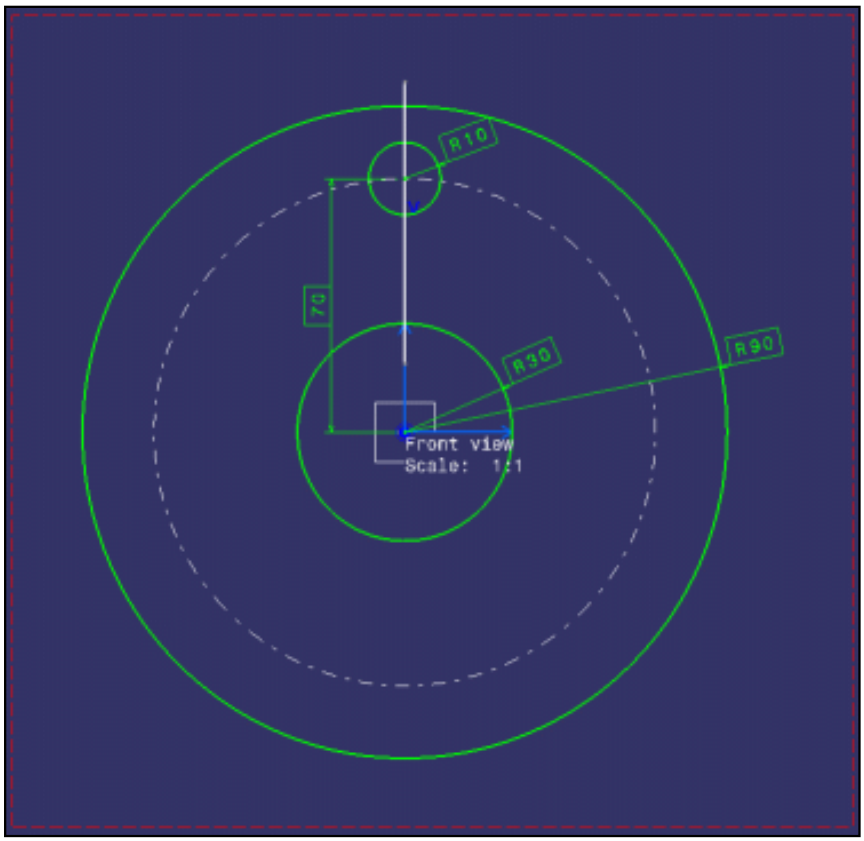
In this task, you will continue the preliminary design of the part you've started designing in the previous task. This involves the following steps:

- creating a section view
- hiding the 2D and 3D backgrounds
- defining the view content using folding lines
- fixing the geometry together
- adding dress-up
- creating dimensions

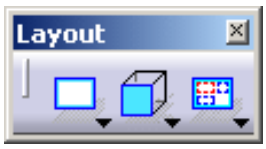
Your layout should still be open from the previous task. If not, open the Disk2.CATPart document.

Creating a section view

1. Click the **Line** icon in the **Geometry Creation** toolbar.
2. Use the vertical axis to define the cutting profile as shown below, and double-click to end the line creation.



3. Click the **New Section/Auxiliary View** icon in the **Layout** toolbar (**Views** sub-toolbar).



4. Select the line you have just created as the cutting profile.

5. In the **Tools Palette** which is automatically displayed, select the **Section View** icon .

This option is also available from a contextual menu.

6. Click in the layout at the location where you want the section view to be positioned.

Positioning the view also defines the section view direction, as if it were a left or a right projection view.

A section view is created. Additionally, the Section view item is added to the specification tree. Note that the 2D background is shown in the section view, enabling you to see the cutting profile from the front view.

Hiding the 2D and 3D backgrounds

At this stage, you will hide both the 2D background (i.e. the 3D representation of 2D elements which do not belong to the current view, but to other views) and the 3D background (i.e. the representation of all 3D elements, including edges, faces and 3D wireframe) from the front and section views.

1. Right-click the front view and select **Background -> Invisible**.

2. Repeat this operation for the section view. The 2D background is now hidden from the section view (you do not see the cutting profile anymore).

You can also multi-select the views and then perform this operation.

Defining the view content using folding lines

At this stage, you will see how to add geometry in the view using folding lines as a guide. You can use folding lines for any kind of view, as long as the planes they correspond to are not parallel. For example, you cannot have folding lines between a front view and a rear view.

1. Double-click the section view to activate it.

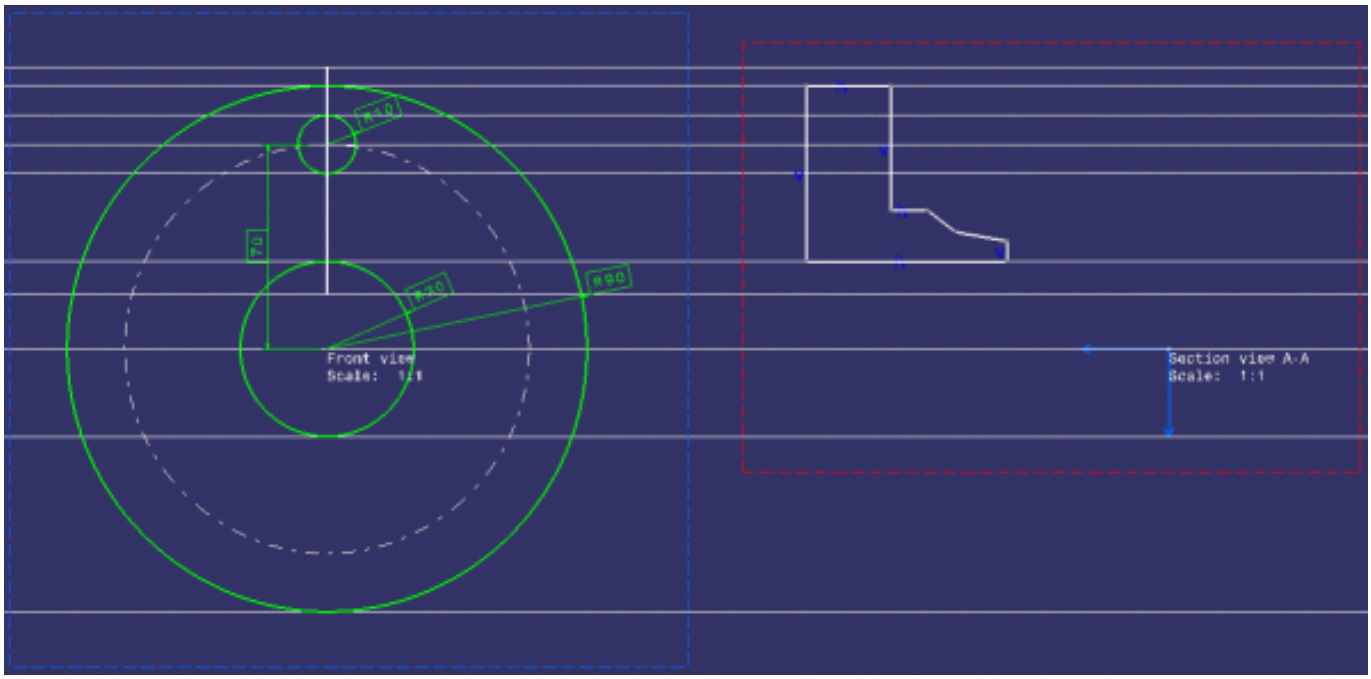
2. Right-click the front view to display the contextual menu.

3. Select **Front view object -> Show Folding Lines**. The folding lines are displayed.

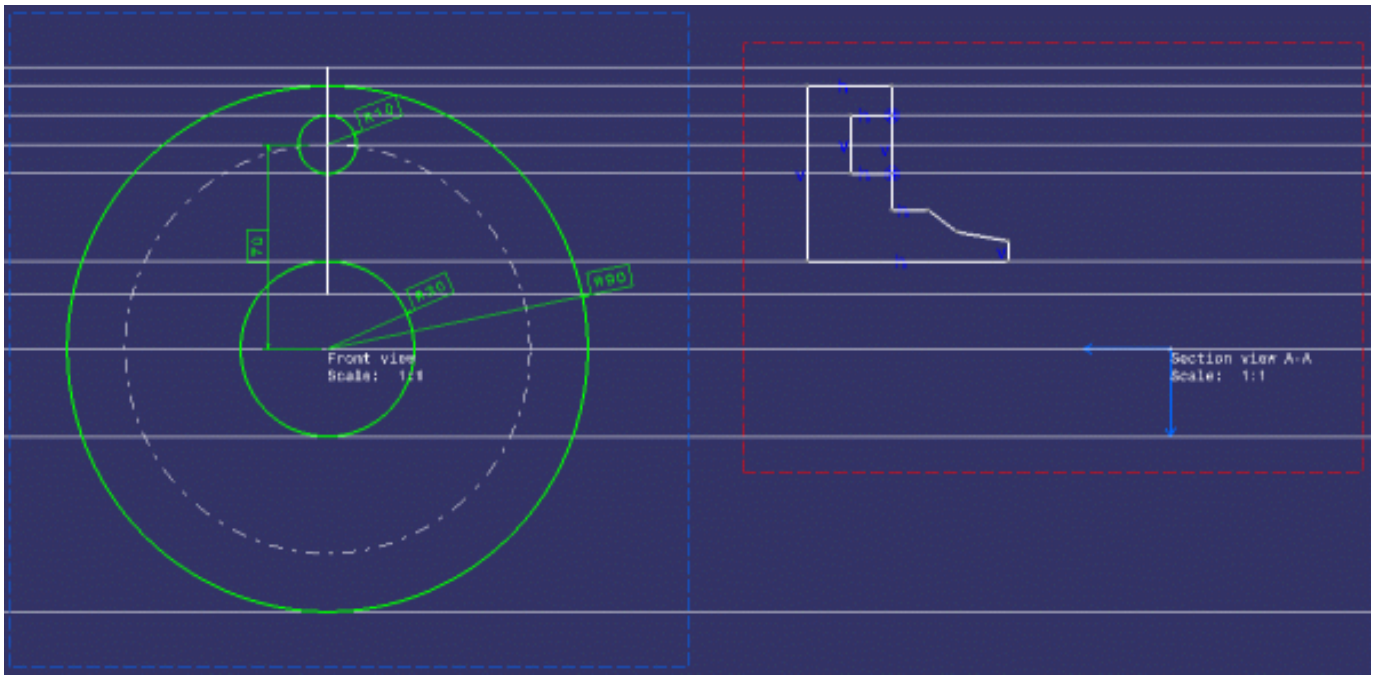
4. Click the **Profile** icon in the **Geometry Creation** toolbar.

5. In the section view, define the profile as shown below, using the folding lines as a guide, and double-click when

done.



6. Repeat steps 4 and 5 to define another profile for the hole.



7. Right-click the front view to display the contextual menu.

8. Select **Front view object -> Hide Folding Lines**. The folding lines are hidden.

Fixing the geometry together

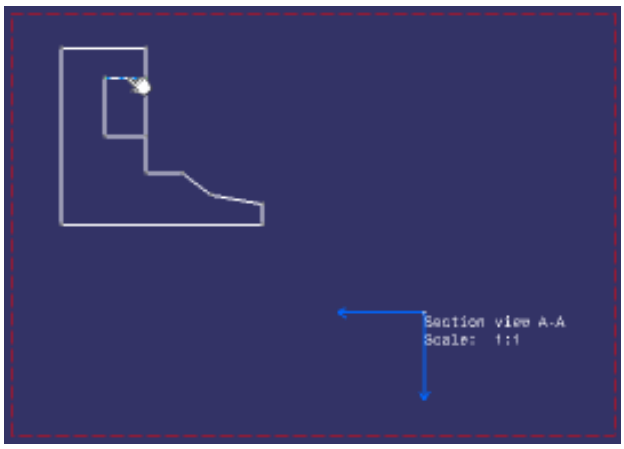
1. Click the **Fix Together** icon in the **Constraint** toolbar.
2. Using the Ctrl key, multi-select the profile you created in step 4 of the previous task (i.e. the external profile, not the hole profile). The **Fix Together Definition** dialog box is displayed.
3. Click **OK**. The geometry in the section view is now rigidly constrained.

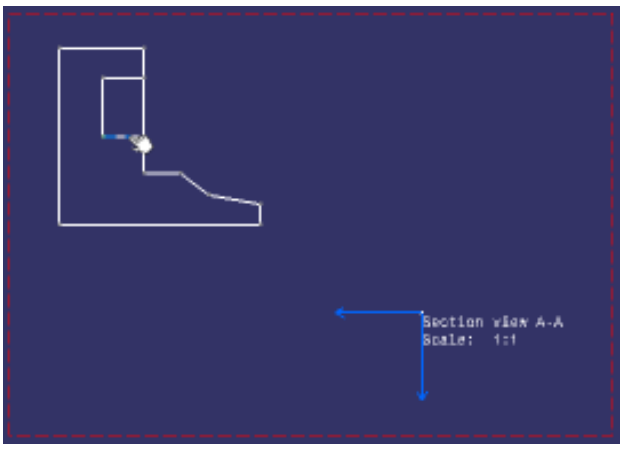
Adding dress-up

At this stage, you will add dress-up elements to the section view. This will make your layout clearer.

You may now want to hide constraints. To do this, in the **Visualization** toolbar, deactivate the **Show Constraints** icon.

1. Click the **Axis Line** icon in the **Dress-up** toolbar (**Axis and Threads** sub-toolbar).
2. Select the first and then the second line of reference as shown below.

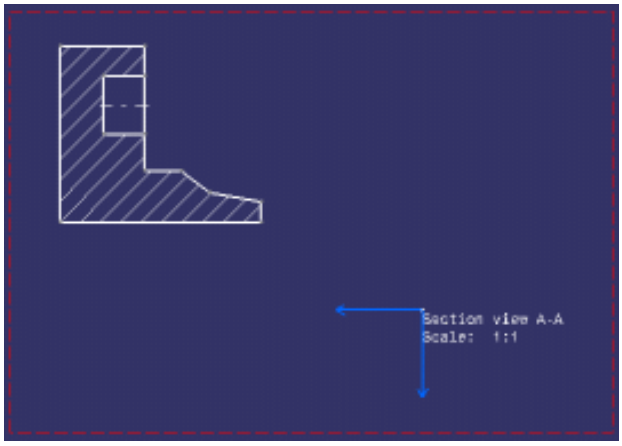




The axis line is created.

3. Click the **Area Fill** icon in the **Dress-up** toolbar. The **Area detection** dialog box is displayed.
4. Leave the default option (**Automatic**) selected, and click inside the section view profile area.

The area fill is created.

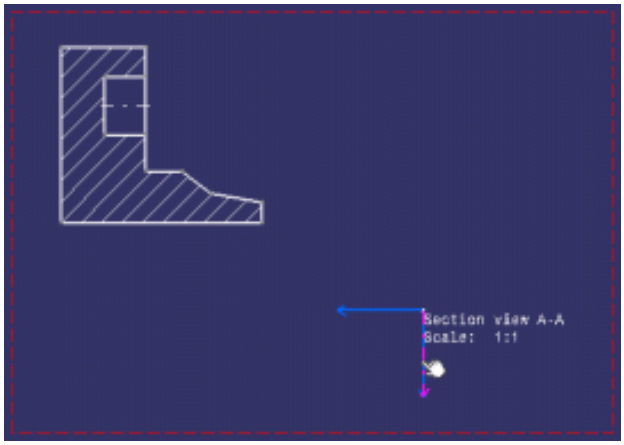


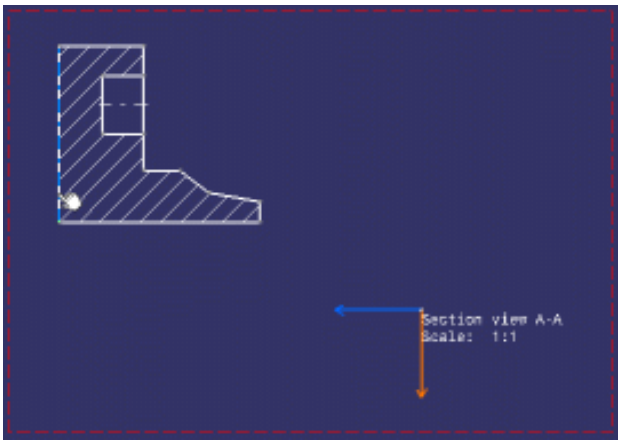
Creating dimensions

The dimensions that you will be creating in this task will be driving dimensions, as defined in the previous task when configuring your options.

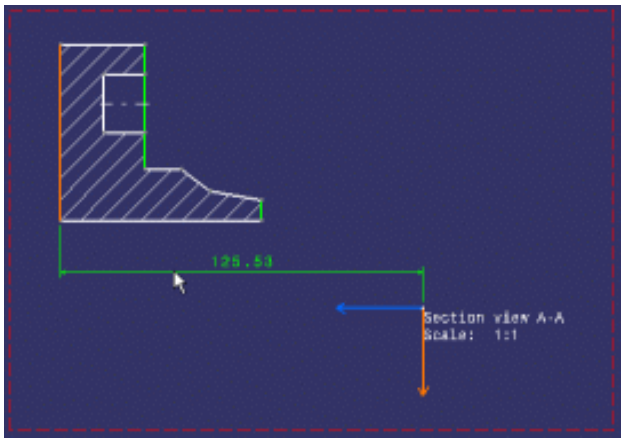
1. Click the **Dimensions** icon in the **Dimensioning** toolbar. The **Tools Palette** is automatically displayed.

2. Select the section view vertical axis, and then the line as shown below.





A preview of the dimension to be created is displayed.



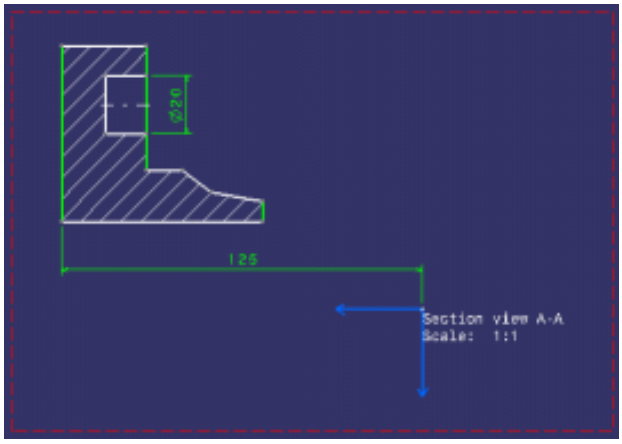
3. If the previewed value is not 125, type 125 in the **Value** field of the **Tools Palette** and then press enter. The whole geometry is moved accordingly.

4. Click at the location where you want to position the dimension. The dimension is created.

5. Click the **Diameter Dimensions** icon in the **Dimensioning** toolbar.

6. Select the first and then the second line defining the hole.

7. Click at the location where you want to position the dimension. The dimension is created, with a value of 20 (if you properly defined the hole profile using the folding lines).



8. If you wish, you can continue creating dimensions until the geometry in the section view is fully iso-constrained. The whole geometry should be green, as defined for iso-constrained elements in the **Diagnostic colors** dialog box. This setting is available via **Tools -> Options -> Mechanical Design -> Drafting -> Geometry** tab, **Colors** button next to the **Visualization of Diagnostic** option.

9. Re-position your dimensions if necessary.

Creating a formula

At this stage, you will be creating a formula specifying that the diameter dimension value (in the section view) is equal to the radius dimension value of the hole (in the front view) multiplied by 2. The radius dimension value will then drive the diameter dimension value.

1. Click the **Formula** icon in the **Knowledge** toolbar.

   The **Formulas: Layout** dialog box is displayed.

2. Select the diameter dimension you created in steps 6 and 7 of the previous task. The parameters list is updated with the parameters associated to this dimension.

3. Make sure the parameter (Offset) that specifies the dimension value is selected.

4. Click the **Add Formula** button. The **Formula Editor** dialog box is displayed.

5. Select the R10 dimension in the front view to add it to the formula field.

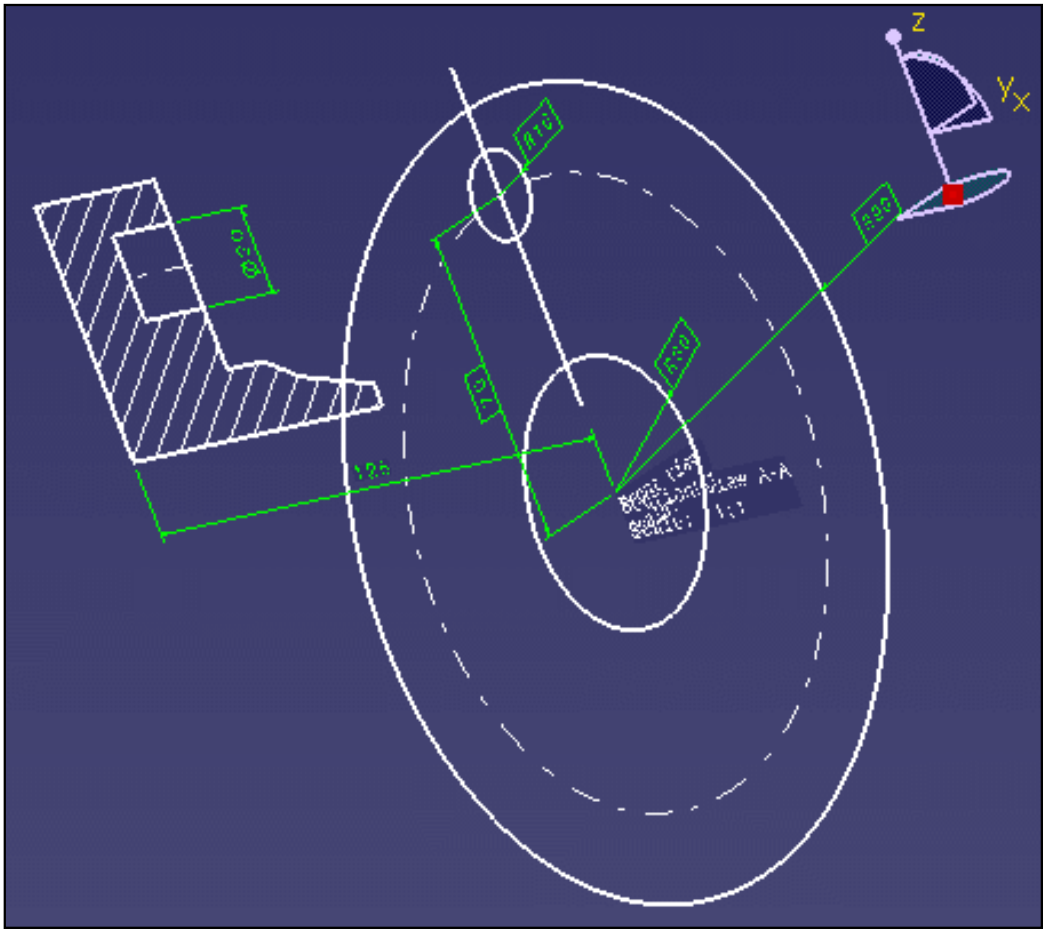
6. Still in the formula field, type *2.

7. Click **OK** to close the **Formula Editor** dialog box. The formula you have just created is listed in front of the associated parameter in the **Formulas: Disk** dialog box.

8. Click **OK** to validate and close the **Formulas: Disk** dialog box.

   If you now edit the radius dimension value from 10 to 11, for example, you will notice that the diameter dimension value changes to 22.

Your preliminary design is now finished. Notice how the layout is previewed in the 3D window.



You can now create the 3D part from this preliminary design.

Creating the 3D Part

In this task, you will create the 3D part from the preliminary design you have finished in the previous task. This involves the following steps:

- creating a 3D profile
- creating a 3D profile on a support plane parallel to the 3D view plane
- creating a 3D plane
- creating a shaft
- creating a pocket
- creating a circular pattern
- checking your layout

Your layout should still be open from the previous task. If not, open the Disk3.CATPart document.

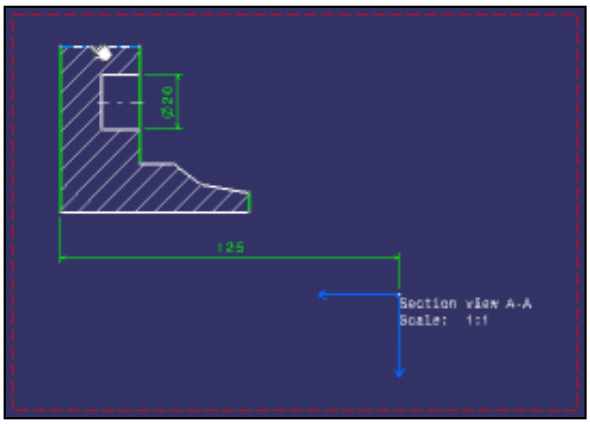
Tile the 2D and 3D windows vertically as you now need to have both windows displayed.

In the 3D window, you can see that elements are pre-positioned, but no 3D element is created. If you right-click the views in the 2D window and then select **Visualization -> Hide in 3D**, you will see that the geometry is hidden, and that no part exists. To display the geometry again, right-click the views again and select **Visualization -> Show in 3D**.

Creating a 3D profile

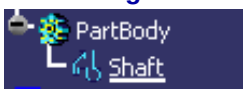
At this stage, you will create a 3D profile to use as a reference element when creating the shaft.

1. In the 2D window, make sure the section view is still active from the previous task. If not, double-click to activate it.
2. Click the **3D Profile** icon in the **3D Geometry** toolbar.
3. Select the line as shown below.



The **Profile Definition** dialog box is displayed.

4. Enter a name for your 3D profile, Shaft for example.
5. Make sure the **Wire (Automatic Propagation)** option is selected from the **Mode:** drop-down list.
6. Click **OK** to validate and close the dialog box. The 3D profile is created, on the same plane as the section view, and it is listed in the specification tree, under the PartBody node.

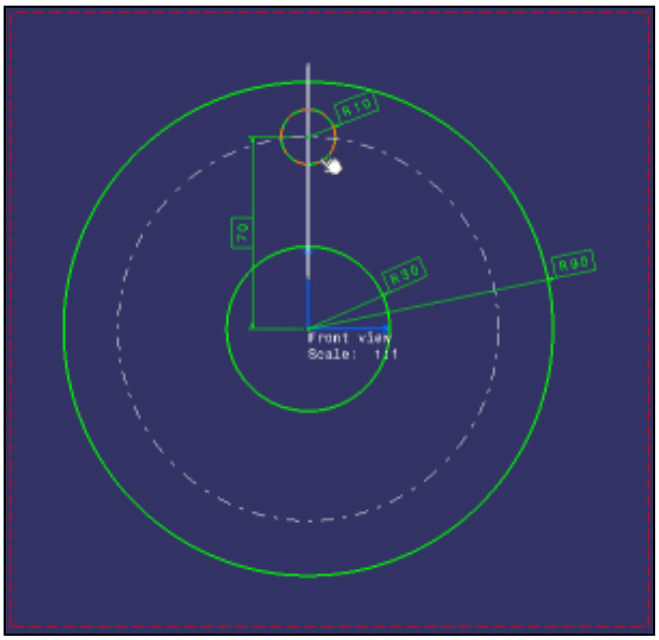


Of all elements created from 2D geometry in 2D Layout for 3D Design, only 3D profiles and planes belong to the current part body.

Creating a 3D profile on a support plane parallel to the 3D view plane

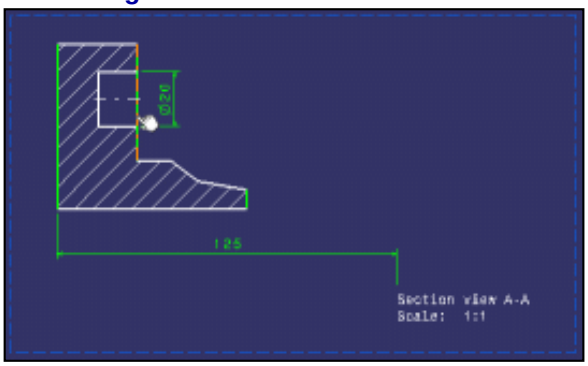
At this stage, you will create a 3D profile on a plane which is parallel to the 3D view plane. This parallel plane will be used as a support plane when creating the pocket.

1. Double-click the front view to activate it.
2. Click the **3D Profile** icon in the **3D Geometry** toolbar.
3. Select the R10 circle as shown below.



The **Profile Definition** dialog box is displayed.

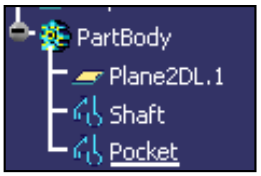
4. Right-click inside the **Support Plane** field.
5. Select the **Create Plane** option in the contextual menu which is displayed.
6. Select the line as shown below.



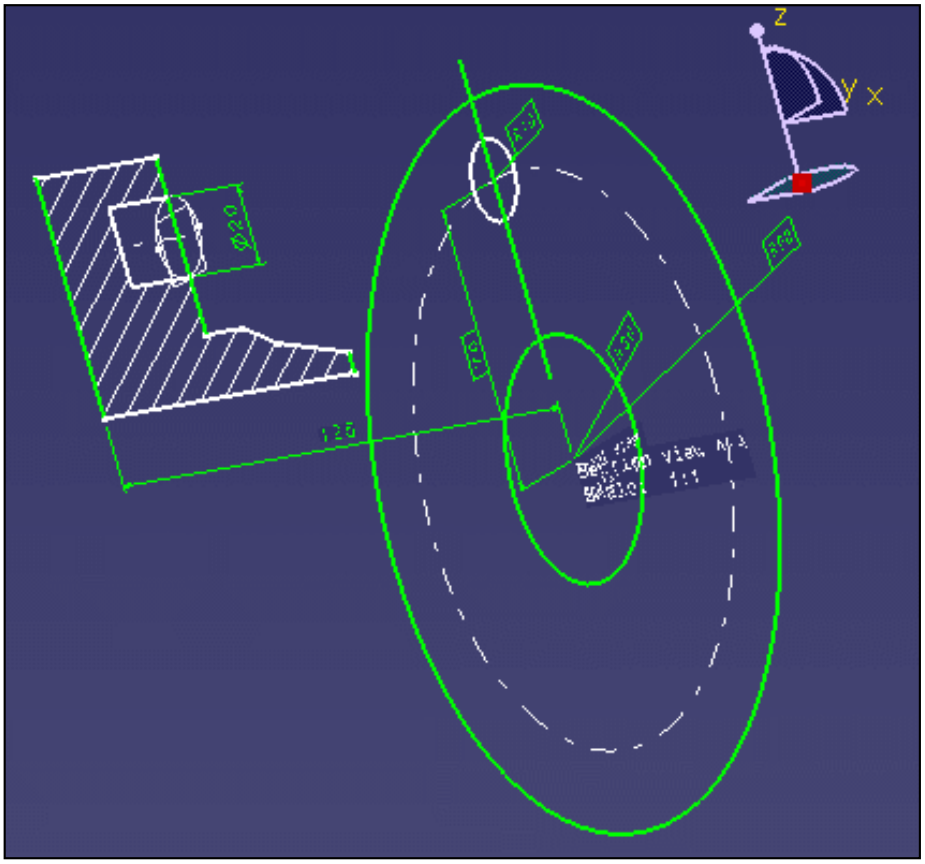
The 3D plane, Plane2DL.1, is created and listed in the specification tree, under the PartBody node.

7. Back in the **Profile Definition** dialog box, enter a name for your 3D profile, Pocket for example.
8. Make sure Plane2DL.1 is selected in the **Support Plane** field.
9. Click **OK** to validate and close the dialog box.

The 3D profile of the circle is created on the support plane which is parallel to the front view. It is listed in the specification tree under the PartBody node.



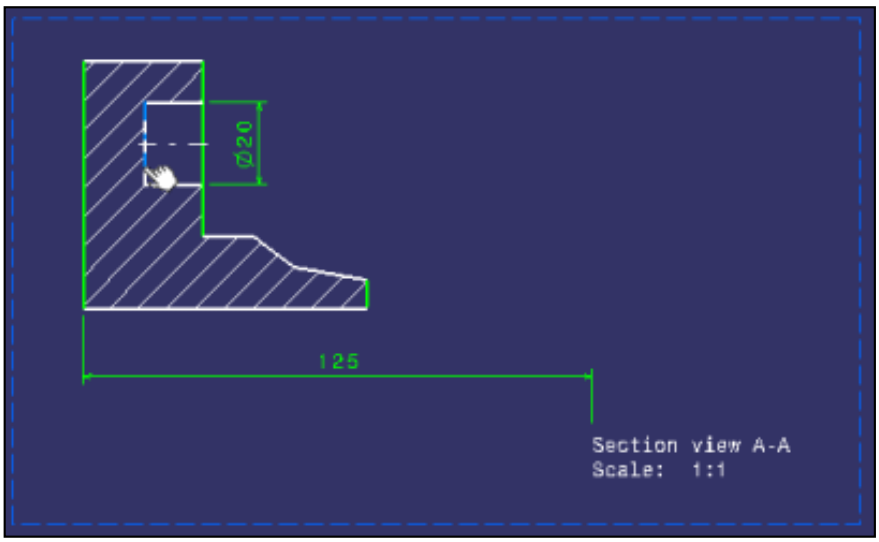
Furthermore, the 3D plane and 3D profile are displayed in the 3D window.



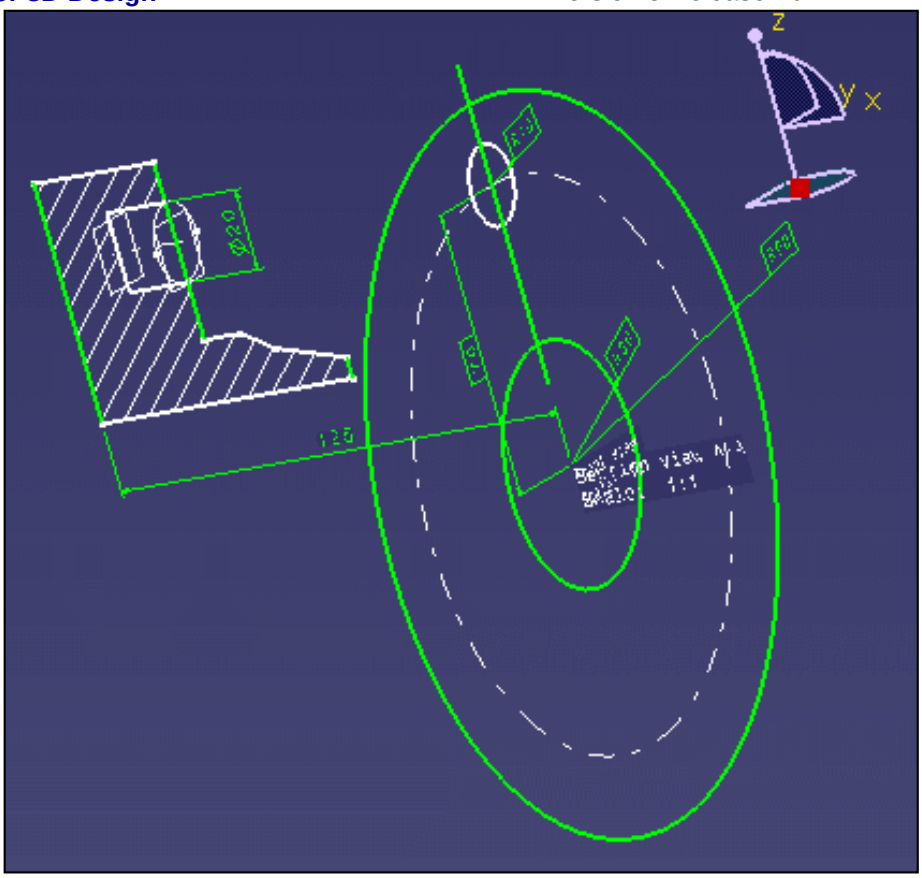
Creating a 3D plane

At this stage, you will create a 3D plane to use as a limit when defining the pocket depth.

1. Make sure the front view is still active from the previous task. If not, double-click to activate it.
2. Click the **3D Plane** icon in the **3D Geometry** toolbar.
3. Select the line as shown below.

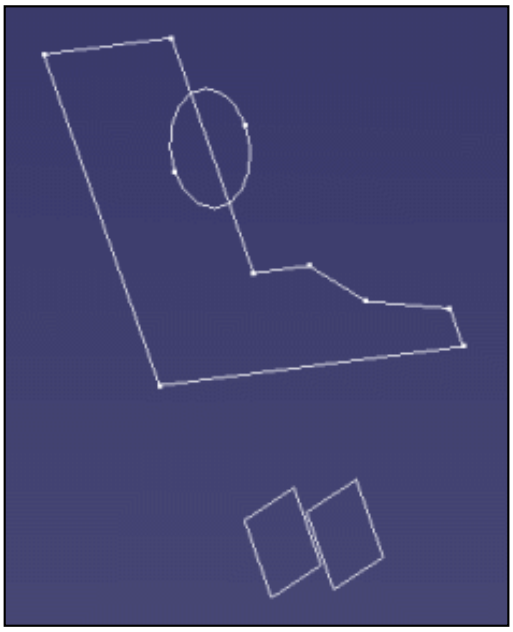


The 3D plane is created, on the same plane as the previous one. It is displayed in the specification tree as the Plane2DL.2 feature, as well as in the 3D window.



Creating a shaft

1. Optionally, right-click the views in the 2D window and select **Visualization -> Hide in 3D** from the contextual menu to hide the 2D geometry from the 3D window. Only the elements that will be used to create the solid are visualized.

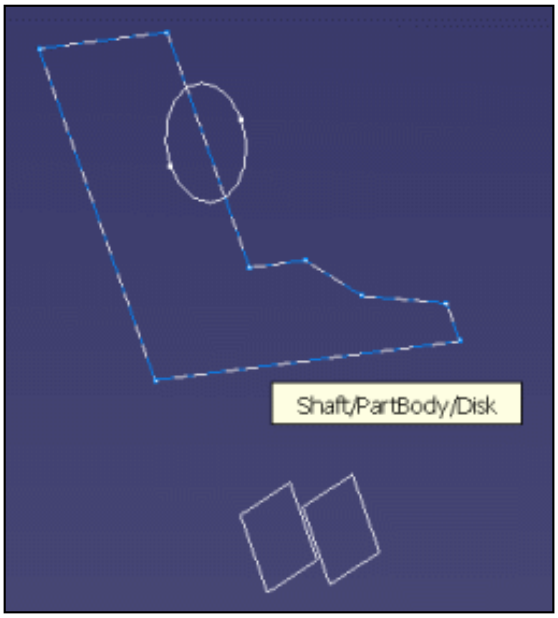


To display the geometry again, right-click the views again and select **Visualization -> Show in 3D** from the contextual menu.

2. Activate the 3D window.

3. Click the **Shaft** icon . The **Shaft Definition** dialog box is displayed.

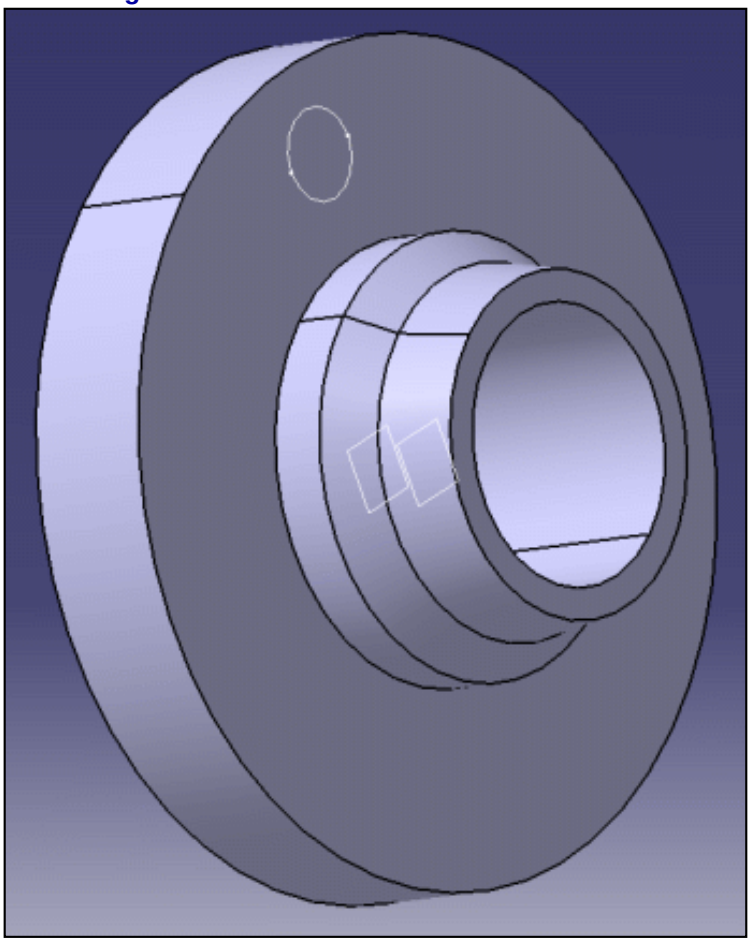
4. Select the Shaft feature as the profile, either from the 3D geometry area or from the specification tree.



5. In the **Axis** area, right-click the **Selection** field.

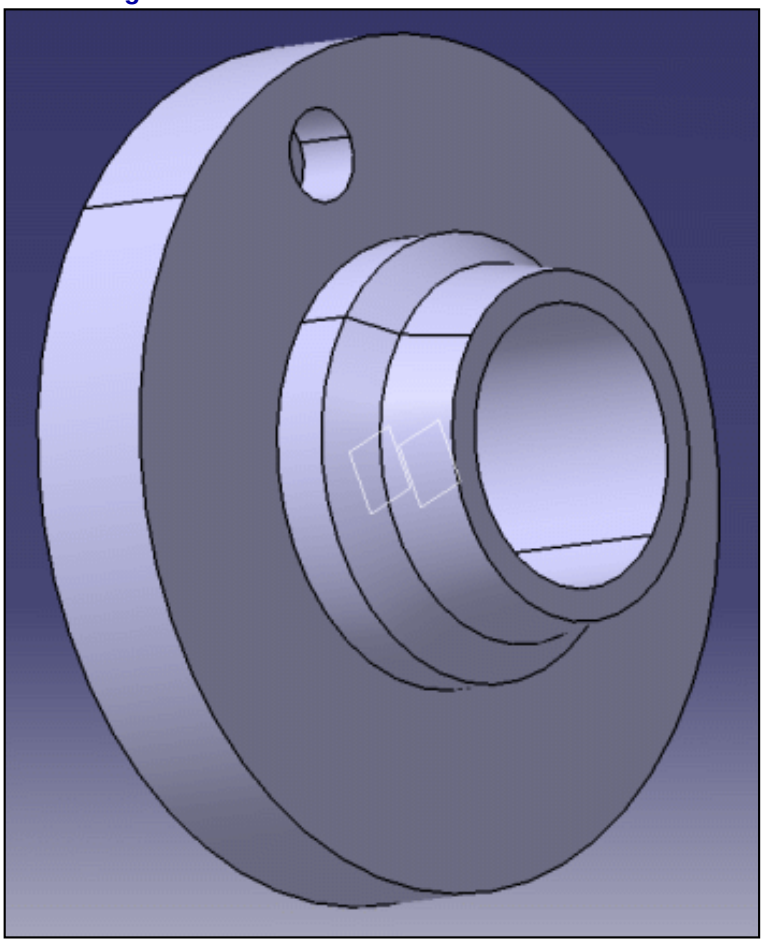
6. Select **X Axis** as the axis for the shaft in the contextual menu which is displayed. A preview of the shaft is displayed.

7. Click **OK** to validate and close the dialog box. The shaft is created.



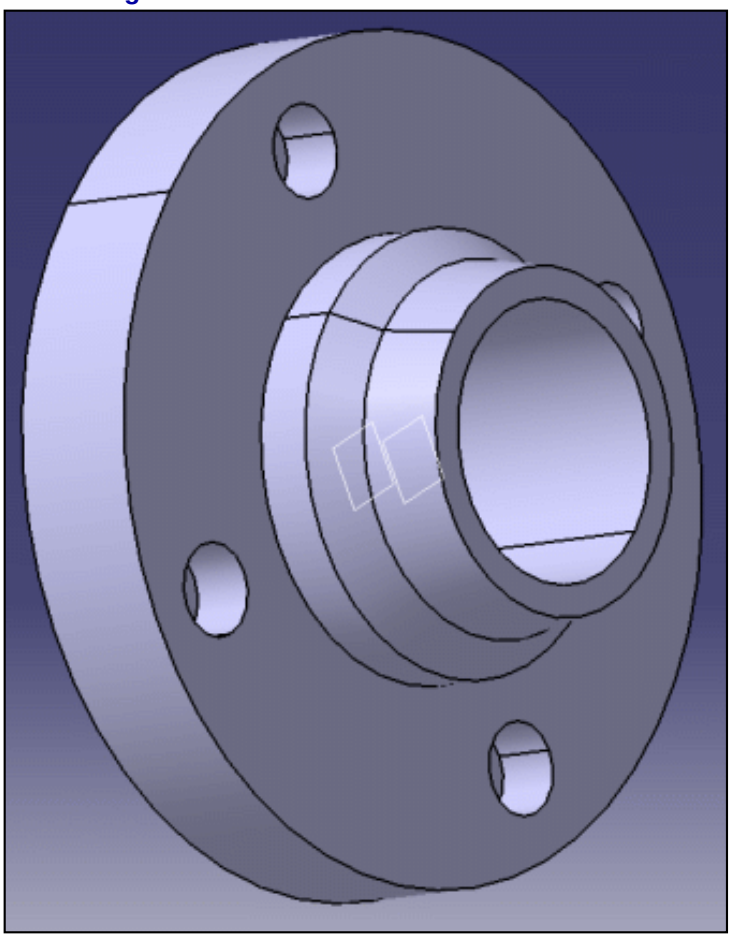
Creating a pocket

1. Still in the 3D window, select the Pocket feature as the profile, either from the 3D geometry area or from the specification tree.
2. Click the **Pocket** icon . The **Pocket Definition** dialog box is displayed.
3. Select **Up to plane** as the type.
4. From the specification tree, select Plane2DL.2 as the limit for the pocket.
5. Click **OK** to validate and close the dialog box. The pocket is created.



Creating a circular pattern

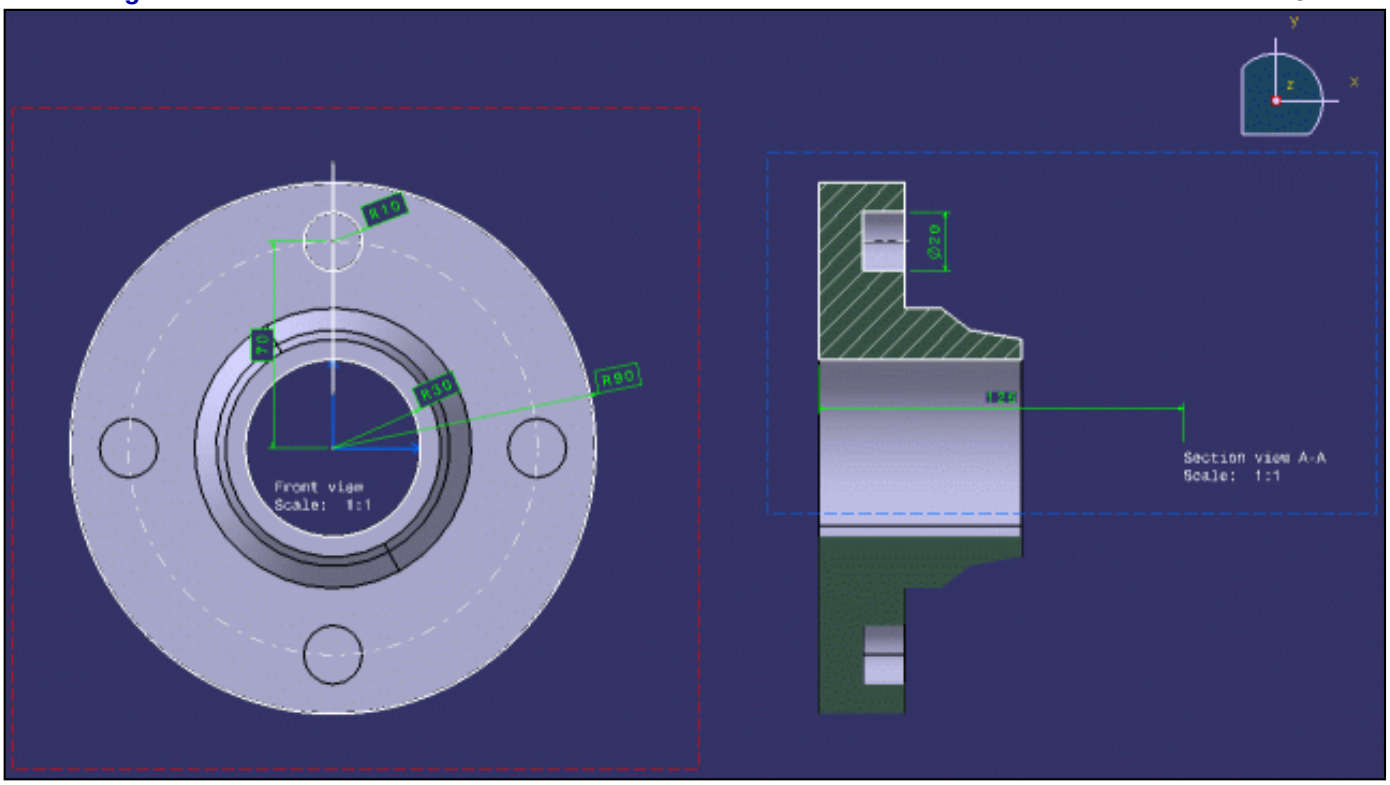
1. Still in the 3D window, select Pocket.1 from the specification tree.
2. Click the **Circular Pattern** icon in the **Transformation** toolbar (**Pattern** sub-toolbar). The **Circular Pattern Definition** dialog box is displayed.
3. Select **Complete Crown** from the **Parameters** drop-down list.
4. Enter 4 in the **Instance(s)** field.
5. As the reference element, right-click the field and select X axis from the contextual menu.
6. Select the pocket (Pocket.1) as the object to pattern.
7. Click **OK** to validate and close the dialog box. The circular pattern is created.



Checking your layout

Now that you have created your 3D part, you can check your layout to make sure it is correct.

1. Activate the Layout window.
2. Right-click the front view and select **Background ->** **Standard**.
3. Repeat this operation for the section view. The 3D background of your layout is displayed in the 2D window.
4. In the **Visualization** toolbar, activate the **Cutting Plane** icon. The 3D background of the Section view is cut along its definition plane.



You have now finished the Getting Started scenario. If you wish, you can open the Disk4.CATPart document to make sure that your layout is similar to our sample.

For more in-depth information about the various functionalities available in 2D Layout for 3D Design, refer to the User Tasks chapter.

User Tasks

The information you will find in this section is listed below:

Layout Tools
Copying, Cutting, Pasting and Deleting
Layout Creation and Edition
Layout Sheets
View Creation
View Management
2D Geometry
2D Geometry Modification
2D Components
Dimensioning
Constraints
Annotations
Dress-up
3D Outputs
Use-Edges
Integration with the Drafting Workbench
Creating View Filters
Printing a Layout
Properties

Layout Tools

The 2D Layout for 3D Design workbench provides a number of tools that you can use when designing a layout. These tools are available using the following toolbars:

- Tools
- Visualization
- Tools Palette

Tools

The **Tools** toolbar displays a number of options. This toolbar is situated at the bottom right of the screen. If you cannot see it properly, just undock it.

The **Tools** toolbar provides the following options:

- **Snap to Point**
- **Create Detected Constraints**
- **Dimension system selection mode**
- **Update 3D profile**

Snap to Point

If activated, this option makes your geometry (as well as 2D components) begin or end on the points of the grid. As you create geometry, points are forced to the intersection points of the grid. Note that this option is also available via **Tools -> Options -> Mechanical Design -> Drafting -> General** tab.

You can use autodetection (the SmartPick capability) even if this option is activated. For more information, refer to the SmartPick task in the *Sketcher User's Guide*.

Create Detected Constraints

If activated, this option creates lasting constraints. If you do not activate this option, the constraints you create are temporary: the geometry is only temporarily constrained, which means that it can then be moved without being constrained.

Dimension system selection mode

This option applies to dimension systems. With this option activated, clicking a dimension system enables you to select the dimension system as a whole. When this option is de-activated, you will be able to select a single dimension within a dimension system.

Update 3D profile

Click this icon to update the 3D profile (and corresponding part) that corresponds to a given layout. For example, if you perform modifications in a layout which impact the 3D profile, the part will be shown as being not up-to-date. Clicking this icon lets you reflect your latest modifications in the 3D model.

Visualization

The **Visualization** toolbar displays a number of visualization-related options. This toolbar is situated at the bottom right of screen, after the **Tools** toolbar. If you cannot see it properly, just undock it.

- **Sketcher Grid**
- **Cutting Plane**
- **Display Backgrounds as Specified for Each View**
- **Show Constraints**
- **Display View Frame as Specified for Each View**
- **Analysis Display Mode**

Sketcher Grid

Activate this option to display the grid in your session. The grid will help you draw geometry in given circumstances. For example, the grid will make it easier to draw profiles requiring parallel lines. The grid depends on the active view position, orientation and scale. Note that this option is also available through **Tools -> Options -> Mechanical Design -> Drafting -> General** tab.

Cutting Plane

Activate this option to cut the 3D background of layout views along each view's definition plane. This can be useful to validate the geometry of a design view, as the view content and 2D background will remain visible, even if their view plane is behind the cutting plane. Since it is the view plane that is used as the cutting plane, the result will depend on the position of the layout in 3D space.

Note that the cutting plane is applied globally to a layout, that is it is either active or inactive for all views at once.

Refer to Using the Cutting Plane for more information.

Display Backgrounds as Specified for Each View

Activate this option to display the 2D and 3D backgrounds as specified for each view.

Refer to Managing the Layout View Background for more information.

Show Constraints

Activate this option if you want existing constraints to be visualized.

Constraints are only visualized in the 2D Layout for 3D Design window, not in the 3D window.

If you cannot visualize constraints even though this option is active, go to **Tools -> Options -> Mechanical Design -> Drafting -> Geometry** tab and select **Display constraints**. You can also modify the constraint color and/or width.

Display View Frame as Specified for Each View

When activated, this option lets you display the view frame which is available for each view, as specified in each view's properties (**Edit -> Properties**, **View** tab, **Display View Frame** check box). In this case:

- the view frame will be displayed for all views for which the **Display View Frame** check box is selected.
- the view frame will be hidden for all views for which the **Display View Frame** check box is cleared.

Deactivating this option hides the view frame for all views.

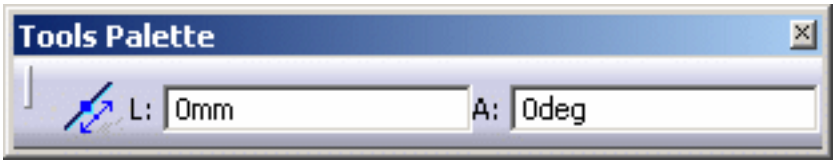
Analysis Display Mode

This option lets you visualize different types of dimensions (for example, dimensions driving 2D geometry and true dimensions) using a specific color for each.

These colors are those customized in the **Options** dialog box. To modify these colors, go to **Tools -> Options -> Mechanical Design -> Drafting -> Dimension** tab. Select **Activate analysis display mode** and, if needed, click the **Types and colors** button to assign the desired colors to specific dimension types.

Tools Palette

The **Tools Palette** appears whenever you select a command for which specific options or value fields are available. This enables you to know immediately when tools are available for a command.

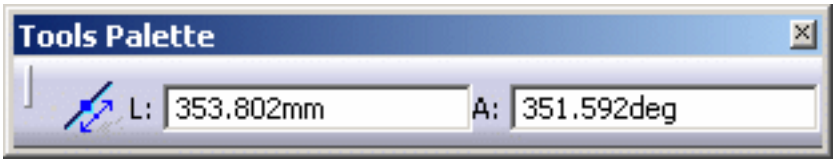


The options or fields available in the **Tools Palette** depend on the command you selected. A single example is provided here. Other options will be described in context, in the relevant documentation scenarios.

Example when creating geometry

Let's take an example such as creating a line. The values of the elements you are sketching appear in the **Tools Palette** as you move the cursor. In other words, as you are moving the cursor, the Length (L) and Angle (A) fields display the coordinates corresponding to the cursor position.

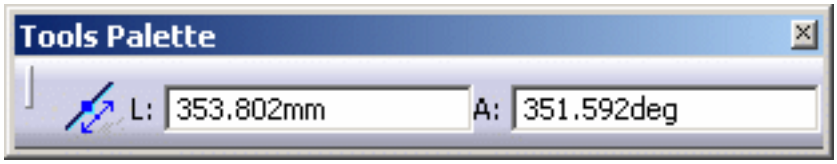
The Horizontal (**H**) and Vertical (**V**) fields are optionally displayed, depending on whether the Show H and V fields in the **Tools Palette** option is selected in **Tools > Options > Mechanical Design > Drafting > Geometry** tab.



You can also use these fields for entering values of your choice. In the following scenario, you are going to sketch a line by entering values in the appropriate fields.

1. Click the **Line** icon from the **Geometry Creation** toolbar.

The **Tools Palette** displays information on value fields.



2. Enter the length (L) of the line.
3. Enter the value of the angle (A) between the line to be created and the horizontal axis. The line is created with the specified values.

Copying, Cutting, Pasting and Deleting

The 2D Layout for 3D Design workbench lets you perform copy, cut, paste and delete operations. However, there are a number of things that you need to keep in mind when performing such operations.

In this task, you will find information on the following subjects:

- Copying, cutting and pasting elements
- About copying, cutting and pasting views
- About copying, cutting and pasting 2D components
- Copy, cut and paste restrictions
- Deleting elements
- Delete restrictions

Copying, cutting and pasting elements

1. Select the element you want to cut or copy.
2. To copy, you can either:
   - click the **Copy** icon ,
   - select the **Edit->Copy** command,
   - select the **Copy** command in the contextual menu.

   This places what you copy in the clipboard.
3. To paste, you can either:
   - click the **Paste** icon ,
   - select the **Edit->Paste** command,
   - select the **Paste** command in the contextual menu.

About copying, cutting and pasting views

Remember the following points when copying, cutting and pasting views:

- A pasted view has the same type, view plane definition, 2D position and associated view box as its original view.
- Copying, cutting and pasting a multi-selection of design views keeps the parent-child relationship of the original views. Therefore, if you multi-select and copy-cut-paste views which belong to a single view set, then the resulting pasted views also belong to a single view set.
  However, if you select and copy-cut-paste individually views which belong to a single view set, then the resulting pasted views belong to independent view sets. As a result, the parent-child relationship and links of the original views are not kept.
- When pasting to another layout a view to which view a filter is applied, the filter is not copied. Refer to Creating View Filters for more information.
- Views cannot be copied/cut from a layout and pasted to a drawing document.
- Views cannot be copied/cut from a drawing document and pasted to a layout.

About copying, cutting and pasting 2D components

Remember the following points when copying, cutting and pasting 2D components:

- When cutting a 2D component which is used as an instance, a warning message is displayed prompting you to confirm the operation.
- 2D components can only be pasted to a layout detail sheet.
- 2D components can be copied/cut from a layout and pasted to any detail sheet of any layout (whether in the same document or not).
- 2D components cannot be copied/cut from a layout and pasted to a drawing document.
- 2D components cannot be copied/cut from a drawing document and pasted to a layout.

Copy, cut and paste restrictions

The following restrictions apply to copy, cut and paste operations in 2D Layout for 3D Design:

- Main views and background views cannot be cut, copied or pasted.
- Layout features (in the specification tree) cannot be cut, copied or pasted.
- You cannot copy, cut and paste elements from the 2D Layout for 3D Design workbench to the Drafting workbench, and vice-versa.
- You cannot copy, cut and paste views from the 2D Layout for 3D Design workbench to the Part Design workbench.
  However, you can copy, cut and paste view sub-elements that are valid in a sketch (such as 2D geometry) from 2D Layout for 3D Design to Part Design.
- You cannot copy, cut and paste sketch sub-elements from Part Design to 2D Layout for 3D Design.

Deleting elements

1. Select the element you want to delete.
2. You can either:
   - right-click and select **Delete**,
   - select the **Edit -> Delete** command,
   - press the Del key.

Delete restrictions

The following restrictions apply to delete operations in 2D Layout for 3D Design:

- Layout features (in the specification tree) cannot be deleted.
- In the case of a layout containing a single sheet, this sheet cannot be deleted.

Layout Creation and Edition

Create a layout: Enter the 2D Layout for 3D Design workbench, create a layout and the related part.

Open a layout: Open, in a 3D window, a part document containing a layout, and then open the layout in a 2D window.

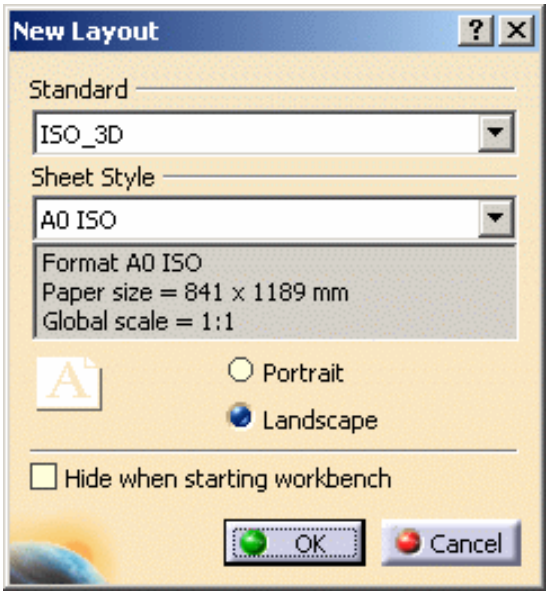
Navigate between windows: Switch back and forth between the 2D and 3D windows.

Creating a Layout

In this task, you will learn how to enter the 2D Layout for 3D Design workbench, create a layout and the related part.

1. Select **Start -> Mechanical Design** from the menu bar.
2. Select the **2D Layout for 3D Design** workbench.

   The **New Layout** dialog box is displayed, allowing you to choose a standard, a sheet style and an orientation for your new layout. Among other things, the sheet style defines the sheet format, paper size, scale and default orientation.



3. Select the **JIS_3D** standard.

   Standards and sheet styles are defined by the administrator in the Standards Editor, who can add an unlimited number of them. Note that any customized standard is based on one of the four international standards (ANSI, ISO, ASME or JIS) as far as basic parameters are concerned. For more details, see Sheet Format Definition and Sheet styles in the Administration Tasks chapter.

   In the **New Layout** dialog box, standards suffixed with _3D are specifically designed for 2D Layout for 3D Design layouts. For example, colors have been customized for optimized display. For more information, refer to Layout Views Customization in the Administration Tasks chapter.

4. Select the **A4 JIS** sheet style.
5. Optionally change the default orientation from **Landscape** to **Portrait**.
6. If you do not want the New Layout dialog box to appear the next time you enter the 2D Layout for 3D Design workbench via the **Start** menu, select the **Hide when starting workbench** option.

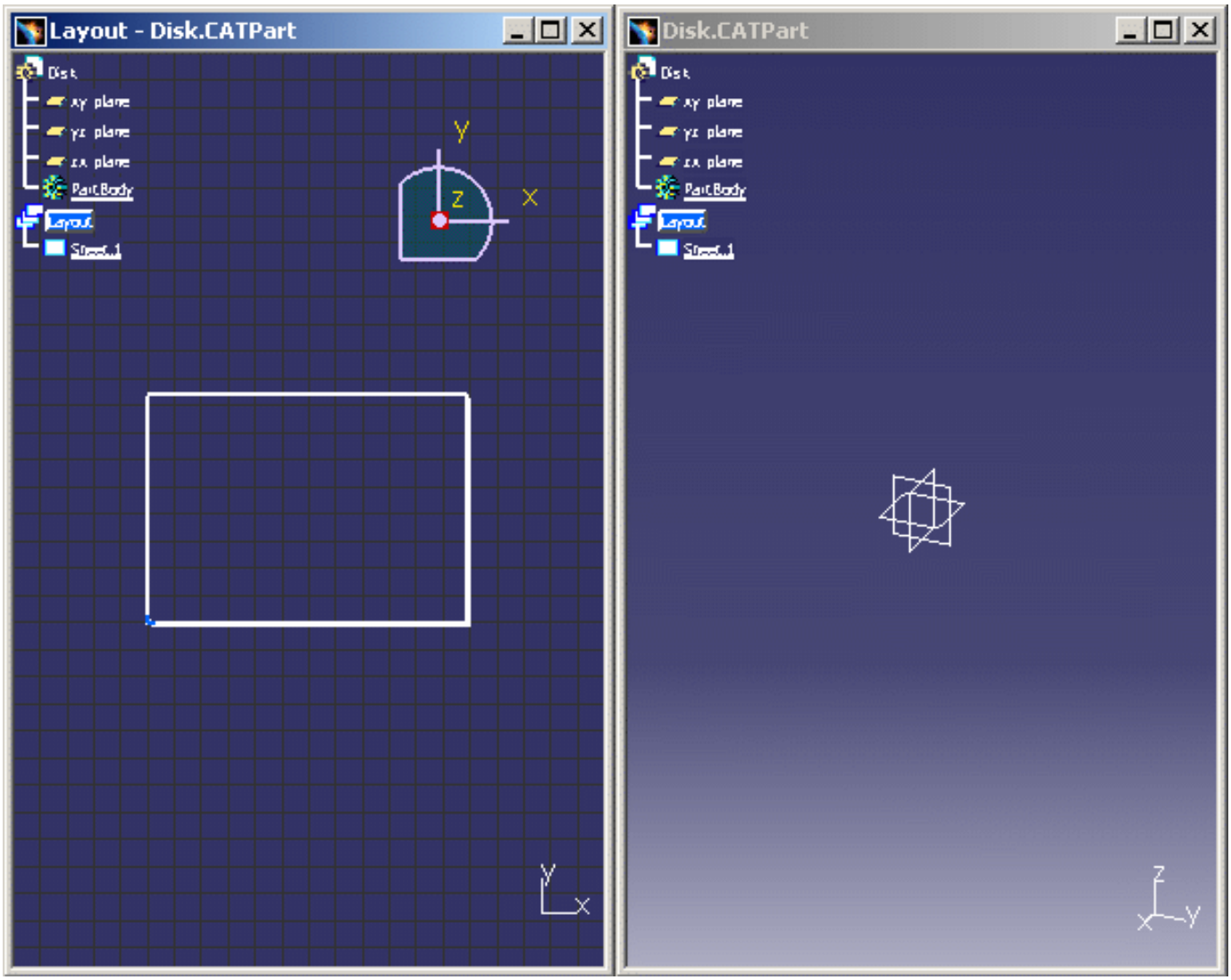
   In this case, the last selected standard, sheet style and orientation will be used by default when creating a layout. You will always be able to reactivate this dialog box by unselecting the **Hide when starting workbench** option available through **Tools -> Options -> Mechanical Design -> Drafting -> General** tab.

7. Click **OK**.

   The **New Part** dialog box is displayed.

8. Enter a name for the part that will be associated to your layout (Disk, for example) and click **OK**.

An empty sheet is created in a specific 2D window, and the associated part document is created and opened in a 3D window.



- The sheet appears in the specification tree (under the Disk.CATPart item) both in the 2D and 3D windows. Pressing the F3 key lets you show or hide the specification tree as desired.
- The creation of a layout cannot be undone.
- Local transformations are planar, which means that the sheet cannot be rotated in 3D.
- By default, the background is blue and not graduated in the 2D window, which enables you to differentiate it from the 3D window which is blue and graduated.
- Refer to Navigating Between Windows to learn how to switch back and forth between the 2D and 3D windows.

About standards

- At any time after defining a sheet, you can change the standard (which you can update), sheet style or orientation. Refer to Modifying a Sheet for more information.
- Only one standard can be associated to a given part, which guarantees standard homogeneity within the part. This may have consequences in the case of a part containing a mix of layout elements and 3D annotations (Functional Tolerancing and Annotation elements):
  - The standard selected when creating a layout also becomes the standard used for 3D annotations. If 3D annotations previously used a different standard, this implies that:
    - the visualization of some of these annotations may change.

  - you cannot use an older standard for the layout (for example, you cannot use a standard created in R15 for the layout if 3D annotations used a standard created in R16).

- Likewise, when creating a 3D annotation for the first time in a part which already contains a layout, the standard used for 3D annotations also becomes the standard used for the layout. This implies that if the layout previously used a different standard, the visualization of some layout elements may change.
- Changing the standard of a layout also changes the standard used for 3D annotations, and vice-versa.

To prevent confusion, administrators are free to provide one standard only to their users.

As a reminder, the standard used for a part is defined by the **Default standard at creation** option available in **Tools -> Options -> Infrastructure -> 3D Annotations Infrastructure -> Tolerancing** tab.

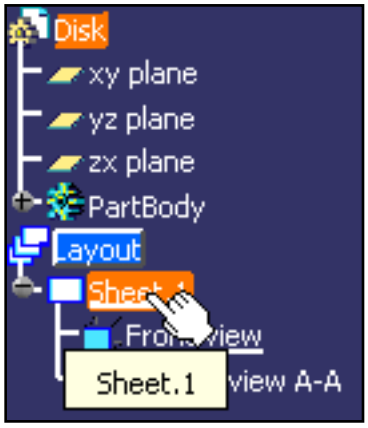
Opening a Layout

In this task, you will learn how to open a part document containing a layout in a 3D window, and then open the layout in a 2D window.

1. Select **File -> Open** from the menu bar. The **File Selection** dialog box is displayed.
2. Browse to select your existing part document. For example, go to the online\lo1ug_C2\samples folder (in the documentation installation folder) and select a part document, such as Disk4.CATPart.
3. Click the **Open** button. The part document appears in the 3D window.

   To visualize the part, you need to zoom out.

4. To open the layout in the 2D window, you can either:
   - Select **Start -> Mechanical Design -> 2D Layout for 3D Design** from the menu bar.
   - Double-click the Sheet.1 feature from the specification tree.



The 2D window with its layout is now open alongside the 3D window. It is the active window.

Refer to Navigating Between Windows to learn how to switch back and forth between the 2D and 3D windows.

Navigating Between Windows

In this task, you will learn how to switch back and forth between the 2D and 3D windows.

Open the Disk4.CATPart document. Select **Start -> Mechanical Design -> 2D Layout for 3D Design** to open the layout in the 2D window, alongside the 3D window.

You can use several methods to switch back and forth between the 2D and 3D windows, such as:

- Directly clicking the window you want to use.

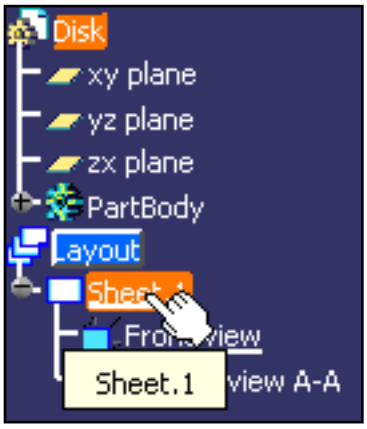
  The behavior may depend on your system requirements.

  To keep multi-selected elements while switching from the 3D to 2D window, trap the selection in the 3D window and then either:
  - click the 2D window title bar with the left mouse button.
  - click the 2D window with the middle mouse button.

  This method lets you use the selection to create use-edges, for example.

- Selecting the window you want to use from the **Window** menu.

- Double-clicking a specific feature from the specification tree: for example, PartBody to switch to the 3D window, or Sheet.1 to switch to the 2D window.



Layout Sheets

Edit a sheet and/or its background: edit a sheet, as well as the background of a sheet.

Modify a sheet: change the standard, sheet style and orientation of a layout sheet; update the standard of a sheet; insert the background view from a drawing sheet.

Add a sheet to a layout: add a sheet to an existing layout.

Set a sheet as current: set a sheet as current, in a layout which contains several sheets.

Editing a Sheet and/or its Background

In this task, you will learn how to edit a sheet, as well as how to edit the background of a sheet.

Editing a sheet sets it as the current one (if necessary) and activates the main view, which supports the geometry directly created in the sheet.

Editing a sheet background sets the sheet as the current one (if necessary) and activates the background view, which is dedicated to frames and title blocks and to the instantiation of 2D components.

Open the Disk4.CATPart document. Select **Start -> Mechanical Design -> 2D Layout for 3D Design** to open the layout in the 2D window. The main view is currently active.

1. To activate the background view, use one of the following methods:
   - From the specification tree, right-click the sheet (Sheet.1) and select **Edit Sheet Background**.
   - Select **Edit -> Background**.

   The background view is activated. You can start adding a frame and a title block, or instantiating 2D components.

2. To go back to the main view, use one of the following methods:
   - From the specification tree, right-click the sheet (Sheet.1) and select **Edit Sheet Working Views**.
   - From the specification tree, double-click the sheet (Sheet.1) or a view (in this case, the selected view will be activated).
   - Select **Edit -> Working Views**.

   The main view is activated. You can add geometry, dimensions, annotations, dress-up and so on.

Modifying a Sheet

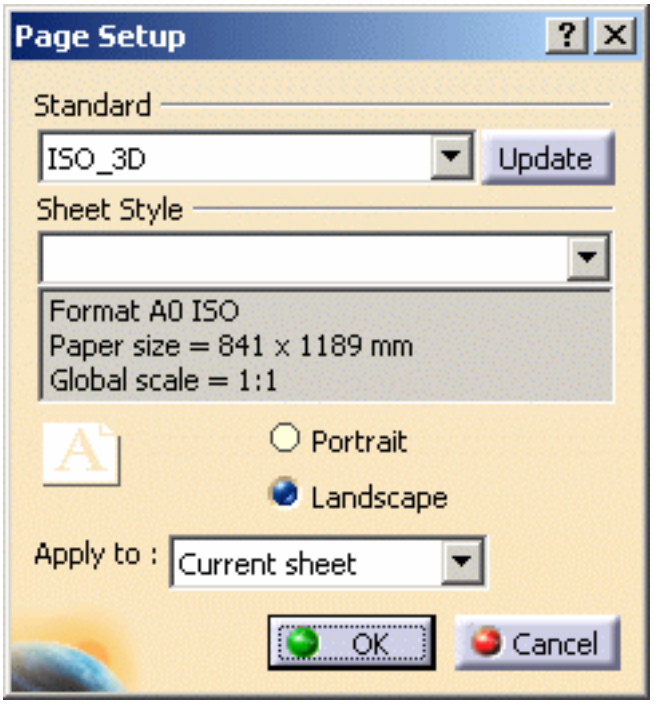
In this task, you will learn how to:

- change the standard, sheet style and orientation of a sheet. This overrides the options you selected in the **New Layout** dialog box when creating the layout.
- update the standard (in the case the current standard file is modified).

Open the Disk4.CATPart document. Select **Start -> Mechanical Design -> 2D Layout for 3D Design** to open the layout in the 2D window.

Changing the standard, sheet style and orientation of a sheet

1. Select **File -> Page Setup** from the menu bar. The **Page Setup** dialog box is displayed.



2. Select the **JIS_3D** standard. A message informs you that this action cannot be undone.

   The parameters of the chosen standard are copied into the drawing and replace the previous parameters. This may have an immediate impact on the appearance of the elements inside the drawing.

- Styles are not affected by this change, that is styles in this standard file that are different from the previous standard file will not be re-applied to existing elements. Indeed, styles are applied when creating elements (as they define the default values to be used for creation). If needed, style parameters can be re-applied to an element using the Style toolbar: simply select the element whose style you want to update and select the updated style in the Style toolbar.

- Sheet styles are re-applied to existing sheets when you are switching to another standard.

3. Click **OK** to continue. The **A0 JIS** sheet style is automatically selected. Among other things, the sheet style defines the sheet format, paper size, scale and default orientation.

4. Optionally choose another sheet style.

Another way to change the sheet style (also called format) is through the **Properties** dialog box: to open it, right-click the sheet, and select **Properties**.

5. Optionally change the default orientation from **Landscape** to **Portrait**.

6. Click **OK** to validate and exit the dialog box. The sheet is modified accordingly.

- Only one standard can be associated to a given part, which guarantees standard homogeneity within the part. In the case of a part containing a mix of layout elements and 3D annotations (Functional Tolerancing and Annotation elements), changing the standard of a layout also changes the standard used for 3D annotations. Refer to the About standards section in Creating a Layout for more information.
- You cannot replace a standard by a older one (for example, you cannot replace a standard created in R16 by a standard created in R15).

Updating the standard

Performing this task requires that your standard file has been modified by the administrator.

When a standard file is modified by the administrator, there is no automatic update of the sheets which use this standard. Each sheet contains a copy of the standard it uses, and retains this version until you explicitly update this copy or change the standard as explained previously.

1. Select **File -> Page Setup**. The **Page Setup** dialog box is displayed.

2. If your standard file has been modified by the administrator, click the **Update** button. A message informs you that this action cannot be undone.

3. Click **OK** to continue. The most recent version of the standard file is embedded in the sheet, thus reflecting the latest changes performed by the administrator.

- The most recent version of the updated standard is copied into the layout sheet and the previous standard parameter values are replaced by the latest ones, reflecting the latest changes an administrator or user may have performed in the standard file. This may have an immediate impact on the appearance of the elements in the sheet.

- Note that styles are not affected by this update, that is styles modified in the updated standard file will not be re-applied to existing elements. Indeed, styles are applied when creating elements (as they define the default values to be used for creation). If needed, new style parameters can be re-applied to an element using the Style toolbar: simply select the element whose style you want to update and select the updated style in the Style toolbar.

4. Click **OK** to validate and exit the dialog box. The sheet is modified accordingly.

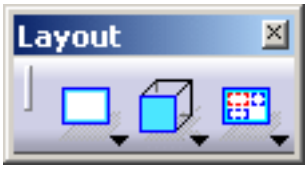
Adding a New Sheet to a Layout

In this task, you will learn how to add a new sheet to an existing layout.

Open the Disk4.CATPart document. Select **Start -> Mechanical Design -> 2D Layout for 3D Design** to open the layout in the 2D window.

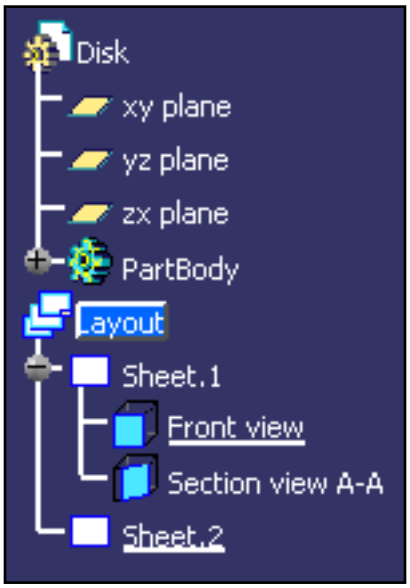
To add a sheet to a layout, use one of the following methods:

- Click the **New Sheet** icon from the **Layout** toolbar.



- Select **Insert -> Layout -> Sheets -> New Sheet**.

The new sheet, Sheet.2, is created and automatically set as the current one. It is listed in the specification tree.



Setting a Sheet as Current

In this task, you will learn how to set a sheet as current, in a layout which contains several sheets. Setting a sheet as current means defining the sheet as the current one, which restores the last active view of the sheet.

Open the Disk4.CATPart document. Select **Start -> Mechanical Design -> 2D Layout for 3D Design** to open the layout in the 2D window. Add a sheet to the layout. The newly created sheet is automatically set as the current one.

To set a sheet (Sheet.1 in our example) as current, use one of the following methods:

- From the specification tree, right-click the sheet you want to set as current, and either:
  - select **Set As Current Sheet**.
  - select **Sheet.X object -> Definition**.
- From the specification tree, double-click the sheet you want to set as current.

The selected sheet, Sheet.1 in our example, is set as the current one. You can start working on it.

View Creation

Before you begin: learn about the basic concepts behind view creation in 2D Layout for 3D Design.

Create a projection view: create a projection view in an existing layout.

Create a section/auxiliary view: create a section view, a section cut or an auxiliary view.

Create a section from 2 planes: create two aligned/offset section views or section cuts using two existing 3D planes as supports.

Create a view from another element: create a view from an existing view, a 3D plane or a Functional Tolerancing and Annotation view.

Before You Begin Creating Views

Before you begin creating views, you should be familiar with some important concepts:

- Layout views
- View box
- Primary views
- View set
- Isometric views

Layout Views

In 2D Layout for 3D Design, a sheet contains a set of views:

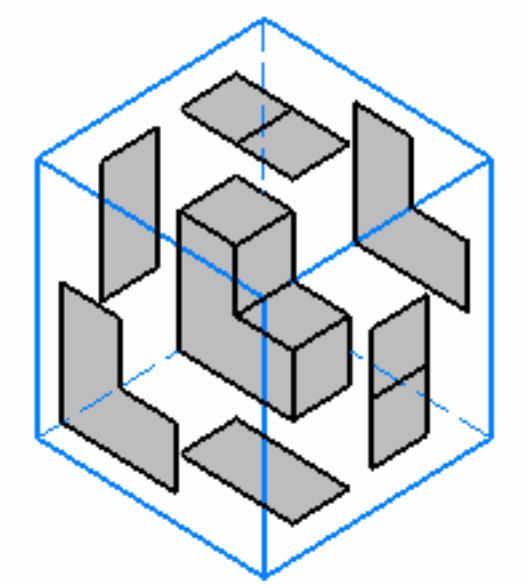
- design views: views in which you design and whose content can be visualized and output in 3D context.
- isometric views: views visualized only in the 2D Layout for 3D Design workbench (not in a 3D window).

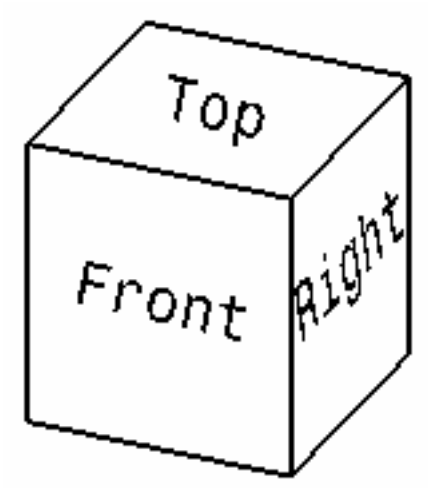
View Box

Defining layout views is an important step in the layout design process. As a designer, you need to have a general idea of the overall dimensions of your design, as well as of its position in space.

The data needed to fully define the layout of a view set in the 2D window, as well as the position of each view in the 3D space, is defined in the standards, using a "view box". This data is made up of:

- Primary view type
- 3D axis
- View box anchor point
- 3D axis origin
- View box overall dimensions





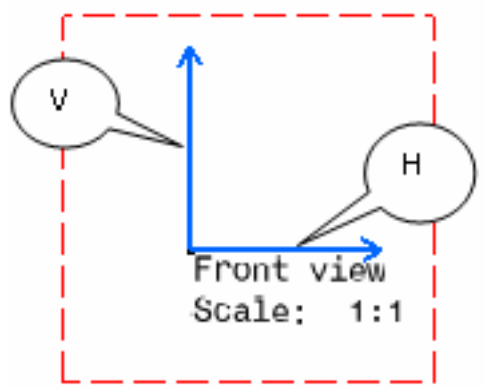
A single, default, view box is defined for each standard (for example, there is a single view box for the ISO_3D standard). The standard definition is retrieved when creating the first view of a view set: the view box definition is associated to each view set. Therefore, an update of the standard does not impact existing views, or views added to an existing view set.

For more information, refer to Layout Views Customization in the Administration Tasks chapter.

3D axis

The view plane definition corresponds to the specific position of design views in the 3D space. It is defined in the standard.

In 2D Layout for 3D Design, coordinates are always expressed as local coordinates (H, V). Local coordinates are transformed to be expressed in 3D space using the view plane definition.

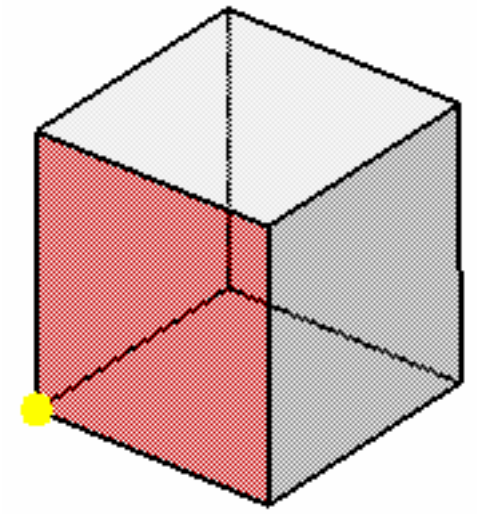


The 3D axis gives the primary view orientation in space. Its first direction corresponds to the view local direction H while its second direction corresponds to the view local direction V. The third direction is deduced from the H and V directions in order to form an axis system.

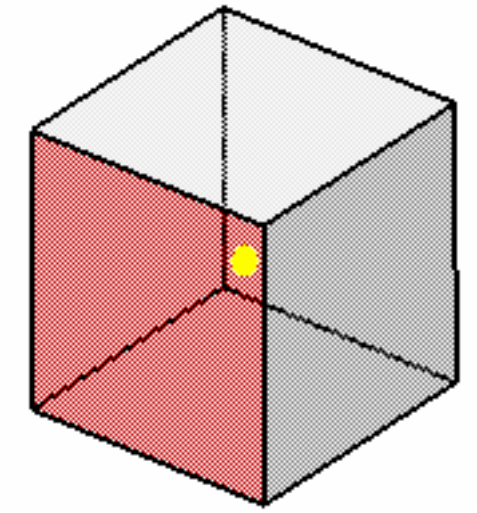
View box anchor point

The view box anchor point is the 3D point from which the view box position in 3D space is defined. It is defined in the standard. Two anchor points are available (the primary view is displayed in red in the images below):

- At the bottom left corner of the view box.



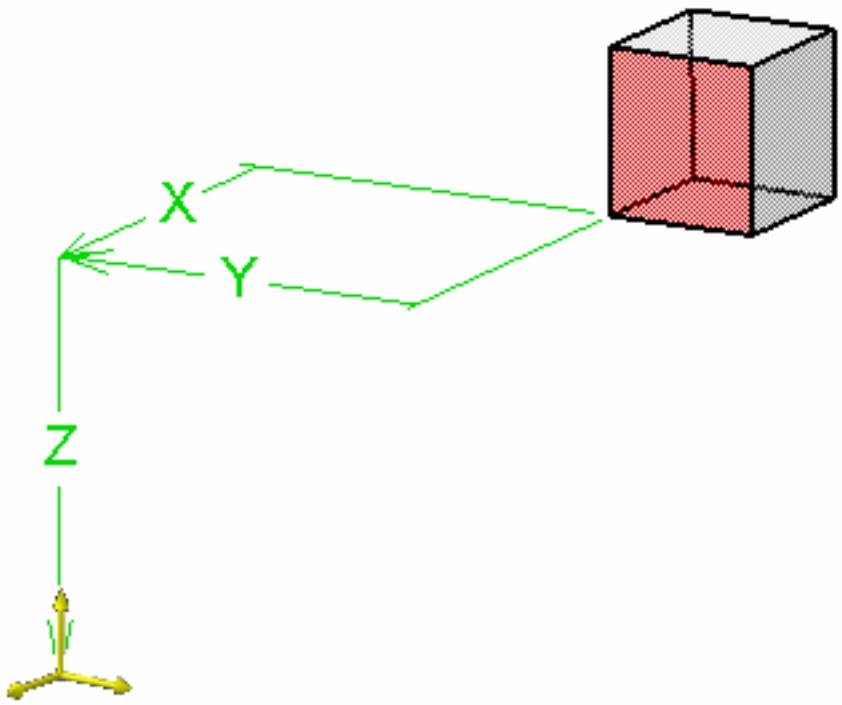
- At the center of the view box.



3D axis origin

The 3D axis origin corresponds to the position of the view box's anchor point in 3D space.

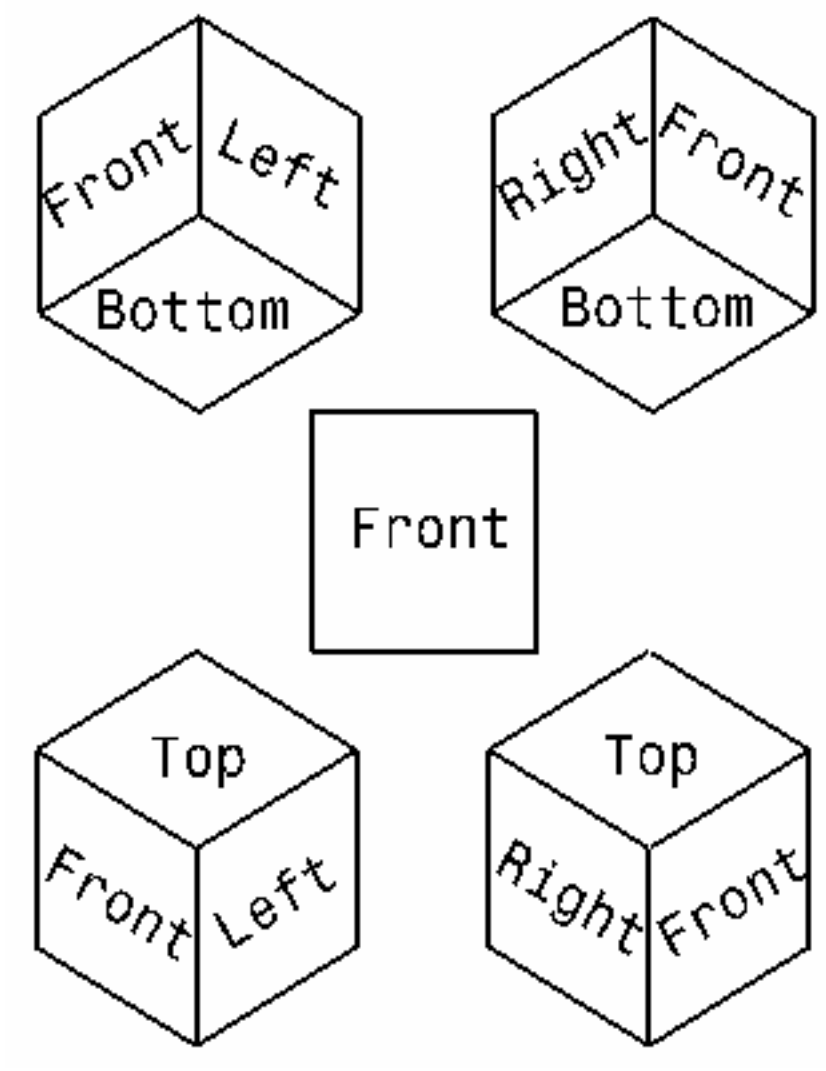
In the example shown here, the anchor point is placed at the bottom left corner of the view box.



Specifying this position is particularly important when the part is designed in an assembly (as for a rear bumper). When the part is designed in order to be multi-instantiated in the assembly, this position is generally null (as for wheels).

The 3D axis orientation of projection views, isometric views, section views, section cuts and auxiliary views depends on the following factors:

- The active view 3D axis orientation
- The position of the cursor on the layout
- The projection method

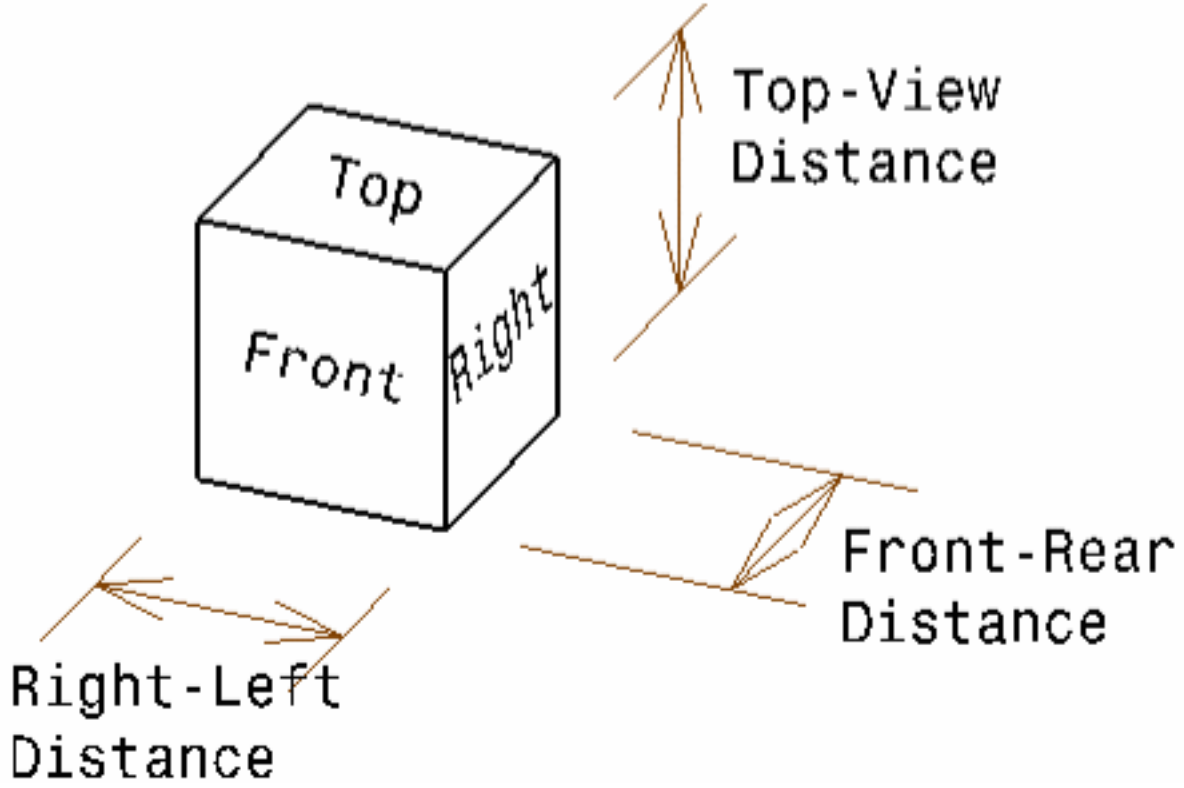


View box overall dimensions

The overall dimensions are as follows:

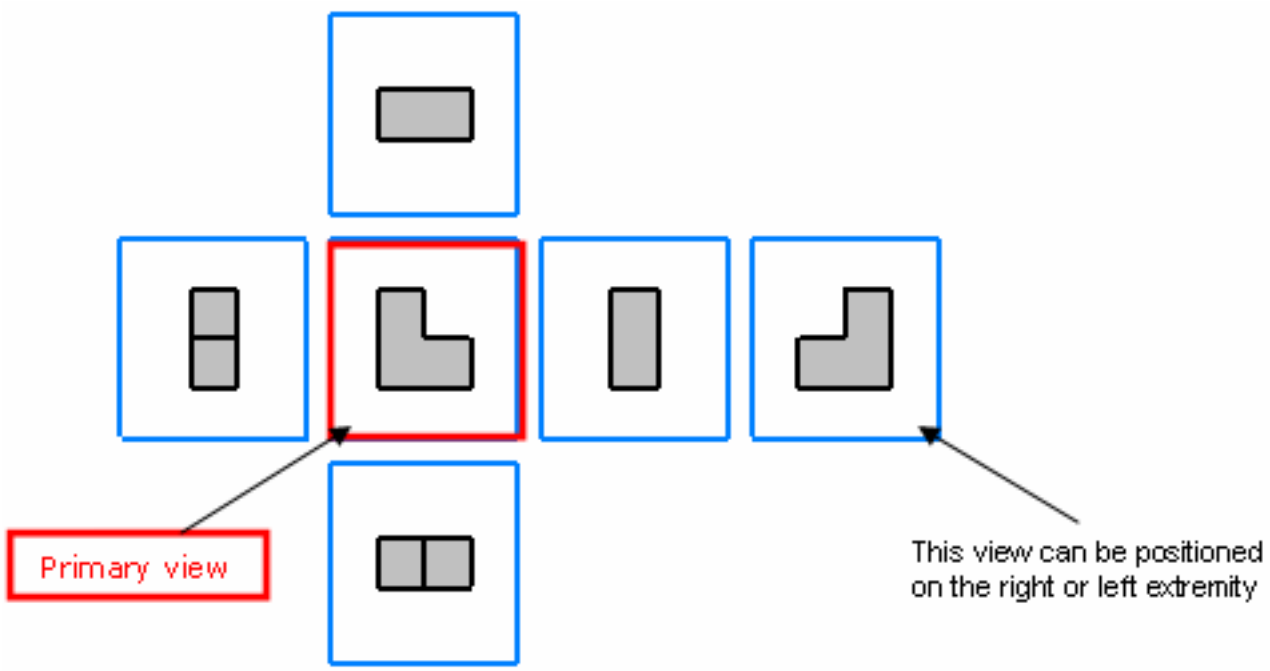
- The distance between the Front and Rear views
- The distance between the Right and Left views
- The distance between the Top and Bottom views

These dimensions define approximately the size of the design. They can be either smaller or larger than the actual design size.



Primary Views

Primary views are design views which have no parent view and set the distribution of their children views, grand-children views, and so on.



The primary view type can be any projection view type (Front, Left, Right, Top, Bottom or Rear) and it is determined by the standard file embedded in the part document. There is no associative property between the primary view type and the standard. Therefore, the type of existing primary views will not be changed when updating the embedded standard with a standard which has another primary view type.

The name of a primary view corresponds to its view type. For example, a primary view of the left type is named "Left view". The specification tree does not distinguish primary views from other views. (To know which view is the primary one, you may use knowledge capabilities).

Layout sheets may have several primary views of different types.

A view box is associated to each new primary view. Thus, design views created from primary views are created according to their associated view boxes and not from the standard's definition, which keeps the position of linked views in 3D space coherent.

View Set

A view set can be associated to an independent view box. This is what happens when you create a primary view, for example. However, a view set does not have an associative view box when it is initiated from a view from 3D plane creation.

It is not possible to create several projection views of a same type within a set of views (two Right views, for example). If needed, you can either start a new view set (that is create a new primary view) or create an auxiliary view from the related view. However, it is possible to create several isometric views or several section views/cuts.

Existing view sets are not impacted by changing standards as they are linked to independent view boxes, if any.
If you need to create views according to a view box different from the one stored in the document's standard, then you first need to switch to a standard containing the new definition, and finally to start a new view set.

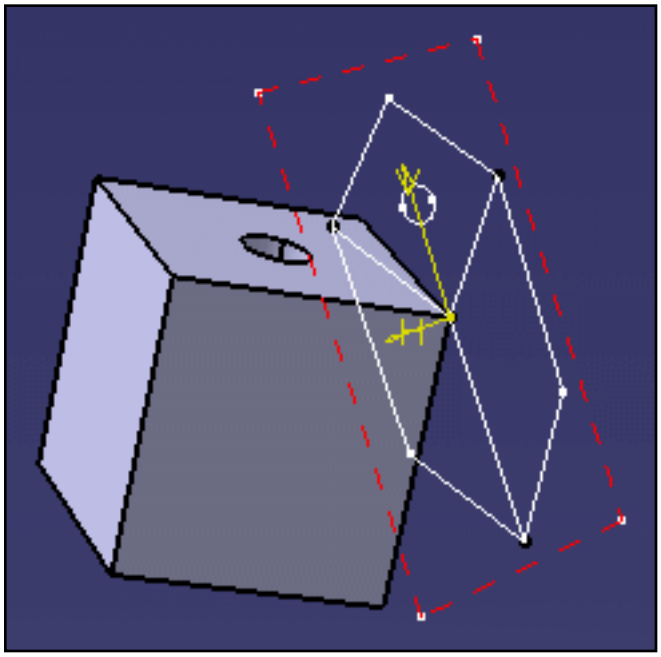
Existing view sets can still be extended after a standard update. The definition of new projection views is found from the view box associated to the set of views.

When defining the view box, you can invert the naming of the Left and Right views if you want the Right view to be called Left view, and vice-versa. This only inverts the name (not the type) of the views.

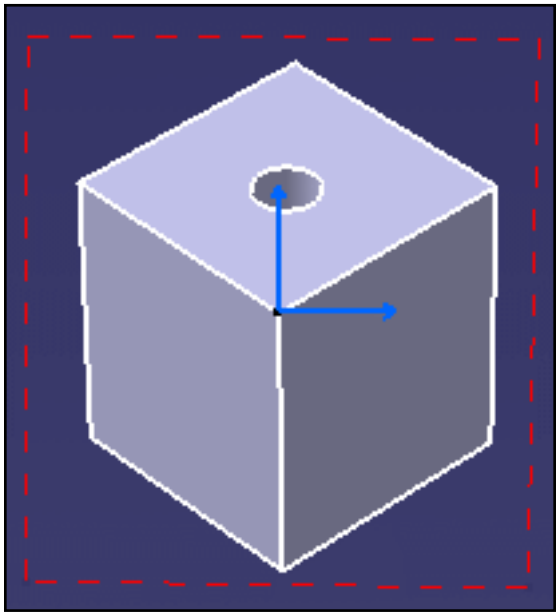
Isometric views

Isometric views differ from projection views to a certain extent. In 3D space, isometric views would be located at the corners of the view box. However, they are not visualized outside the 2D Layout for 3D Design workbench.

The example here shows the positioning of an isometric view in 3D space.



The purpose of isometric views is to check by transparency the validity of the 3D design, that is to compare the result of a part creation (from one or several design views) with the expected original design.



Creating a Projection View

In this task, you will learn how to create a projection view in an existing layout.

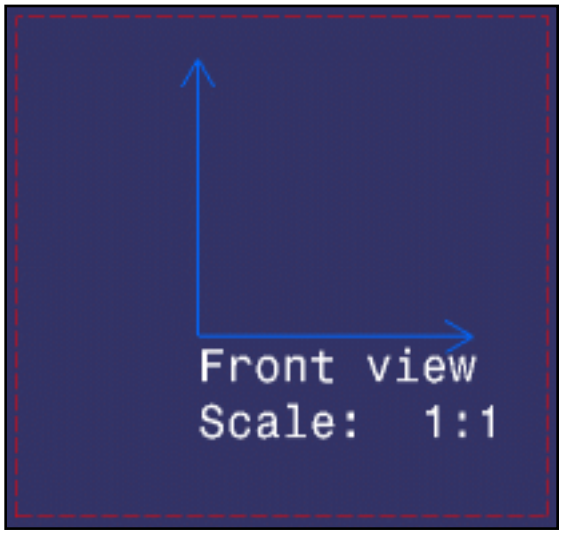
Open the Disk.CATPart document. Select **Start -> Mechanical Design -> 2D Layout for 3D Design** to open the empty layout in the 2D window, alongside the 3D window.

1. Click the **New View** icon in the **Layout** toolbar.
2. Click on the sheet to position the new view.

Note how the view is previewed in the part window. You need to zoom out, as the view box defined in the ISO_3D standard used by the current layout has sides of 1000mm. For more information on the standards, see Administration Tasks.

An empty primary view is created, displaying a blue axis in a red frame, as well as the view name and scale. Additionally, the Front View item is added to the specification tree.

In our scenario, the primary view is a front view. The view type for the primary view is defined in the current standard (ISO_3D in our scenario).



You can now create 2D geometry in your view or continue creating more projection views as detailed below.

3. Click the **New View** icon again and click in the layout at the location where you want the

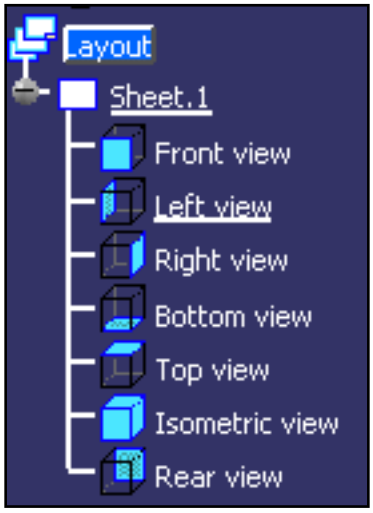
projection view to be positioned: you can create Left, Right, Top, Bottom and Isometric views.

Create a Left view, for example.

- Projection views are always created from the active (current) view.
- Positioning the view defines the projection view direction, in accordance with the projection method and the primary view type (a front view in our scenario).
  The projection method (First angle standard or Third angle standard) is defined by the sheet style, as specified in the standard used by the layout. For more information, refer to Sheet Styles in the Administration Tasks chapter. You can change the projection method by editing the layout sheet properties (through **Edit > Properties**).
- Remember that it is not possible to create several projection views of a same type within a set of views (two Right views, for example).
- Projection views are always created using the sheet scale. The primary view scale, when it is different from the sheet scale, is not propagated to other projection views.
- If you change the sheet scale (defined in the sheet properties), the scale of all existing views (defined in the view properties) is multiplied by that of the sheet (for example, if existing views already have a scale of 1/10, and if you change the sheet scale to 1/10, then existing views will now have a scale of 1/100).

The view is created. The specification tree is updated again to show the newly created view.

4. Double-click the **New View** icon to create several projection views one after the other: this time, create Right, Top, Bottom and Isometric views. The specification tree is updated again.
5. Press Escape when you have created your views.
6. Activate one of the projection views by double-clicking it. For example, activate the Left view.
7. Click the **New View** icon again: this time, you can create Rear or Isometric views from the left view. Create a rear view, for example. The specification tree is updated again.



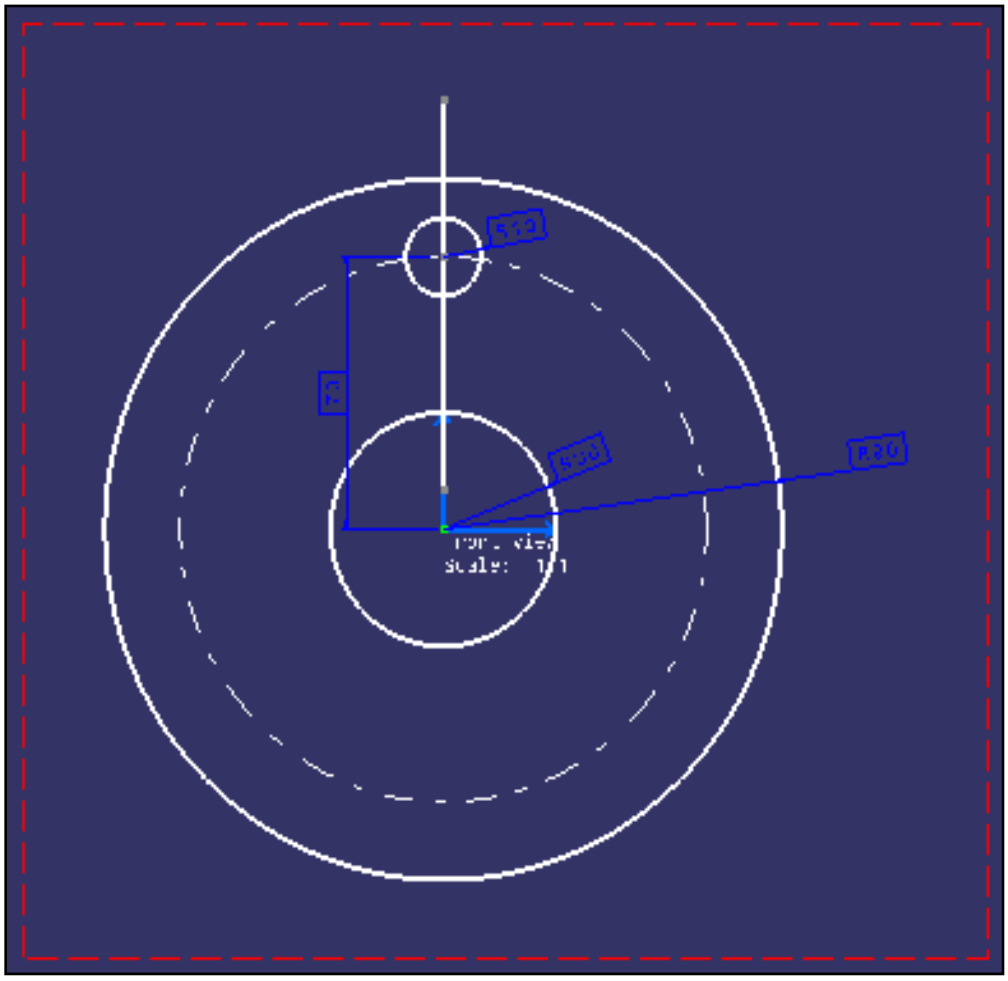
Creating a Section/Auxiliary View

In this task, you will learn how to create a section view, a section cut or an auxiliary view in an existing layout.

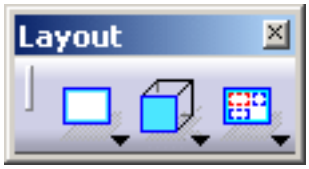
Open the Disk2.CATPart document. Select **Start -> Mechanical Design -> 2D Layout for 3D Design** to open the layout in the 2D window, alongside the 3D window.

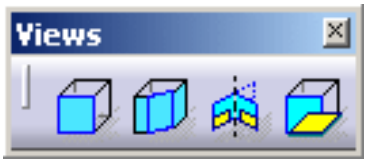
1. Click the **Line** icon in the **Geometry Creation** toolbar.
2. Use the vertical axis to define the cutting profile as shown below, and double-click to end the line creation.

   You could also draw a profile consisting in multiple segments to create an aligned section view/cut or an offset section view/cut.



3. Click the **New Section/Auxiliary View** icon in the **Layout** toolbar (**Views** sub-toolbar).





4. Select the line you have just created as the cutting profile.

You cannot select geometry which does not belong to the active view.

The **Tools Palette** is automatically displayed with the following options:

- **Auxiliary View**: creates an auxiliary view.
- **Section View**: creates a section view.
- **Section Cut**: creates a section cut.

These options are also available from a contextual menu.

5. Select the **Section View** icon .
6. Click on the sheet at the location where you want the section view to be positioned.

- Positioning the view defines the section view direction, in accordance with the projection method.
  The projection method (First angle standard or Third angle standard) is defined by the sheet style, as specified in the standard used by the layout. For more information, refer to Sheet Styles in the Administration Tasks chapter. You can change the projection method by editing the layout sheet properties (using **Edit > Properties**).
- Section/auxiliary views are always created using the sheet scale. The primary view scale, when it is different from the sheet scale, is not propagated to other section/auxiliary views.
- If you change the sheet scale (defined in the sheet properties), the scale of all existing views (defined in the view properties) is multiplied by that of the sheet (for example, if existing views already have a scale of 1/10, and if you change the sheet scale to 1/10, then existing views will now have a scale of 1/100).

Note how the view is previewed in the part window. You need to zoom out, as the view box defined in the ISO_3D standard used by the current layout has sides of 1000mm. For more information on the standards, see Administration Tasks.

An empty section view is created, with its plane perpendicular to the active view. Additionally, the Section view item is added to the specification tree.

7. Double-click the section view to activate it. You can now create 2D geometry in this view.

More about creating a section/auxiliary view

Section views, section cuts or auxiliary views are created using an existing 2D profile (a single line or a multi-segment profile) in the active view. Such views are based on a plane perpendicular to the active view which contains the directional element (the selected line). The projection direction depends on two factors:

- the cursor location
- the projection method

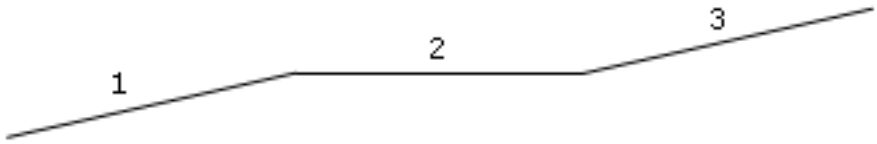
Once created, the new view is not associative to the directional element, and this element is not transformed into a callout. Therefore, it is impossible to edit section profiles after the view has been created.

Section views, section cuts or auxiliary views are positioned according to their reference view. Therefore, moving a reference view also moves its linked views.

The case of multi-segment profiles

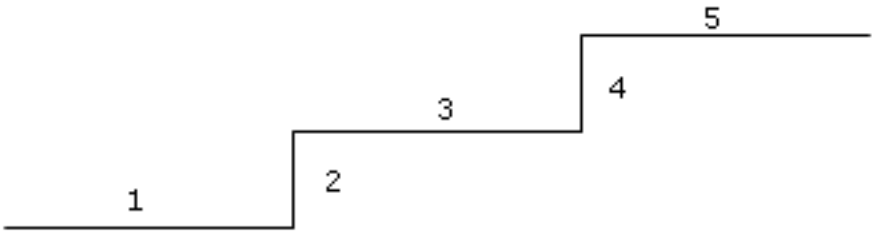
Selecting a profile made of multiple segments lets you create several section views/cuts in one shot. There are two possible scenarios:

- Aligned profile:



When selecting such a profile, a view is created for each segment.

- Offset profile:



When selecting such a profile, a view is created for each segment parallel to the first one (for

example, 1; 3; 5).

When creating views from a multi-segment profile, remember the following points:

- Make sure you select the first segment in the profile (the first-created one) if you want to create section views/cuts. If you do not select the first segment, then only the selected segment is taken into account (and not the whole profile), and an auxiliary is created.
- The views share the same origin in the layout sheet, and their local axes (H and V) overlap.
- The first view of a multi-segment profile has folding lines corresponding to each segment, and the other views only have folding lines corresponding to the current segment.
- The name of each view is suffixed to indicate the view's rank.
- Only section views and section cuts can be created from a multi-segment profile.
- The first segment of a multi-segment profile defines the section orientation.

Creating a Section From Two Planes

In this task, you will learn how to create two section views or section cuts using two existing 3D planes as supports. These 3D planes can be defined in the same document or in another document.

Open the Disk4.CATPart document. Select **Start -> Mechanical Design -> 2D Layout for 3D Design** to open the layout in the 2D window, alongside the 3D window.

1. Click the **New Section From 2D Planes** icon in the **Layout** toolbar (**Views** sub-toolbar).
2. Select a plane from the specification tree or from the geometry area (the xy plane, for example).
3. Select another plane from the specification tree or from the geometry area (the yz plane, for example).

   - You cannot select two parallel planes.
   - You could also select a 3D plane created using the **3D Plane** command. For more information, refer to Creating a 3D Plane.
   - Do not select a plane in the 3D window, as this will stop the command. You must select both planes in the 2D window.

   The **Tools Palette** is automatically displayed with the following options:

   - **Section View**: creates a section view.
   - **Section Cut**: creates a section cut.

   These options are also available from a contextual menu.

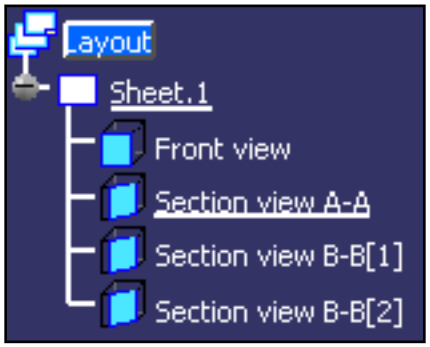
4. Select the **Section Cut** icon.

   Click on the sheet at the location where you want the view to be positioned.

- Section views/cuts are always created using the sheet scale. The primary view scale, when it is different from the sheet scale, is not propagated to other section views/cuts.
- If you change the sheet scale (defined in the sheet properties), the scale of all existing views (defined in the view properties) is multiplied by that of the sheet (for example, if existing views already have a scale of 1/10, and if you change the sheet scale to 1/10, then existing views will now have a scale of 1/100).

Note how the view is previewed in the part window. You need to zoom out, as the view box defined in the ISO_3D standard used by the current layout has sides of 1000mm. For more information on the standards, see Administration Tasks.

Two empty aligned section views are created. Additionally, two new Section view items are added to the specification tree.



You can now create 2D geometry in these views.

More about creating a section from two planes

The two section views or cuts have a number of common characteristics. However, the first-created view has a few characteristics of its own. To enable you to distinguish between the views, their name is suffixed ([1], [2]) to indicate their rank: the suffix [1] indicates the first-created view.

The first-created view has the following characteristics:

- Its origin and normal are identical to those of its related 3D plane.
- Its H axis is on its related 3D plane and oriented towards the intersection of both planes.

The two views share the following characteristics:

- Their V axes are along the intersection of both planes.
- They share the same origin in the layout sheet, their local axes (H and V) overlap.
- They have folding lines which correspond to the intersection of both planes.
- Their axes and origins are not associative to the 3D planes.
- They cannot be linked in position with another view.

Creating a View From Another Element

In this task, you will learn how to create a view from another element in an existing layout:

- creating a view from an existing view
- creating a view from a 3D plane

Open the Disk4.CATPart document. Select **Start -> Mechanical Design -> 2D Layout for 3D Design** to open the layout in the 2D window, alongside the 3D window.

Creating a view from an existing view

1. Click the **New View From** icon in the **Layout** toolbar (**Views** sub-toolbar).
2. Select a view from the specification tree or from the geometry area, the front view for example.

   A new Front view item is added to the specification tree.
3. Click on the sheet at the location where you want the view to be positioned.

- "Views from" are always created using the sheet scale. The primary view scale, when it is different from the sheet scale, is not propagated to other "views from".
- If you change the sheet scale (defined in the sheet properties), the scale of all existing views (defined in the view properties) is multiplied by that of the sheet (for example, if existing views already have a scale of 1/10, and if you change the sheet scale to 1/10, then existing views will now have a scale of 1/100).

Note how the view is previewed in the part window. You need to zoom out, as the view box defined in the ISO_3D standard used by the current layout has sides of 1000mm. For more information on the standards, see Administration Tasks.

An empty front view is created, with the same position in space as the front view from which it was created.

You can now create 2D geometry in this view.

More about creating a view from an existing view

You can use an existing layout view as a template for creating a new layout view, in the same document or another. The newly created view has the same type and position in space as its template view. However:

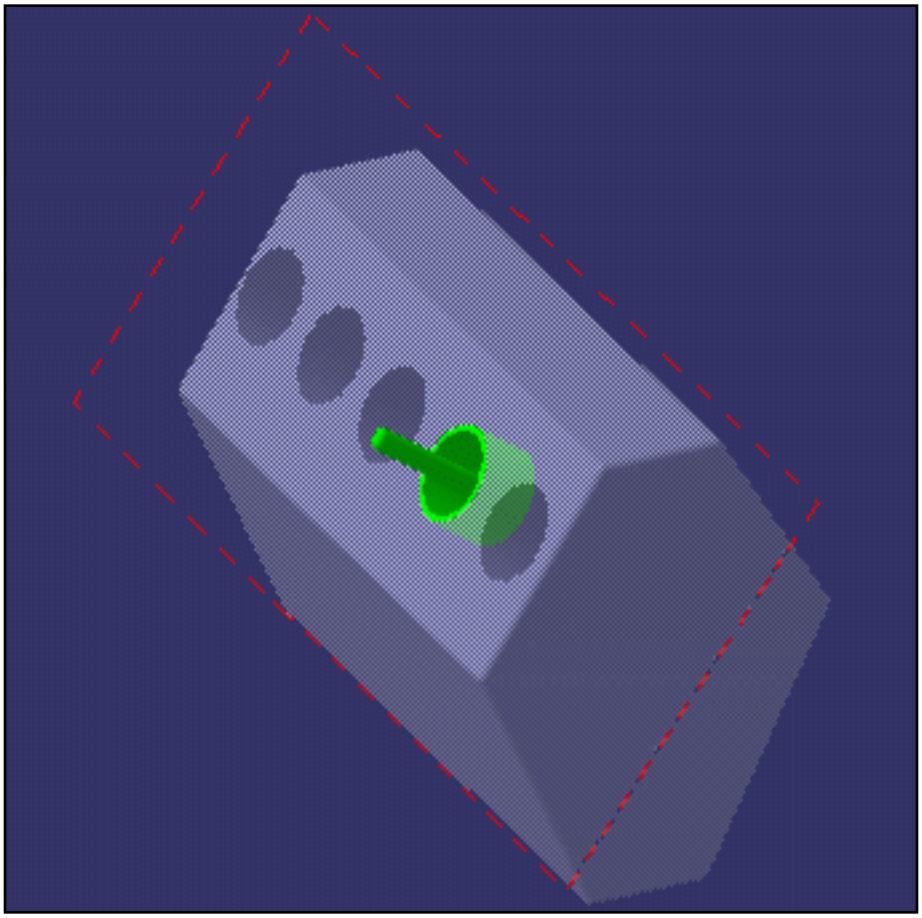
- There is no parent/child relationship between these views.
- The created view cannot be linked in position with another view.
- The view is created empty.
- If the created view is a projection view, then it also features an associated view box which is equivalent to that of the template view. Thus, it is possible to add projection views to this new view set.

Why create a view from an existing view?

When designing a part in context, for example, you may want to re-use the definition of an existing view to design a new part in accordance with an existing one.

Let's take the engine shown here as an example.

In a first step, the designer defined the overall dimensions of the engine. Then, he created an auxiliary view to design the cylinders. To design the pistons, the designer will find it useful to create a view from the view used for the cylinders.



Creating a view from a 3D plane

1. Click the **New View From** icon in the **Layout** toolbar (**Views** sub-toolbar).
2. Select a plane from the specification tree or from the geometry area (the xy plane, for example). A new auxiliary view item is added to the specification tree.

   - You could also select a 3D plane created using the **3D Plane** command . For more information, refer to Creating a 3D Plane.
   - You could also select a view from the Functional Tolerancing and Annotation workbench.

3. Click on the sheet at the location where you want the view to be positioned.

   An empty view is created and added to the specification tree, displaying a blue axis in a red frame, as well as the view name and scale.

   You can now create geometry in this view.

More about creating a view from a 3D plane

You can use an existing 3D plane as a support for creating a new design view, whether this 3D plane is defined in the same document or not. You may also use a Functional Tolerancing and Annotation (FTA) view for the same purpose.

A design view created from a 3D plane (or from an FTA view) has the following characteristics:

- It is either an auxiliary view, a section view or a section cut.
- The view axis and origin are identical to those of the 3D plane.
- The view axis and origin are not associative to the 3D plane.
- The created view cannot be linked in position with another view.
- The view is created empty.

View Management

Use the cutting plane: cut the 3D background of layout views along each view's definition plane.

Use the clipping frame: clip the 2D and 3D backgrounds of layout views using a frame.

Use the back-clipping plane: clip the 2D and 3D backgrounds of layout views using a back-clipping plane.

Manage the layout view background: display and manage the 2D and 3D backgrounds differently for each view.

Using the Cutting Plane

In this task, you will learn how to cut the 3D background of layout views along each view's definition plane.

This can be useful to validate the geometry of a design view, as the view content and the 2D background will remain visible, even if their view plane is behind the cutting plane. Since it is the view plane that is used as the cutting plane, the result will depend on the position of the layout in 3D space.

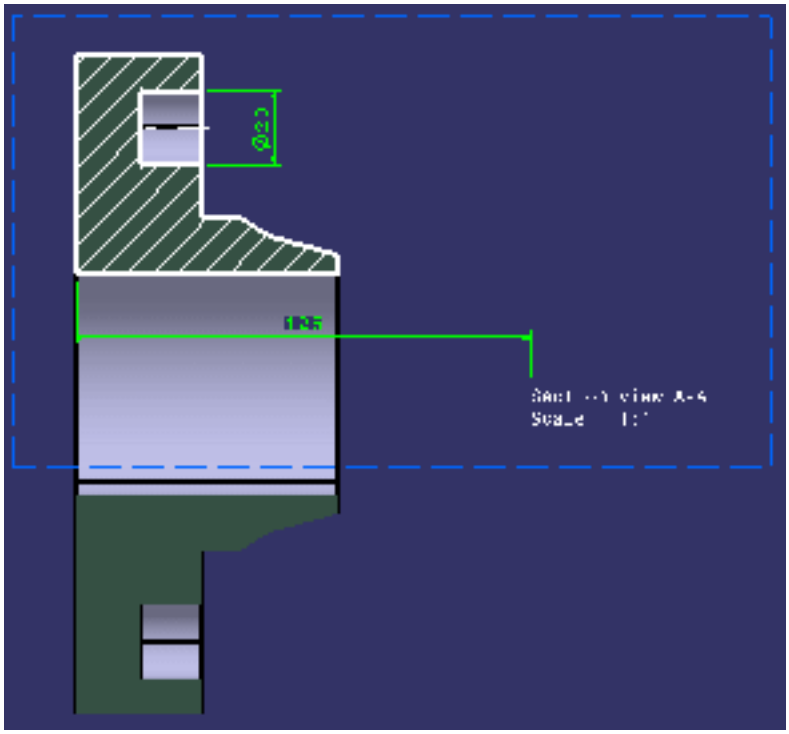
Note that the cutting plane is applied globally to a layout, which means it is either active or inactive for all views at once.

Open the Disk4.CATPart document. Select **Start -> Mechanical Design -> 2D Layout for 3D Design** to open the layout in the 2D window.

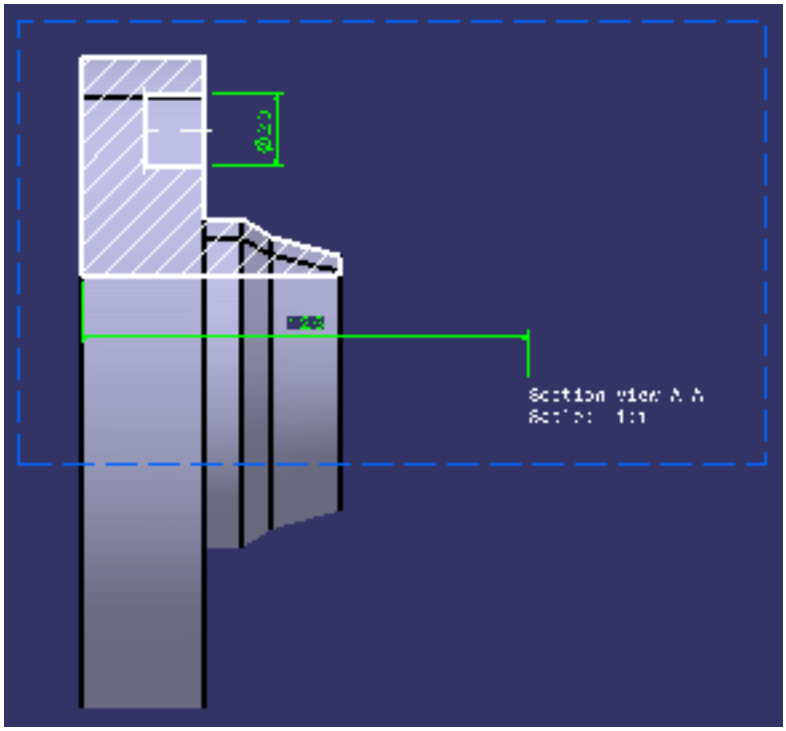
If necessary, activate the **Display Backgrounds as Specified for Each View** icon in the **Visualization** toolbar to display the layout view background.

1. Activate the **Cutting Plane** icon in the **Visualization** toolbar.

Notice how the 3D background of the Section view is cut along its definition plane.



2. Now, de-activate the **Cutting Plane** icon and notice how the 3D background of the Section view is no longer cut along its definition plane.



- The cutting plane is taken into account when printing in raster mode and when creating a drawing view.
- For more information on the other visualization-related options available in the Visualization toolbar, refer to Layout Tools.

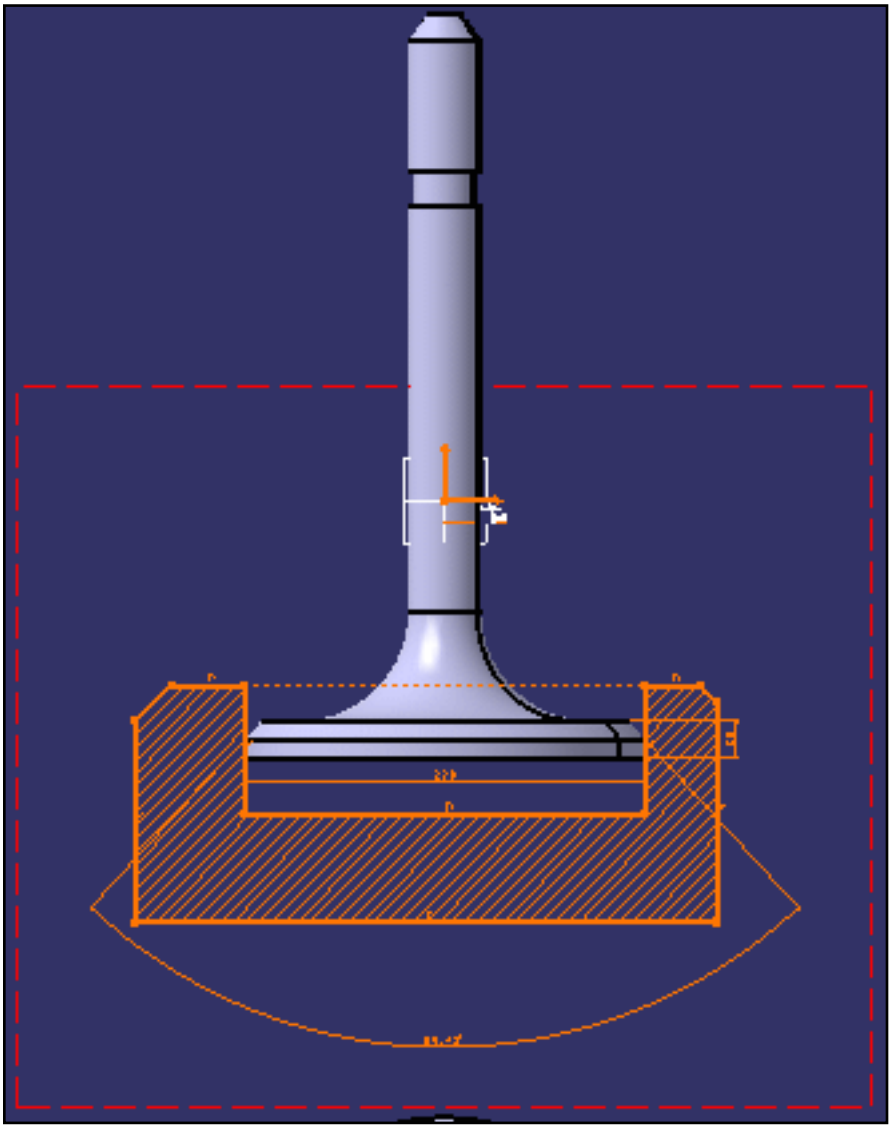
Using the Clipping Frame

In this task, you will learn how to clip the 2D and 3D backgrounds of layout views using a rectangular frame.

The clipping frame is composed of four planes that are parallel to the layout view axis, and makes all background elements that are outside of the frame's boundaries invisible.

Open the Valve.CATPart document. Double-click Sheet.1 in the specification tree to open the layout in the 2D window.

Most of the front view's background is of no use for the design and can be clipped.

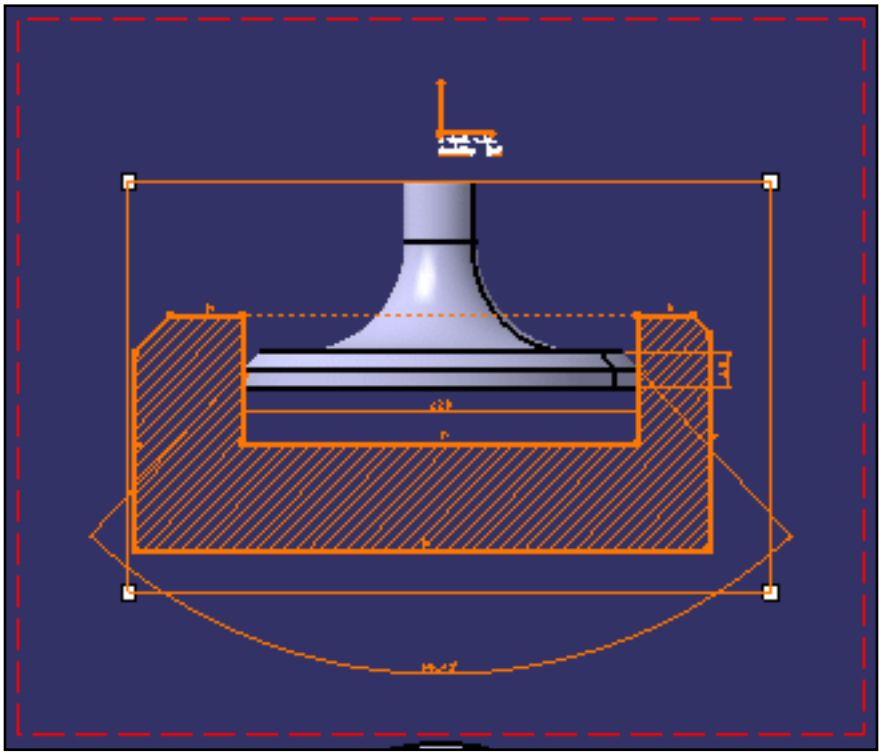


1. Right-click the front view in the specification tree or in the sheet and select **Activate clipping frame**.

   You can also use this alternative method:
   a. Right-click the front view and select **Properties**.
   b. In the **Properties** dialog box, click the **Visualization** tab.
   c. In the Clipping area, select the **Activate clipping frame** check box.
   d. Click **OK** to validate and exit the dialog box.

   The front view now contains a frame that clips the background that is outside of the frame's boundaries.

   Note that all 2D elements of the layout view itself are still visible, independently of the frame's size and position. This lets you work in the layout view as usual, adding geometry, annotations and dress-up for example.



2. You can now reframe, resize and/or move the clipping frame according to your design's needs:

   You can perform the following operations providing the view has been selected first.

   - To reframe the clipping frame on the view content, right-click the frame and select **Reframe on view content**. Thus reframed, the clipping frame is slightly larger than the layout view frame.

   - To reframe the clipping frame on the whole view background, right-click the frame and

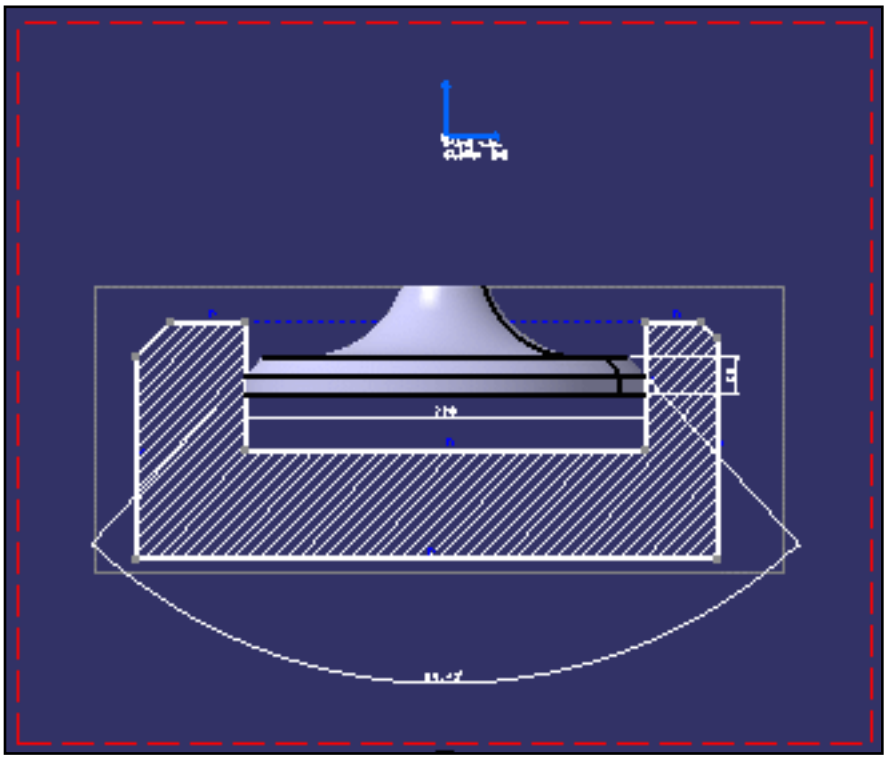
select **Reframe on view background**. Thus reframed, the clipping frame shows the whole 2D and 3D background.

- Prior to performing this operation, you may want to hide all elements on the sheet (views, geometry, annotations, and so on) from the view's 3D background. To do this, multi-select all views, right-click them and then select **Visualization -> Hide in 3D**.
- This option computes the clipping frame according to a square bounding box composed of all background elements. As a result, the clipping frame may end up being larger than the background.

- To resize the clipping frame, drag the manipulators at the four corners of the frame.

If necessary, you can resize the clipping frame beyond the layout view frame.

- To move the clipping frame, drag the frame itself using one of its lines.



3. To deactivate the clipping frame, right-click the front view and select **Deactivate clipping frame**.

You can also use this alternative method:

a. Right-click the front view and select **Properties**.
b. In the **Properties** dialog box, click the **Visualization** tab.
c. In the Clipping area, clear the **Activate clipping frame** check box.
d. Click **OK** to validate and exit the dialog box.

The front view's whole background is displayed again as the frame is de-activated.

- To activate or deactivate the clipping frame for several views at once, multi-select the views and use the **Properties** dialog box. Note that when the selected views do not all have the same status (clipping frame activated or deactivated for all views), the check box is grayed out.
- The clipping frame can also be activated by default for each newly created view. Refer to View Creation > Clipping in the Customizing chapter for more information.
- The clipping frame is taken into account:
  - when printing in raster mode: all background elements that are outside of the frame's boundaries, as well as the frame itself, are not printed.
  - when creating a drawing view: all background elements that are outside of the frame's boundaries are not generated. Note that the frame itself is not generated, and that the boundary of clipped elements is displayed using a specific, un-customizable, line type. Also note that the clipping frame is not associative: if you activate/deactivate it, or modify its size/position after the drawing has been generated, you will need to update the drawing (even if it appears as being up-to-date) in order for your modifications to be taken into account.

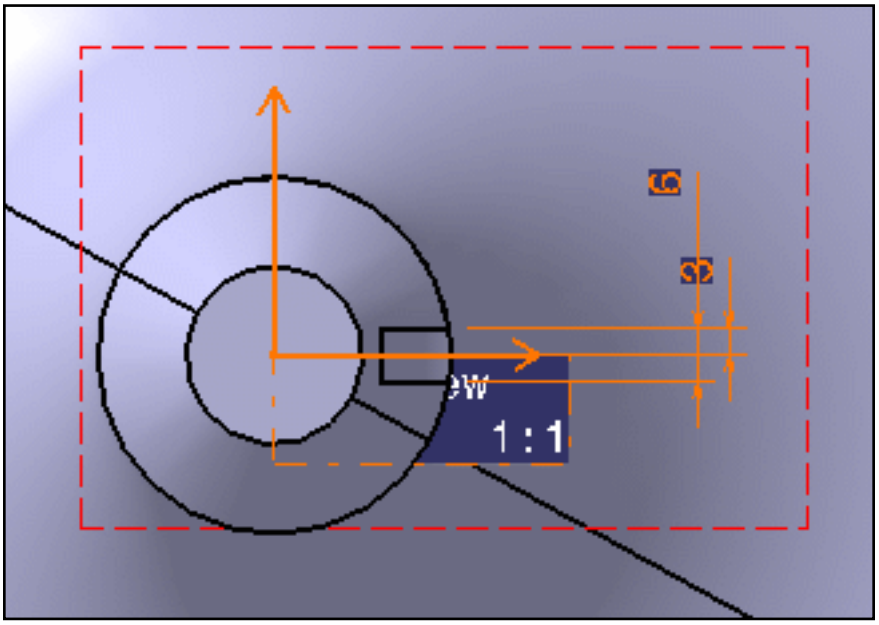
Using the Back-Clipping Plane

In this task, you will learn how to clip the 2D and 3D backgrounds of layout views using a back-clipping plane.

The back-clipping plane is parallel to, and located behind, the layout view plane. It makes all background elements that are behind it invisible.

Open the Valve.CATPart document. Double-click Sheet.1 in the specification tree to open the layout in the 2D window.

Most of the top view's background is of no use for the design and can be back-clipped.



1. Right-click the top view in the specification tree or in the sheet and select **Activate back-clipping plane**.

   You can also use this alternative method:
   a. Right-click the front view and select **Properties**.
   b. In the **Properties** dialog box, click the **Visualization** tab.
   c. In the Clipping area, select the **Activate back-clipping plane** check box.
   d. Click **OK** to validate and exit the dialog box.

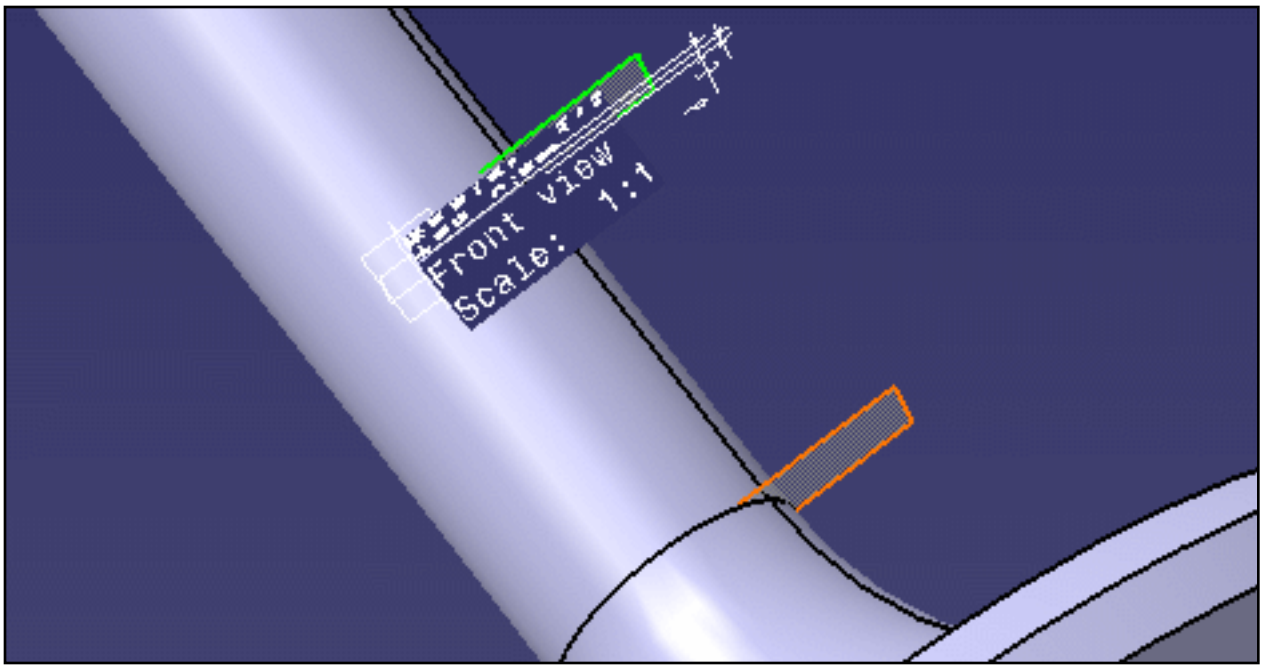
   The top view now contains a plane that clips the background that is behind it.

   Note that all 2D elements of the layout view itself are still visible, independently of the plane's size and position. This lets you work in the layout view as usual, adding geometry, annotations and dress-up for example.

2. You can now define the back-clipping plane's position in 3D space according to your design's needs.

This position is defined by the distance between the layout view plane and the back-clipping plane.

To do so, right-click the front view and select **Edit back-clipping plane**. A 3D viewer is displayed. The back-clipping plane is visualized using an orange frame and the layout view plane using a green frame.



3. You can either:
   - specify the plane's position relatively to the layout view plane by entering a value in the **Depth** field.

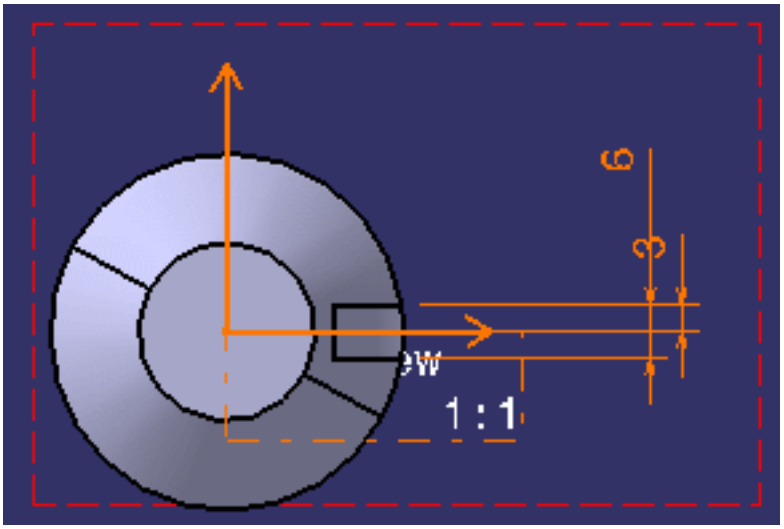
     The unit value is defined in **Tools** > **Options** > **General** > **Parameters and Measure** > **Units** tab.

   - drag the back-clipping plane.

4. Click **Apply** to view your modifications.

5. When you are satisfied with the back-clipping plane's position, click **Close** to validate and exit the dialog box.

The background of the top view is back-clipped as specified.



6. To deactivate the back-clipping plane, right-click the top view and select **Deactivate back-clipping plane**.

You can also use this alternative method:

a. Right-click the front view and select **Properties**.
b. In the **Properties** dialog box, click the **Visualization** tab.
c. In the Clipping area, clear the **Activate back-clipping plane** check box.
d. Click **OK** to validate and exit the dialog box.

The top view's whole background is displayed again as the back-clipping plane is de-activated.

- To activate or deactivate the back-clipping plane for several views at once, multi-select the views and use the **Properties** dialog box. Note that when the selected views do not all have the same status (back-clipping plane activated or deactivated for all views), the check box is grayed out.
- When a view is rotated, its clipping frame is rotated using the same angle.
- When the scale of a view is modified, the position and size of its clipping frame is scaled as well.
- The back-clipping plane can also be activated by default for each newly created view. Refer to View Creation > Clipping in the Customizing chapter for more information.
- The back-clipping plane is taken into account:
  - when printing in raster mode: all background elements that are behind it are not printed.
  - when creating a drawing view: all background elements that are behind it are not generated. Note that the boundary of back-clipped elements is displayed using a specific, un-customizable, line type. Also note that the back-clipping plane is not associative: if you activate/deactivate it, or modify its position after the drawing has been generated, you will need to update the drawing (even if it appears as being up-to-date) in order for your modifications to be taken into account.

Managing the Layout View Background

In this task, you will learn how to display and manage the 2D and 3D backgrounds differently for each view.

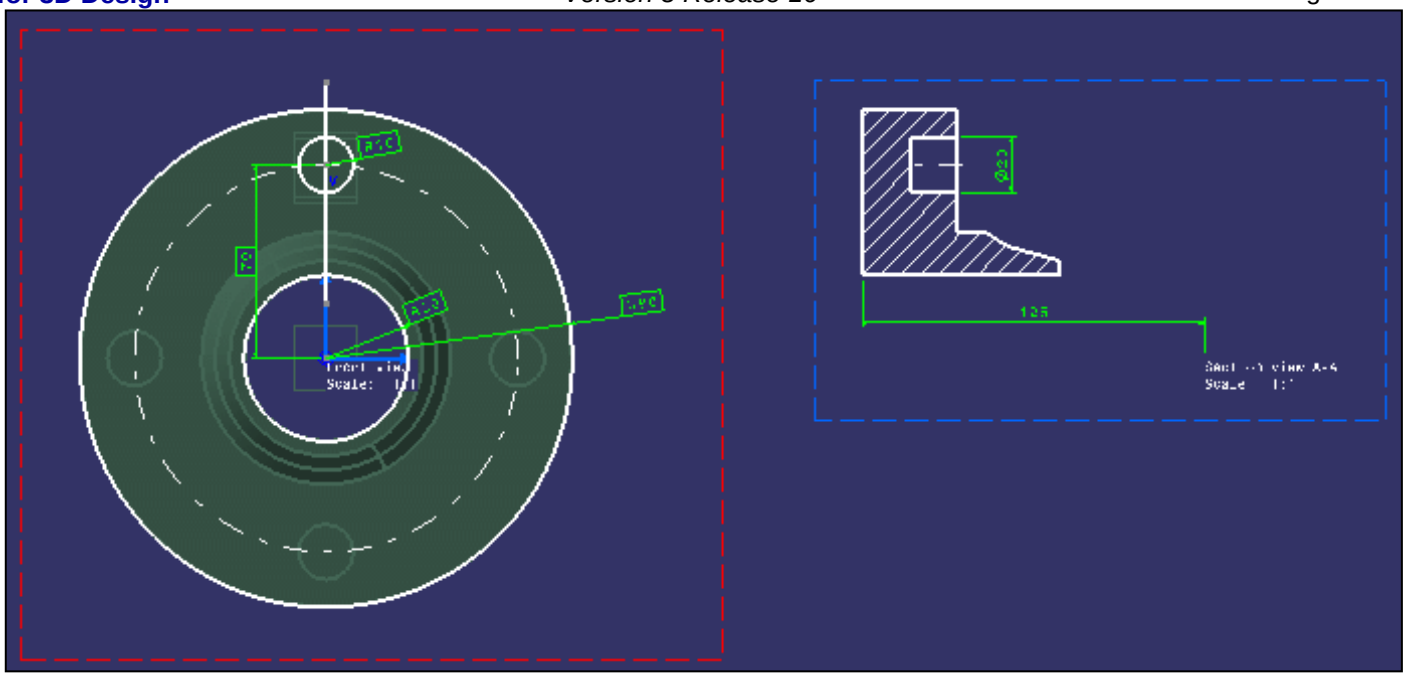
Open the Disk4.CATPart document. Select **Start -> Mechanical Design -> 2D Layout for 3D Design** to open the layout in the 2D window.

1. Activate the **Display Backgrounds as Specified for Each View** icon in the **Visualization** toolbar. This will display each view according to the option you choose for it in the following steps.

2. Right-click the Front view from the geometry area or the specification tree.
3. Point to **Background**. A number of options are available from the menu:
   - **Standard**: shows both the 2D and 3D backgrounds.
   - **Invisible**: hides both the 2D background (the 3D representation of 2D elements which do not belong to the current view, but to other views) and the 3D background (the representation of all 3D elements, including edges, faces and 3D wireframe).
   - **Unpickable**: prevents selecting elements in both the 2D and 3D backgrounds, even though you can see them. You can just handle 2D elements which belong to the current view.
   - **Low-intensified**: dims all elements in both the 2D and 3D backgrounds.
   - **Unpickable low-intensified**: dims all elements in both the 2D and 3D backgrounds. Additionally, although you can see these elements, you cannot select them. You can just handle 2D elements in the current view.
4. Select **Unpickable low-intensified**.

   The Front view is displayed with all elements dimmed in both the 2D and 3D backgrounds, and you cannot select the elements contained in the backgrounds.
5. Right-click the Section view from the geometry area or the specification tree.
6. Select **Background -> Invisible**.

   Both the 2D and 3D backgrounds are hidden from the Section view.



- You can also manage the background of each layout view using the view properties. For more information, refer to Editing View Properties.
- For more information on the other visualization-related options available in the Visualization toolbar, refer to Layout Tools.

2D Geometry

The 2D Layout for 3D Design workbench enables you to create 2D geometry.

As 2D geometry commands work exactly as in the Sketcher workbench, this section of the documentation actually provides links to the *Sketcher User's Guide*. As such, the information detailed in this section is presented in a Sketcher context. You should note that the *Sketcher User's Guide* contains images that correspond to the Sketcher workbench and therefore illustrate geometry in an environment that is different from the 2D Layout for 3D Design environment (symbols and background color, for example).

Before you begin creating 2D geometry in 2D Layout for 3D Design

Before you begin creating 2D geometry in 2D Layout for 3D Design, make sure you are familiar with such concepts as:

- The Tools toolbar and the Tools Palette.
- SmartPick, an easy-to-use tool designed to assist you when creating geometry. For more information, refer to the SmartPick task in the *Sketcher User's Guide*.
- Construction elements. For more information, refer to Creating Standard or Construction Elements in the *Sketcher User's Guide*.
- Multi-selection. For more information, refer to the Selecting Objects chapter in the *Infrastructure User's Guide*.

Remember the following points:

- Construction elements contained in 2D geometry are displayed only in the current view (in the 2D window).
- To ensure that 2D geometry is not altered once it has been created, geometry edition is only allowed in the active view. Therefore, if you want to edit or move 2D geometry, you need to activate the view which contains the geometry. You can prevent 2D geometry from being involuntarily moved (and distorted) in active views by unselecting **Allow direct manipulation** from **Tools -> Options -> Mechanical Design -> Drafting - > Geometry** tab.
- While creating 2D geometry, you can create detected constraints automatically by activating the **Create Detected Constraints** icon in the **Tools** toolbar. You can view the created constraints by activating the **Show Constraints** icon.
- You can create as many 2D geometry elements of a given type as needed by double-clicking the appropriate icon (instead of single-clicking it).

Create a profile: Use the Tools Palette or click to define lines and arcs.

Create a rectangle: Use the Tools Palette or click the rectangle vertices one after the other.

Create an oriented rectangle: Use the Tools Palette or click to define a first side for the rectangle and then a point corresponding to the rectangle length.

Create a parallelogram: Use the Tools Palette or click to define a first side for the parallelogram and then a point corresponding to the parallelogram length.

Create an elongated hole: Use the Tools Palette or click to define the center to center axis and then a point corresponding to the curved oblong profile length and angle.

Create a cylindrical elongated hole: Use the Tools Palette or click to define the center to center circular axis and then a point corresponding to the curved oblong profile length and angle.

Create a keyhole profile: Use the Tools Palette or click to define the center to center axis and then two points corresponding to both radii.

Create an hexagon: Use the Tools Palette or click to define the hexagon center and dimension.

Create centered rectangles: Use the Tools palette to define the rectangle center and dimensions.

Create centered parallelograms: Use the Tools palette to define a first side for the parallelogram and then a point corresponding to the parallelogram length.

Create a circle: Use the Tools Palette or click to define the circle center and then one point on the circle.

Create a three point circle: Use the Tools Palette or click to define the circle start point, second point and end point one after the other.

Create a circle using coordinates: Use the Circle Definition dialog box to define the circle center point and radius.

Create a tri-tangent circle: Click three elements one after the other to create a circle made of three tangent constraints.

Create a three point arc: Use the Tools Palette or click to define the arc start point, end point and second point one after the other.

Create a three point arc with limits: Use the Tools Palette or click to define the arc start point, end point and second point one after the other.

Create an arc: Use the Tools Palette or click to define the arc center, then the arc start point and end point.

Create a spline: Click the points through which the spline will go.

Connect curves with a spline: Click the first, then the second element to connect.

Create an ellipse: Use the Tools Palette or click to define the ellipse center, major semi-axis and minor semi-axis endpoints one after the other.

Create a parabola by focus: Click the focus, apex then the two extremity points.

Create a hyperbola by focus: Click the focus, center and apex, then the two extremity points.

Create a conic: Click the desired points and excentricity for creating an ellipse, a circle, a parabola or a hyperbola, using tangents, if needed.

Create a line: Use the Tools Palette or click the first and second points of the line.

Create an infinite line: Use the Profile toolbar or click the first and second points of the infinite line.

Create a bi-tangent line: Click two elements one after the other to create a line that is tangent to these two elements.

Create a bisecting line: Click two lines.

Create a line normal to a curve: Click a point and then the curve.

Create a point: Use the Tools Palette or select the point horizontal and vertical coordinates.

Create a points using coordinates: Enter in the Point Definition dialog box cartesian or polar coordinates.

Create an equidistant point: Enter in the Equidistant Point Definition dialog box the number and spacing of the points to be equidistantly created on a line or a curve-type element.

Create a point using intersection: Create one or more points by intersecting curve type elements.

Create a point using projection: Create one or more points by projecting points onto curve type elements.

Creating Profiles

This task shows how to create a closed profile. A profile may also be open (if you click the profile end point in the free space). Profiles may be composed of lines and arcs which you create either by clicking or using the Sketch tools toolbar.

1. Click the **Profile** icon:

   The **Sketch tools** toolbar displays values for defining the profile.

   Three profile mode options are available:

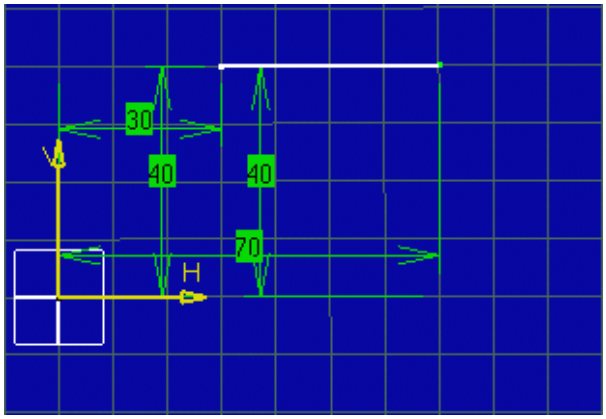
   - **Line**:
   - **Tangent Arc**:
   - **Three Point Arc**:

   Line is the default mode.

2. Type in the **Sketcher tools** toolbar for the first point: **H=30mm**, **V=40mm** and press **Enter**.

3. Type in the **Sketcher tools** toolbar for the end point: **H=70mm**, **V=40mm** and press **Enter**.

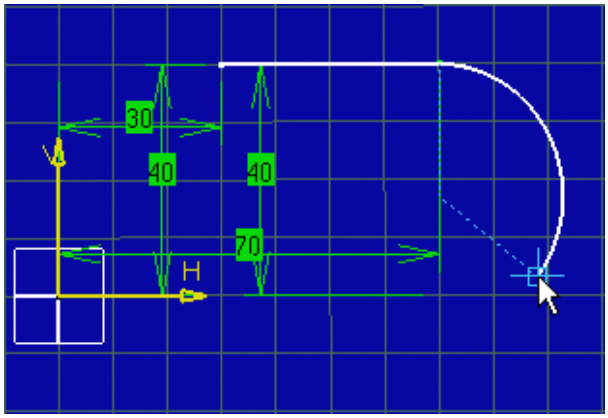
   The line appears as shown here, with the constraints corresponding to the line created via the **Sketch tools** toolbar options.



   Note that at this step, you may also enter length **L** and angle **A** values.

4. Select the **Tangent Arc** option:

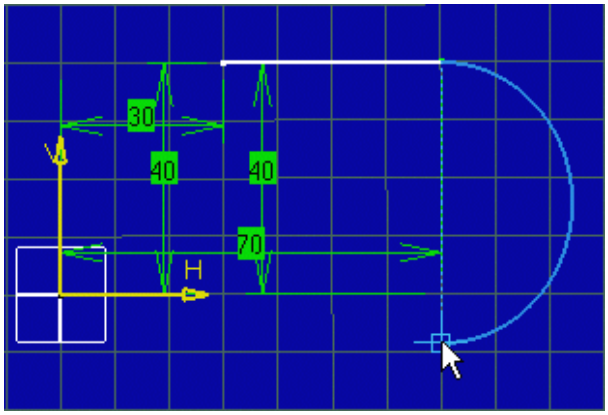
   A rubberbanding arc follows the cursor, showing the tangent arc to be created.



When you sketch a profile using the cursor (in other words without using the **Sketch tools** toolbar fields) to define the end point of the current line or arc, and before clicking this end point, you can hold the CTRL key then click the end point to activate the **Tangent Arc** mode. A rubberbanding rectangle appears representing the arc of circle.

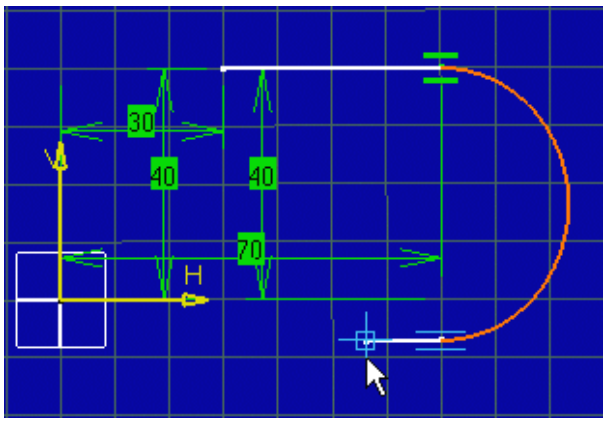
5. Click to indicate the arc end point.

   Tangent arcs are always positioned in the direction of the element previously created.

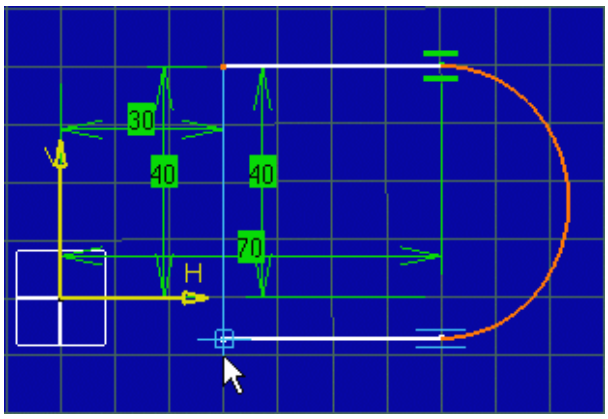


The default mode is back to **Line**.

6. Start dragging another line.

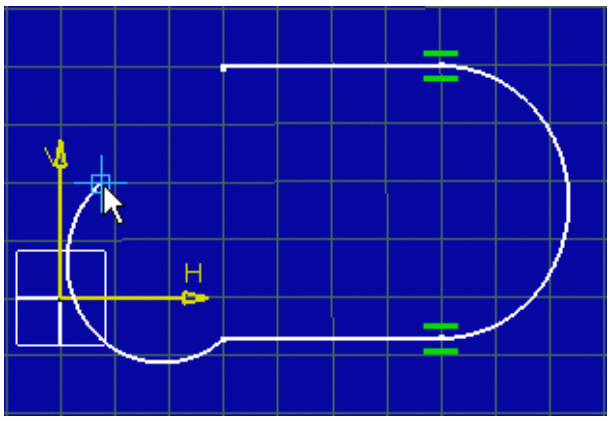


7. Click to indicate the line end point.



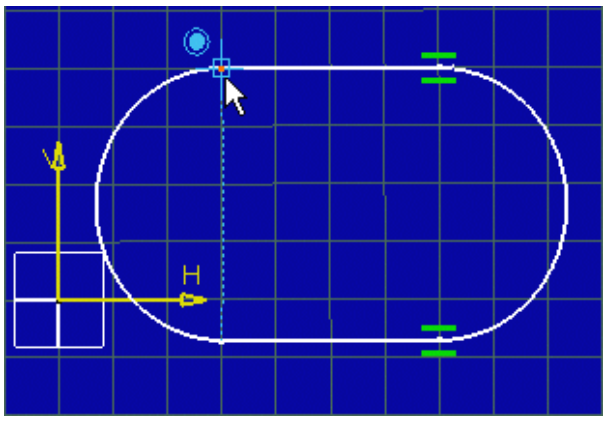
8. Click the **Three Points Arc** mode:

9. Click to indicate a point which the profile is going to go through (arc second point).

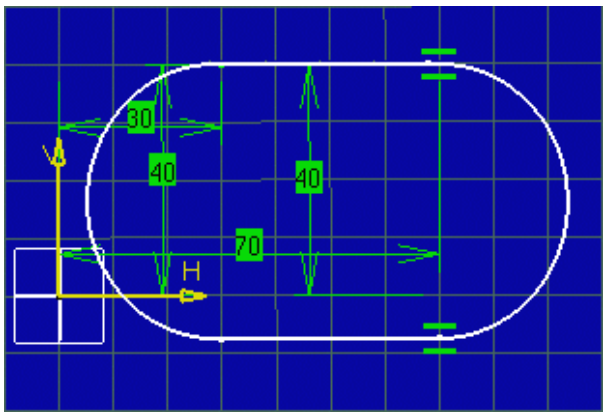


10. Click the start point of the line first created.

    You thus define the three point arc end point.



The profile results as shown here:


30
40
40
70
H

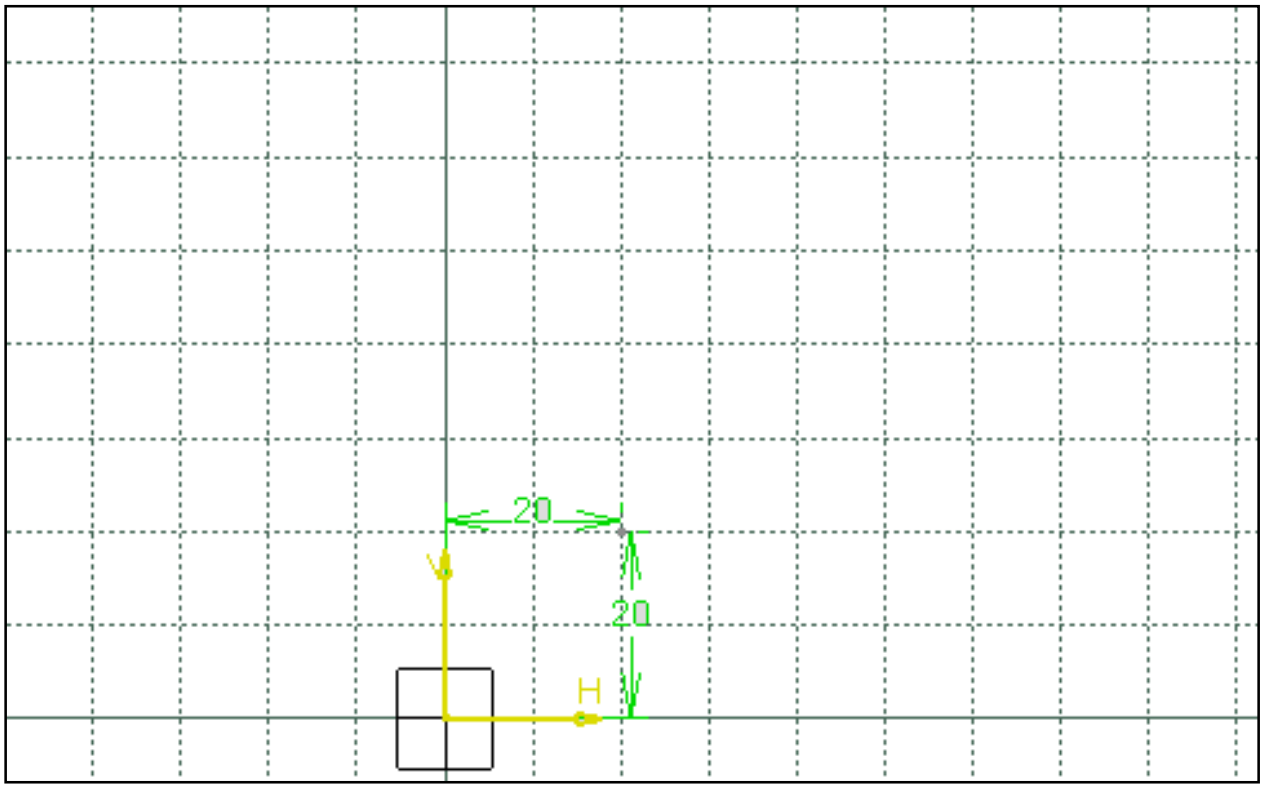
Creating Rectangles

This task shows how to create a rectangle. In this task, we will use the Sketch tools toolbar but, of course you can create this rectangle manually. For this, move the cursor to activate SmartPick and click as soon as you get what you wish.

1. Click the **Rectangle** icon .

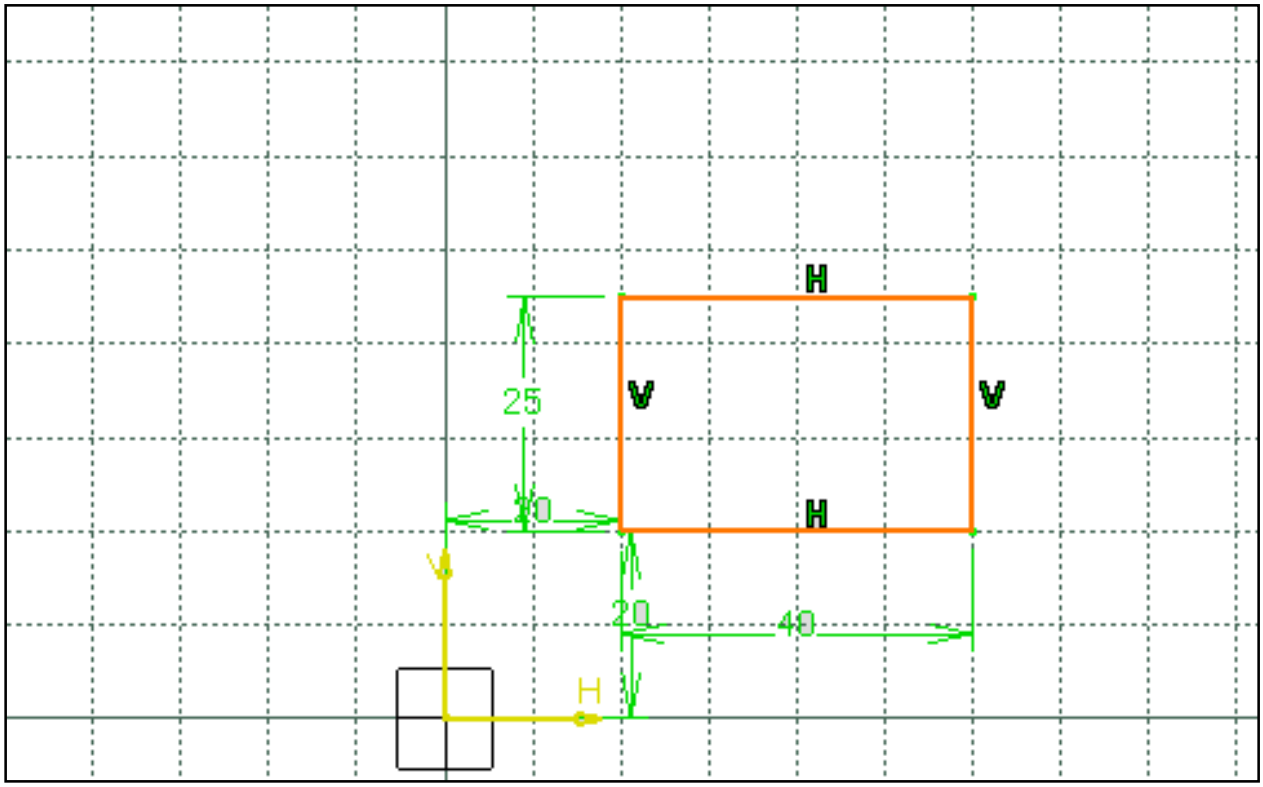
   The **Sketch tools** toolbar displays values for defining the rectangle. For more information, see Using Tools for Sketching in the Sketcher User's Guide

2. Position the bottom-left point at: **H=20mm**, **V=20mm.**



3. Position the top-right corner from the first point: **Width=40mm**, **Height=25mm.**

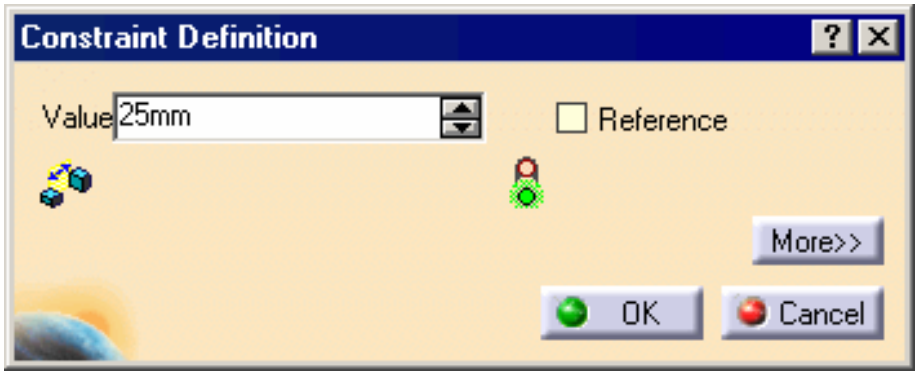
   The rectangle is created.



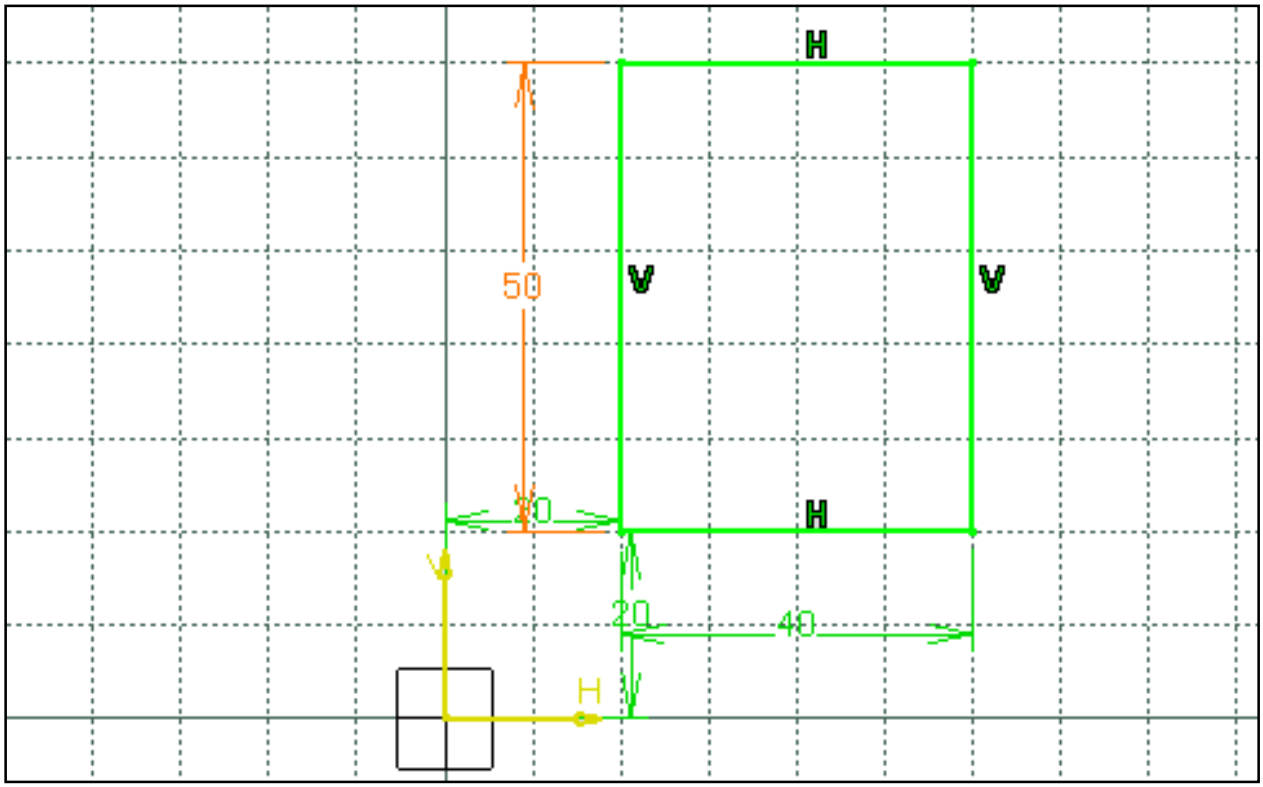
Constraints are similarly assigned to this rectangle on the condition you previously activated the **Dimensional Constraints** option in the **Sketch tools** toolbar.
As a result, to modify the position of this rectangle, you will perform as follows:

4. Double-click the constraint corresponding to the value to be modified.

   The **Constraint Definition** dialog box appears.



5. Enter **50mm** and click **OK**.


H
50
V
V
20
H
20
40
H
V

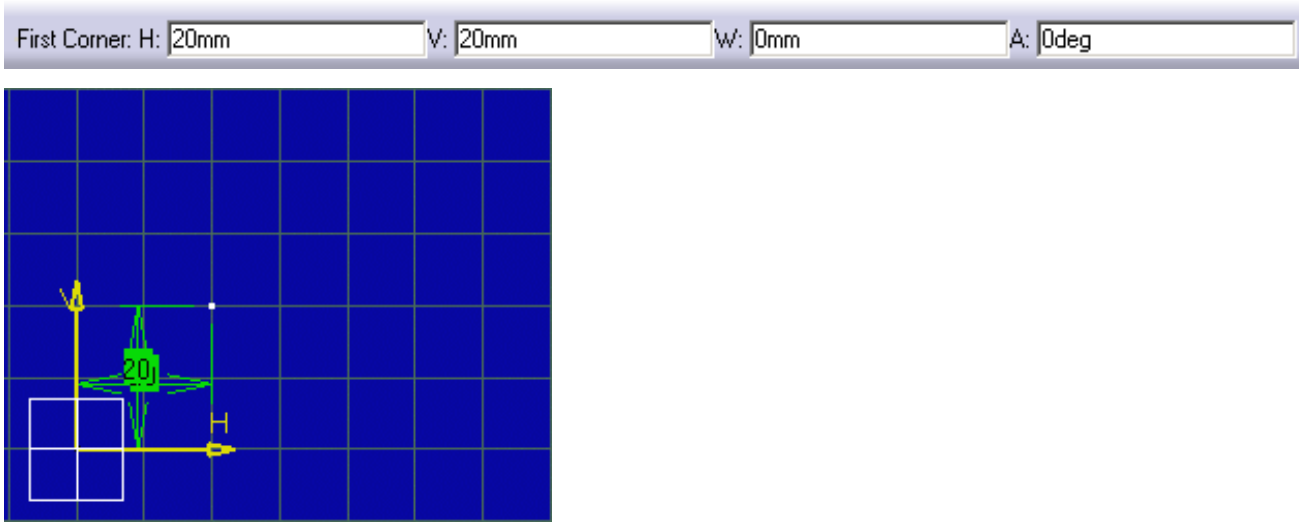
Creating Oriented Rectangles

This task shows how to create a rectangle in the direction of your choice by defining three extremity points of the rectangle. In this task, we will use the Sketch tools toolbar but, of course you can create this oriented rectangle manually. For this, move the cursor to activate SmartPick and click as soon as you get what you wish.

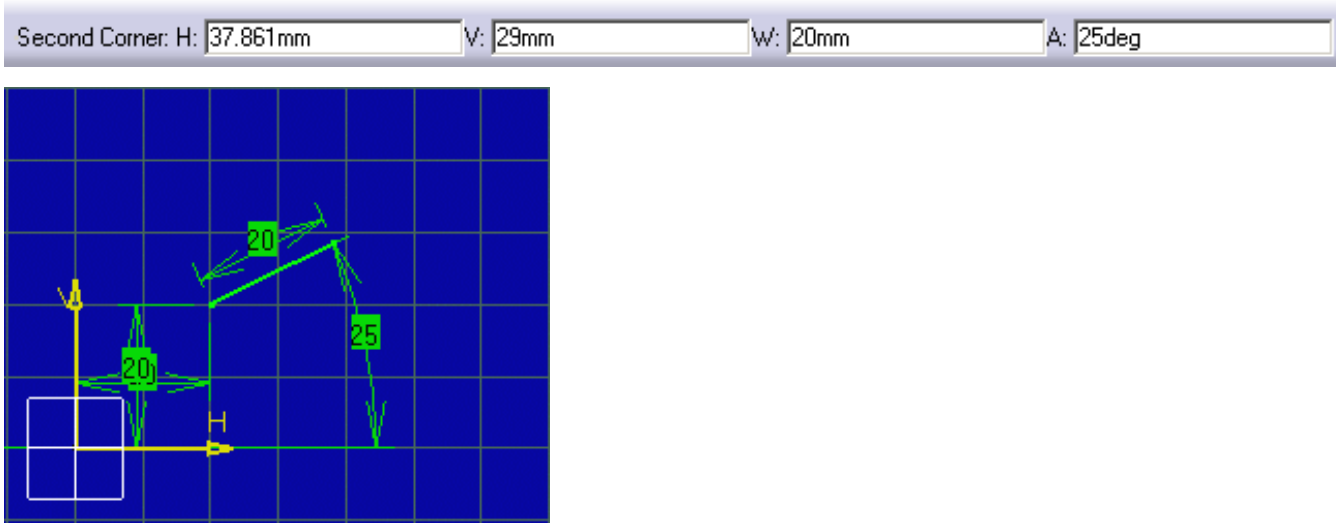
Enter the Sketcher workbench.

1. Click the **Oriented Rectangle** icon .
   The **Sketch tools** toolbar now displays values for defining the first side of the oriented rectangle (both points) and then either one point on the second side or directly the oriented rectangle height.

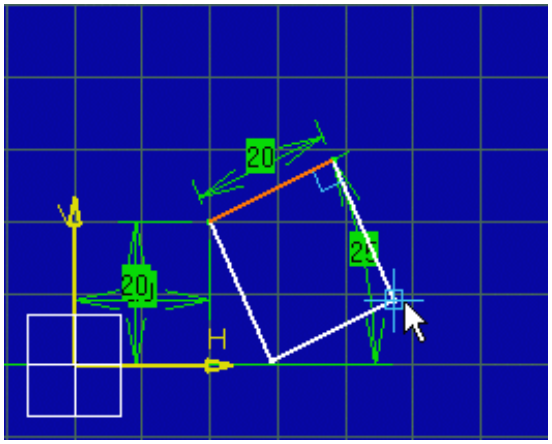
2. Type in the **Sketcher tools** toolbar for the first corner: **H=20mm**, **V=20mm** and press **Enter**.



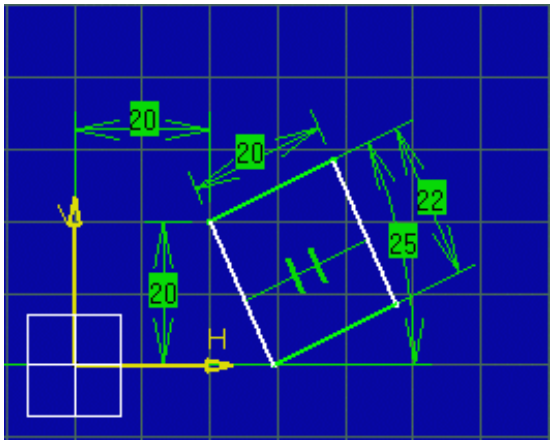
3. Type in the **Sketcher tools** toolbar for the second corner: **W=20mm**, **A=25deg** and press **Enter**.



4. Type in the **Sketcher tools** toolbar for the third corner: **Height=-22mm** and press **Enter**.



The oriented rectangle is created and corresponding constraints appear as shown here.



Creating Parallelograms

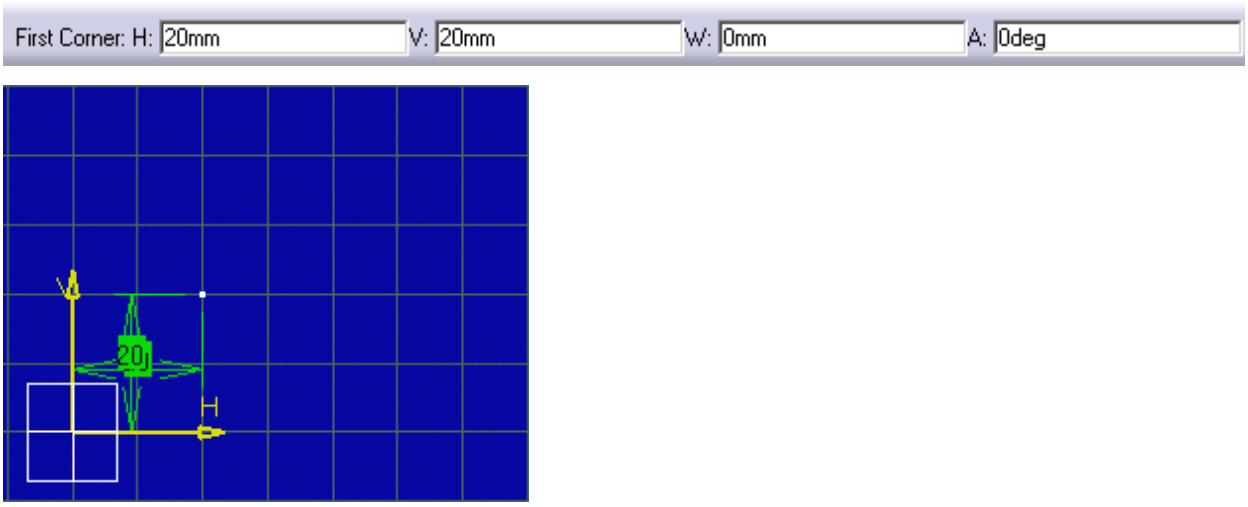
This task shows how to create a parallelogram by clicking. In this task, we will use the Sketch tools toolbar but, of course you can create this parallelogram manually. For this, move the cursor to activate SmartPick and click as soon as you get what you wish.

Enter the Sketcher workbench.

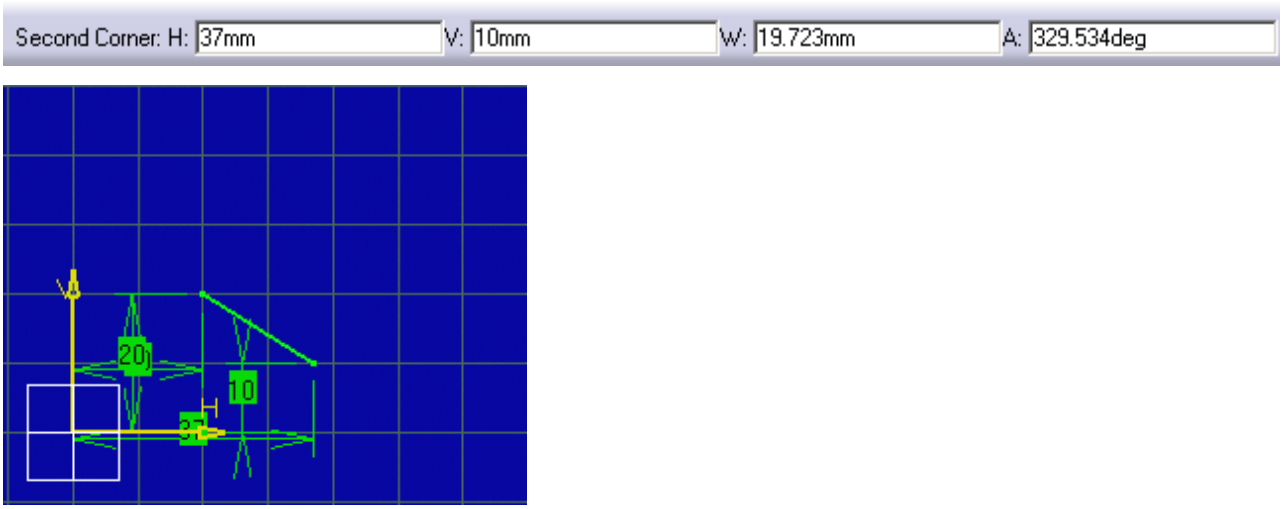
1. Click the **Parallelogram** icon:

   The **Sketch tools** toolbar now displays values for defining the first point of the parallelogram.

2. Type in the **Sketcher tools** toolbar for the first corner: **H=20mm**, **V=20mm** and press **Enter**.

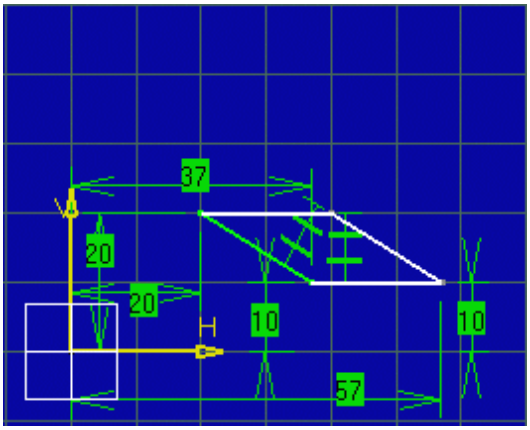


3. Type in the **Sketcher tools** toolbar for the second corner: **H=37mm**, **V=10mm** and press **Enter**.



4. Type in the **Sketcher Tools** toolbar for the third point: **H=57mm**, **V=10mm** and press **Enter**.

The parallelogram and corresponding constraints appear as shown here.


37
20
20
H
10
10
57

Creating Elongated Holes

This task shows how to create an elongated hole by clicking. In this task, we will use the Sketch tools toolbar but, of course you can create this elongated hole manually. For this, move the cursor to activate SmartPick and click as soon as you get what you wish.

1. Click the Elongated Hole icon from the Profiles toolbar (Predefined Profile sub-toolbar).

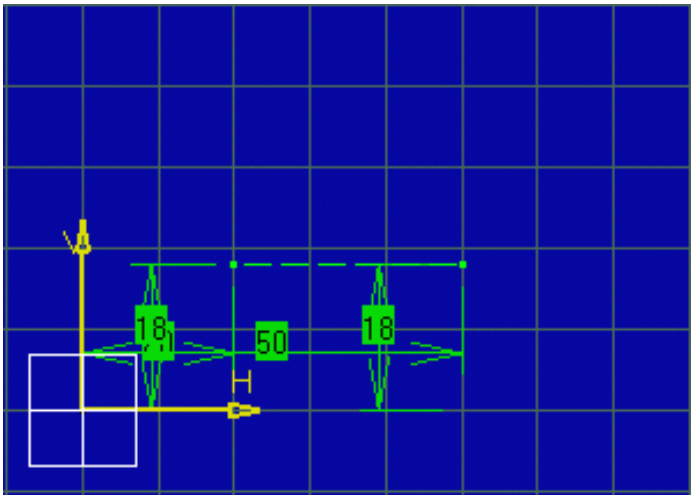
2. The **Sketch tools** toolbar now displays values for defining the elongated hole center to center axis (first and second center point) and then either the elongated hole radius or a point on this elongated hole. Position the cursor in the desired field (**Sketch tools** toolbar) and key in the desired values.

First Center

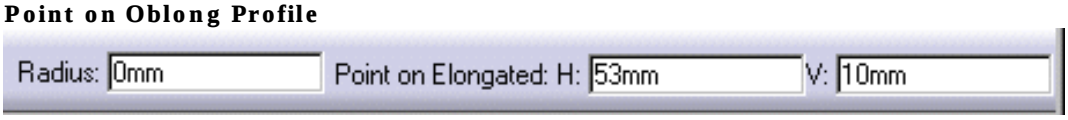
Second Center

For example, key in the coordinates of both center points of the elongated hole: a first point (H: 20mm and V: 18mm) and a second point (H: 50mm and V: 18mm).

You just defined the profile major axis using points. What you can also do is enter both the length and angle of this axis.



Point on Oblong Profile

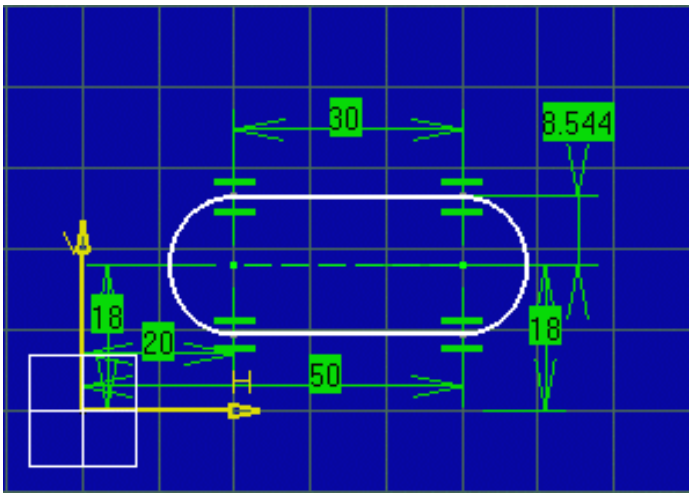


For example, key in the coordinates of a point on the elongated hole (H: 53mm and V: 10mm).

In other words, you just defined the profile minor axis or the elongated hole width applying a given radius to the profile extremity.

At this step, what you can also do is enter the elongated hole radius.

The elongated hole appears as shown here.



Creating Cylindrical Elongated Holes

This task shows how to create a cylindrical elongated hole. A construction arc assists you in creating this element. In this task, we will use the **Sketch tools** toolbar but, of course you can create this cylindrical elongated hole manually. For this, move the cursor to activate SmartPick and click as soon as you get what you wish.

Enter the Sketcher workbench.

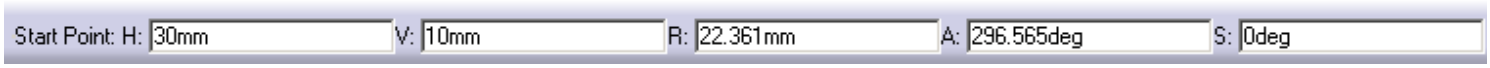
1. Click the **Cylindrical Elongated** icon:

   The **Sketch tools** toolbar now displays values for defining the cylindrical elongated hole.

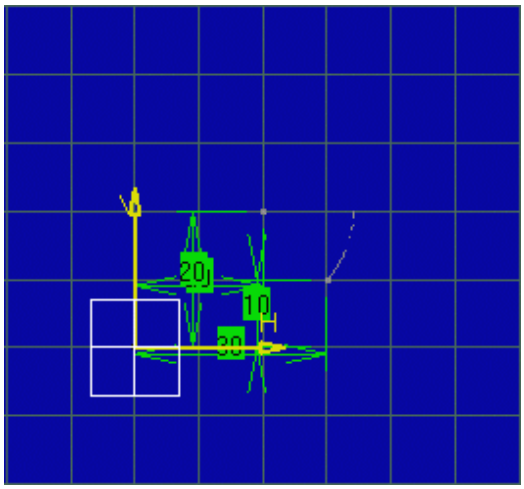
2. Type in the **Sketcher tools** toolbar for the circle center: **H=20mm**, **V=20mm** and press **Enter**.

   The center point will be used to create both the big radius (radius and angle of the cylindrical elongated hole) and the small radius (circular extremities used to define the cylindrical elongated hole).

3. Type in the **Sketcher tools** toolbar for the arc start point : **H=30mm**, **V=10mm** and press **Enter**.



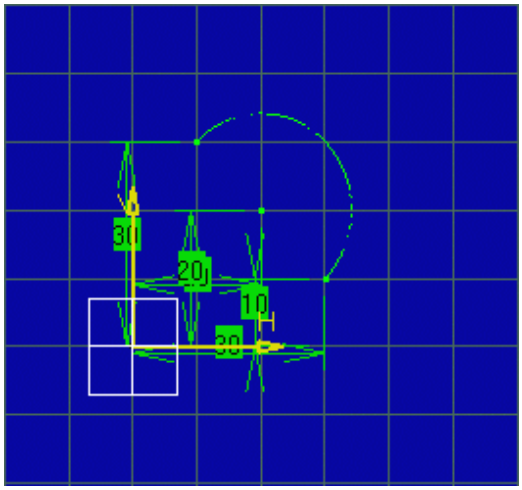
   The arc appears as a construction arc.



   At this step, you may also define the arc big radius **R** and angle **A**.

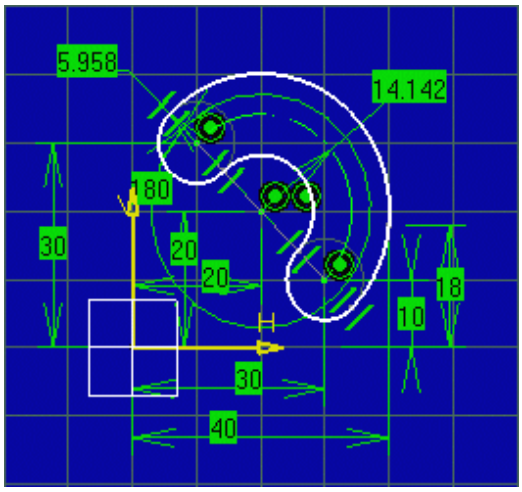
4. Locate the cursor close to **H=10mm** and **V=30mm**

5. Type in the **Sketcher tools** toolbar for the arc end point : **H=10mm** and press **Enter**.



At this step, you cannot define the arc big radius **R** and angle **A**.

6. Type in the **Sketcher tools** toolbar for the point on cylindrical elongated hole: **H=40mm**, **V=18mm** and press **Enter**.



In other words, you are defining what we call the small radius (**Radius: 5.958mm**). This small radius corresponds to the width of the cylindrical elongated hole, relatively to the circle center.

Creating Keyhole Profiles

This task shows how to create a keyhole profile. In this task, we will use the Sketch tools toolbar but, of course you can create this keyhole manually. For this, move the cursor to activate SmartPick and click as soon as you get what you wish.

Enter the Sketch workbench.

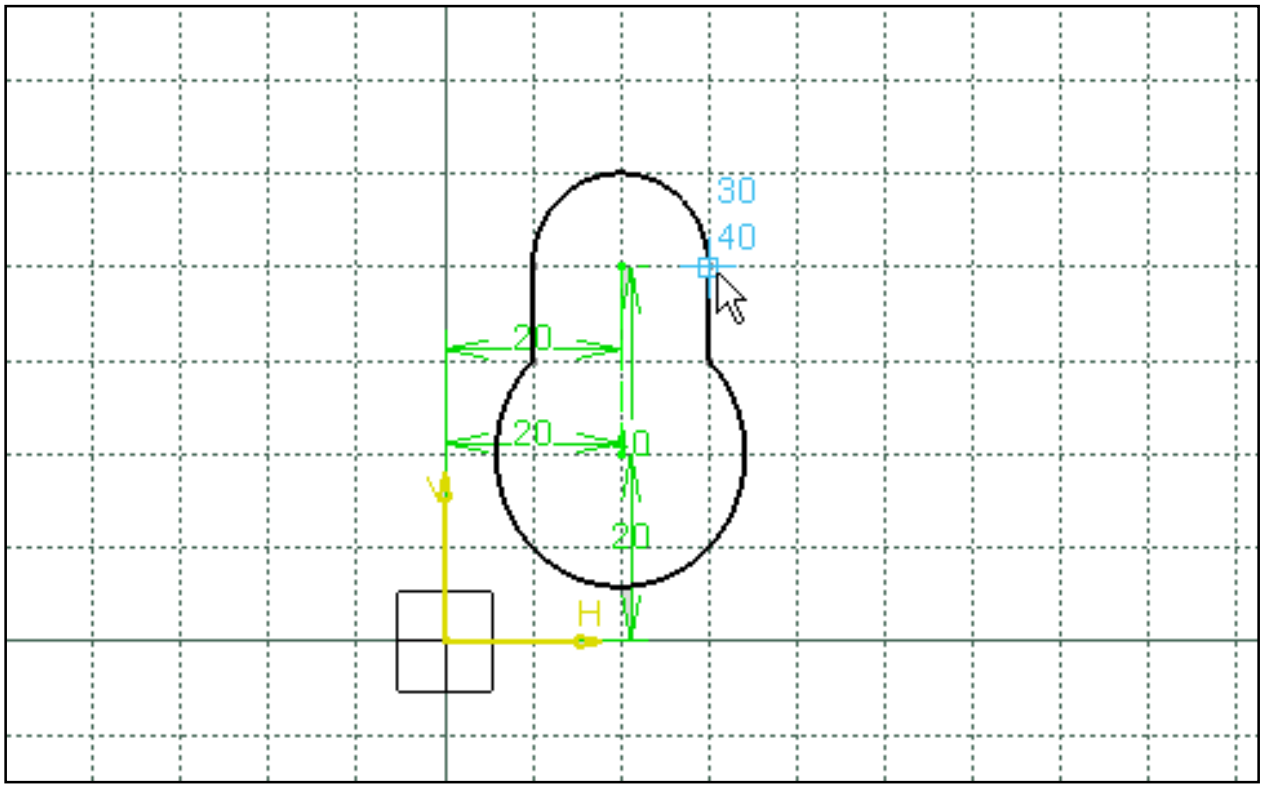
1. Click the **Keyhole Profile** icon .

   The **Sketch tools** toolbar now displays values for defining the keyhole profile. See Using Tools for Sketching for more information.

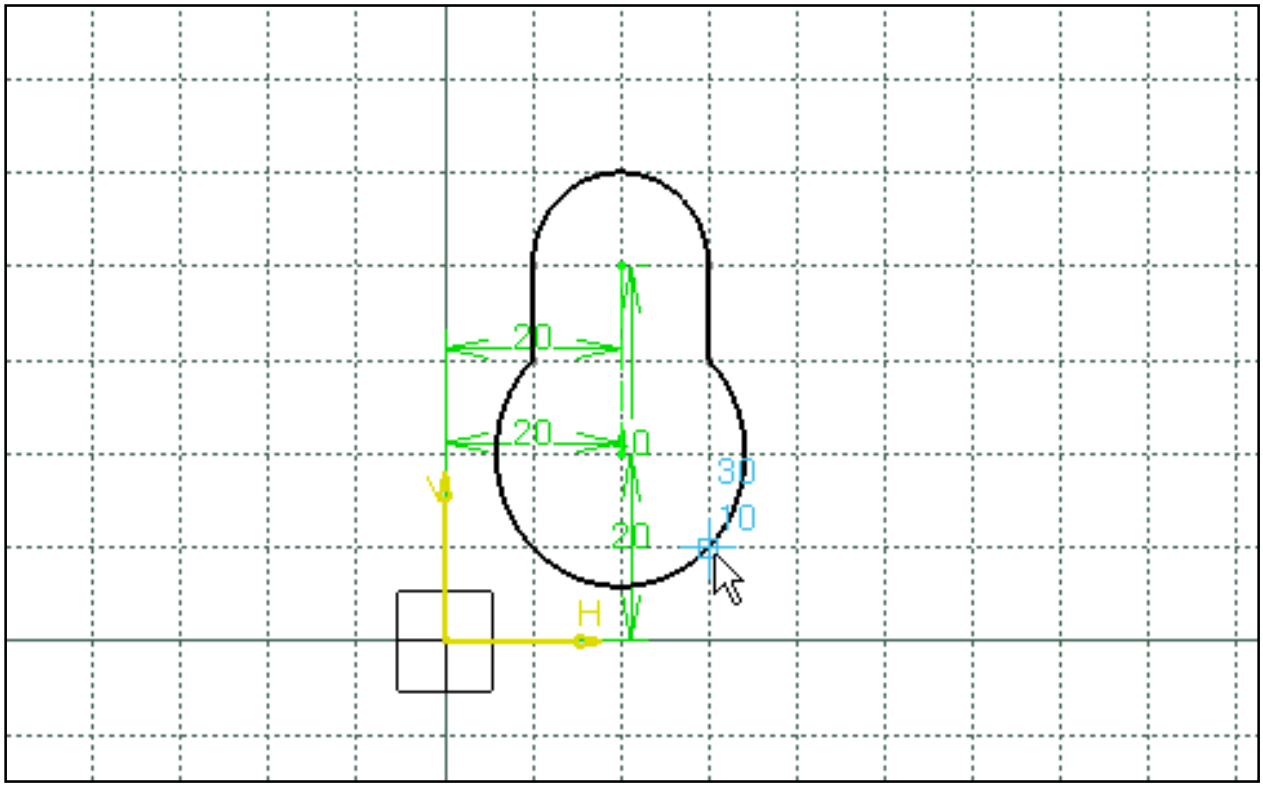
2. Position the stating point at **H=20mm**, **V=20mm**

3. Position the small radius center point at **H=20mm**, **V=40mm**

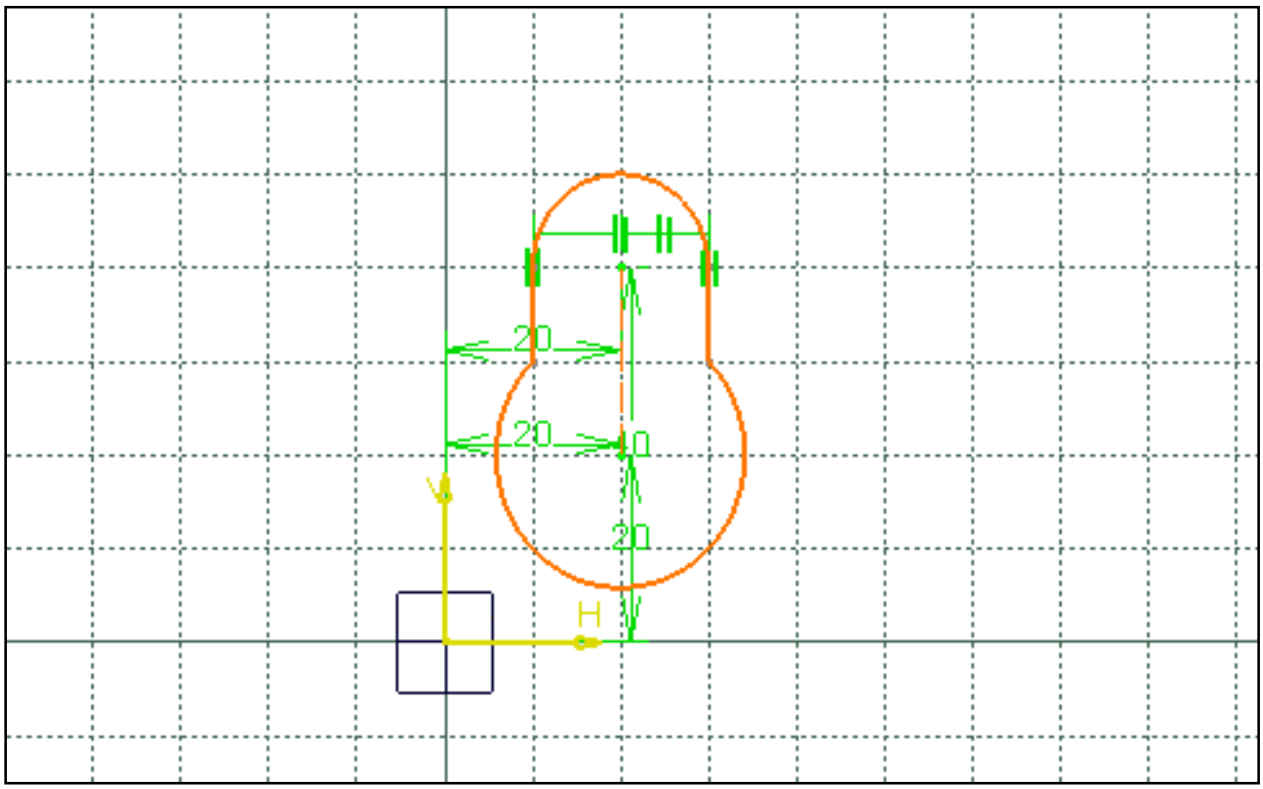
4. Click to define the small radius.



5. Click to define the large radius.



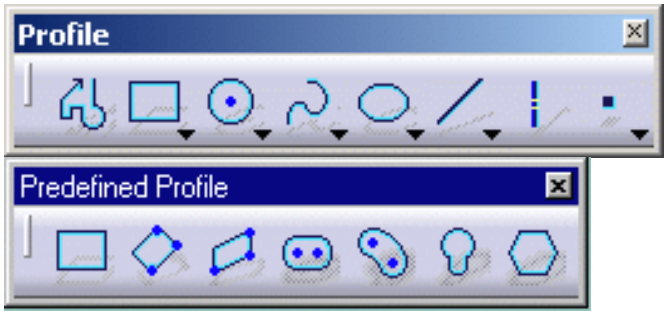
The resulting keyhole is as shown here.



Creating Hexagons

This task shows you how to create an hexagon. A construction circle assists you in creating this profile. In this task, we will use the Sketch tools toolbar but, of course you can create this hexagon manually. For this, move the cursor to activate SmartPick and click as soon as you get what you wish.

1. Click the Hexagon icon from the Profiles icon (Predefined Profile subtoolbar).

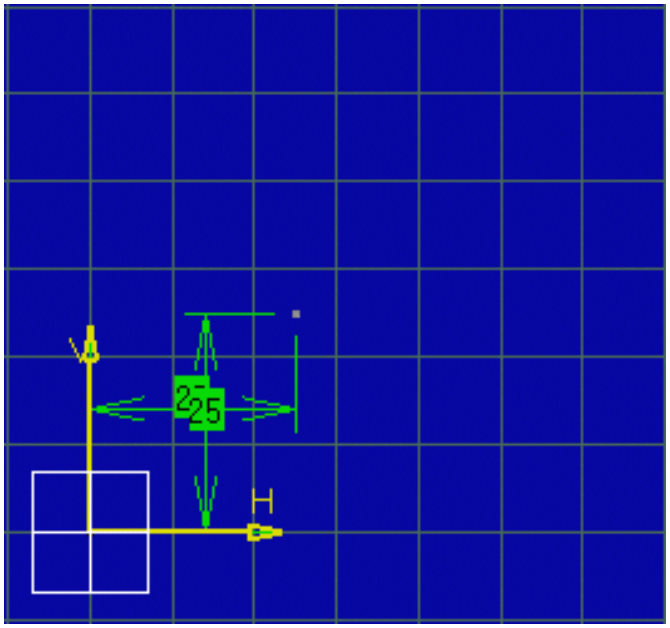


The **Sketch tools** toolbar now displays values for defining the hexagon center and then either a point on this hexagon or the hexagon dimension and angle.

2. Position the cursor in the desired field (**Sketch tools** toolbar) and key in the desired values.

Hexagon Center

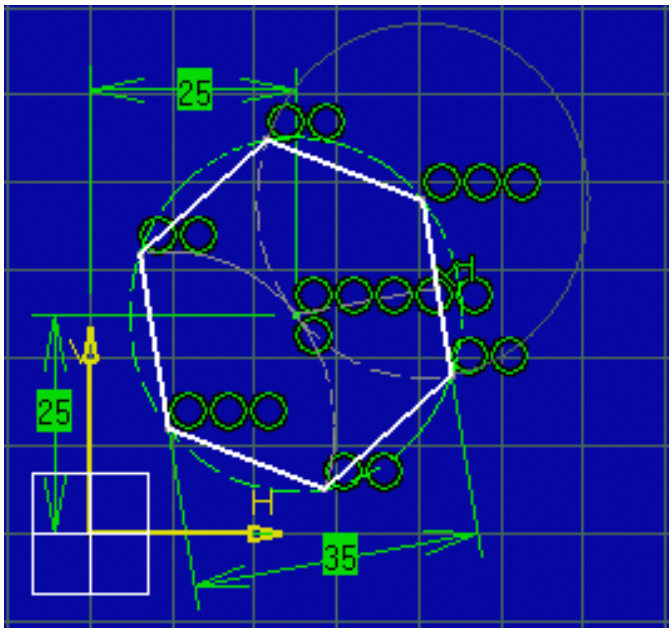
For example, key in the coordinates of the center of the hexagon (H: 25mm and V: 25mm).



Point on Hexagon

For example, key in the dimension (35mm) and Angle (10deg) of the hexagon.

The hexagon is created.



Be careful: if you fix one extremity of the hexagon and try to move the hexagon using another extremity point, this hexagon can result twisted. To avoid this, you must drag the hexagon step by step releasing the mouse button regularly.

Creating Centered Rectangles

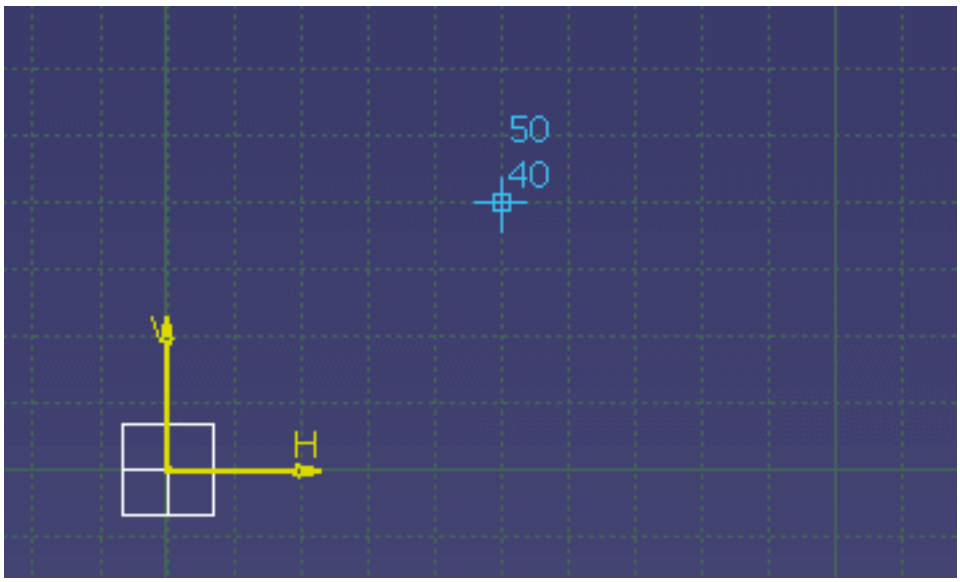
This task shows you how to create a centered rectangle.

Enter the Sketcher workbench:

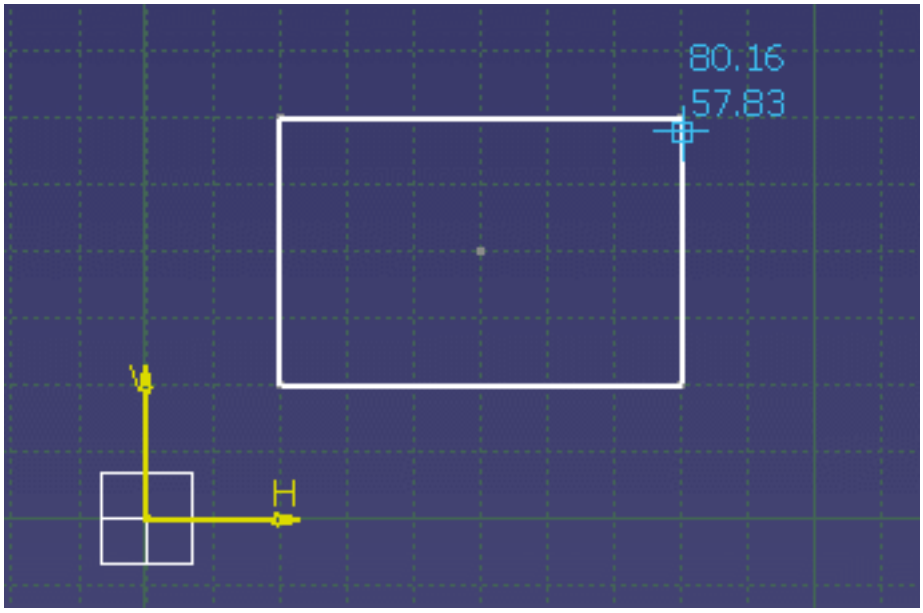
- Ensure that the **Geometrical Constraints** and the **Dimensional Constraints** options are deactivated.

1. Click the **Centered Rectangle** icon .

2. Click a point in the geometry area or select an existing one.



3. Drag the cursor to create the centered rectangle.



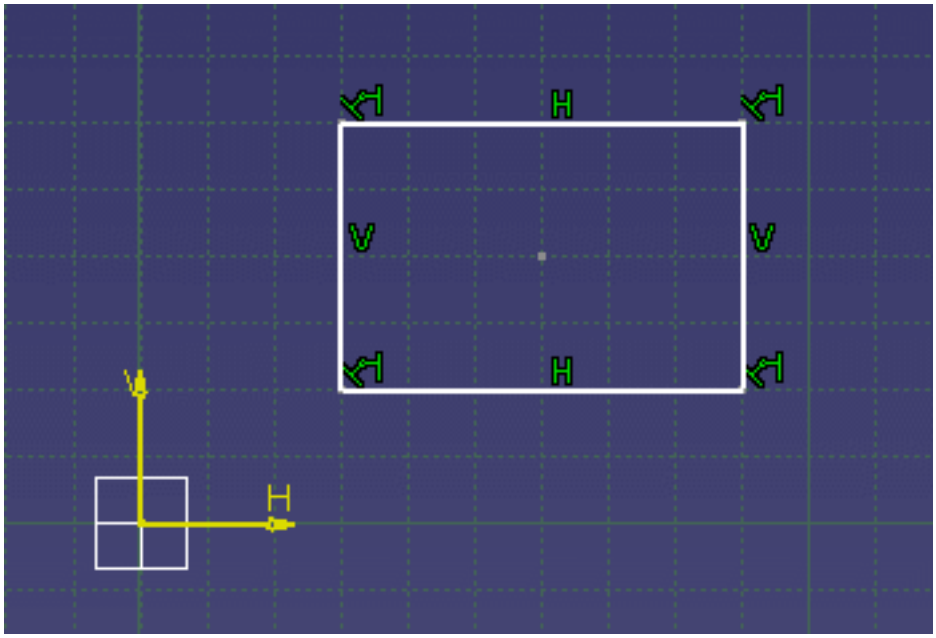
Applying Constraints

4. Activate the **Geometrical Constraints** option and the **Dimensional Constraints** option

5. Click the **Centered Rectangle** icon:

6. Click a point in the geometry.

7. Drag the cursor to specify the rectangle dimensions.



- Equidistant constraints are applied automatically on the opposed lines accordingly to the center point.
- Dimensional and Geometrical constraints are activated by default.

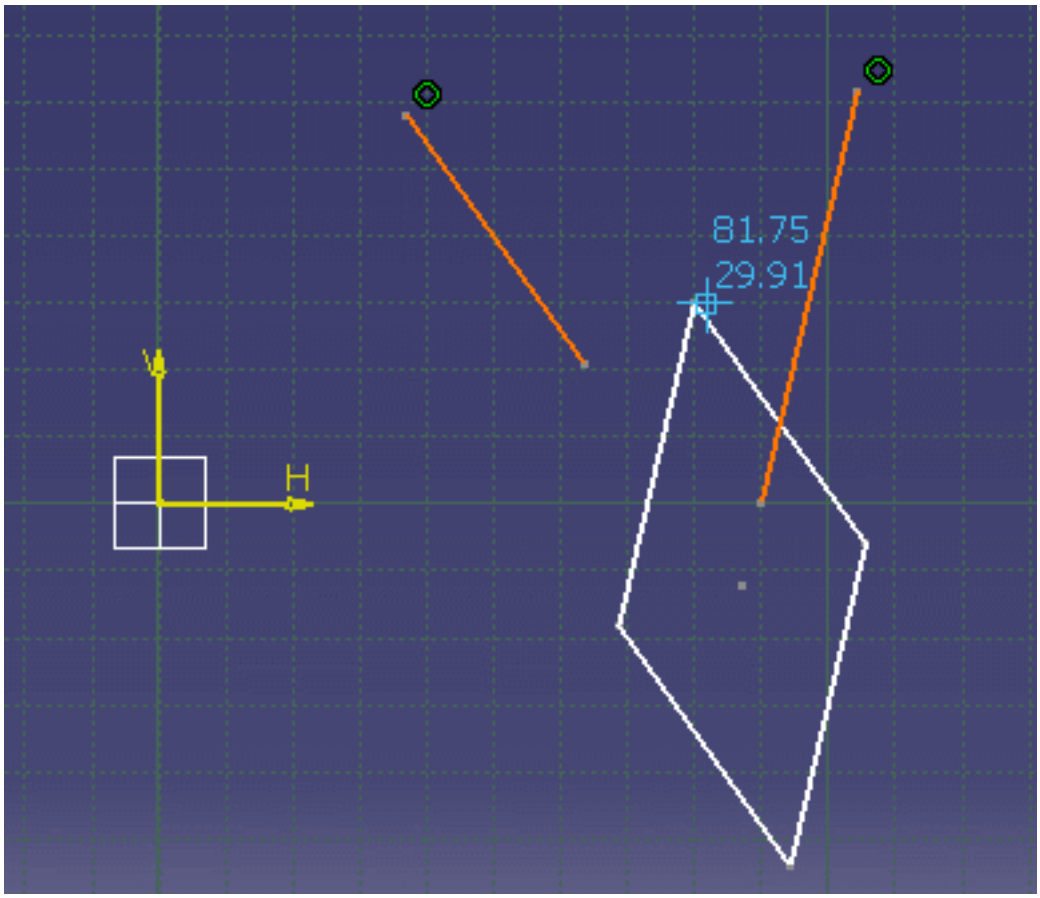
Creating Centered Parallelograms

This task shows you how to create a centered parallelogram.

Enter the Sketcher workbench and create two lines.

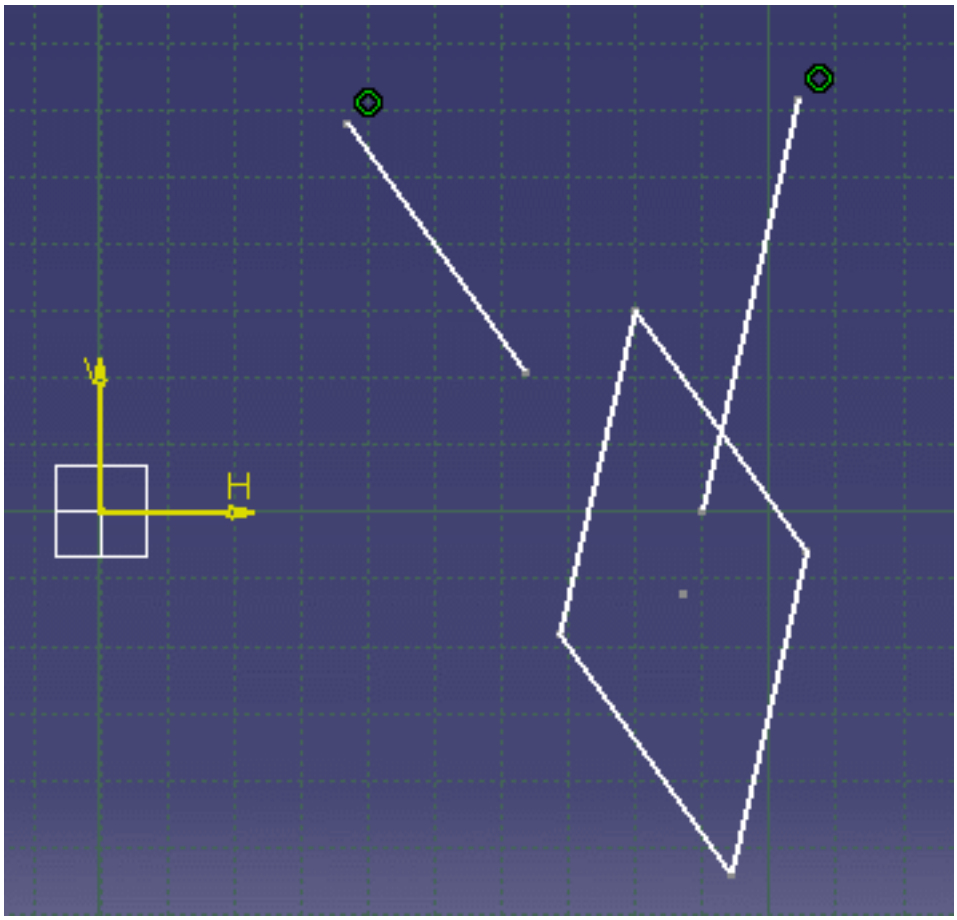
- Ensure that the **Geometrical Constraints** and the **Dimensional Constraints** options are deactivated.

1. Click the **Centered Parallelogram** icon:
2. Select a first line (or an axis).
3. Select a second line (or an axis).
4. Drag the cursor to specify the rectangle dimensions.



The parallelogram is created:

- it is centered on the intersection point of the two lines.
- its edges are parallel to the selected lines.



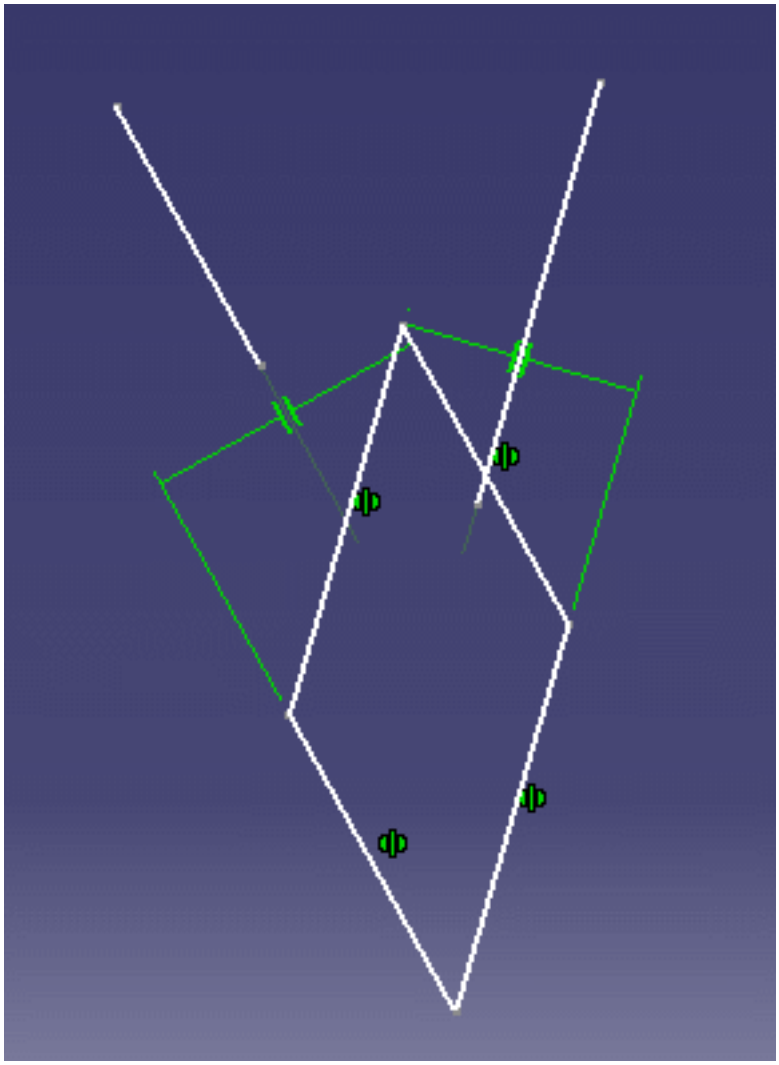
Applying Constraints

5. Activate the **Geometrical Constraints** option and the **Dimensional Constraints** option

6. Click the **Centered Parallelogram** icon:

7. Select the two lines one after the other.

8. Drag the cursor to specify the rectangle dimensions.



- Two parallelism constraints are created as long as two symmetrical constraints which are based on the two lines selected before the parallelogram creation.
- Dimensional and Geometrical constraints are activated by default.

Creating Circles

This task shows how to create a circle. In this task, we will use the Sketch tools toolbar but, of course you can create this circle manually. For this, move the cursor to activate SmartPick and click as soon as you get what you wish.

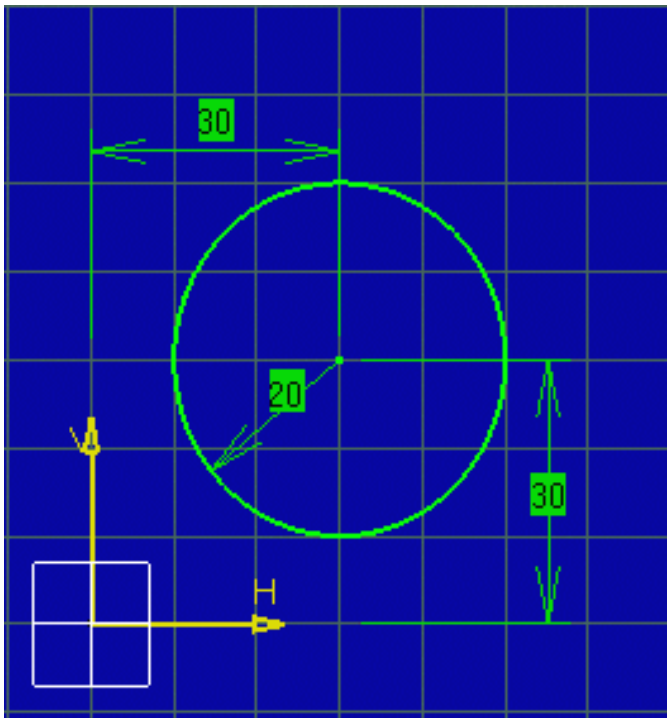
By default, circle centers appear on the sketch. In case you create circles by clicking, if you do not need them, you can specify this.

1. Click the **Circle** icon .

   The **Sketch tools** toolbar displays values for defining the circle.

2. Type in the **Sketcher tools** toolbar for the circle center: **H=30mm**, **V=30mm** and press **Enter**.

3. Type in the **Sketcher tools** toolbar for the point on circle: **R=20mm** and press **Enter**.

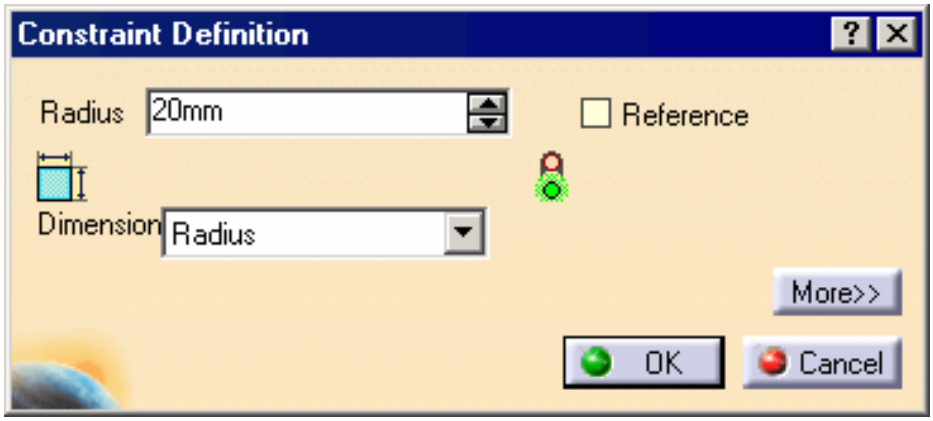


The circle is created.

Constraints are similarly assigned to this circle on the condition you previously activated the **Dimensional Constraints** option in the **Sketch tools** toolbar.

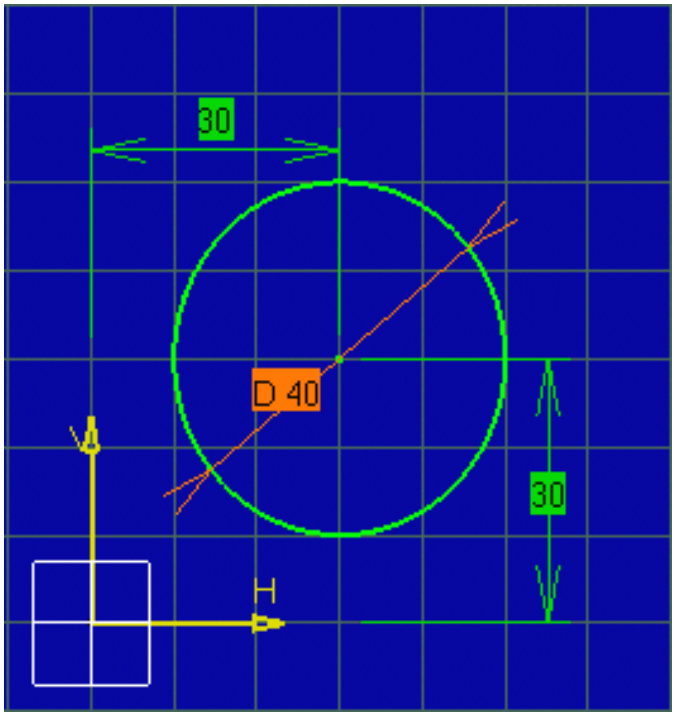
4. Double-click to edit the offset constraint corresponding to the radius.

   The **Constraint Definition** dialog box appears.



5. Select **Diameter** in the **Dimension** combo list and click **OK**.

   The offset constraint type has been changed to diameter.



Copying the Circle Radius Parameters

Once you have created one circle, you can create any other and in the meantime use the radius parameter from the circle first created. To do this:

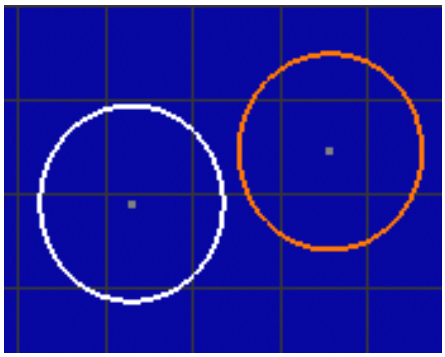
6. Click the **Circle** icon:

7. Right-click the first circle and select **Parameter** > **Copy Radius** from the contextual menu.

   The new circle is automatically created with the radius of the circle first created but not positioned.

8. Click to indicate the second circle location or use the **Sketch tools** toolbars to specify the circle center.

   The new circle is positioned.



Changing the Circle Radius

Once you have created a circle, you can change its radius. To do this, you can:

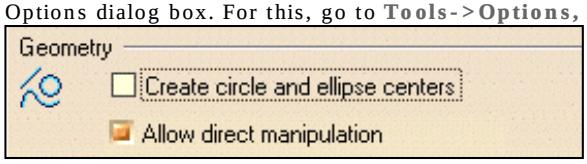
- If the offset constraint corresponding to the radius exists:
  - Double-click the offset constraint and modify the radius value in the **Constraint Definition** dialog box that appears.
- Otherwise:
  - Double-click the circle and modify the radius value in the **Circle Definition** dialog box that appears.
  - Drag the circle until you are satisfied with its new radius.

If the circle center is fixed (or iso-constrained), you can change the circle radius by using one of the methods explained above.

Creating Three Points Circles

This task shows how to create a circle that goes through three points. In this task, we will use the Sketch tools toolbar but, of course you can create this circle manually. For this, move the cursor to activate SmartPick and click as soon as you get what you wish.

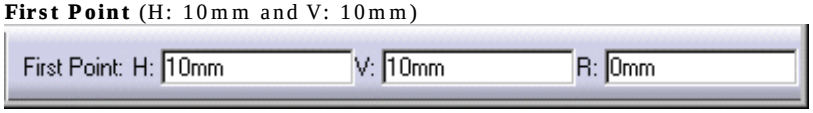
By default, circle centers appear on the sketch. In case you create circles by clicking, if you do not need them you can specify this in the Options dialog box. For this, go to **Tools->Options, Mechanical Design > Sketcher** option (**Sketcher** tab).



1. Click the **Three Point Circle** icon from the Profiles toolbar (Circle subtoolbar).

2. The **Sketch tools** toolbar displays one after the other the values for defining the three points of the circle: values for defining the horizontal (H) and vertical (V) values of a point on the circle or else the radius of this circle.
   Position the cursor in the desired fields and key in the desired values.

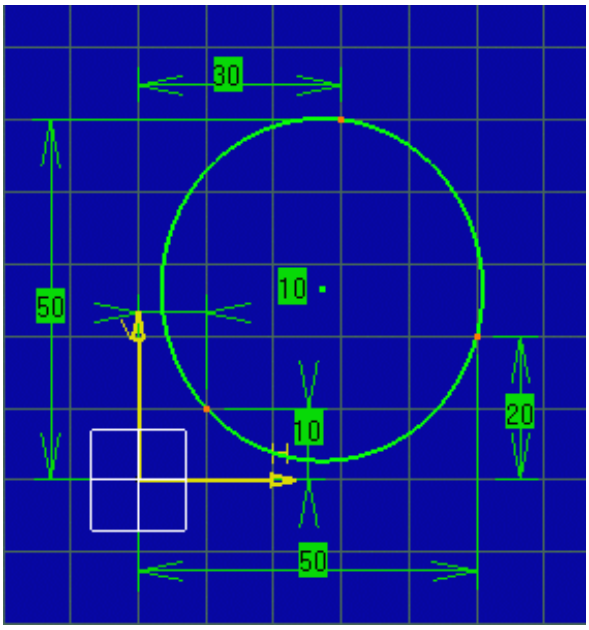
First Point (H: 10mm and V: 10mm)



Second Point (H: 50mm and V: 20mm)

Last Point (H:30mm and V: 50mm)

The three point circle appears:



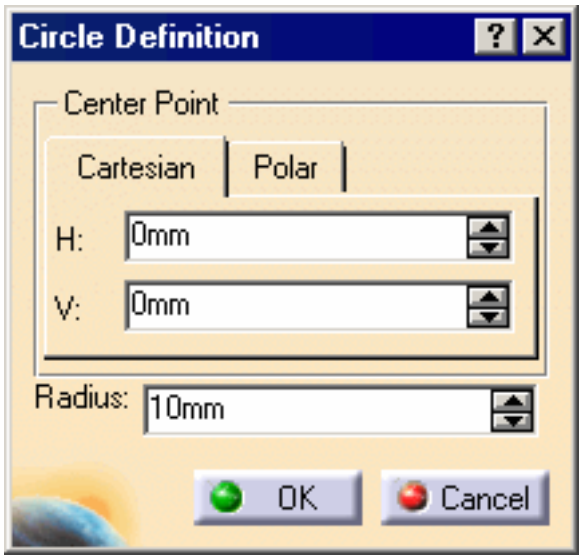
Creating Circles Using Coordinates

This task shows how to create a circle using center point coordinates. In this particular case, we will use cartesian coordinates. Still, you can also use polar coordinates.

By default, circle centers appear on the sketch. In case you create circles by clicking, if you do not need them, you can specify this as indicated in the Customizing section of this User's Guide.

1. Click the **Circle Using Coordinates** icon .

   The **Circle Definition** dialog box appears. The default point coordinates that appear in the **Circle Definition** dialog box are the origin axis coordinates. The default circle radius is **10mm**.

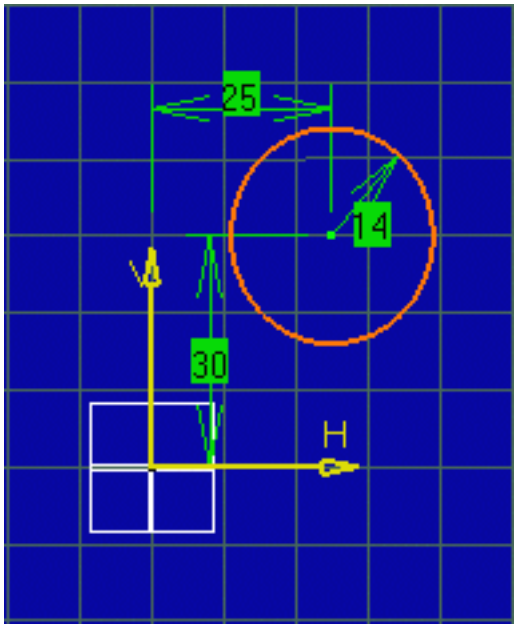


   If, before clicking the **Circle Using Coordinates** icon, you select an existing point, this point will be used as a reference point and the coordinates of the center point will be set from this point.

2. Type in the **Circle Definition** dialog box for the circle center point: **H=25mm**, **V=30mm** and **Radius=14mm**

3. Click **OK**.

   The circle and its center point are created.

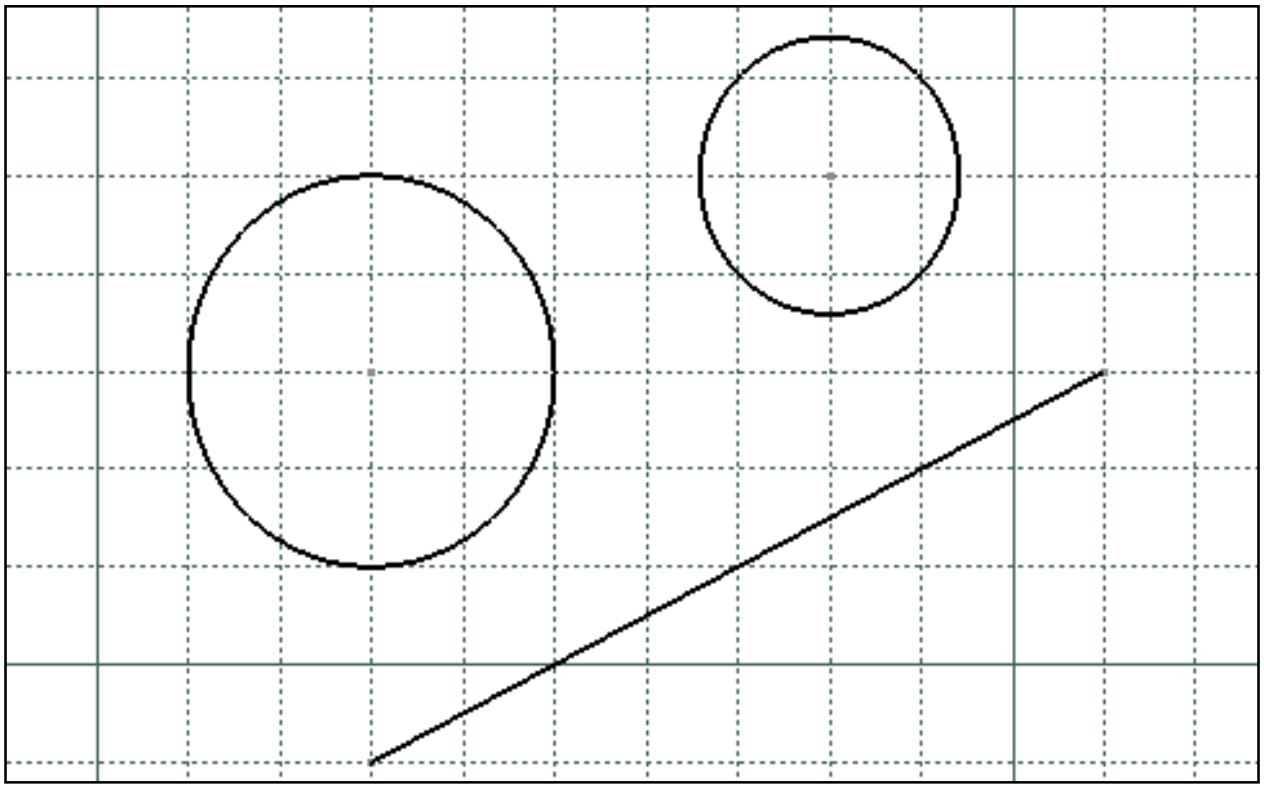

25
14
30
V
H

Creating a Tri-Tangent Circle

This task shows how to create a tri-tangent circle by creating three tangents.

By default, circle centers appear on the sketch. In case you create circles by clicking, if you do not need them, you can specify this, see the customizing section of this user's guide.

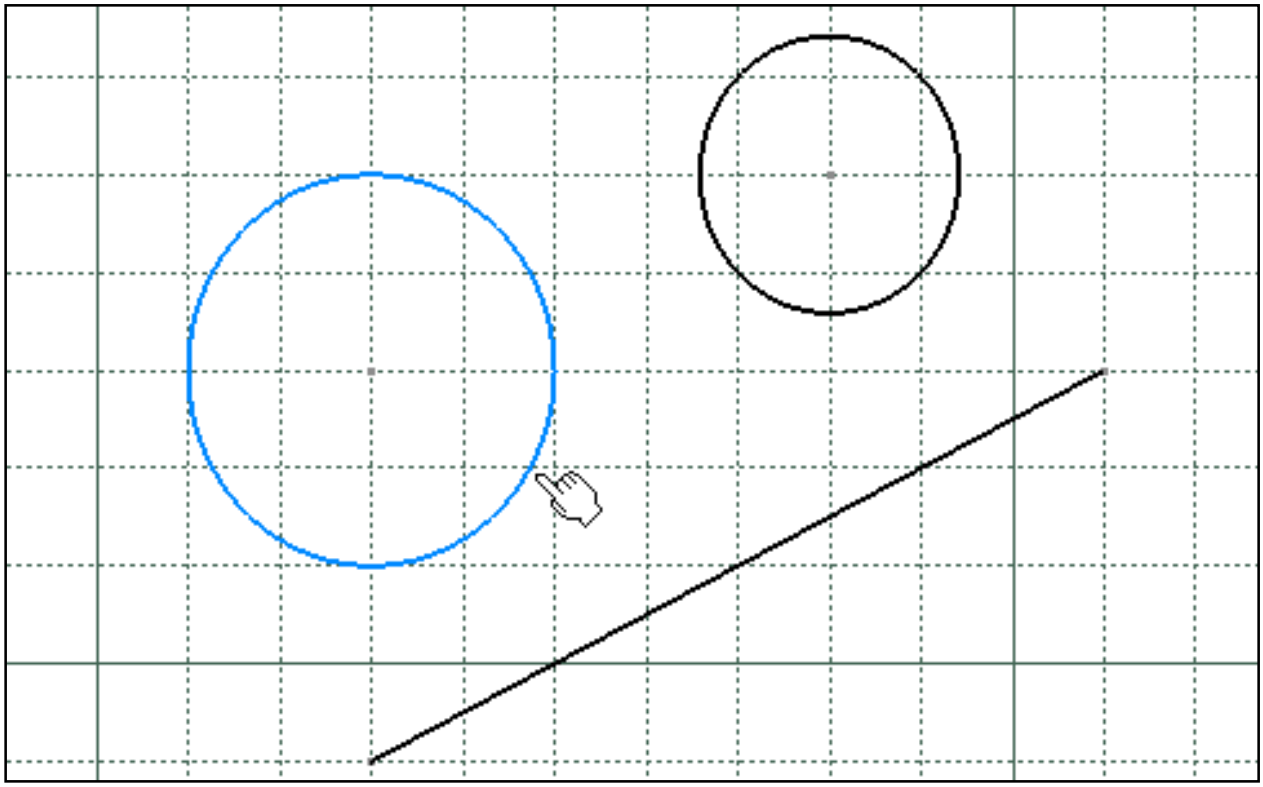
Enter the Sketcher workbench and create two circles and a line.



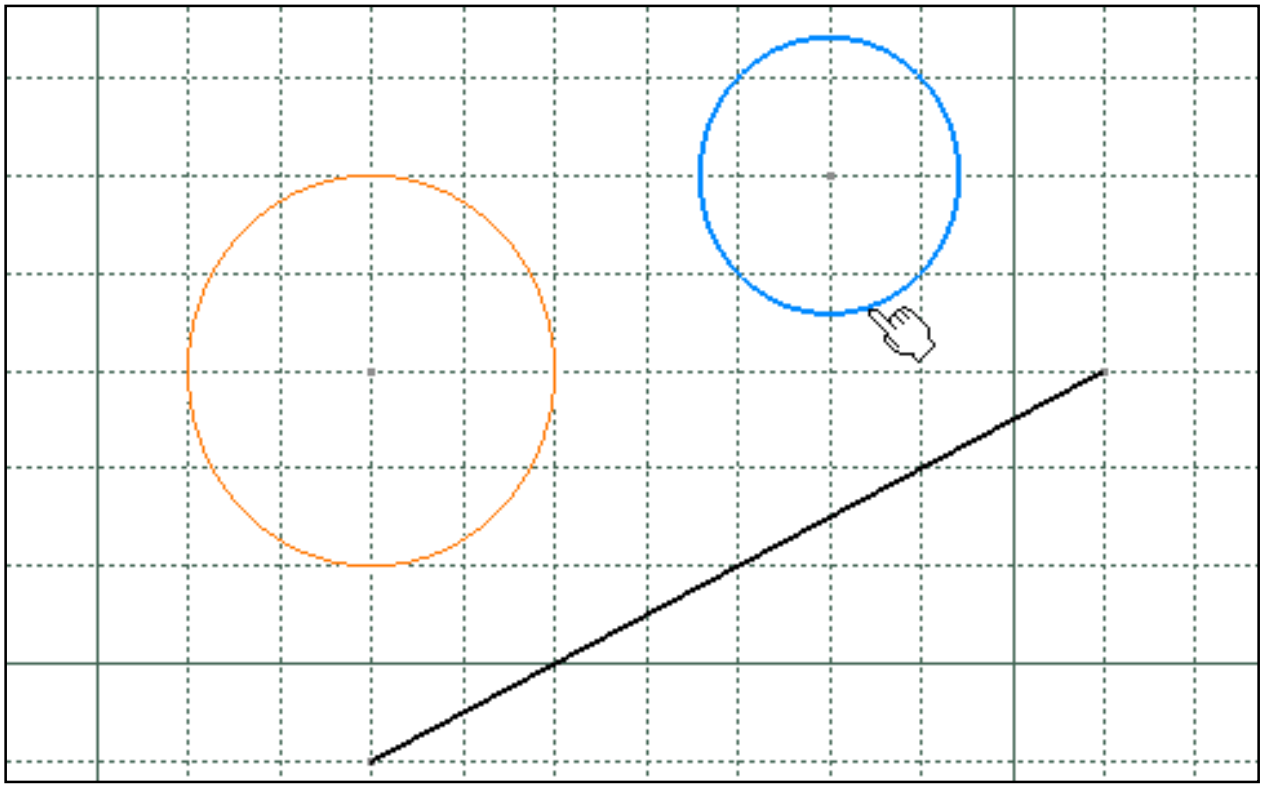
1. Click the **Tri-Tangent Circle** icon:

   The **Circle Definition** dialog box appears. The default point coordinates that appear in the **Circle Definition** dialog box are the origin axis coordinates. The default circle radius is **10mm**

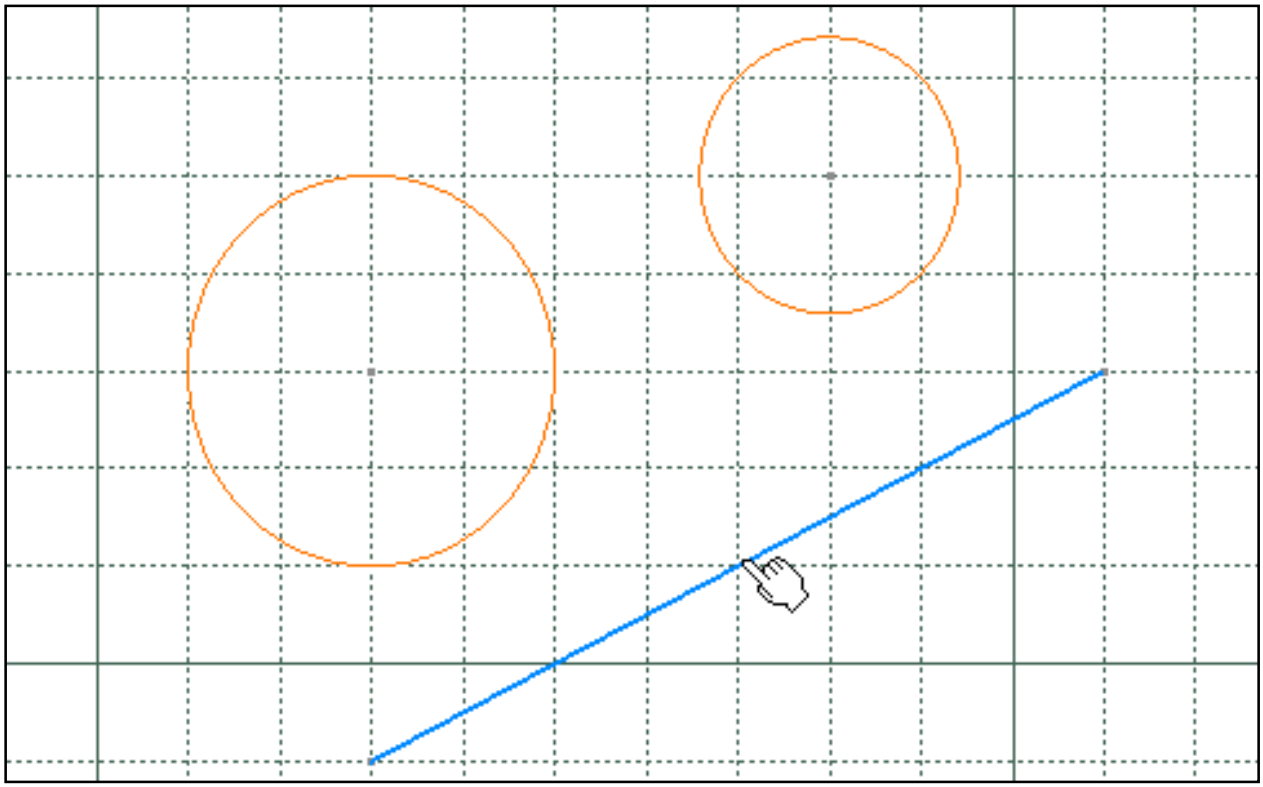
2. Select the first circle.



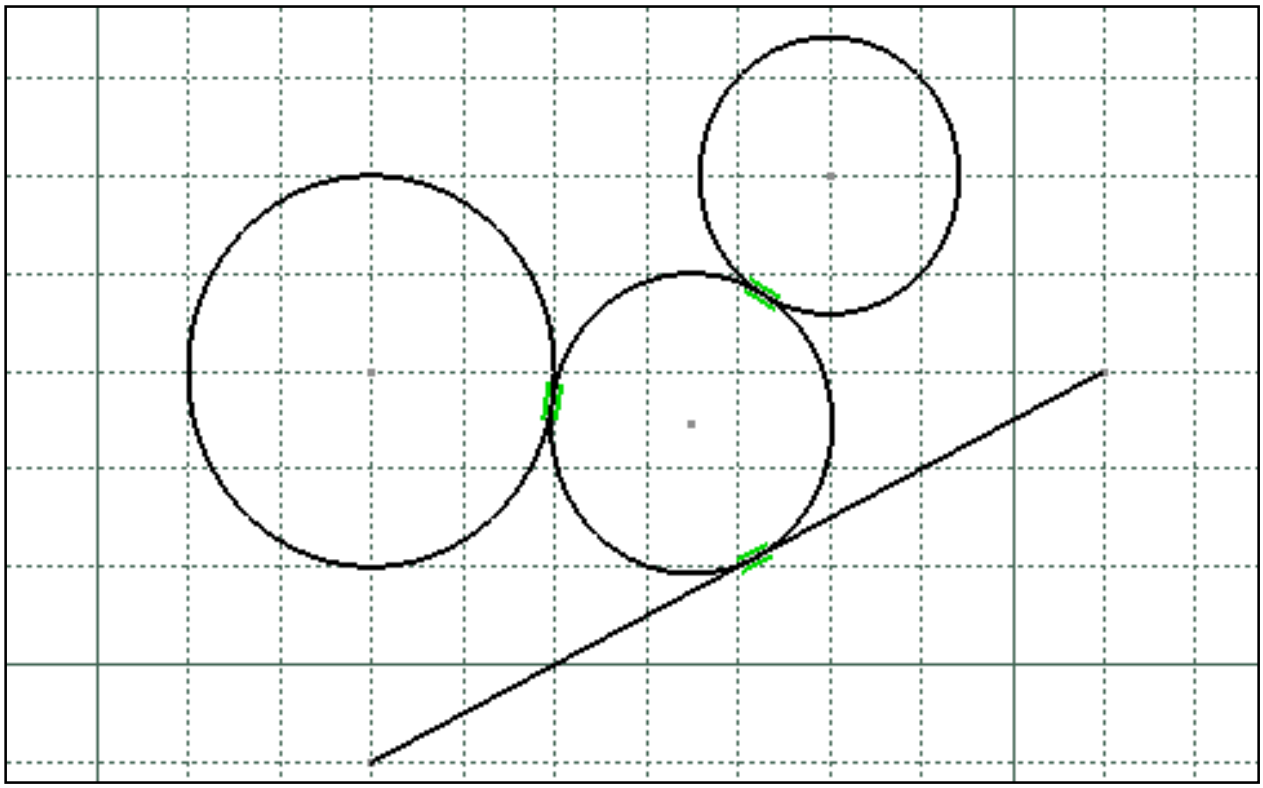
3. Select the second circle.



4. Select the line.



The tri-tangent circle is created.



Constraints are similarly assigned to this circle on the condition you previously activated the **Geometrical Constraints** option in the **Sketch tools** toolbar.

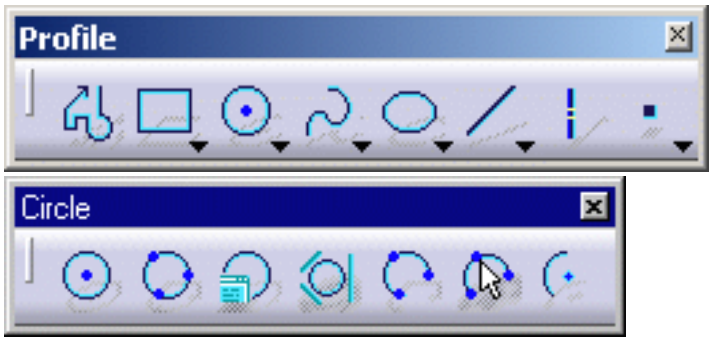
- If you select a point the created constraint is a coincidence.
- As there are several tangencies for a considered curve (circle, conic, spline, etc), tangent is created as close as possible to where you clicked on the curve.

Creating Three Points Arcs

This task shows how to create an arc using three reference points in order to define the required size and radius. In this task, we will use the Sketch tools toolbar but, of course you can create this arc manually. For this, move the cursor to activate SmartPick and click as soon as you get what you wish.

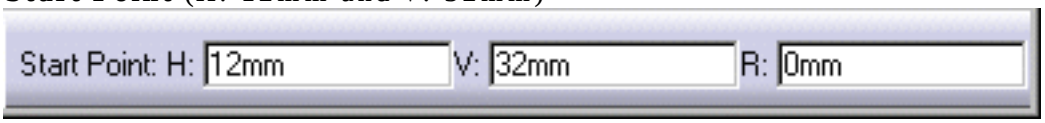
By default, arc centers appear on the sketch and are associative. In case you create arcs by clicking, if you do not need them you can specify this in the Tools->Options dialog box. For this, go to **Tools->Options, Mechanical Design > Sketcher** option at the left of the dialog box (**Sketcher** tab)

1. Click the Three Point Arc icon from the Profiles toolbar (Circle subtoolbar).

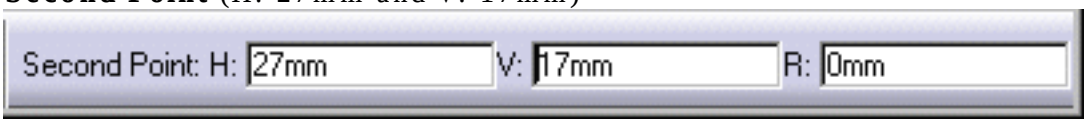


2. The Sketch tools toolbar will display one after the other values for defining the three points of the circle: defining the horizontal (H) and vertical (V) values of three points on the arc. Position the cursor in the desired fields and key in the desired values.

Start Point (H: 12mm and V: 32mm)

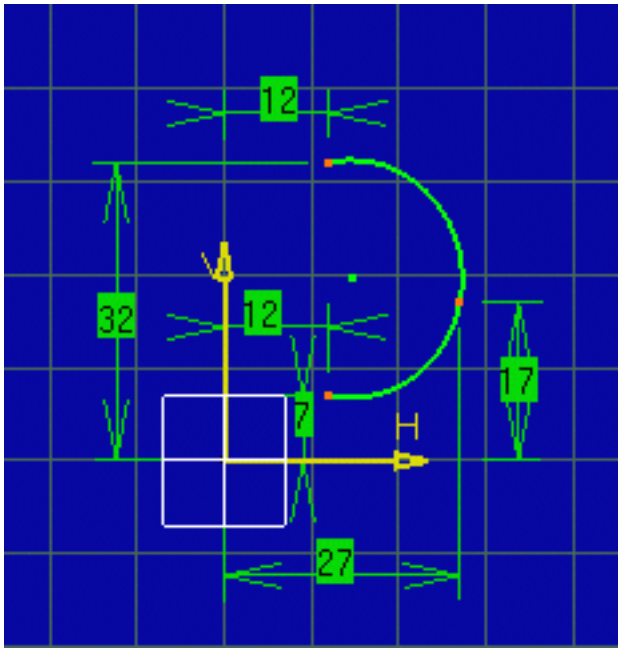


Second Point (H: 27mm and V: 17mm)



End Point (H: 12mm and V: 7mm)

The arc results as shown here.



Creating Three Points Arcs Using Limits

This task shows how to create a three point arc by starting creating the arc limits first. In this task, we will use the Sketch tools toolbar but, of course you can create this arc manually. For this, move the cursor to activate SmartPick and click as soon as you get what you wish.

By default, arc centers appear on the sketch and are associative. In case you create arcs by clicking, if you do not need them you can specify this in the Options dialog box.

To do so, go to **Tools->Options, Mechanical Design > Sketcher** option at the left of the dialog box (**Sketcher** tab).

1. Click the Three Point Arc Starting with Limits icon from the Profiles toolbar (Circle subtoolbar).

The Sketch tools toolbar will display one after the other values for defining the three points of the circle: values for defining the horizontal (H) and vertical (V), values for defining the arc start, end or second points

or else the radius of this arc.

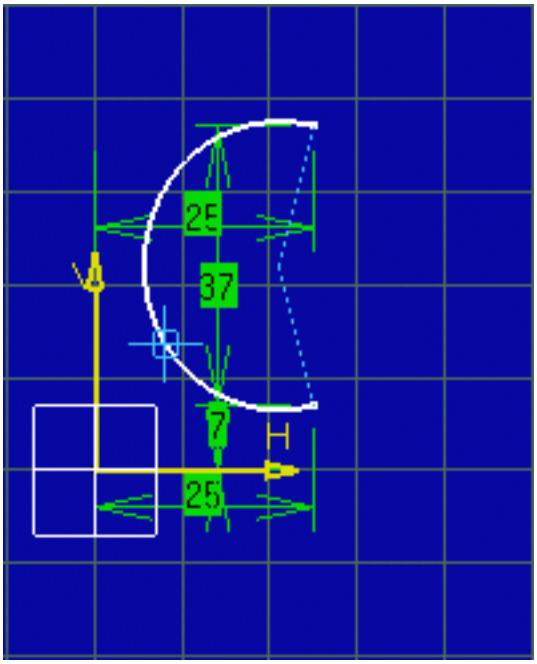
2. Position the cursor in the desired fields and key in the desired values.

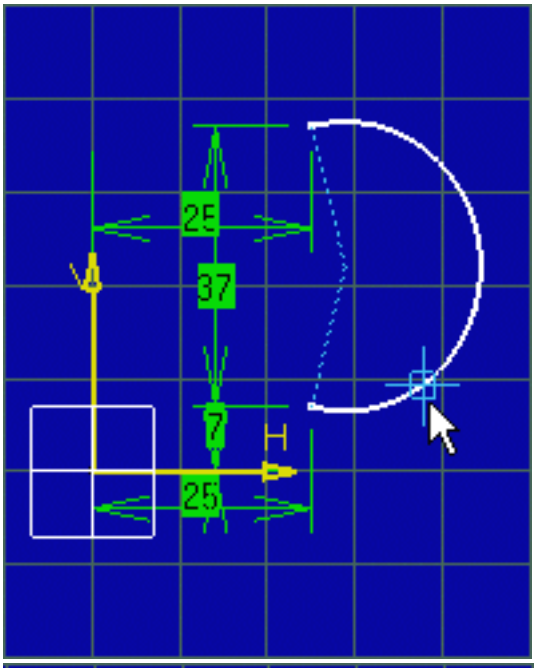
Start Point (H: 25mm and V: 37mm)

End Point (H: 25mm and V: 7mm)

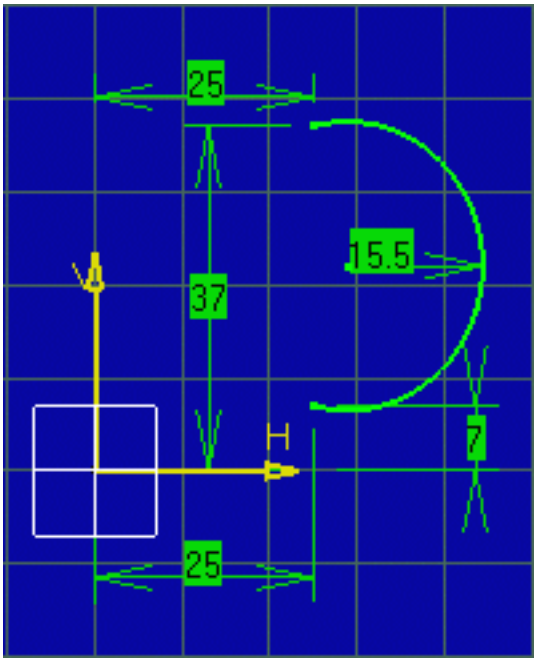
Second Point (R: 15.5mm)

3. Drag the cursor and click to create the arc intermediate point (the point which the arc will go through).





The three point arc appears:

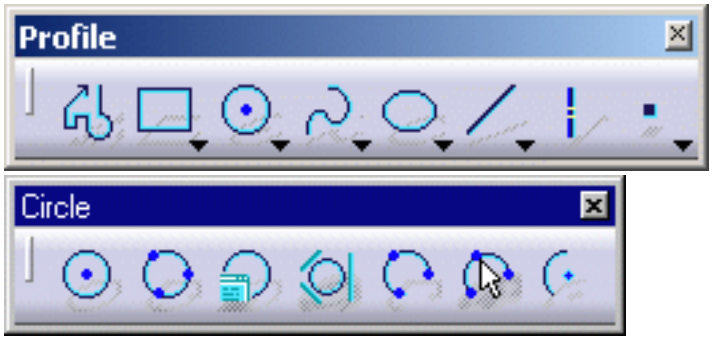


Creating Arcs

This task shows how to create an arc. In this task, we will use the Sketch tools toolbar but, of course, you can create this arc manually. For this, move the cursor to activate SmartPick and click as soon as you get what you wish.

By default, arc centers appear on the sketch and are associative. In case you create arcs by clicking, if you do not need them you can specify this in the Options dialog box. For this, go to **Tools->Options, Mechanical Design > Sketcher** option at the left of the dialog box (**Sketcher** tab).

1. Click the Arc icon from the Profiles toolbar (Circle subtoolbar).



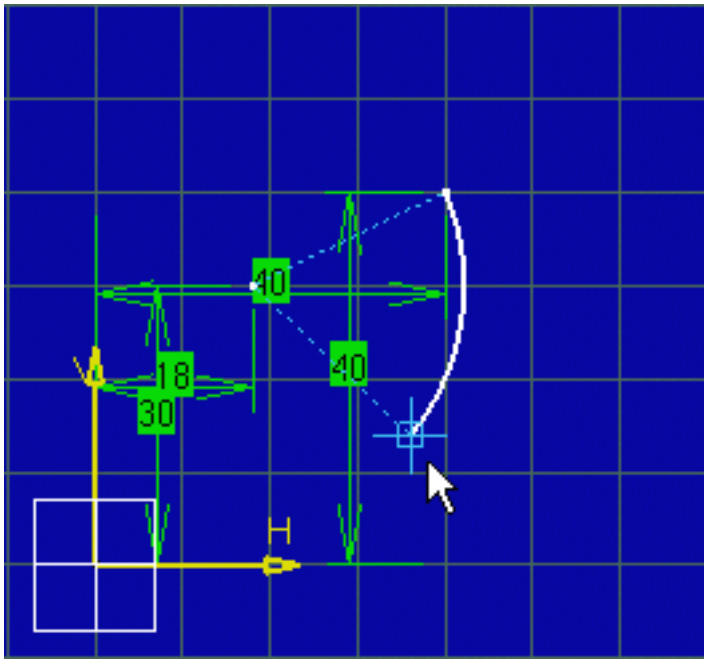
2. The Sketch tools toolbar now displays values for defining one after the other the arc center point, start point and end point. Position the cursor in the desired field (Sketch tools toolbar) and key in the desired values.

Arc Center

Start Point

For example, enter H: 18mm and V: 30mm (Circle Center) and then H: 40mm and V: 40mm (Start Point).

The arc center and start point appear.

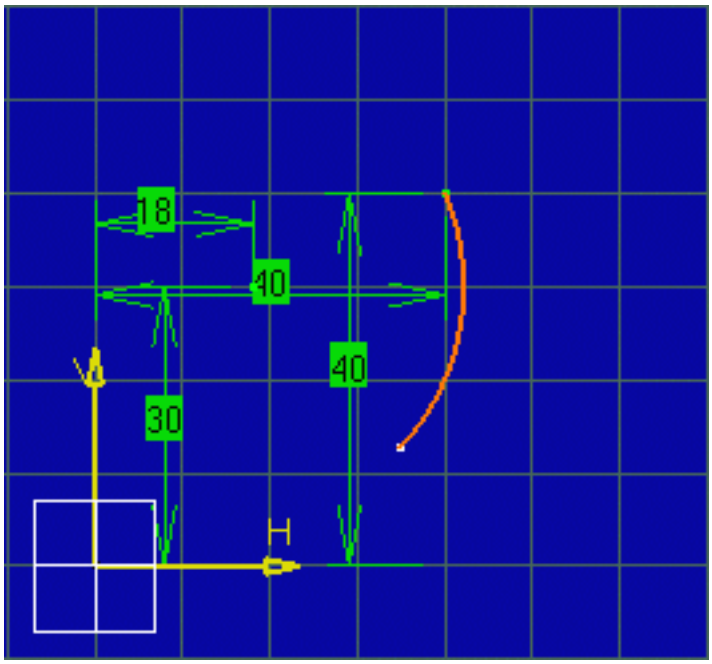


The arc will now appear according to the position you assign to the cursor. In this particular case, the cursor position is at the bottom extremity of the arc.

End Point

For example, enter S: -70deg (Angular Sector).

The arc appears as shown here.



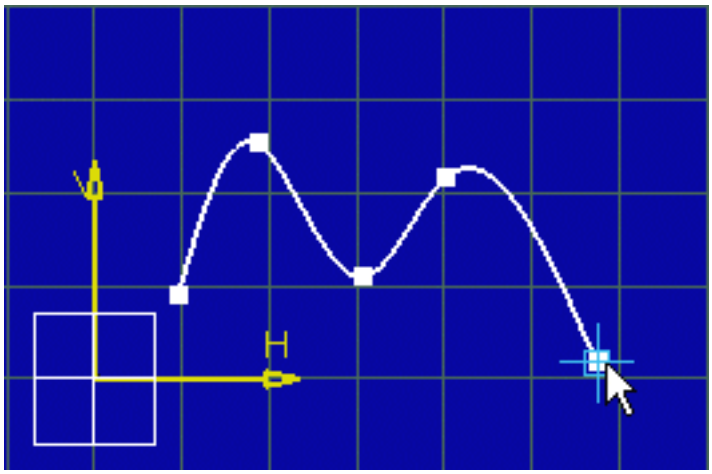
Creating Splines

This task shows you how to create a spline and then modify the spline control points (coordinates or clicking).

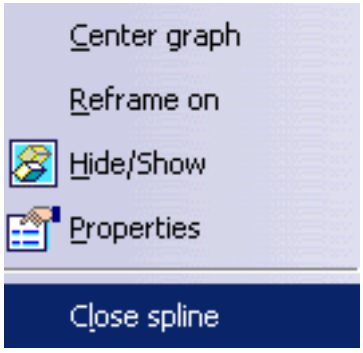
Creating a spline

1. Click the Spline icon from the Profiles toolbar.

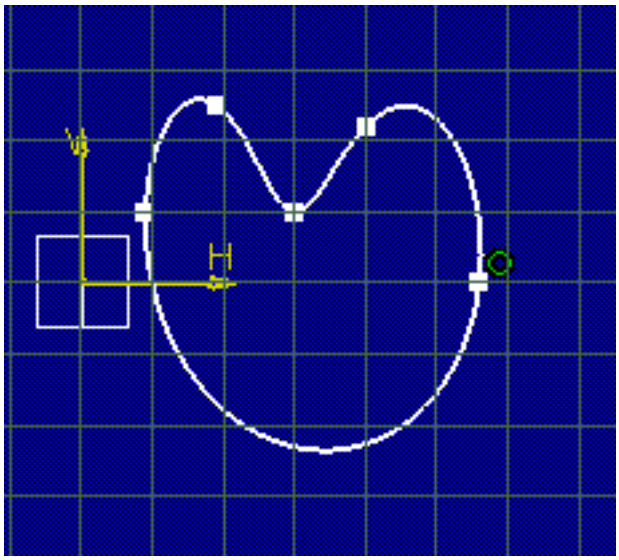
2. Click in the geometry to indicate the points through which the spline goes.
3. Double-click the last point you have created to finish the spline creaton. (Clicking again on the Spline icon or another command also ends the spline creation.)



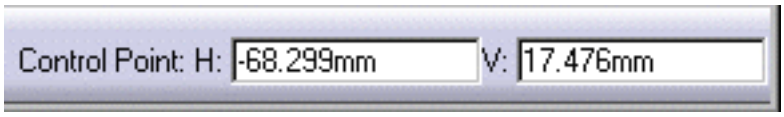
At any time when creating a spline, you can close it by right-clicking the last point and selecting **Close spline** from the contextual menu.



The spline is closed in such a way that it is continuous in curvature.



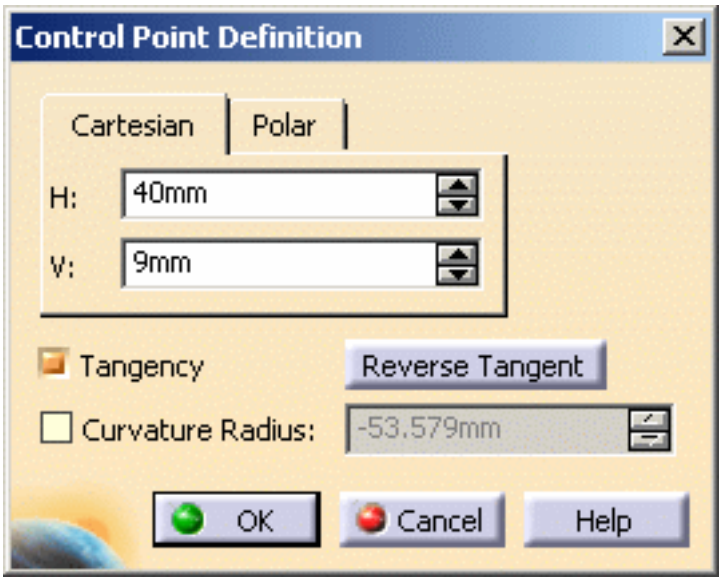
- Keep in mind that using the displayed **Sketch tools** toolbar also allows creating a spline. In addition, two constraints will be created (**H** and **V**).



Modifying the spline control points

1. Double-click the control point you wish to edit.

   The **Control Point Definition** dialog box appears.



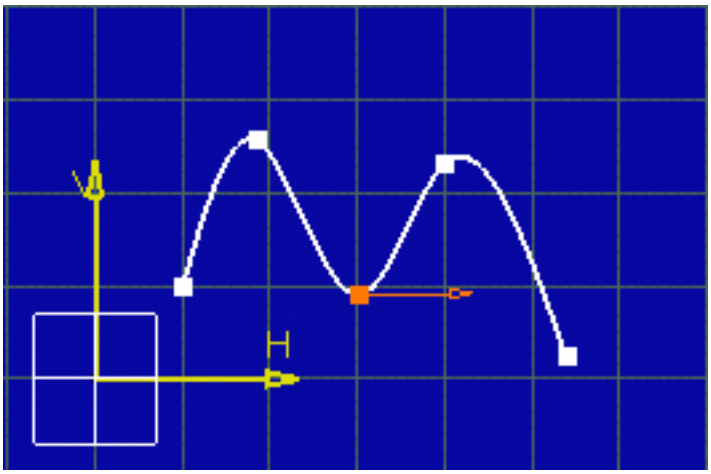
2. Enter new coordinates. For example, **v: 9mm** (vertical).

3. Check the **Tangency** option to impose a tangency on this control point.

   You can invert the tangent direction clicking the **Reverse tangent** button.

4. Click **OK**.

The point is moved and an arrow appears on this point to indicate a tangency.



You can also check the **Curvature** option to activate the **Curvature editor** and impose a curvature on the previously selected control point.

Keep in mind that selecting a point then dragging it will modify the spline shape.

Tangents can be constrained.

Connecting Curves with a Spline

This task shows you how to connect two elements of the curve type, using a connecting curve (a spline) that goes through their end points.

A connecting curve is associative, and it can be continuous in point, in curvature or in tangency with its support curves. You can define the tension value and the direction of the continuity at each connecting point, as well as add constraints to the connecting curve. Moving a connecting curve will change the shape of the support curves accordingly.

Open the Connect_Curves.CATPart document.

1. Click the **Connect** icon .

   The **Sketch tools** toolbar now displays connection and continuity options for defining the connection:

   Connection options are:

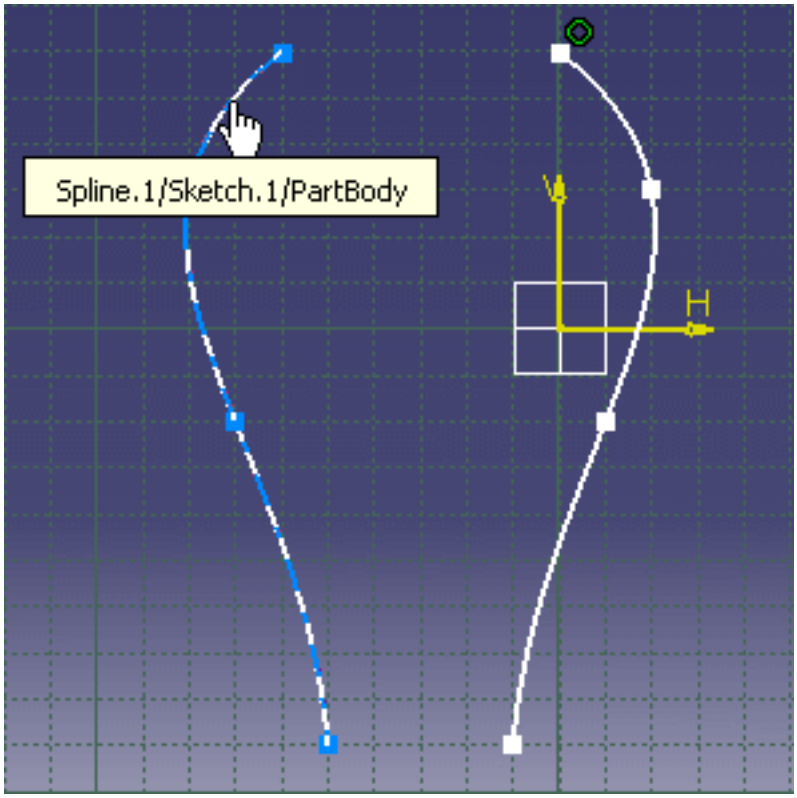
   - **Connect with an Arc**:
   - **Connect with a Spline**: (selected by default).

   Continuity options are (available with **Connect with a Spline** option only):

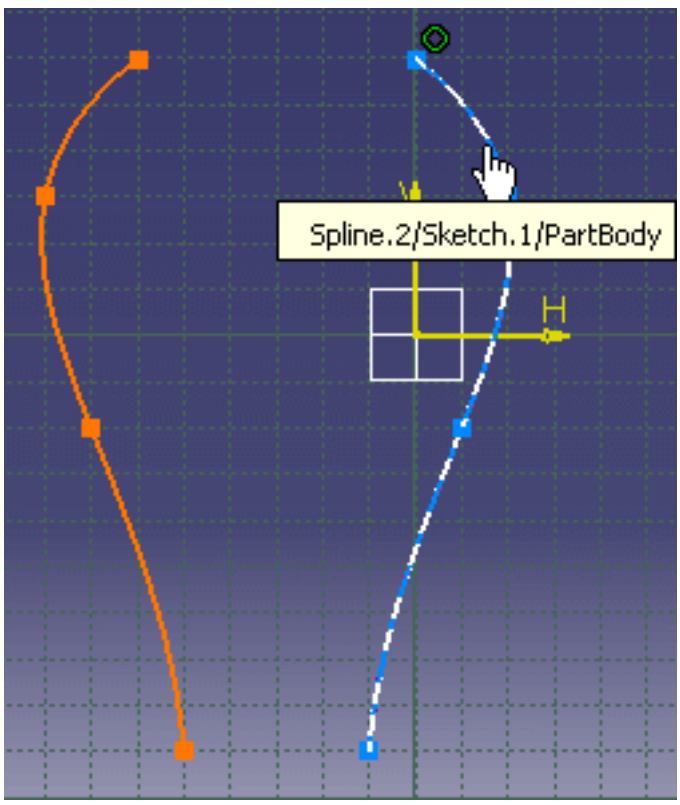
   - **Continuity in point**:
   - **Continuity in tangency**:
   - **Continuity in curvature:** (selected by default.)

   Tension value corresponds to a multiplying coefficient applied to the tangent vector norm (available with **Continuity in tangency** and **Continuity in curvature** options only). The default value is 1 and the 0 value corresponds to a continuity in point.

2. Select the first spline to be connected.

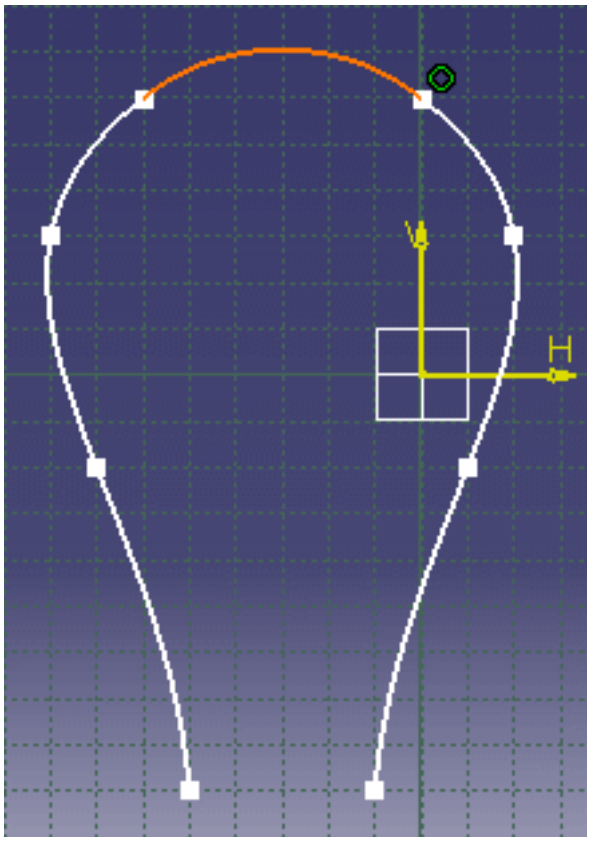


3. Select the second spline to be connected.



Locations where you click to select the first and the second element are important: the closest point to where you click will be automatically used as the starting point and the end point of the connecting curve. Always click close to the point you want to connect, or click the point itself.

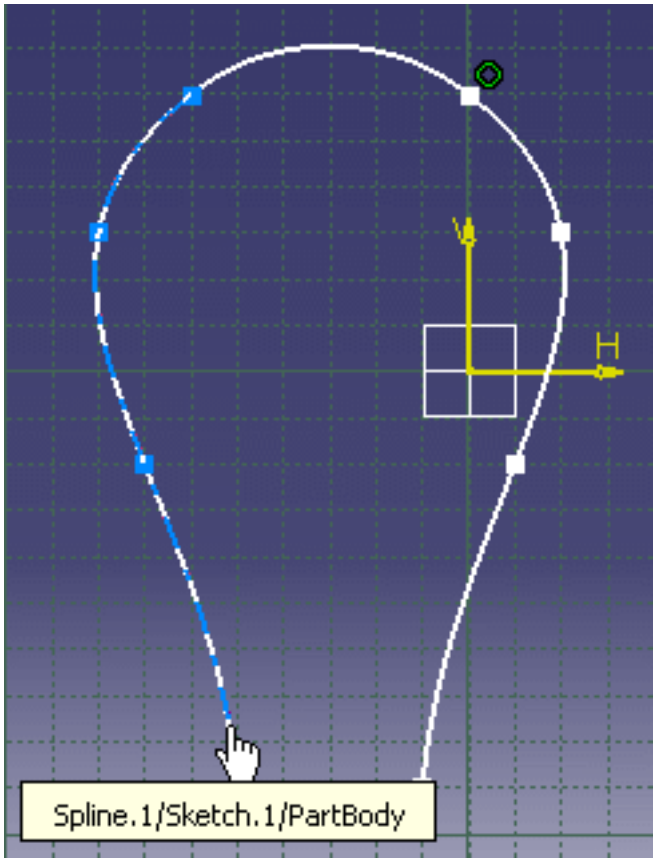
A connecting spline appears: it is continuous in curvature to both selected elements.



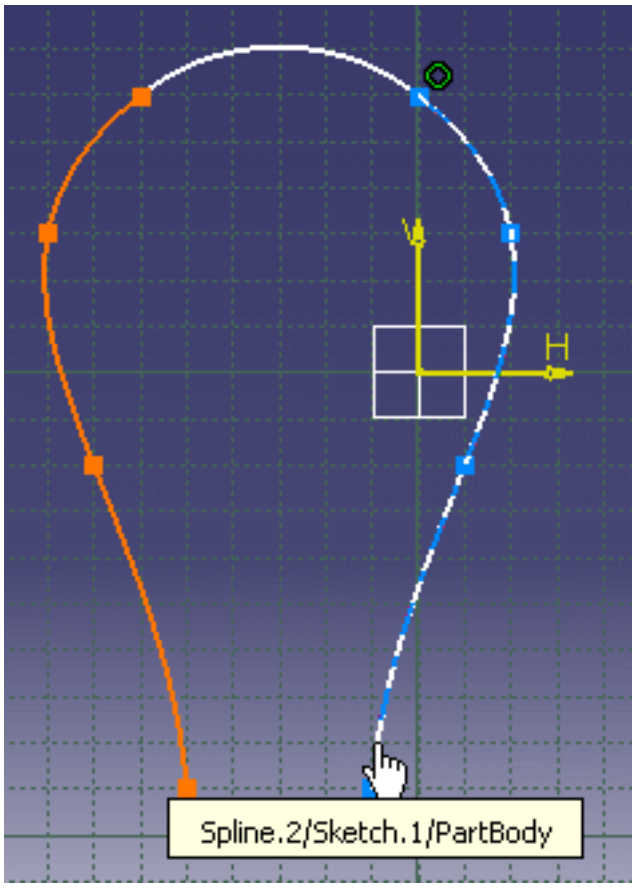
4. Click the **Connect** icon .

5. Select the **Continuity in point** option .

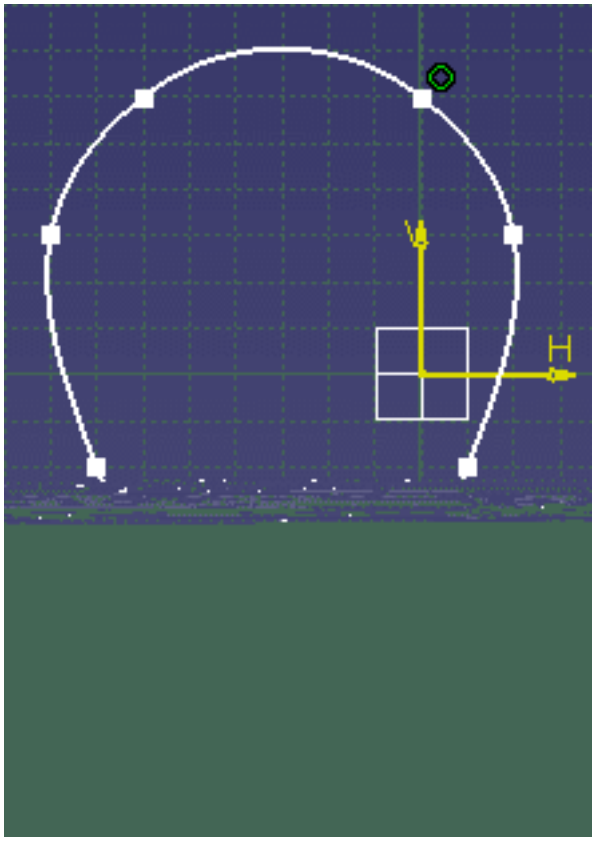
6. Select the first spline to be connected.



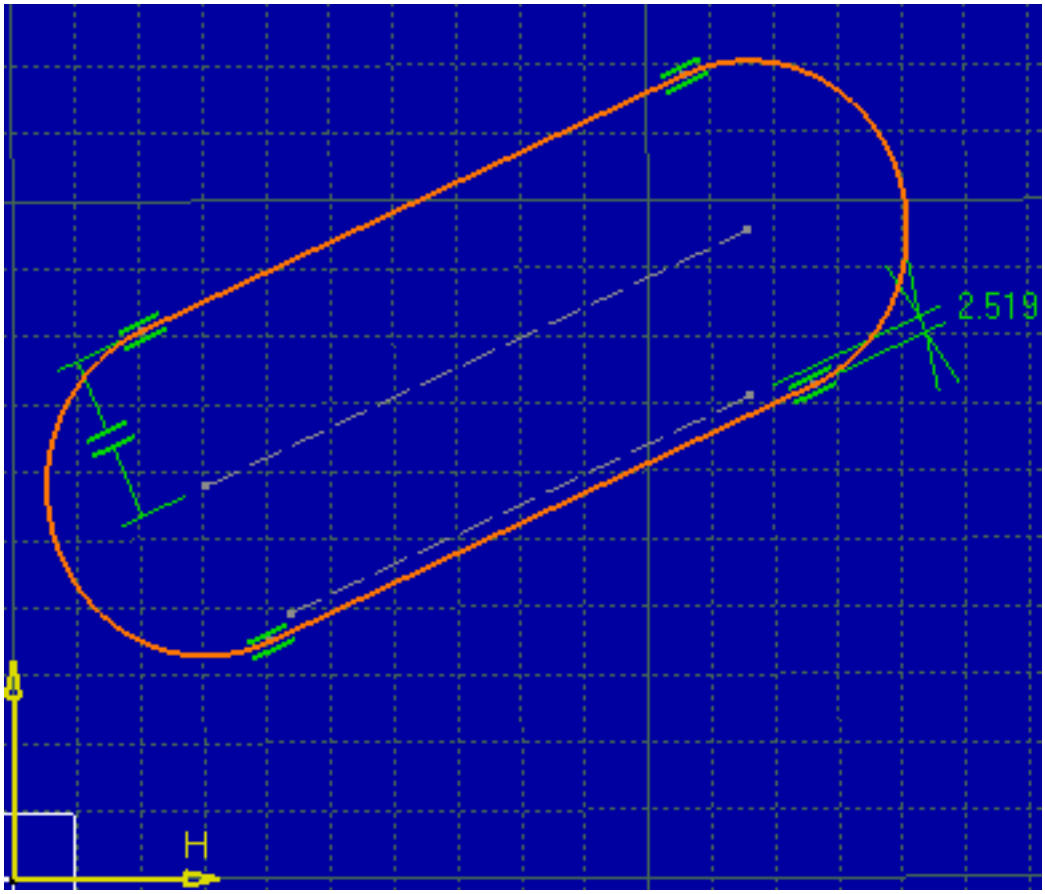
7. Select the second spline to be connected.



A connecting spline appears: it is continuous in point to both selected elements.



- You can edit the connecting curve, as well as add constraints to it.
- You can also move the connecting curve: in this case, the shape of the support elements will change accordingly, as shown here for example.
- You cannot trim or break a connecting curves.



- Internal constraints are preserved but revalued.
- External constraints:
  - geometrical constraints are deleted.
  - dimensional constraints are modified and revalued.

Offsetting Elements

This task shows you how to duplicate an element of the following type: line, arc or circle.

You can also duplicate by offset one of the following: an edge, a face (all the boundaries of this face are offset) or a geometrical feature (for example, by selecting a join or another sketch in the specification tree).

Select a topic:

- Offset 2D geometry,
- Use offset tools,
- Offset 3D geometry,
- Modify a 3D geometry offset.

Offsetting 2D Geometry

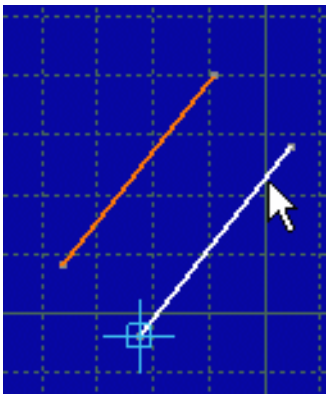
Create a line.

1. Click the **Offset** icon from the **Operations** toolbar (**Transformation** subtoolbar).

OR

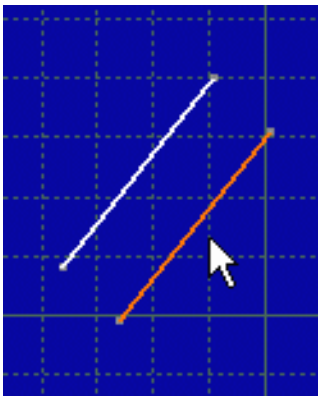
1. Select the **Insert** >**Operation**>**Transformation**>**Offset** command from the menu bar.

2. There are two possibilities, depending on whether the line you want to duplicate by offset is already selected or not:
   - If the line is already selected, the line to be created appears immediately.
   - If the line is not already selected, select it. The line to be created appears.



3. Select a point or click where you want the new element to be located.

   The selected line is duplicated. Both lines are parallel.



- If you were offsetting circles or arcs, these two circles would be concentric.

- If the **Geometrical Constraints** icon is active in the **Sketch tools** toolbar when offsetting an element, constraints are automatically created, based on the type of element you are offsetting. Thus, if you move an element, or change its geometry, the other element will be moved or modified accordingly.

Using offset tools

You can also apply one or more offset instances to profiles made of several elements:

- by using tangency propagation or point propagation,
- by creating an offset element that is tangent to the first one,
- by creating several offset instances.

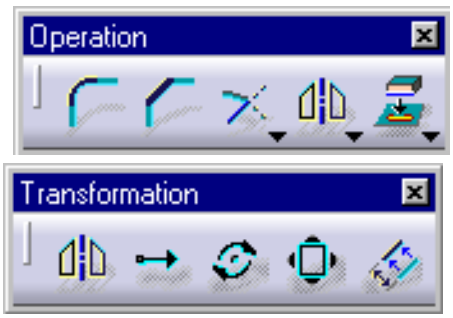
This is not true for generated elements (Generative Drafting workbench).

If the multi-selected elements do not make up a closed profile, the offset will be applied to the selected elements only. As a result, you will have as many offset elements as the first multi-selected elements.

Previews are not available when creating several offset instances (i.e. when the value in the **Instance(s)** field of the **Sketch tools** toolbar is higher than one).

Open the Offset.CATPart document.

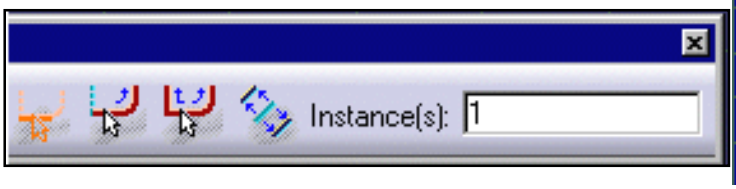
1. Click the **Offset** icon from the **Operations** toolbar (Transformation subtoolbar).

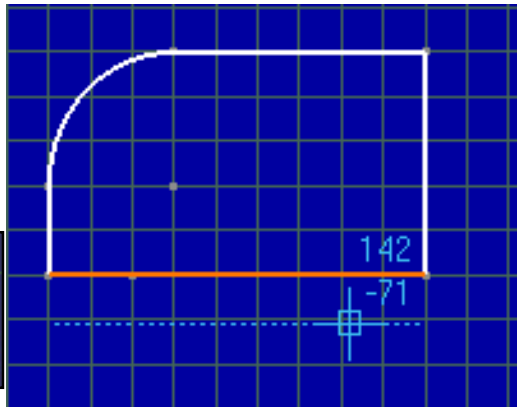


2. Select the desired option from the displayed **Sketch tools** toolbar and if needed, enter the desired number of instances. (These options are described further down in this section).
3. Select the element you want to offset. The element to be created is previewed.
4. Select a point or click where you want the new element to be located.

To offset a single element:

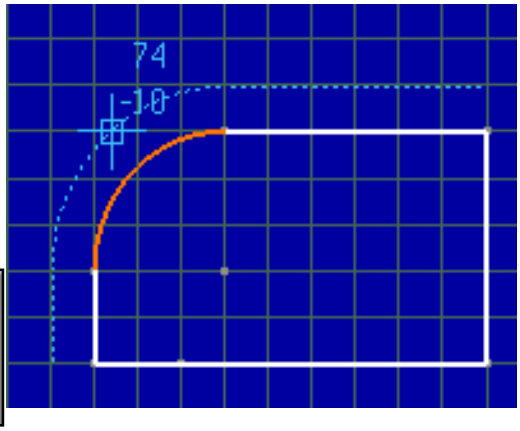
Activate the **No Propagation** icon.





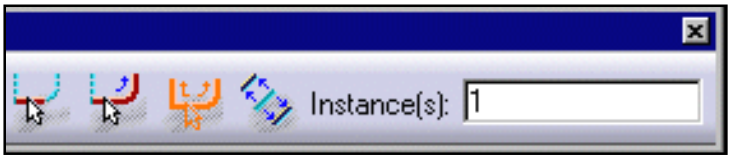
To offset an element and elements which are tangent to it:

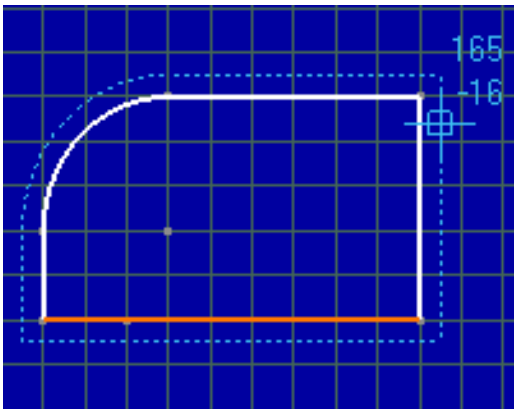
Activate the **Tangent Propagation** icon.



To offset an element using Point Propagation:

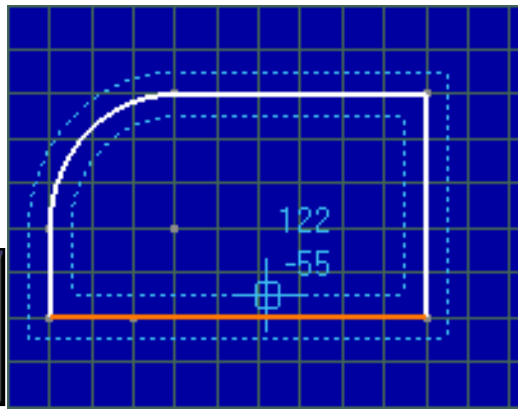
Activate the **Point Propagation** icon.





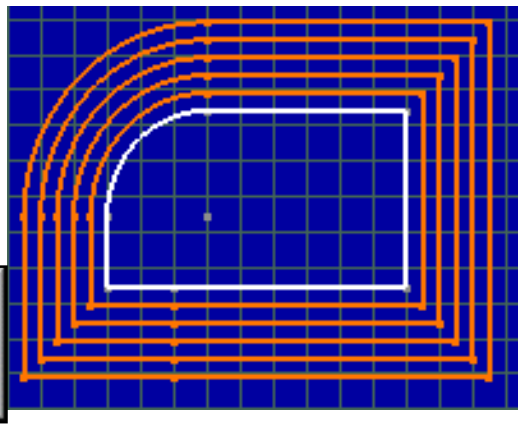
To offset an element symmetrically to another:

Activate the **Both Side Offset** icon.

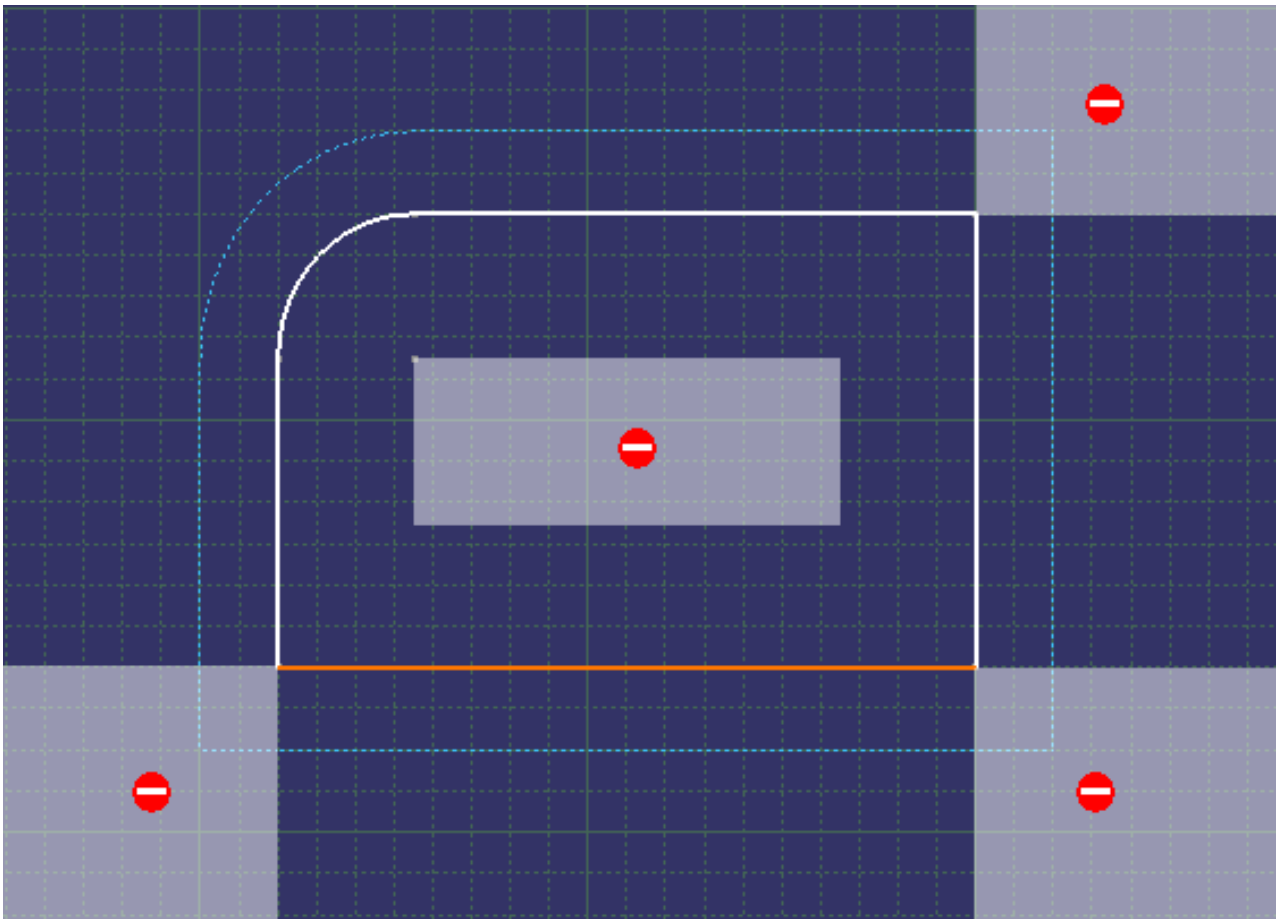


To offset and duplicate multiple elements:

Type the number of elements you want to create in the **Instance(s)** field.



Note that if you position the cursor outside the zone that is allowed for creating a given element, the symbol appears.



Drafting Workbench

You can create offset geometry using 2D component elements and dress-up elements (axis lines, center lines and threads). Note that by doing this, you will not create offset 2D components or dress-up elements, but you will create offset geometry.

- You can offset them only element by element.
- You cannot offset complex curves.
- This will only work if you first select the command and then the element to offset.

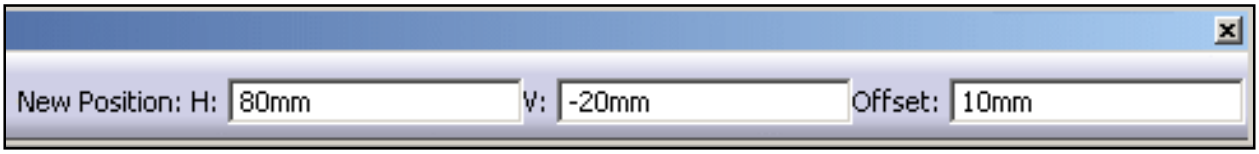
Offsetting 3D Geometry

You can create an associative offset with a 3D element.

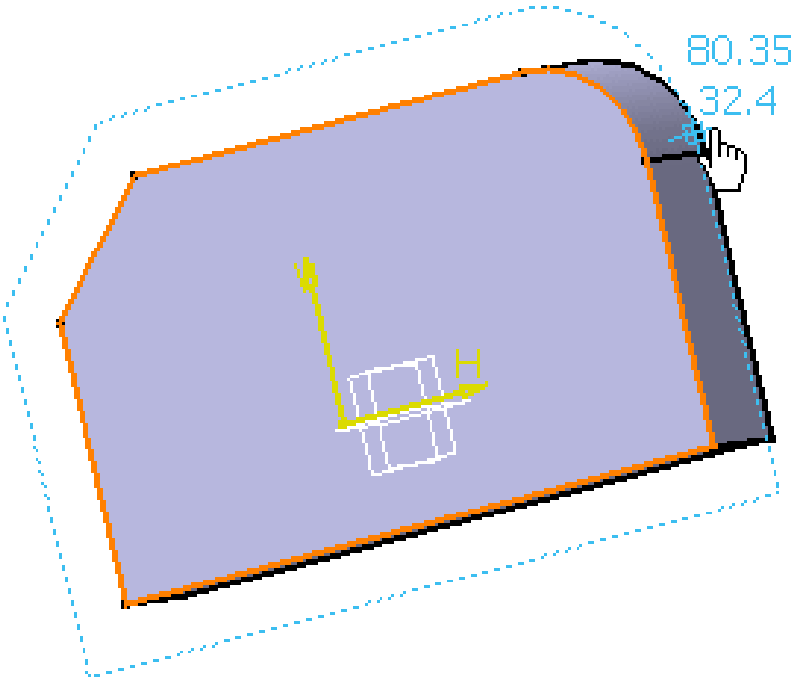
Open the Offsetpad.CATPart document.

1. Click the **Offset** icon from the **Operations** toolbar (**Transformation** subtoolbar).
2. Select the 3D surface to offset, Face.1 for example. The profile to be created is previewed.
3. You can do one of the following:

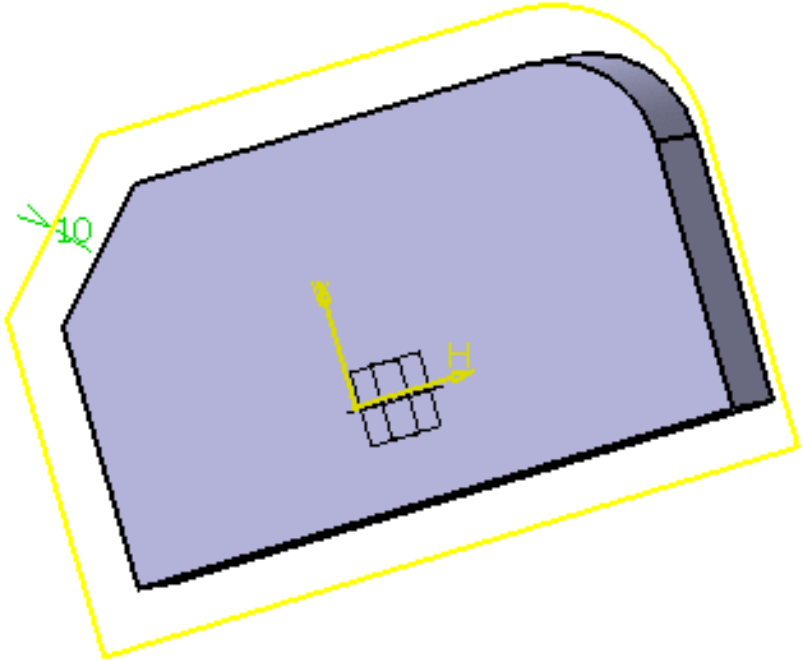
- specify the offset position and value in the **Sketch tools** toolbar and press Enter to validate.



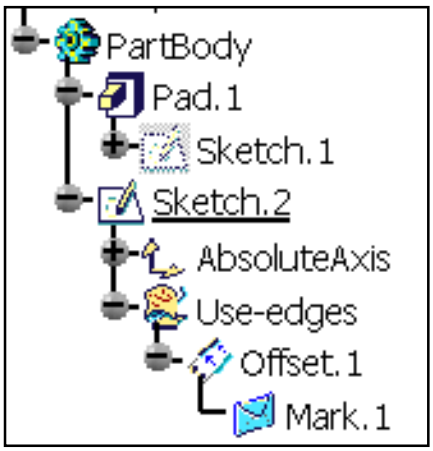
- drag the cursor till the correct offset appears in the sketch, then click to validate the position.



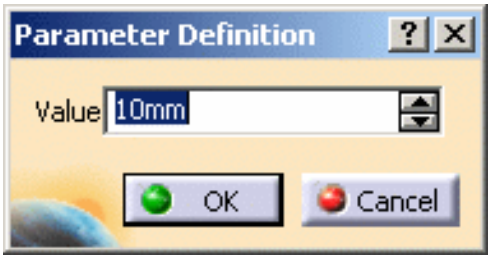
The offset is created, with the offset value displayed.



It appears as Mark.1 in the specification tree:



If you want to edit the offset value, you can double-click it and enter a new value in the dialog box which is displayed.

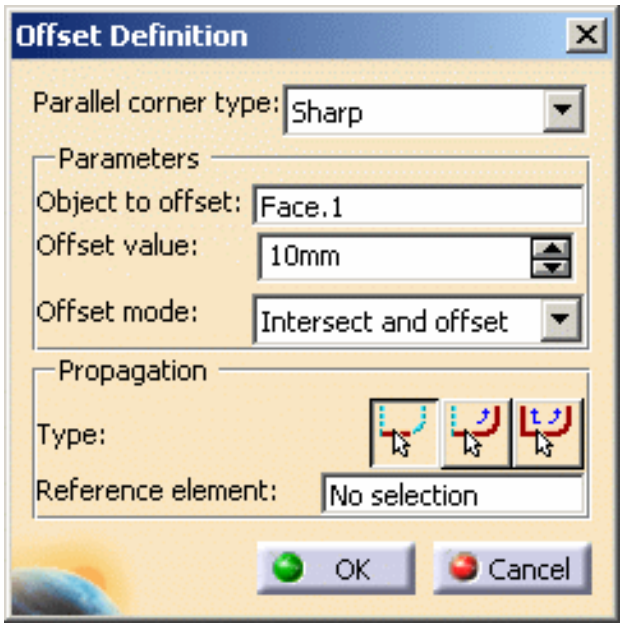


- When offsetting a face, if there is an intersection between the face and the sketch plane, by default, it is this intersection which is offset (rather than the projection of the face edges). In this case, if you want to offset the projection of the face edges, you can modify the offset as explained in the section below.
- You can offset the intersection between a face and a sketch plane without explicitly creating this intersection.
- If you offset a multi-domain face, the face that is closer from the cursor is offset.

- If you isolate a composite mark, as many simple geometry elements as the mark was containing are created, and associativity will not be available anymore.

Modifying a 3D Geometry Offset

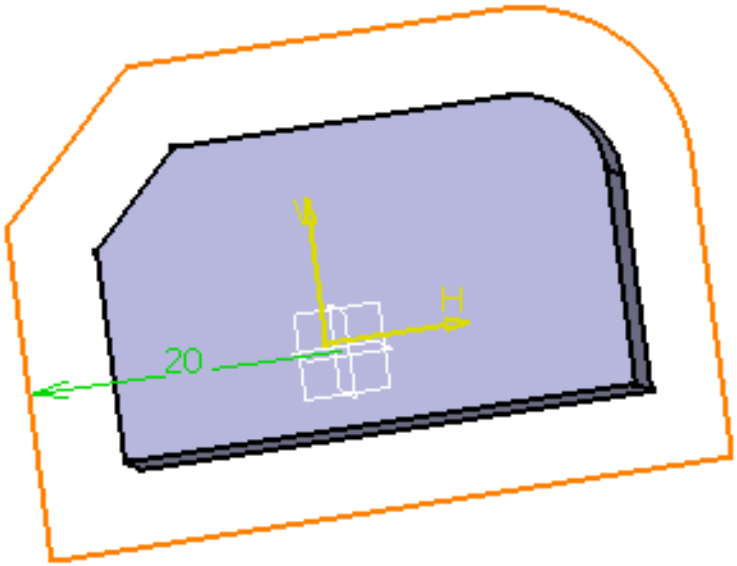
1. Double-click the offset in the specification tree or on the sketch. The **Offset Definition** dialog box is displayed.



In this dialog box, you can modify the offset definition.

- **Parallel corner type:** specifies whether corners should be round or sharp (when applicable).

Note that this option applies only when the offset results in extrapolated curves (as is the case in our example, for instance).



Parameters

These options let you specify the offset parameters.

- **Object to offset**: indicates which 3D element is offset. To offset another element, select this field and then select the new element in the sketch.
- **Offset value**: indicates the offset value. You can modify it by typing a new value in this field.
- **Offset mode**: when offsetting a face, specify whether you want to intersect and offset or to project and offset the face by selecting the appropriate option from the list.

Propagation

These options let you offset a 3D element using the propagation of an edge.

- **Type**: specifies what type of offset propagation should be applied to the selected reference element: **No propagation**, **Tangent propagation**, or **Point propagation**. Click the appropriate icon.
- **Reference element**: indicates which edge should be used as a reference for the propagation. Select this field and then select the reference edge in the sketch.

2. In the **Offset value** field, type 20mm.
3. Choose **Project and offset** from the **Offset mode** field.
4. Click **OK** to validate. The offset is modified.

- Only 3D elements can be offset with associativity.
- There is no propagation on 3D edges.
- Typing a negative offset value reverses the offset direction.
- Multi-domain elements cannot be offset in one shot.
- If you apply the **Parents/Children...** command to a sketch containing an offset obtained after selecting a face or an edge, the **Parents** command shows the last solid feature that modified the offset geometry. To see an example of this, refer to Parents/Children paragraph of Projecting 3D Elements onto the Sketch Plane.

2D Components

Before you begin with 2D components: You should be familiar with important concepts.

Create a 2D component: Create a detail sheet, and create 2D components on this sheet.

Instantiate a 2D component: Create an instance of a 2D component previously created on a detail sheet.

Edit a 2D component: Add a leader to a 2D component instance, modify text in 2D component instances and replace the reference of a 2D component instance.

Explode a 2D component: Individually explode a 2D component instance so that you can then modify it as desired.

Instantiate a 2D component from a catalog: Instantiate a 2D component previously referenced in a catalog.

Expose a 2D component from a catalog: Expose a 2D component to cut any existing link between this 2D component and its reference in a catalog.

Before You Begin With 2D Components

Before you begin creating 2D components, you should be familiar with some important concepts:

- What is a 2D Component?
  - What are 2D component references?
  - What are 2D component instances?
- What is a Component Catalog?
- Moving or Scaling 2D Components

What is a 2D Component?

2D components are either references or instances.

What are 2D component references?

A 2D component reference (also called "reference") is a re-usable set of geometry and annotations. It is stored either:

- in a layout detail sheet (in the same document as its instances)
- in a component catalog (in a drawing document).

A 2D component reference can be edited like any other item in a view. It serves as a reference when creating 2D component instances.

What are 2D component instances?

Instantiating a 2D component means re-using a 2D component reference, by creating 2D component instances (from a layout detail sheet or from a component catalog) in a layout view, once or several times. Each instance has a specific orientation, position and scale. When you instantiate a 2D component, you create a 2D component instance, also known as an "instance".

Depending on where the 2D component reference is stored, there are different ways of instantiating a 2D component:

- using the **Instantiate 2D Component** command. 2D components can be instantiated in a layout sheet, provided that they are stored in a detail sheet belonging to the same layout. Refer to Instantiating a 2D Component for more information.
- from the Catalog browser: 2D components can be instantiated from a component catalog in a layout sheet. The component catalog is based on a drawing (.CATDrawing) document.
- using Cut/Copy/Paste: 2D components instances can be copied/cut from a layout and pasted to any sheet of the same layout. Refer to Copying, Cutting and Pasting for more information.

What is a Component Catalog?

The component catalog is a separate file which references detail views from drawing documents, letting you group 2D components, classify them, and add information and attributes to them. This allows overall management of the components.

The Catalog browser lets you choose a 2D component and instantiate it in a layout sheet. Associativity is kept between a 2D component in a catalog and existing instances in a layout (through the drawing which serves as a basis for the component catalog). Therefore, in the event a 2D component is modified in a component catalog, you can synchronize (update) any existing instances. To do this, go to **Edit -> Links** and click the **Synchronize** button from the displayed dialog box.

For more information on component catalogs, refer to 2D Components > Before you Begin in the *Interactive Drafting User's Guide*, and to the *Component Catalog Editor User's Guide*.

Moving or Scaling 2D Components

By default, a 2D component (as a whole) can be moved or scaled using the mouse. If you want to prevent this, go to **Tools -> Options -> Mechanical Design -> Drafting -> Annotation and Dress-up** tab and select **Prevent direct manipulation** and/or **Prevent direct scaling**.

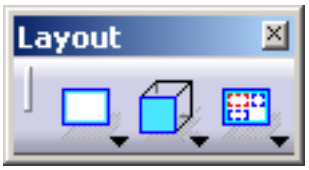
Creating a 2D Component Reference

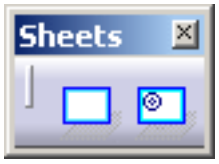
In this task, you will learn how to create a detail sheet and then position a 2D component reference on this detail sheet.

Differentiating the design sheet from the detail sheet lets you assign a structure to the document, in that the layout elements are separated from the re-usable components.

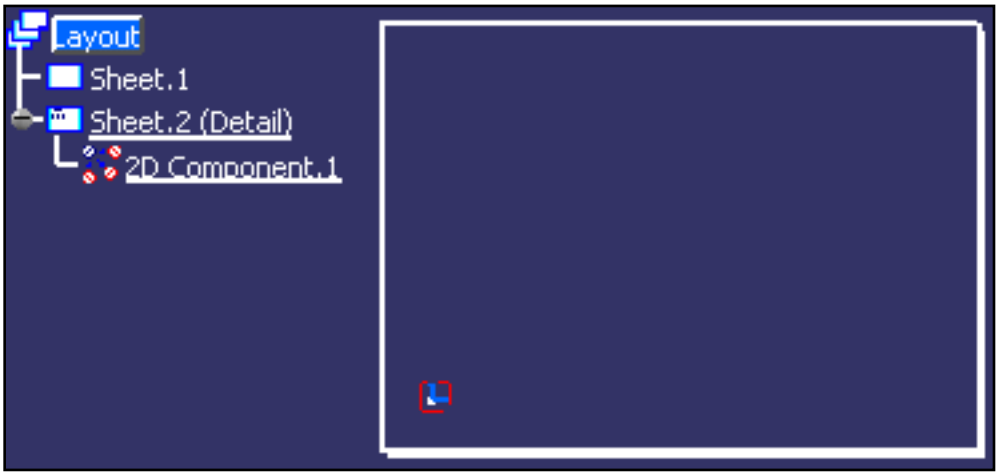
Create an empty layout as explained in Creating a Layout.

1. Click the **New Detail Sheet** icon from the **Layout** toolbar (**Sheets** sub-toolbar).

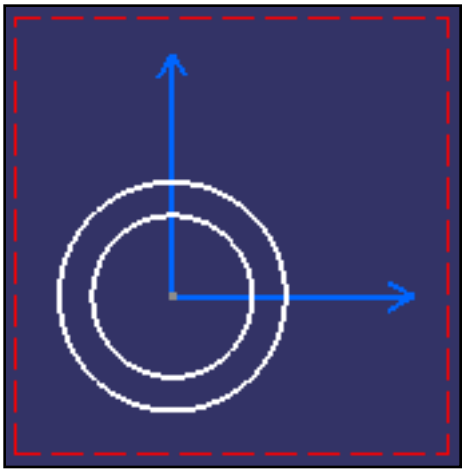




The detail sheet is created, with a 2D component view at its lower left-hand corner. It is listed in the specification tree.



2. Create geometry inside this view: two circles, for example.



You can now instantiate this 2D component on a layout design sheet.

3. Click the **2D Component** icon from the **Layout** toolbar.
4. Click on the detail sheet to position the new 2D component view.
5. Create geometry inside this view.
6. Repeat steps 3 to 5 to create as many 2D components as wanted. You can then instantiate them on a layout design sheet.

Instantiating a 2D Component

This task shows you how to instantiate a 2D component previously created on a detail sheet.

Instantiating a 2D component means re-using it (from a layout detail sheet or from a component catalog) in a layout, once or several times. When you instantiate a 2D component, you create what is known as a "2D component instance", or an "instance".

Open the 2DComponent.CATPart document. Select **Start -> Mechanical Design -> 2D Layout for 3D Design** to open the layout in the 2D window, alongside the 3D window. Make sure Sheet.1 is active.

1. From the geometry area or from the specification tree, double-click the view in which you want to instantiate the 2D component (the right view, for example). This view is now active.

2. Click the **Instantiate 2D Component** icon .

   Double-clicking the **Instantiate 2D Component** icon lets you position several 2D components on the sheet while keeping the scale and angle properties for all components.

3. From the specification tree, click the 2D component under the detail sheet **Sheet.2 (Detail)** item.

A preview of the 2D component is displayed in the view, and the **Position** icon box is displayed.

It contains the following options:

- **Change Component Origin**: Click this icon to change the component's origin, and then click the point that you want to use as the origin.

- **Change Component Angle**: Click this icon to change the component's angle, and then click in the view once the component angle axis corresponds to the position

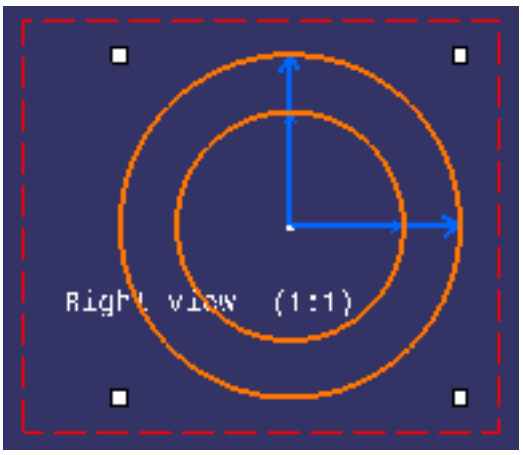
you want to assign to the component.

- **Flip Component Horizontally**: Click this icon to flip the 2D component instance on the horizontal axis of the 2D component reference.

- **Flip Component Vertically**: Click this icon to flip the 2D component instance on the vertical axis of the 2D component reference.

For the purpose of this scenario, you do not need to use these options.

4. Click at the location where you want to position the 2D component instance. The 2D component instance is created.



5. If needed, use the displayed manipulators to scale the instance, or move the instance by dragging it.

- You can easily edit the 2D component reference by right-clicking the 2D component instance, and choosing **Edit Reference Component**.

- Remember that if you selected **Prevent direct manipulation** and/or **Prevent direct scaling** from **Tools -> Options -> Mechanical Design -> Drafting - > Annotation and Dress-up** tab, you will not be able to move and/or scale the component.

Editing a 2D Component Instance

This task shows you how to edit a 2D component instance. This includes:

- Adding a leader to a 2D component instance
- Modifying text in 2D component instances
- Replacing the reference of a 2D component instance

Open the Disk4.CATPart document. Select **Start -> Mechanical Design -> 2D Layout for 3D Design** to open the layout in the 2D window. Instantiate a 2D component (in the Front view, for example) as described in Instantiating a 2D Component.

Adding a leader to a 2D component instance

1. Right-click the 2D component instance and select **Add Leader**.
2. Select the element to which you want the 2D component instance to be associated, or click in empty space if you do not want the 2D component to be associated with an element.

Modifying text in 2D component instances

In your layout, double-click the 2D component reference from the specification tree (**2D component.1**). This activates the detail sheet and the view which contains the 2D component reference. Add a text to the 2D component reference.

1. Right-click the 2D component reference text.
2. From the contextual menu, select **Modifiable in instance**. This makes the text modifiable in existing instances as well as in newly-created instances.
3. Create a 2D component instance (in the Front view, for example). There should now be two instances in your Front view.
4. If you now double-click the text in the first 2D component instance, you are able to modify it. You can then repeat this operation for the text in the 2D component instance you have just created.

More about texts in 2D components

- Attribute links may be added in the text content.
- Once a text becomes modifiable, it is not possible to make it non-modifiable.
- For 2D component instances created with a catalog, if a text becomes modifiable in the catalog, you have to synchronize the external reference to make the 2D component instance text modifiable too (refer to Creating and Editing a Catalog in the *Component Catalog Editor User's Guide*).
- If you create a 2D component reference (called MYREF for example) containing a 2D component instance with a modifiable text, the text will not be modifiable in instances of MYREF.

- If you want to use as symbols 2D components with text, activate both the **Apply Scale** property for the text (in **Edit -> Properties -> Text** tab) and the **Create with a constant size** setting (in **Tools -> Options -> Mechanical Design -> Drafting -> Annotation and Dress-up** tab): the size of both the 2D component and its text will then be independent from the view scale.

- Remember that if the **Allow direct manipulation** check box is cleared in **Tools -> Options -> Mechanical Design -> Drafting - > Annotation and Dress-up** tab, you cannot manipulate the component. In particular, you cannot modify the text strings in 2D component instances.

Replacing the reference of a 2D component instance

Create a 2D component reference (this time defining two rectangles, for example) so that your detail sheet now contains two 2D component references.

1. Activate Sheet.1.
2. Right-click the existing 2D component instance in the Front view, and select **2D component.X (instance) object > Replace Reference**.
3. From the specification tree, select the 2D component reference you just created (2D component.2).

   You could also select another instance in order to take its reference into account.

   The reference of the 2D component instance is replaced, and the instance is modified accordingly.

More about replacing references

- You cannot use a catalog to replace a 2D component instance reference. To bypass this, use an instance created with this catalog.
- When replacing the reference of a 2D component instance, any existing text in the original 2D component instance is also replaced, even if this existing text had been previously modified (see Modifying text in 2D Component Instances for more information on this point).

Exploding a 2D Component

This task shows you how to individually explode a 2D component that was instantiated from a detail sheet.

Exploding a 2D component instance means removing the link between it and its reference (the associativity with the detail sheet is cut): you can then modify the exploded 2D component instance as desired, as it behaves as independent geometry.

Instantiate a 2D component as described in Instantiating a 2D Component.

Right-click the 2D component instance, and select **2D Component.X (Instance) object > Explode 2D Component** from the contextual menu.

The component is now exploded. You can therefore modify the geometry and/or graphical properties of its elements.

After an Explode operation, all dress-up elements added to the instance are deleted, texts lose their associativity with the detail sheet and dimensions change color (according to the color defined for **Not-up-to-date dimensions** in the **Types and colors of dimensions** dialog box available through **Tools -> Options -> Mechanical Design -> Drafting -> Dimension** tab, **Analysis Display Mode** area, **Types and colors...** button. By default, this color is fuchsia. For more information, refer to Dimension > Analysis Display Mode in the Customizing Settings chapter).

Instantiating a 2D Component from a Catalog

This task shows you how to instantiate a 2D component you previously referenced in a catalog.

Performing this task assumes that you have already created a component catalog using the Interactive Drafting workbench. For more information, refer to Creating a Component Catalog in the *Interactive Drafting User's Guide*. To enable you to perform this scenario, a component catalog sample is provided.

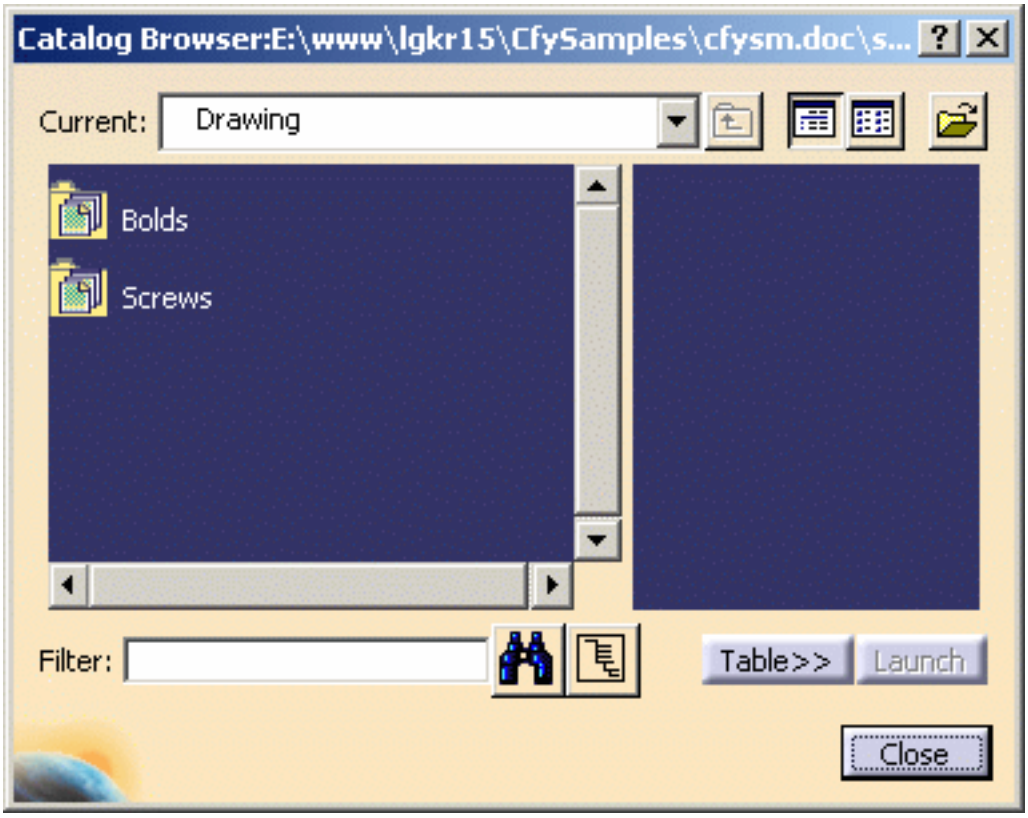
Create a layout and a projection view. You will be using the Detail1.catalog document.

1. Click the **Catalog Browser** icon from the **Catalog** toolbar. The **Catalog Browser** dialog box appears.

   It contains the following information:

   - the name of the currently open catalog
   - the catalog chapter tree
   - a preview of the selected component
   - the possibility to perform a query on available components (refer to the *Knowledge Advisor User's Guide* for more details on formulas).

2. Click the **Browse another catalog** option from the **Catalog Browser** dialog box and open the Detail1.catalog document (from the \online\cfyugsm_C2\samples\Drafting directory).



3. Double-click Bolds chapter from the chapter tree.

   The list with the components included in the Bolds chapter appears in the dialog box.

4. Click Special K10 from the list available.

   This lets you preview the component you just selected.

5. Double-click Special K10 from the list.

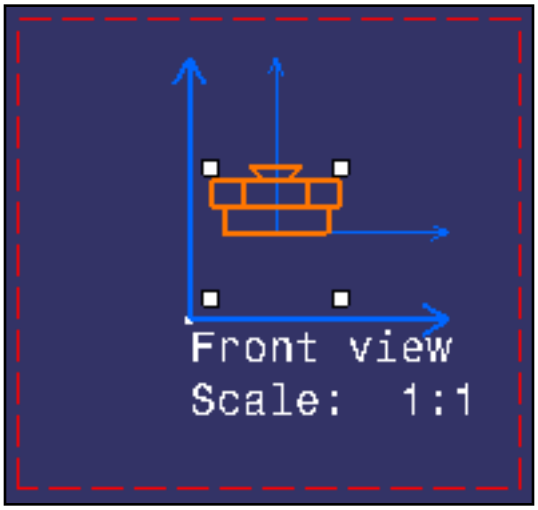
   You can also right-click the chosen component from the dialog box, and select the **Instantiate Component...** option.

   A preview of the 2D component is displayed in the view, and the **Position** icon box is displayed.

   For more information on this icon box, refer to Instantiating a 2D Component.

6. Click at the location where you want to position the 2D component instance in the layout view. The 2D component instance is created.

7. If needed, use the displayed manipulators to scale the instance, or move the instance by dragging it.



8. Repeat steps 4 to 7 if you want to instantiate additional 2D components from the catalog.

9. Click Close in the **Catalog Browser** dialog box when you are done.

Exposing a 2D Component from a Catalog

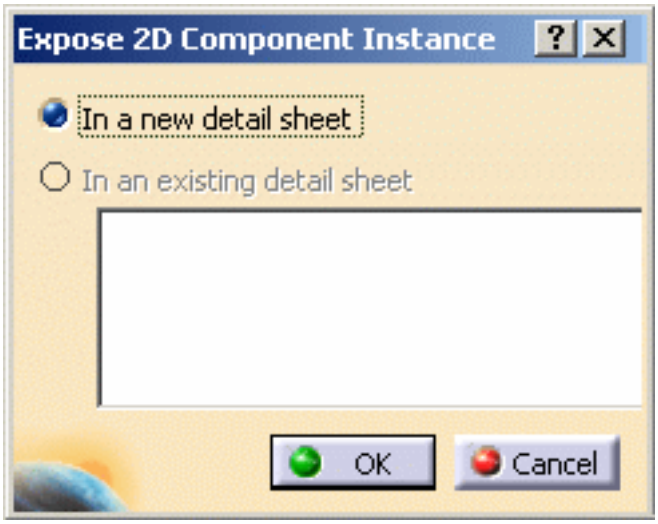
In this task, you will learn how to expose a 2D component from a catalog.

Exposing a 2D component lets you cut any existing link between a 2D component instance and its reference in a catalog, by creating (from the 2D component catalog reference) a 2D component reference on a detail sheet.

In a given document, all 2D component instances that were linked to the former 2D component catalog reference will then link to the exposed 2D component reference. This will also allow you to modify the 2D component reference locally (without modifying the 2D component catalog reference).

Instantiate a 2D component from a catalog as explained in Instantiating a 2D Component from a Catalog.

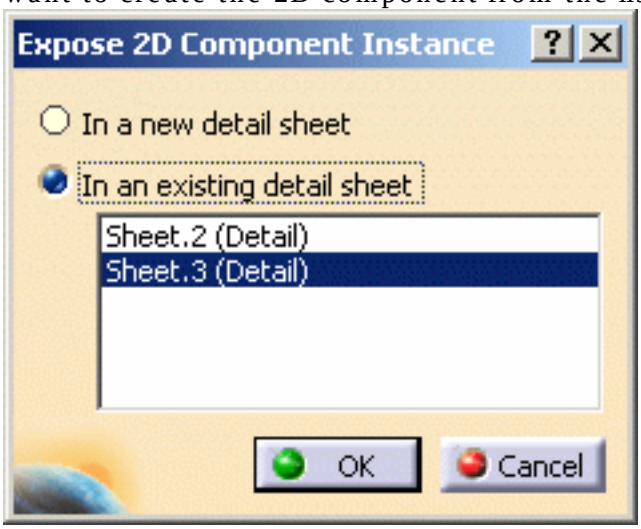
1. Right-click the 2D component instance in the layout view, and select **Special K10.X object -> Expose 2D Component**. The **Expose 2D Component Instance** dialog box is displayed.



As no detail sheet exists in this drawing, a detail sheet will be created for the 2D component.

Therefore, the **In a new detail sheet** option is selected. The other option is unavailable.

In the case of a drawing with an existing detail sheet, you can specify whether you want the 2D component to be created in a new detail sheet or in an existing detail sheet by selecting the appropriate option. In the latter case, you can then select the detail sheet in which you want to create the 2D component from the list underneath.



3. Click **OK**. All links are now cut between the 2D component instance and its catalog reference.

4. In the detail sheet, you can now modify the 2D component reference. For example, add a text indicating the component's name.

In this scenario, text is modified as an example of 2D component reference modification. However, the **Modifiable in instance** contextual command is specifically intended for modifying texts in 2D component instances. See Re-Using a 2D Component for more information on modifying text in 2D component instances.

The 2D component instance has been modified in the sheet. On the other hand, the 2D component reference in the catalog is left unchanged.

Dimensioning

The 2D Layout for 3D Design workbench enables you to create:

- 2D dimensions (measured or driving)
- associative 3D dimensions (measured only)
- associative hybrid dimensions between 2D and 3D elements (measured only)

As dimensioning commands work as in the Interactive Drafting workbench, most tasks included in this section provide links to the *Interactive Drafting User's Guide*. As such, the information detailed in these tasks is presented in an Interactive Drafting context. You should note that the *Interactive Drafting User's Guide* contains images that correspond to the Interactive Drafting workbench and therefore illustrate dimensions in an environment that is different from the 2D Layout for 3D Design environment (symbols and background color, for example).

Refer to Dimensioning in a 2D Layout for 3D Design Context which explains what is specific to creating dimensions in the 2D Layout for 3D Design workbench as opposed to doing so in the Interactive Drafting workbench.

Dimensioning in a 2D Layout for 3D Design context: explains what is specific to creating dimensions in the 2D Layout for 3D Design workbench as opposed to doing so in the Interactive Drafting workbench.

Dimensions and tolerances (tasks documented in the *Interactive Drafting User's Guide)*: Create and modify all types of dimensions and tolerances.

Dimension systems (tasks documented in the *Interactive Drafting User's Guide)*: Create and modify dimension systems.

Dimensioning in a 2D Layout for 3D Design Context

The 2D Layout for 3D Design workbench enables you to create:

- 2D dimensions (measured or driving)
- associative 3D dimensions (measured only)
- associative hybrid dimensions between 2D and 3D elements (measured only)

As dimensioning commands work as in the Interactive Drafting workbench, most tasks included in the Dimensions and Tolerances and Dimension Systems sections provide links to the *Interactive Drafting User's Guide*. However, there are a few particularities about dimensioning in 2D Layout for 3D Design, as opposed to doing so in Interactive Drafting, which you will learn in this section.

In this section, you will learn about:

- Selecting elements to dimension
- Dimensioning: use cases
  - Dimension on view content
  - Dimension on 2D background
  - Dimension on 3D background
  - Dimension on view content and 2D background
  - Dimension on view content and 3D background
  - Dimension on 2D background and 3D background
- Dimensioning: particular cases
  - True length dimensions
  - Dimension systems
  - Chamfer dimensions
  - Dimensions along a direction
- Before you begin creating dimensions in 2D Layout for 3D Design

Selecting elements to dimension

Dimension commands provide a visual feedback indicating whether it is possible to create dimensions on a given element.

However, you should be aware of the following rule: in a given part layout, it is impossible to create a dimension for another part. You can only create dimensions within a single part layout. For example, in Part.1, it is not possible to store a dimension measuring an element of Part.2.

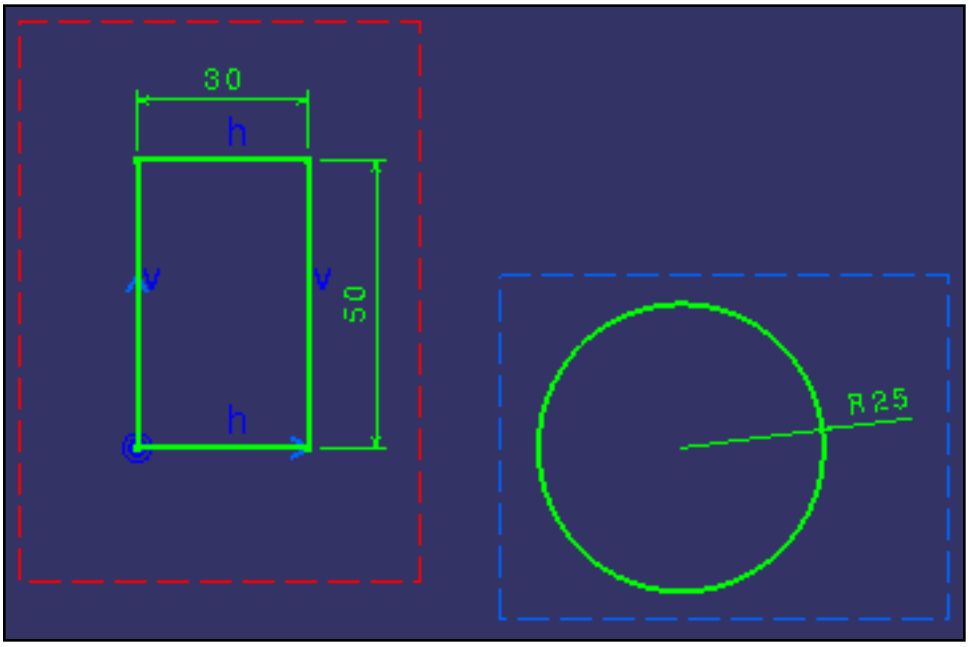
When selecting elements to dimension, remember the following points:

- Dimensions can be created in any view, even a non-active one.
- After starting a dimension command, the view in which you select the first element is the view of creation (that is the view where the dimension will be created).
- You can always select an element belonging to the view content.
- Once you have selected the first element, you can only select the other elements in the view of creation.
- You cannot select as the first element a 2D background element.
- You can select an element which belongs to the 3D background of a part layout only if this element belongs to the current layout.

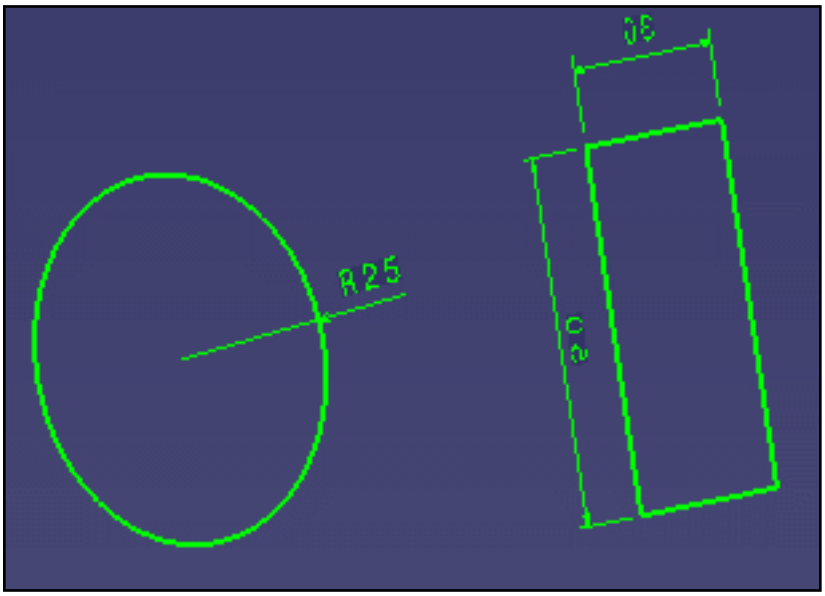
Dimensioning: use cases

Dimension on view content

Such dimensions can have 1, 2 or 3 reference elements which are either 2D geometry or dress-up. They behave exactly as they do in the Interactive Drafting workbench.



Dimensions on view content in the 2D Layout for 3D Design window



Dimensions on view content in the 3D window

Available dimensions

Distance, offset, length, curvilinear length, circular length, angle, radius, diameter, chamfer, thread, coordinate, chained, cumulated, stacked.

Non-supported dimensions

True length dimensions.

Dimension type

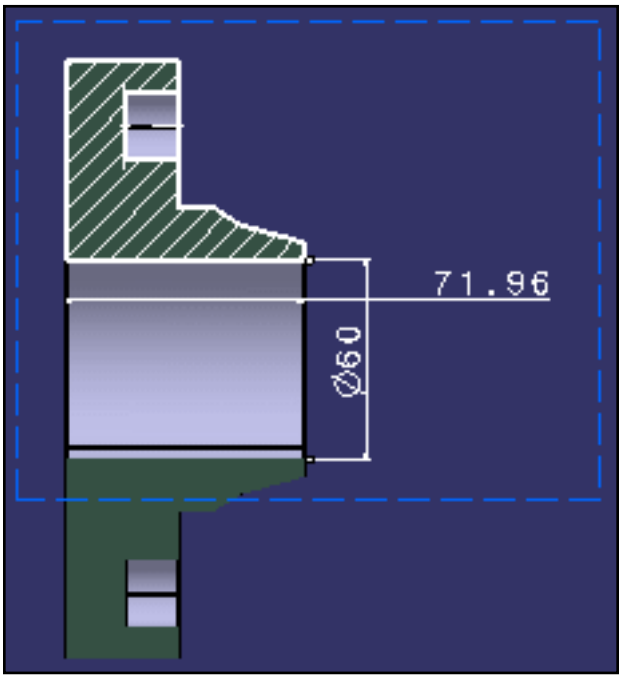
Measured, driving (view content element).

Dimension on 2D background

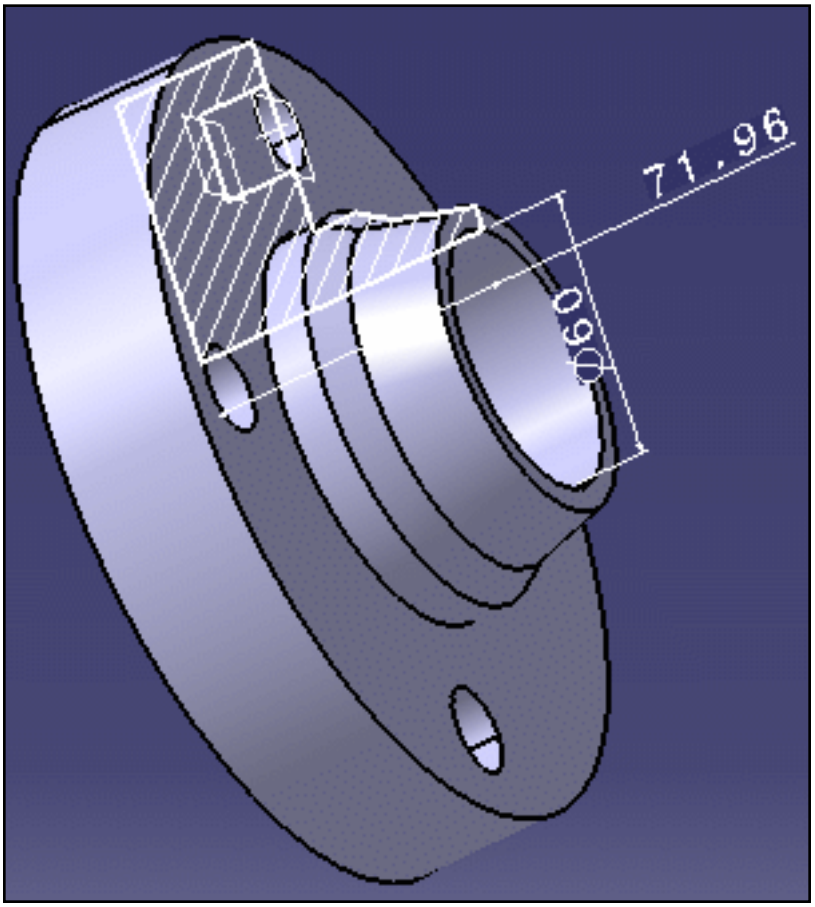
Dimensions on 2D background (that is with reference elements belonging exclusively to the 2D background) are not supported.

Dimension on 3D background

Such dimensions have reference elements which belong exclusively to the 3D background. Such dimensions behave exactly as they do in the Functional Tolerancing and Annotation workbench. For more information, refer to the *Functional Tolerancing and Annotation User's Guide*.



Dimensions on 3D background in the 2D Layout for 3D Design window



Dimensions on 3D background in the 3D window

Available dimensions

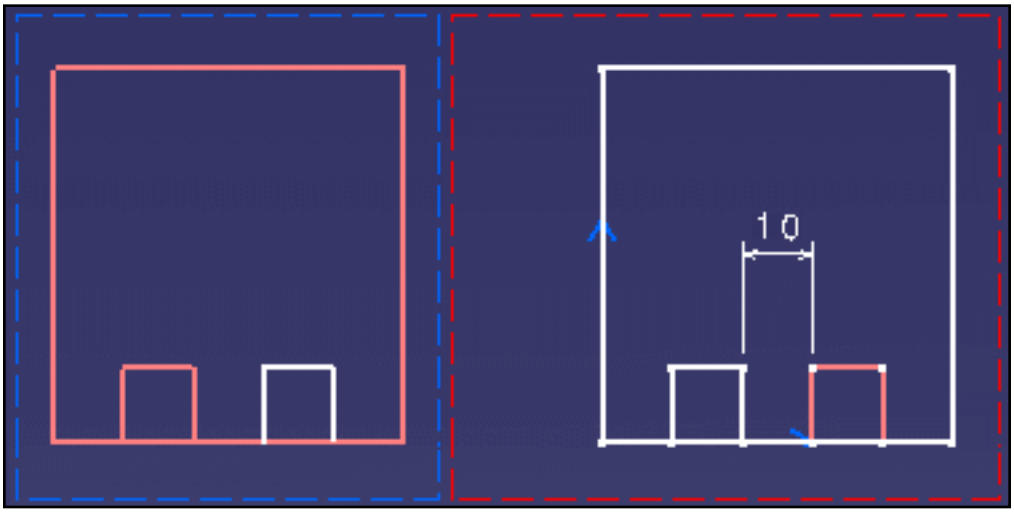
Distance, length, curvilinear length, circular length, angle, radius, diameter, chamfer, chained, cumulated, stacked.

Dimension type

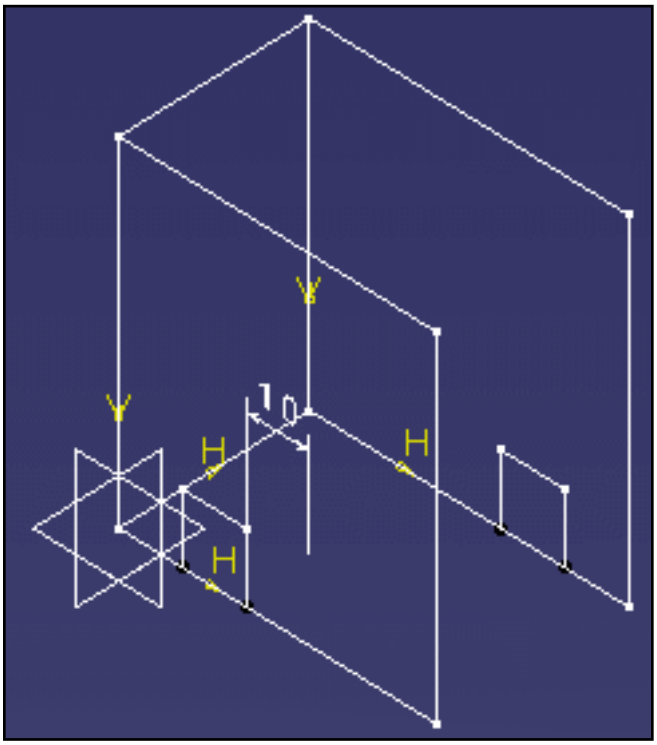
Measured.

Dimension on view content and 2D background

Such dimensions have two reference elements which are either 2D geometry or dress-up. Both reference elements (the one in the view content and the one in the 2D background) belong to the same part layout.



Dimensions on view content and 2D background in the 2D Layout for 3D Design window



Dimensions on view content and 2D background in the 3D window

Available dimensions

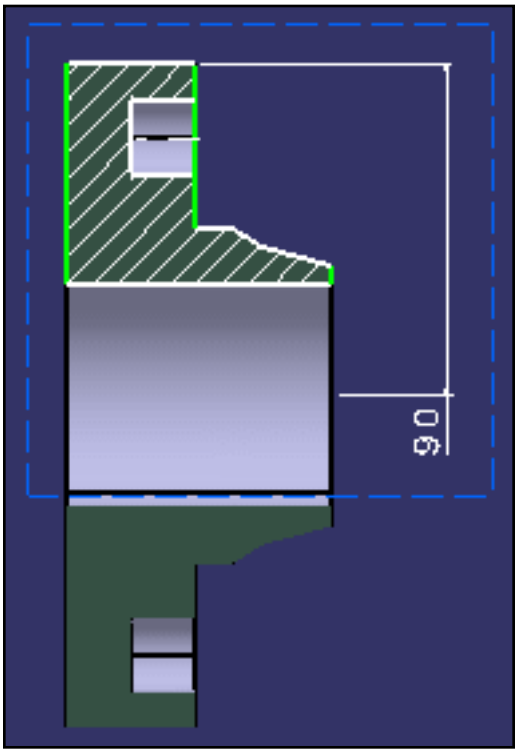
Distance, angle.

Dimension type

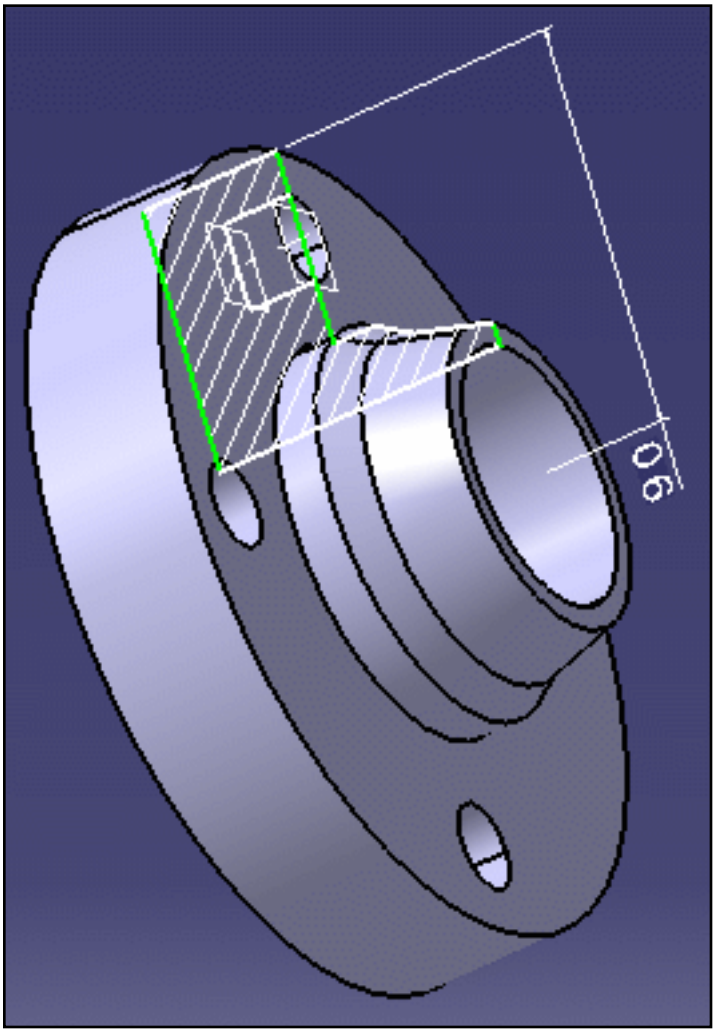
Measured.

Dimension on view content and 3D background

Such dimensions have two reference elements, one of which is either 2D geometry or dress-up, while the other is a 3D element (edge, face, 3D wireframe). Both reference elements (the one in the view content and the one in the 3D background) belong to the same part layout.



Dimensions on view content and 3D background in the 2D Layout for 3D Design window



Dimensions on view content and 3D background in the 3D window

Available dimensions

Distance, angle.

Dimension type

Measured.

Dimension on 2D background and 3D background

Dimensions on 2D background and 3D background (that is with a reference element belonging to the 2D background and the other one to the 3D background) are not supported.

Dimensioning: particular cases

True length dimensions

True length dimensions are not supported in 2D Layout for 3D Design.

Dimension systems

Dimensions individually behave as classical dimensions. For example, you can create a cumulated dimension system by selecting first an element in the view content (that is the reference), and then elements either in the view content, 2D background or 3D background. However, if the reference is a 2D background element, then the other elements must belong to the view content.

Chamfer dimensions

Every selected element must be of the same kind. That is to say you cannot dimension a chamfer which would contain an element in the view content and one in the 3D background.

Dimensions along a direction

The directional element must belong to the same part layout.

Before you begin creating dimensions in 2D Layout for 3D Design

Before you begin creating dimensions in 2D Layout for 3D Design, make sure you are familiar with:

- The Tools toolbar and the Tools Palette.
- SmartPick, an easy-to-use tool designed to assist you when creating annotations. For more information, refer to the SmartPick task in the *Sketcher User's Guide*.
- Multi-selection. For more information, refer to the Selecting Objects chapter in the *Infrastructure User's Guide*.

Dimensions and Tolerances

As dimension commands work as in the Interactive Drafting workbench, this section of the documentation provides links to the *Interactive Drafting User's Guide*. As such, the information detailed in this section is presented in an Interactive Drafting context. You should note that the *Interactive Drafting User's Guide* contains images that correspond to the Interactive Drafting workbench and therefore illustrate dimensions in an environment that is different from the 2D Layout for 3D Design environment (symbols and background color, for example).

Refer to Dimensioning in a 2D Layout for 3D Design Context which explains what is specific to creating dimensions in the 2D Layout for 3D Design workbench as opposed to doing so in the Interactive Drafting workbench.

Before you begin: You should be familiar with basic concepts.

Create dimensions: Create dimensions by clicking elements.

Create half-dimensions: Create half-dimensions on distance, angle, diameter, cylinders, diameter edges and diameter tangents but not on cumulated dimensions.

Create explicit dimensions: Create dimensions by selecting the desired icon and the required geometrical elements.

Create/modify angle dimensions: Create an angle dimension and perform the following kinds of modification: new angle sector or turn an angle sector into a supplementary sector.

Creating fillet radius dimensions: Create a fillet radius dimension in a projected view.

Create chamfer dimensions: Create a chamfer dimension using selection.

Create thread dimensions: Create associative thread dimensions.

Create/modify coordinate dimensions: Automatically create coordinate dimensions on elements.

Create/modify curvature radius dimensions: Create and modify a curvature radius dimension. This lets you know the curvature radius at a given point on a curve (spline, ellipse and so on).

Create overall curve dimensions: You can create dimensions on the overall size of any kind of curve, whether it is canonical or not (line, circle, ellipse, spline and so on). You can also create dimensions on the overall size between two curves, or between a curve and a line, for example.

Create curvilinear length dimensions: You can create dimensions for the curvilinear length of a curve, that is measure the overall length of a curve.

Create partial curvilinear length dimensions: You can create dimensions for the curvilinear length of a curve portion, that is measure the partial length of a curve.

Create dimensions along a reference direction: You can create dimensions along a direction of measure. In other words, you can measure the projection of a segment/distance onto a direction.

Create dimensions between intersection points: You can create dimensions between an intersection point and an element or between two intersection points.

Create dimensions between an element and a view axis: Create dimensions between an element and a view axis (one of the two axes or the origin).

Create driving dimensions: Create dimensions that will drive associated constrained geometry.

Modify the dimension type: Modify the dimension type as you create a dimension. In other words, you modify the dimension attributes.

Re-route dimensions: Recalculate dimensions taking into account new geometrical elements.

Interrupt one or more extension lines: Interrupt manually one or more extension lines of one or more dimensions, either using the contextual menu or the **Insert** menu bar command.

Modify the dimension line location: Use the mouse to modify dimension line location either before or after creating dimensions.

Modify the dimension value text position: Use the cursor to modify dimension value text position.

Specify the dimension value position: Automatically or explicitly position the dimension value inside or outside the area between extremity symbols.

Add text before/after the dimension value: Insert text before or after the dimension value.

Modify the dimension overrun/blanking: Modify the dimension extension line overrun and/or blanking either together or separately.

Scaling a dimension: Apply (or not) a scale to the dimension text when a scale is applied to the object containing the dimension.

Line up dimensions (free space): Line up dimensions relative to a point in the free space.

Line up dimensions (reference): Line up dimensions according to a given reference.

Create a datum feature: Use the Datum Feature Creation dialog box to create a datum feature.

Modify a datum feature: Modify a datum feature by editing it.

Create a geometrical tolerance: Use the Geometric Dimensioning And Tolerancing Parameters dialog box to create geometrical tolerances.

Modify a geometrical tolerance: Use the Geometric Dimensioning And Tolerancing Parameters dialog box to modify geometrical tolerances.

Copy a geometrical tolerance: Copy an existing geometrical tolerance and then edit the content for creating a new one.

Before You Begin

Before you begin creating dimensions, you should be familiar with the concepts described in this section.

First of all, bear in mind that dimension creation in Drafting follows the general rules which apply to geometry creation in V5: the geometry to dimension must be contained within a "box" whose dimensions are 2.e+6mm (the coordinates can vary from -1,000,000 mm to +1,000,000 mm). Therefore, it is impossible to create dimensions for elements exceeding these dimensions.

Creating Dimensions

You can create (and therefore modify) the following types of dimensions:

Dimensions created on one element:

- Length dimensions
- Diameter dimensions
- Radius dimensions
- Radius curvature dimensions

Dimensions created on two elements:

- Distance dimensions
- Angle dimensions
- Diameter/Radius Cylinder dimensions

Note that you can create half-dimensions on distance, angle, diameter cylinder, diameter edge and diameter tangent dimensions but not on cumulate dimensions.

Modifying the Dimension Attributes

You can modify the following attributes at any time before you click to validate the dimension creation:

Modify while creating:

- Type
- Measure direction. Refer to Using Tools for more information.
- Angle sector
- One symbol
- Diameter/Radius center

Modify while or just after creating:

- Value position

- Extension line overrun/blanking (either one or both)
- Text before/after
- Properties (see further down)
- Swap to diameter/radius

Manipulating Dimensions

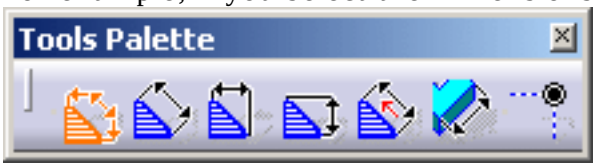
By default, when manipulating dimensions, you will use the following functionalities:

- dimension following the cursor: go to **Tools -> Options -> Mechanical Design -> Drafting -> Dimension** tab, to use automatic positioning
- global move: go to **Tools -> Options -> Mechanical Design -> Drafting -> Dimension** tab, to move precisely dimension line, dimension value, secondary part of a dimension line.
- blanking manipulators (available when modifying a dimension): go to **Tools -> Options -> Mechanical Design -> Drafting -> Manipulators** tab, not to visualize blanking manipulators or to visualize other manipulators either when creating or when modifying a dimension (**Overrun**, **Blanking**, **Insert text before**, **Insert text after**, **Move value**, **Move dimension line, Move DimLine Secondary Part**).
- value snapped between the dimension lines symbols: go to **Tools -> Options -> Mechanical Design -> Drafting -> Dimension** tab, if you do not want to have the possibility to snap the dimension value between both symbols of the dimension line and/or you want to snap the dimension position on the grid.
- during creation: to switch temporarily the **Dimension following the cursor** option, hold on the **ctrl** key.
- during creation and edition: to switch temporarily the **Activate Snapping** option, hold on the **shift** key. Clicking on the dimension symbols will invert them.
- during angle dimension creation: if the **Dimension following the cursor** option is activated, you can swap the angle sector according to the mouse position holding on the **ctrl** and **shift** keys. If the **Dimension following the cursor** option is not activated, you can swap to the complementary angle sector holding on the **ctrl** key and clicking on the dimension line.

Dimension Tools

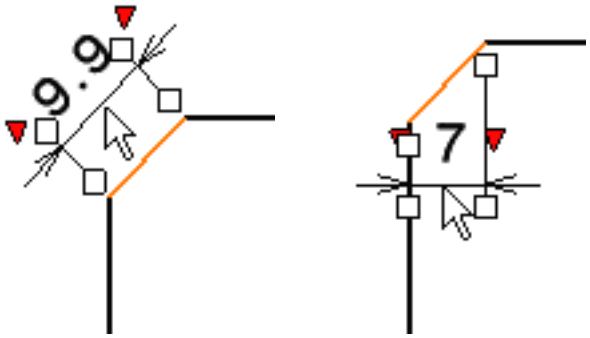
The Tools palette appears whenever you select a command for which specific options or value fields are available. This enables you to know immediately when specific tools are available for a command. The options or fields available in the Tools Palette depend on the command you selected. Only a few examples are provided here.

For example, if you select the Dimensions command, the Tools Palette may provide the following options:

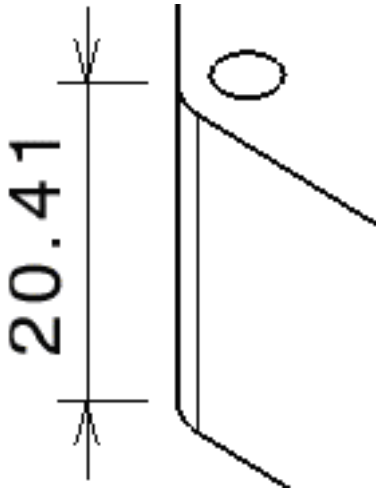


Projected/Forced/True Length Dimension

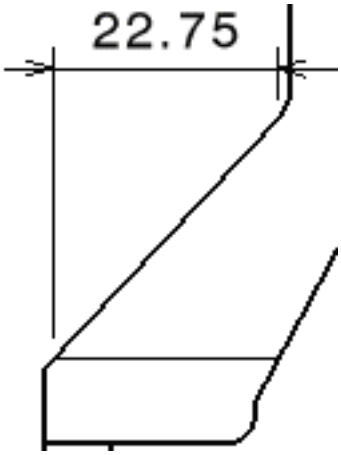
Projected Dimension (according to the cursor position)



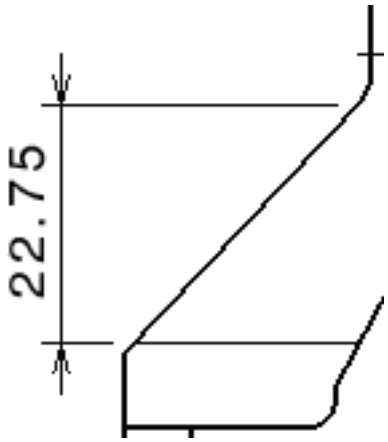
Force Dimension on Element



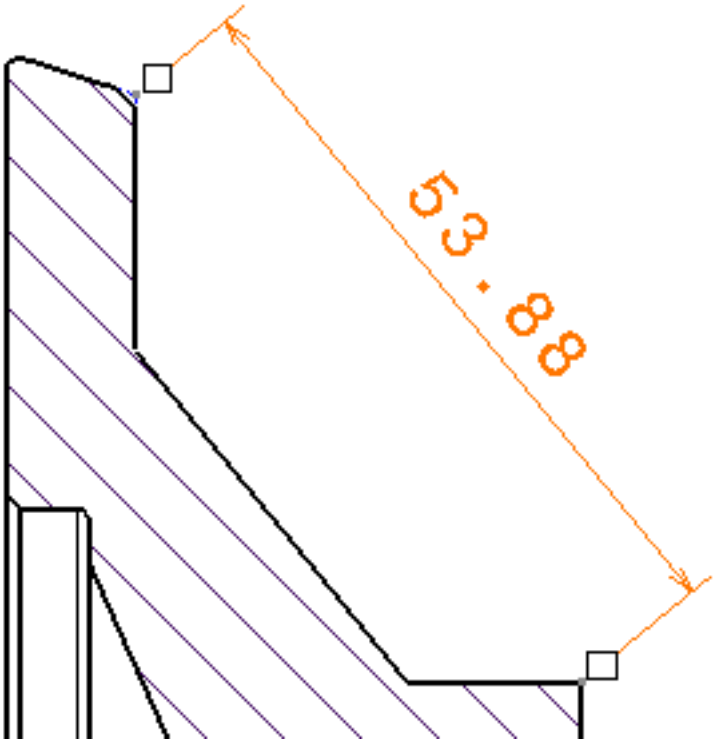
Force Horizontal Dimension in View



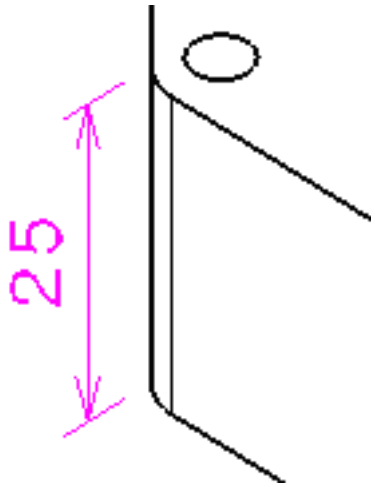
Force Vertical Dimension in View



Force Dimension along a direction



True Length Dimensions (for isometric views only)



Remember that as you create the dimension in one mode, you can use the contextual menu and select another mode.

Dimension Properties

You can apply given properties to all the dimensions you are going to create. For this, use the Dimension Properties toolbar.

- Line type (regular, two parts, one part leader, or two parts leader)
- Tolerance type
- Tolerance value
- Numerical Display Format
- Precision.

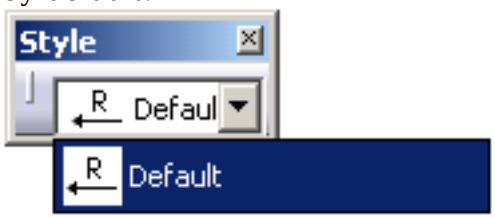
- For the ISOCOMB combined tolerance, use the following type of syntax in the tolerance value field: H6 (+0.5 / -0.3)

- When creating a new drawing, the Unit field (here: NUM.DIMM) drives the unit of the dimensions to be created.
  The value which is used by default in this field for each type of dimension is usually defined by the dimension styles (**Tools -> Standards -> Styles -> [dimension style] -> ValueDisplayFormat -> MainValue -> Name**). However, if no value is defined by the styles, the one which will be used by default is that defined as your default unit choice in **Tools -> Options -> General -> Parameters and Measure -> Units** tab.
- When editing an existing drawing, if you change your default unit choice in **Tools -> Options -> General -> Parameters and Measure -> Units** tab, then the numerical display format which best corresponds to the selected unit is automatically selected in the toolbar instead of the current default value.

Using Styles

You can use styles (i.e. a set of default values for each kind of element) when creating dimensions in drawings created with version V5 R11 and later (or pre-R11 drawings whose standard has been updated or changed in V5 R11 and later). Styles are defined in the standard used by the drawing and managed by the administrator.

When creating a dimension, the Style toolbar displays the styles available for this type of dimension. (By default, the Style toolbar is situated at the top left of screen.) If only one style is available, it will be used by default.



If several styles are available for this type of dimension, you can choose the style that you want to use to create this dimension by selecting it from the Style toolbar.

Refer to Using Styles for more information.

In drawings created with versions up to V5 R10, you can create dimensions using default values. Refer to Setting Properties As Default in Pre-R11 Drawings and to Using Properties Set as Default in Pre-R11 Drawings for more information.

Creating Dimensions

In this task, you will learn how to create dimensions. When creating dimensions on elements, you can preview the dimensions to be created.

This task deals with:

- Creating Dimensions
- More About Dimensions
  - Properties
  - Associativity
  - Driving Dimensions
  - True Dimensions
  - Half Dimensions
  - Extension Line Anchor

Creating Dimensions

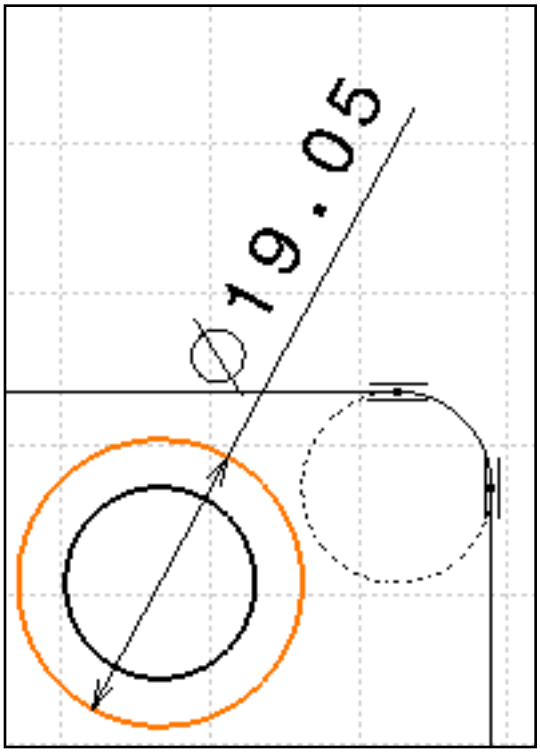
Open the Brackets_views02.CATDrawing document.

1. Click the **Dimensions** icon on the **Dimensioning** toolbar.

2. Click a first element in the view. For example, a circle.

3. If needed, click a second element in the view.

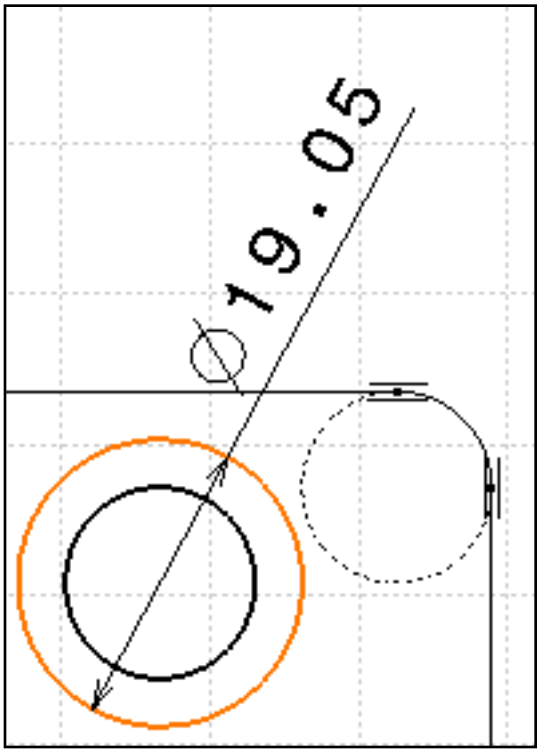
   The dimension type is automatically defined according to the selected elements (or in the **Tools Palette**).



At this step, the options in the **Tools Palette** ( ) allow you to position the dimension using one of the following modes: Projected or Forced modes. These options are also available in the contextual menu.

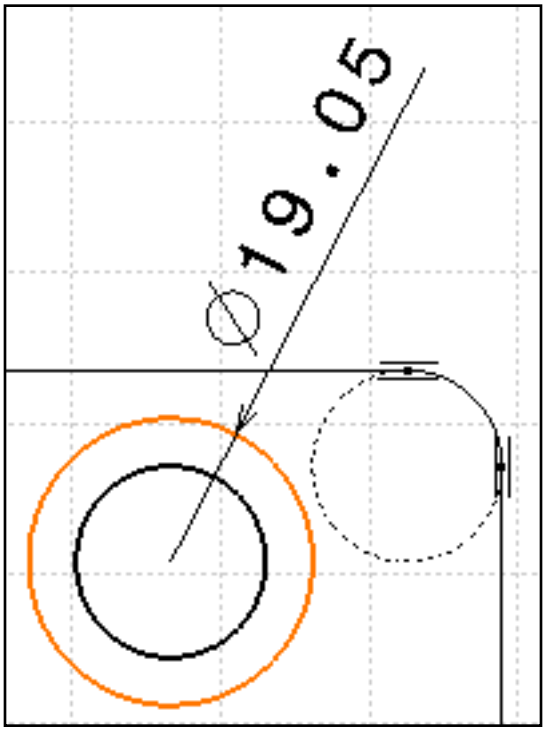
This toolbar is situated at the bottom right of screen. If you cannot see it properly, just undock it.

4. Click the **Force Dimension on element** icon from the **Tools Palette**.



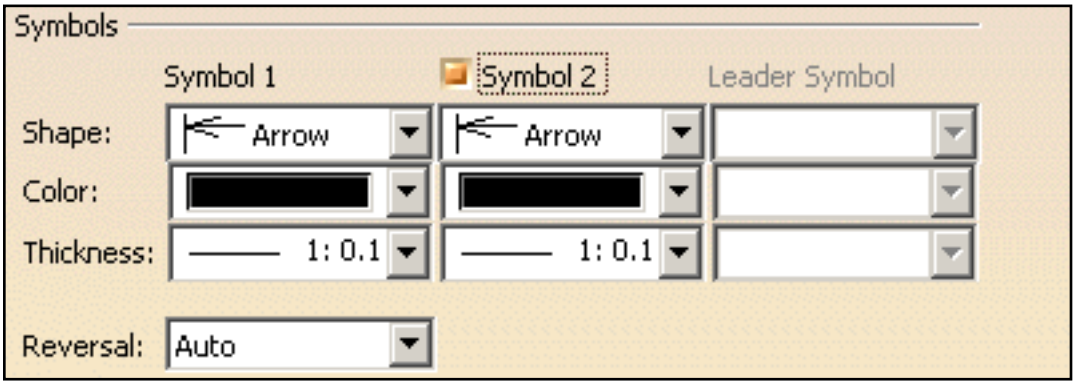
5. Right-click to access the contextual menu and select **1 symbol.**

The dimension becomes a one-symbol dimension.

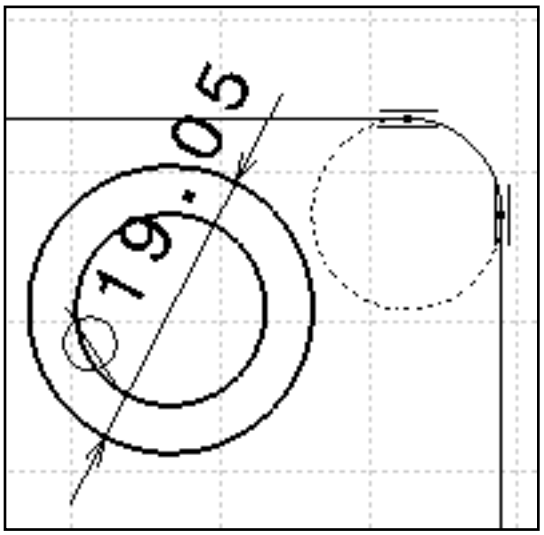


During the dimension creation step, you can switch between one-symbol or two-symbols dimension by selecting or deselecting **1 symbol** in the contextual menu.

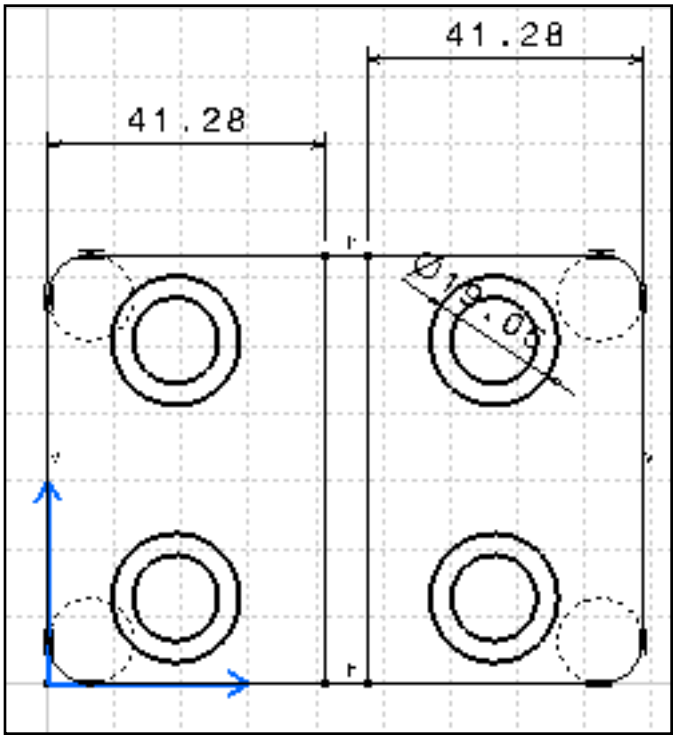
Once the dimension has been created, you must use the **Properties** menu to specify whether you want to use one or two symbols. Right-click the dimension and in the contextual menu, choose **Properties.** Click the **Dimension Line** tab and then select **Symbol 2** to display two-symbols dimension, or clear this check box to display one-symbol dimension.



6. Click in the drawing window to validate the dimension creation.

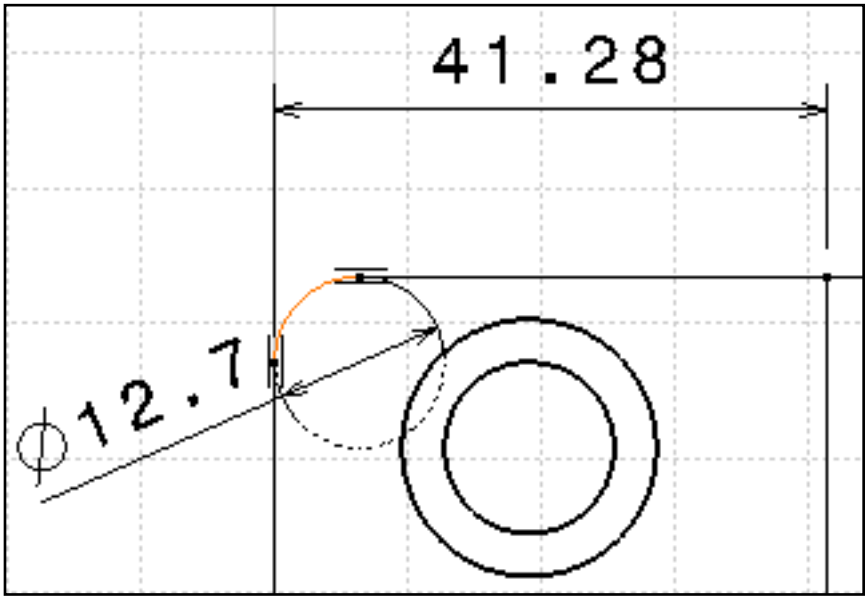


7. Create two other dimensions on a line as shown.



8. Select the two dimensions with the **Ctrl** key (you can move them both).

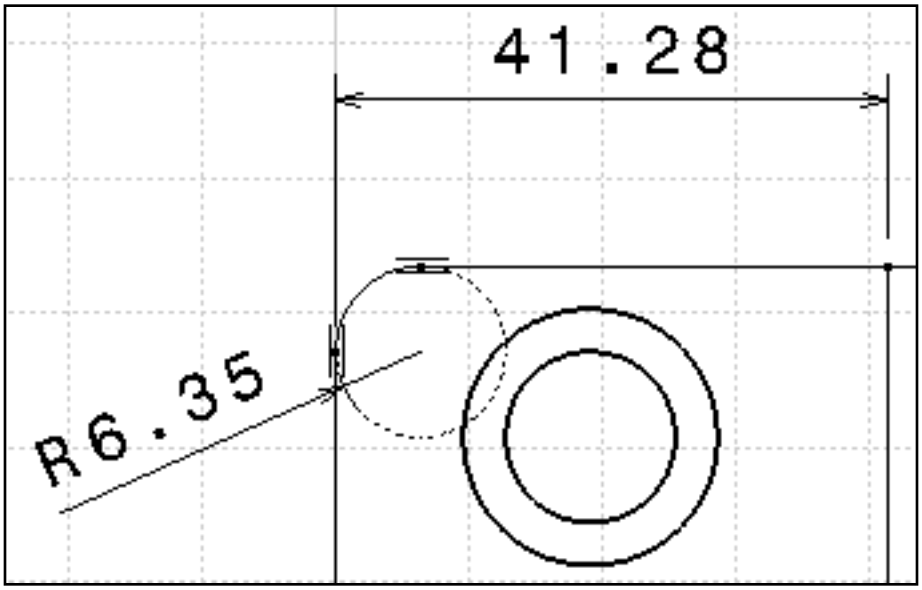
9. Start creating another dimension: click the **Dimensions** icon and select another circle.



10. Click in the drawing to validate the dimension creation.

11. Right-click the dimension you just created and in the contextual menu, choose **Dimension.3 Object** and select **Swap to Radius**.

    The diameter dimension has swapped to radius dimension.

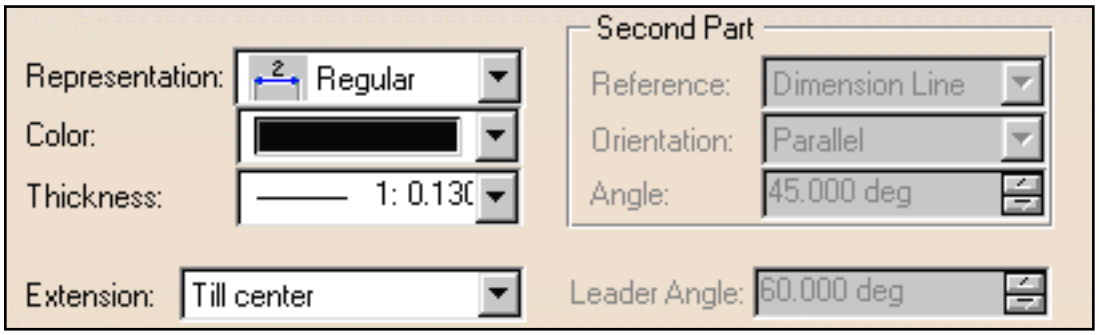


12. Right-click the dimension again, and in the contextual menu, choose **Dimension.3 Object**, and uncheck **Extend to Center**.

    The radius extension line is not extended up to the center anymore.

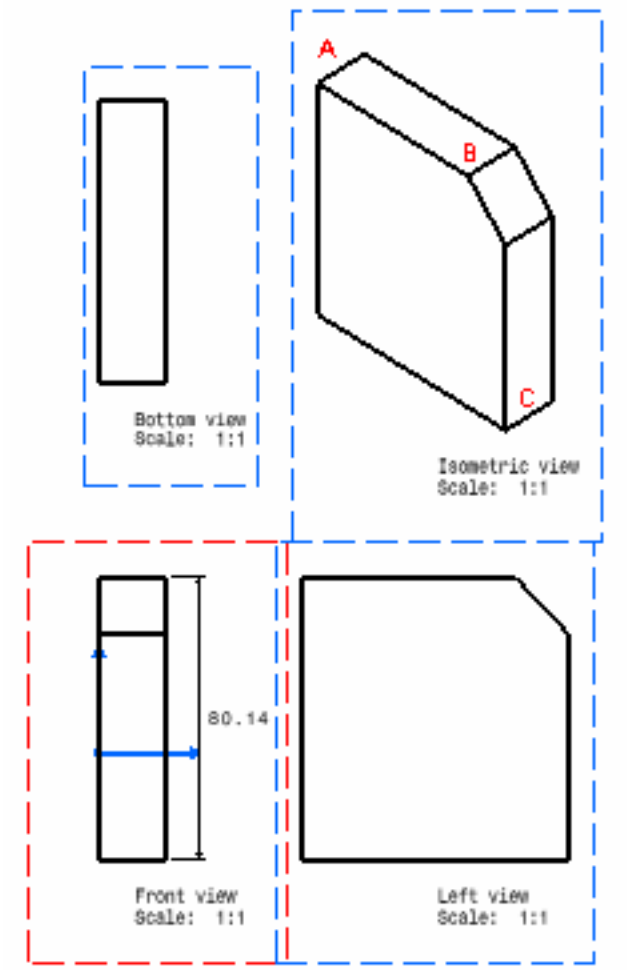
More About Dimensions

- You can use this functionality through the Properties menu: right-click on the dimension and choose **Properties**. On the **Dimension Line** tab, select the type of extension you want from the **Extension** list: **From standard, Till center** or **Not till center**.

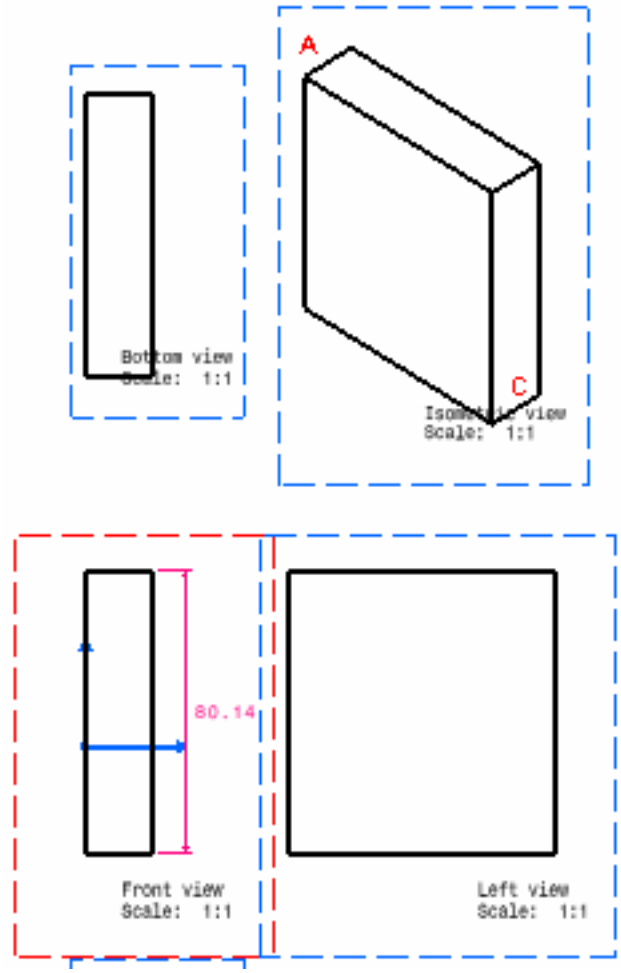


- This functionality works with radius dimension and one-symbol diameter dimension.

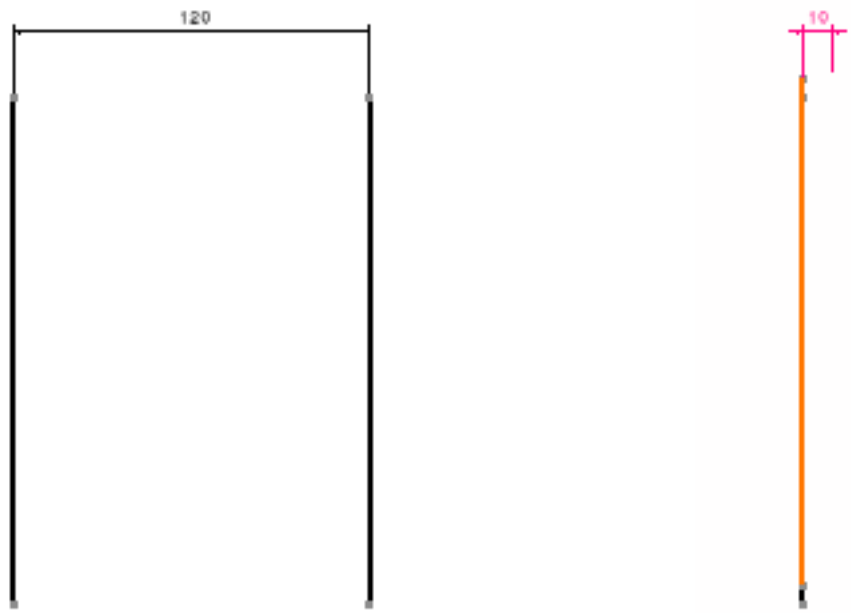
- When you create a dimension between a generated element in a broken view and a sketched element, the dimension value may be false to let the user set a fake dimension value.
- When you create a dimension between an axis and another element, the dimension created by the software is automatically an half dimension.
  To bypass this problem, during creation, uncheck Half Dimensions in the contextual menu (right-click).
- You can generate errors when refreshing the dimensions in the following cases:
  - In this drawing the dimension "80.14" is measured from the line B to the line C:



If the corresponding part is modified and the chamfer removed, when the drawing is refreshed the dimension is colored in fuchsia because the line B was removed with the chamfer:



- If the two elements separated by the dimension value are moved then merged, an error is generated and the dimension turns to fuchsia by default (or according to the color defined for **Not-up-to-date dimensions** in the **Types and colors of dimensions** dialog box available via **Tools -> Options -> Mechanical Design -> Drafting -> Dimension** tab, **Analysis Display Mode** area, **Types and colors...** button).
  Note that in this case, it is not possible to create a null value. Should you need to, you should create a driving dimension and set its value to 0.



Properties

If you right-click the dimension before creation, a contextual menu lets you modify the dimension type and value orientation as well as add funnels. Using this contextual menu once the dimension is created, you can also access the Properties options.

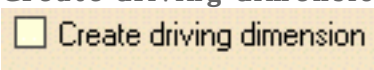
Associativity

If one parent element of the dimension is deleted or deactivated, as soon as you update the drawing, the dimension turns to the color defined for **Not-up-to-date dimensions** in **Tools -> Options -> Mechanical Design -> Drafting**, **Dimension** tab (provided the **Analysis Display Mode** is active).

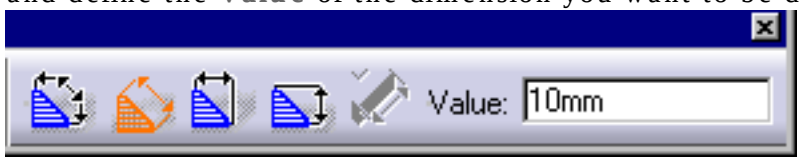
Driving Dimensions

You can create dimensions that will, by default, drive the geometry. For this:

- Go to **Tools -> Options-> Mechanical Design -> Drafting**, **Dimension** tab, and activate the **Create driving dimension** option.

- Create and/or modify the desired dimension on the geometry. If needed, you can use the Tools Palette and define the **Value** of the dimension you want to be driving.

For more information, refer to Creating Driving Dimensions.

True Dimensions

True Length dimensions can be created using the True Length Dimensions option from the Tools Palette or using the contextual menu.

Before using true dimensions, make sure that you have not set the **only create non-associative dimensions** option in **Tools -> Options -> Mechanical Design -> Drafting**, **Dimension** tab, **Associativity on 3D**. In order to work, this functionality must be applied to an associative dimension.

Half-Dimensions

You can create half-dimensions. For this, right-click the dimension as you create it and select the Half-dimension option from the contextual menu.

Extension Line Anchor

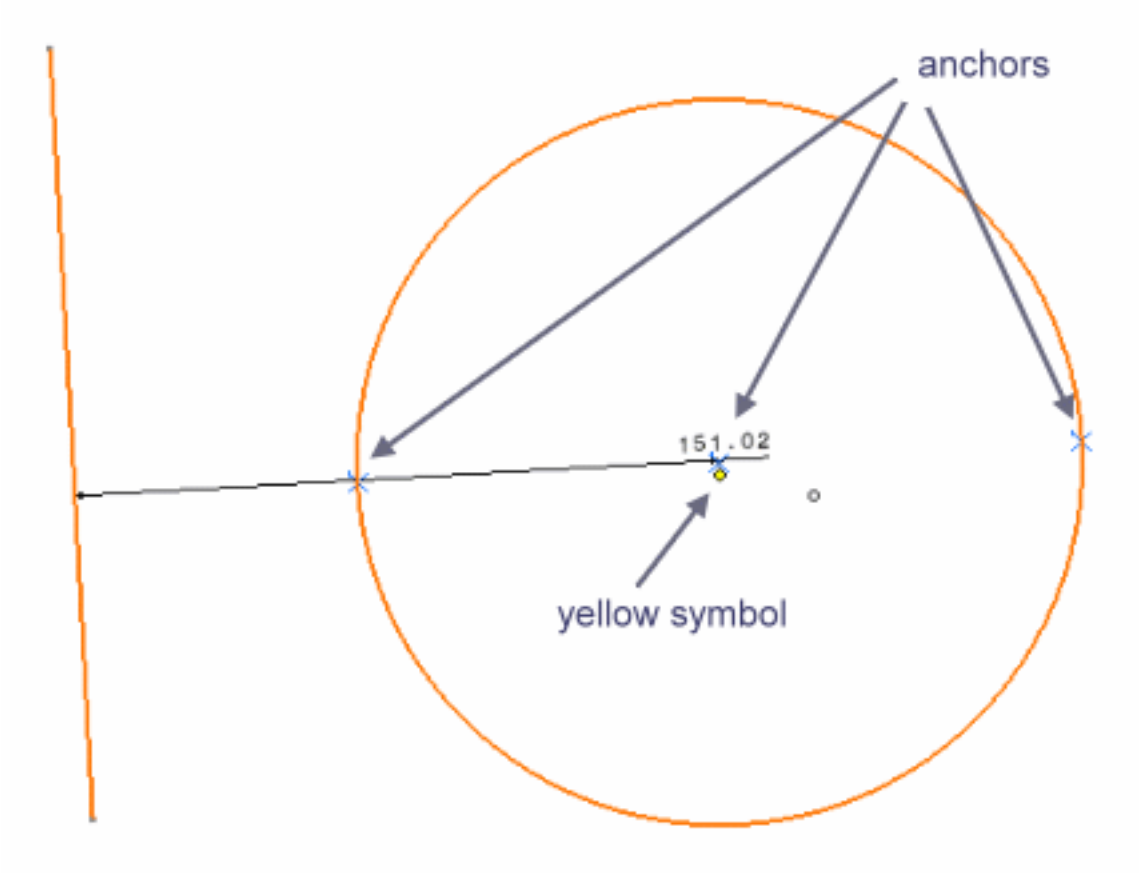
As you create a dimension between two elements, one of these elements being a circle, you can select the extension line anchor. To do tor this, you can :

- use the contextual menu (positioned on the dimension) and select one of the available Extension Line anchor options.

You will thus position the extension line:

  - at one extremity of the circle (First Anchor)
  - at the center of the circle (Second Anchor)
  - at one extremity of the circle (Third Anchor)

- drag the yellow symbol to the one of the anchors (anchors appear when the cursor is over the yellow symbol):



Note that if you selected the **Dimension following the mouse** option in **Tools -> Options -> Mechanical Design -> Drafting -> Dimension** tab, then, to move the extension line anchor, you must press the **Crtl** key before selecting the yellow symbol (to switch temporarily the option).

Creating Half-Dimensions

This task will show you how to create a half-dimension. You can create half-dimensions on distance, angle, diameter cylinders, radius cylinders made out of two selections, diameter edges and diameter tangents. You cannot create half-dimensions on cumulated dimensions.

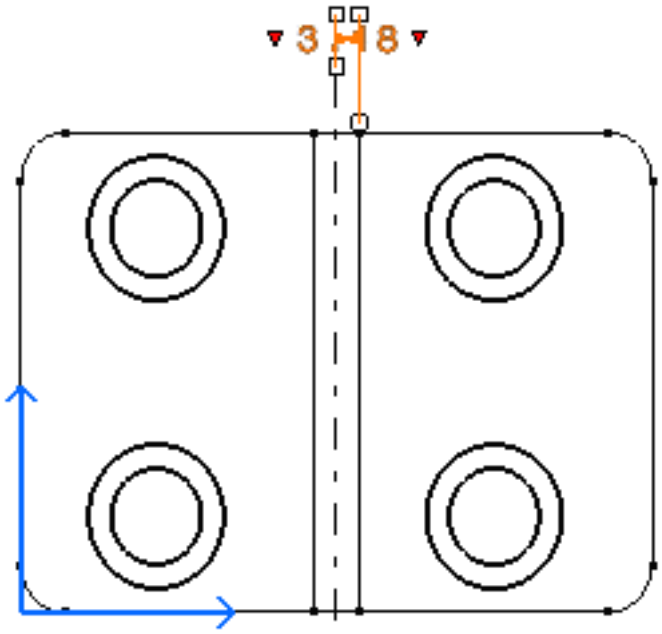
Half-dimensions are useful in the case of revolved features or elements using a plane symmetry, as they let you create the dimensions only on half the geometry.

Open the Brackets_views05.CATDrawing document.

1. Click the **Dimensions** icon from the **Dimensioning** toolbar.

2. Click a first element in the view. For example, an edge.

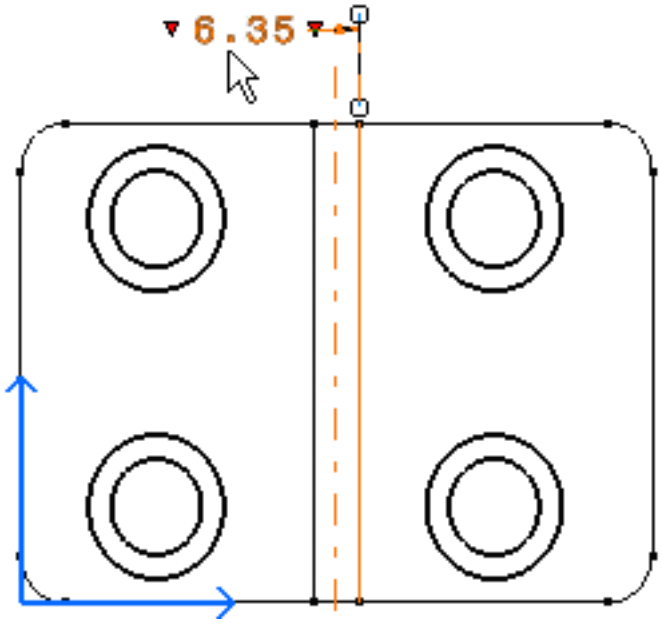
3. If needed, click a second element in the view. For example, another edge.



4. Right-click the dimension and select **Half Dimension**.

   The half-dimension appears. Only one extension line is displayed. The dimension line is shortened

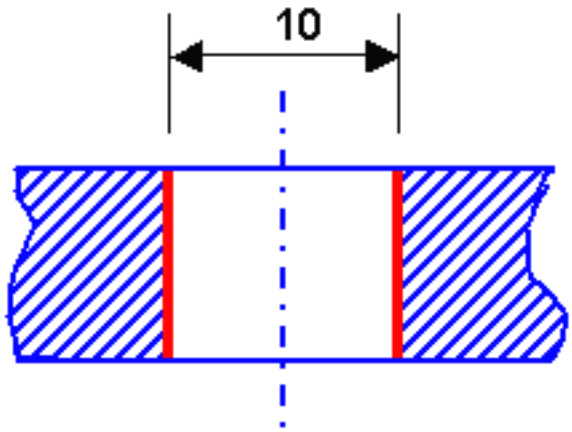
with specific overrun, gap and length. The value is not centered on the dimension line. The attributes mentioned in Dimension parameters drive the dimension graphic display.

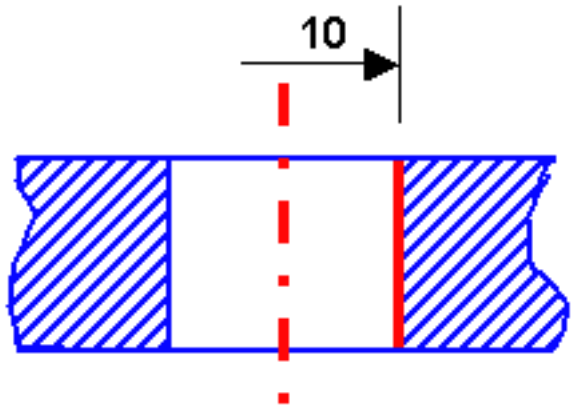


- Once you select the half-dimension option from the contextual menu, all the following dimensions you create will be assigned the half-dimension mode. If you want to create dimensions in the standard mode, go back to the contextual menu and de-activate the Half Dimension option.
- You can create a half-dimension directly by selecting first an axis line and then an other element (which is not an axis). The half-dimension value will be the double of the measured value between the elements. If you don't want a half-dimension to be created when selecting such elements, uncheck Half Dimensions from the contextual menu (right-click) when creating the dimension.

- The dimension value is doubled when they are made out of two selections (distance, angle, 2D diameter cylinder, radius cylinder) but not for dimensions made out of one selection (angle on cone, 3D diameter cylinder, diameter edge, diameter tangent).
- Associativity in the case of half-dimensions is different from associativity in the case of standard dimensions. For example, the half distance dimension below is associated to the axis and the element, whereas a standard dimension is associated to both symmetrical elements.

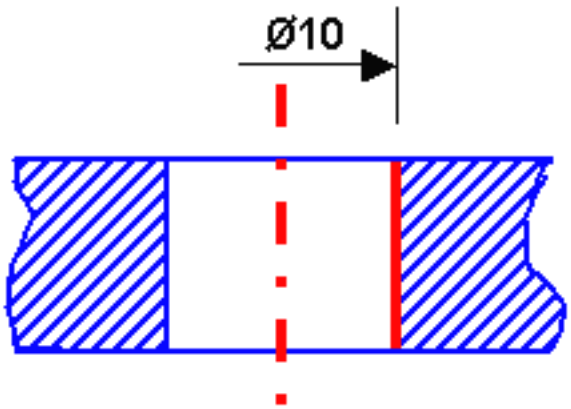
Standard distance dimension: Half distance dimension:





Diameter and radius dimensions are usually created with one selection in 3D. If the dimension is created with two selections, for instance an edge coming from a 3D revolution and another element, the dimension will be not associative. To create the dimension below, you must select only the left or the right side of the cylinder and then right-click on the dimension and select **Half Dimension**.

Half diameter dimension:



Creating Explicit Dimensions

This task will show you how to create a dimension you explicitly decide to be:

- a length/distance dimension
- an angle dimension
- a radius dimension
- a diameter dimension

You will select the required elements. Note that when entering the command dedicated to the creation of a given type of dimension, the default orientation will be the most adequate one.

Open the Brackets_views02.CATDrawing document.

1. Click the desired icon from the **Dimensioning** toolbar (**Dimensions** sub-toolbar).

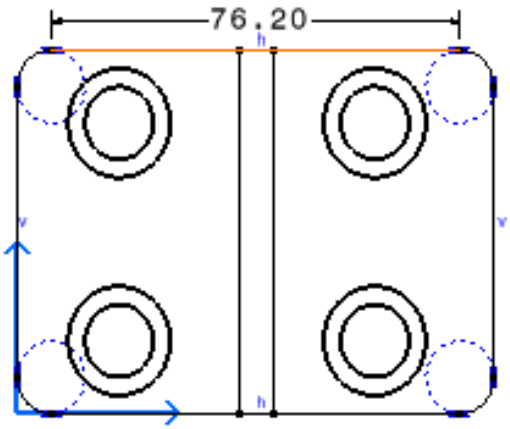
   - **Length/distance dimension**
   - **Angle dimension**
   - **Radius dimension**
   - **Diameter dimension**

2. Click as many elements as required in the view.

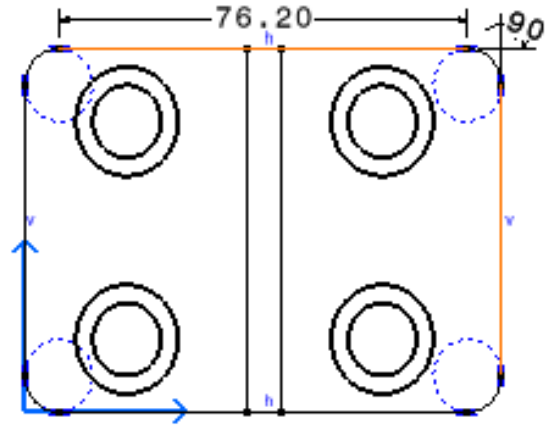
   The Tools Palette automatically appears, displaying dimension modes, except in the case of angle dimensions.

3. If needed, define the dimension mode in the **Tools Palette** using one of the following modes: Projected, Forced or True Length modes. These options are also available in the contextual menu.

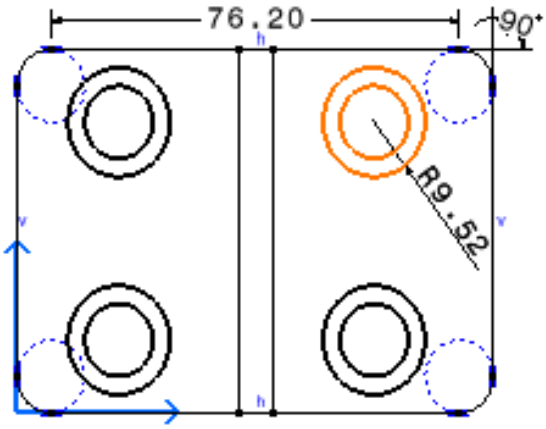
   Length/Distance Angle

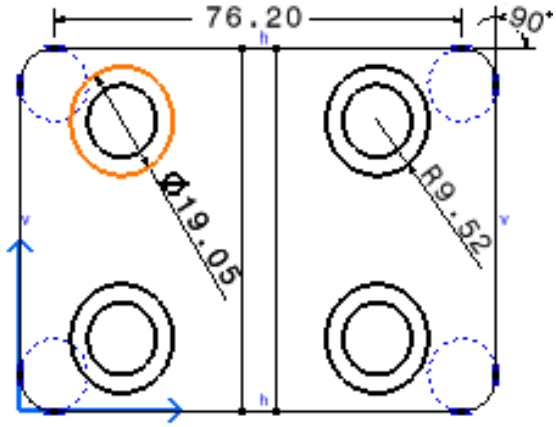


Radius

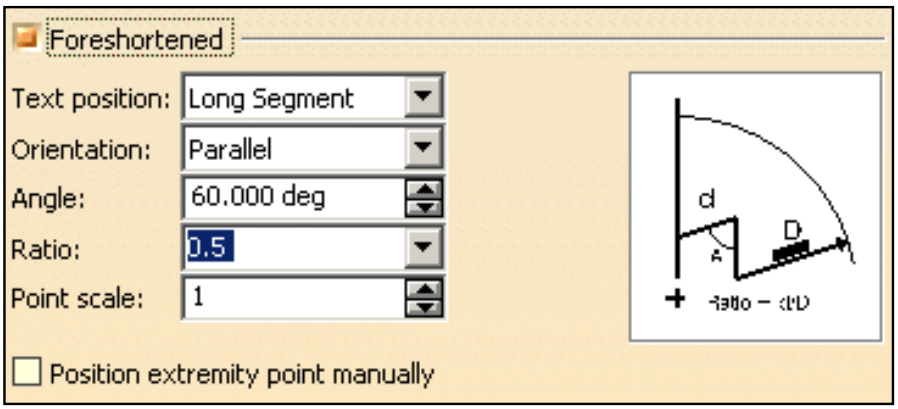


Diameter





For radius dimensions, you can select the **Foreshortened** check box via **Properties -> Dimension Line.**



It allows you to transform a radius dimension line into a foreshortened radius dimension line. Then you can choose the text position (on long segment or short segment), the dimension text orientation according to the dimension line (parallel or convergent), the angle value, the ratio value (short segment/long segment), and the point scale value.

You can also specify whether you want to position manually the extremity point of the foreshortened dimension line (in this case, you will be able to move the extremity point using a yellow manipulator).

Creating/Modifying Angle Dimensions

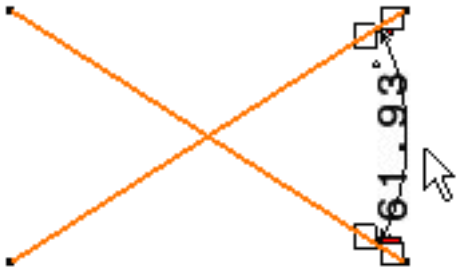
This task will show you how to create an angle dimension and perform the following kinds of modifications: new angle sector or turn an angle sector into a supplementary sector.

Create two lines.

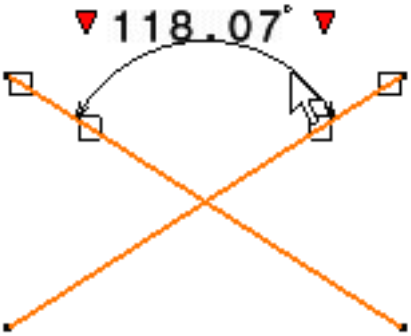
1. Select the **Dimension** icon from the Dimensioning toolbar.

2. Select both lines to be dimensioned, one after the other.

   The angle dimension appears in the sector associated to both selected lines.

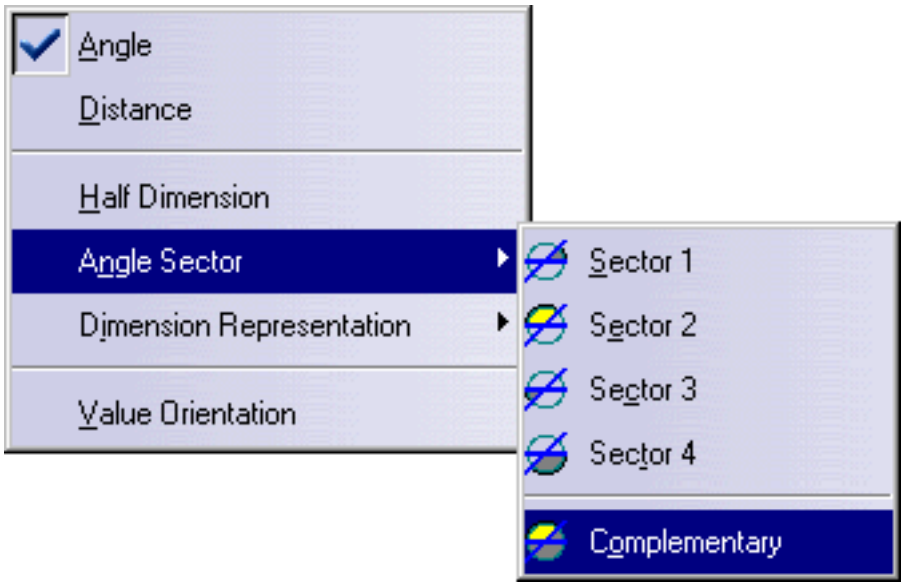


3. Drag the angle dimension line to the desired quadrant (or sector).

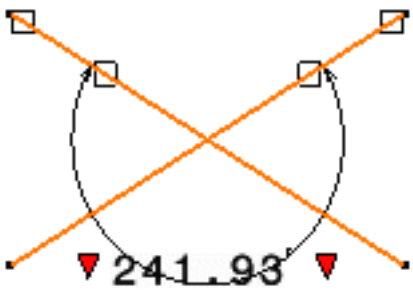


You can move the dimension to a new sector by using the contextual menu:

- Right-click the angle dimension and from the contextual menu, **Dimension.x object > Angle Sector**, either select a given angle sector or the **Complementary** Angle sector.



- You can also CTRL-click the dimension line.



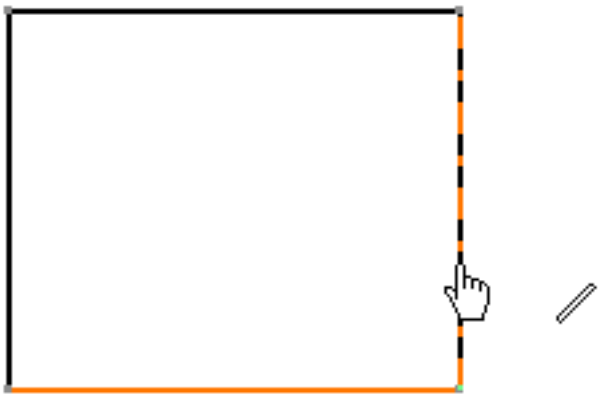
4. Click anywhere to create the angle dimension.

You can edit the angle sector of an existing angle dimension, by right-clicking the angle dimension and selecting the **Dimension_name object -> Angle Sector** command from the contextual menu.

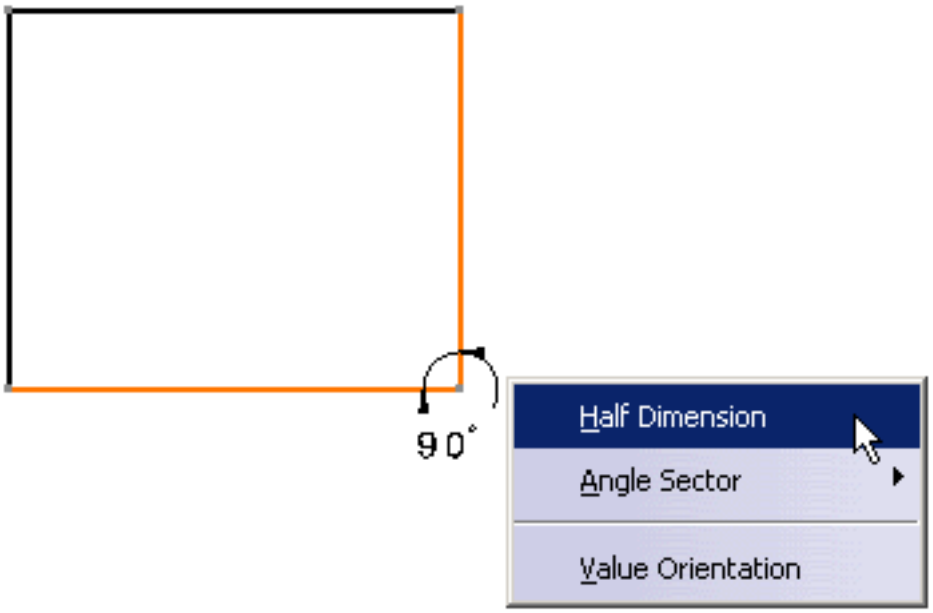
5. Create a rectangle.

6. Select the Angle Dimensions icon from the Dimensioning toolbar.

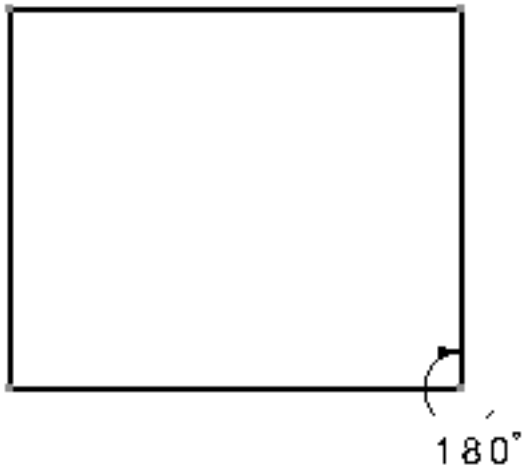
7. Select two lines in the rectangle to create an angle dimension.



8. Click to create the angle dimension, then right click on the dimension and select **Half Dimension** in the contextual menu.

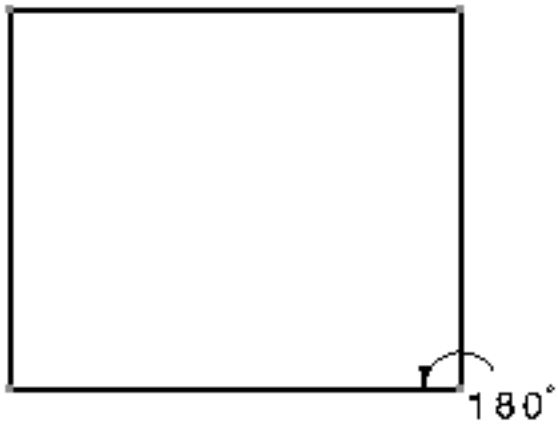


9. Click in a free space to create the dimension.



Note that the half angle dimension's orientation depends on the order of selection of the lines to be dimensioned.

If you had you selected the vertical line first, then the horizontal one, the orientation of the dimension would had been the following:



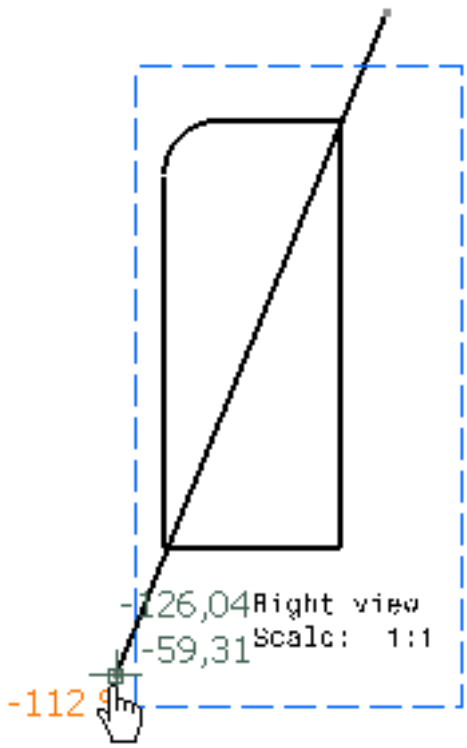
If you need to modify the dimension, use the Re-route dimension command .

Creating Fillet Radius Dimensions

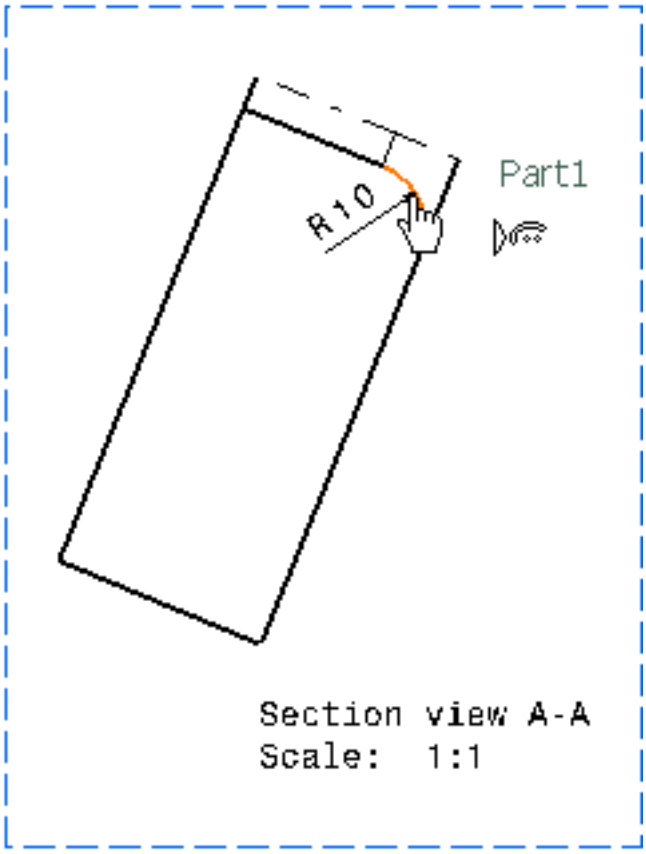
This task will show you how to create a fillet radius dimension in a projected view.

Open the Fillet_Radius.CATDrawing document.

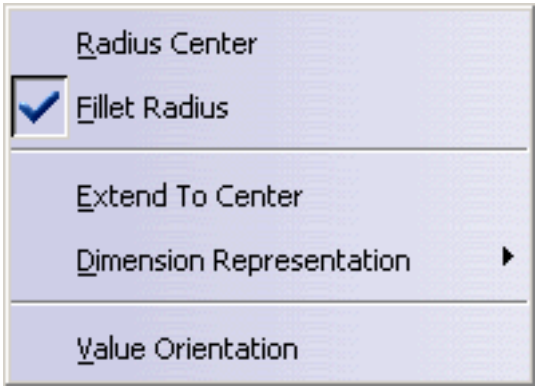
1. Click on the Section View icon to create a section view of the right view so as to display the fillet angle.



2. In the newly created section view, click on Radius icon and select the fillet to dimension.

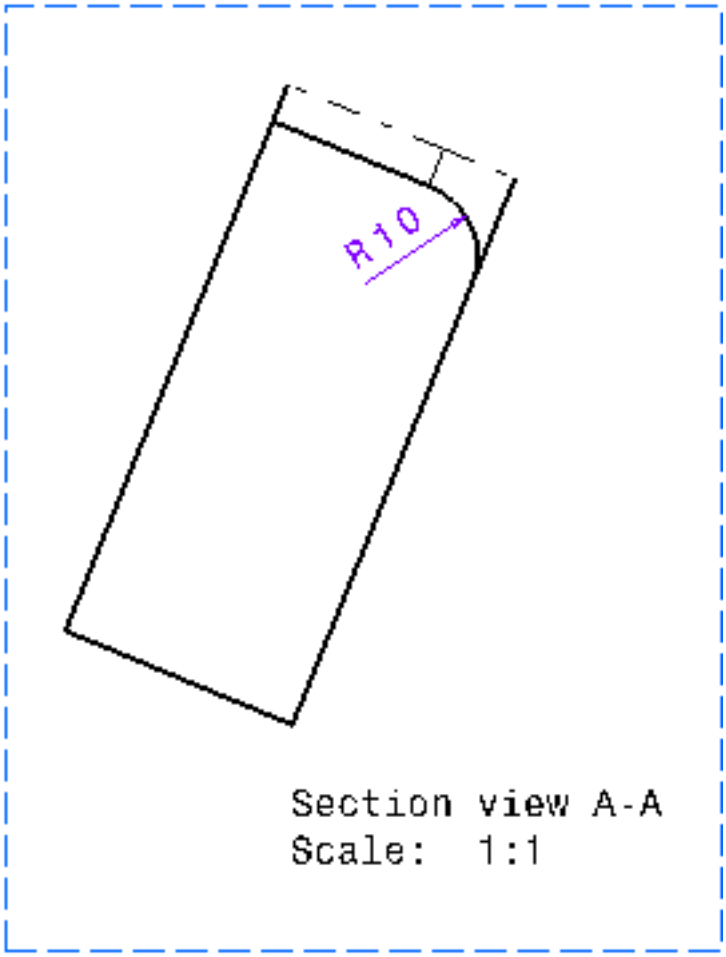


3. Right-click on the fillet and in the contextual menu select **Fillet Radius** to create a dimension on the fillet radius.



4. Click in the free space to create the fillet radius dimension.

   The dimension is created and its color changes indicating it is a true dimension.



- Variable fillet radius can be dimensioned but in that case, their value corresponds to a minimal value.
- Fillet radius dimensions are not supported in Visualization mode.
- Dimension Generation does not support fillet radius dimensions.

Creating Chamfer Dimensions

This task will show you how to create a chamfer dimension.

This task deals with:

- Creating chamfer dimensions manually
- Creating chamfer dimensions using chamfer detection
- Remarks about chamfer dimensions

Creating chamfer dimensions manually

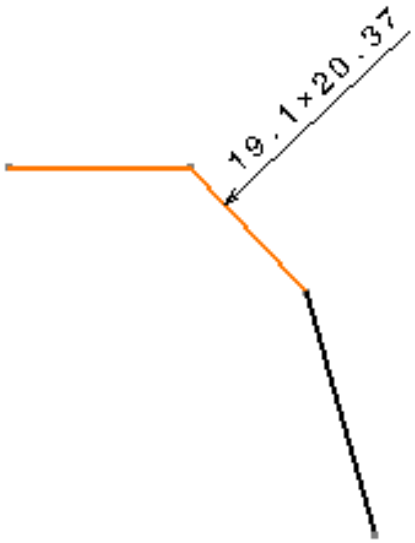
Open the IntDrafting_Dim_Chamfer.CATDrawing document.

1. Go to **Tools -> Options -> Mechanical Design -> Drafting -> Dimensions** tab and make sure the **Detect chamfer** check box is not selected.
2. Click the **Chamfer Dimensions** icon from the **Dimensioning** toolbar (**Dimensions** sub-toolbar).

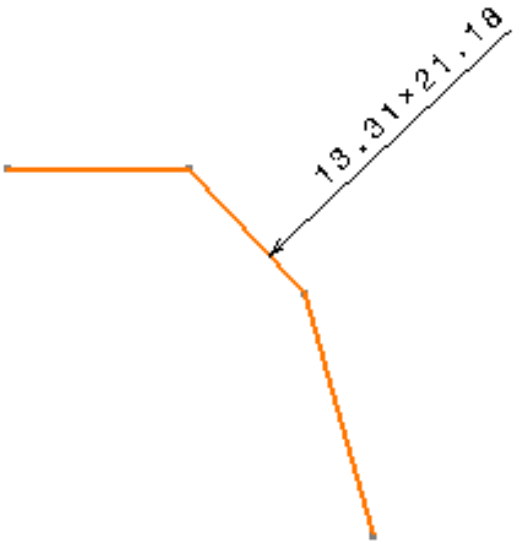
3. In the **Tools Palette** which is displayed, you can choose:
   - The format of the dimension:
     - **Length x Length** (19,1 x 19,1 in our example)
     - **Length x Angle** (19.1 x - 46deg84'8" in our example)
     - **Angle x Length** (- 46deg84'8" x 19.1 in our example)
     - **Length** (19,1)
   - The representation mode:
     - **One symbol**
     - **Two symbols**

You can also access these options using the contextual menu: at any time during the chamfer dimension creation, you can right-click to display the contextual menu.

4. Choose the **Length x Length** format and the **One symbol** mode .
5. Select the element to be dimensioned.
6. Select a reference line or surface.
7. You have two options:



- Click on the sheet to end the dimension creation. The chamfer dimension is computed with an implicit second reference line that is *perpendicular* to the first one.



- Select a second reference line or surface. In this case, the chamfer dimension is computed according to both reference lines you selected.

  In a Generative Drafting context (i.e. in the case of a generative view), you must do this, i.e. you must explicitly select the second reference line.

In any case, the dimension is associated to all the elements you selected.

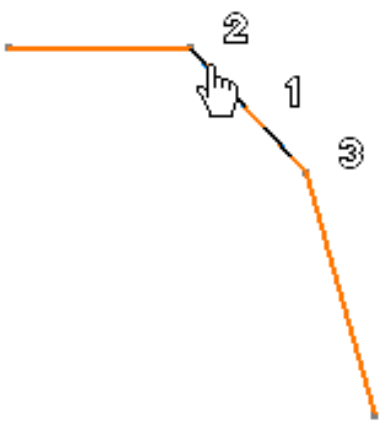
Creating chamfer dimensions using chamfer detection

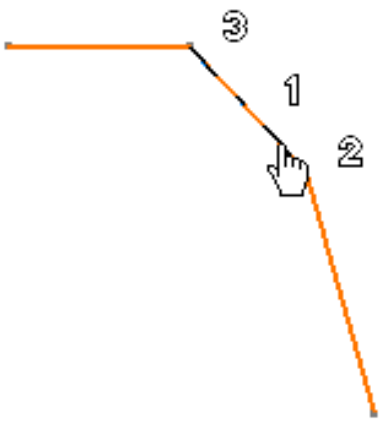
Note that chamfer detection is provided as a help in selecting chamfers. However, depending on the geometrical configuration, it may not detect all chamfer types. If your chamfer is not detected, you can still create the chamfer dimension manually as explained below.

Open the IntDrafting_Dim_Chamfer.CATDrawing document.

1. Go to **Tools -> Options -> Mechanical Design -> Drafting -> Dimensions** tab and make sure the **Detect chamfer** option is selected.
2. Click the **Chamfer Dimensions** icon from the **Dimensioning** toolbar (**Dimensions** sub-toolbar).
3. In the **Tools Palette** which is displayed (as well as in the contextual menu), you can choose the format of the dimension and the representation mode. For more information, refer to step 2 in Creating chamfer dimensions manually.

   Choose the **Length x Length** format and the **One symbol** mode .
4. Fly the mouse over the element to be dimensioned. You can notice that, depending on where you position the cursor, the auto-detection agent indicates a different order for taking elements into account when creating the chamfer dimension:
   - 1 indicates the element to be dimensioned.
   - 2 indicates the line which will be used as the first reference.
   - 3 indicates the line which will be used as the second reference.

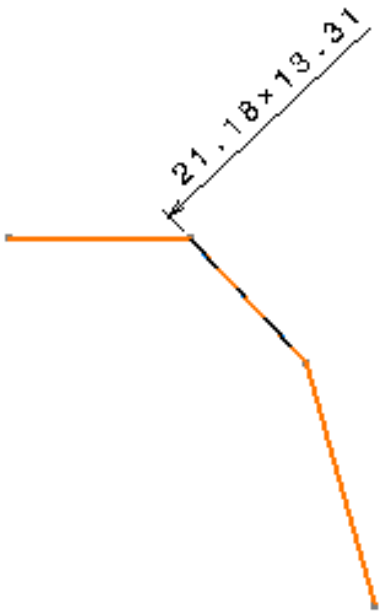




5. Click when you are satisfied with the order offered by the auto-detection agent. For example, click to accept the 3 - 1 - 2 order. The chamfer dimension is computed according to the first and the second auto-detected reference lines.

At this stage, if you are not satisfied with the order you just accepted, or if your chamfer is not detected, you can still click to select the first reference line, and, optionally, the second reference line. This amounts to creating the chamfer dimension manually.

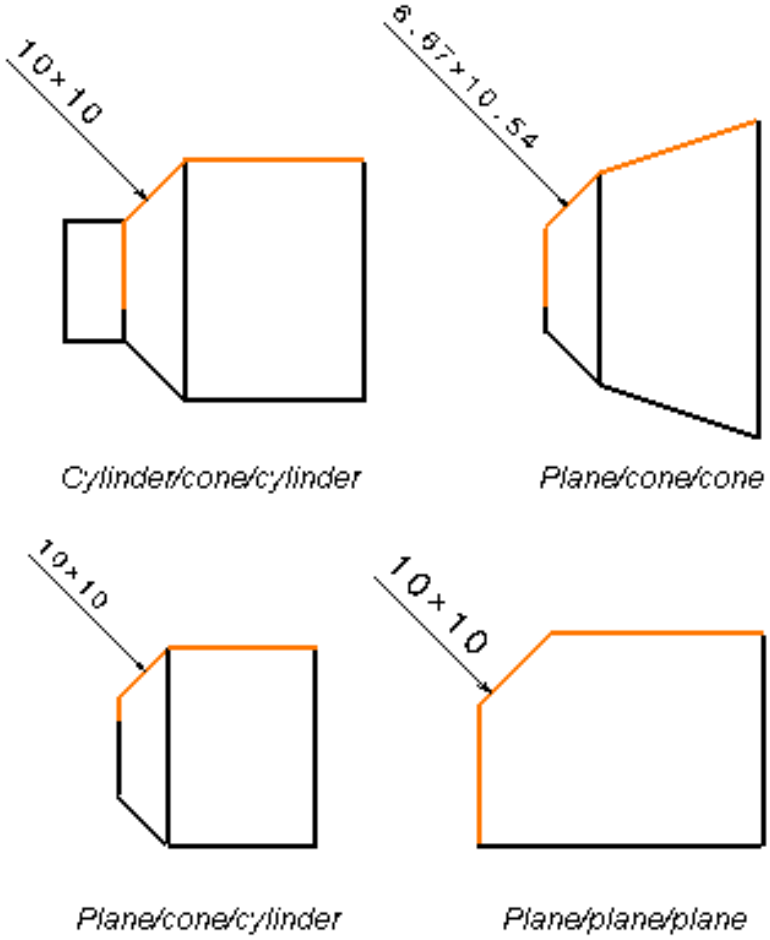
6. Click to end the chamfer dimension creation.



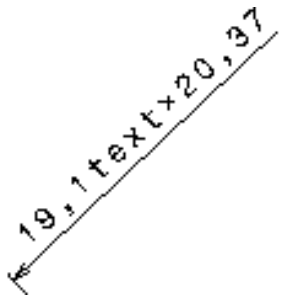
The dimension is associated to all auto-detected elements.

Remarks about chamfer dimensions

- In a Generative Drafting context, you can create chamfer dimensions for the following types of cylindrical shapes: cylinder/cone/cylinder, plane/cone/cone, plane/cone/cylinder, plane/plane/plane.



- When creating chamfer dimension on cylindrical shapes in a Generative Drafting context, remember that:
  - in the case of projection views, the projection plane needs to be parallel to the cylinder axis.
  - in the case of section views or section cuts, the section plane needs to to be parallel to, and to go through, the cylinder axis.
  - the sketched profile on which the cylinder (or the cone) is based must be a circle.
- All settings defined in **Tools -> Option -> Mechanical Design -> Drafting** (**Dimensions** and **Manipulators** tabs) are taken into account when creating chamfer dimensions.
- When editing chamfer dimension text properties (**Edit -> Properties** command, **Dimension Texts** tab), any information (e.g. associated text, fake dimension, tolerance, text before/after, etc.) added to the main value, will actually be positioned according to the first value (excluding the "x" symbol, e.g. "19,1").
  This information will be positioned in the following order: Text Before/Prefix/first value/Tolerance/Suffix/Text After/second value (including the "x" symbol, e.g. "x 20,37"). An example is provided below, with a Text After.



- When re-routing chamfer dimensions which have only two reference elements (the element to be dimensioned and a single reference line or surface), you will need to select three reference elements.

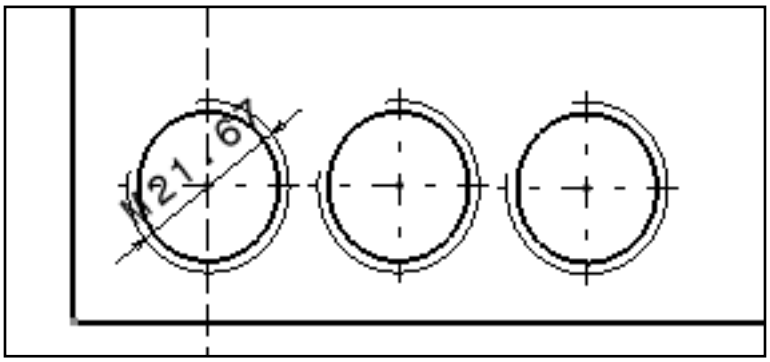
Creating Thread Dimensions

This task will show you how to create thread dimensions.

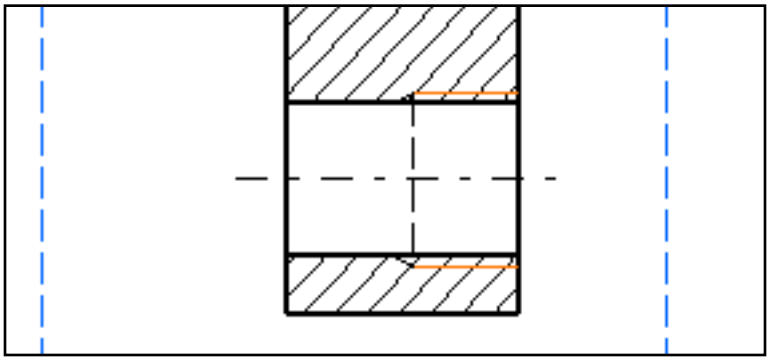
Open the intthread.CATDrawing document.

1. Click the **Thread Dimension** icon from the **Dimensioning** toolbar (**Dimensions** sub-toolbar).

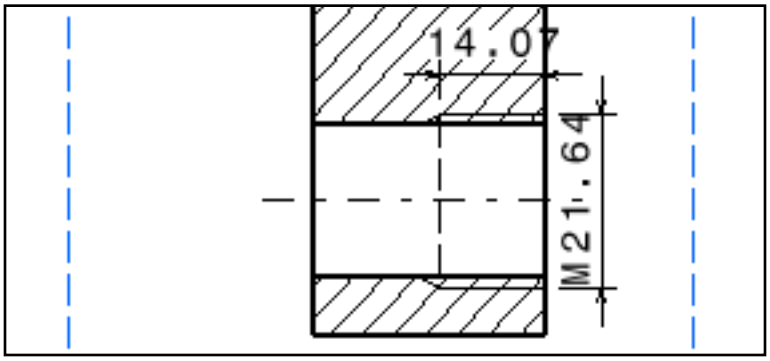
2. In the front view, select the thread to be dimensioned. The thread diameter dimension appears.



3. Click the **Thread Dimension** icon again.
4. In the section view, select the two lines representing the thread to be dimensioned.



Two thread dimensions appear, representing the thread diameter as well as the thread depth.



- In top views, you can modify the orientation of threads dimensions.
- For information on creating associative thread dimensions in a Generative Drafting context, refer to Creating Associative Thread Dimensions in the *Generative Drafting User's Guide*.

Creating/Modifying Coordinate Dimensions

This task will show you how to automatically create 2D or 3D coordinate dimensions on elements. Coordinate dimensions allow you to define the coordinates of a point relative to the X, Y, and possibly Z, axes.

Open the PointSketch.CATDrawing document.

1. Click **Coordinate Dimension** in the **Dimensioning** toolbar (**Dimensions** sub-toolbar).

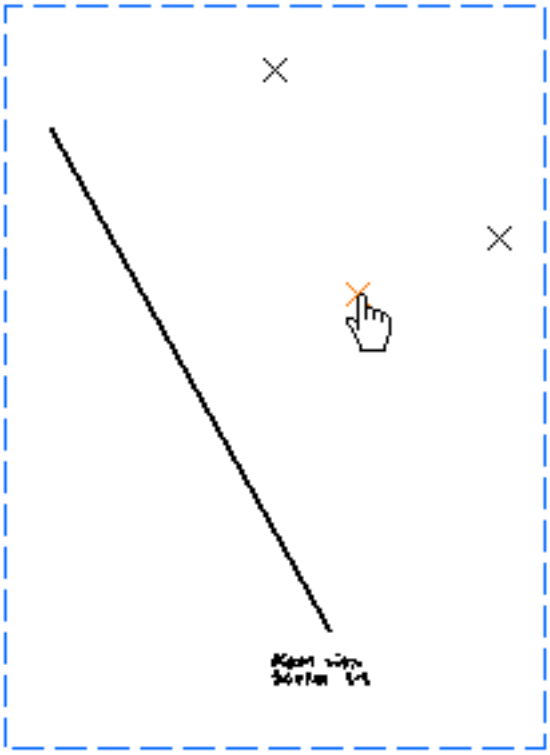
   The **Tools** palette appears with two options:

   - **2D Coordinates** lets you create 2D (x, y) coordinate dimensions for interactive geometry.
   - **3D Coordinates** lets you create 3D (x, y, z) coordinate dimensions for generative geometry.

   - These options are also available via the contextual menu.
   - This choice of options is valid for generative geometry only. In the case of a generative drawing, or in the case of a drawing containing a mix of generative and interactive elements, both options will be available, but if you select sketched (i.e. interactive) geometry, the 2D Coordinates option will be applied automatically (even if you selected the 3D Coordinates option). In the case of a purely interactive drawing, the options will not be displayed at all, and only the 2D Coordinates option will be applied.

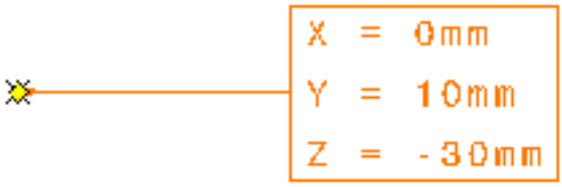
2. Select **3D Coordinates** in the **Tools Palette**, as you will be dimensioning elements generated from the 3D.

3. Select the element for which you want to create the 3D coordinate dimension. The coordinate dimension is created immediately.

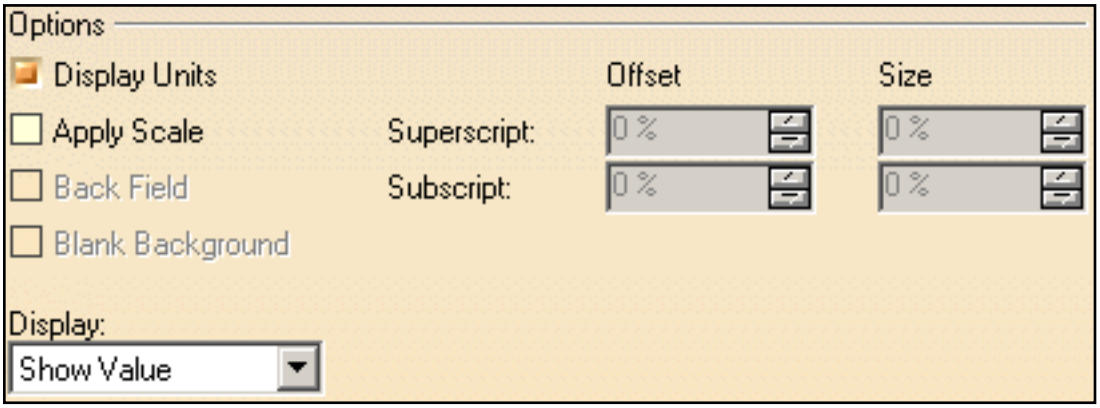


- At this point, you can right-click to display the contextual menu, which allows you add a breakpoint to the leader, or to choose the leader symbol.
- You can also select a set of elements by trapping them with the mouse, to create several coordinate dimensions in one shot.

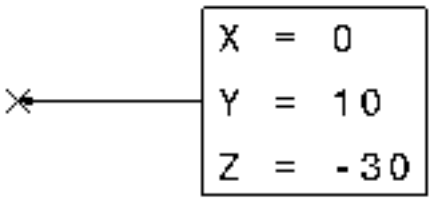
4. Click in the free space to end the dimension creation.
5. Select the coordinate dimension to modify its position. The dimension is highlighted and its anchor point appears in yellow.
6. Drag the dimension to a new position.



- Coordinates are relative to the absolute axis system except for views created by selecting a 3D local axis system.
- The yellow anchor point is associative and is linked to the element you dimensioned.
- If you create a coordinate dimension on the origin, this dimension is invariably non-associative. In this case, the leader symbol may be different from the leader symbol used for associative coordinate dimensions. Refer to Dimension Styles > Coordinate Dimension Styles for more information.
- If you need to hide the coordinate dimension's unit, you can do so by editing the properties of the coordinate dimension (via **Edit -> Properties**): select the **Text** tab and uncheck the **Display Units** option.



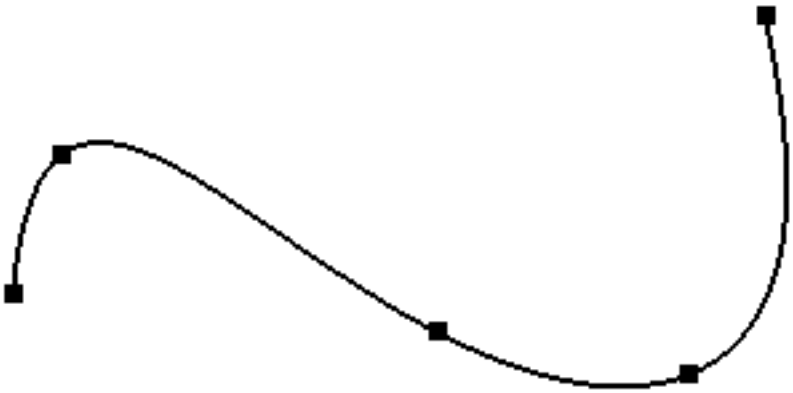
Then click **OK** to update the view: the units should not be displayed anymore.



Creating Curvature Radius Dimensions

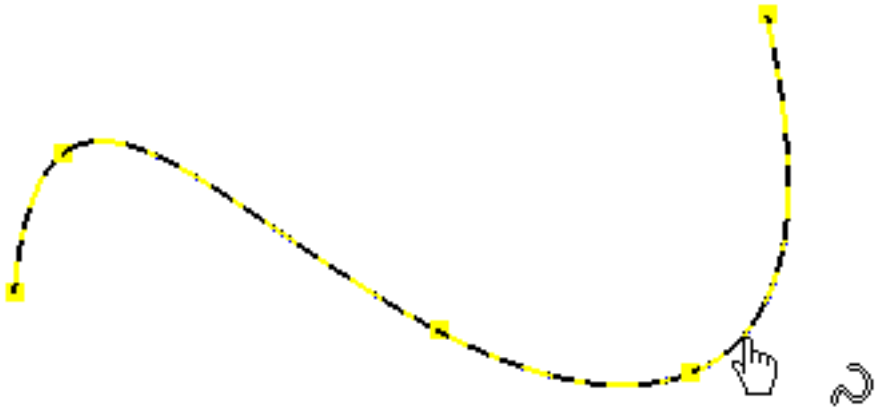
This task will show you how to create and modify a curvature radius dimension. A curvature radius dimension lets you know the curvature radius at a given point on a curve (spline, ellipse, etc.).

Create a spline.



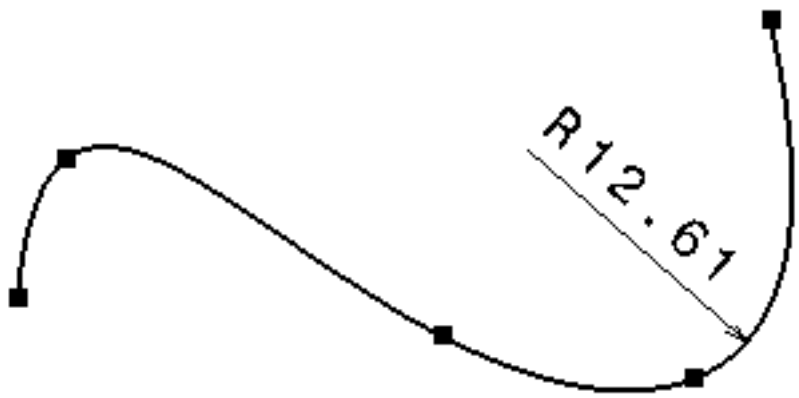
1. Select the **Dimension** icon from the **Dimensioning** toolbar.

2. Move the cursor over the spline. You can notice that the cursor changes to indicate that you are going to create a dimension on a spline.

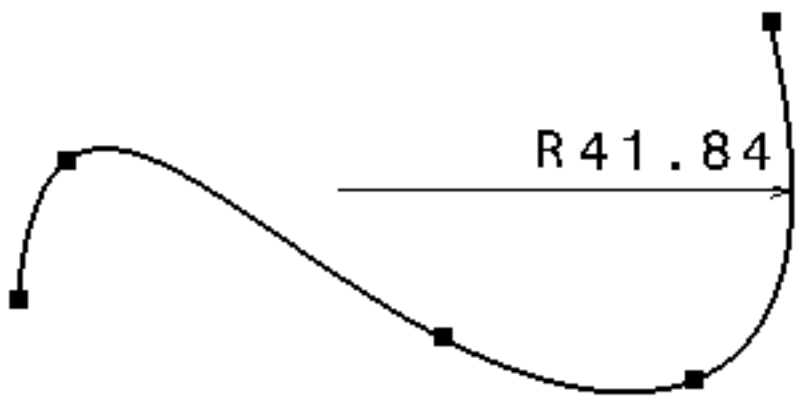


3. On the spline, click the point where you want to create the curvature radius dimension. A preview of the curvature radius dimension is displayed.

4. Click to validate the dimension creation.



5. Move the dimension over the spline to modify the dimension.



Creating Overall Curve Dimensions

This task will show you how to create overall dimensions on curves. You can create dimensions on the overall horizontal or vertical size of any kind of curve, whether it is canonical or not (e.g.: ellipse, spline, etc.). You can also create dimensions on the overall size between 2 curves, or between a curve and a line, for example.

Go to **Tools -> Options -> Mechanical Design -> Drafting.** On the **Dimension** tab, uncheck **Dimension following the cursor (CTRL toggles)**.

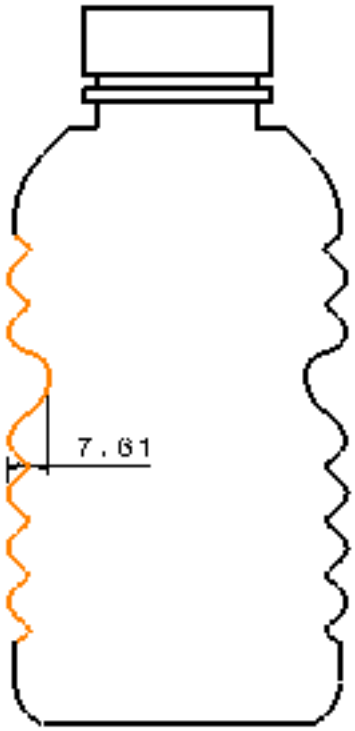
Open the Dimension_Spline.CATDrawing document.

1. Click the **Dimension** icon from the **Dimensioning** toolbar.

2. In the Tools Palette, click the **Force horizontal dimension in view** icon to specify that you want to create the dimension based on the horizontal direction.

The direction of overall curve dimensions can only be horizontal or vertical.

3. Select a spline. A preview of the dimension is displayed.



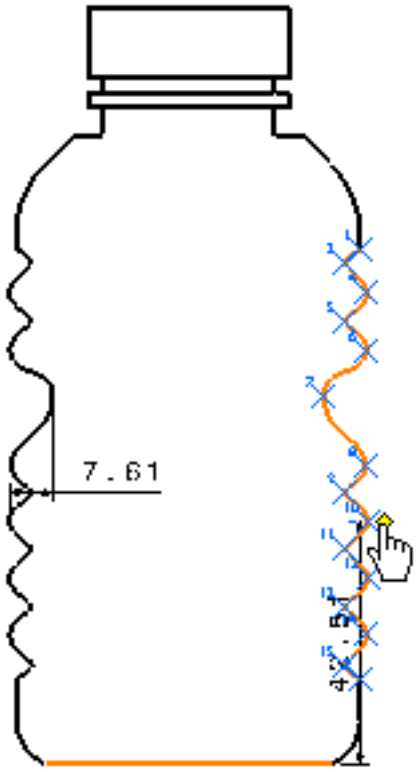
If the preview shows a curvilinear length dimension instead of an overall curve dimension, right-click to display the contextual menu and select **Overall** instead of **Curvilinear Length**.

4. Click elsewhere in the drawing to validate the dimension creation. The dimension you created indicates the overall horizontal size of the spline.

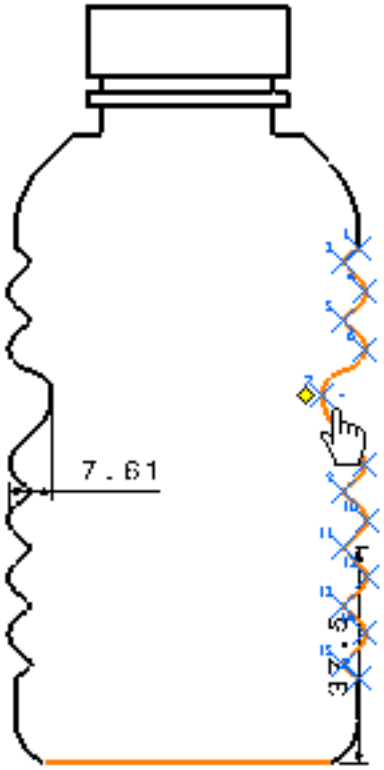
5. Again, click the **Dimension** icon .

6. In the Tools Palette, click the **Force vertical dimension in view** icon to specify that you want to create the dimension based on the vertical direction.

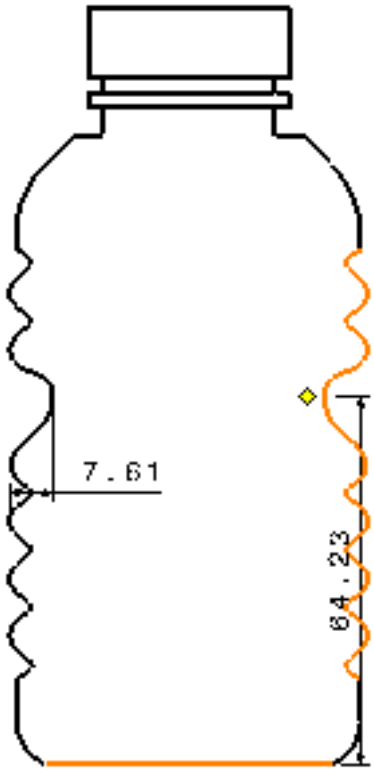
7. Select the bottom line and the other spline. A preview is displayed. Yellow manipulators and point indicators appear: these let you select precisely the points that you want the dimension to take into account.



8. Move the spline dimension manipulator to point 7 on the spline, for example.



The preview is updated.



9. Click in the drawing to validate the dimension creation. The dimension you created indicates the overall vertical distance between the bottom line and point 7 of the spline.

You can edit the dimension representation of an existing dimension, by right-clicking the dimension and selecting the **Dimension_name object -> Dimension Representation** command from the contextual menu.

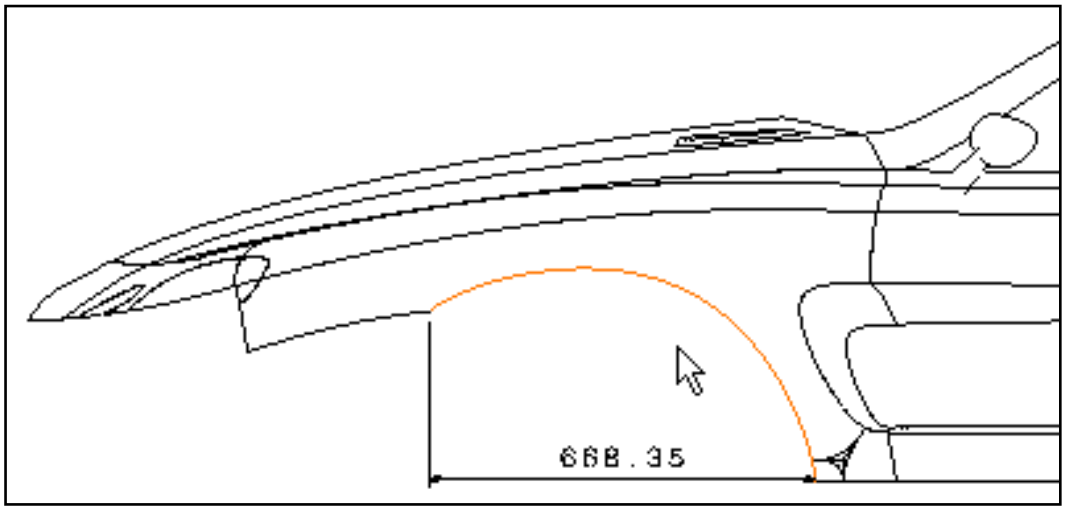
Creating Curvilinear Length Dimensions

This task will show you how to create dimensions for the curvilinear length of a curve, i.e. to measure the overall length of a curve.

Open the CurvilinearDimension.CATDrawing document.

1. Click the **Dimension** icon from the **Dimensioning** toolbar.

2. Select a curve. A preview of the dimension is displayed. By default, this preview shows an overall curve dimension.

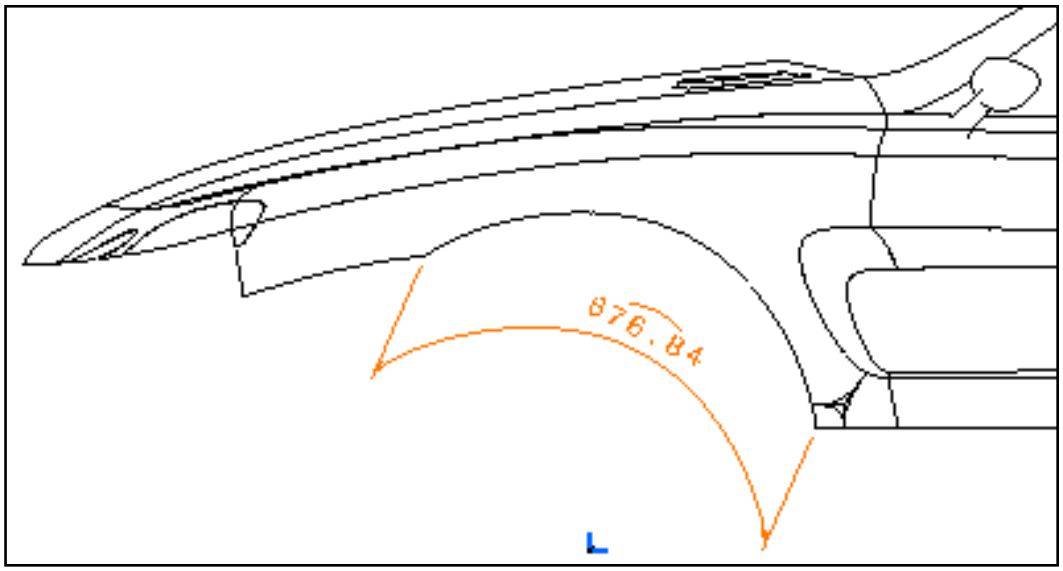


3. Right-click to display the contextual menu and select **Curvilinear Length** instead of **Overall**.
4. Still in the contextual menu, select a representation mode for the dimension line:
   - **Offset** displays the dimension line as an offset of the measured curve.
   - **Parallel** displays the dimension line as a translation of the measured curve.
   - **Linear** displays the dimension line as linear.

Select **Parallel**, for example.

5. Optionally drag the dimension line and/or the dimension value to position them as wanted.

6. Click elsewhere in the drawing to validate the dimension creation. The semi-arc symbol displayed over the dimension value symbolizes a curvilinear length dimension. You can now handle the dimension just like any other dimension.

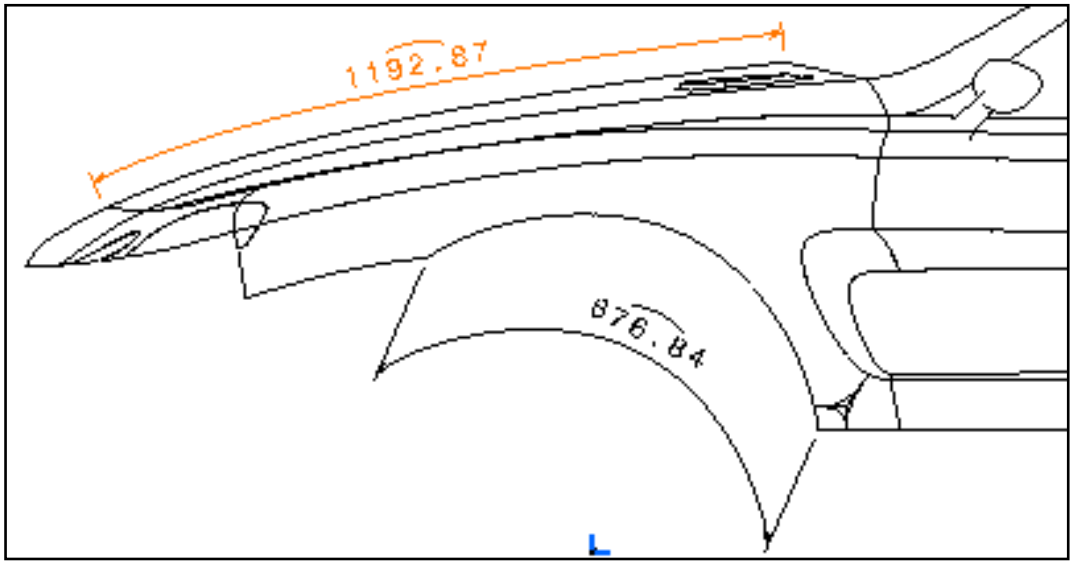


7. Again, click the Dimension icon .

8. Select another curve. This time, the preview of the dimension shows a curvilinear length dimension (your previous selection was memorized).

9. Once again, right-click to display the contextual menu and select **Offset** as the representation mode for the dimension line.

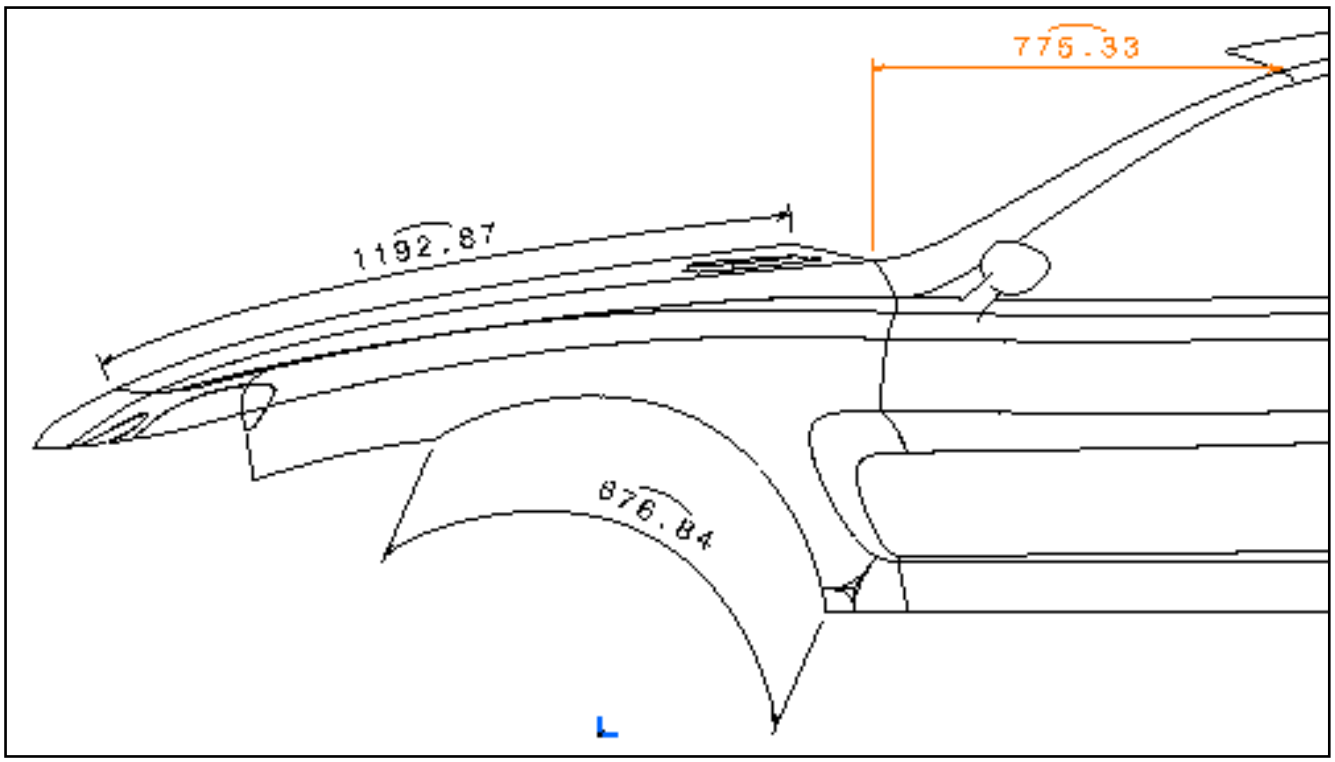
10. Click in the drawing to validate the dimension creation.



11. Repeat steps 7 to 9, this time selecting **Linear** as the representation mode for the dimension line.

12. Still in the contextual menu, select **Dimension Representation -> Force Horizontal Dimension in View** to specify the dimension line orientation.

13. Click in the drawing to validate the dimension creation.



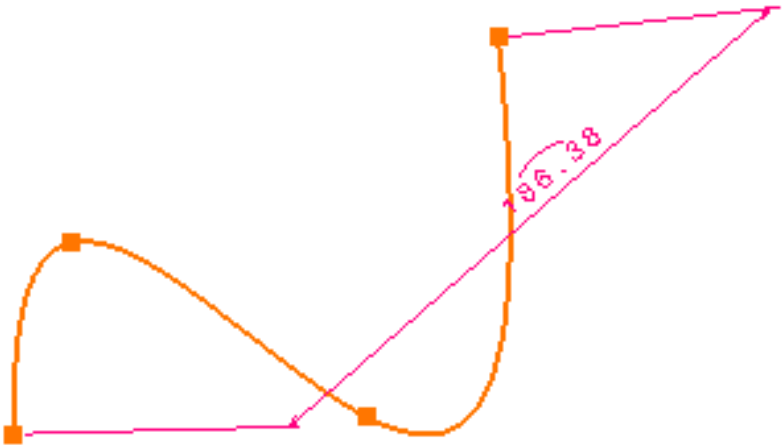
More About Curvilinear Length Dimensions

General remarks

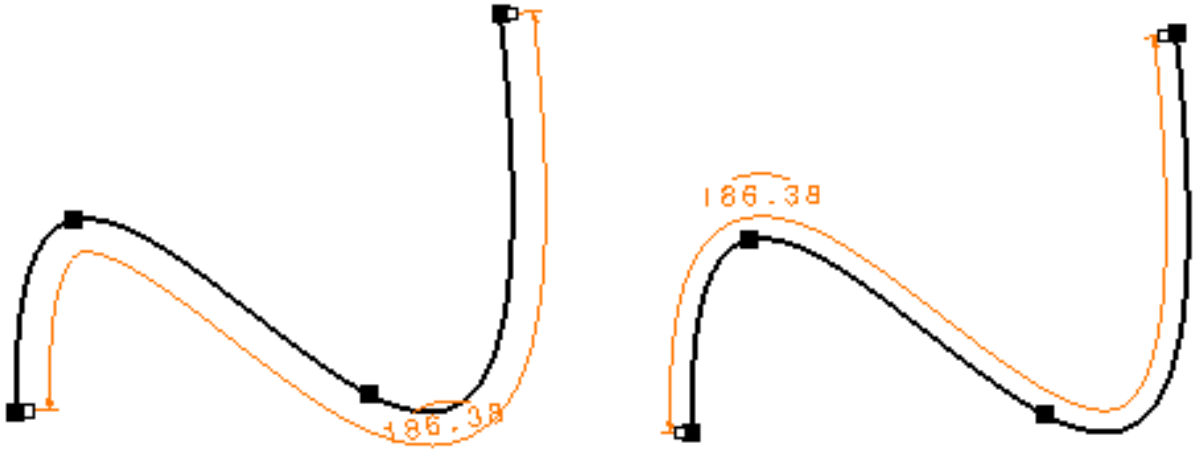
- Curvilinear length dimensions can be created using the **Dimensions** and **Length/Distance Dimensions** commands; they cannot be created using the **Stacked Dimensions** and **Cumulated Dimensions** commands.
- You can create curvilinear length dimensions for all types of curves: splines, circles, arcs of circle, conics, etc. Note that in the case of circles and arcs of circle, they will be called *circular* length dimensions.
- The curvilinear length symbol is defined by the administrator in the standards.
- The linear representation mode for the dimension line is:
  - forbidden in the case of closed curves.
  - the only authorized representation mode for True Length dimensions.

Restrictions

- You cannot change the dimension line representation mode or orientation after the dimension has been created.
- In the case of the parallel and offset representation modes, the dimension value cannot be moved out of the curve limits, except for circles and arcs of circle. As a result, you cannot specify the dimension value position (Inside, Outside, Auto).
- In some cases, depending on the curve and on the offset value, the offset representation mode cannot be computed:
  - In certain cases, when switching from another representation mode to the offset mode, the dimension will be previewed as being not-up-to-date (i.e. using the color configured in **Tools -> Options -> Mechanical Design -> Drafting -> Dimension** tab, **Analysis Display Mode**): try to move the cursor closer to the dimension.



  - In other cases, you will not be able to position the dimension further than a certain limit. The examples below show the limits for positioning a curvilinear length dimension in offset mode for a spline.



- In the case of curvilinear length dimensions in offset mode, it is recommended to activate the **Constant offset between dimension line and geometry** setting in **Tools > Options > Mechanical Design > Drafting > Dimension** tab. This will ensure that the dimension remains associative if the geometry is moved.
- When dimensioning a 3D curve that is not planar, the extension line of the curve will extend to the projection of the endpoints of the curve in the view plane of the dimension. As a result, the dimension may seem to point nowhere.
- Curvilinear dimensions cannot be measured along a direction.
- Curvilinear dimensions cannot be driving dimensions.

Creating Partial Curvilinear Length Dimensions

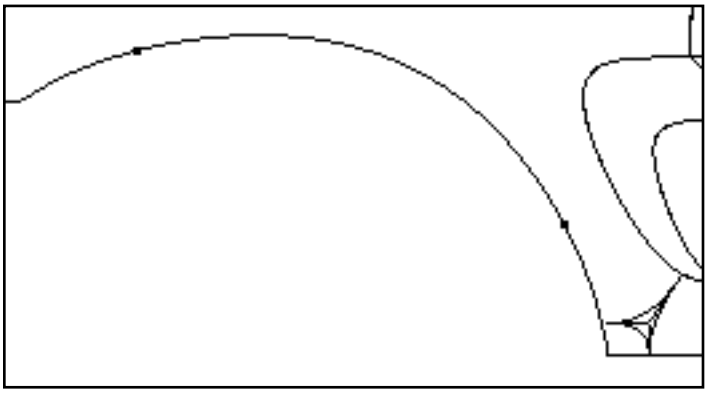
This task will show you how to create dimensions for the curvilinear length of a curve portion, i.e. to measure the partial length of a curve.

Partial curvilinear length dimensions are defined using points. You can use two different methods:

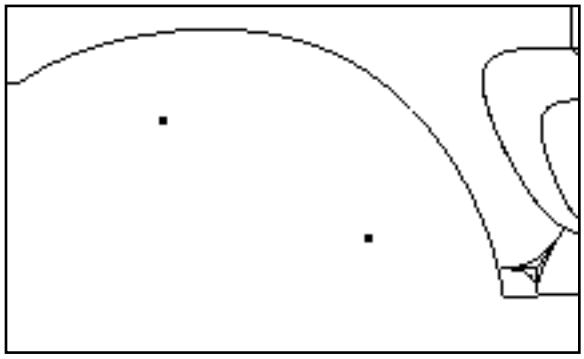
- Creating partial curvilinear length dimensions using existing points
- Creating partial curvilinear length dimensions using points created on-the-fly

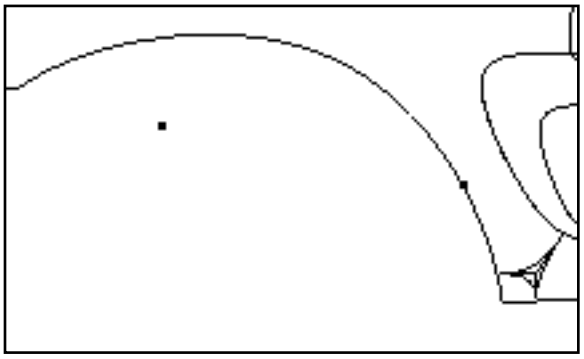
Creating partial curvilinear length dimensions using existing points

Open the CurvilinearDimension.CATDrawing document. Create two points on a curve, for example. These points will be used to define the extremities of the curve portion to dimension.



You can also use spline control points (but there is none in the sample provided for this scenario), or points created in free space. In the case of points in free space, the partial curvilinear length dimension will be computed according to the normal projection of these points on the curve. So, when creating such points, you need to make sure that they will be projected on the curve, as shown below for example.





1. Click the **Dimension** icon from the **Dimensioning** toolbar.

2. Select the curve on which you created the points. A preview of the dimension is displayed. By default, this preview shows an overall curve dimension.

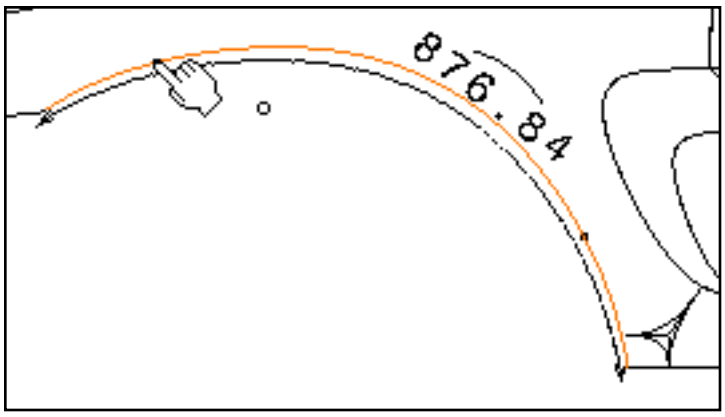
3. Right-click to display the contextual menu and select **Partial Curvilinear Length** instead of **Overall**.

4. Still in the contextual menu, select a representation mode for the dimension line:

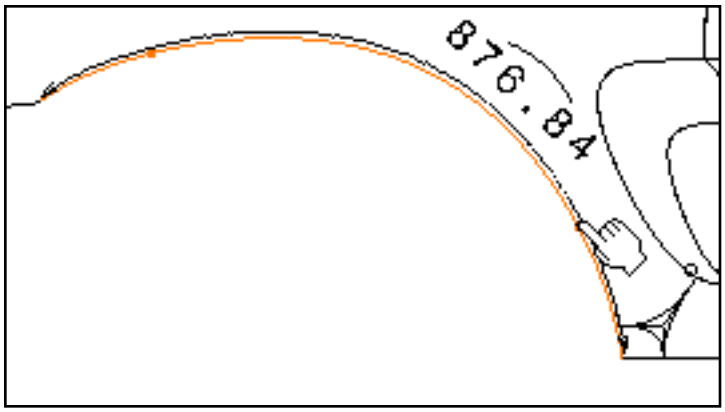
- **Offset** displays the dimension line as an offset of the measured curve.
- **Parallel** displays the dimension line as a translation of the measured curve.
- **Linear** displays the dimension line as linear.

For the purpose of this scenario, select **Parallel**.

5. On the curve, select the existing point that defines the first extremity of the curve portion to dimension.

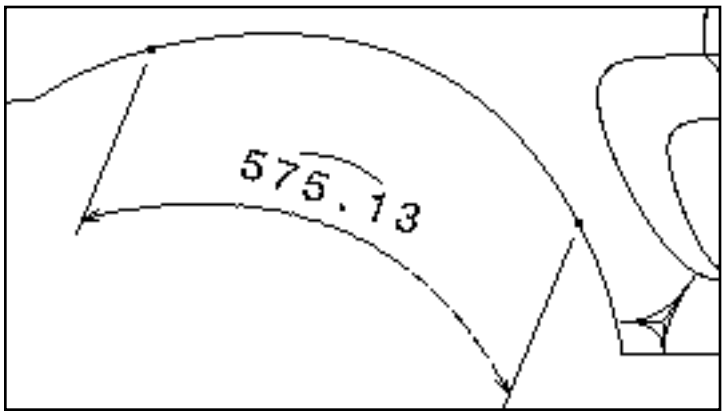


6. Select the point that defines the second extremity of the curve portion to dimension.

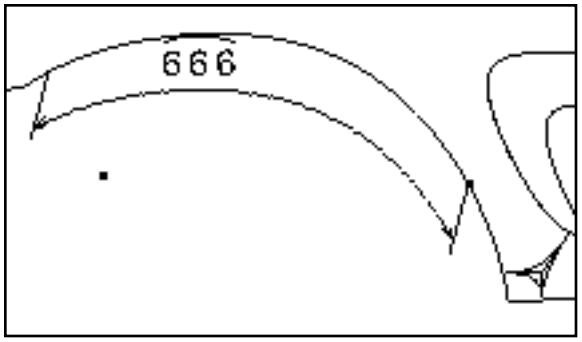


7. Optionally drag the dimension line and/or the dimension value to position them as wanted.

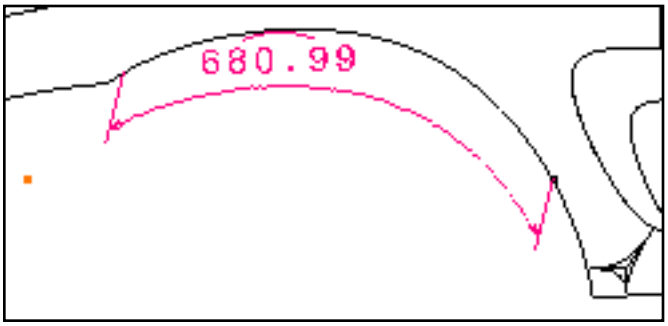
8. Click elsewhere in the drawing to validate the dimension creation. The semi-arc symbol displayed over the dimension value symbolizes a curvilinear length dimension (whether partial or not). You can now handle the dimension just like any other dimension.



9. Move one or both points, on the line or in free space. The dimension is re-computed (if you moved the point in free space, it is re-computed according to the normal projection of the points on the curve.)



If you move a point in such a way that it cannot be projected on the curve anymore, the dimension becomes not-up-to-date.



Creating partial curvilinear length dimensions using points created on-the-fly

Use the CurvilinearDimension.CATDrawing document from the previous scenario, but delete the points and the dimension you created previously.

1. Click the **Dimension** icon from the **Dimensioning** toolbar.

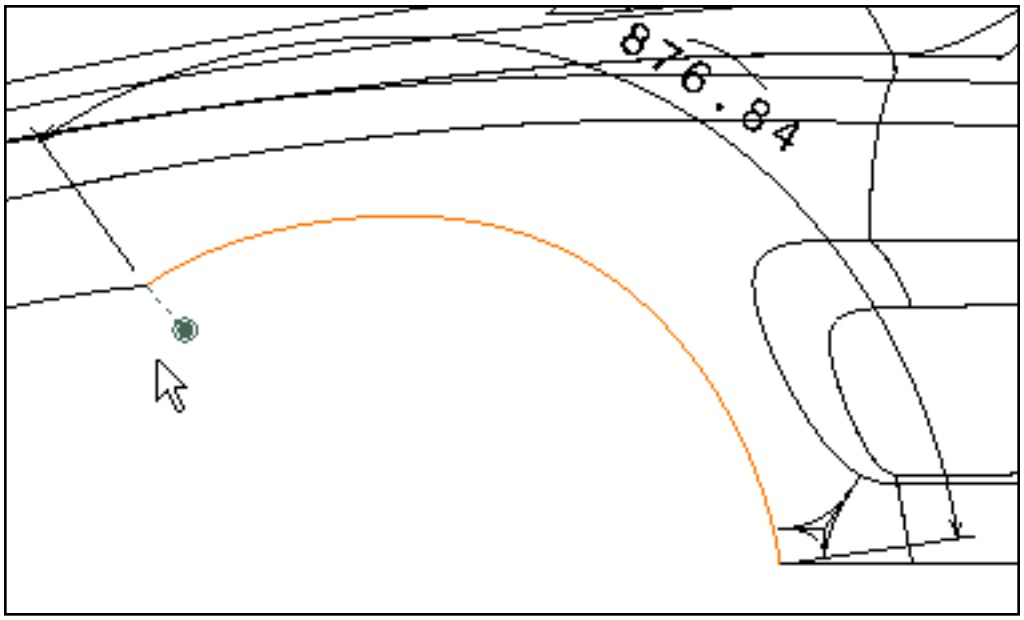
2. Select a curve. A preview of the dimension is displayed.

3. Right-click to display the contextual menu and make sure **Partial Curvilinear Length** is selected.

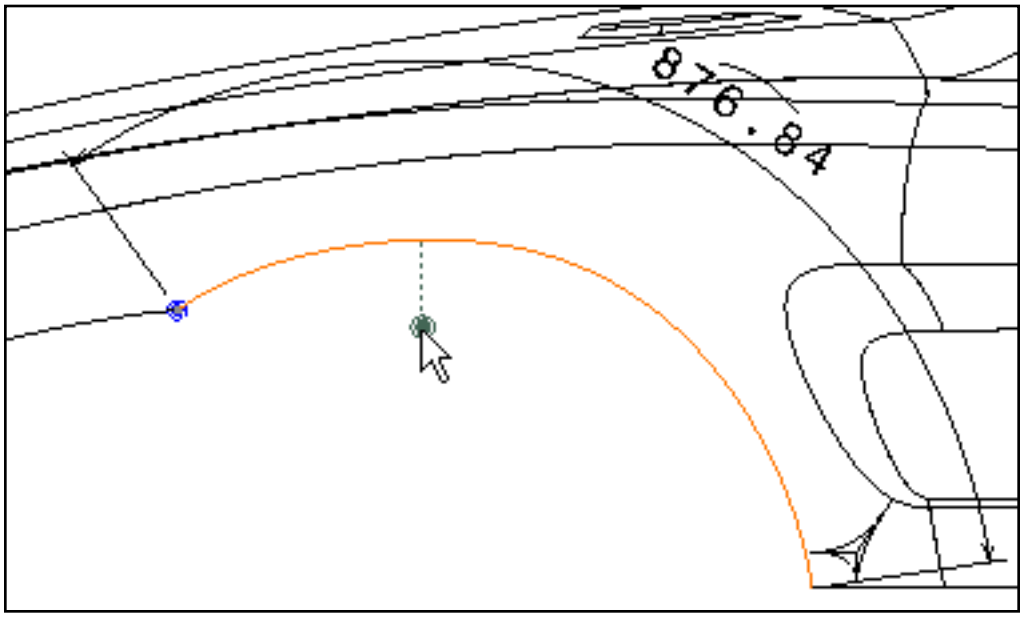
4. Still in the contextual menu, select a representation mode for the dimension line: for the purpose of this scenario, select **Offset**.

5. On the curve, select the point that defines the first extremity of the curve portion to dimension. You can click on the curve, or in the free space.

Note that the indicated point cannot go further than the extremity of the curve itself.

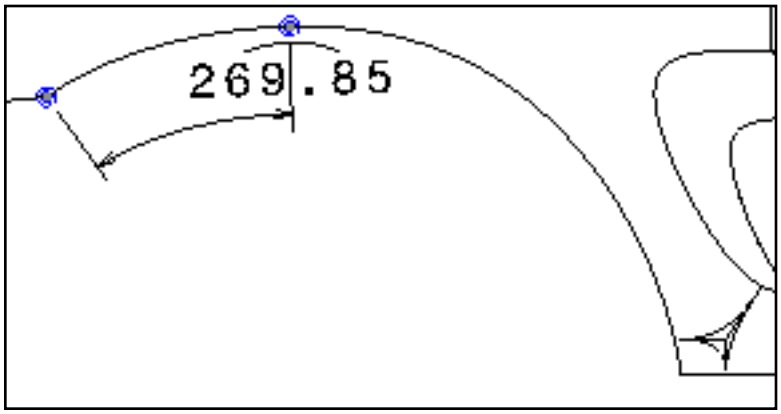


6. Select the point that defines the second extremity of the curve portion to dimension.



Note that two points, as well as two coincidence constraints, have been created on the curve, at the projection point of where you clicked.

7. Optionally drag the dimension line and/or the dimension value to position them as wanted.
8. Click elsewhere in the drawing to validate the dimension creation. The semi-arc symbol displayed over the dimension value symbolizes a curvilinear length dimension (whether partial or not). You can now handle the dimension just like any other dimension.



More About Partial Curvilinear Length Dimensions

General remarks

- Partial curvilinear length dimensions can be created using the **Dimensions** and **Length/Distance Dimensions** commands; they cannot be created using the **Stacked Dimensions** and **Cumulated Dimensions** commands.
- You can create partial curvilinear length dimensions for all types of curves: splines, circles, arcs of circle, conics, etc. Note that in the case of circles and arcs of circle, they will be called *partial circular length* dimensions.
- You can create partial length dimensions for lines.
- The curvilinear length symbol is defined by the administrator in the standards. The same symbol is used for partial curvilinear dimensions and for curvilinear dimensions.
- Partial curvilinear length dimensions cannot be True Length dimensions.
- If you delete a point that defines a dimension, the dimension becomes not-up-to-date, and its color changes to fuchsia by default (or according to the color defined for **Not-up-to-date dimensions** in the **Types and colors of dimensions** dialog box available via **Tools -> Options -> Mechanical Design -> Drafting -> Dimension** tab, **Analysis Display Mode** area, **Types and colors...** button). If you delete both points, the dimension becomes a regular curvilinear dimension.

Restrictions

- You cannot change the dimension line representation mode or orientation after the dimension has been created.

- In the case of the parallel and offset representation modes, the dimension value cannot be moved out of the curve limits, except for circles and arcs of circle. As a result, you cannot specify the dimension value position (Inside, Outside, Auto).
- In some cases, depending on the curve and on the offset value, the offset representation mode cannot be computed.
- In the case of partial curvilinear length dimensions in offset mode, it is recommended to activate the **Constant offset between dimension line and geometry** setting in **Tools -> Options -> Mechanical Design -> Drafting -> Dimension** tab. This will ensure that the dimension remains associative if the geometry is moved.
- Partial curvilinear dimensions cannot be measured along a direction. However, partial length dimensions can be measured along a direction.
- Partial curvilinear dimensions cannot be driving dimensions.
- When creating partial circular length dimensions on circles, you cannot select a circular sector.

Creating Dimensions along a Reference Direction

This task will show you how to create dimensions along a reference direction, i.e. measure the projection of a segment/distance onto a direction. This direction is determined using either a linear element, a fixed angle in the view or a combination of both.

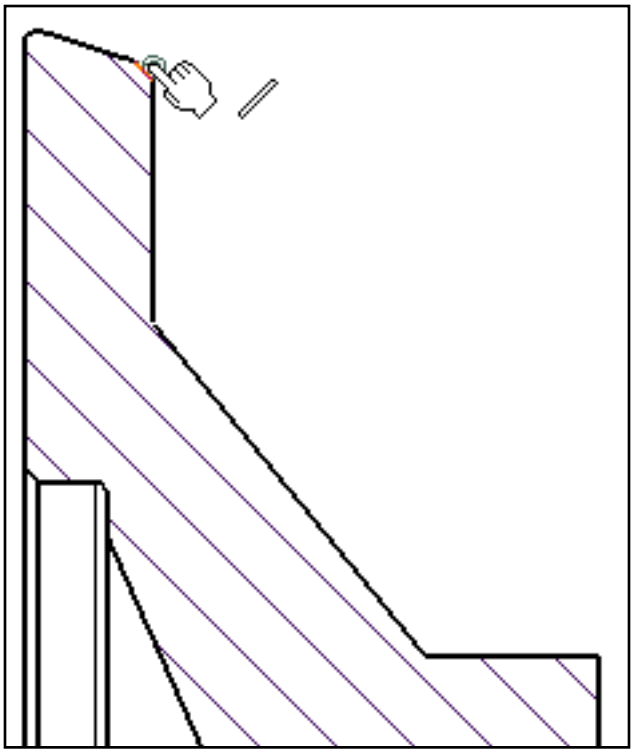
Dimensions along a reference direction can be created for length, distance, diameter tangent, radius tangent, and overall curve dimensions, as well as on linear (i.e. not angular) cumulated or stacked dimensions.

Open the GEAR-REDUCER2.CATDrawing document.

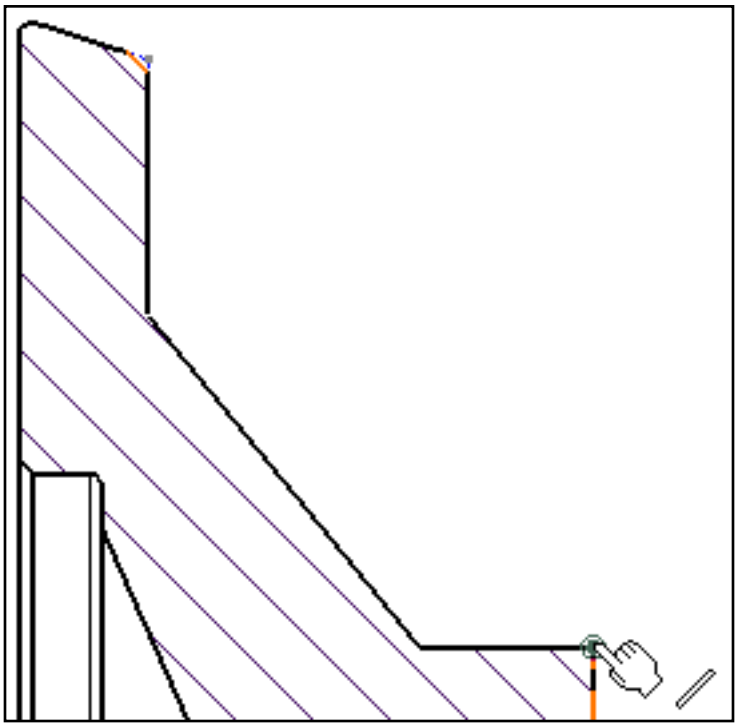
1. Click the **Dimension** icon from the **Dimensioning** toolbar.

2. In the Tools Palette, click the **Intersection Point Detection** icon . Refer to Creating dimensions between intersection points for more information about this functionality.

3. Click the first element, in this case, an intersection point.



4. Click the second element.



The dimension to be created is previewed. In the Tools Palette, click the **Force dimension along a direction** icon:

Several options are then displayed in the Tools Palette:

- **Dimension along a direction** creates the dimension using a linear element (line, axis line, center line) as the reference direction, or using an angle to define the reference direction relatively to a linear element. In the latter case, key in a value in the **Angle** field.

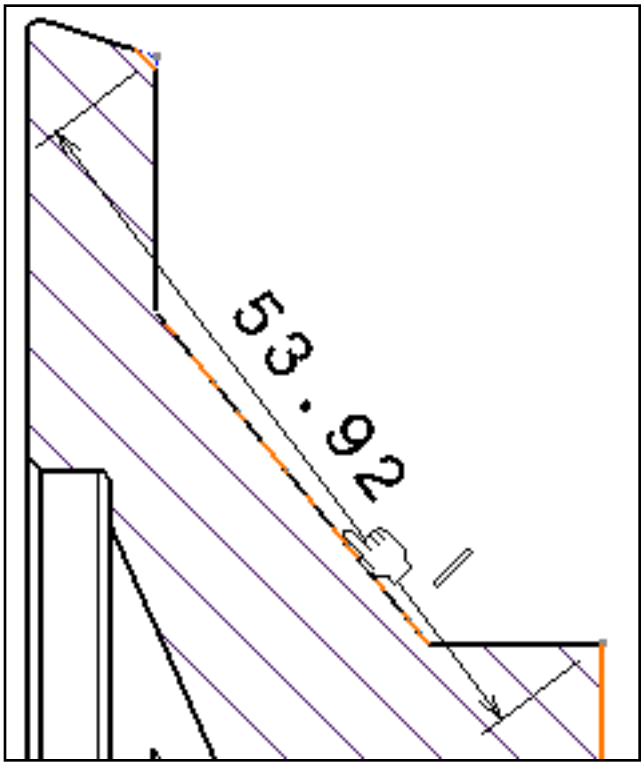
- **Dimension perpendicular to a direction** creates the dimension perpendicularly to a linear element.

- **Dimension along a fixed angle in view** creates the dimension using a fixed angle in the view. In this case, key in a value in the **Angle** field.
  Note that such a dimension follows the view rotation. Thus, a dimension line with a 30 deg angle in a view which is set at 45 deg (relatively to the sheet) will be equivalent to a dimension line with a 75 deg angle relatively to the sheet.

These options are also available in the contextual menu that you can display during the dimension creation.

5. Click the **Dimension along a direction** icon . For the purpose of this scenario, leave the **Angle** field set to 0 deg.

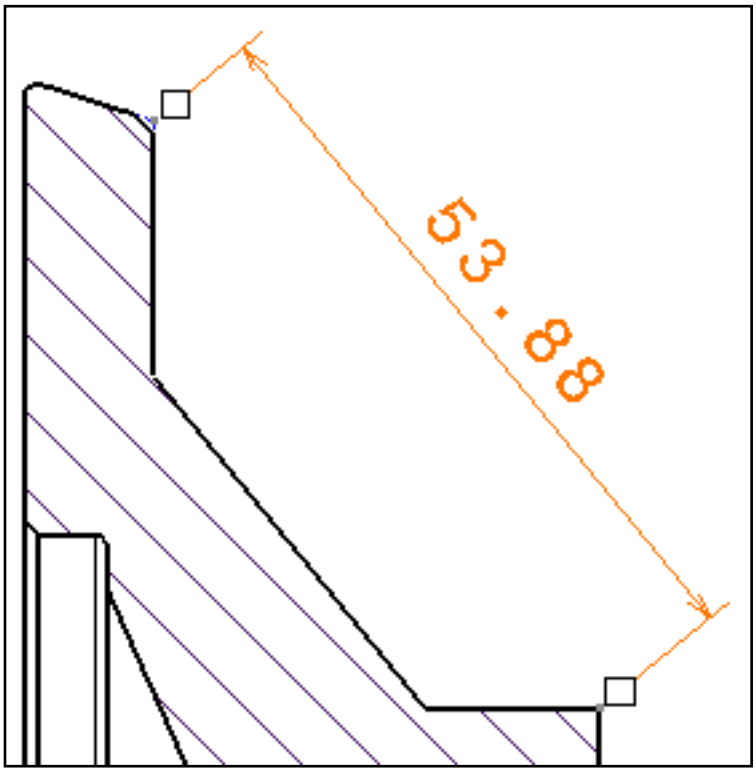
6. Select a linear element to use as the reference direction. Once created, the dimension will be associative to this element.



The dimension is updated so as to measure the distance between the selected points once projected onto the reference direction.

7. Drag the mouse to position the dimension as wanted.

8. Click to validate the dimension creation.



More About Dimensions Along a Reference Direction

- In the case of a dimension along or perpendicular to a direction, if you delete the linear element used as the reference direction, the dimension will be automatically converted into a dimension along a fixed angle in view (the angle being that of the reference element in the view before its deletion).

- The behavior of a dimension along or perpendicular to a direction will actually depend on whether the **Only create non-associative dimensions** option is activated in **Tools > Options > Mechanical Design > Drafting > Dimension** tab, **Associativity on 3D** button:
  - If it is activated, then the dimension will actually be a dimension along a fixed angle in the view (the angle being that of the reference element in the view).
  - If it is not activated, then the dimension will always match the direction of the element defining the reference direction.

- Once a dimension along a reference direction has been created, you cannot modify the elements that define the direction of measure, i.e. either the linear element used as the reference direction or the fixed angle in view.
- The reference direction will not be taken into account when re-routing dimensions (**Re-route Dimension** command).
- Dimensions along a reference direction cannot be driving dimensions. So, if the **Create driving dimension** option is activated in **Tools -> Options -> Mechanical Design -> Drafting -> Dimension** tab, you will not be able to drive dimensions when dimensioning along a direction.
- Dimensions created in a shot (i.e. cumulated/stacked dimensions, or dimensions sharing the same type as the first one) all have the same reference direction.

Creating Dimensions between Intersection Points

This task will show you how to create dimensions between an intersection point and an element or between two intersection points.

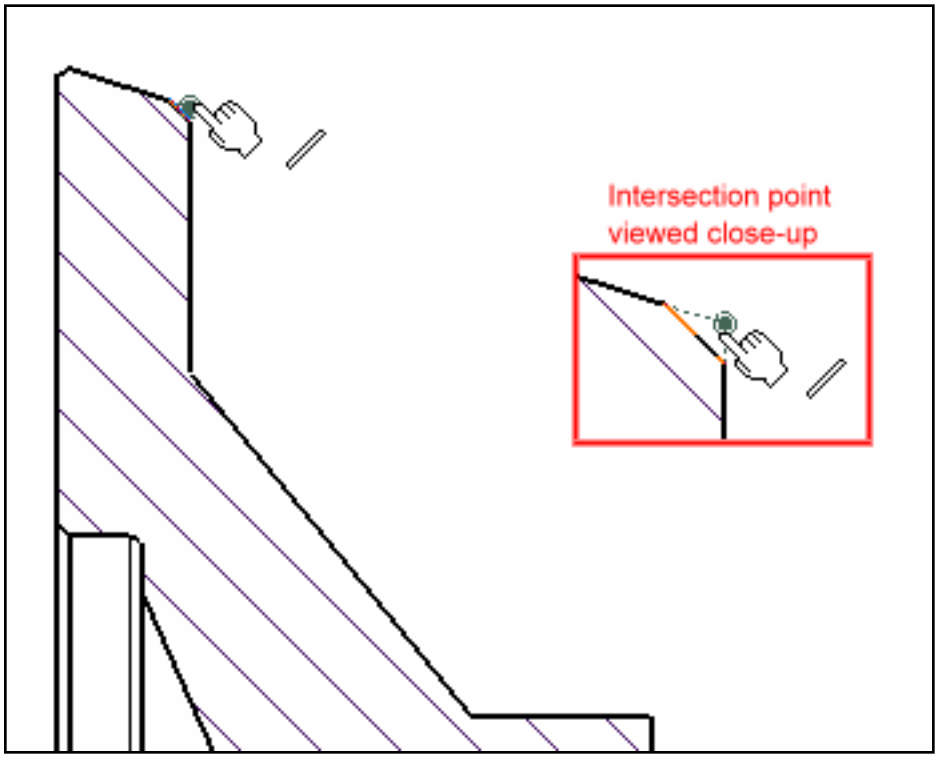
Open the GEAR-REDUCER2.CATDrawing document.

1. Click the **Dimension** icon from the Dimensioning toolbar.

2. In the Tools Palette, click the **Intersection Point Detection** icon .
3. Position the mouse over the first intersection point. An intersection point is the meeting point of:

- 2 extensions lines (as shown in this example)
- 2 lines
- a line and an extension line

A preview of the intersection point is displayed.

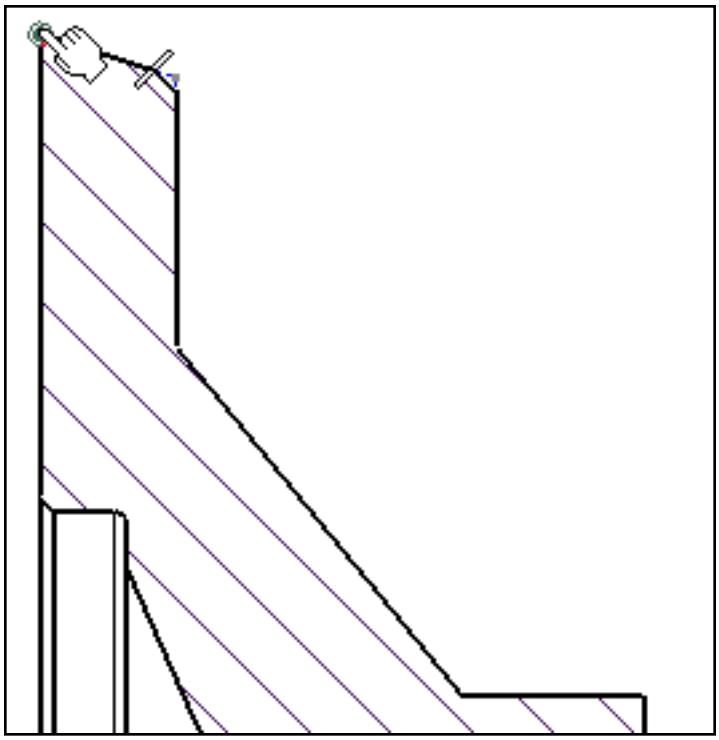


- In the case of drawings with many elements displayed on screen, intersection points may sometimes be difficult to detect. If this happens (i.e. if the intersection point is not previewed or if the previewed intersection point is not the one you want), simply position the mouse over the first and then the second reference element. The proper intersection point will then be previewed.
- In the case of a generative view created with the Approximate generation mode, detection of intersection points is not available. In this case, you need to position the mouse over the first and then the second reference element.

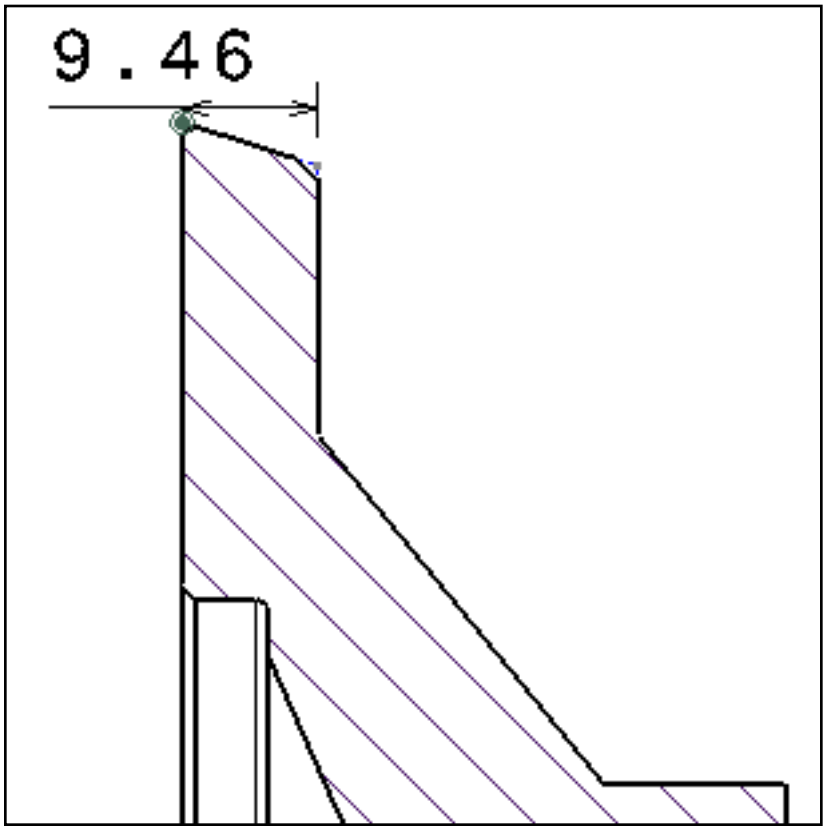
4. Click to create the intersection point. The point is created, as well as construction lines and coincidence constraints between the point and its reference elements.

The display and behavior of intersection points is defined by the administrator in the standards. Indeed, the administrator can specify the style that should be applied to the intersection point and construction line, whether the intersection point can be printed or not, and whether construction lines should be displayed and/or printable.

5. Now, position the mouse over the second intersection point.



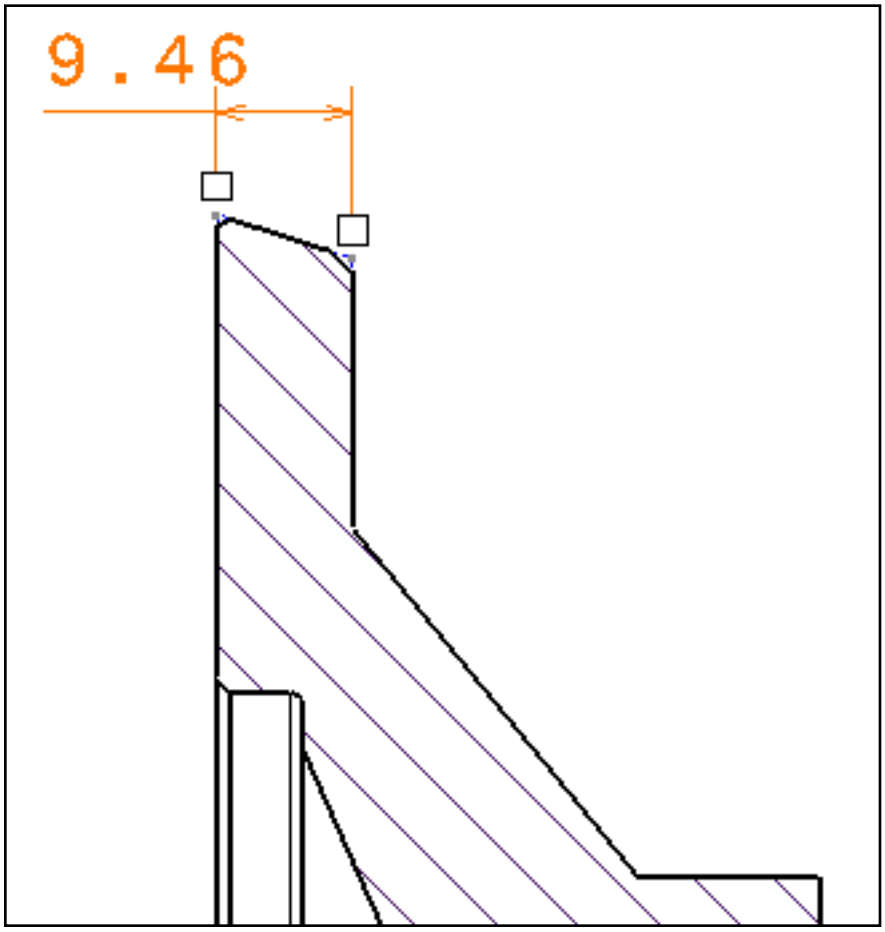
6. Click to create the intersection point. A preview of the dimension is displayed. By default, this dimension is a distance dimension.



At this point, if you want to create a diameter dimension or a radius dimension rather than a distance dimension, you can right-click to display a contextual menu in which you will be able to change the dimension type from the default **Distance** to **Diameter Edge** or **Radius Edge**.

For the purpose of this scenario, leave the default option, **Distance**, selected.

7. Using the mouse, position the dimension as wanted.
8. Click to validate and end the dimension creation.


9.46

Creating Dimensions between an Element and a View Axis

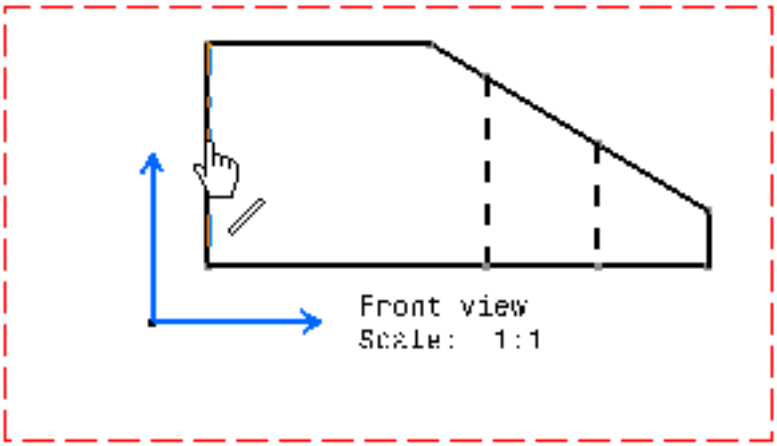
In this task, you will learn how to create dimensions between an element and a view axis (one of the two axes or the origin).

Open the IntDrafting_Viewplane_Front.CATDrawing document.

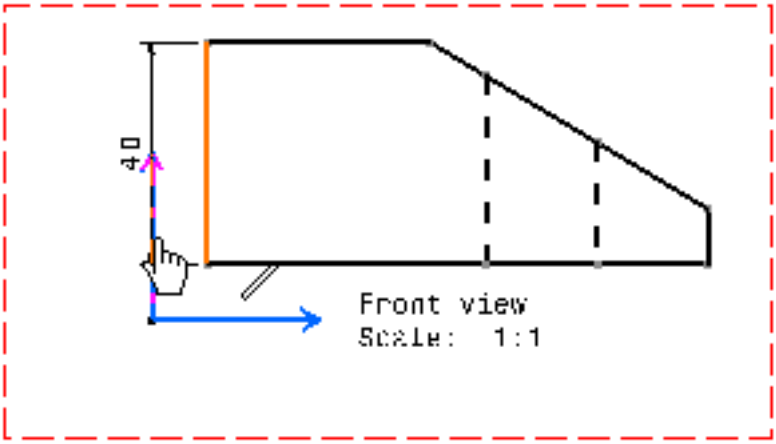
Go to **Tools -> Options -> Mechanical Design -> Drafting -> General** and check **Display in the current view** to display the view axis.

1. Click the **Dimensions** icon from the **Dimensioning** toolbar.

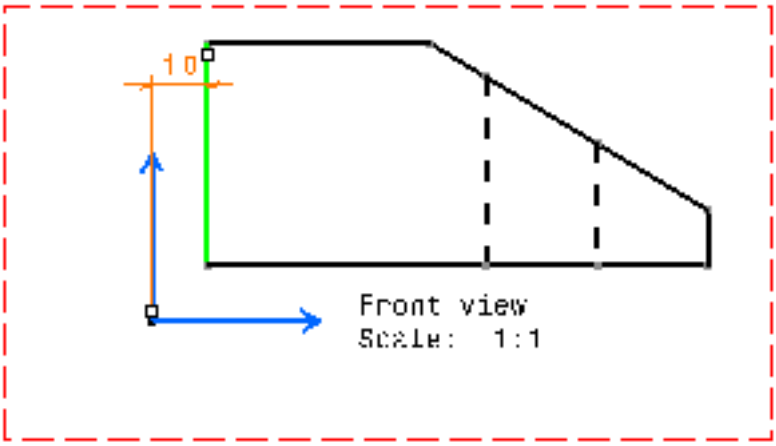
2. Click a first element in the view.



3. Select one of the two view axes or the origin.



4. Click anywhere in the drawing window to confirm the dimension creation.



The dimension is created.

Creating Driving Dimensions

This task shows you how to create dimensions that will drive associated geometry.

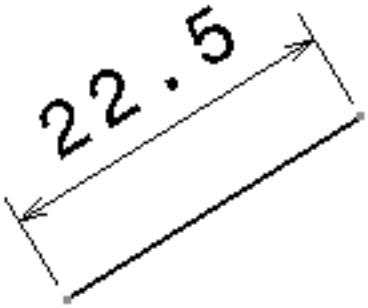
You can create the following types of driving dimensions:

- length
- distance (and distance offset in the case of two concentric circles)
- angle
- radius
- diameter.

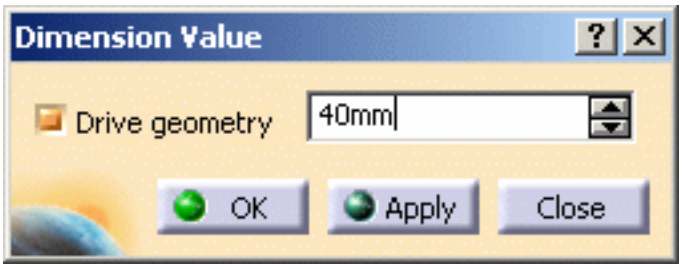
Go to **Tools -> Options -> Mechanical Design -> Drafting -> Dimension** and **select Activate analysis display mode**. Then, click the **Types and colors** button. The Types and colors dialog box is displayed.

Make sure the **Dimensions driving 2D geometry** check box is selected, and identify the color that will be assigned to driving dimensions (you can change it if you want).

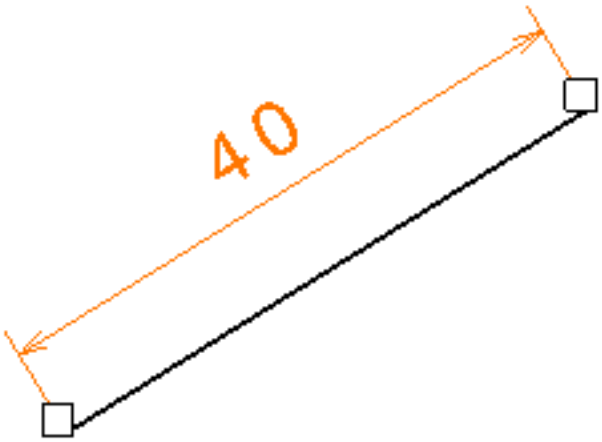
Create a line. Click the **Dimensions** icon from the **Dimensioning** toolbar and create a length dimension on this line.



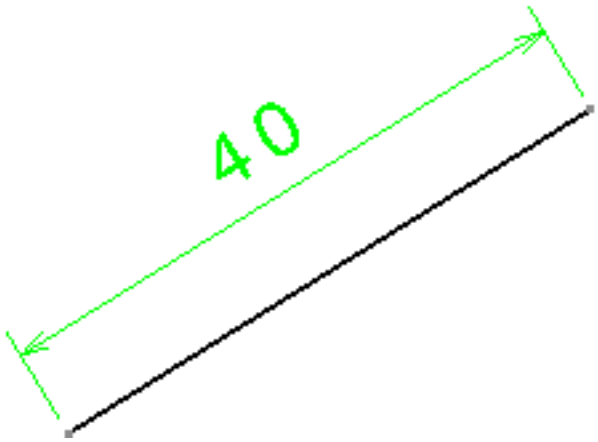
1. Double-click the dimension. The Dimension value dialog box is displayed.
2. Make sure the **Drive geometry** check box is selected. This dimension will now drive the geometry.
3. Modify the dimension value, entering 40 millimeter as the new length.



4. Click **OK** to validate and exit the dialog box. The geometry is updated according to the new driving dimension value.



5. Click elsewhere in the drawing to deselect the dimension. You can see that the driving dimension is assigned the colors defined in the Types and colors dialog box.

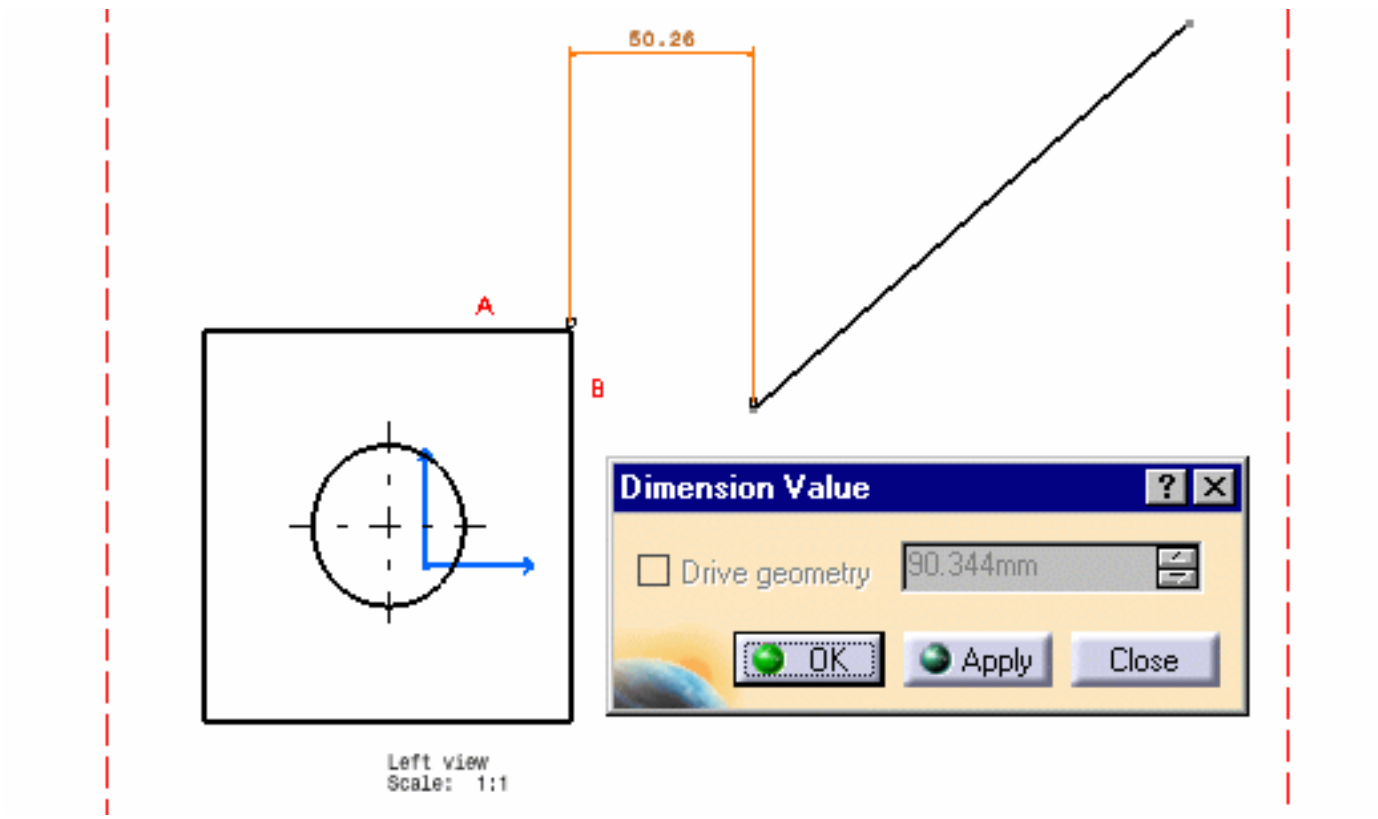


More about driving dimensions

Once the **Drive geometry** check box is selected, you can access a contextual menu and customize the values properties according to your needs. For more information on the available options, refer to CATIA Knowledgeware Infrastructure - Tips and Techniques - Summary, available from the Using Knowledgeware Capabilities section in the *Infrastructure User's Guide*.

You cannot create driving dimensions between the following types of elements (in this case, the **Drive geometry** option is deactivated when double-clicking the dimension):

- Between an interactive element and a generated circle center.
  To bypass this problem, create a point that is concentric with the center of the circle and create the dimension between this new point and the other element.
- Between an interactive circle and a generated circle.
- Between an interactive line and a generated one that are not parallel.



To bypass this problem, create a point that will be coincident with line A and line B at the same time and create the dimension between this new point and the other element.

- Between two fixed elements, that is to say:
  - generated elements
  - axis lines
  - center lines
  - 2D components
  - interactive elements that are fixed by constraints or driving dimensions.

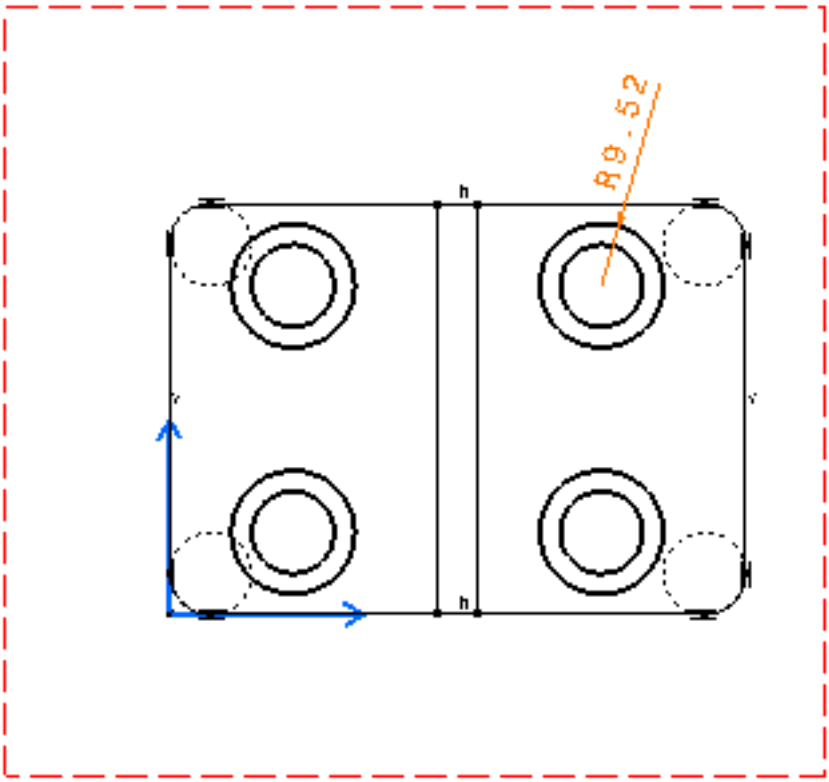
Modifying the Dimension Type

This task will show you how to modify the dimension type as you create a dimension. In other words, you modify the dimension attributes. In this particular example, we will apply a Radius Center dimension type to a hole.

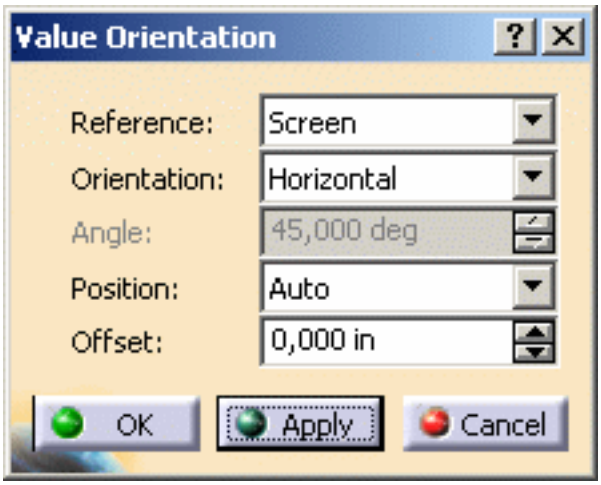
Open the Brackets_views02.CATDrawing document.

1. Click the **Dimensions** icon and then select a hole, for example. Make sure you do not click in the drawing or on the dimension, as this would validate the creation.
2. Right-click the dimension.
3. Select the required dimension type from the displayed contextual menu. For example, **Radius Center**.

   The diameter dimension is automatically turned into a radius dimension.
4. Click in the drawing to validate the dimension creation. If needed, you can modify the dimension location.



When you display the contextual menu during the dimension creation, you can define the value orientation with the **Screen**, **View** or **Dimension line** as reference, or still **Horizontal**, **Vertical** or according to a **Fixed angle**. These options are available in the **Value Orientation** dialog box.



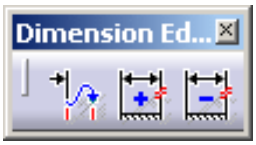
Re-routing Dimensions

This task will show you how to re-route dimensions, i.e. to recalculate dimensions taking into account new geometry elements which are compatible with the re-routed dimension type.

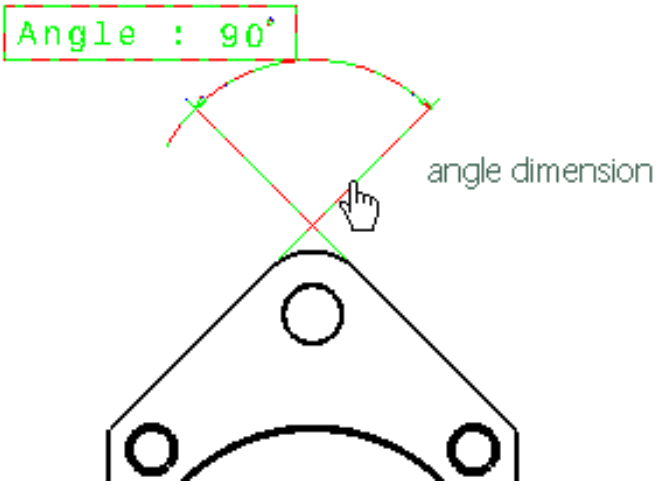
Re-routing dimensions can be particularly useful in the case of isolated dimensions resulting from V4 to V5 migration. Indeed, re-routing isolated dimensions to the geometry enables you make them associative.

Open the Reroute_Dimensions.CATDrawing document. You can notice that the dimension properties are customized.

1. Select the **Re-route Dimension** icon from the **Dimensioning** toolbar (**Dimension Edition** sub-toolbar).

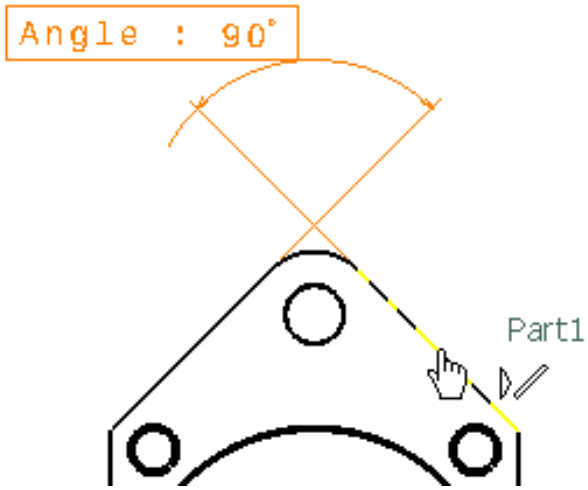


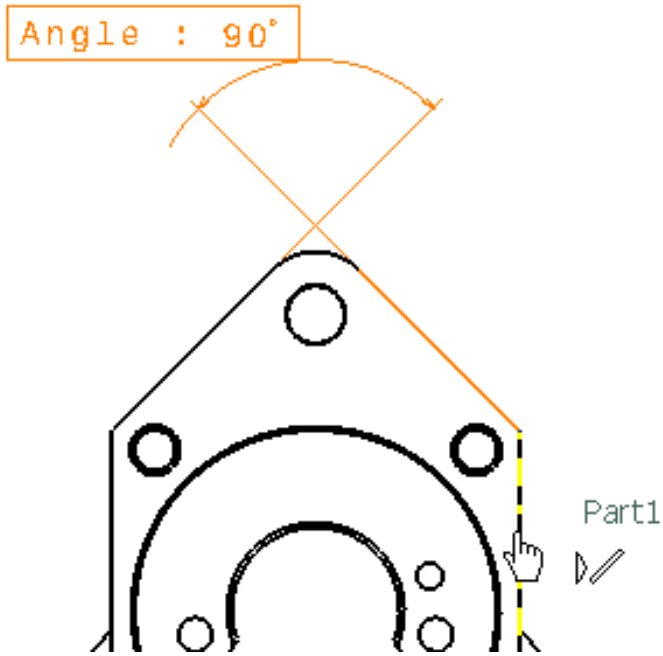
2. Select the angle dimension. You can notice that the cursor indicates the type of dimension you are selecting.



3. Select the first element you want to take into account for the dimension re-routing, and then the second element.

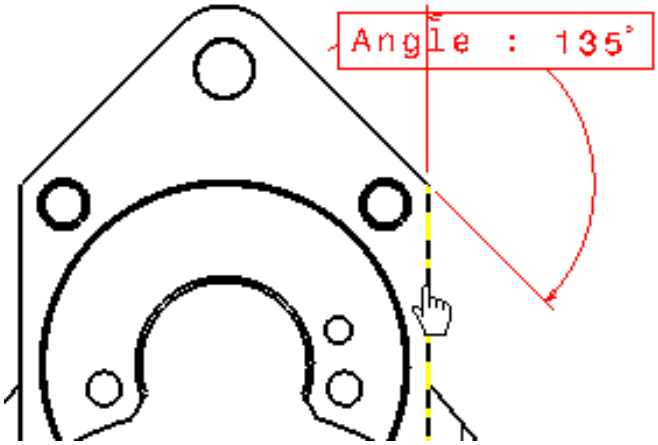
Select the first element. *Then, select the second element.*



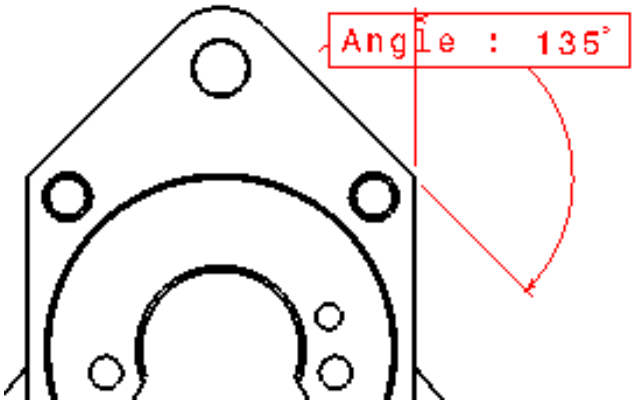


During this operation, the cursor gives a graphic preview of what type of element you are selecting (in this case, lines).

A preview of the re-routed angle dimension is displayed.



4. Click to validate the dimension creation.



5. You can proceed in the same manner to re-route the other dimension types available on the drawing.

- Always make sure that the element(s) to which you are re-routing dimensions are compatible with the re-routed dimension type. For example, when re-routing a radius dimension, you need to select a curved element.
- In a Generative Drafting context, you cannot re-route dimensions generated via the Generate Dimensions command.
- Re-routing dimensions preserves dimension properties when you customized them.
- When re-routing chamfer dimensions which have only two reference elements (the element to be dimensioned and a single reference line or surface), you will need to select three reference elements.
- When re-routing a dimension from one circle to another, note that:

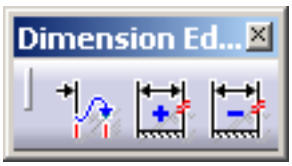
A dimension associated to the circle's...	will always be re-routed to the other circle's...	and cannot be re-routed to the other circle's...
edge	edge	center
center	center	edge

Interrupting Extension Lines

This task will show you how to interrupt manually one or more extension lines of one or more dimensions.

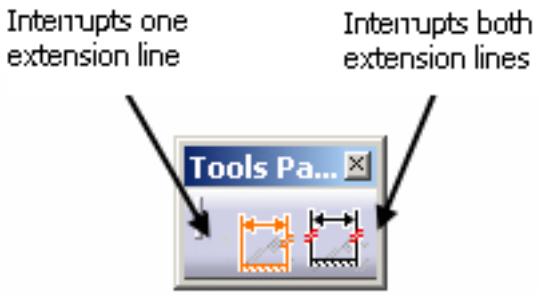
Open the Interruption_ExtLine01.CATDrawing document.

1. You have several possibilities:
   - Right-click a dimension and select the **Dimension.1 Object -> Create Interruption(s)** option from the contextual menu.
   - Select a dimension and click **Insert -> Dimensioning -> Dimension Edition -> Create Interruption(s)** from the menu bar.
   - Select a dimension and click on the **Create Interruption(s)** icon in the **Dimensioning** toolbar (**Dimension Edition** sub-toolbar).



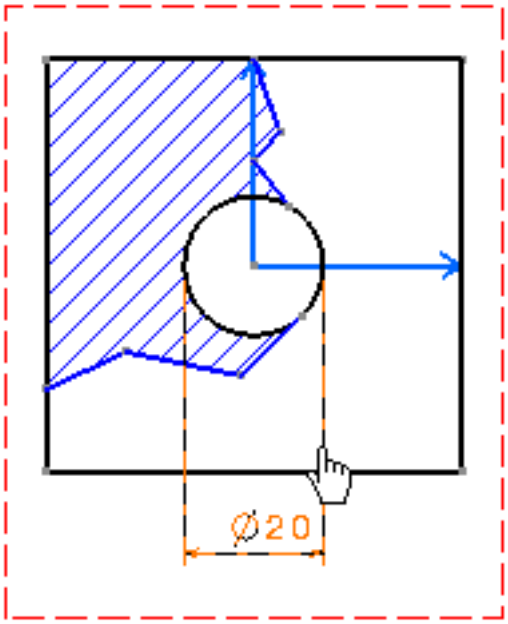
   - You can also select the interruption command first, and then the dimension.
   - You can multi-select several dimensions either using the Ctrl key or the mouse.

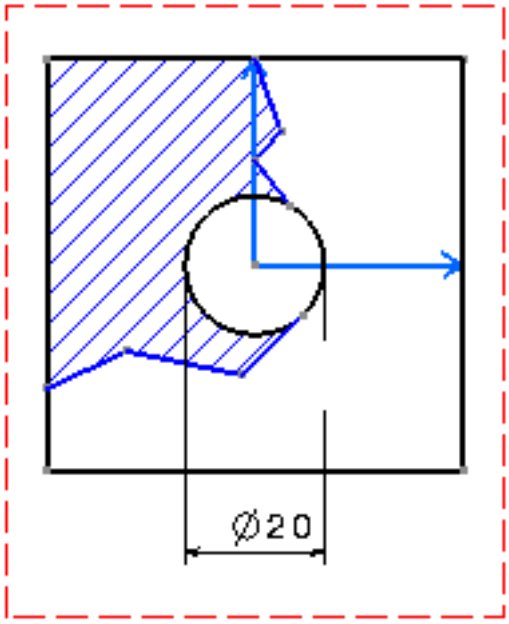
2. In the Tools Palette, indicate if you want to create the interruption on one extension line or on both extension lines.



3. Click to indicate the first point defining the interruption to be created.

4. Click to indicate the second point defining the interruption to be created.





If you have chosen to create the interruption on one extension line, the interruption is automatically created on the extension line which is closest to where you click.

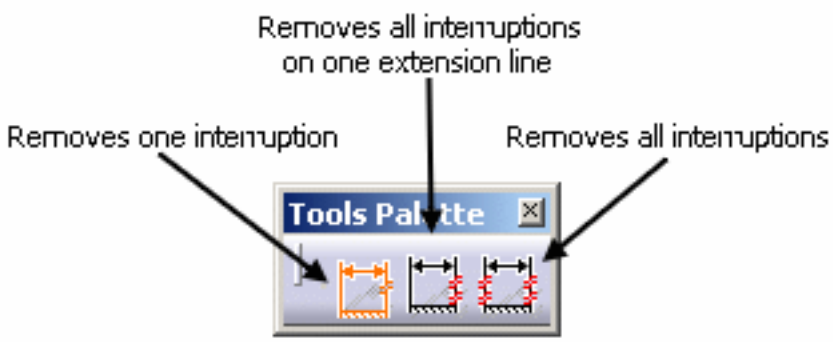
5. To remove the interruption you created, you have several possibilities:

- Right-click the dimension and select **Dimension.1 Object -> Remove Interruption(s)** from the contextual menu.

- Select the dimension and click **Insert -> Dimensioning -> Dimension Edition -> Remove Interruption(s)** from the menu bar.

- Select the dimension and click on the **Remove Interruption(s)** icon in the Dimensioning toolbar (Dimension Edition sub-toolbar).

6. In the Tools Palette, indicate if you want to remove a single interruption on an extension line, all interruptions on an extension line, or all interruptions on both extension lines. In this case, leave the **Remove One Interruption** icon selected.



7. Click to indicate the extension line from which you want to remove the interruption. The interruption is removed from the extension line which is closest to where you click.

- When creating or removing interruptions, you can select the dimension either before or after selecting the appropriate command.
- If you move the dimension, the interruption will remain as you created it.
- If you modify either the overrun and / or the blanking, the interruption also remains the same.
- You can perform interruptions on dimension systems, both on single dimensions of a system and the whole system.
  However, for stacked and cumulated dimension systems, the reference line cannot be interrupted.

- You can apply a maximum of eight interruptions to an extension line.
- Extension lines with funnels cannot be interrupted. Likewise, you cannot add funnels to extension lines with interruptions.

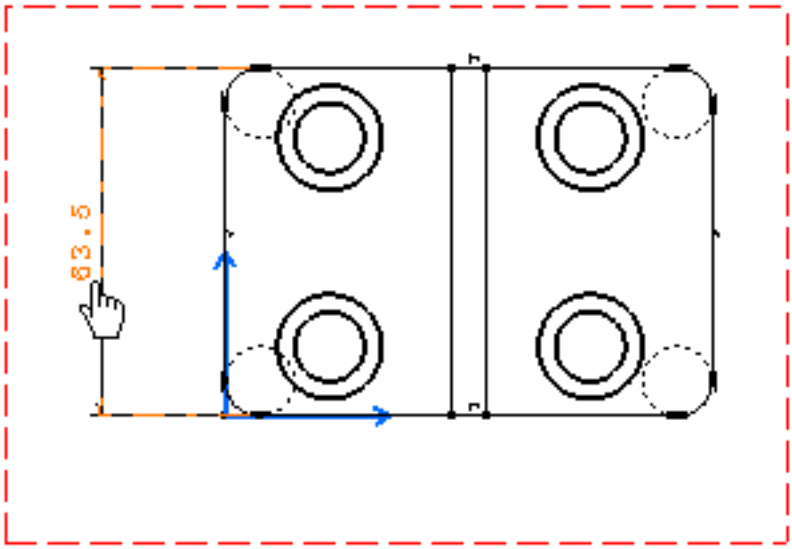
Modifying the Dimension Line Location

This task will show you how to modify dimension line location either as you create or after creating dimensions.

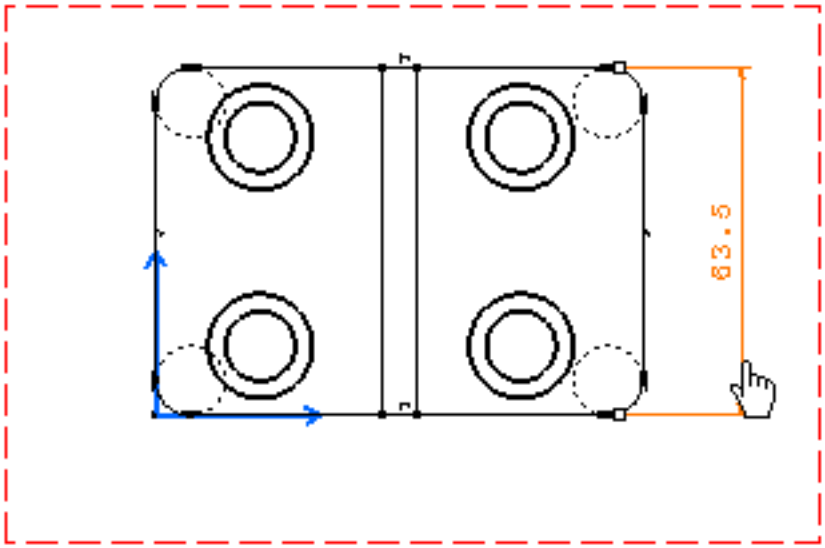
Open the Brackets_views02.CATDrawing document. Create a distance dimension, for example.

1. Click the Select icon , if needed.
2. Select the dimension to be modified. For example, a distance dimension.

   The distance dimension is highlighted.



3. Select the dimension line.
4. Drag the dimension line to the new position.



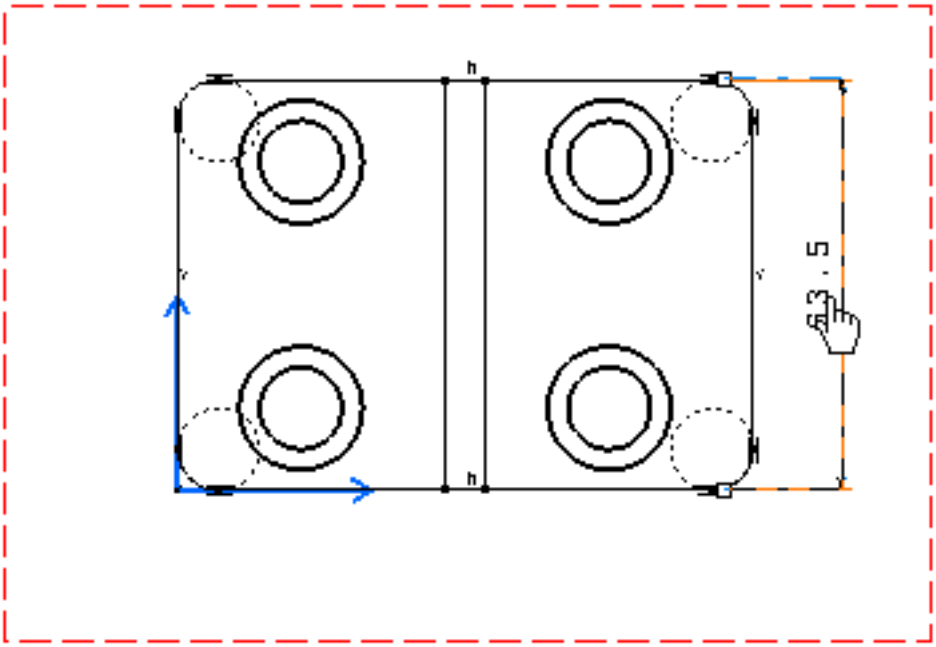
- You can also modify the dimension line location using the extension line.
- As a useful help, you can press the Shift key to temporarily activate/deactivate snapping (depending on whether the **Snap by default** check box is selected in **Tools > Options > Mechanical Design > Drafting > Annotation and Dress-up** tab).

Modifying the Dimension Value Text Position

This task will show you how to modify the position of the dimension value text using the mouse.

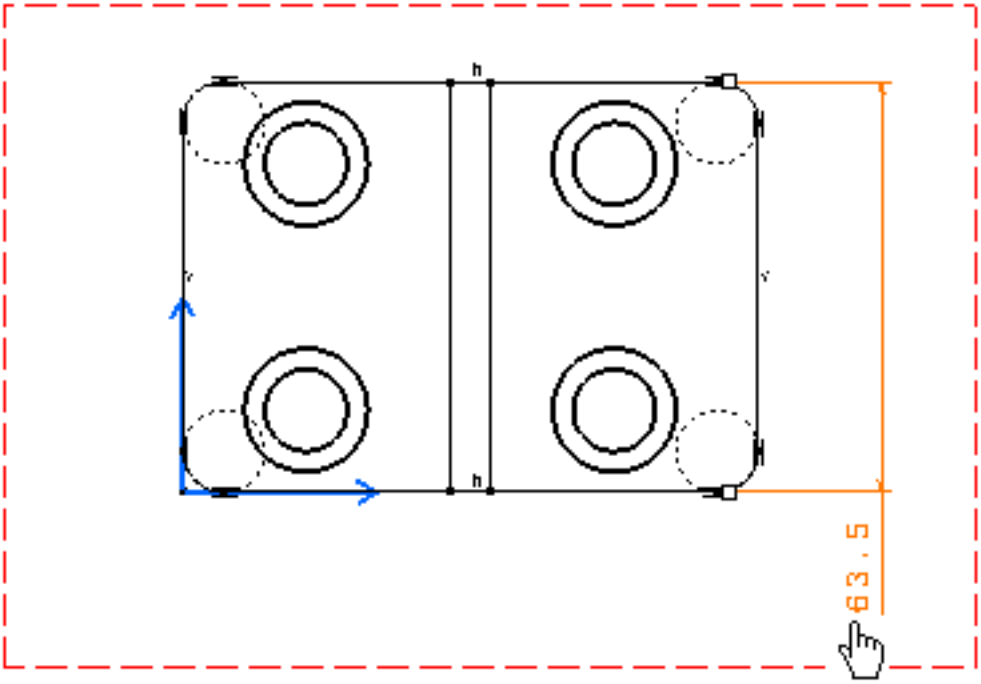
Open the Brackets_views02.CATDrawing document. Create a distance dimension, for example.

1. Click the Select icon , if needed.
2. Select the dimension value text.



3. Drag the value text to the new position.
4. Click to validate the position.

   Note that as a useful help, you can press the Shift key to use Snapping temporarily (as long as you keep the button pressed).



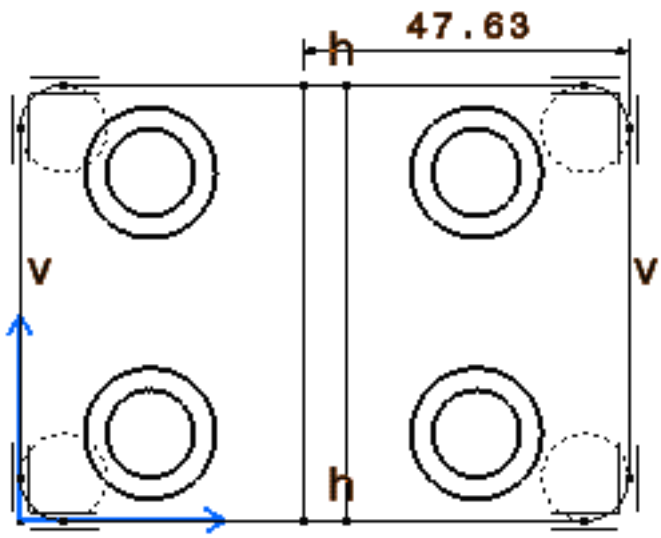
At any time, you can restore the original value text position. To do this, right-click the dimension you positioned and select **Restore Value Position** from the contextual menu.

Specifying the Dimension Value Position

In this task, you will learn how to place automatically the value outside the area between extremity symbols when this area is too small, or how to explicitly position the value inside or outside the area between extremity symbols.

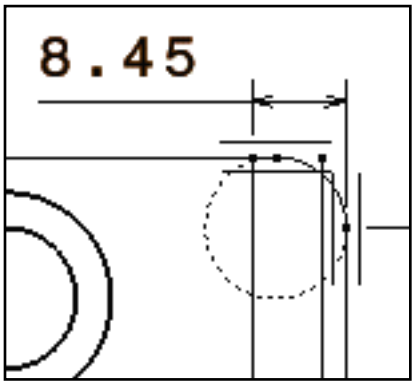
Open the Brackets_views02.CATDrawing document. Create a distance dimension, for example.

1. Click the **Select** icon , if needed.
2. Right-click the dimension to be modified.



3. In the contextual menu, select **Properties**. The **Properties** dialog box is displayed.
4. Click on the **Value** tab.
5. In the **Value Orientation** area, there are three options in the **Position** field:
   - **Auto**: positions the value inside the area between extremity symbols whenever this is possible; otherwise, positions it outside.
   - **Inside**: positions the value inside the area between extremity symbols.
   - **Outside**: positions the value outside the area between extremity symbols.
6. Select **Auto**.

   If you change the dimension from now on, and the value does not fit inside the area between extremity symbols, the value will be automatically positioned outside. Try it by reducing the dimension as shown in our example.



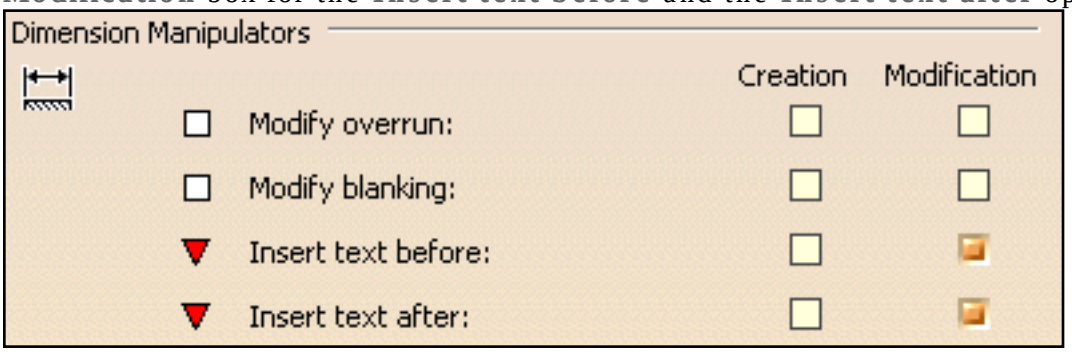
- The **Auto** position of the dimension value will be disabled if you modify the position of the dimension value text using the mouse (i.e. if you manually move it). You can restore the original position of the dimension value by right-clicking the dimension and selecting **Restore Value Position** from the contextual menu.
- If you switch between **Auto**, **Inside**, and **Outside**, make sure the dimension value is properly positioned by restoring the original position of the dimension value (use the **Restore Value Position** option from the contextual menu).

Adding Text Before/After the Dimension Value

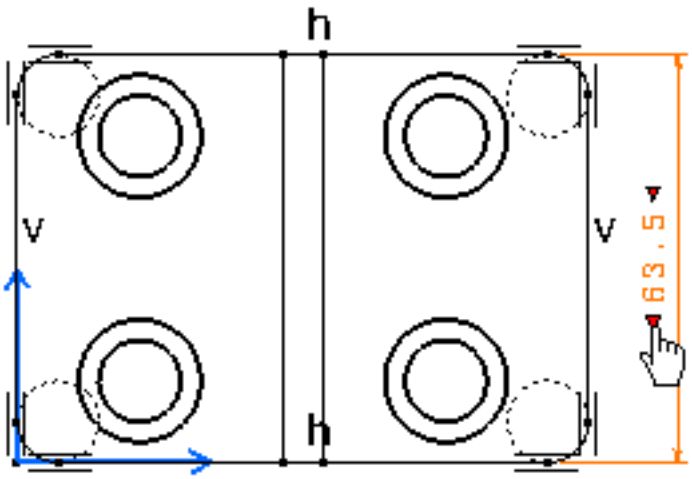
This task will show you how to insert text before or after the dimension value.

Open the Brackets_views02.CATDrawing document. Create a distance dimension, for example.

Go to **Tools -> Options -> Mechanical Design -> Drafting -> Manipulators** tab, and check the **Modification** box for the **Insert text before** and the **Insert text after** options.

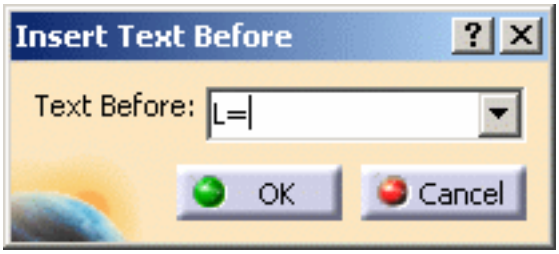


1. Click the dimension to be modified. The dimension is highlighted and two manipulators appear, both before and after the dimension value.
2. Click the manipulator before the dimension value, for example.

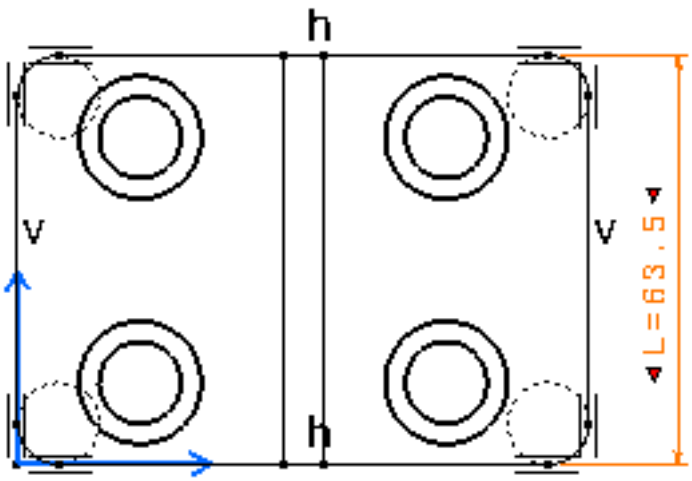


The **Insert Text Before** dialog box is displayed.

3. Enter the text that you want to add before the dimension value, L= for instance.



4. Click **OK**. The text is automatically inserted before the dimension value.



Note that any created Text Before is automatically added to the drop-down list in the dialog box and can therefore be selected again from this list.

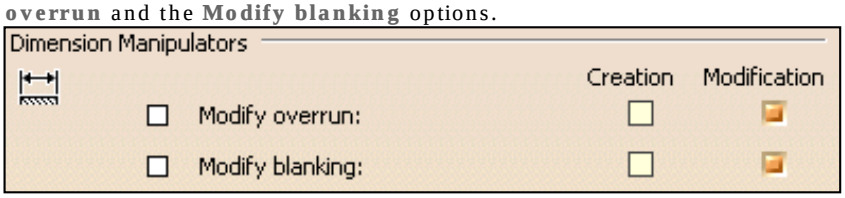
5. Click in the free space.

Modifying the Dimension Overrun/Blanking

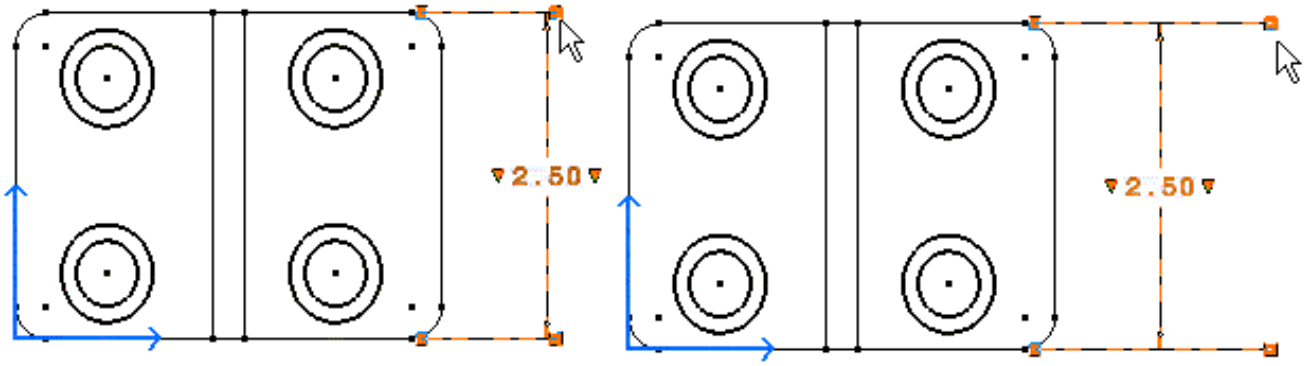
This task shows how to modify dimensions extension line overrun and/or blanking either together or separately.

Open the Brackets_views02.CATDrawing document. Create a distance dimension, for example.

Go to **Tools -> Options -> Mechanical Design -> Drafting -> Manipulators** tab, and check the **Modification** box for the **Modify overrun** and the **Modify blanking** options.

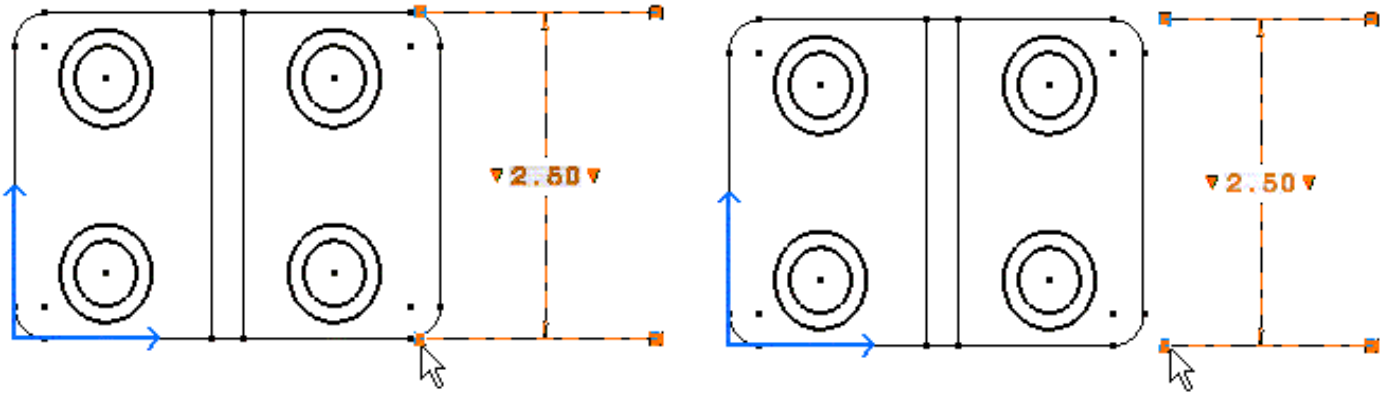


1. Drag the overrun manipulator(s) to a new position.



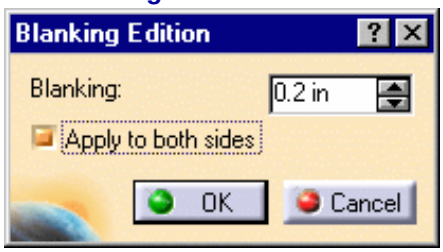
If you want to modify one extension line only, press the **Ctrl** key and drag the desired manipulator.

2. Drag the blanking manipulator(s) to a new position.



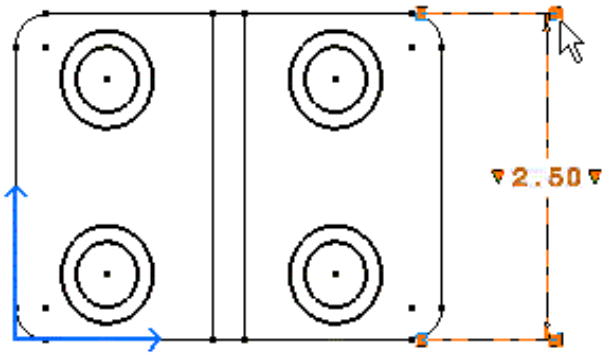
3. If you need to be more precise, double-click the manipulator. The **Blanking Edition** dialog box is displayed.

4. Enter the desired value to modify the blanking.



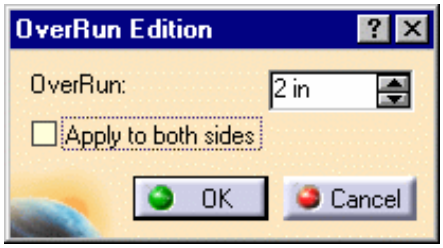
You can also modify the overrun/blanking of only one extension line of the dimension.

5. Double-click the overrun manipulator(s).

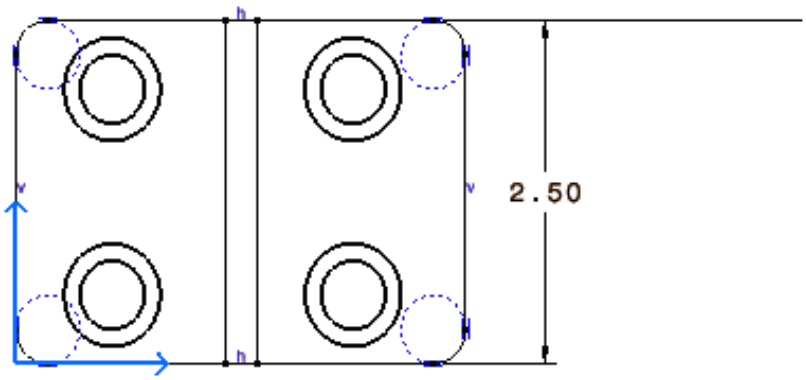


The **Overrun Edition** dialog box appears.

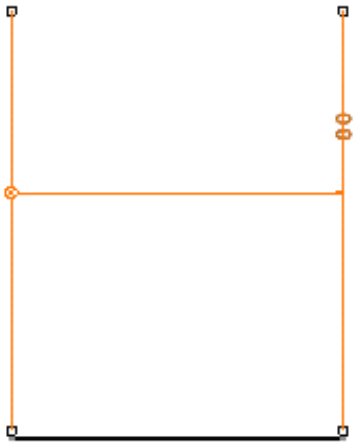
6. Enter the desired overrun value and un-check the **Apply to both sides** option from the **Overrun Edition** dialog box.



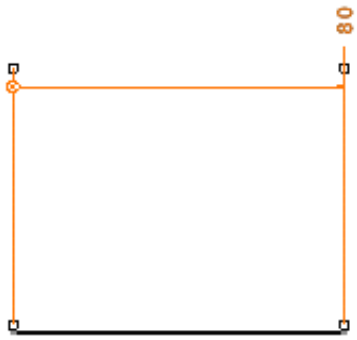
The overrun is applied to one side only.



- Note that you can also right-click the dimension and select the **Edit -> Properties** option from the displayed contextual menu. The **Properties** dialog box appears. Select the **Extension Line** tab and modify the desired value(s) of the **Overrun / Blanking Extremities** option(s).
- To set Cumulate dimension extension line length and text position, customize the Cumulate Dimension parameters in the standards.
- **Overrun** is the overrun minimum value. As an example, for a cumulated dimension (for ISO Standard):



You can increase the overrun size.



You cannot decrease it below the minimum value.

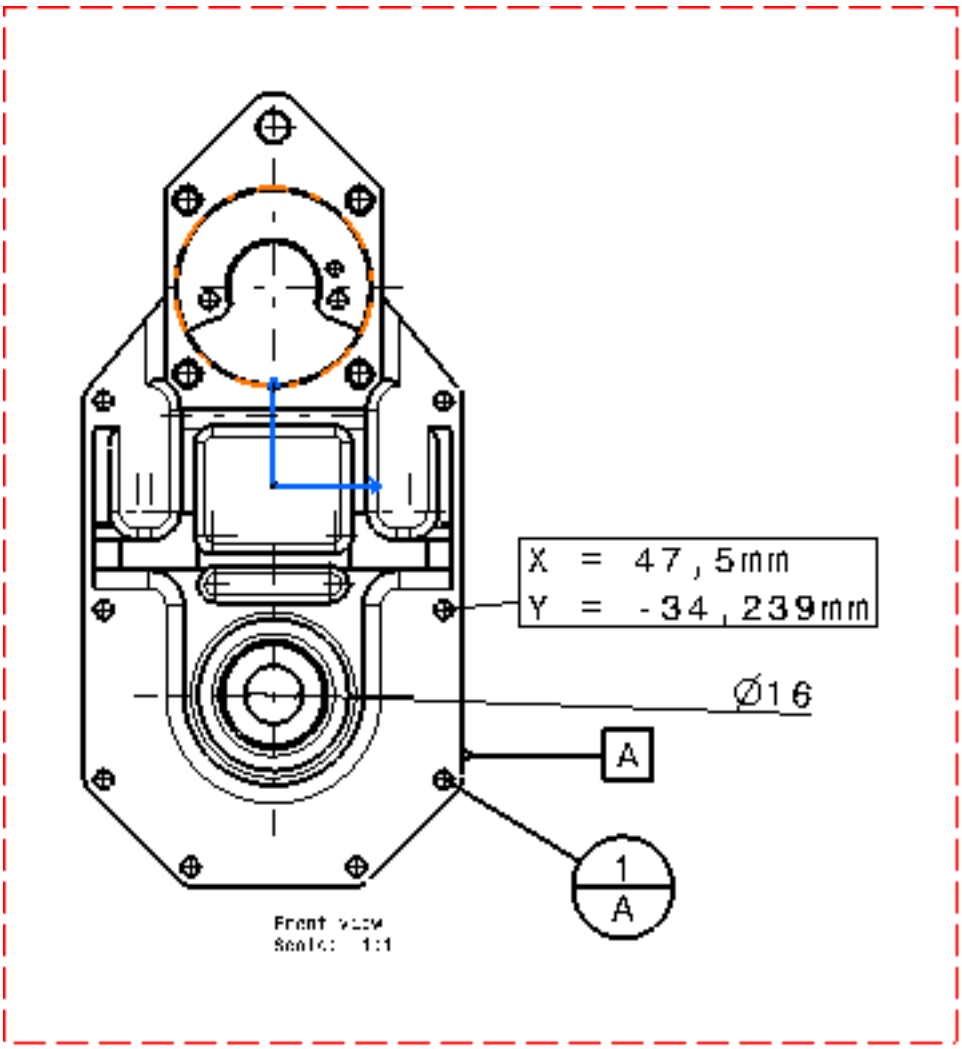
Scaling a Dimension

This task will show you how to scale a dimension in a view, a 2D component, a 2D component instance or a 2D component from a catalog.

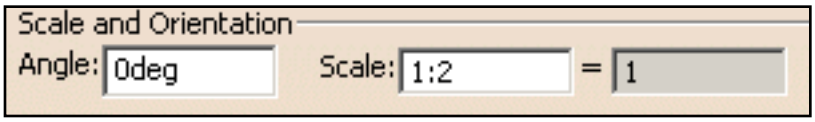
Open the GenDrafting_front_view.CATDrawing document.

In a View

1. Create a diameter and a coordinate dimension, as well as a datum feature and a datum target.
2. Multi-select the dimensions and annotations, then set the font size to 7,5mm.

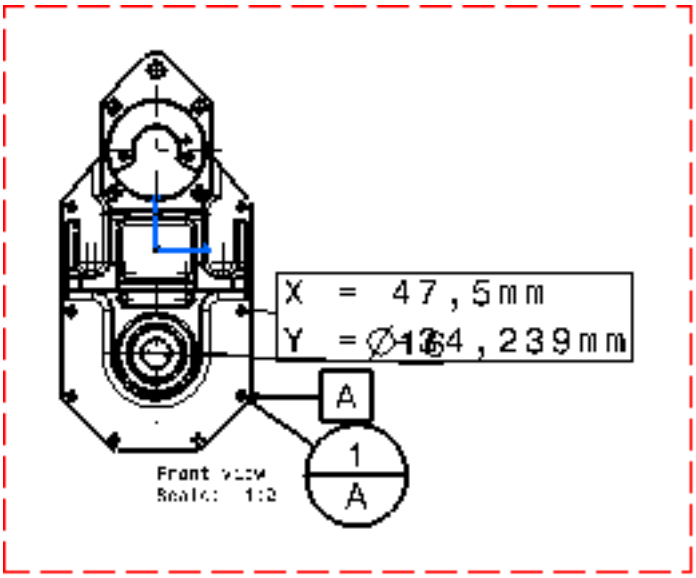


3. Right-click on the frame of the front view.
4. Select **Properties** in the contextual menu.
5. Enter 1:2 as new scale.



6. Click on OK.

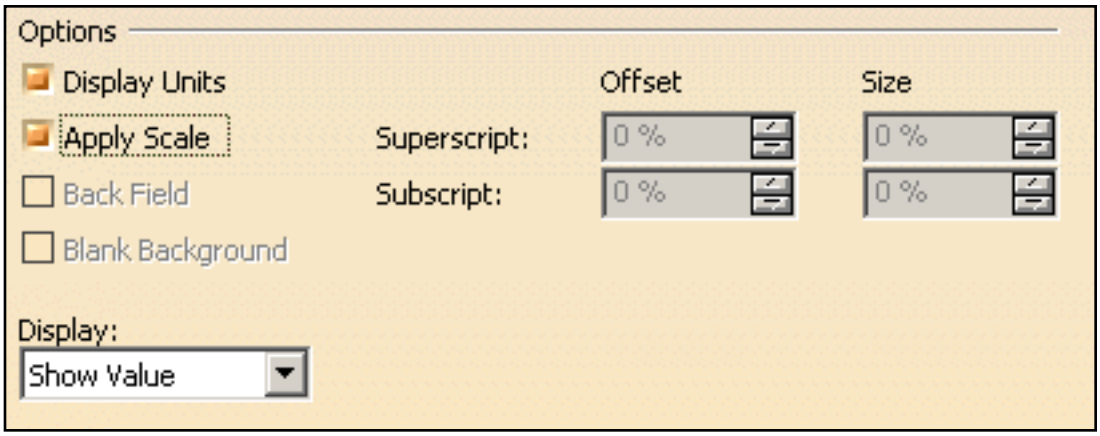
The view is updated but the dimensions and annotations are not scaled.



7. Multi-select the dimensions and annotations again.

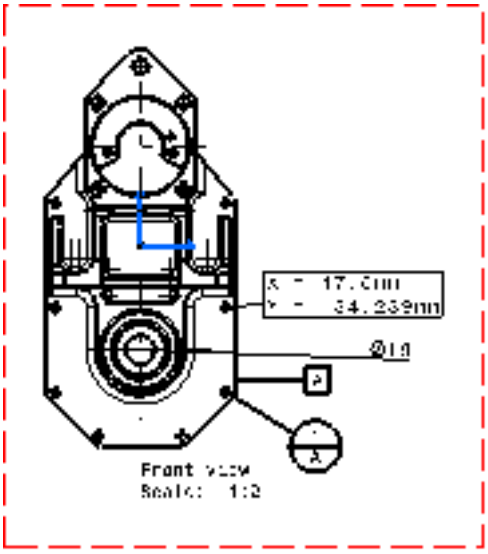
8. Then right-click and select **Properties** in the contextual menu.

9. In the Text tab, check **Apply Scale**.



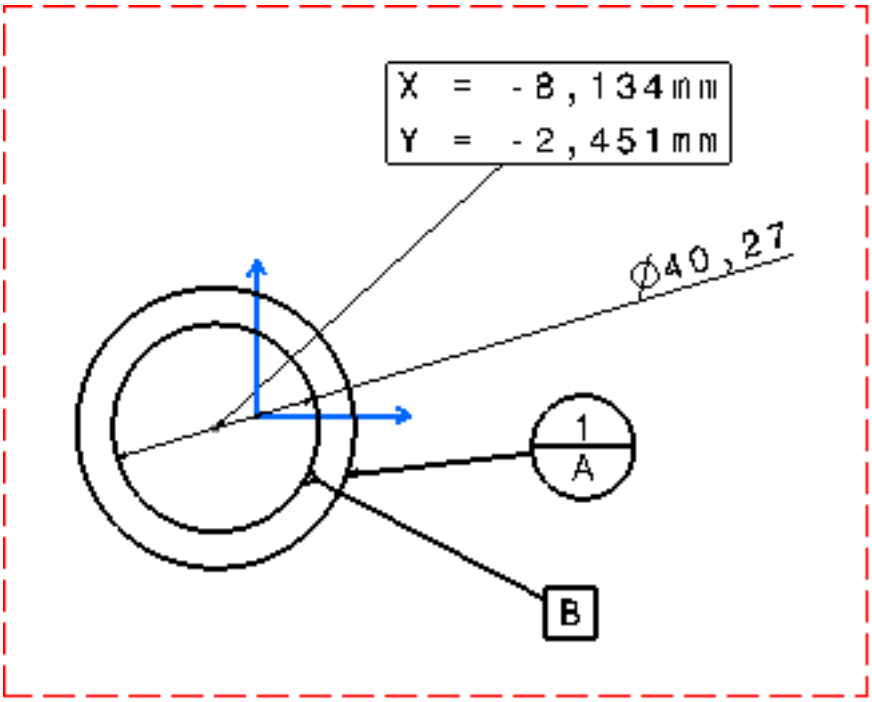
10. Click on **OK**.

The dimensions and the annotations are scaled.



In a 2D Component

1. Create a 2D component in a new detail sheet.
2. Create a diameter and a coordinate dimensions, as well as a datum feature and a datum target.
3. Multi-select the dimensions and annotations, then set the font size to 5mm.

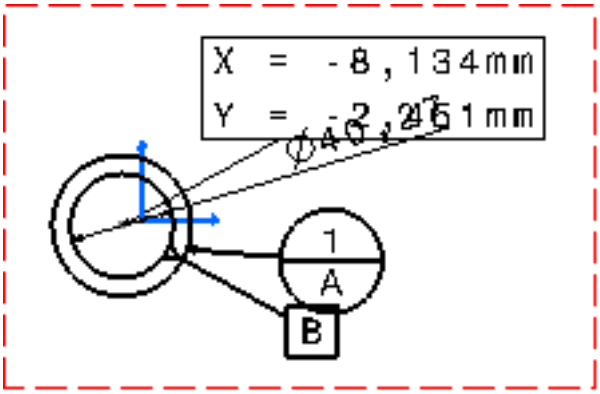


4. Right-click on the frame of the view in the detail sheet.
5. Select **Properties** in the contextual menu.

6. Enter 1:2 as new scale.

7. Click on **OK**.

   The view is updated but the dimensions and annotations are not scaled.



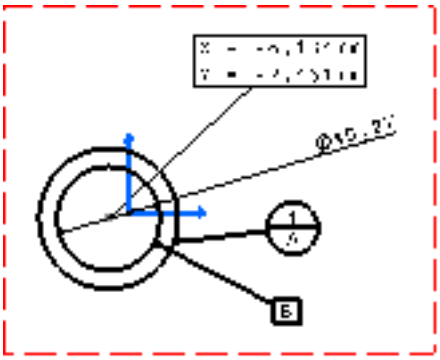
8. Multi-select the dimensions and annotations again.

9. Then right-click and select **Properties** in the contextual menu.

10. In the Text tab, check **Apply Scale**.

11. Click on **OK**.

    The dimensions and the annotations are scaled.



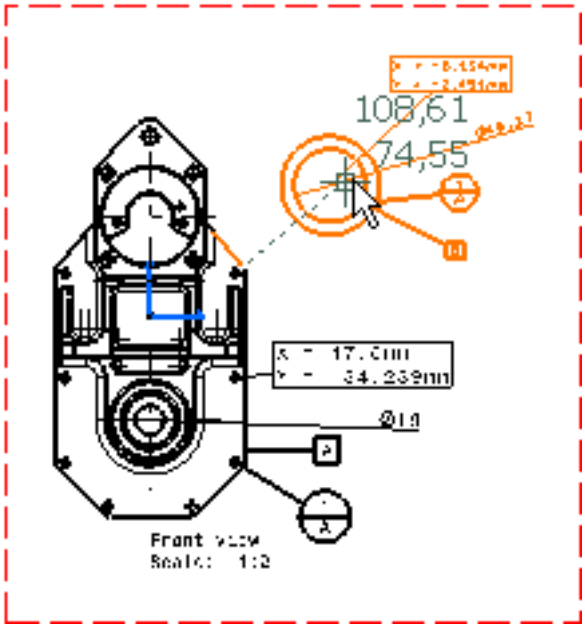
In a 2D Component instance

1. Double-click on the front view.

2. Click on the **Instantiate 2D Component** icon .

3. In the detail sheet Sheet.2 Detail, click on the component.

4. Position the component in the front view.



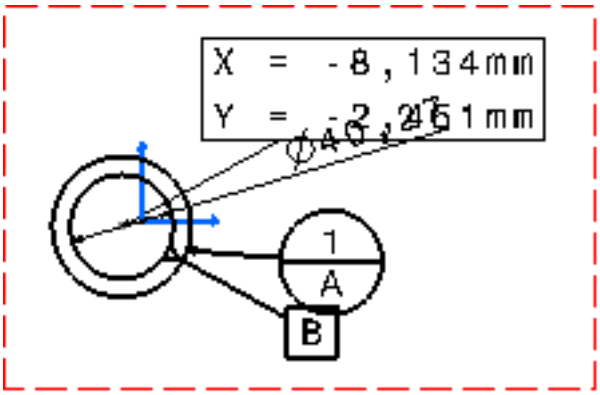
5. Back in the detail sheet, multi-select the dimensions and annotations.

6. Then right-click and select **Properties** in the contextual menu.

7. In the Text tab, uncheck **Apply Scale**.

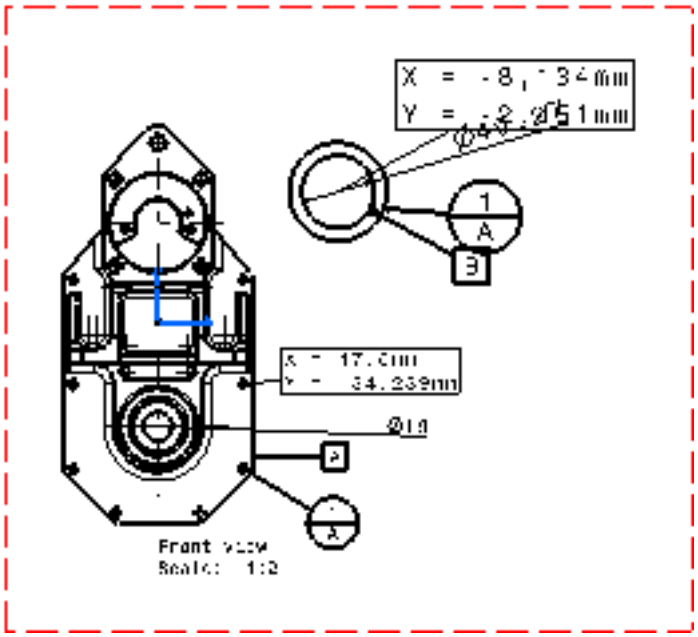
8. Click on **OK**.

   The dimensions and annotations are not scaled anymore.



9. Go back to the front view.

The dimensions and annotations in the front view are scaled, but those in the 2D component instance are not.



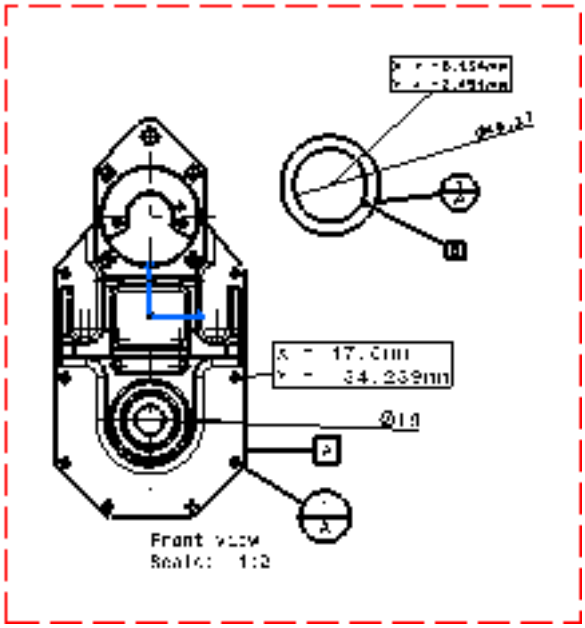
10. Back in the detail sheet, multi-select the dimensions and annotations.

11. Then right-click and select **Properties** in the contextual menu.

12. In the Text tab, check **Apply Scale**.

13. Click on **OK**.

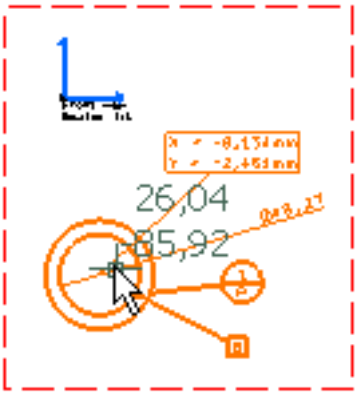
The dimensions and annotations in the front view and those in the 2D component instance are scaled.



In a 2D Component from a Catalog

1. in the detail sheet, make sure that **Apply Scale** is checked in the **Properties** panel for all dimensions and annotations.
2. Select **File->Save as**.
3. In the **Save as type** field, select catalog the click on **OK**.
4. Click on **New** and select **Drawing**.
5. Click on **OK** to create a new drawing, then create a new view.
6. Click on the **Catalog browser** icon from the Catalog toolbar .
7. In the Catalog dialog box, double-click on Sheet.2 (detail).
8. Then, double-click on 2D Component.1.

   A preview of the component is displayed.
9. Drag the component to position it in the view.



Then click once to end the positioning.

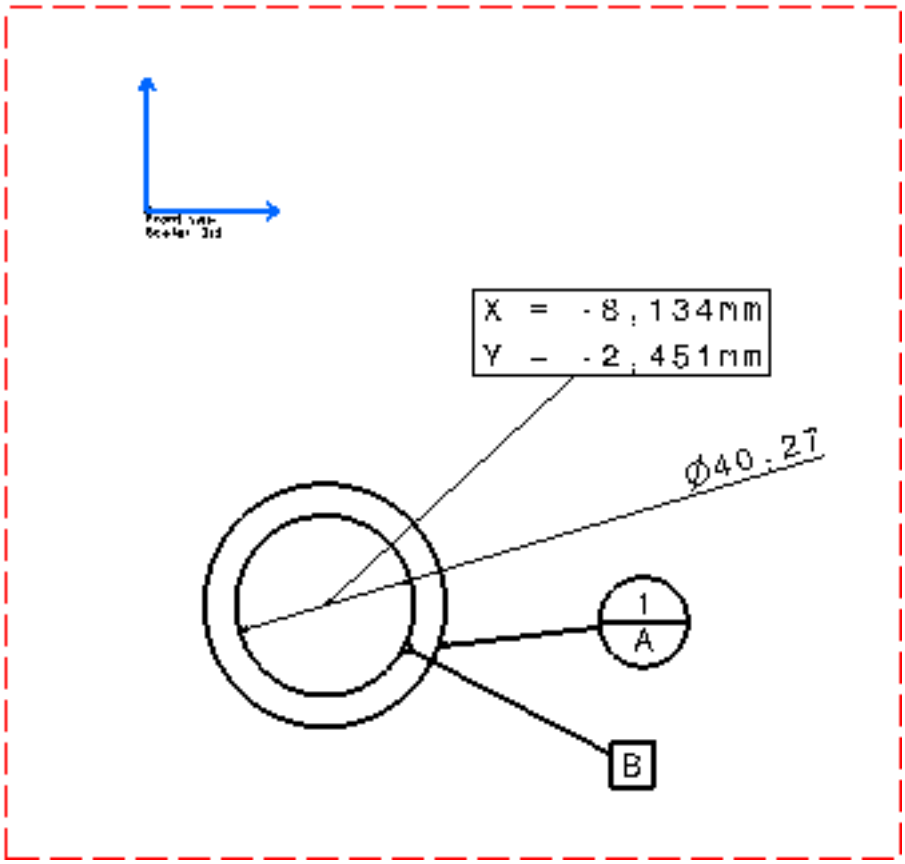
10. Right-click on the frame of the front view.

11. Select **Properties** in the contextual menu.

12. Enter 2:1 as new scale.

13. Click on **OK**.

The dimensions and annotations in the front view are scaled.



If you have save a 2D component with **Apply Scale** unchecked, the dimensions and annotations of the imported 2D component will not scale when modifying the view's scale.

- Note that the behavior of dimensions and annotations when modifying the scale is homogeneous.
- However, if you use pre-R14 data, the original behavior of dimensions and annotations when modifying the scale is preserved.

Lining up Dimensions (Free Space)

This task will show you how to line up the following dimensions relative to a point in the free space:

- Length dimensions
- Distance dimensions
- Radius dimensions
- Diameter dimensions
- Angle dimensions

In other words, you are going to organize dimensions into a system with an offset. The offset will align the dimensions according to each other, based on the position of the smallest dimension.

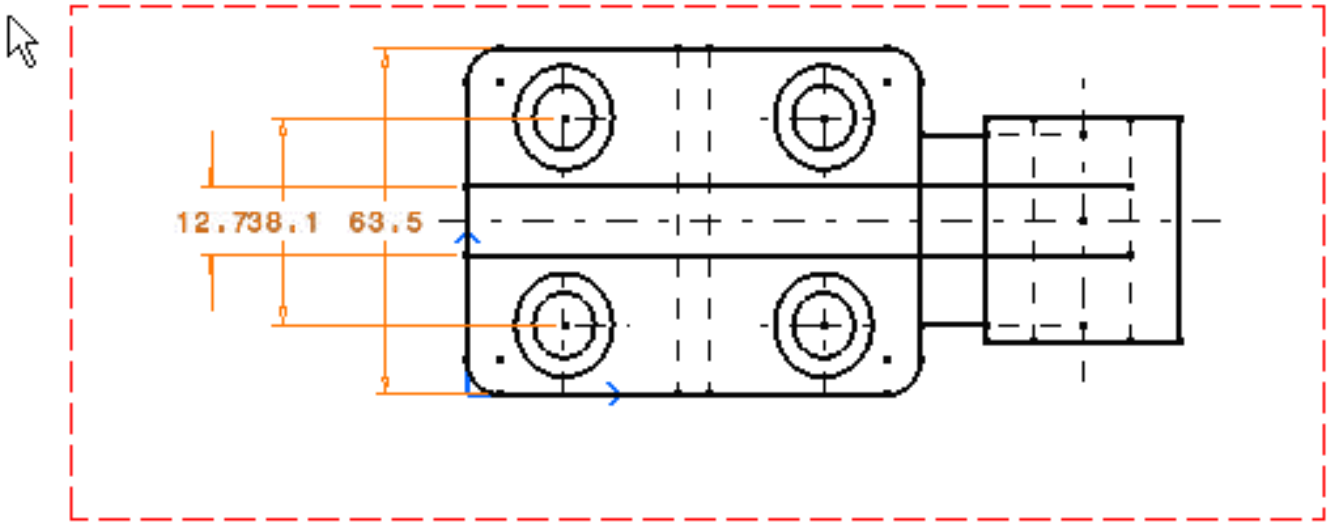
Open the LineUp_Dimensions01.CATDrawing document.

1. Select the dimensions to be lined up.

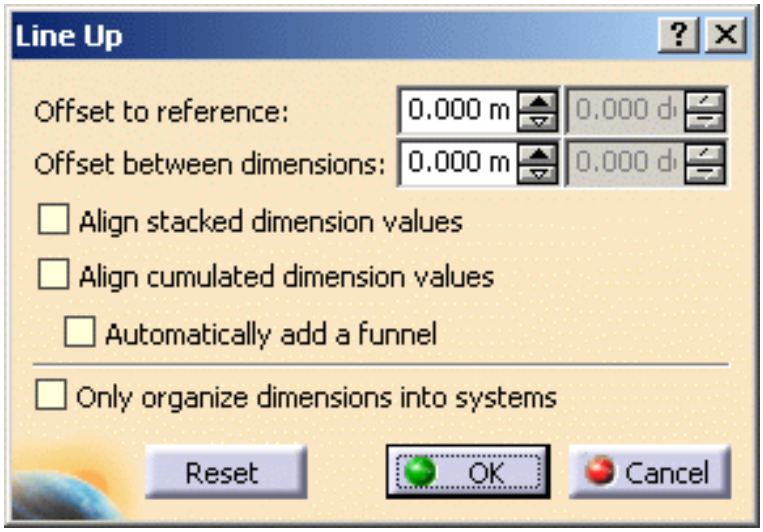
   When selecting the dimensions, make sure that they belong to a single, coherent system (if you select dimensions which could form two different systems, you could get unexpected results).

2. Do either of the following:
   - Right click and select **Line-up** from the contextual menu,
   - click the **Line-Up** icon ,
   - select **Tools->Positioning->Line-up** from the menu bar.

3. Click anywhere on the drawing.



The **Line Up** dialog box appears.



4. Enter the desired value for the offset to reference. For example, **20 mm**.

5. Enter the desired value for the offset between dimensions. For example, **20 mm**.

Two fields are available for both these options: the first field is dedicated to length, distance and angle dimensions and the second field (grayed out in our example) is dedicated to radius and diameter dimensions. Whether a field is active depends on the type of dimension selected.

6. Select **Align stacked dimension values** to align all the values of a group of stacked dimensions on the value of the smallest dimension of the group.

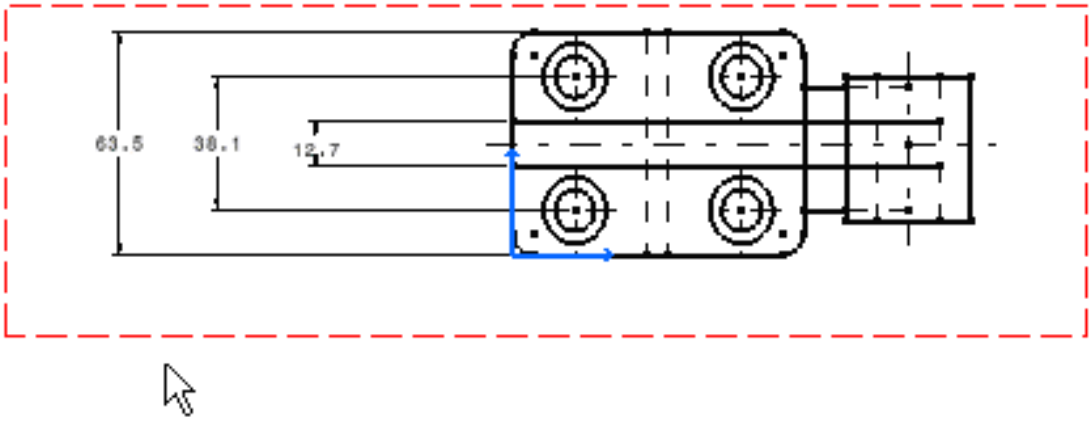
   Note that, in the case of cumulated dimensions, **Align cumulated dimension values** would align all the values of a group of cumulated dimensions on the value of the smallest dimension of the group.

7. Optionally, select **Automatically add a funnel** to display a funnel automatically whenever the dimension value cannot be displayed correctly without one.

8. Click the **Only organize into systems** option: neither the smallest dimension of each system nor dimensions that cannot be organized into systems will be moved.

9. Click **OK** to validate.

   The dimensions are now aligned. The position of the smallest system dimension is not modified. The stacked dimensions are aligned according to the smallest dimension.



When you click in the free space, the linear offset between the smallest dimension and the reference is automatically set to **0** value. The space between two dimensions will be the space defined in the **Options** dialog box (**Tools** > **Options** > **Mechanical Design** > **Drafting**> **Dimension** tab, **Line Up** paragraph). See Dimension Creation in the Customizing chapter for more details.

Lining up Dimensions (Reference)

This task will show you how to line up the following dimensions according to a given reference:

- Length dimensions
- Distance dimensions
- Radius dimensions
- Diameter dimensions
- Angle dimensions

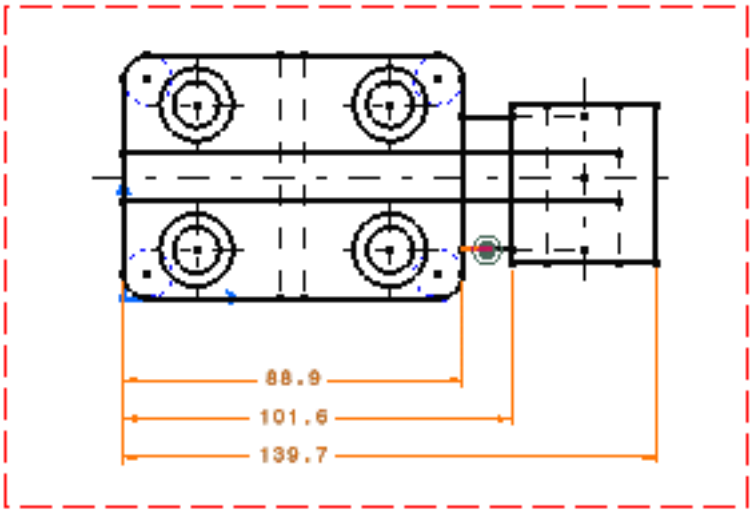
Open the LineUp_Dimensions02.CATDrawing document.

1. Select the dimensions to be lined-up.

   When selecting the dimensions, make sure that they belong to a single, coherent system (if you select dimensions which could form two different systems, you could get unexpected results).

2. Right-click and select **Line-up** from the contextual menu, or select **Tools -> Positioning -> Line-up** from the menu bar.

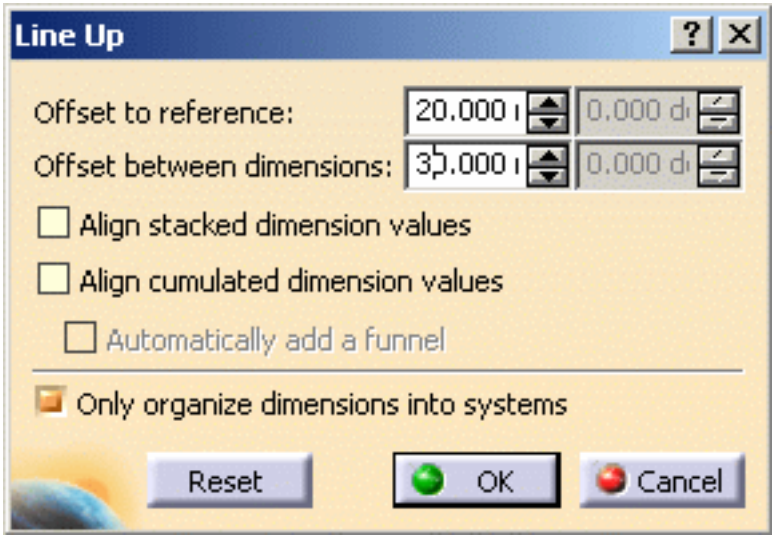
3. Select the element that will be used as reference for positioning dimensions, as show here:



   The **Line Up** dialog box appears.

4. Enter the desired value for the offset to reference. For example, **20 mm**.

5. Enter the desired value for the offset between dimensions. For example, **30 mm**.



Two fields are available for both these options: the first field is dedicated to length, distance and angle dimensions and the second field (grayed out in our example) is dedicated to radius and diameter dimensions. Whether a field is active depends on the type of dimension selected.

6. Optionally, select **Align stacked dimension values** to align all the values of a group of stacked dimensions on the value of the smallest dimension of the group.

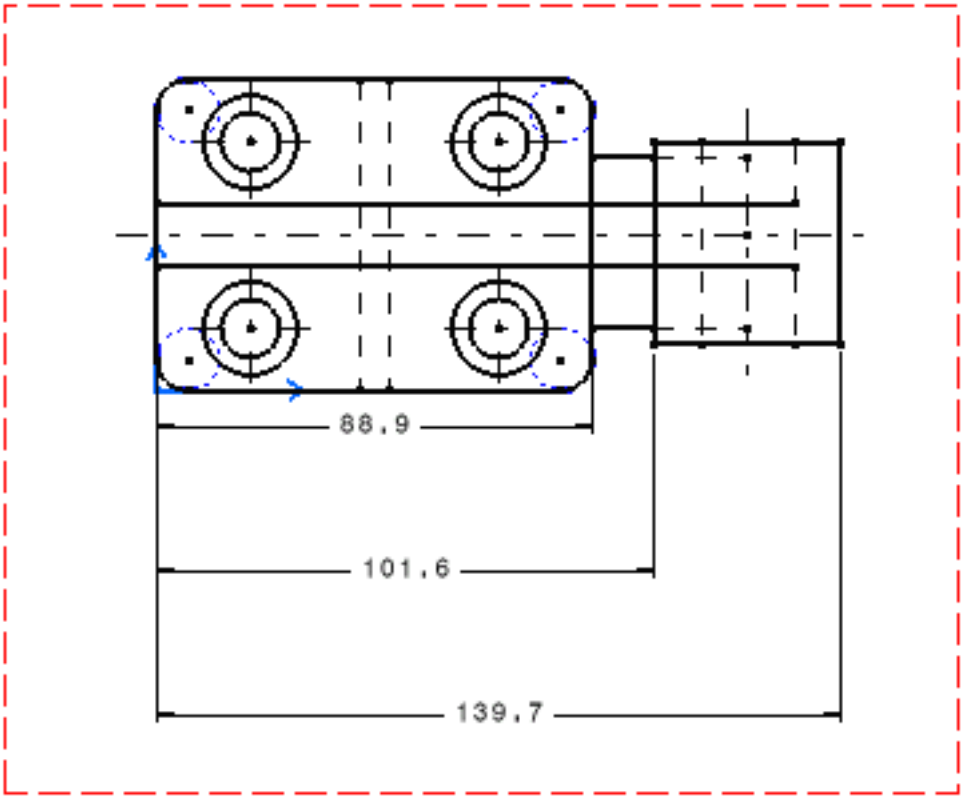
7. Optionally, select **Align cumulated dimension values** to align all the values of a group of cumulated dimensions on the value of the smallest dimension of the group.

8. Make sure the **Only organize into systems** option is not selected.

9. Click **OK** to validate.

   The smallest dimension is positioned with an offset of 20 mm according to the selected element.

   The offset between each dimension is equal to 30mm.


88.9
101.6
139.7

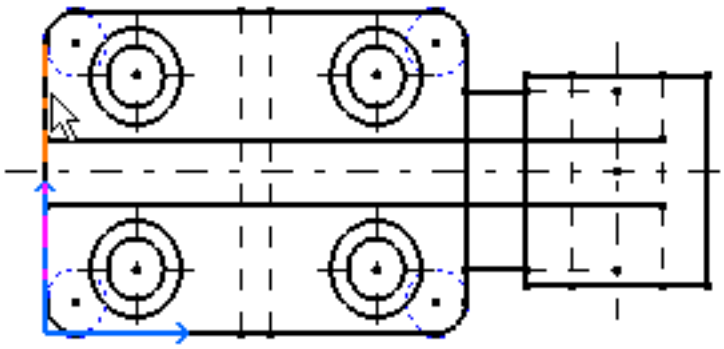
Creating a Datum Feature

This task will show you how to create a datum feature.

Open the Brackets_views08.CATDrawing document.

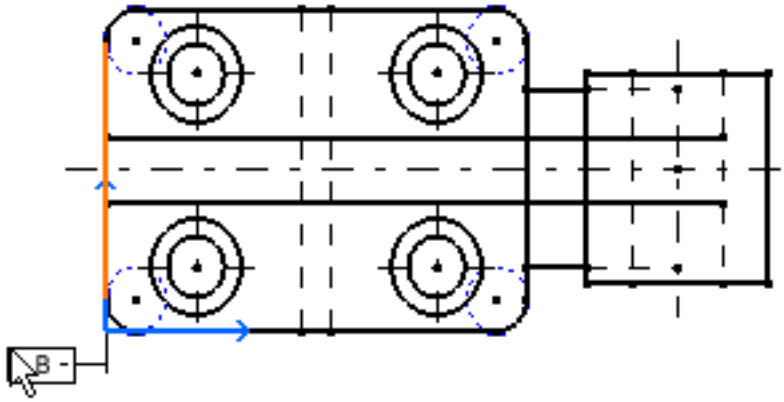
1. Click the **Datum Feature** icon from the **Dimensioning** toolbar.

2. Select the point at which you want the datum feature to be attached (attachment point).
3. Select the point at which you want the datum feature to be anchored (anchor point).



The **Datum Feature Creation** dialog box is displayed with A as default value (incremental value).

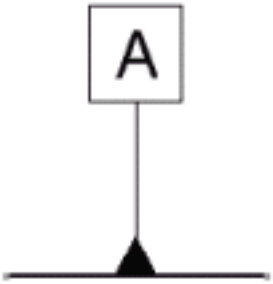
4. Change the value, if needed.



5. Click **OK**. The datum feature is created, and an extension line is automatically created on the datum feature.

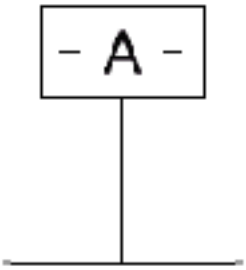
- The character string that is edited in the Datum Feature Creation dialog box is simultaneously previewed on the drawing.
- When you create more than one datum feature, the character string of this datum feature is automatically incremented.
- If the drawing uses an ANSI standard, you can change the Datum Feature ANSI representation to ASME representation. To do this, change the TXTDatumMode parameter of the standard file. Refer to Dimension parameters for more information.

ASME



TXTDatumMode = 1
(Normal)

ANSI



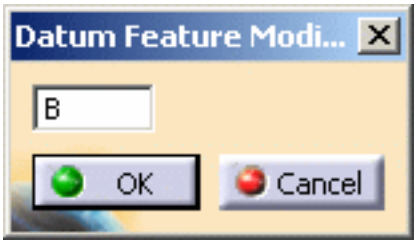
TXTDatumMode = 2
(Flag)

Modifying a Datum Feature

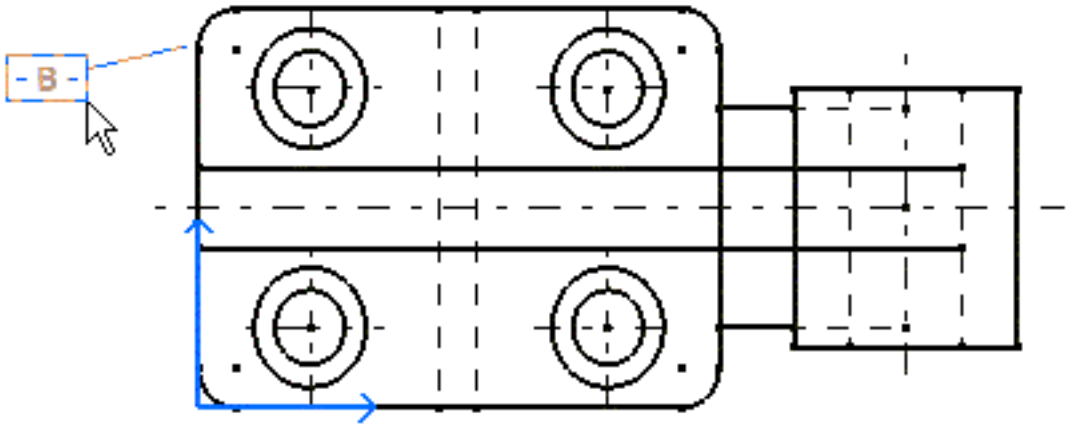
This task shows you how to modify a datum feature by editing it.

Open the Brackets_views03.CATDrawing document. Create a datum feature with a value of A.

1. Double-click the datum feature you want to modify. The **Datum Feature Modification** dialog box is displayed.
2. Modify the datum feature value. For example, enter B instead of A.



3. Click **OK**. The datum feature is modified.
4. Optionally drag the datum feature to move it.



Note that depending on the type of element to which the datum feature is attached, you may not be able to move the datum feature as wanted. For example, if the datum feature is attached to a dimension line, you will only be able to move the datum feature along the dimension line direction.

Creating a Geometrical Tolerance

This task shows you how to create a geometrical tolerance (annotation).
You can also copy an existing geometric tolerance. You can set text properties either before or after you create the text.

- Creating a geometrical tolerance
- Leader orientation
- Geometrical tolerance orientation

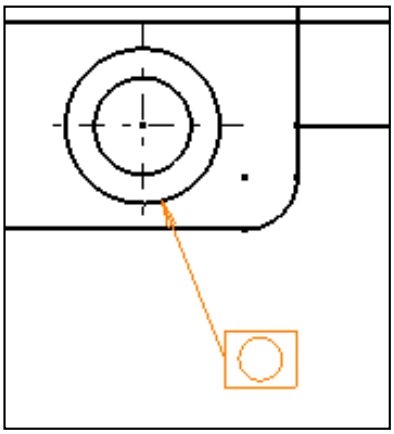
Creating a Geometrical Tolerance

Open the Brackets_views03.CATDrawing document.

1. Click the Geometric Tolerance icon from the Dimensioning toolbar (Tolerancing sub-toolbar).

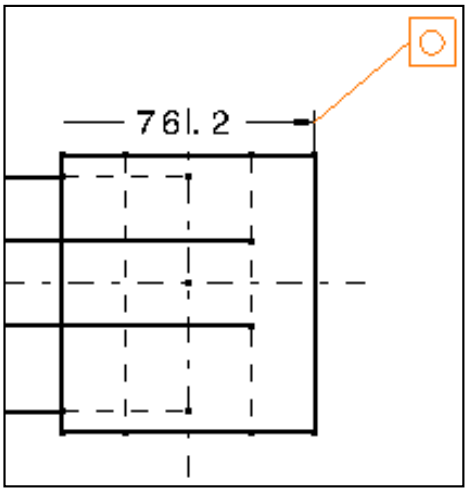
2. Select an element (geometry, dimension, dimension value, text or point) or click in the free space to position the anchor point of the geometrical tolerance.

- If you select an element, the anchor point will be an arrow. Note that you can modify this symbol by editing the annotation leader.

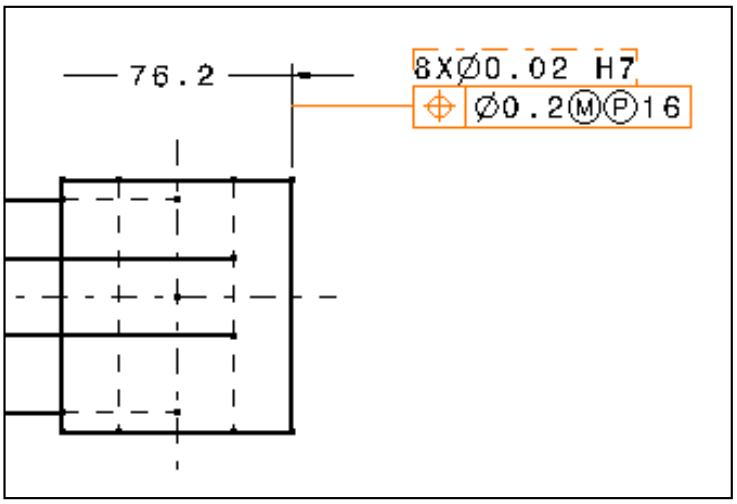


- If you select a point in the free space, the anchor point will be a small balloon.

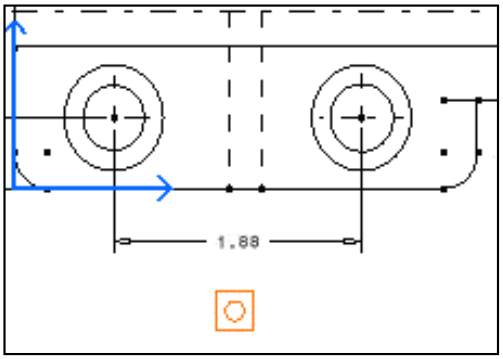
- If you select a dimension, the anchor point will be at the intersection of the dimension line and the extension line.



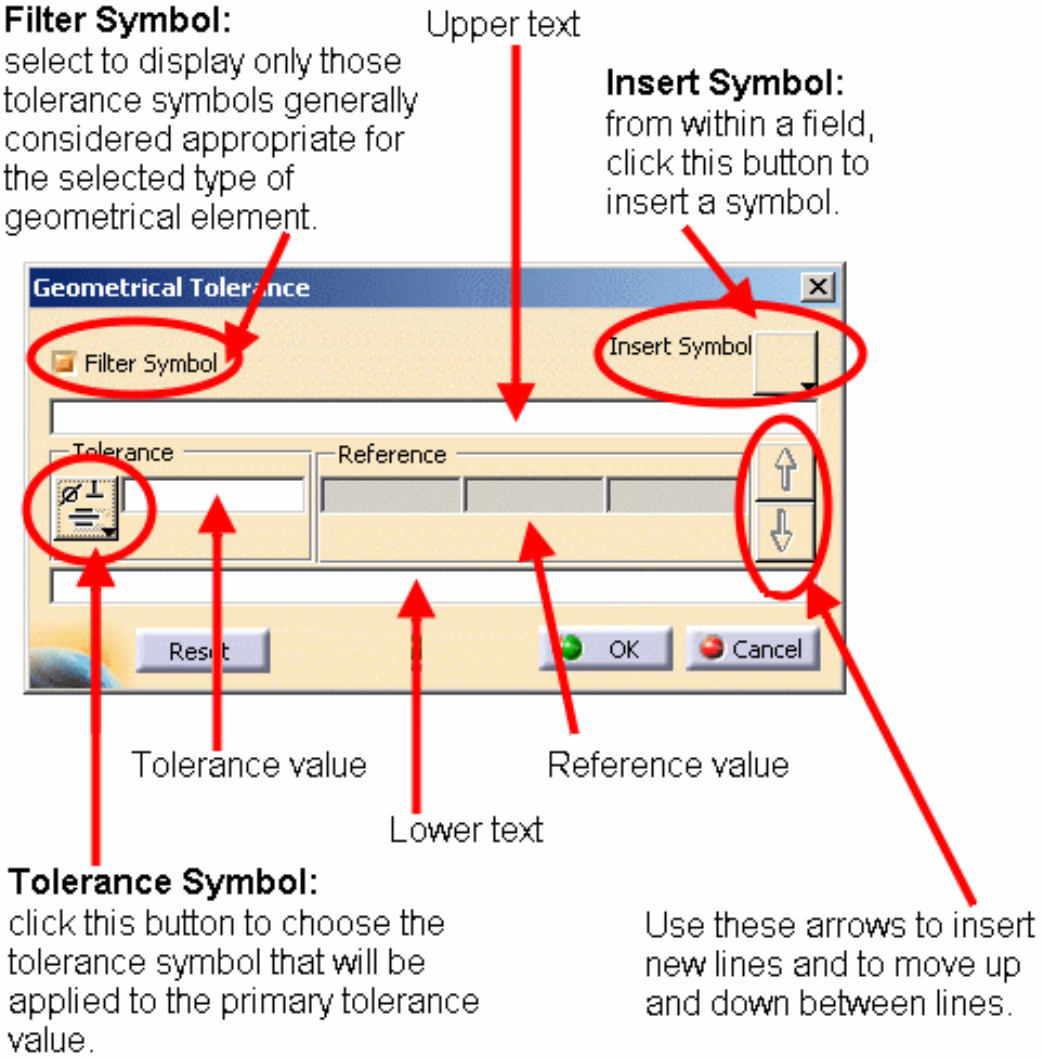
- If you press the Shift key and select the extension line, the leader is perpendicular to the extension line and the anchor point corresponds to the position of the cursor when you click to create the geometrical tolerance.



- If you select a dimension value or a text, no leader will be created. The geometric tolerance will be displayed just below and parallel to the element you selected.



3. Move the cursor to position the geometrical tolerance and then click at the chosen location. The Geometrical Tolerance dialog box appears.



- At this step, you can apply the parameter values of an existing geometric tolerance to the tolerance you are creating: to do this, simply select the existing geometric tolerance.
- If you have selected the **Use style values to create new objects** option in **Tools -> Options -> Mechanical Design -> Drafting -> Administration** tab, the Geometrical Tolerance dialog box is pre-filled with custom style values (as defined in the Standards Editor). In this case, Properties toolbars and the Tools Palette are disabled during the creation of the geometrical tolerance.
  On the other hand, if you have not selected this option, the Geometrical Tolerance dialog box is pre-filled with the last entered values (if any). In this case, Properties toolbars and the Tools Palette are active during the creation of the geometrical tolerance.
- You can reset the current style values in the Geometrical Tolerance dialog box at any time using the **Reset** button.

4. Select the Filter Symbol option to filter the available tolerance symbols according to the type of geometrical element you selected (if any).

If you did not select any geometrical element, the tolerance symbols will not filtered.

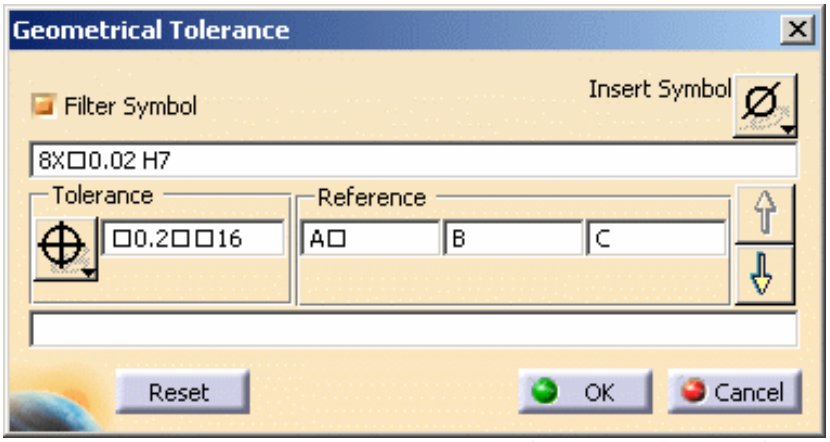
5. Specify the tolerance type by clicking the Tolerance Symbol button and selecting the appropriate symbol.
6. Type the tolerance value in the Tolerance value field, adding symbols as needed. To do this, position the cursor at the proper location in the field, and click the Insert Symbol button to choose the appropriate symbol.

You can add symbols to the tolerance and reference value as well as to the upper and lower text.

7. Type the reference values in the Reference value fields, adding symbols as needed.

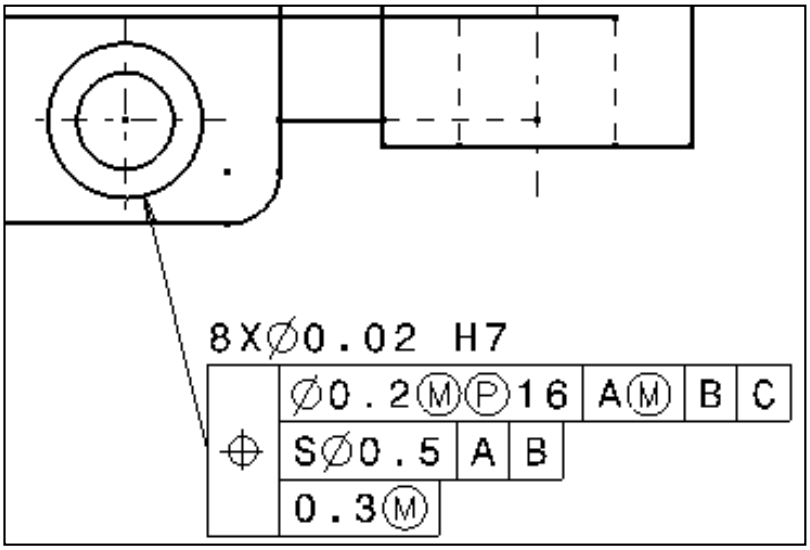
8. To add a new geometrical tolerance, click the Next line arrow button and repeat steps 4 to 5.

9. Type the upper and lower texts in the appropriate fields. You may also add symbols if you want to.

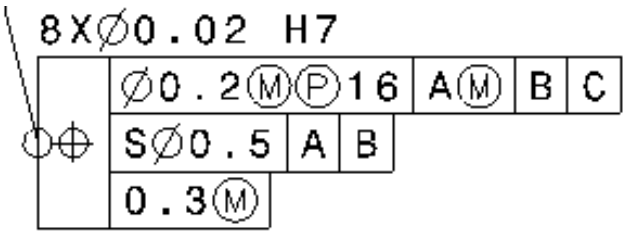


The geometric tolerance is updated as you define values for each field.

10. Click OK when you're done. The geometrical tolerance is created.



11. You can add an all-around symbol to the leader. To do this, select the geometrical tolerance, right-click the yellow manipulator on the arrow and select All Around from the contextual menu.



Specifying Leader Orientation

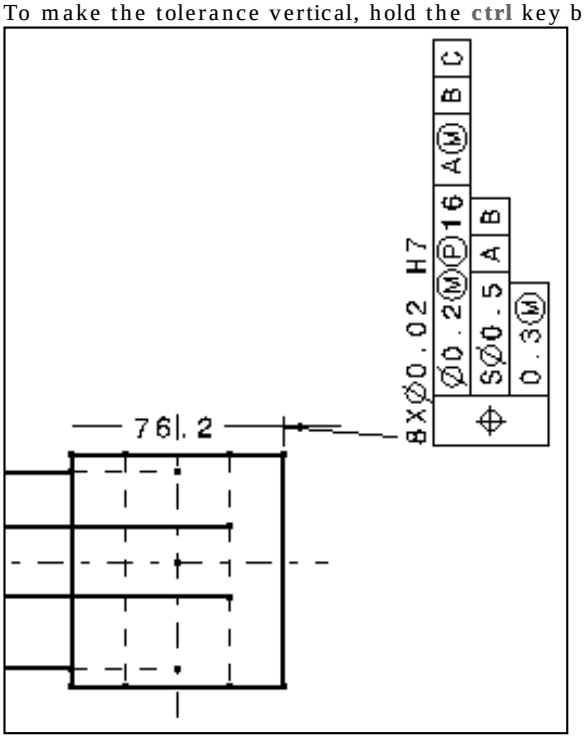
You can orient the geometrical tolerance leader perpendicularly to the element to which it is associated (for example, if the leader is associated to a dimension, you can position the leader parallel to the dimension line and orthogonal to the extension line).

For this, you have two different possibilities:

- Either go to **Tools -> Options -> Mechanical Design -> Drafting -> Annotation and Dress-up** tab and check **Activate snapping (SHIFT toggles)**. Then, click the **Configure** button and select either **On orientation** or **Both**. To orient directly the geometrical tolerance leader perpendicularly to the associated element, press the **Shift** key before clicking in the drawing to position the tolerance (see previous scenario, step 3).
- Or go to **Tools -> Options -> Mechanical Design -> Drafting -> Annotation and Dress-up** tab and check **Geometrical tolerance** in **Annotation Creation**. The leader will be oriented perpendicularly to the geometry by default. In this case, pressing the **Shift** key will let you orient it differently.

Specifying Geometrical Tolerance Orientation

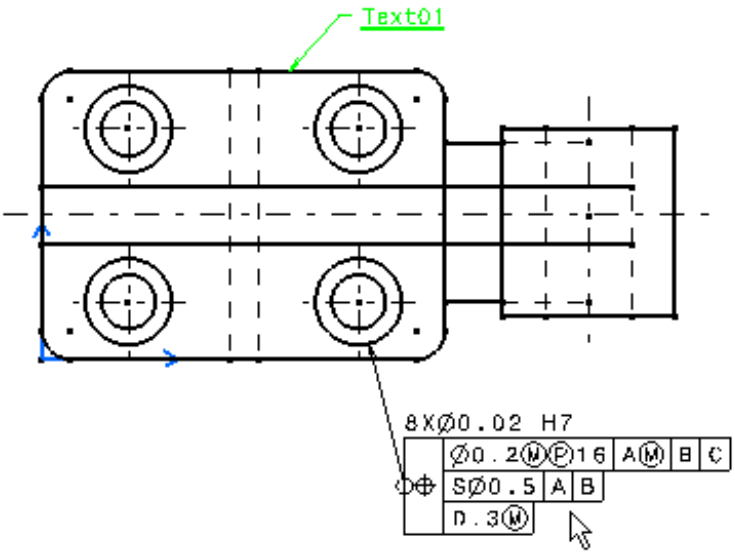
To make the tolerance vertical, hold the **ctrl** key before clicking in the drawing to position the tolerance (previous scenario, step 3).



Modifying Geometrical Tolerances

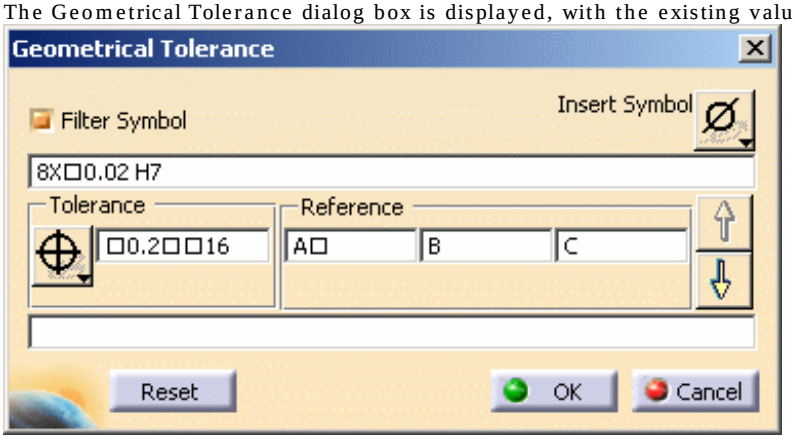
This task shows you how to modify a geometrical tolerance.

Open the Brackets_views03.CATDrawing document. Create a geometrical tolerance.



1. Double-click the geometrical tolerance you want to modify.

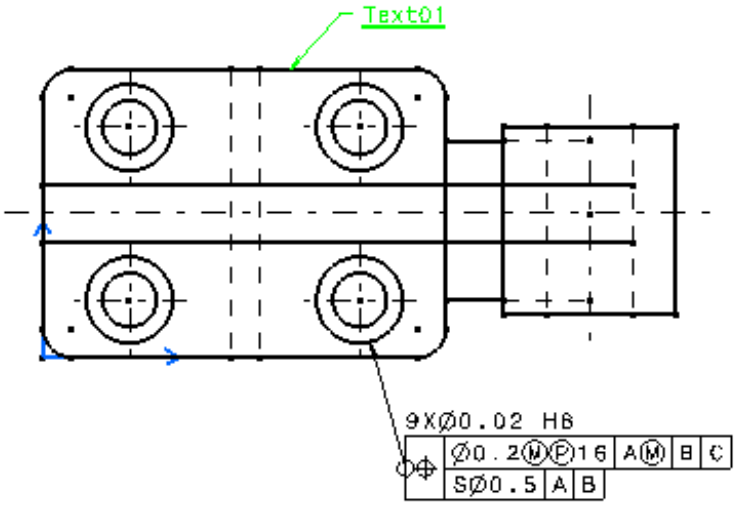
The Geometrical Tolerance dialog box is displayed, with the existing values pre-entered.



You can reset the current style values in the Geometrical Tolerance dialog box at any time using the **Reset** button.

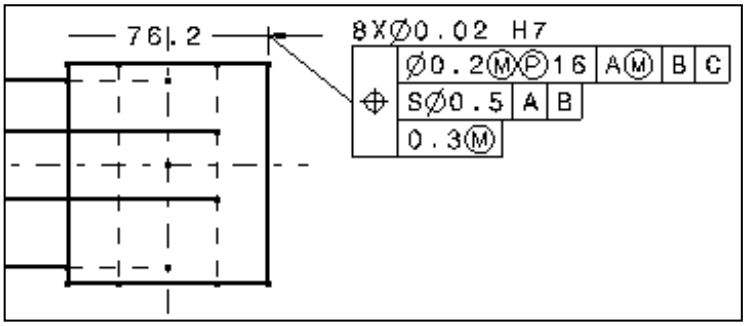
2. Modify the values as desired, as explained in Creating a Geometrical Tolerance.

3. Click OK.

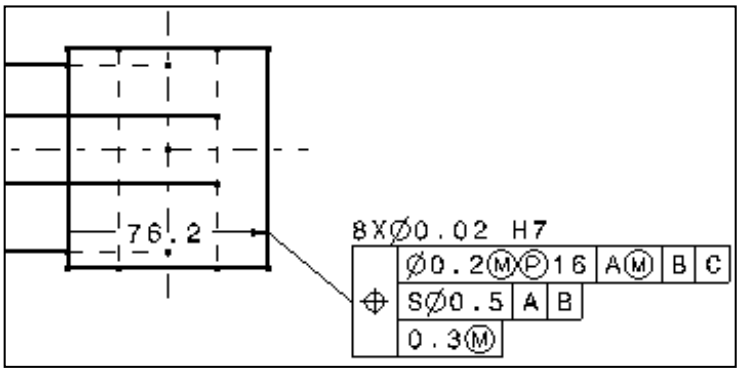


4. Click in the free space to validate the geometrical dimension modification.

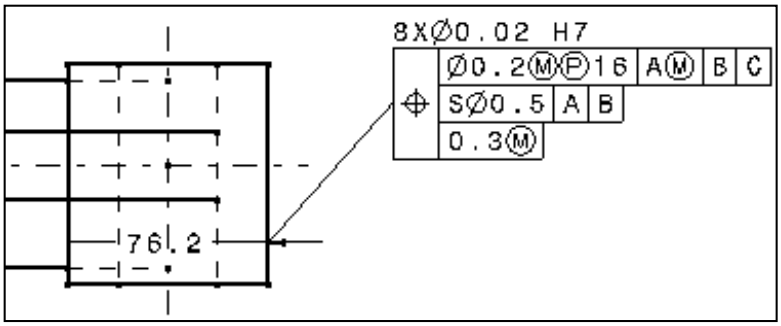
5. Create a dimension, then create a geometrical tolerance on it.



6. Click on the extension line and move the dimension.
You can see that the geometrical tolerance follows the dimension.



7. Click on the geometrical tolerance and move it.
You can see that it has not impact on the position of the dimension.



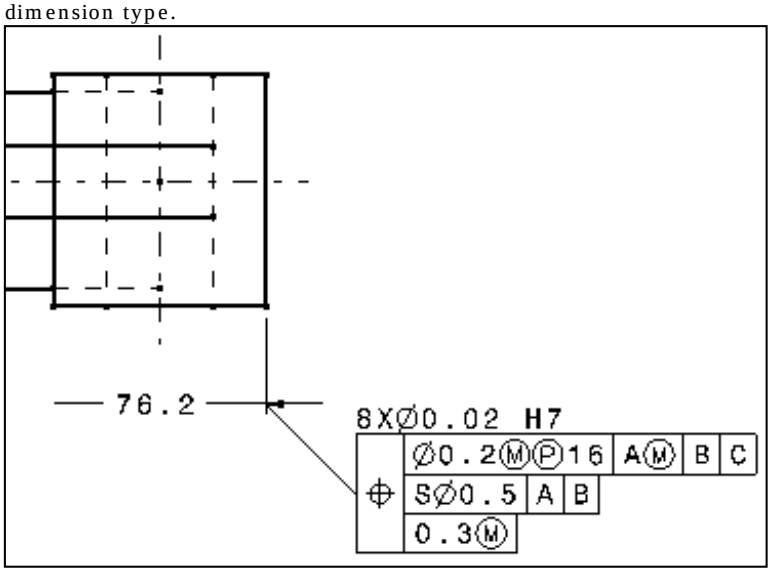
The behavior is the same if a geometrical tolerance is created on the dimension value.

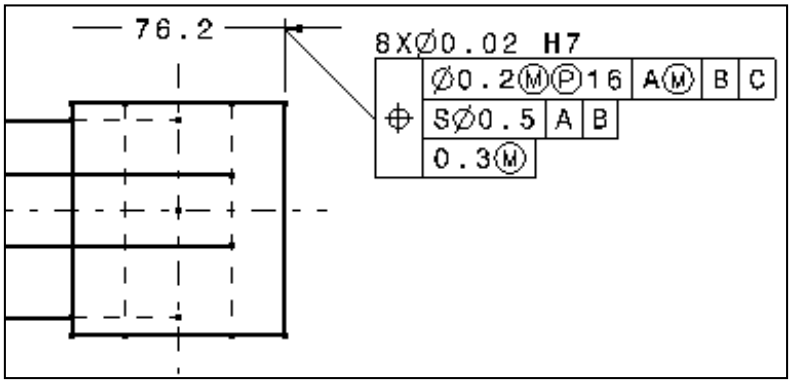
Associativity between the dimension and the geometrical tolerance

1. Create a dimension.

2. Create a geometrical tolerance on it, selecting the dimension line.

3. Move the dimension.
You will note that the positioning and the length of the tolerance leader remain constant when moving the dimension, whatever the dimension type.



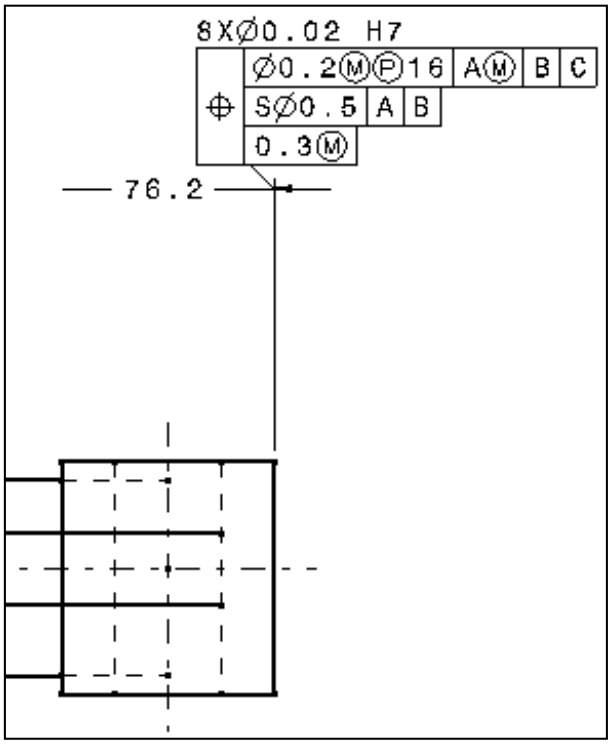


5. Create a dimension.

6. Create a geometrical tolerance on it, selecting the extension line.

7. Move the dimension.
You will note the length of the tolerance leader is recomputed as long as the dimension is moved.
This is due to the fact that the ratio between the lengths of the leader's projection and the extension line's projection remain constant.

The positioning of the leader is also recomputed so that the distance between the anchor point of the tolerance and the extension line remain constant.



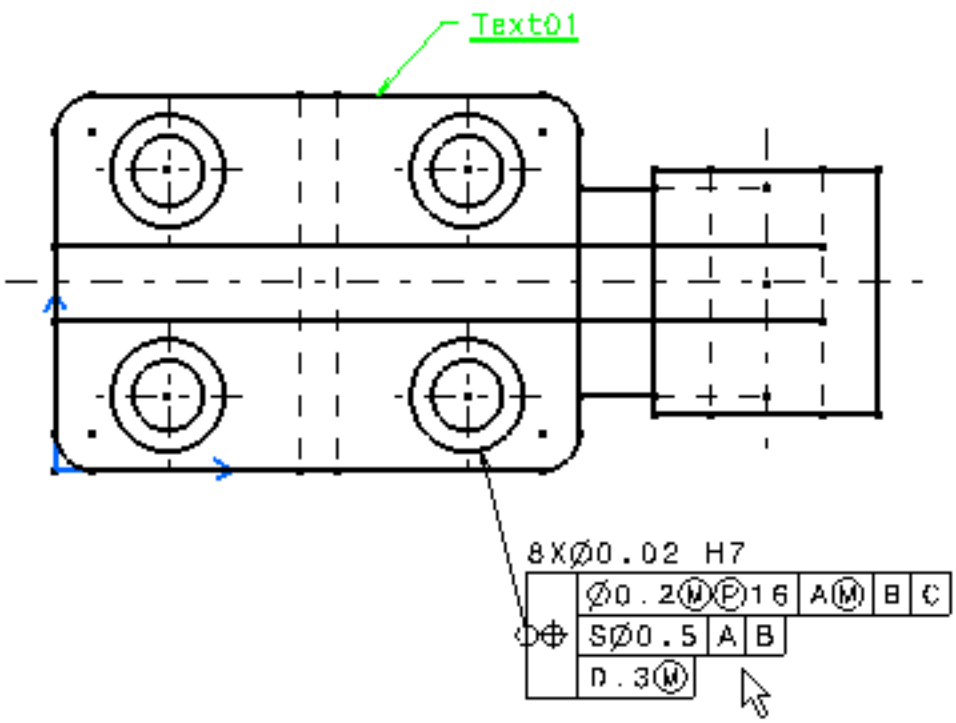
When a tolerance is created on an angle dimension, the positioning and the length of the leader remain constant if the dimension is moved.

Copying Geometrical Tolerances

This task will show you how to copy an existing geometrical tolerance and then edit the content for creating a new one.

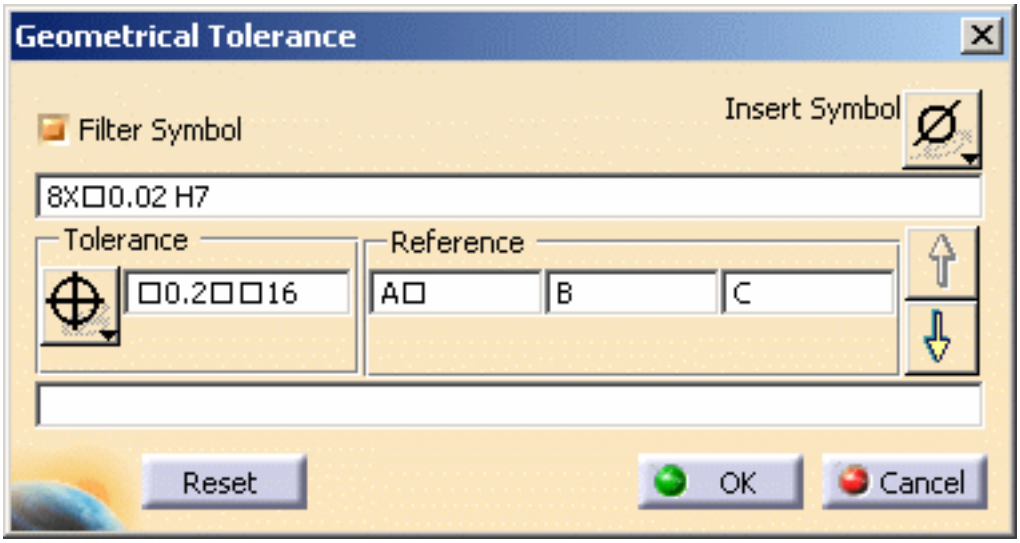
Open the Brackets_views03.CATDrawing document. Create a geometrical tolerance.

1. Click on the geometrical tolerance you want to copy.



2. Right-click and select **Copy** from the contextual menu.
3. Select the element to which you want the geometrical tolerance to be associated.
4. Right-click and select **Paste** from the contextual menu.
5. Move the copied geometrical tolerance to position it as desired.
6. Double-click the copied geometrical tolerance.

   The Geometrical Tolerance dialog box is displayed, with the existing values pre-entered.



7. Make sure the **Filter Tolerance** box is selected. This will display only those tolerance symbols generally considered appropriate for the type of geometrical element selected.

   Unselecting this box displays all symbols, regardless of the selected type of element.

8. Modify the values as desired, as explained in Creating a Geometrical Tolerance.

9. After you are done entering values in a given field, press the Tab key to move to the next field.

   The geometrical tolerance is updated as you define values for each field.

10. Click **OK** to confirm your operation and close the dialog box.

11. Click anywhere in the drawing to validate.

Dimension Systems

As dimension system commands work as in the Interactive Drafting workbench, this section of the documentation provides links to the *Interactive Drafting User's Guide*. As such, the information detailed in this section is presented in an Interactive Drafting context. You should note that the *Interactive Drafting User's Guide* contains images that correspond to the Interactive Drafting workbench and therefore illustrate dimension systems in an environment that is different from the 2D Layout for 3D Design environment (symbols and background color, for example).

Refer to Dimensioning in a 2D Layout for 3D Design Context which explains what is specific to creating dimensions in the 2D Layout for 3D Design workbench as opposed to doing so in the Interactive Drafting workbench.

Before you begin: You should be familiar with basic concepts.

Create chained dimension systems: Create a chained dimension system using selection.

Create cumulated dimension systems: Create a cumulated dimension system using selection.

Create stacked dimension systems: Create a stacked dimension system using selection.

Modifying a dimension system: Modify a dimension system or a dimension within a dimension system.

Line-up dimension systems: Line-up dimensions according to a given reference.

Before You Begin

A dimension system is a set of dimensions which can be handled either globally or individually.

Creating Dimension Systems

You can create (and therefore modify) the following types of dimension systems:

- Chained Dimension Systems
- Cumulated Dimension Systems
- Stacked Dimension Systems

Note that you can create half-dimensions on stacked dimension systems only.

Manipulating Dimension Systems

- By default, when manipulating dimension systems, **Dimension following the cursor** is activated. Go to **Tools -> Options -> Mechanical Design -> Drafting -> Dimension** tab, to use or not automatic positioning.
- A click over a dimension system enables you to select the whole dimension system. However, you may want to reverse this behavior to select a single dimension. Click on the **Dimension system selection mode** icon in the **Tools** toolbar to deactivate the dimension system. Selections will be now focused on dimensions rather than on the whole dimension system. Click again on the icon to activate the dimension system.

Modifying the Dimension System Attributes

You can modify the following attributes while you have no more than one dimension in the dimension system:

Modify while creating:

- Type
- Measure direction. Refer to Using Tools for more information.
- Angle sector

Modify after creating:

- Properties (see further down)

- When creating: to activate temporarily the **Dimension following the cursor** option, hold on the **ctrl** key.
- When creating and editing: to activate temporarily the **Activate Snapping** option, hold on the **shift** key.
- When creating an angle dimension: if the **Dimension following the cursor** option is activated, you can swap the angle sector of the first dimension according to the mouse position holding on the **ctrl** and **shift** keys. If the **Dimension following the cursor** option is not activated, you can swap to the complementary angle sector holding on the **ctrl** key and clicking on the dimension line.

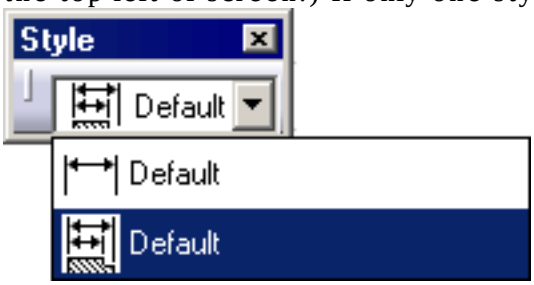
Dimension System Properties

You can apply given properties to dimension systems. For this, use the edit **Properties**, see Editing Dimension System Properties. Note, chained dimension systems have no specific system properties.

Using Styles

You can use styles when creating dimension systems in drawings created with version R14 and later (or pre-R14 drawings whose standard has been updated or changed in R14 and later). Styles are defined in the standard used by the drawing and managed by the administrator.

When creating a dimension system, the Style toolbar displays the styles available for this type of dimension system and the styles available for its dimensions. (By default, the Style toolbar is situated at the top left of screen.) If only one style is available, it will be used by default.



If several styles are available for this type of dimension system, you can choose the style that you want to use to create this dimension system by selecting it from the Style toolbar.

If several styles are available for dimension in the dimension system, you can choose the style that you want to use to create this dimension by selecting it from the Style toolbar.

Refer to Using Styles for more information.

Creating Chained Dimension Systems

This task will show you how to create a chained dimension system.

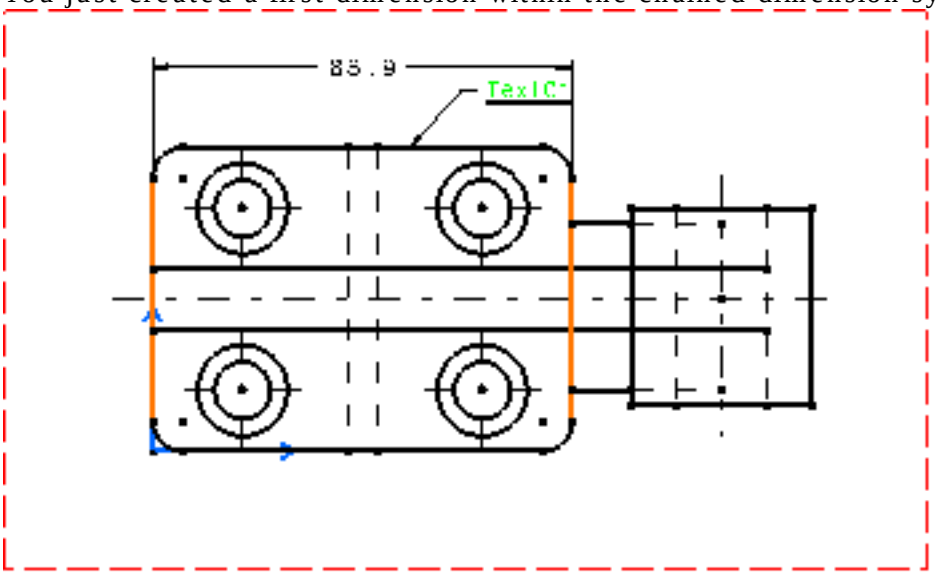
Open the Brackets_views03.CATDrawing document.

1. Click the **Chained Dimensions** icon from the **Dimensioning** toolbar (**Dimensions** sub-toolbar).

2. Click a first point on the view.

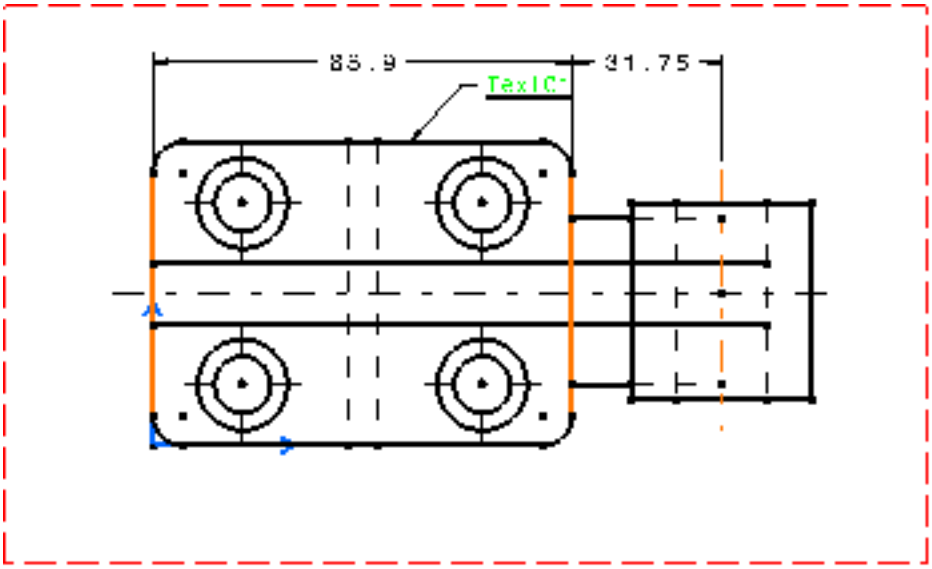
3. Click a second point on the view.

You just created a first dimension within the chained dimension system.



4. Click a third point on the view.

You now created a second chained dimension in the system. You can create as many chained dimensions as desired.



Note that if you move one dimension line as you create a chained dimension, all the lines will move accordingly. In the same way, clicking on one dimension line highlights all the lines showing the whole system is selected.

5. Click in the free space to end the chained dimension system creation.

The Chained Dimension System works for distance and angle dimensions only.

You can interrupt manually extension lines on both single dimensions of a system and the whole system. For more information, refer to the Interrupting Extension Lines section.

Creating Cumulated Dimension Systems

This task will show you how to create a cumulated dimension system on a view.

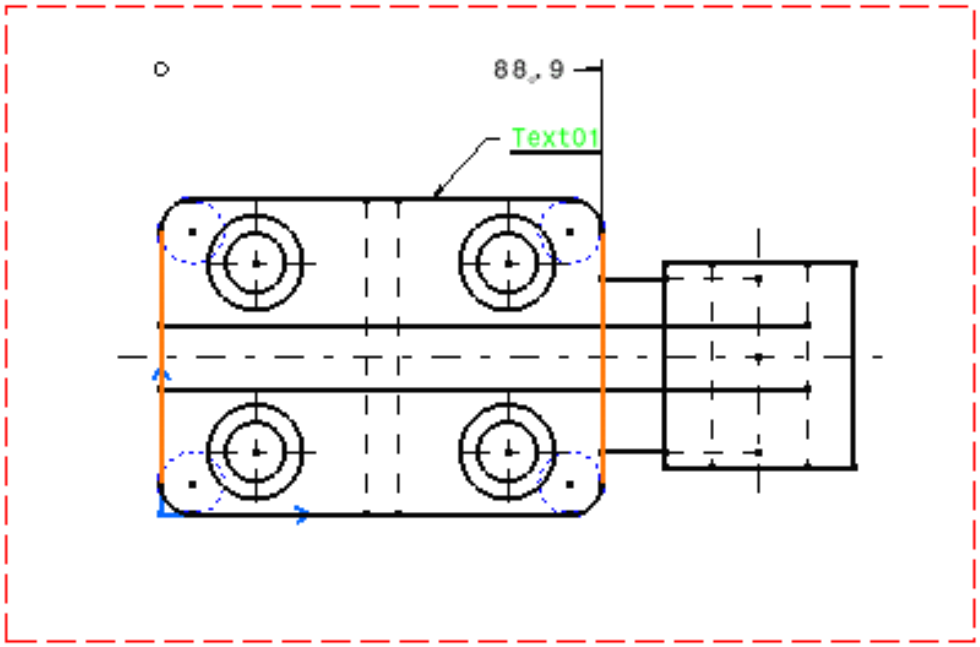
Open the Brackets_views03.CATDrawing document.

1. Click the **Cumulated Dimensions** icon from the **Dimensioning** toolbar (**Dimensions** sub-toolbar).

2. Click a first point on the view.

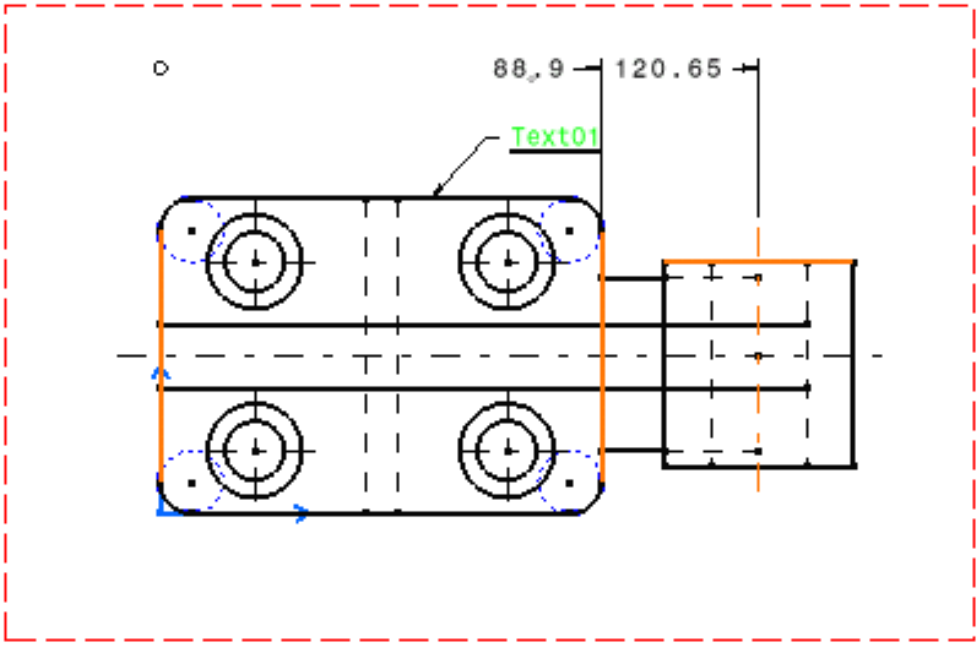
3. Click a second point on the view.

   You just created a first dimension within the cumulated dimension system.



4. Click a third point on the view.

   You now created a second cumulated dimension in the system. You can create as many cumulated dimensions as desired.



Note that if you move one dimension line as you create a cumulated dimension, all the lines will move accordingly. In the same way, clicking on one dimension line highlights all the lines, thus showing that the whole system is selected.

5. Click in the free space to end the cumulated dimension system creation.

- If the cumulated dimensions are set with the value oriented along dimension line, set the Parameters applying only if value orientation reference is Dimension Line (Cumulate Dimension: Value Orientation Reference = 1) in the standards.
- You can interrupt manually extension lines on both single dimensions of a system and the whole system. For more information, refer to the Interrupting Extension Lines section.

- The Cumulated Dimension System works for distance and angle dimensions only.

Creating Stacked Dimension Systems

This task will show you how to create a stacked dimension system on a view.

Stacked dimensions are parallel lines with a common extension line.

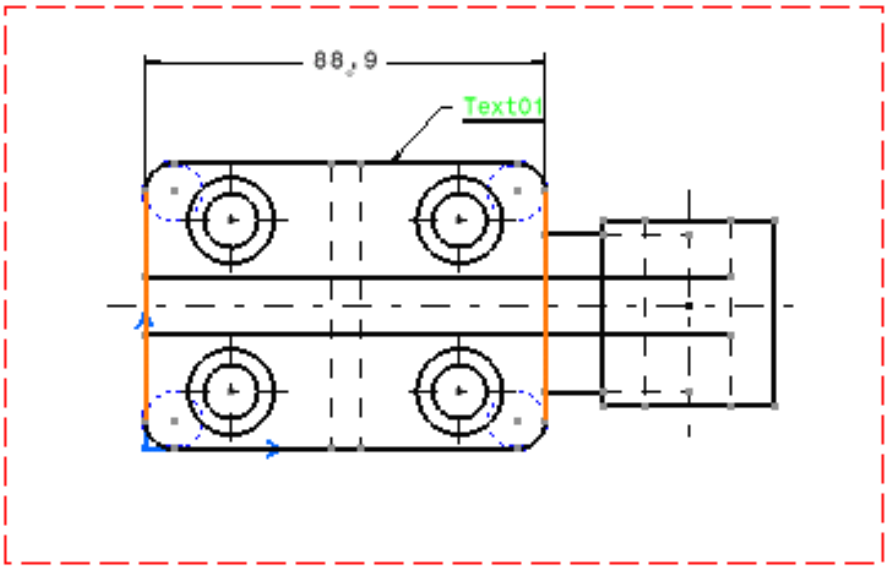
Open the Brackets_views03.CATDrawing document.

1. Click the **Stacked Dimensions** icon from the **Dimensioning** toolbar (**Dimensions** sub-toolbar).

2. Click a first point on the view.

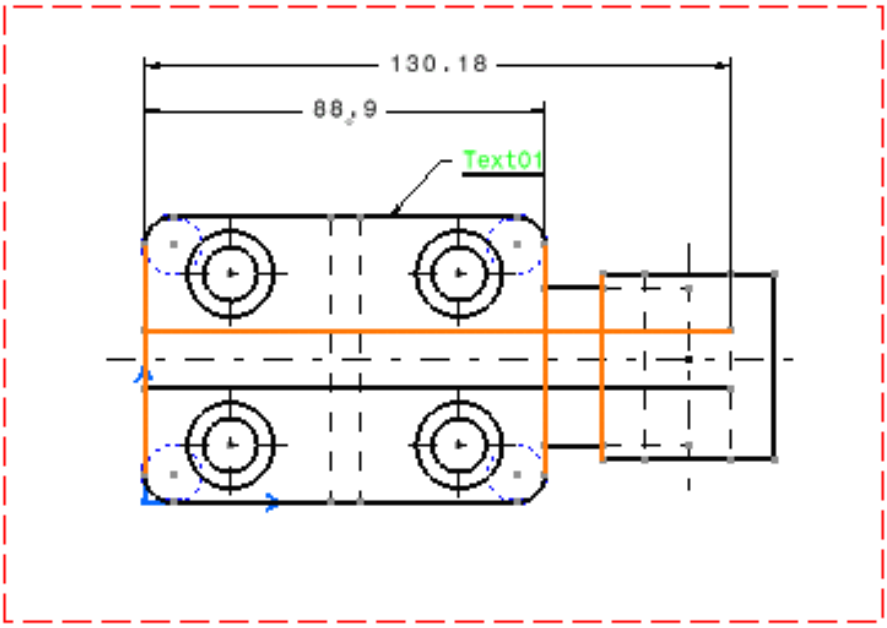
3. Click a second point on the view.

   You just created a first dimension within the stacked dimension system.



4. Click a third point on the view.

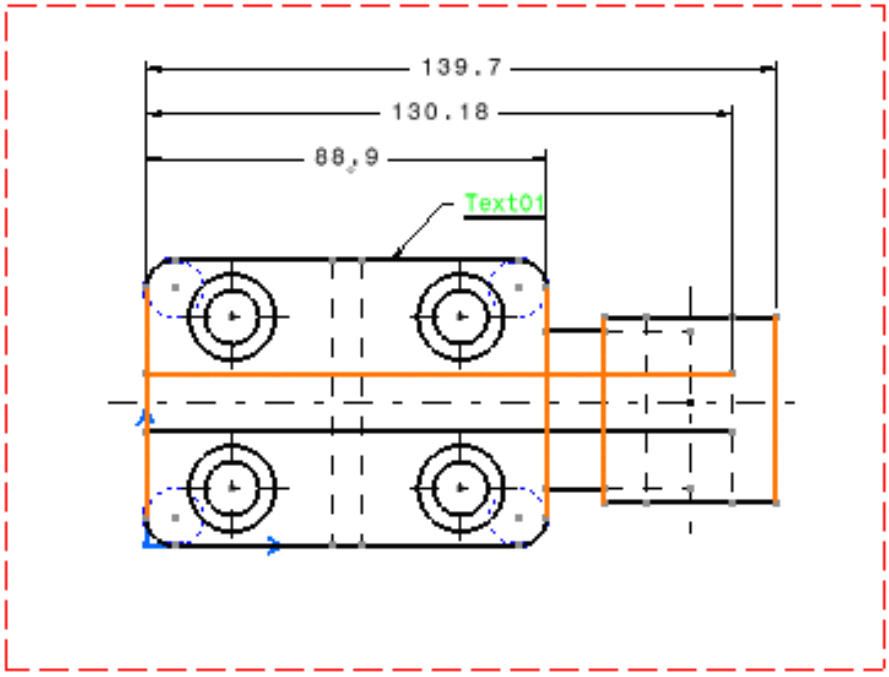
   You now created a second stacked dimension in the system.



5. Click a third point on the view.

   You now created a third stacked dimension in the system. Note that this stacked dimension is inserted properly into the system.

   You can create as many stacked dimensions as desired.



6. Click in the free space to end the stacked dimension creation.

The Stacked Dimension System works for distance and angle dimensions only.

You can interrupt manually extension lines on both single dimensions of a system and the whole system. For more information, refer to the Interrupting Extension Lines section.

Modifying a Dimension System

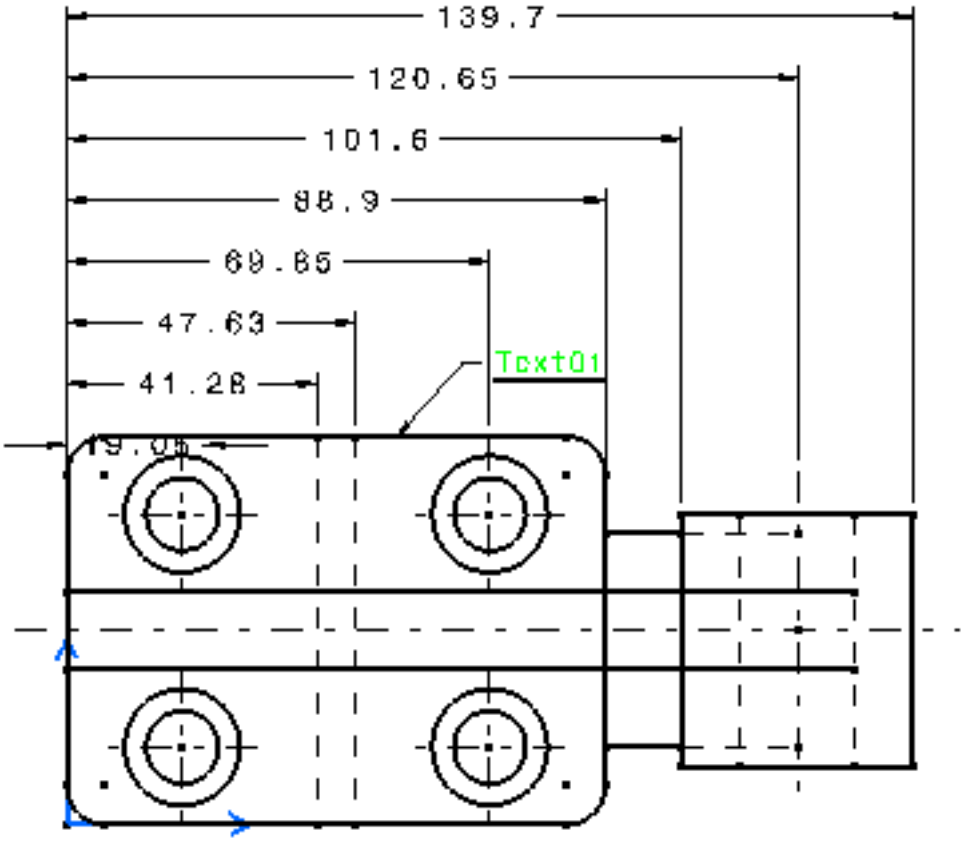
This task will show you how to modify a dimension system or a dimension within a dimension system.

In our example we chose to create a stacked dimension system and to perform the following actions:

- Moving a dimension system
- Moving a dimension
- Aligning a dimension system
- Restoring a value position
- Adding a dimension into a system
- Deleting a dimension from a system
- Copying a dimension system

Open the Brackets_views03.CATDrawing document.

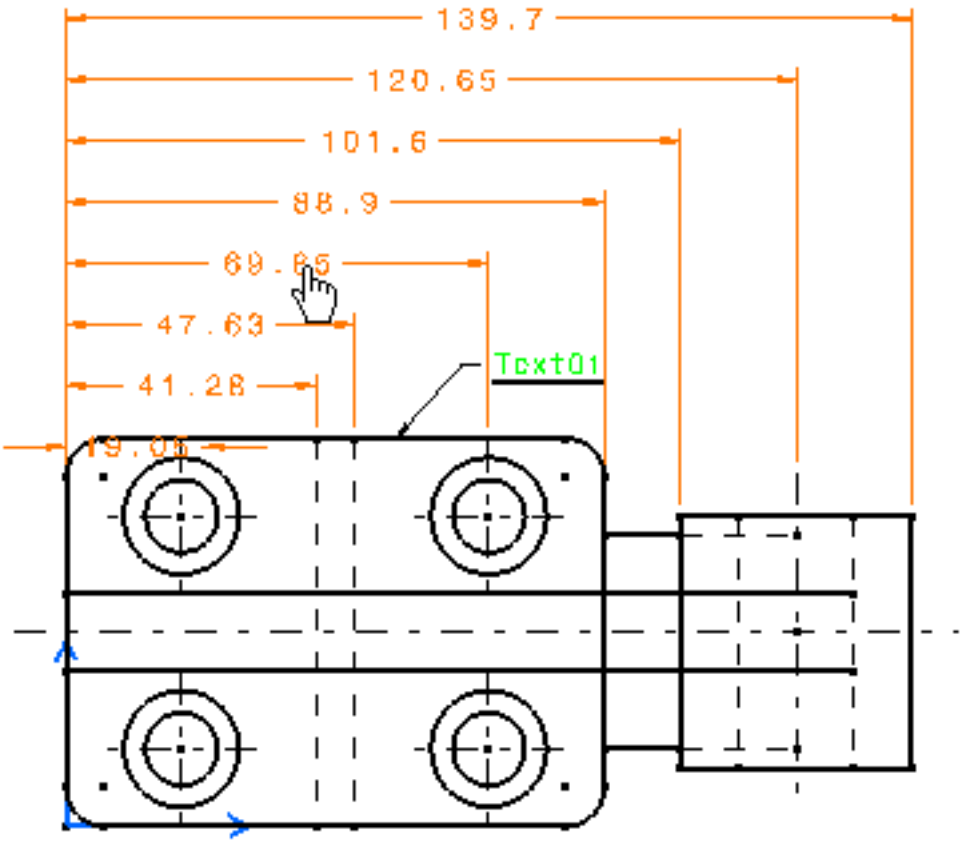
1. Create a stacked dimension system including several dimensions as shown below.



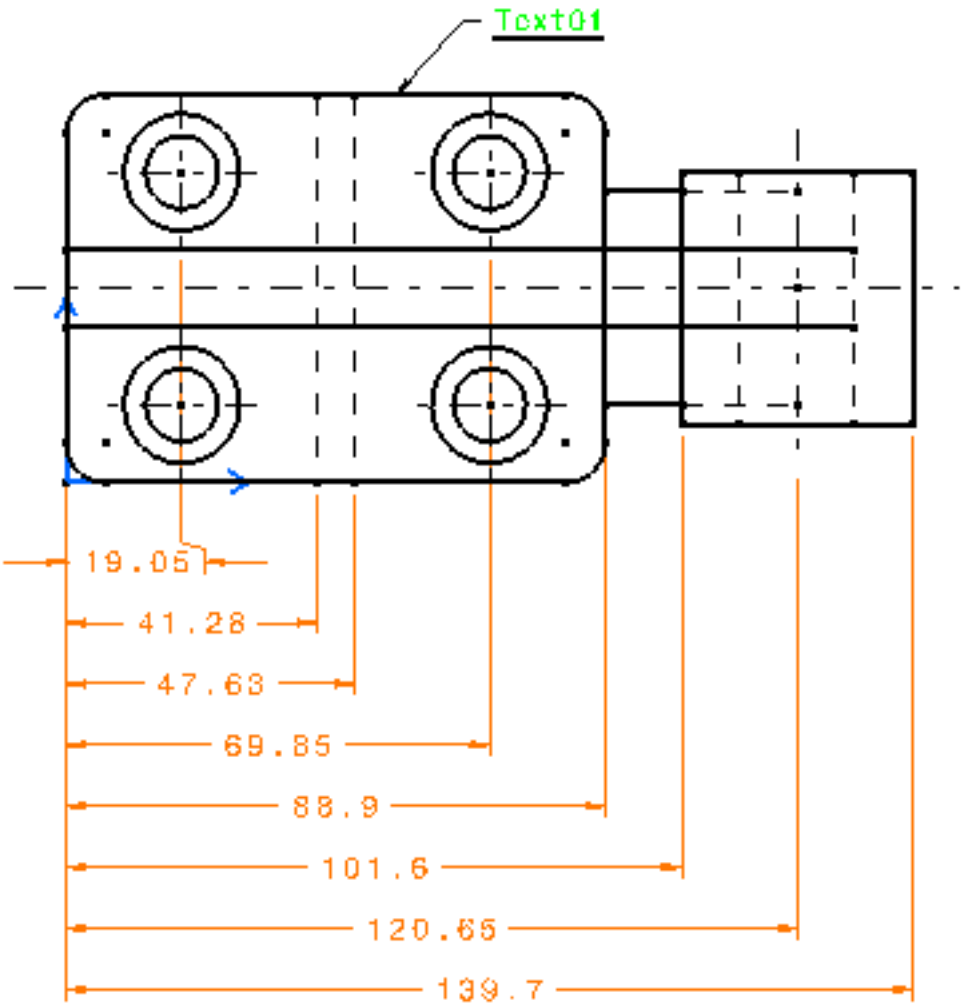
Moving a Dimension System

1. Click over the dimension system to select it.

   The whole system is highlighted.



2. Drag the whole system below the drawing.

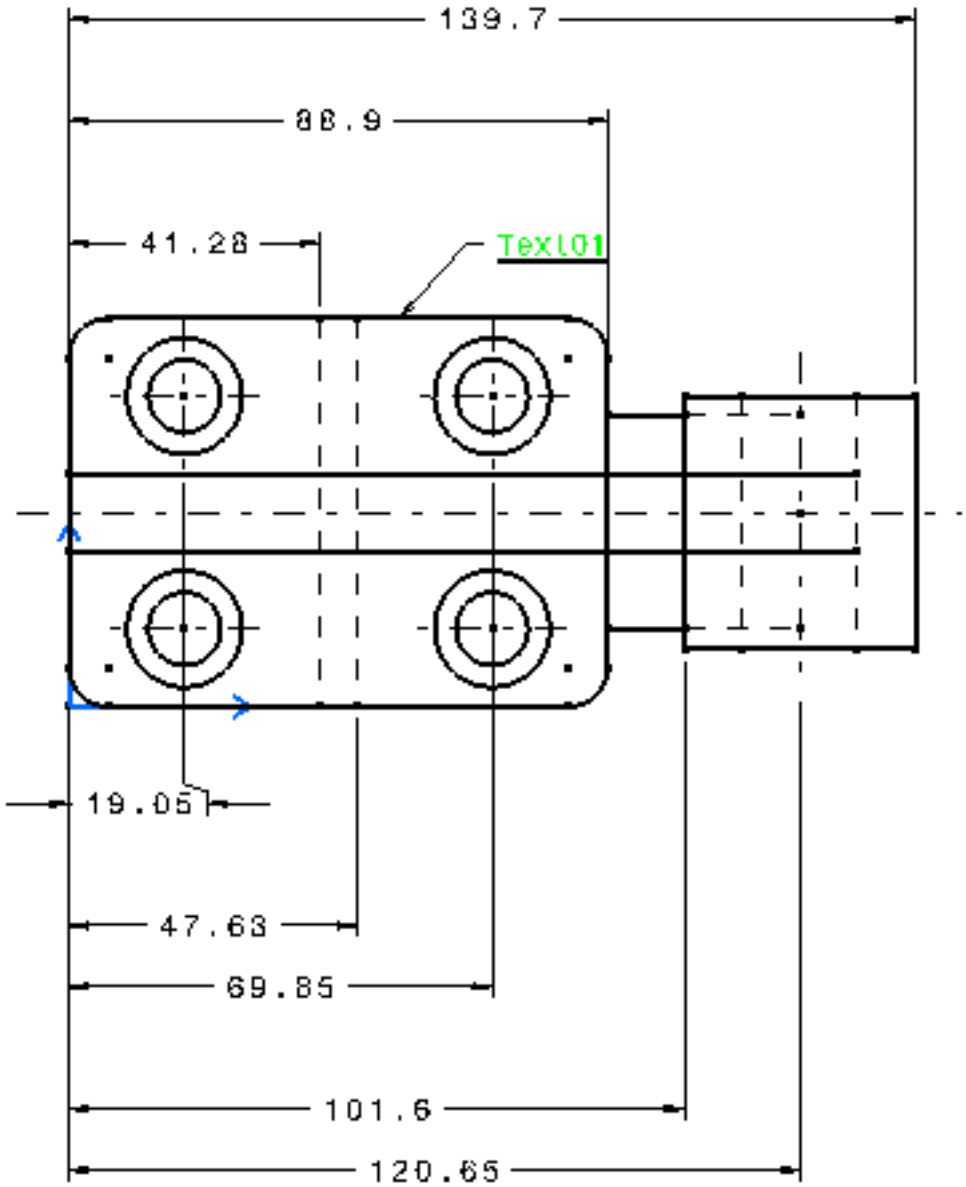


3. Click in the free space to end the stacked dimension system selection.

Moving a Dimension

1. Right-click on the Dimension system and select **Properties** in the contextual menu.
2. In the System tab, set the Offset mode to **Free**, then click on OK.

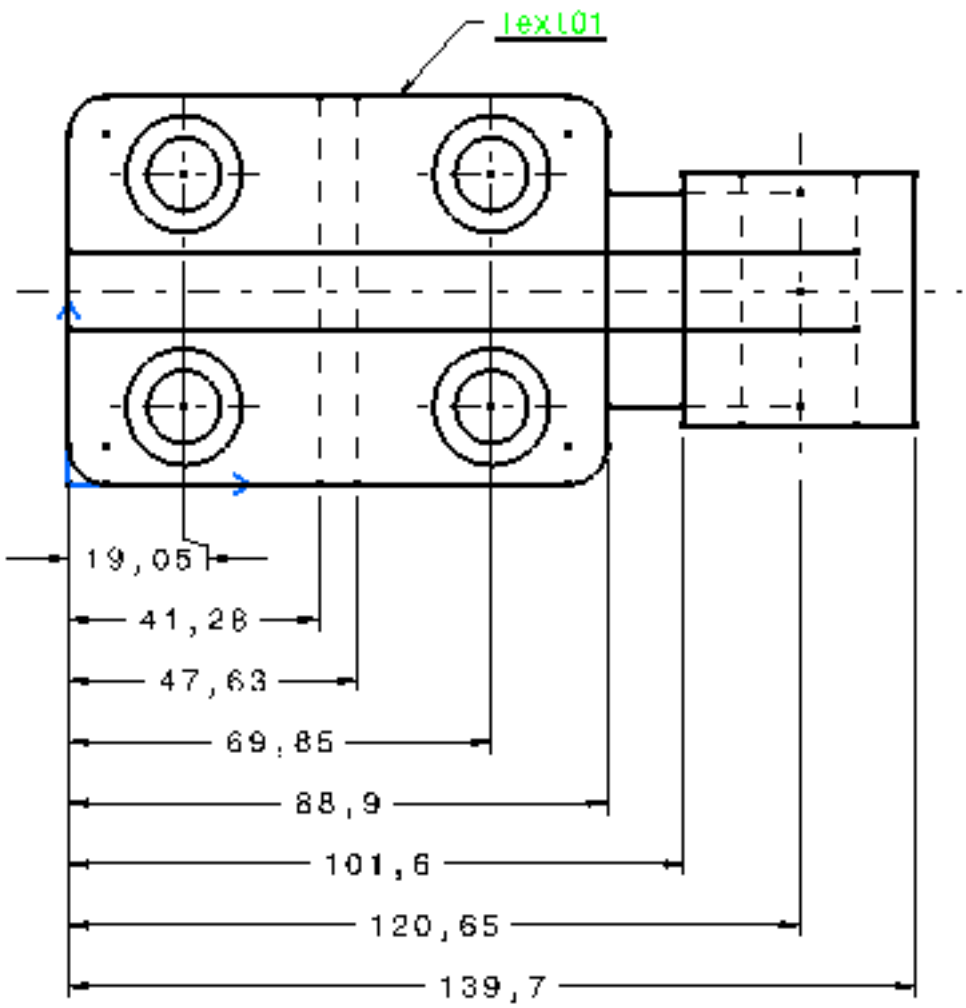
   See Editing Dimension System Properties for further information on the Properties' panel.
3. In the Tools toolbar, click on the Dimensions system selection mode icon to deactivate the dimension system.
4. Select several dimensions and drag them above the dimension system.
5. Click in the free space to end the dimension selection.



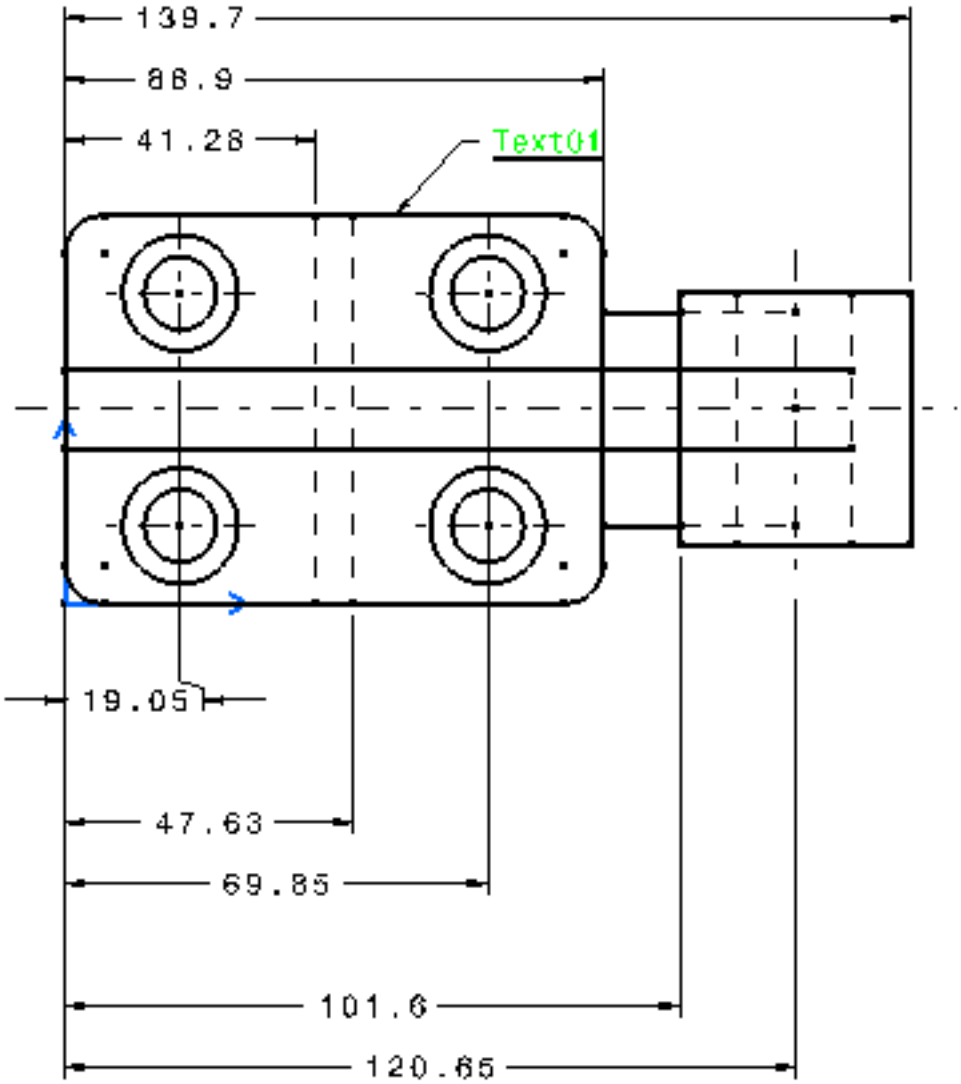
Moving a dimension can be performed only on stacked dimension system.

Aligning a Dimension System

1. Make sure Dimensions system selection mode icon is activated in the Tools toolbar.
2. Click over the dimension system to select it.
3. Right-click and select **Align into System**.
   The dimensions are aligned into the system as before.

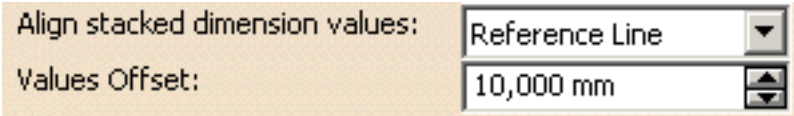


You can perform an **Align into System** on a whole system as well on several dimensions after multi-selecting them. To do so, set the system's offset mode to free (see Editing Dimension System Properties), then click on the Dimensions system selection mode icon to deactivate the dimension system. The dimensions you selected are aligned into a system of their own as shown below.

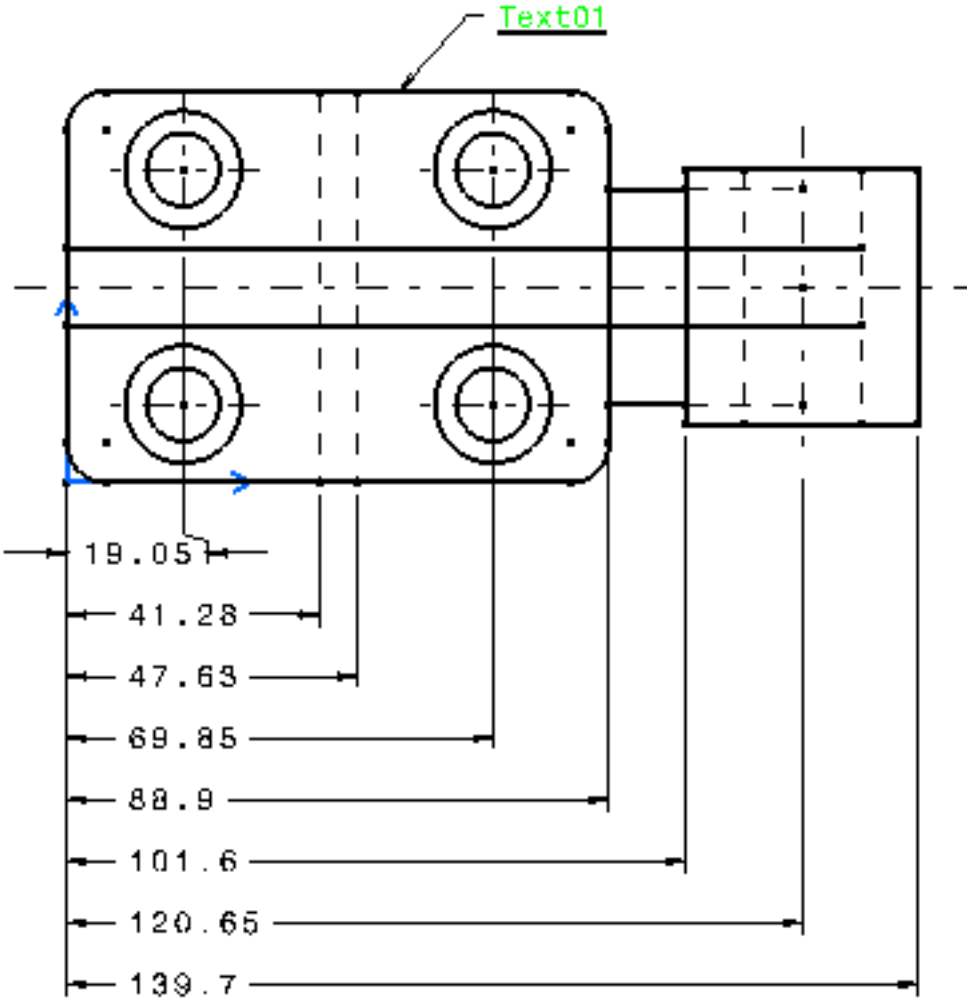


Restoring a Value Position

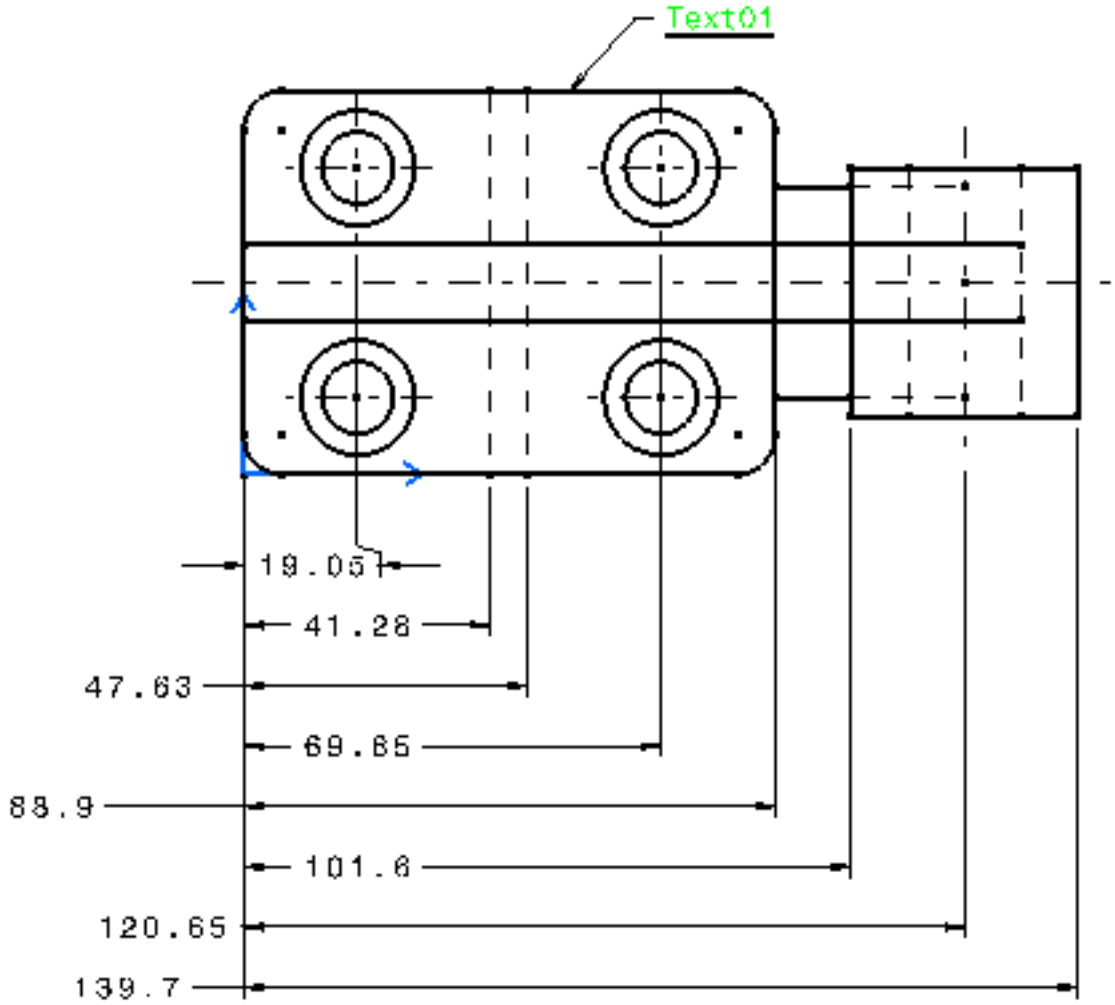
1. Make sure Dimensions system selection mode icon is activated in the Tools toolbar.
2. Right-click on the Dimension system and select **Properties** in the contextual menu.
3. In the System tab, set the Dimension values alignment at 10mm from the Reference line.



4. Click on Apply to visualize the Dimension values alignment.



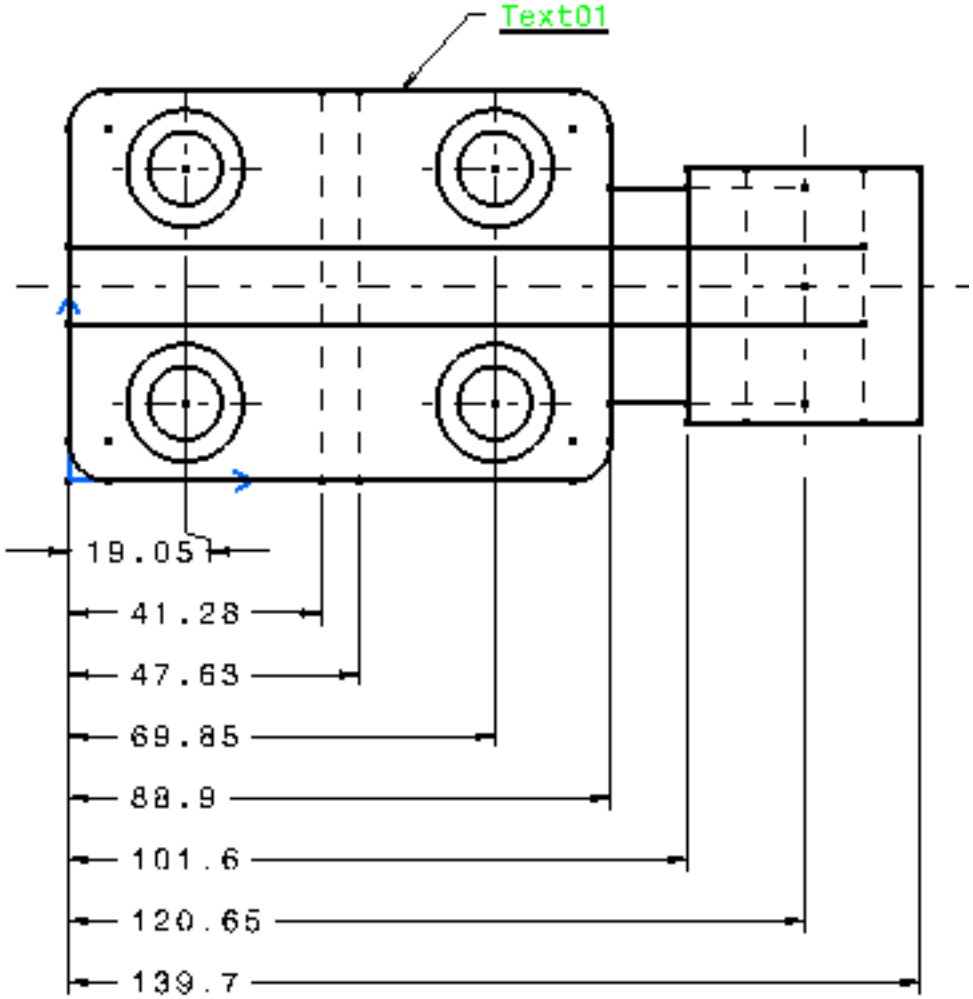
5. Click on the Dimensions system selection mode icon to deactivate the dimension system.
6. Move some of the dimensions.



7. Click on the Dimensions system selection mode icon to activate the dimension system again.

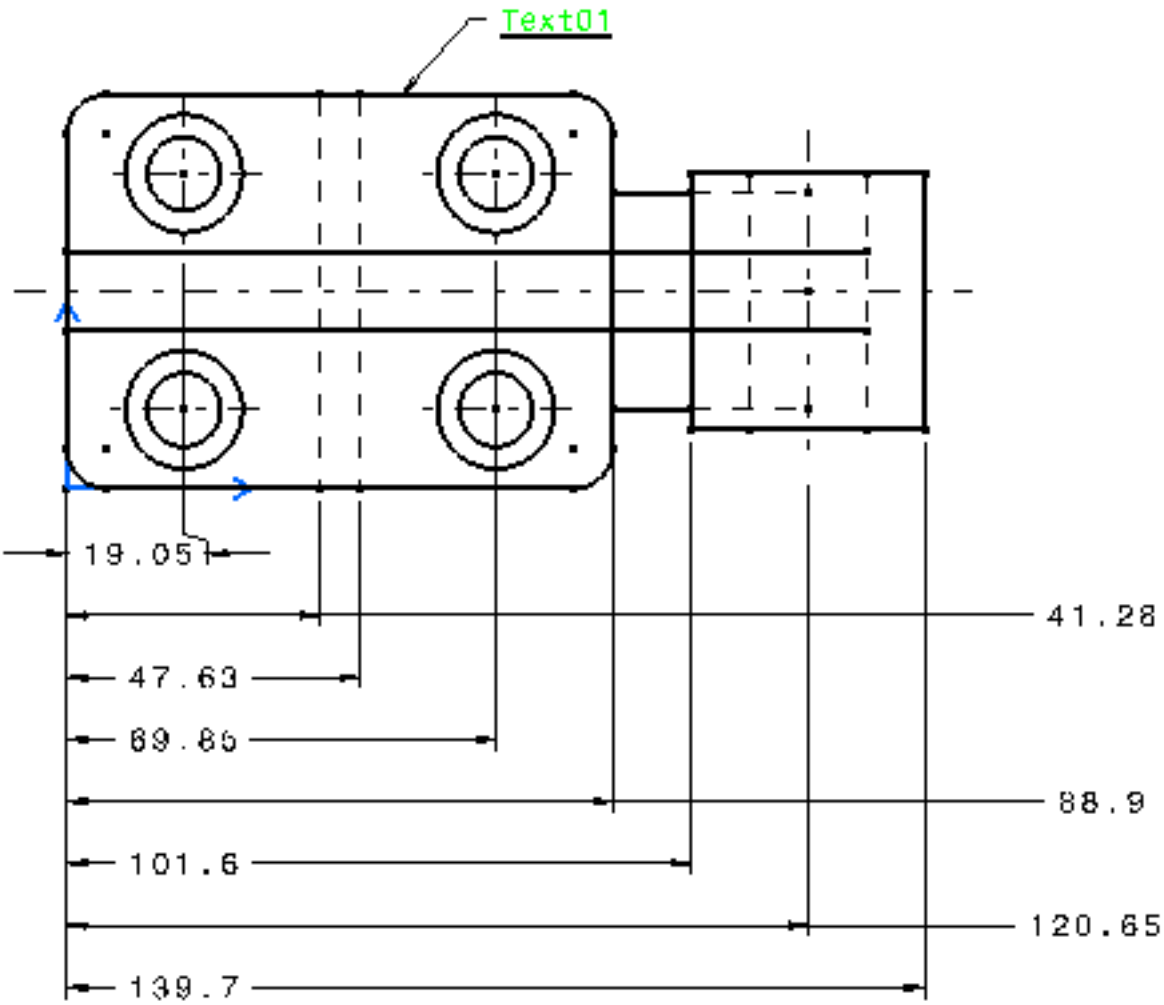
8. Right-click on the dimension system and select **Restore Value Position**.

   The dimensions' values are back to their initial position.



9. Click on the Dimensions system selection mode icon to deactivate the dimension system.

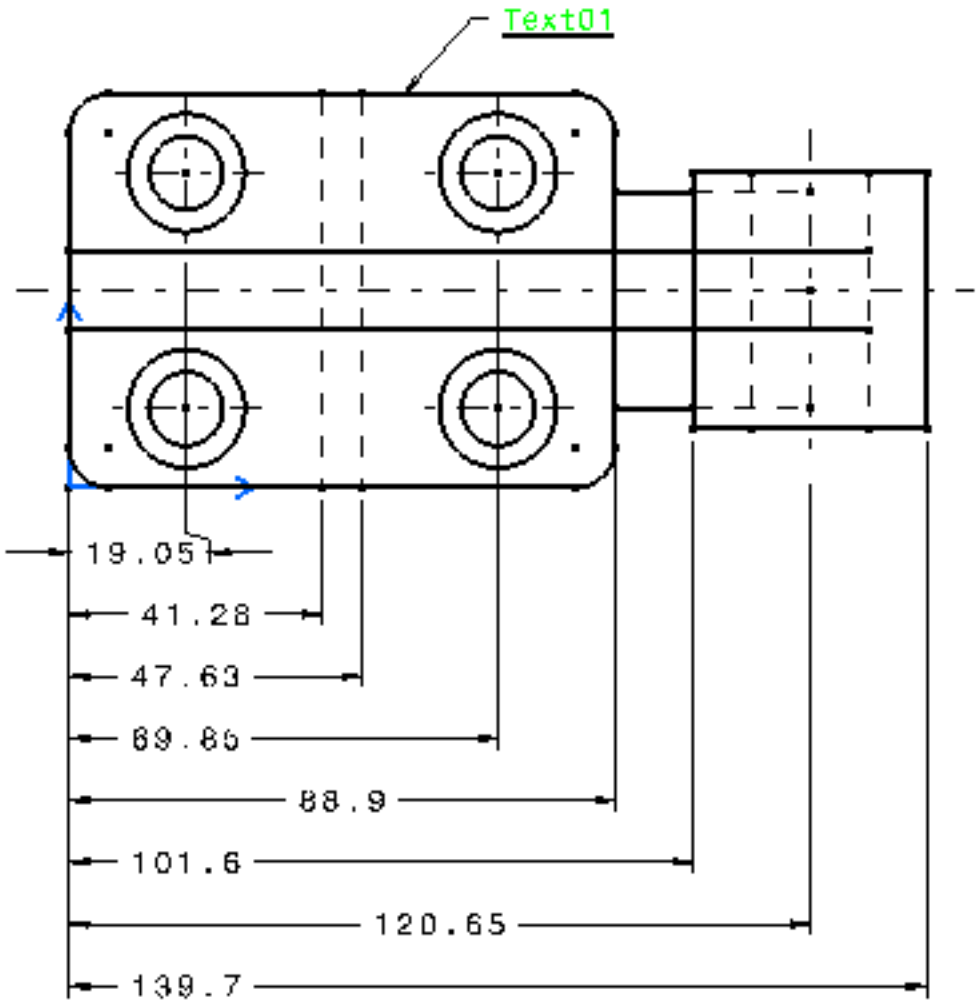
10. Drag several dimension values to modify their position.



11. Multi-select the dimensions.

12. Right-click and select **Restore Value Position** in the contextual menu.

    The dimensions' values are centered back in the system.

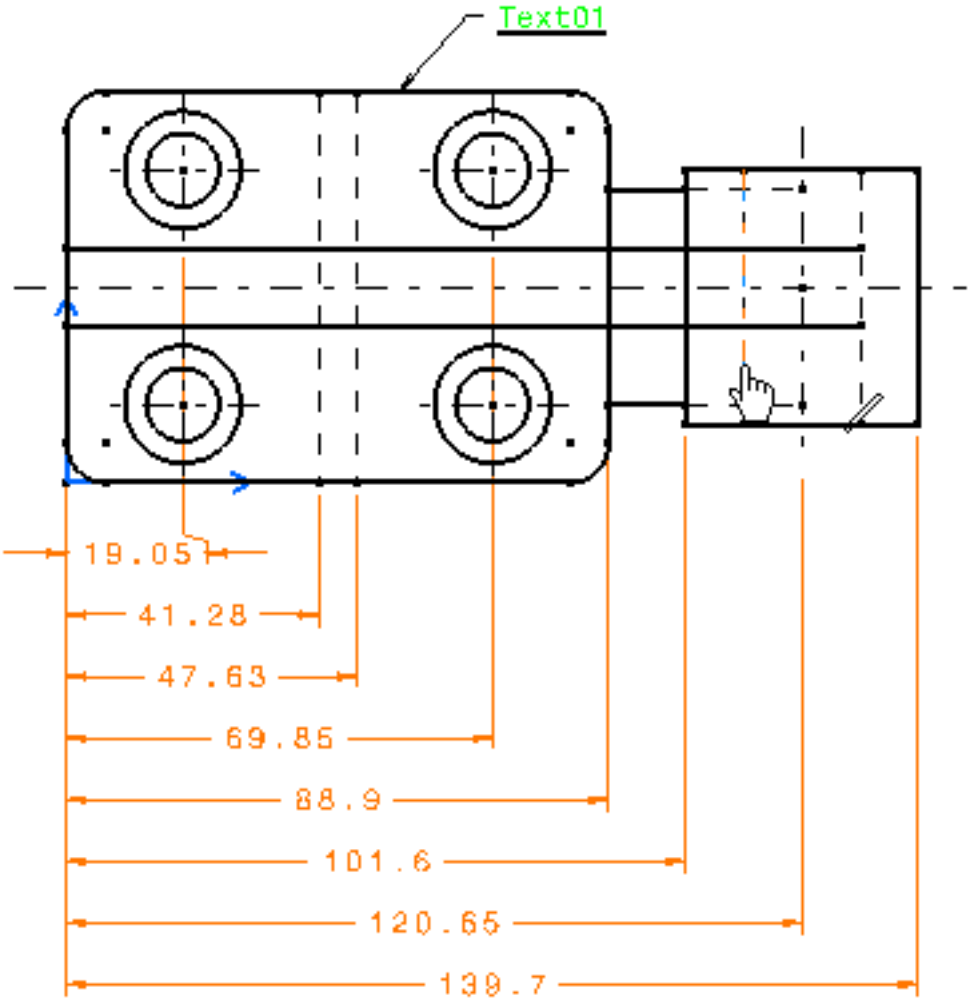


Adding a Dimension into a System

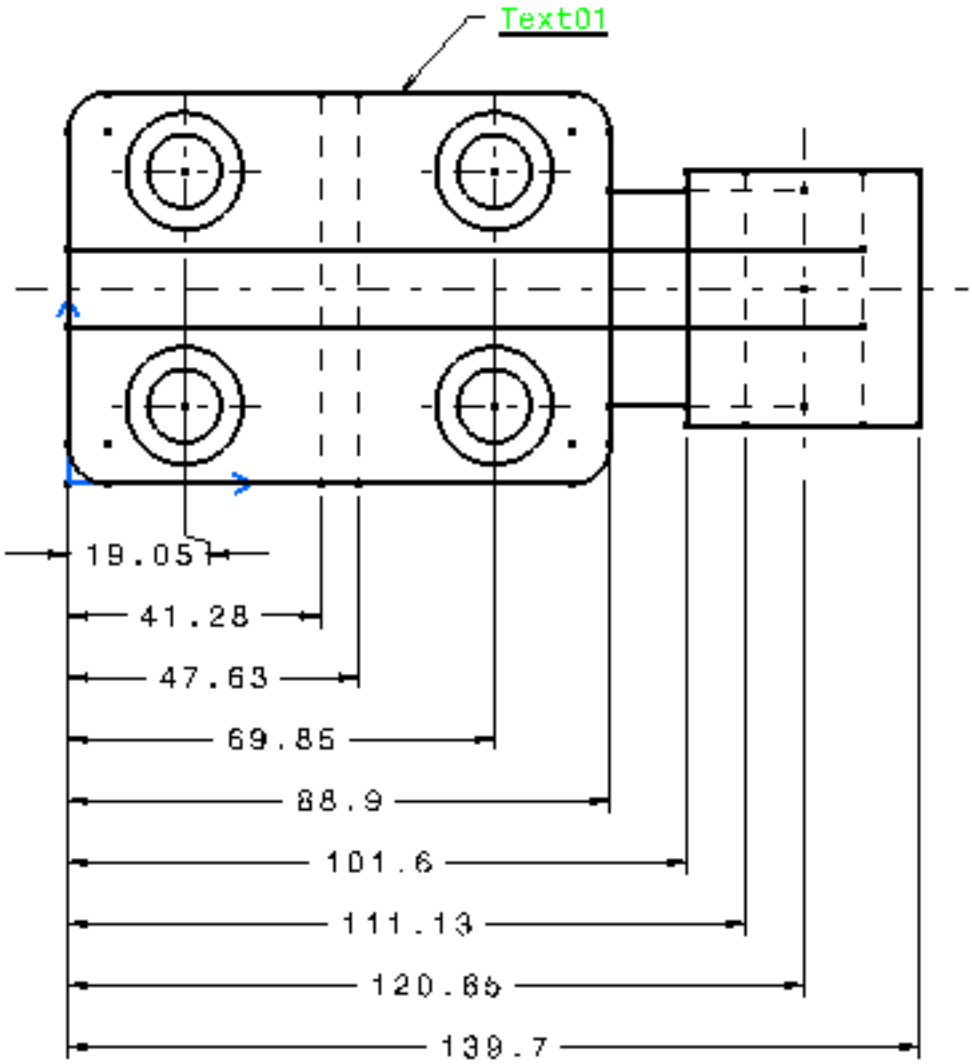
You have to select the icon corresponding to create the dimension system you would like to modify i.e. a chained, cumulated or stacked dimension system.

1. Make sure Dimensions system selection mode icon is activated in the Tools toolbar.
2. In the System tab, set the Offset mode to **Constant**, then click on OK.

   See Editing Dimension System Properties for further information on the Properties' panel.
3. Go to menu **Insert->Dimensioning->Dimensions** and select **Stacked dimensions.**
4. Select the system you want to insert a dimension to.
5. Select the geometry to dimension.



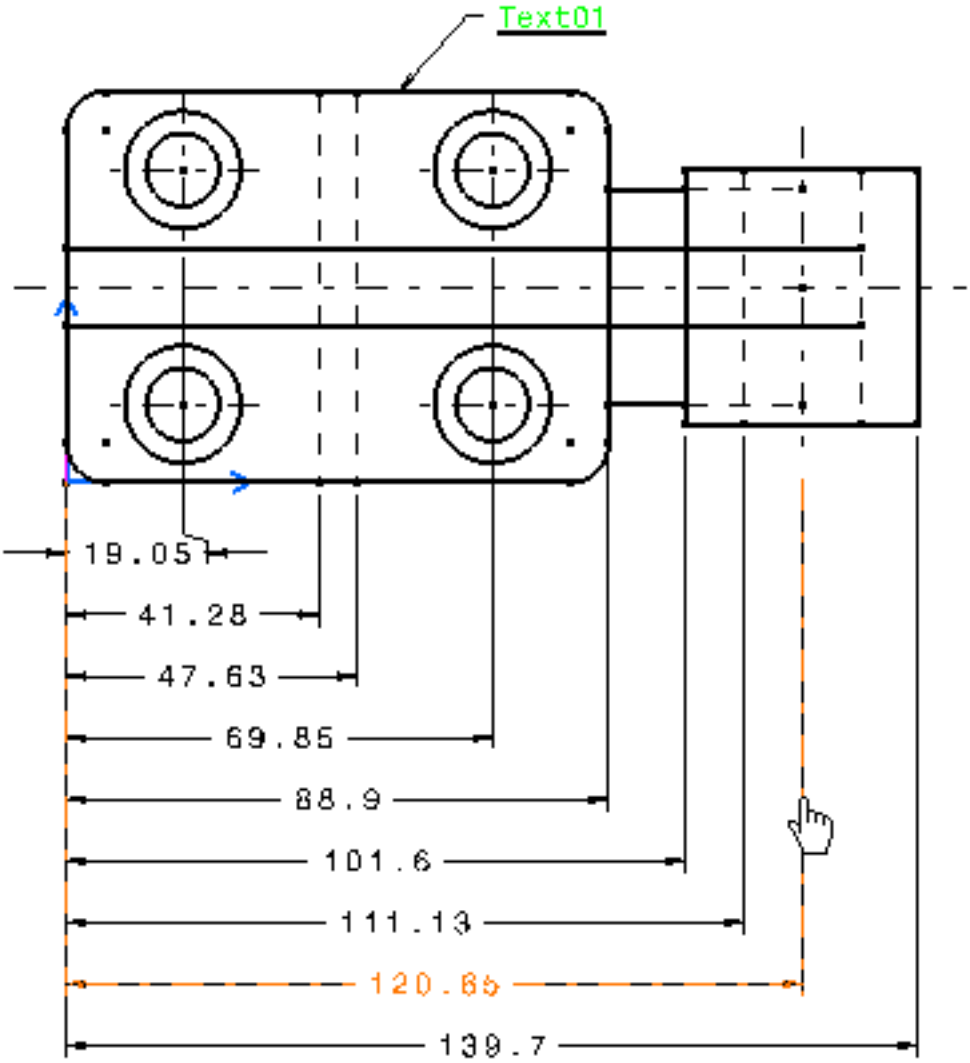
The new dimension is inserted into the system and is automatically aligned into the system.



Deleting a Dimension from a System

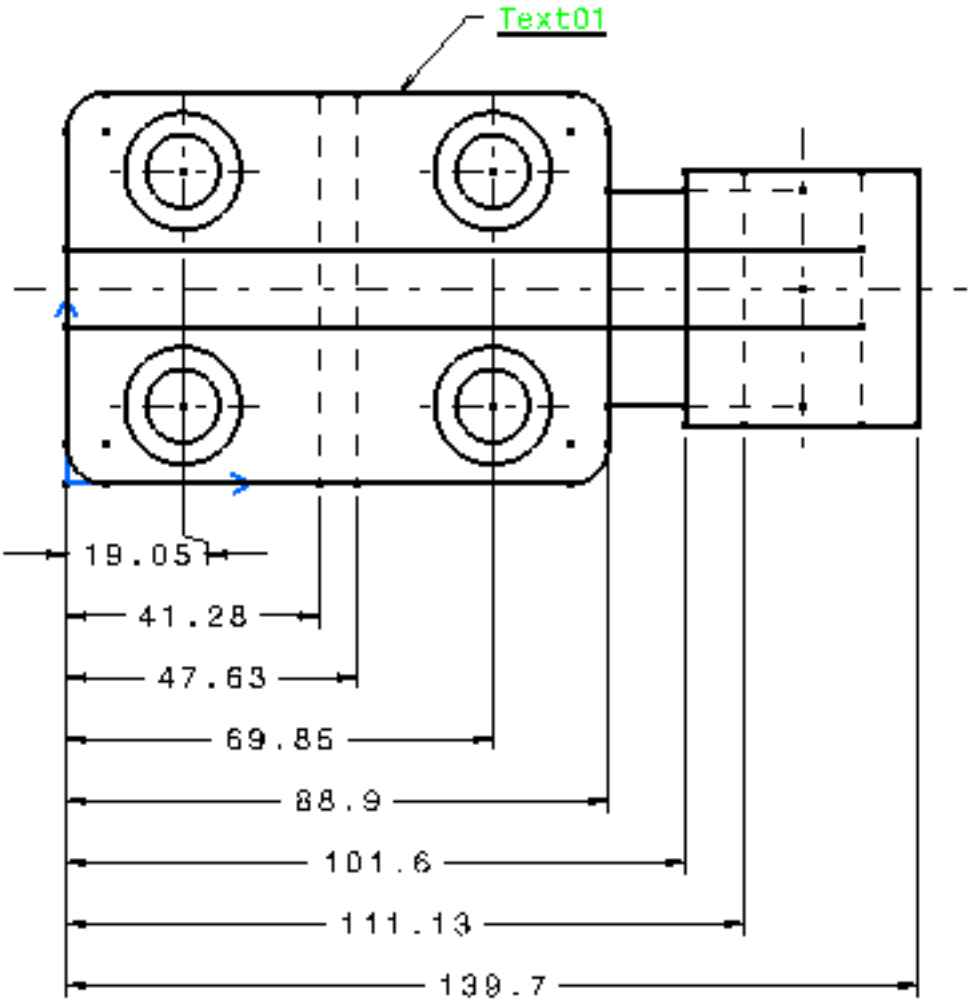
1. Make sure Dimensions system selection mode icon is deactivated in the Tools toolbar.
2. Select a dimension.

   Note that only the selected dimension is highlighted, not the whole system.



3. Press **Del** key.

The dimension system is updated accordingly automatically aligned.



Should you want to delete the whole dimension system, make sure Dimensions system selection mode icon is activated in the Tools toolbar and press **Del** key.

Copying a Dimension System

1. Make sure Dimensions system selection mode icon is activated in the Tools toolbar.
2. Select the geometry and its associated dimension system and click on the Copy icon.
3. Create a new view and select it.
4. Click on the Paste icon:

   The geometry and its dimension system is pasted in the view.

If the dimension system offset mode is set to **Free**, modifying one or several dimensions of the system will not impact the system 's alignment.

If the dimension system offset mode is set to **Constant**, an automatic line-up is applied to dimension lines and values in case you perform any of the following action:

- Adding a dimension to a system
- Deleting a dimension to a system
- Updating the 3D geometry
- Applying a style to a system

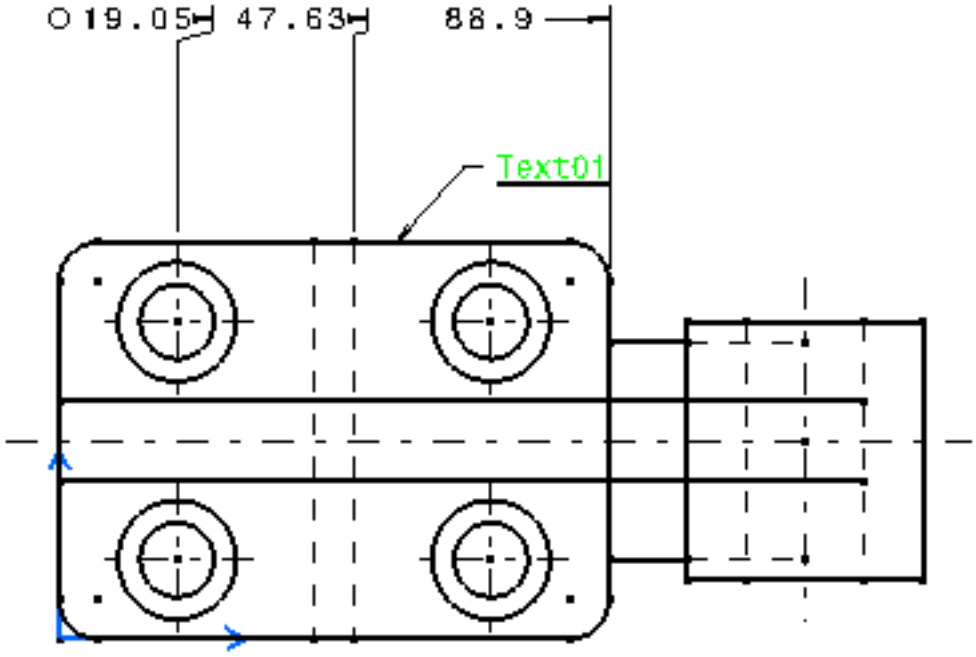
For more information on the Constant and Free Offset mode, refer to the Editing Dimension System Properties section.

Lining Up Dimension Systems

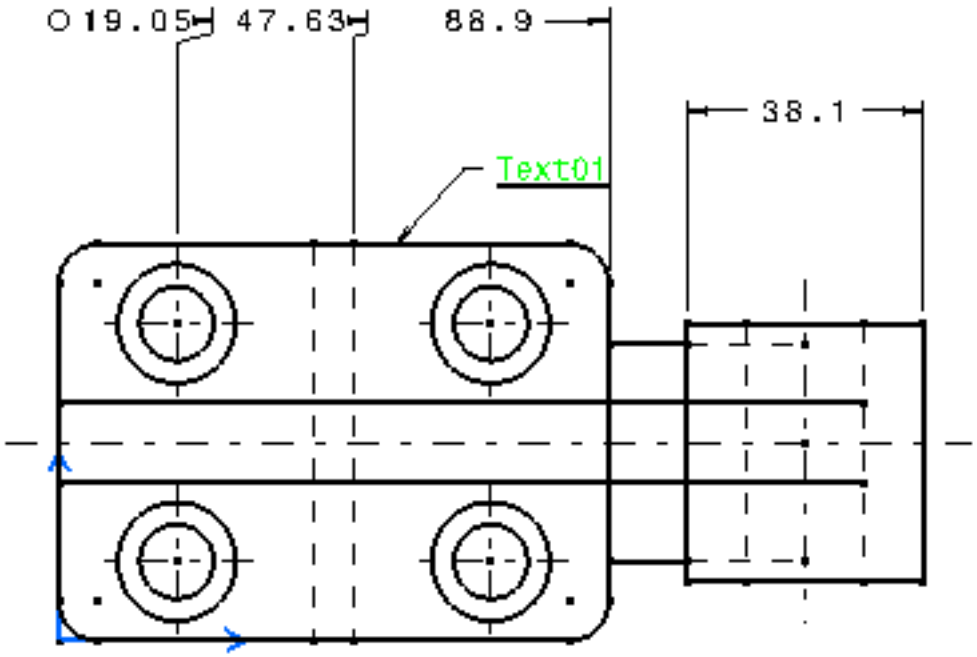
This task will show you how to line-up a dimension system.

Open the Brackets_views03.CATDrawing document.

1. Create a cumulated dimension system including several dimensions as shown below.

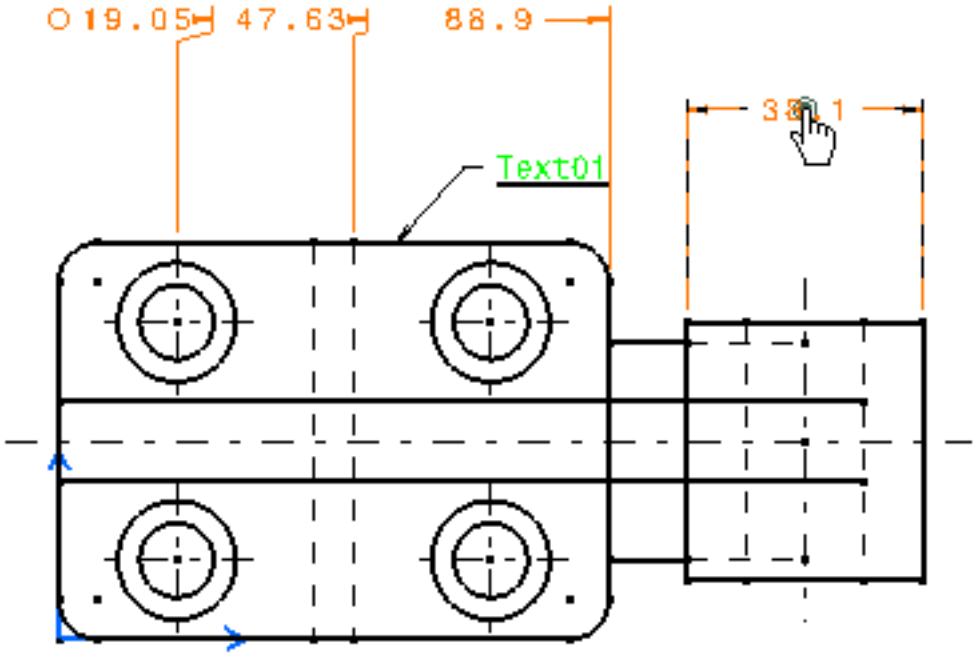


2. Click on the **Dimension** icon to create a dimension.

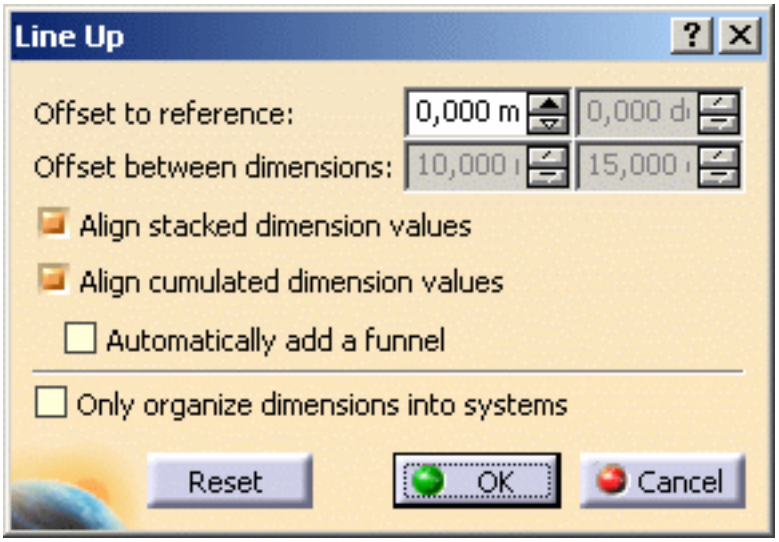


3. Right-click on your dimension system and select **Line-up** in the contextual menu.

4. Position the mouse on the dimension you just created to align the dimension system on it.



The **Line Up** dialog box is displayed.

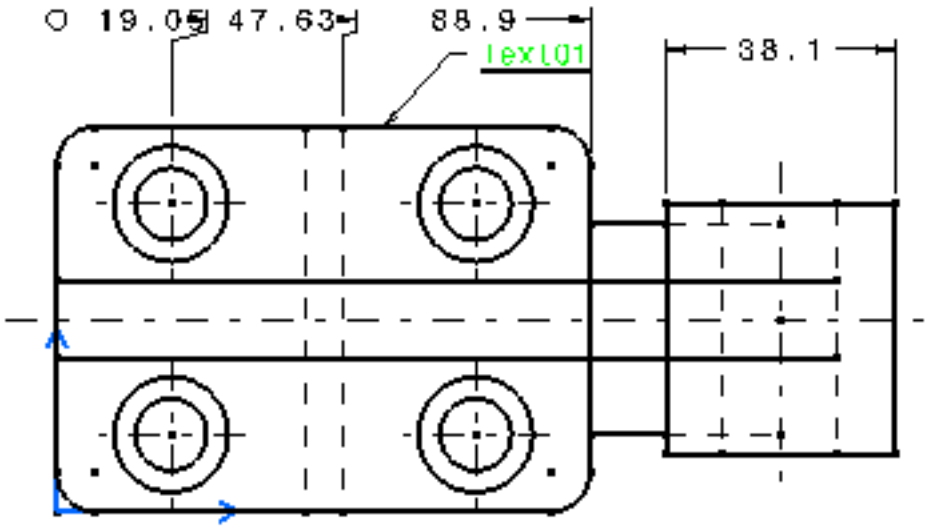


For a dimension system only **Offset to reference**, **Align stacked dimension values** and **Align cumulated dimension values** are taken into account.

5. Modify the **Offset to reference** value to 5mm.

6. Click on **OK**.

The dimension system is aligned on the dimension with an offset of 5mm.



The dimension system is aligned according to its dimension system properties.

Constraints

The 2D Layout for 3D Design workbench lets you create geometrical constraints, which specify explicitly how the geometry should behave.

As constraint commands work as in the Interactive Drafting and Sketcher workbenches, the tasks included in this section provide links to the *Interactive Drafting User's Guide* or to the *Sketcher User's Guide*. As such, the information detailed in these tasks is presented in the context of these workbenches. You should note that these user's guides contain images that correspond to the related workbench and therefore illustrate geometry in an environment that is different from the 2D Layout for 3D Design environment (symbols and background color, for example).

Before you begin creating constraints in 2D Layout for 3D Design

Before you begin creating constraints in 2D Layout for 3D Design, remember the following points:

- A constraint applies to up to three elements.
- Constraints are created either through the constraint commands or via SmartPick.
- When you use SmartPick, you detect geometrical constraints dynamically. But SmartPick can simply be used to automatically detect constraints without necessarily creating them. For more information, refer to the SmartPick task in the *Sketcher User's Guide*.
- In the 2D Layout for 3D Design workbench, dimensional constraints do not exist as such. It is by creating driving dimensions that you can drive constrained geometry.
- If you want constraints to be created, make sure the **Show Constraints** icon , and optionally the **Create Detected Constraints** icon , are active in the **Tools** toolbar, before you start creating constraints. Refer to Layout Tools for more information on these options.

Creating Quick Constraints (task documented in the *Interactive Drafting User's Guide*): quickly set geometrical constraints.

Fixing Elements Together (task documented in the *Sketcher User's Guide*): constrain a set of geometrical elements; once constrained, the set is considered as rigid and can be easily moved just by dragging one of its elements.

Creating Constraints via a Dialog Box (task documented in the *Interactive Drafting User's Guide*): set geometrical constraints via a dialog box.

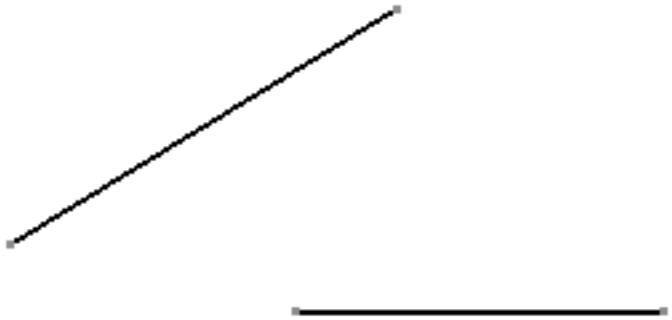
Creating a Contact Constraint (task documented in the *Sketcher User's Guide*): apply a constraint with a relative positioning that can be compared to contact.

Creating Constraints via SmartPick (task documented in the *Interactive Drafting User's Guide*): detect, create and visualize constraints using SmartPick.

Creating Quick Constraints

This task shows you how to create geometrical constraints quickly.

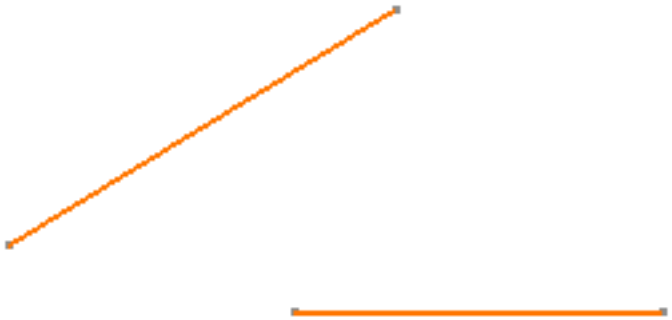
Create two lines.



Make sure the **Show Constraints** icon is active in the **Tools** toolbar.

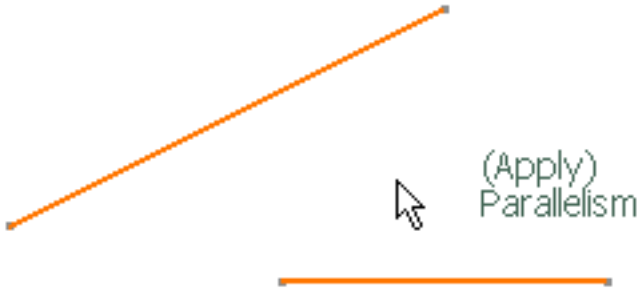
For the purpose of this scenario, also make sure that the **Create Detected Constraints** icon is active in the **Tools** toolbar: this option creates lasting constraints (if you do not activate this icon, the geometry is only temporarily constrained, which means that it can subsequently be moved without being constrained.).

1. Select the geometrical elements to be constrained to each other. For the purpose of our scenario, select the two lines you created.

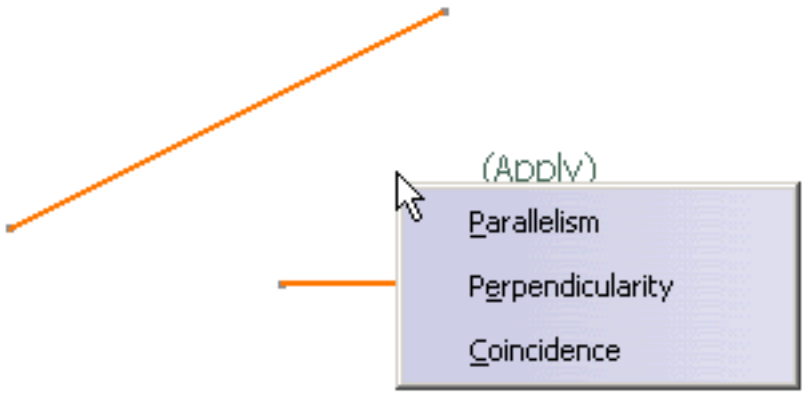


2. Click the **Geometrical Constraint** icon from the Geometry Modification toolbar.

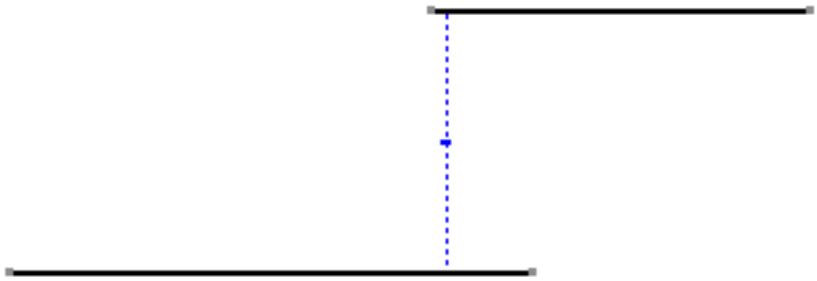
Based on the elements you selected, the software automatically offers to create a parallelism constraint, as shown at the tip of the cursor.



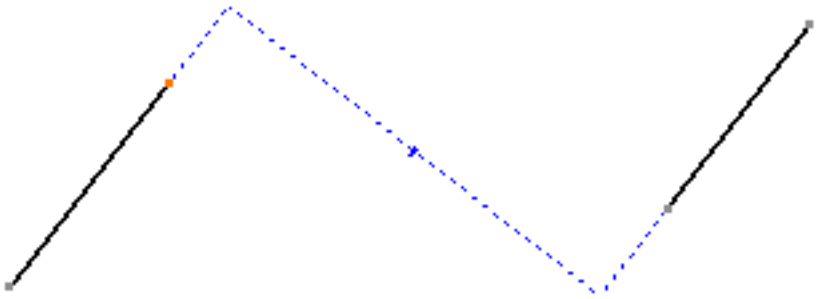
3. At this time, you can right-click on the drawing, to display a contextual menu offering the other types of constraints available for the selected elements.



For the purpose of the scenario, simply click on the drawing to accept the parallelism constraint. Both lines are now constrained as parallel to each other.



4. Modify the position of one of the lines, by moving one of its end points, for example.



As you can see, the lines are constrained so as to remain parallel to each other, whatever the new position and/or length you assign to one of them.

Even though you set a constraint relation between two elements, constraints are not necessarily visualized. If you cannot visualize constraints even though the Show Constraints option is active in the Tools toolbar, go to **Tools -> Options -> Mechanical Design -> Drafting -> Geometry** tab and select **Display Constraints**. (You can also modify the constraint color and/or width.)

Fixing Elements Together

In this task, you are going to attach sketcher elements together by using the Fix Together command. This capability lets you constrain a set of geometric elements even if constraints or dimensions are already defined for some of them. Once constrained, the set is considered as rigid and can be easily moved just by dragging one of its elements.

One of the interest of this capability is that it also allows you to make 2D kinematics studies in the Sketcher.

This task shows you how to make two elongated holes perpendicular, then how to position them inside a rectangle, while using the Fix Together command.

- Making Elongated Holes Perpendicular (scenario)
- Selecting Geometrical Elements
- Degrees of Freedom
- Positioning Holes in the Rectangle (scenario)
- Editing a Fix Together Constraint
- Additional Constraints
- Removing Geometrical Elements
- Applying Operations onto a Fix Together Constraint
- Methodology

Enter the Sketcher workbench and create a rectangle and two non-constrained elongated holes next to it.

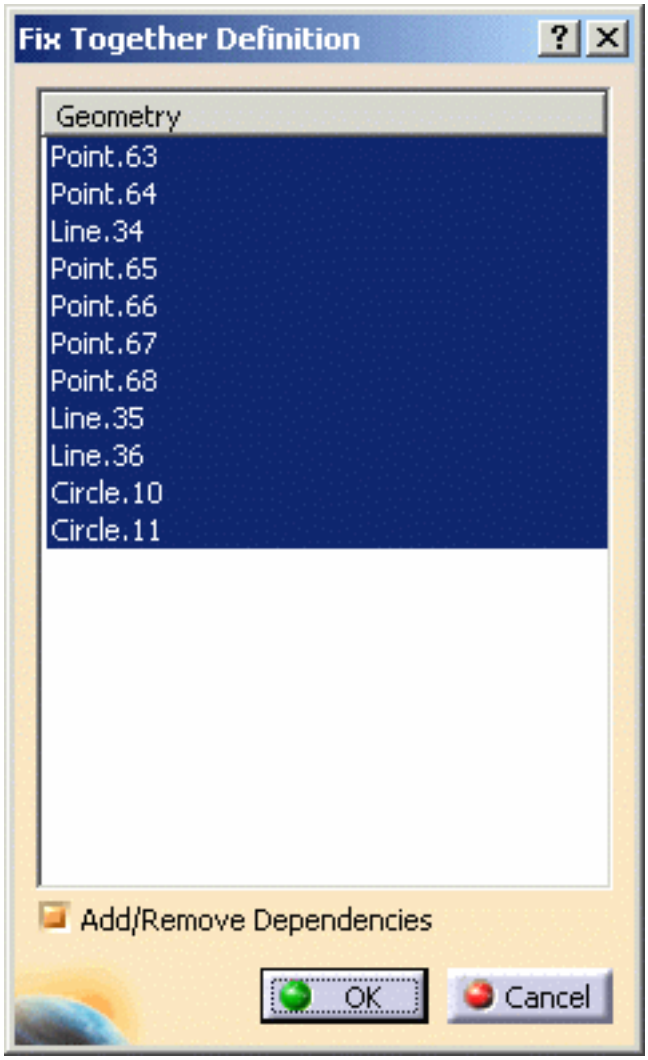
Making Elongated Holes Perpendicular

Prior to using the **Fix Together** command, consider the following scenario: to make both elongated holes perpendicular to each other, you could be tempted to select one oblong hole then drag it next to the second one, and eventually set a perpendicular constraint. The fact is that setting the constraint if no other constraints are set, deforms the holes. To quickly achieve the desired geometry, follow the steps as explained below.

1. Select one elongated hole by using the selection trap.

2. Click the **Fix Together** icon .

   The **Fix Together Definition** dialog box that appears displays all selected geometrical elements.



Selecting Geometrical Elements

Dependencies

To assume that the rigid body behavior can be managed, by default the application includes element dependencies. This is indicated by the activated **Add/Remove Dependencies** button. When adding a spline for instance, all its control points and control point tangencies are automatically added even if they were not selected.

Note that you can deactivate this behavior for advanced uses cases by deselecting the **Add/Remove Dependencies** option.

The following table lists geometric elements and their corresponding dependencies:

Geometric Element	Dependency
Line	Start point + End point
Circle/Ellipse	Center point
Arc of circle/Ellipse	enter point + Start point + End point
Parabola/Hyperbola	Start point + End point
Conic by two points	Start point + End point + (Start Tangent curve + End Tangent curve or Tangent Intersection point) + (Passing point or Not)
Conic by four points	Four points + One Tangent curve
Conic by five points	Five points
Connect Curve	First point + Second point + First curve + Second curve
Spline	Control points + Tangent directions

Number of Elements

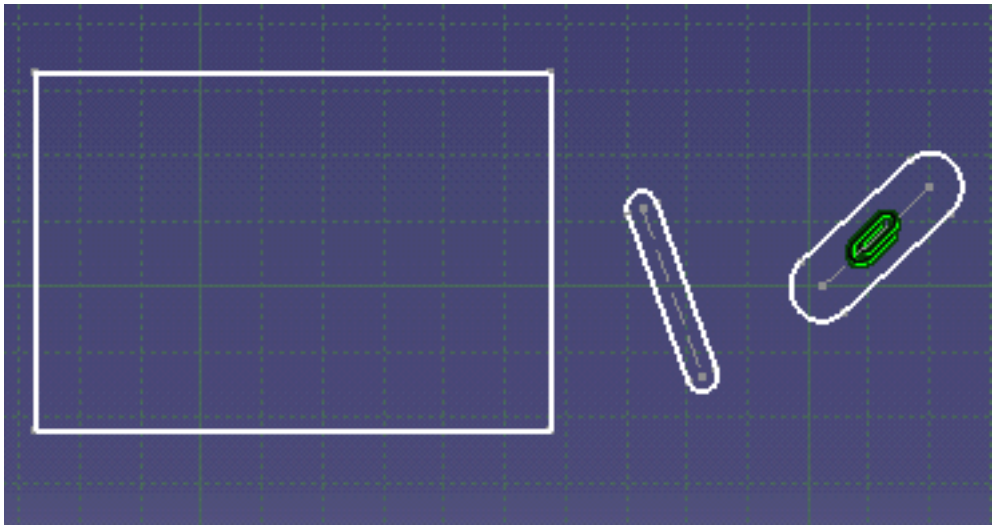
You can select as many geometrical elements as you wish, but just remember that a geometrical element can be used by only one Fix Together constraint.

Absolute Axis

You can select the origin, the H or V Direction of the sketch absolute axis. These three elements cannot be selected at the same time by a selection trap. You need to explicitly select them one by one.

3. Click **OK** to confirm.

The **Fix together** constraint is created as indicated by a green paper clip symbol.



Degrees of Freedom

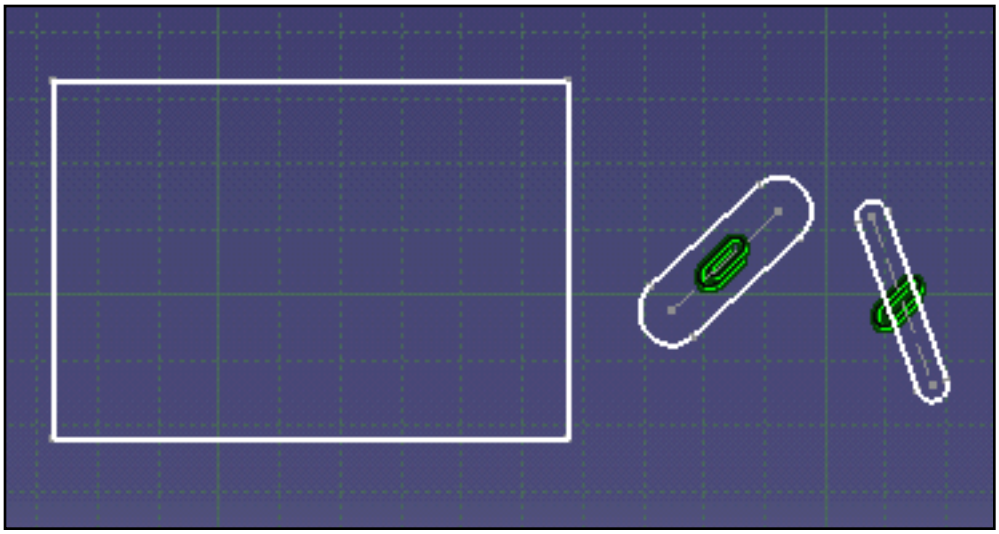
The set of geometric elements constrained by **Fix Together** has three degrees of freedom whatever the number of elements. In order to be fully defined, the set needs to be dimensioned to fix geometry taking up at least the three degrees of freedom (one rotation and two translations).

A geometric element included in a set of elements constrained via **Fix Together** can also be constrained using . If, for instance, a Fix Together constraint contains a fixed line, the set of geometric elements has one single degree of freedom which is along the direction of the line.

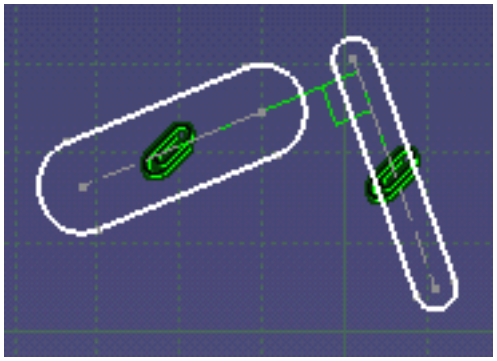
4. Repeat the operation for the second elongated hole.

Just to check that you can now manipulate each hole by keeping its rigid body.

5. Select them and drag them to any location.



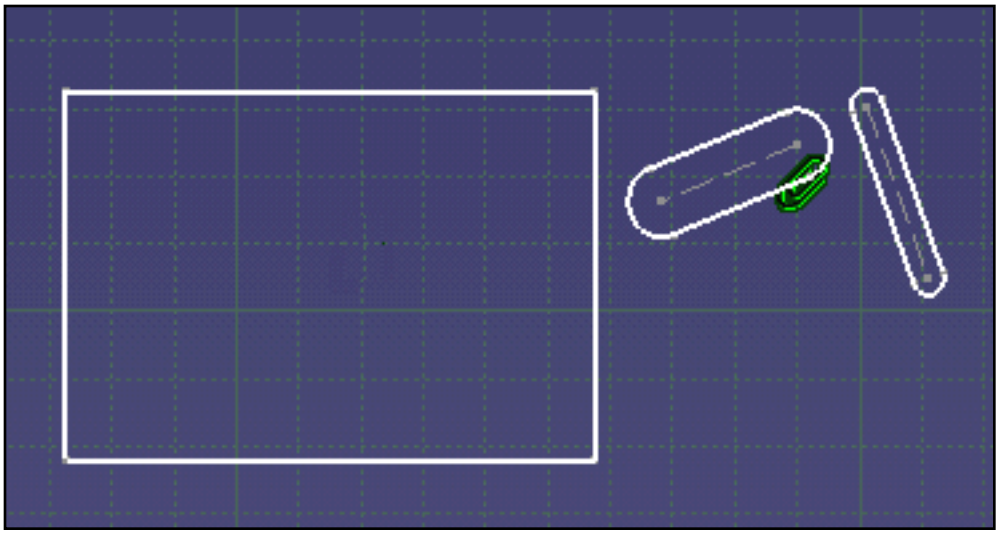
6. Set the perpendicular constraint.



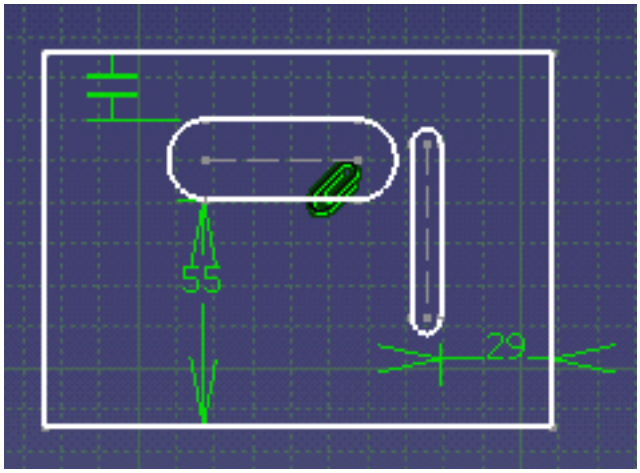
Positioning Holes in the Rectangle

7. To position both holes inside the rectangle, delete the constraints you previously set.

8. Create only one **Fix Together** for both holes.



9. Drag the holes all together inside the rectangle after selecting any of their geometrical element and add constraints between the rectangle and the holes to specify their exact positions.

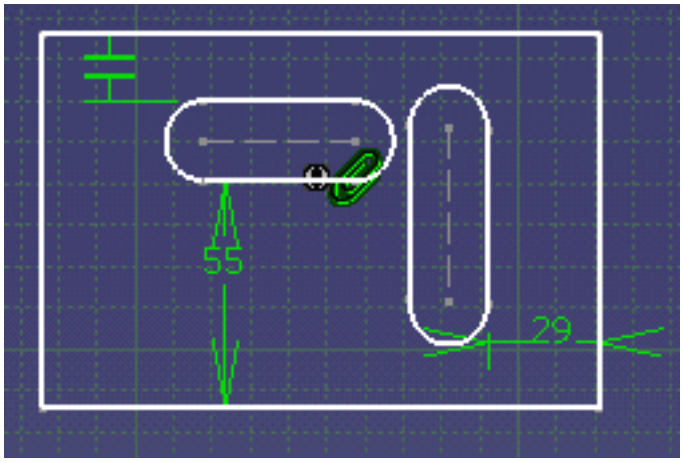


10. Select the **Fix Together** constraint attaching the holes and use the **Fix-Together.x object > Deactivate** contextual menu item.

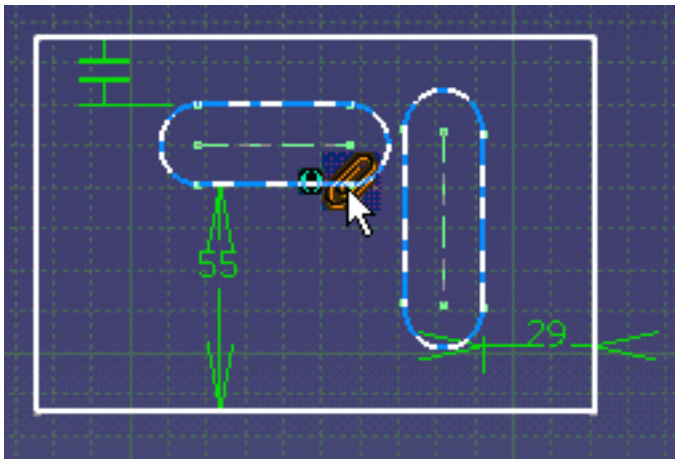
    Note that if the Fix Together constraint is deactivated, the geometric elements are always seen by the application as belonging to a rigid set. So selecting them remains impossible for defining another Fix Together constraint.

    You can now modify the shapes of the holes as the constraint is deactivated.

11. Enlarge the right hole.



12. Note that passing the cursor over an activated or not Fix Together constraint highlights the associated geometry.



Editing a Fix Together Constraint

You can add a geometrical element to a **Fix Together** constraint provided it belongs to the current sketch and is not already included in another Fix Together constraint. The selection or the pre-selection of the elements to add depends on this verification during the Fix Together constraint creation and editing.

You can add several elements at the same time either by using the CTRL key or the selection trap. After a selection:

- Geometrical elements not used for the definition are added
- Geometrical elements that are already part of the definition are removed.

Additional Constraints

- Adding constraints between elements involved in a Fix Together constraint and other elements involved too in a distinct Fix Together constraint or free elements allows you to position the fixed together set.

- Adding a constraint to a fixed together element brings about an over-constrained system. But unlike other types of constraints, when exiting the Sketcher, the application does not detect no inconsistency.
- All existing or added constraints on geometric elements of a Fix Together constraints are seen as over-defined (in purple when solving status is displayed).

Except for Fix constraints, no constraints are solved between geometric elements linked by the same Fix Together constraint. However, no update error appears on such over-defined constraints (between Fixed Together geometric elements) and the part is successfully updated.

Removing Geometrical Elements

There are two ways of removing geometrical elements from a Fix Together constraint:

- by deleting the geometrical elements: dependencies are deleted too.
- by editing the **Fix Together** constraint:
  - select the **Add/Remove dependencies** option and select the geometry to be removed: dependencies are deleted too.
  - unselect the **Add/Remove dependencies** option and select the geometry to be removed: dependencies are not deleted.

When the number of geometric elements in the set is less than two, the Fix Together constraint is NOT automatically deleted.

Applying Operations Onto Fix Together Constraint

Copy/Paste

You can copy and paste a fix together set (not the constraint alone). To do so:

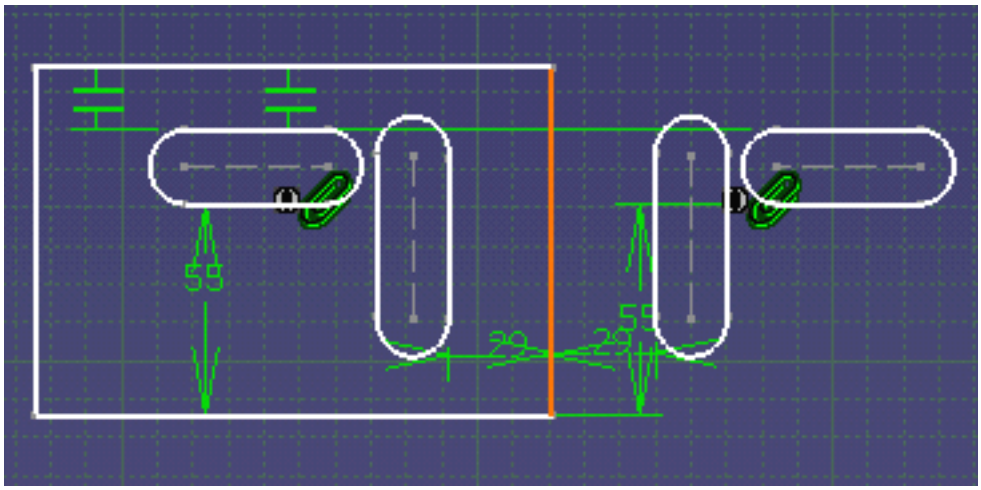
1. Select the paper clip.
2. Use the **Fix Together object > Select Geometrical Elements** contextual menu item.
3. Use Ctrl to include the paper clip in the selection.

   or

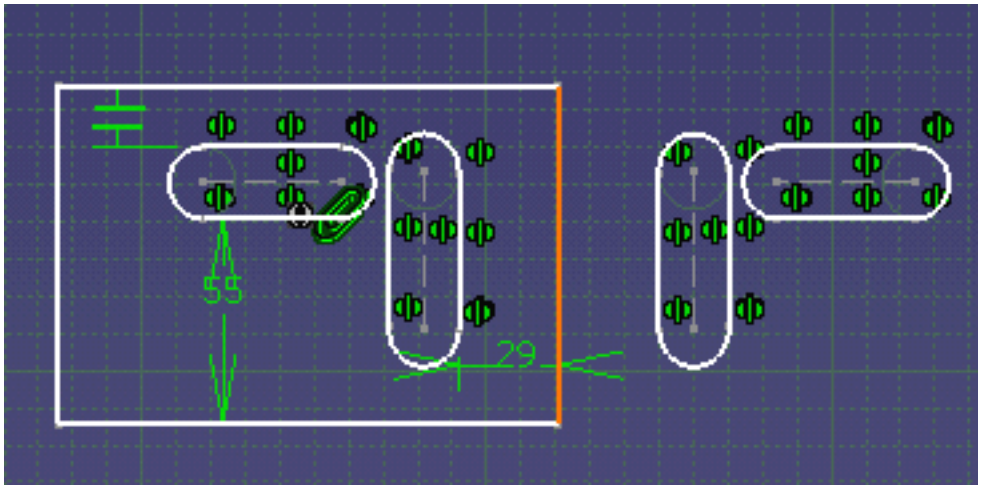
1. Use the selection trap to ensure that the paper clip and the associated constrained geometry are selected.
2. Apply the **Copy** > **Paste** capability.

Mirror

By switching off the Geometric constraints mode, the Fix Together constraint is taken into account like the other constraints when mirroring geometries and keeping the initial constraints:



...otherwise, the application creates symmetry constraints as requested.



Break/Trim/Corner/Chamfer

You can apply the **Break** , **Trim** , **Corner** and **Chamfer** commands onto elements attached by a fix together constraint.

When all the geometrical elements belong to the same Fix Together constraint, the constraint is updated accordingly. For instance, when breaking a curve, the new half curve is automatically added to the definition.

Methodology

Depending on your geometry and your needs, you will use the **Fix Together** command or the the **Auto Constraint** command, bearing in mind that:

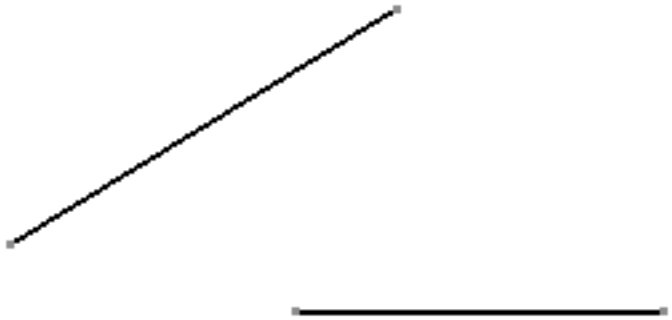
- The **Fix Together** command creates only one constraint for a group or elements.
- The **Auto Constraint** command detects all possible constraints between the selected elements then creates these constraints. This means that sometimes you may create a lot a unnecessary constraints just for imposing a rigid behavior. For more information, refer to Autoconstraining a Group of Elements in CATIA Sketcher User's Guide.

The **Fix Together** command is a way of getting better solving performances as well as solving more complex systems including rigid sub-parts.

Creating Constraints via a Dialog Box

This task shows you how to set geometrical constraints via a dialog box.

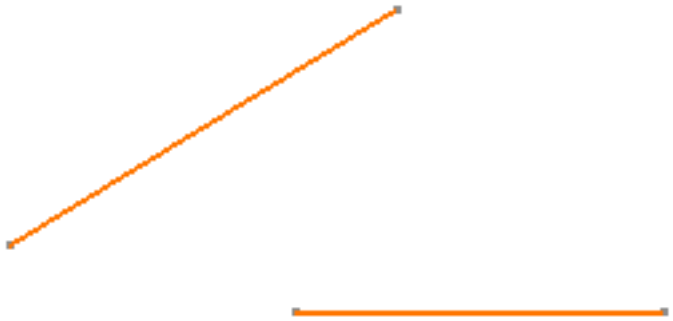
Create two lines.



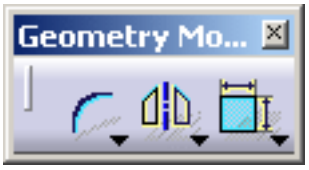
Make sure the **Show Constraints** icon is active in the **Tools** toolbar.

For the purpose of this scenario, also make sure that the **Create Detected Constraints** icon is active in the **Tools** toolbar: this option creates lasting constraints (if you do not activate this icon, the constraints you create are temporary: the geometry is only temporarily constrained, which means that it can subsequently be moved without being constrained.).

1. Select the geometrical elements to be constrained to each other. For the purpose of our scenario, select the two lines you created.

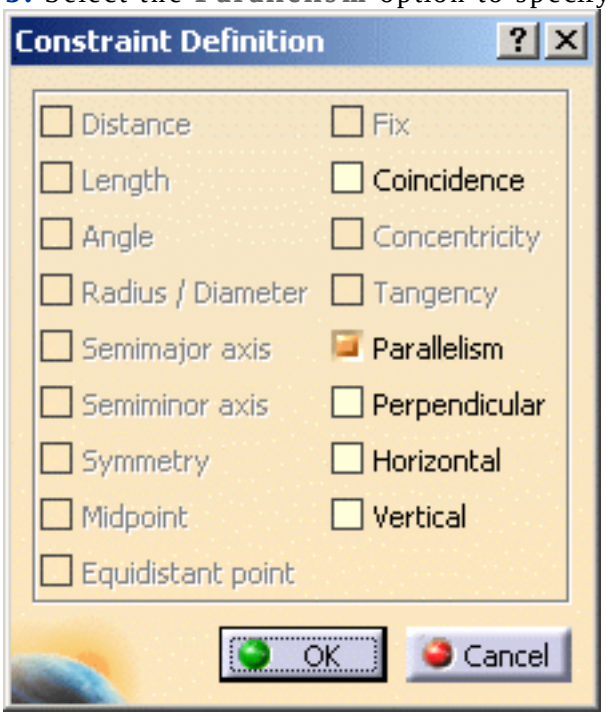


2. Click the **Constraint with Dialog Box** icon from the Geometry Modification toolbar.

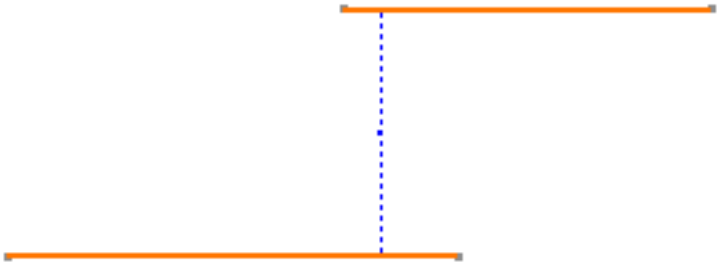


The Constraint Definition dialog box appears. The options corresponding to the various types of constraints you can create for the selected elements are active.

3. Select the **Parallelism** option to specify that the selected lines should be parallel.

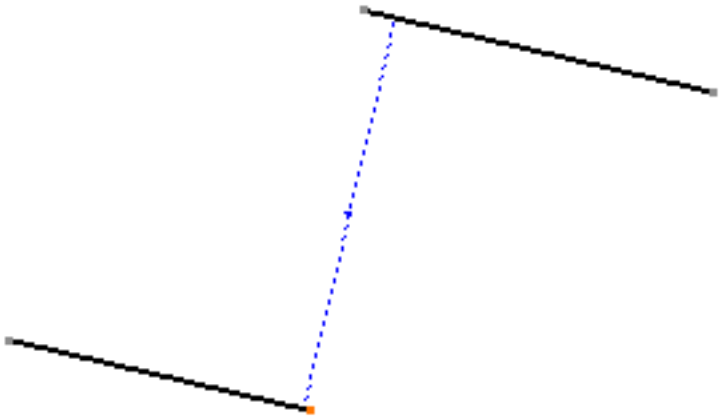


You can preview the result.



4. At this time, you can still select another option from the dialog box if you decide to apply another type of constraint. For the purpose of the scenario, simply click **OK** to validate. Both lines are now constrained as parallel to each other.

5. Modify the position of one of the lines, by moving one of its end points, for example.



As you can see, the lines are constrained so as to remain parallel to each other, whatever the new position and/or length you assign to one of them.

- It is impossible to create constraints between 2D and generated elements via the Constraint Definition dialog box. In the Constraint Definition dialog box, you can only create constraints between similar elements. In other words, you can create constraints either between 2D elements, or between generated elements, but not between a mix of these.
- Even though you set a constraint relation between two elements, constraints are not necessarily visualized. If you cannot visualize constraints even though the Show Constraints option is active in the Tools toolbar, go to **Tools -> Options -> Mechanical Design -> Drafting -> Geometry** tab and select **Display Constraints**. (You can also modify the constraint color and/or width.)

Creating Contact Constraints

This task shows you how to apply a constraint with a relative positioning that can be compared to contact. You can either select the geometry or the command first.

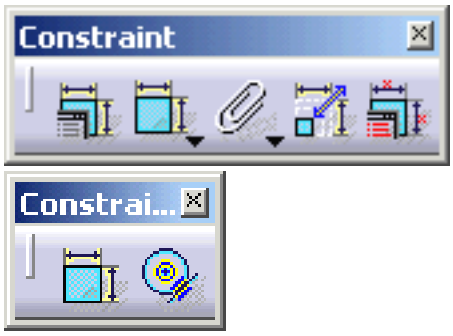
This constraint can be created between either two elements. These constraints are in priority:

- concentricity
- coincidence
- tangency

Use the contextual menu if you want to create other types of constraints.

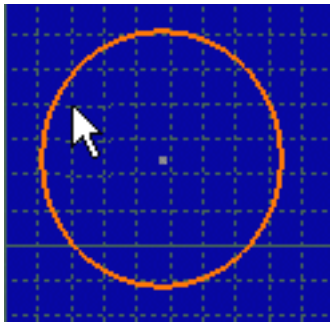
Open the Constraint_Contact.CATPart document.

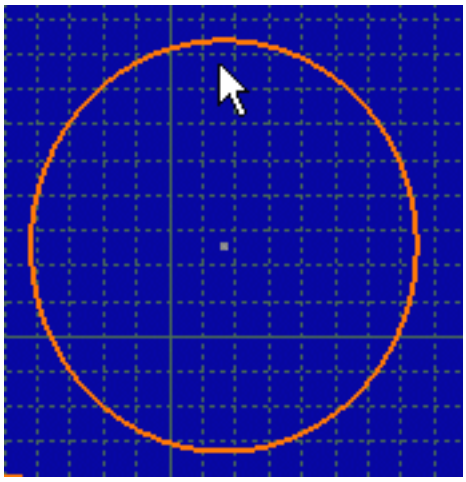
1. Select the **Contact Constraint** icon from the **Constraint** toolbar (**Constraint Creation** subtoolbar).

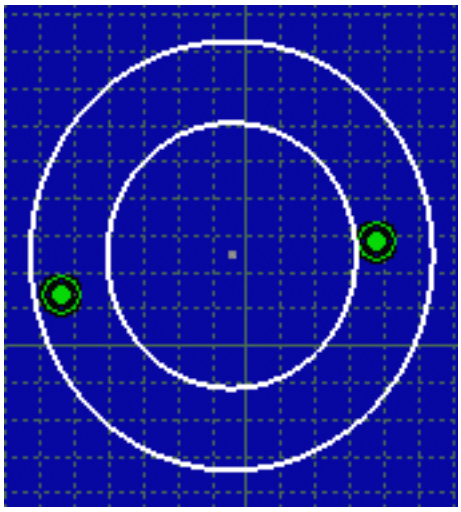


2. Click a first element. For example, click a circle.

3. Click a second element. For example, click another circle.







The Concentricity constraint symbol appears and the constraint is created.

According to the elements you select, a single type of constraint is proposed for defining the contact:

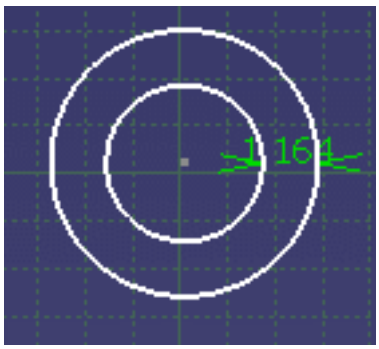
- A point and a line: coincidence
- Two circles: concentricity
- Two lines: coincidence
- Two points: coincidence
- A line and a circle: tangency
- A point and any other element: coincidence
- Two curves (except circles and/or ellipses) or two lines: tangency
- Two curves and/or ellipses: concentricity

Modifying or Deleting Contact Constraints

You can modify or even delete the contact constraint as follows:

1. Make sure either the **Constraint** or the **Contact Constraint** icon is active in the **Constraint** toolbar.
2. Right-click the constraint you want to modify or delete.
3. Select the option corresponding to the desired operation, from the contextual menu. For example, select the **Distance** constraint type to turn the contact constraint into a distance constraint.

   The Distance constraint symbol and value now appear as shown here.



Creating Constraints via SmartPick

This task shows you how to detect, create and visualize constraints using SmartPick. In this scenario, you are going to create two constrained parallel lines.

SmartPick dynamically detects the following geometrical constraints:

- support lines and circles
- alignment
- parallelism
- perpendicularity
- tangency
- concentricity
- horizontality
- verticality
- middle point

You can choose whether or not you want to create the constraints detected by SmartPick. To create the constraints, you can either:

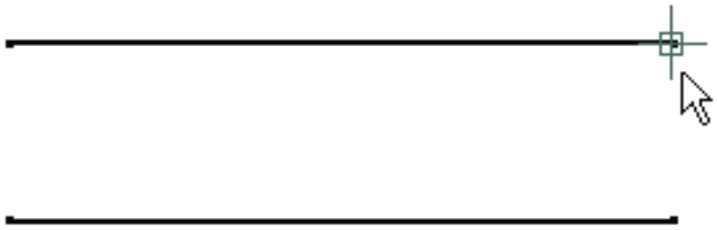
- select the **Create detected and feature-based constraints** check box in **Tools -> Options -> Mechanical Design -> Drafting -> Geometry** tab.
- or activate **Create Detected Constraints** in the **Tools** toolbar.

Make sure **Show Constraints** (to visualize detected and created constraints) and **Create Detected Constraints** are active in the **Tools** toolbar.

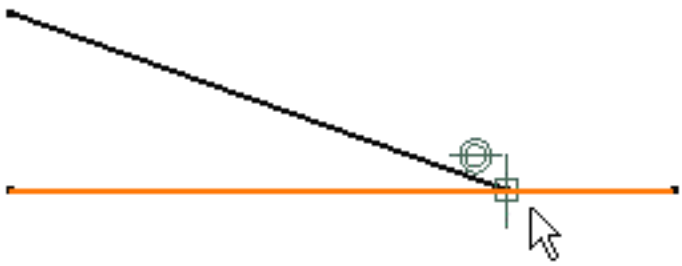
1. Create a first line.
2. Create a second line.

SmartPick can be used to create certain elements on the drawing. More precisely, only the elements which the cursor last went over will be used to apply SmartPick constraints. In other words, you simply need to move the cursor over the element you want to use as reference for a constraint.

No element is picked.



3. To detect parallelism constraints, go over the line to be used as reference.



As a result, a parallelism constraint is detected and created.

- When a constraint is detected by SmartPick, you can temporarily:
  - deactivate this constraint by maintaining the Shift key pressed.
  - lock this constraint by maintaining the Ctrl key pressed.
- In a Generative Drafting context, you cannot create geometrical constraints for extremities and circle centers.

Annotations

The 2D Layout for 3D Design workbench enables you to create:

- 2D annotations
- associative 3D annotations
- associative hybrid annotations between 2D and 3D elements

As annotation commands work as in the Interactive Drafting workbench, most tasks included in this section provide links to the *Interactive Drafting User's Guide*. As such, the information detailed in these tasks is presented in an Interactive Drafting context. You should note that the *Interactive Drafting User's Guide* contains images that correspond to the Interactive Drafting workbench and therefore illustrate annotations in an environment that is different from the 2D Layout for 3D Design environment (symbols and background color, for example).

Refer to Creating annotations in a 2D Layout for 3D Design Context which explains what is specific to creating annotations in the 2D Layout for 3D Design workbench as opposed to doing so in the Interactive Drafting workbench.

Creating annotations in a 2D Layout for 3D Design context: explains what is specific to creating annotations in the 2D Layout for 3D Design workbench as opposed to doing so in the Interactive Drafting workbench.

Before you begin: You should be familiar with basic concepts such as setting the properties of a text (font style, size, justification and so on), using default values, and specifying the position and/ or orientation of a text.

Create a free text: Create a text that either wraps or not, that is assigned an unlimited width text frame, even though this text may reach the frame boundary.

Create an associated text: Create a text which remains associated to an existing element.

Make an existing annotation associative: At any time and once an annotation has been created, you can add a link between an annotation and another element.

Create a text with a leader: Create a text with a leader either in the free space or associated with an element.

Add a leader to an existing annotation: Add a leader to an annotation that was previously created.

Handle annotation leaders: Add or remove breakpoints, extremity or interruptions. Move and position leader breakpoints.

Add frames and sub-frames to existing text: Add a frame or a sub-frame to a text that was previously created.

Copy text graphical properties: Copy the text graphical properties of an annotation or element to other elements.

Create a datum target: Create a datum target on a view.

Modify a datum target: Modify a datum target by editing it in a dialog box.

Create a balloon: Create a balloon using a dialog box.

Modify a balloon: Modify a balloon using a dialog box.

Create a roughness symbol: Create a roughness using a dialog box.

Create a welding symbol: Create a welding symbol using a dialog box.

Modify annotation positioning: Assign new positioning to existing annotations.

Create/modify a table: Create, edit and modify a table.

Find/replace text: Locate and then, if needed, replace strings of characters.

Perform an advanced search: Use the advanced search command.

Query object links: Query object links in a drawing.

Adding attribute links to text: Add one or more attribute links between text that was previously created.

Creating Annotations in a 2D Layout for 3D Design Context

The 2D Layout for 3D Design workbench enables you to create:

- 2D annotations
- associative 3D annotations
- associative hybrid annotations between 2D and 3D elements

As annotation commands work as in the Interactive Drafting workbench, the tasks included in the Annotations chapter provide links to the *Interactive Drafting User's Guide*. However, there are a few particularities about creating annotations in 2D Layout for 3D Design, as opposed to doing so in Drafting, which you will learn in this section.

In this section, you will learn about:

- Selecting elements to annotate
- Annotations in 2D Layout for 3D Design
  - Available commands
  - General behavior
  - Specific behaviors
  - Behavior of annotations with positional or orientation link
- Before you begin creating annotations in 2D Layout for 3D Design

Selecting elements to annotate

Annotation commands provide a visual feedback indicating whether it is possible to create annotations on a given element.

However, you should be aware of the following rule: in a given part layout, it is impossible to create an annotation which is associative in orientation or position to another part, as no link can be created between a part and another one. You can only create associative annotations within a single part layout. For example, in Part.1, it is not possible to store an annotation with a positional or orientation link to an element of Part.2.

When selecting elements to annotate, remember the following points:

- Annotations can be created in any view, even a non-active one.
- After starting an annotation command, the view in which you select the first element is the view of creation (that is the view where the annotation will be created).
- You can always select an element belonging to the view content.
- Once you have selected the first element, you can only select the other elements in the view of creation.
- You cannot select as the first element a 2D background element.
- You can select an element which belongs to the 3D background of a part layout only if this element belongs to the current layout.

Annotation behavior in 2D Layout for 3D Design

Available commands

You can create the following types of annotations: text, balloon, datum feature, datum target, geometrical tolerance, roughness symbol, welding symbol, and table.

Note that in order to be consistent with the way commands have been grouped in toolbars and sub-toolbars, datum targets and geometrical tolerances are documented in the Dimensioning chapter.

In addition, it is possible to add leaders, positional links and orientation links to existing annotations. Regarding positional/orientation links, some restrictions apply, which are detailed in Behavior of annotations with positional or orientation link below.

General behavior

You can create annotations:

- in the main view, in the background view, or in a 2D component view (on a layout detail sheet).
- in any visible design view (projection view, auxiliary view, section view/cut) or isometric view of the current sheet, whether or not it is the active view.

To do so, you need to select (and not just point to) an element of the view in which you want to create the annotation.

Specific behaviors

Welding symbols

In the Drafting workbench, welding symbol leaders are positioned associatively to the intersection of two reference elements. As only one element can be selected in 2D Layout for 3D Design, the leader is simply positioned at the indicated position.

Text with attribute link

In 2D Layout for 3D Design, you can create the same attribute links as in the Drafting workbench.

The update mechanism is the same as in Drafting: if the referenced parameter is located in the same document, the text is automatically updated each time the parameter value is modified. But if the referenced parameter belongs to another document, you need to update the attribute link manually using the **Local Update** command available on the layout, sheet or view contextual menu.

Adding a leader to an existing annotation

You can add a leader to an existing annotation. However, you cannot select any kind of geometry. The leader is associative if you respect the rules detailed in Behavior of annotations with positional or orientation link below. If not, the leader is positioned at the indicated position.

Adding a positional or orientation link to an existing annotation

You can add a positional or orientation link to an existing annotation. However, you cannot select any kind of geometry. The link is created if you respect the rules detailed in Behavior of annotations with positional or orientation link below.

Updating a positional or orientation link when the reference element is modified

When the reference element for a positional or orientation link is modified, the way the link is updated depends on where the reference element is located:

If the reference element belongs...	... then, when the reference element is modified:
to the view content	the positional/orientation link is automatically updated.
to the view 2D background	the positional/orientation link is updated when you have finished modifying the reference element. For example, if the reference element is a line and you drag it, the positional/orientation link is updated when you release the mouse.
to the view 3D background	you need to update the link manually using the **Local Update** command available on the layout, sheet or view contextual menu.

Behavior of annotations with positional or orientation link

When creating annotations with positional or orientation link, there are two behaviors, depending on where you create the annotation.

If you create annotations in the main/background view or in a 2D component view

If you create annotations directly in the main view, in the background view, or in a 2D component view (on a layout detail sheet), no

specific position in 3D space is defined. In this case, annotations are created exactly in the same context as in the Drafting workbench. Only view content elements may be selected. Therefore, annotation creation and edition commands behave exactly as in Drafting.

If you create annotations in a design/isometric view

If you create annotations in a design view or isometric view, a specific position in 3D space is defined. In this case, annotation creation and edition commands behave somewhat differently than in Drafting. Indeed, some restrictions apply regarding whether or not the annotation will be created with a positional/orientation link, as detailed below.

When editing a part layout outside the context of an assembly:

Only elements belonging to the same part or its associated layout are visible in a design/isometric view (including its background).

The table below sums up whether or not the annotation will be created with a positional/orientation link, depending on where the selected element is located and on what type of element you select:

If elements belong...	... and if you select...	... then the created annotation will be associative in position/orientation to the selected element:
to the view content	any geometrical element, annotation, axis line or center line	Yes
to the view 2D background	any element which belongs to another view (through the background of a design view)	Yes
to the view 3D background	• any edge, vertex or 3D wireframe element	Yes
	• a surface or solid element	No

When editing a part layout in the context of an assembly:

Elements belonging to the same part or its associated layout, as well as all elements belonging to any other part or product of the assembly, are visible in a design/isometric view (including its background).

However, remember that in a given part layout, it is impossible to create an annotation which is associative in orientation or position to another part, as no link can be created between a part and another one.

The table below sums up whether or not the annotation will be created with a positional/orientation link, depending on where the selected element is located and on what type of element you select:

If elements belong...	... and if you select...	... then the created annotation will be associative in position/orientation to the selected element:
to the view content	any geometrical element, annotation, axis line or center line	Yes
to the view 2D background	• an element of the same part instance	Yes
	• an element of another part or another instance of the part	No
to the view 3D background	• an element of the same part instance:	
	• any edge, vertex or 3D wireframe element	Yes
	• a surface or solid element	No
	• an element of another part or another instance of the part	
	• any edge, vertex or 3D wireframe element	No
	• a surface or solid element	No

Before you begin creating annotations in 2D Layout for 3D Design

Before you begin creating annotations in 2D Layout for 3D Design, make sure you are familiar with:

- The Tools toolbar and the Tools Palette.
- SmartPick, an easy-to-use tool designed to assist you when creating annotations. For more information, refer to the SmartPick task in the *Sketcher User's Guide*.
- Multi-selection. For more information, refer to the Selecting Objects chapter in the *Infrastructure User's Guide*.

Before You Begin

This task deals with:

- Setting Text Properties
- Specifying the Position and Orientation of a Text
- Annotations: Limitation

Setting Text Properties

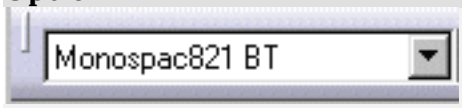
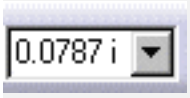
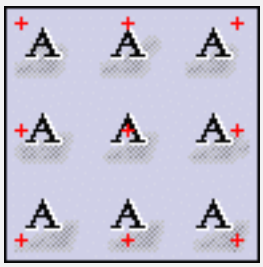
Text properties such as font style, size and justification can be applied to text, dimension text, text with leader, balloon and datum target, as well as to text included in datum features and geometrical tolerances. You can set the properties of a text either before or after creating it.
Create a text.

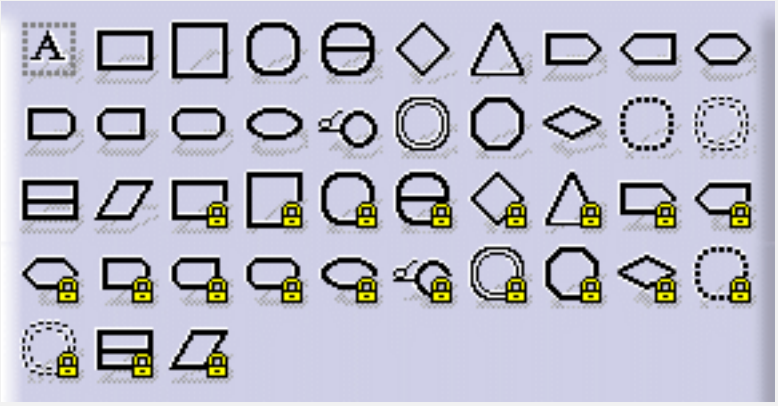
If the **Text Properties** toolbar is not displayed, choose **View -> Toolbars**, and select **Text Properties**.

1. Select the text.

2. Choose the properties you want to apply to this text in the **Text Properties** toolbar. For instance, select **Italic** and **Bold**. The properties you chose are applied to the selected text.

The options available in the Text properties toolbar are listed in the table below:

Option	Name	Description
	*Font Name*	Changes the style of the text.
	*Font Size*	Changes the size of the text. Note that this size cannot be inferior to 0.2 mm.
	Bold	Changes the weight of the text. Toggles between normal and heavy (bold).
	Italic	Changes the angle of the text. Toggles between normal and slanted (italic)
	Underline	Adds a line under the text.
	Strike Thru	Adds a line through the center of the text.
	Overline	Adds a line above the text.
	Superscript	Raises the text above the normal text line.
	Subscript	Lowers the text below the normal text line.
	Left Justify	Aligns multiple lines of text to the left edge of the text frame.
	Center Justify	Centers multiple lines of text within the text frame.
	Right Justify	Aligns multiple lines of text to the right edge of the text frame.
	*Anchor point*	Changes the position of the point that connects the text to the drawing or to an element. Choices are as follows: • Along the top of the text: left, center or right • Along the vertical center of text: left, center or right • Along the bottom of the text: left, center or right
	Frame	Draws a single-line frame around the text. A variety of different shapes is available. You can choose to create



each frame with either a variable or a fixed size. For a rectangular frame, for example, the icon represents the variable-size frame, and the icon (with the padlock) represents the fixed-size frame.

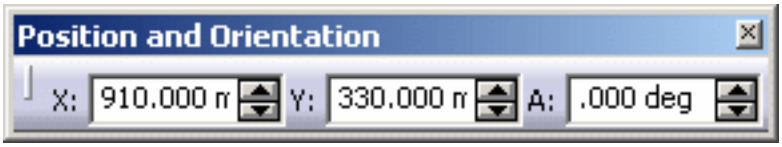
Insert Symbol Insert a symbol if the selected element allows it.

Specifying the Position and Orientation of a Text

You can specify the position (x, y) and/ or orientation (angle) of a text using the **Position and Orientation** toolbar. Note that the options available in this toolbar apply to annotations only (not to dimensions, for example).

Create a text.

If the **Text Properties** toolbar is not displayed, choose **View -> Toolbars**, and select **Position and Orientation**.



1. Select the text whose position and/or orientation you want to specify.
2. Enter the required values in the **Position and Orientation** toolbar.
   - X: sets the horizontal position.
   - Y: sets the vertical position.
   - A: sets the orientation.

Annotations: Limitation

- In the Drafting workbench, it is impossible to store annotations in a selection set.
- If you multi-select annotations and geometry and then try to drag the selected elements, then either annotations or geometry will be moved (depending on what elements are located under the mouse), but not both. To bypass this limitation, you can translate the elements instead of trying to drag them.

Creating a Free Text

This task explains how to create a text, with possible line wrapping. This text is assigned a frame of unlimited width, even though it may reach the frame boundary. You can set the properties of a text (anchor point, font size, justification, etc.) either before or after creating it.

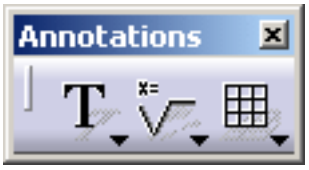
You will learn how to perform the following operations:

- Creating a Free Text
- Specifying Text Orientation

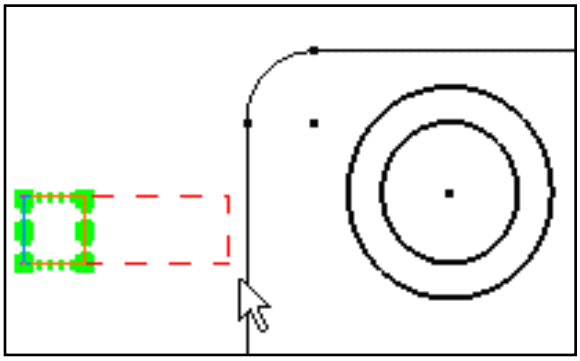
Creating a Free Text

Open the Brackets_views02.CATDrawing document.

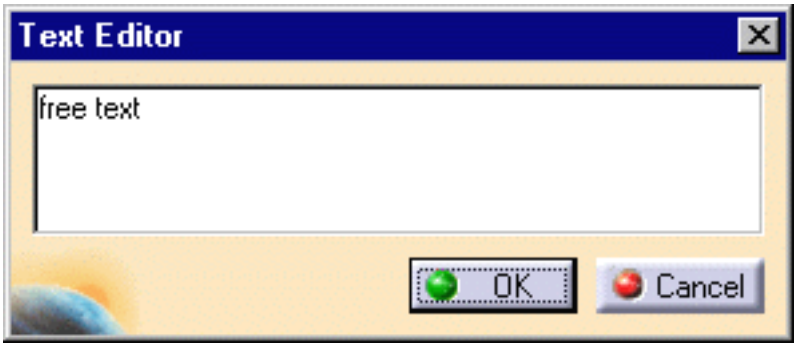
1. Click **Text** in the **Annotations** toolbar.



2. Click where you want to insert the free text on the drawing. A green frame appears, as well as the **Text Editor** dialog box.
3. If you want to specify the horizontal boundary of the text, drag the frame to where you want to place the boundary. If you want the horizontal boundary to adjust to your text, proceed with the following step.



4. Type your text in the **Text Editor** dialog box: "free text", for example.

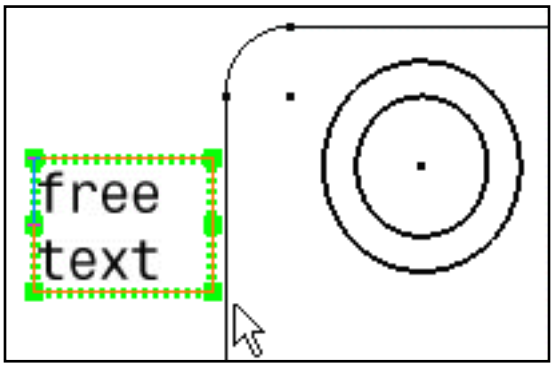


The drawing is automatically updated with the text you are typing in the **Text Editor** dialog box.

- You can copy text from another application and paste it into the **Text Editor** dialog box, and vice-versa. Note that the text layout and properties will not be preserved when copying/pasting.
- When copying/pasting an engineering symbol (such as ϕ Phi for example) in the text editor, note that the symbol is pasted as a plain character. As a result, if the symbol does not exist in the current font, the resulting character in the drawing may be different.
- You cannot copy complex objects (such as tables) from another application.

5. When you are done typing your text, click **OK** in the **Text Editor** dialog box, click anywhere on the drawing, or click any command. You can also click the **Select** icon : in this case, the text will remain selected so you can change its properties for example.

   You can now start setting the properties of the text you just created using the Text Properties toolbar.



Although you can create a text in a view that is not up-to-date, you cannot associate it to geometry. If you try to do so, a message appears, indicating that the selected or active view is not up-to-date.

Specifying Text Orientation

You can associate the text to an element and make it parallel to it. To do this, you can either:

- Go to **Tools -> Options-> Drafting -> Annotation and Dress-Up** tab and check **Text** in the **Annotation Creation** area. From then on, any text you create after having selected an element will be automatically associated to this element.
- When the above option is not activated, you can specify when you want to associate a text to an element. To do so, click **Text** and then press the **Shift** key while selecting the element you want the text to be associated to. You can then type your text.

You can also make the text vertical. To do this, click **Text** and then press the **Ctrl** key while clicking in the drawing where you want to create your free text.

Creating an Associated Text

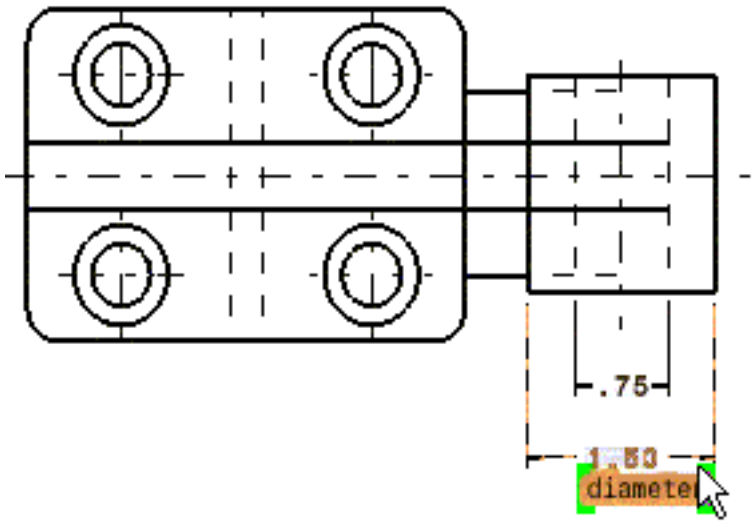
This task shows you how to create a text that will be associated to an existing element. This text will remain associated with this element. You can set text properties either before or after you create the text.

Open the Brackets_views03.CATDrawing document. Create two diameter dimensions, for example.

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

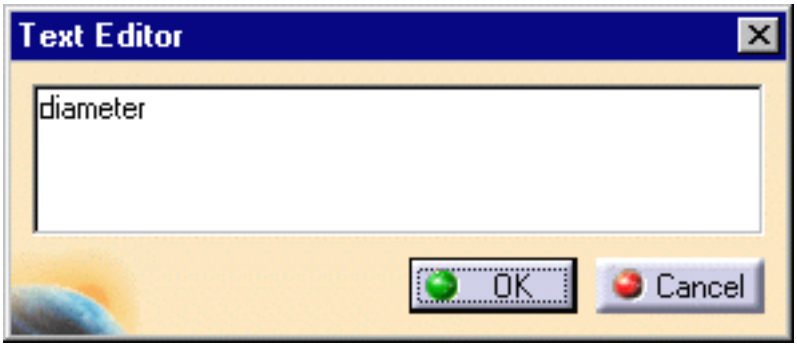
2. Select the element to which you want to associate a text.

   Here, we will use a dimension. You can associate the text either to the dimension line or to the extension line by clicking the appropriate element. Click the dimension line as in our example.



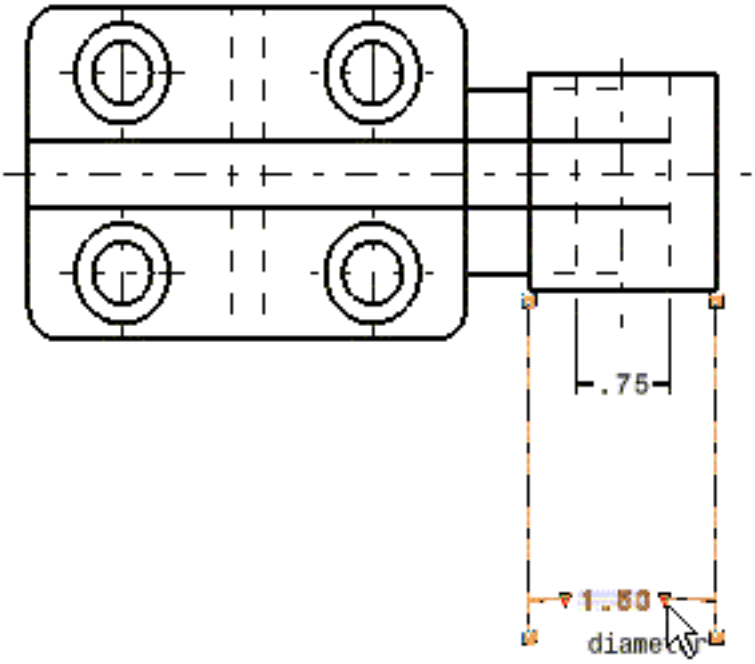
   The green text frame is displayed as well as the Text Editor dialog box.

3. Enter the text to be created in the Text Editor dialog box or directly on the drawing. For example, enter "diameter".

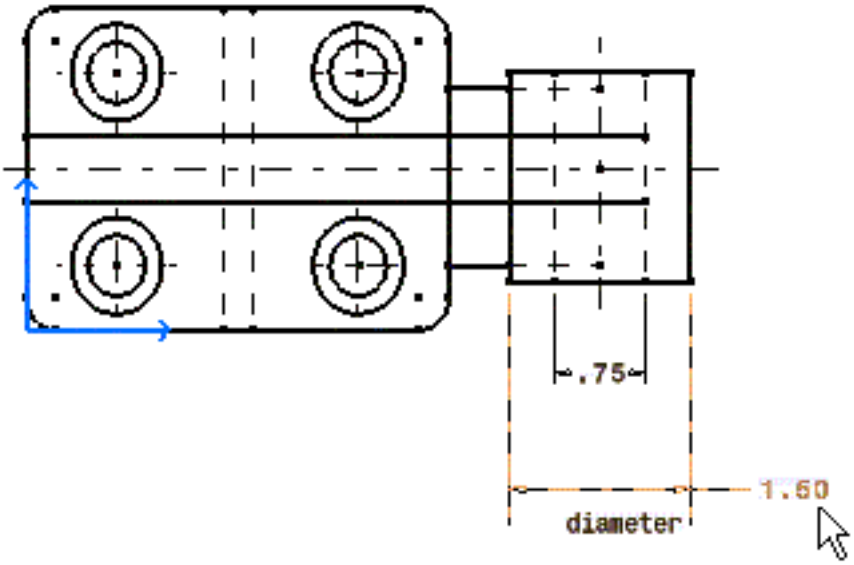


4. Click in the free space or click the Select icon to end the text creation.

5. If needed, select the dimension and move it to the desired location.



The text remains associated to the dimension.



- Note that the text is associative to the whole selected element. In other words, in the case of a dimension, if you move the dimension text exclusively, the associated text will not move accordingly.
- When creating associated texts, pressing the **SHIFT** key lets you change the orientation of the text as regards the element to which it is associated.
- You can associate text to the following elements:
  - Annotations: text, datum feature, datum target, balloon, geometrical tolerance, roughness symbol, weld symbols.
  - Dimensions
  - 2D elements: point, circle, ellipse, parabola, hyperbola.
  - Generative edges

Making an Existing Annotation Associative

This task explains how, at any time and once an annotation has been created, you can add a link between an annotation and another element. You can add two different types of links:

- positional links
- orientation links

The elements that can be linked to annotations are listed below:

- Annotations
  - text
  - datum feature
  - datum target
  - balloon
  - geometrical tolerance
  - roughness symbols
  - weld symbols
- Dimensions
- 2D elements
  - points
  - circles
  - ellipse
  - parabola
  - hyperbola

And, in a Generative Drafting context:

- Generative edges

Positional link

You can create positional links for every type of annotation.

Open the Brackets_views03.CATDrawing document. Create a text.

1. Click the Select icon .
2. Right-click any part of the text (text itself, frame or leader).
3. Select **Positional Link -> Create** from the contextual menu.
4. Select the element to which you want the text's position to be linked.

   The positional link is created. If you now select the linked element and drag it in the drawing, you can notice that the text follows this element.

5. To remove the positional link, right-click the text again, and select **Positional Link -> Delete** from the contextual menu.

Orientation link

You can create orientation links for texts, texts with leader and roughness symbols.

6. Right-click the text.
7. Select **Orientation Link -> Create** from the contextual menu.
8. Select the element to which you want the text's orientation to be linked.

   The orientation link is created. If you now select the linked element and modify its orientation, you can notice that the text's orientation is modified simultaneously.

- If you create a link between an annotation and a dimension system, remember that the link can only be made on a single dimension of the system.
- In a Generative Drafting context, certain types of generated elements (such as Pipe elements) are not associative. For this reason, positional links or orientation links between an annotation and such elements are not taken into account.

Creating a Text With a Leader

This task shows you how to create a text with a leader either in the free space or associated with an element. You can set text properties either before or after you create the text.

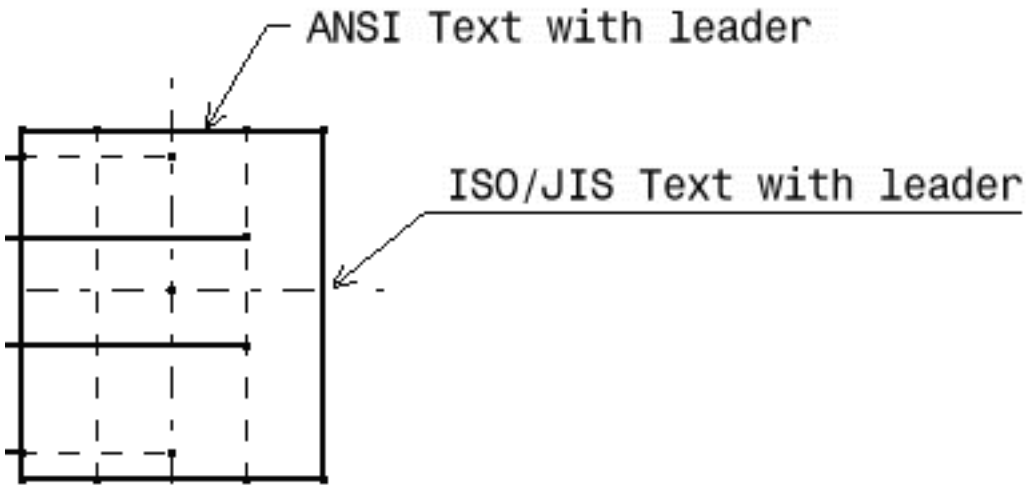
This task deals with:

- Creating a Text with a Leader
- About Texts with Leader
  - Specifying Leader Orientation
  - Specifying Text Orientation
  - Elements that can be Assigned Text with a Leader

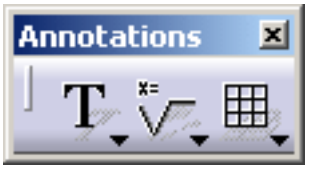
Creating a Text With a Leader

Create a rectangle.

Note that leader lines are displayed in either of the following ways based on the standard set when defining the sheet.



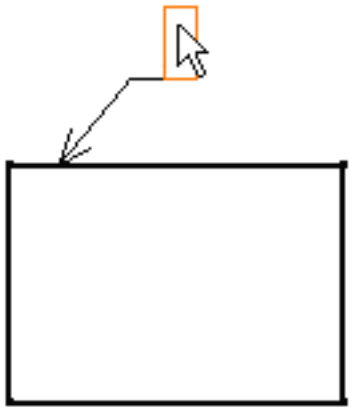
1. Click the **Text With Leader** icon from the **Annotations** toolbar (**Texts** sub-toolbar).



2. Click the point on the element you want the leader to begin (arrow end).

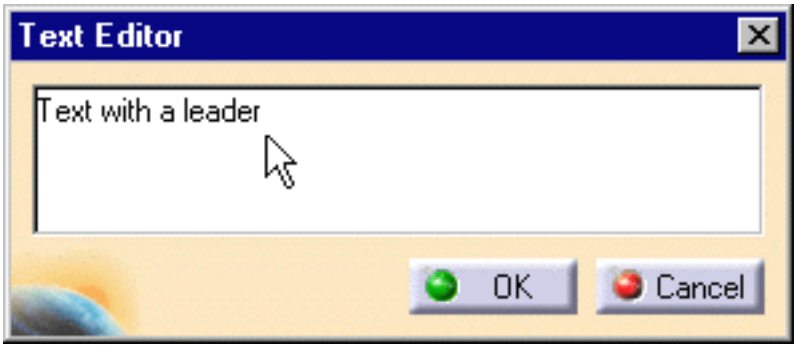
   A red frame appears.

3. Click in the free space to define a location for the text.

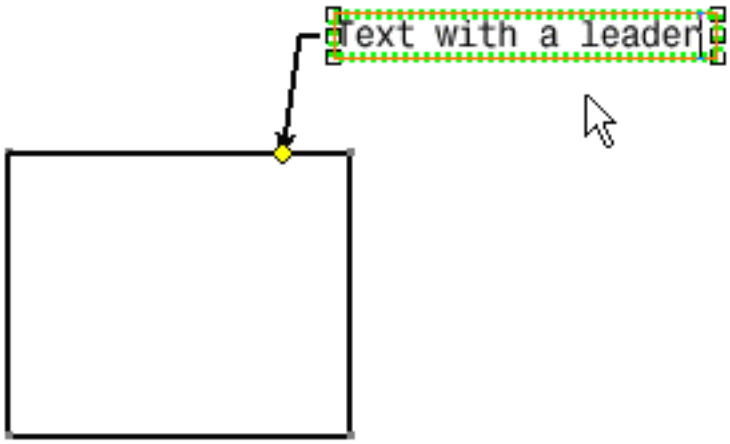


The Text Editor dialog box is displayed.

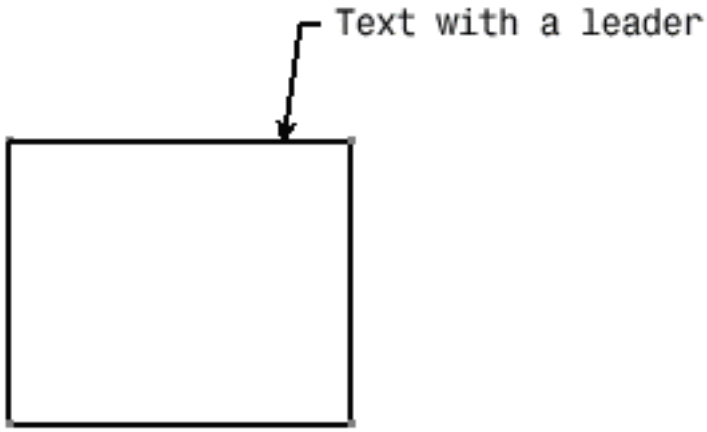
4. Enter the text in the Text Editor dialog box or directly on the drawing: "text with a leader", for example.



5. If needed, re-position or modify the text.



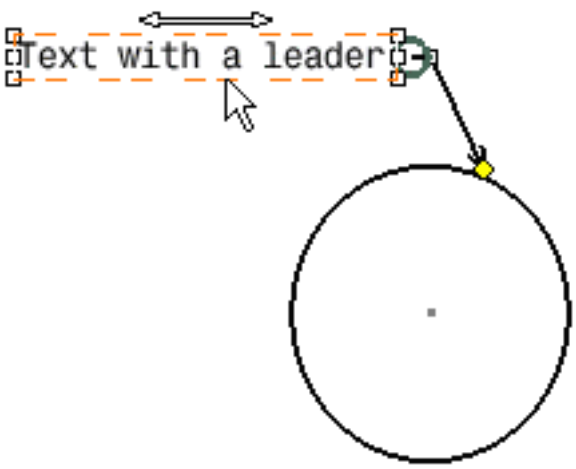
6. To end the text creation, click again in free space or select a command icon.



The leader is associated with the element you selected. If you move either the text or the element, the leader stretches to maintain its association with the element.
If you change the element that is associated with the leader, both the new element and the text with leader remain associative to each other.

7. Create a circle.

8. Drag the text with leader (using the yellow manipulator at the leader's extremity) to associate it with the circle instead of the rectangle.



Although you can create a text in a view that is not up-to-date, you cannot associate it to geometry. If you try to do so, a message appears, indicating that the selected or active view is not up-to-date.

About Texts with Leader

Specifying Leader Orientation

When creating a text with leader, you can orient the leader perpendicularly to the element to which it is associated.

To do this, you have two different possibilities:

- Either go to **Tools -> Options -> Drafting -> Annotation and Dress-up** tab and check **Activate snapping (SHIFT toggles)**. Then, click the **Configure** button and select either **On orientation** or **Both**. To orient directly the leader perpendicularly to the associated element, press the **Shift** key while clicking on the element to which you want to associate the text with leader (previous scenario, step 3).
- Or go to **Tools -> Options -> Drafting -> Annotation and Dress-Up** tab, and check **Text** in the **Annotation Creation** area. The text leader will be oriented perpendicularly to the geometry by default. In this case, pressing the **Shift** key will let you orient it differently.

Specifying Text Orientation

When creating a text with leader, you can make the text vertical. To do this, hold the **Ctrl** key while clicking in the drawing to position the text (previous scenario, step 3).

- You can also add a leader to existing text. To learn how to do this, refer to Adding a Leader to Existing Text.
- You can perform a number of operations on a leader. To learn more, refer to Editing Annotation Leaders.

Elements to which a Text with a Leader Can Be Assigned

You can assign a text with a leader to the following elements:

- 2D elements
  - lines
  - points
  - circles
  - curves
- Generative edges

Adding a Leader to an Existing Annotation

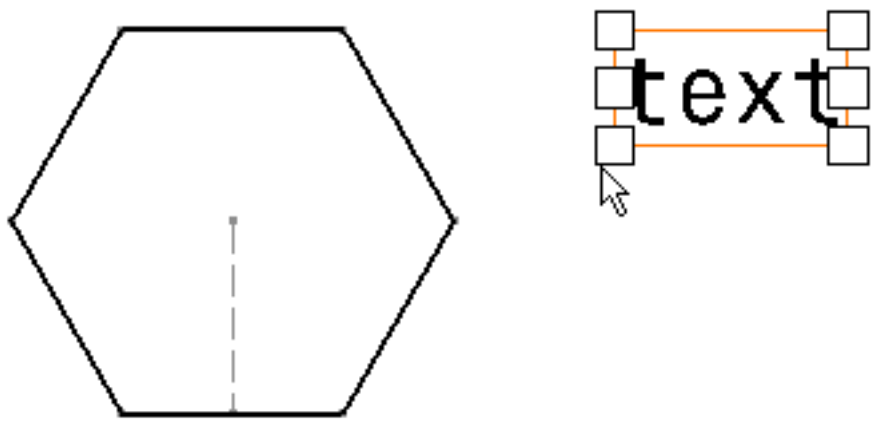
This task shows you how to add a leader to an annotation that was previously created. Leaders can be positioned freely, or using snapping (the leader is oriented perpendicular to the reference element).

For the purpose of this scenario, you will learn how to add a leader to an existing text, but this functionality is available with other annotation types as well.

Go to **Tools -> Options-> Mechanical Design -> Drafting -> Annotation and Dress-Up** tab . Make sure the **Activate snapping (Shift toggles)** option is selected. Then, click on the **Configure** button and select either **On orientation** or **Both**.

Create a hexagon. Create an annotation, a free text for example.

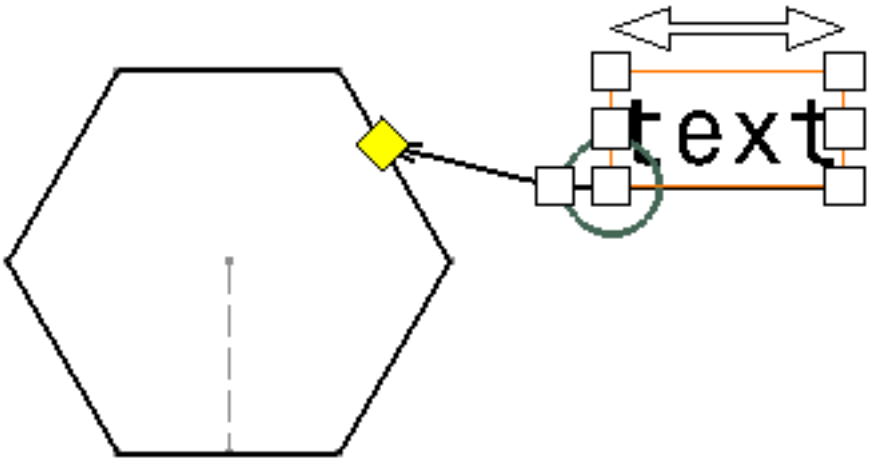
1. Right-click the annotation to which you want to add a leader.



2. Select **Add Leader**.
3. You have two possibilities:
   - If you want to position the leader freely:

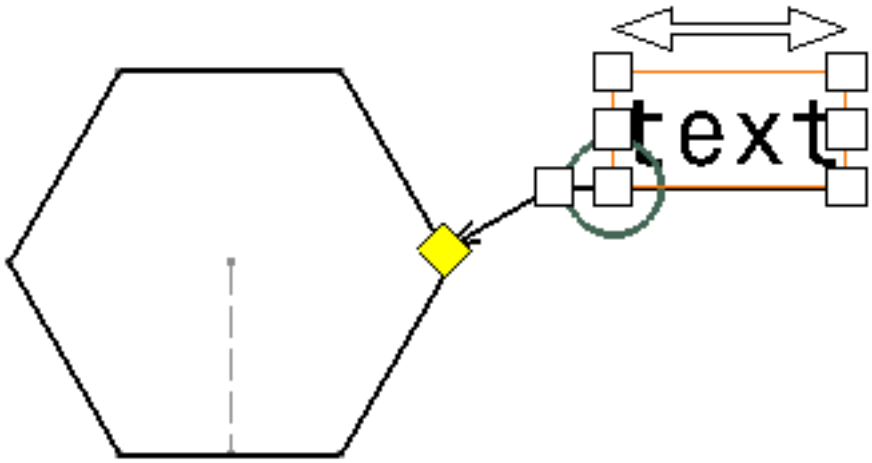
     Click where you want to position the leader head. The leader is created.

     You can then move it to the desired location using the mouse. You can position the leader breakpoint anywhere on the reference element, and snapping is not used.

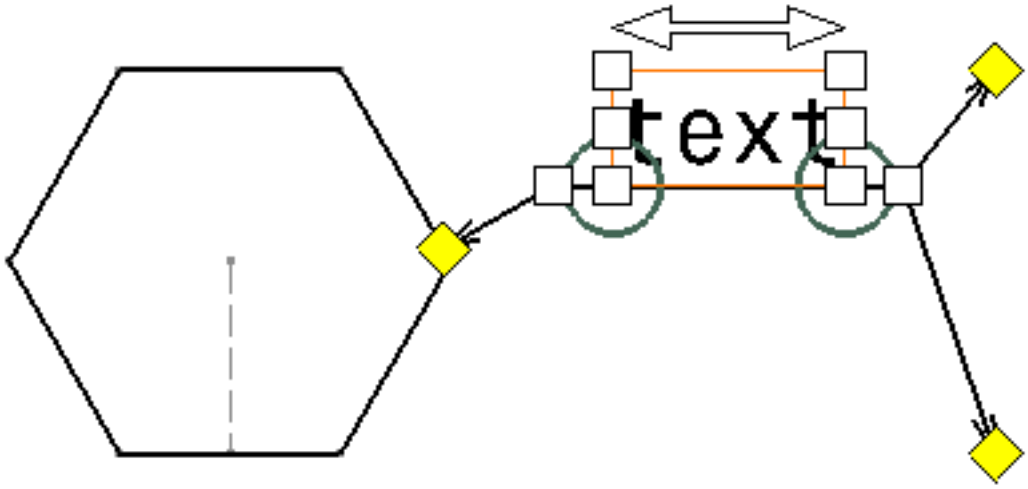


- If you want the leader to be oriented perpendicular to the reference element:

  Press the **Shift** key while clicking where you want to position the leader head. The leader is created: it is snapped, and oriented perpendicular to the element to which it is attached. Release the **Shift** key and the mouse.



To create as many leaders as required for an existing text, go to **Tools -> Customize** and create the **Add Leader** command in a separate toolbar. You will then be able to double-click the **Add Leader** command and click to locate the leader(s) to be created.



If several text elements are selected as you activate the Add Leader command, the selection is cleared. Make sure you select one annotation only.

The leader remains associative to the text even if you modify the text it is associated with.

Handling Annotation Leaders

This task shows you how to do the following:

- handle annotation leaders, by performing such operations as adding or removing a breakpoint, an extremity or an interruption.
- move and position leader breakpoints.

Depending on the type of annotation the leader is associated with, not all operations described in this section will be available.

Multi-selection restrictions:

- For all operations described in the Handling Leaders section below (except for changing the symbol shape - *see next comment*), multi-selection is not taken into account. The operation will be performed only on the leader you right-click in the selection.
- Changing the symbol shape behaves differently depending on whether one or several annotations in the selection have more than one leader:
  - if the leader you right-click is the only one in the annotation, then the symbol is applied to this leader and to all annotations which have only one leader.
  - if the leader you right-click is not the only one in the annotation, then the symbol is applied to this leader only.

About associative and non-associative annotations:

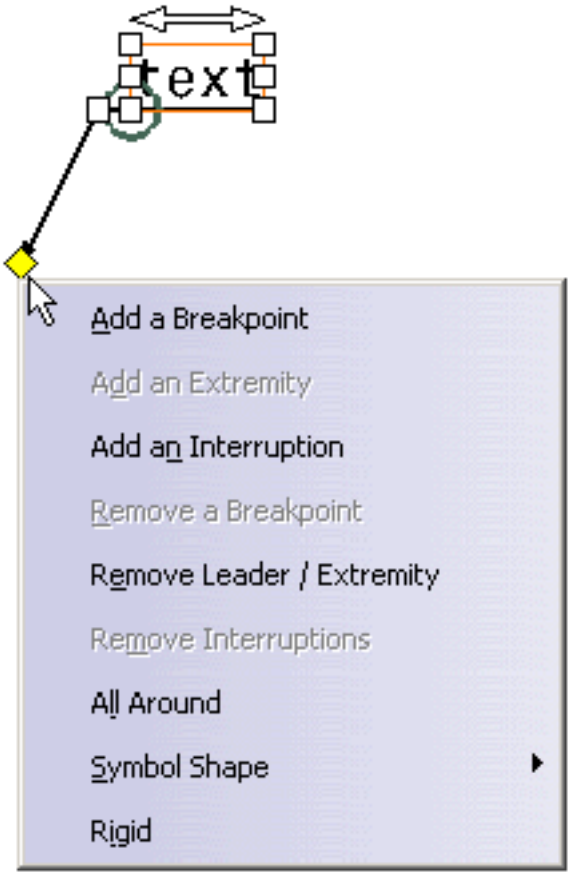
When you create an annotation, a type of leader is automatically set, provided the standard files have not been modified. If you choose the Automatic option, a default symbol will be used, depending on the standard type, on the annotation type, and on whether the leader is associated to an element or not:

- If the leader is associated to an element:
  - Unfilled arrow for ANSI / ASME
  - Open arrow for ISO / JIS
- If the leader is not associated to an element:
  - Unfilled circle for ANSI / ASME
  - Filled circle for ISO / JIS

Handling Leaders

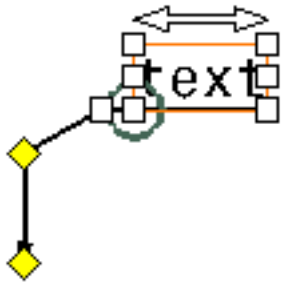
Create a text with a leader.

1. Right-click the yellow control point at the end of the leader. The leader's contextual menu is displayed.

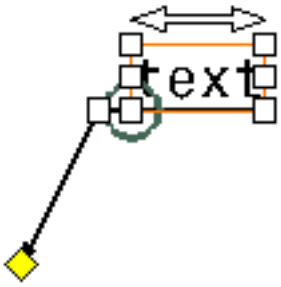


2. Choose from the available options.

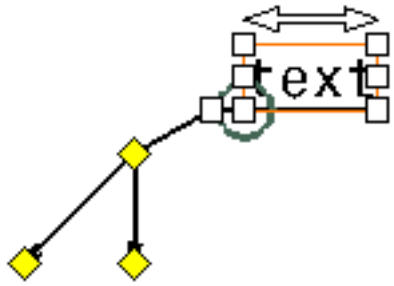
- To add a breakpoint, select Add a Breakpoint.



Then, to remove this breakpoint, right-click on the breakpoint and select Remove a Breakpoint.

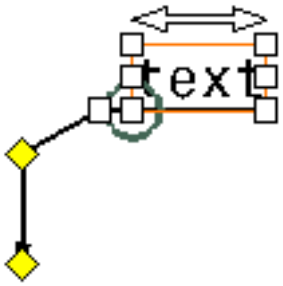


- To add an extremity to an existing breakpoint, right-click on the breakpoint, select Add an extremity, and then click where you want to position the extremity.



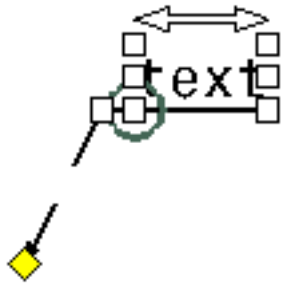
You can add an extremity only in the case of a text or a welding symbol.

Then, to remove this extremity, right-click on the additional extremity and select Remove Leader/Extremity.

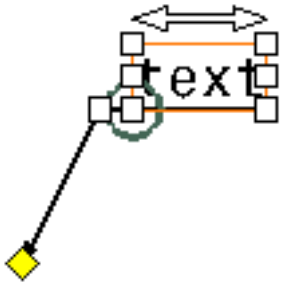


Clicking on the main leader extremity will remove the leader.

- To add an interruption, select Add an Interruption and then, on the leader, click the two points between which you want to add the interruption.

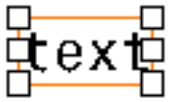


Then, to remove this interruption, right-click on the leader yellow control point and select Remove Interruptions.

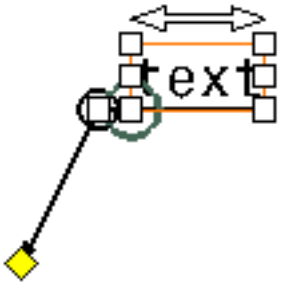


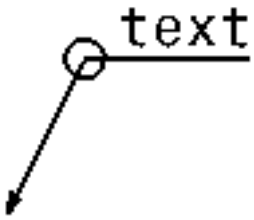
Any existing interruption will be removed from the leader when adding or removing breakpoints.

- To remove the leader, select Remove Leader/Extremity.



- To add an all around symbol, select All Around.

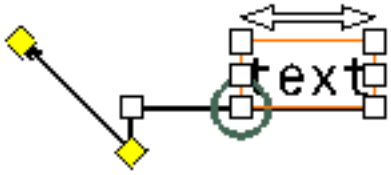




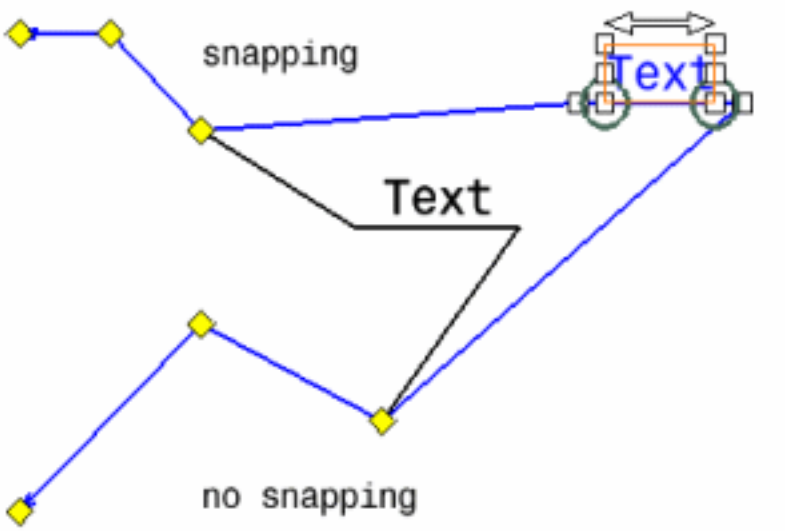
- To modify the leader symbol shape, point to **Symbol Shape**. Then, select **No Symbol** if you do not want a symbol for the leader, or select the symbol you want from the available symbols.

You can remove the leader extremity symbol for all annotations.

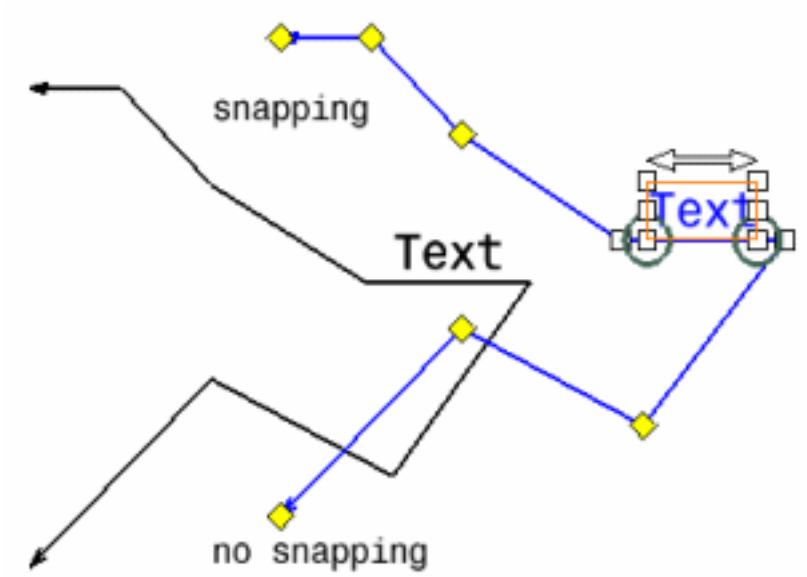
3. You can also move the leader or any existing breakpoints by clicking a yellow control point and moving it using the mouse.



- To move the annotation but not the leader, click the annotation and move it using the mouse.



- To move the leader along with the annotation while making sure the leader keeps its original shape, select Rigid and then move the annotation.



- This functionality is available for texts, welding symbols, 2D components, tables and geometrical tolerances, but not for other annotation types.

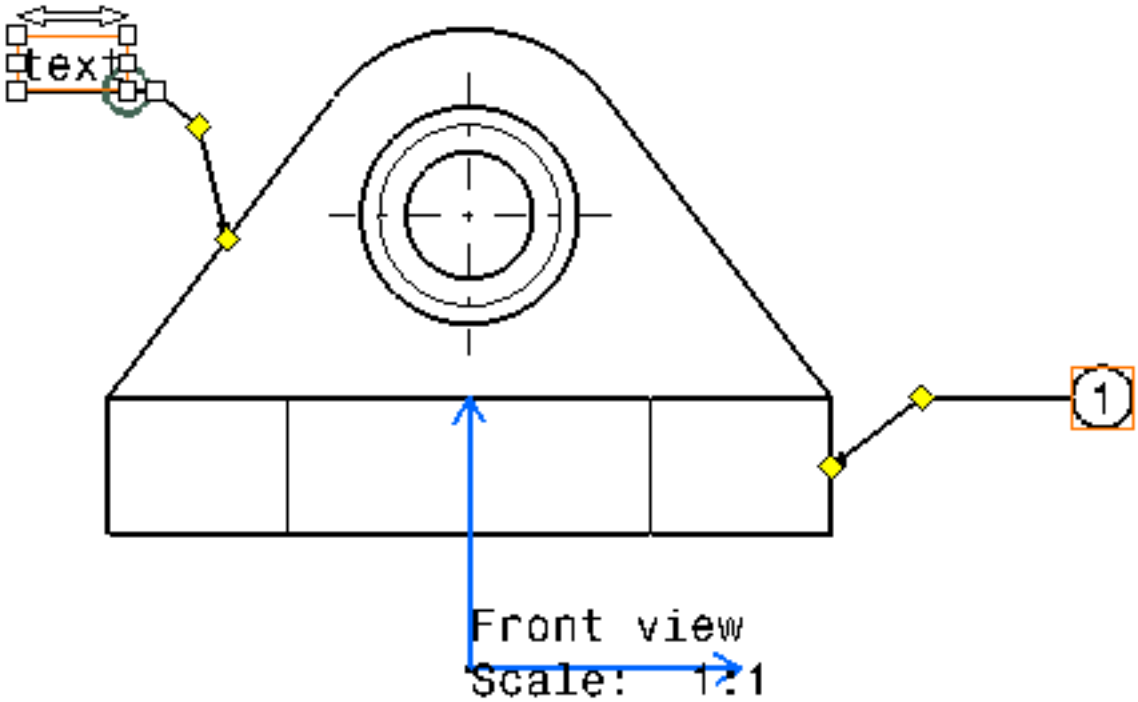
- This functionality also applies when rotating the annotation text using the Free Rotation icon .

Moving and Positioning Leader Breakpoints

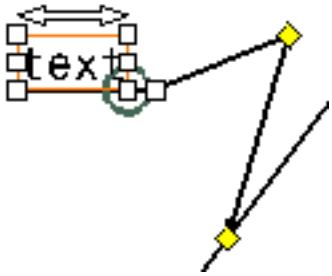
You can move and position leaders breakpoints easily, for all types of annotations. Leader breakpoints are moved and positioned using snapping (the leader is oriented perpendicular to the reference element).

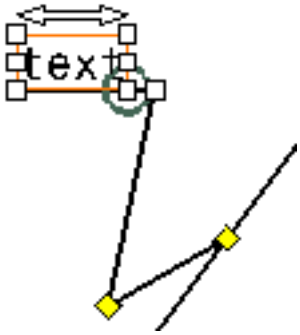
Go to **Tools -> Options-> Drafting -> Mechanical Design -> Annotation and Dress-up** tab. Make sure the **Activate snapping (Shift toggles)** option is selected. Then, click on the **Configure** button and, in the dialog box which is displayed, select either **Leader orientation** or **Both**.

Open the Move_Leaders.CATDrawing document. This document contains a text with leader and a balloon. Add a breakpoint to both annotations, as explained in the previous section.

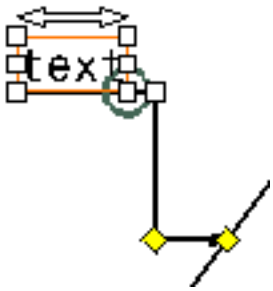


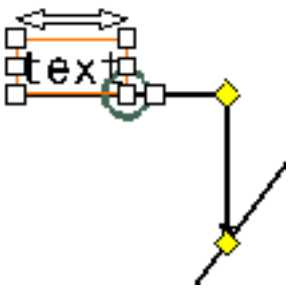
1. Move the text leader breakpoint with the mouse. You can position the leader breakpoint anywhere, and snapping is not used.

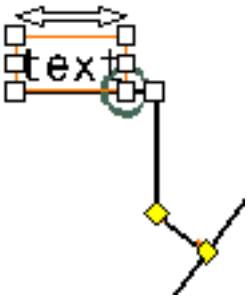




2. Now, press the **Shift** key while moving the leader breakpoint with the mouse. The leader is snapped, and is positioned vertically or horizontally, or perpendicular to the element to which it is attached.

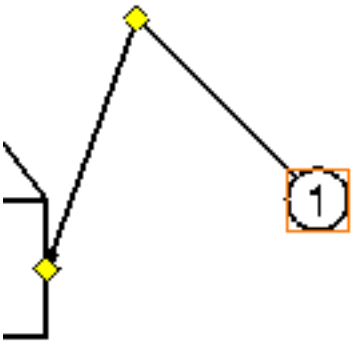


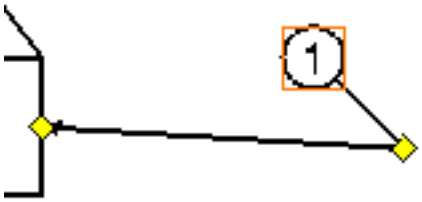




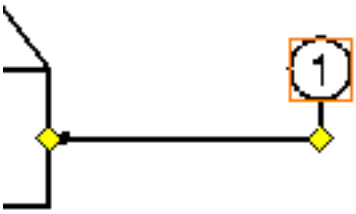
3. Release the **Shift** key and the mouse when you are satisfied with the position of the leader.

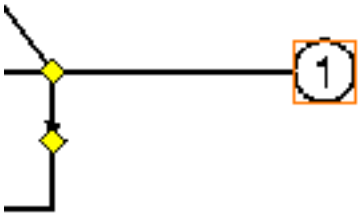
4. Move the balloon leader breakpoint with the mouse. You can position the leader breakpoint anywhere, and snapping is not used.



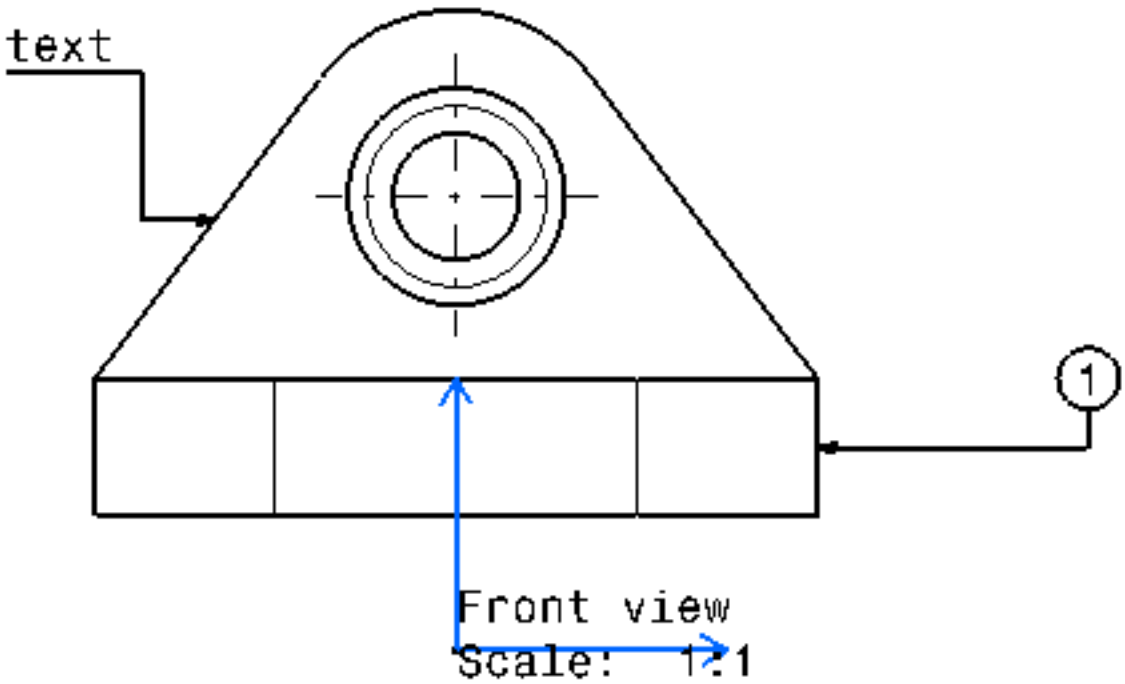


5. Now, press the **Shift** key while moving the leader breakpoint with the mouse. The leader is snapped, and is positioned vertically or horizontally, which happens to be the same orientation as the element to which the leader is attached.





6. Release the **Shift** key and the mouse when you are satisfied with the position of the leader. Both leaders are now positioned properly.

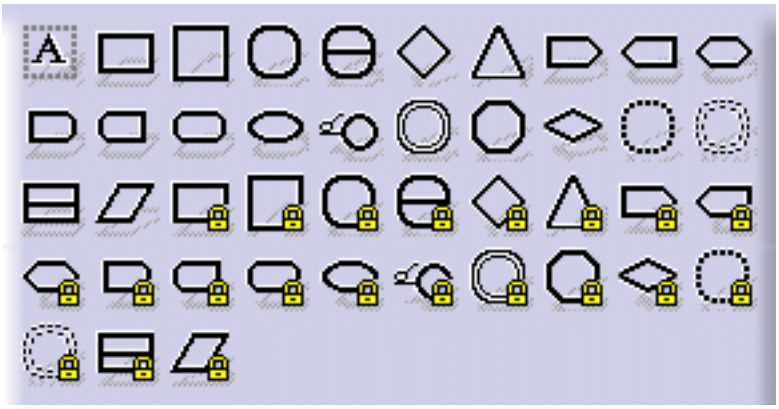

text
1
Front view
Scale: 1:1

Adding Frames or Sub-Frames

This function allows you to add frames and sub-frames to texts and texts with leader.

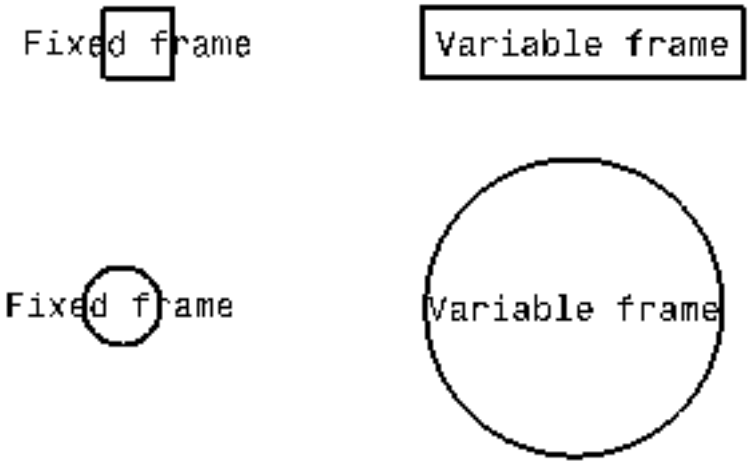
Create a free text.

1. Select the text you have created and click the **Frame** icon in the **Text Properties** toolbar. The **Frames** sub-menu is displayed.



You can choose to create each frame with either a variable or a fixed size. For a rectangular frame, for example, the icon represents the variable-size frame, and the icon (with the padlock) represents the fixed-size frame.

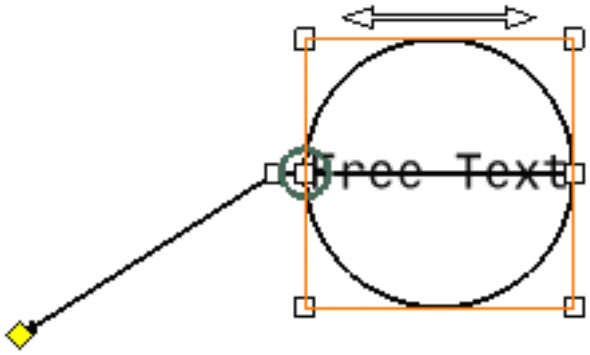
- Variable-size frames adapt to the text length, whereas fixed-size frames always remain as is, no matter what the text length is. So if you choose a fixed-size frame and the length of you text exceeds the frame size, then the text will extend beyond the frame.



- Fixed frame sizes are defined in the standards. For more information, refer to Frames in Administration Tasks > Setting Standard Parameters and Styles > Setting Standard Parameters.

2. Choose a frame in the menu, scored circle, for example, and click to select it.

3. Right-click on the text and in the contextual menu choose the **Add leader** command and click in the free space to end the leader creation.



You can zoom to make it easier to move the leader round the text.

4. Right-click the hanged point.

A contextual menu is displayed, in which **Standard Behavior** is selected by default. Selecting or deselecting this option drives how many anchor points are available when moving the leader around the frame (refer to the *Frames Anchors Table* below, *Standard Behavior OFF* or *Standard Behavior ON* columns).

Frames Anchors Table

Type of frame	Standard Behavior OFF	Standard Behavior ON
Rectangle	1 2 3 o-------o-------o / \ 4 o o 5 \ / o-------o-------o	o-------o-------o / \ 1 o o 2 \ / o-------o-------o
Scored Rectangle		
Square		
Circle	3 __o__ 2 o o 4 | | 1 o o 5 | | 8 o o 6 --o-- 7	__o__ / \ | | 1 o o 2 | | \ / --o--
Scored Circle		
Set		
Fixed Support		
Sym Part		
Sym Set		
Diamond	3 o / \ 2 o o 4 / \ 1 o o 5 \ / 8 o o 6 \ / o 7	o / \ o o / \ 1 o o 2 \ / o o \ / o

Nota

Triangle

```
        3
        o
       / \
    2 o     o 4
     /       \
  1 o----o----o 5
        6
```

```
        o
       / \
      o     o
     /       \
  1 o---------o 2
```

Right Flag

Right Oblong

```
    1       2       3
    o-------o-------o
    |                 \
  4 o                  o 5
    |                 /
    o-------o-------o
    6       7       8
```

```
    o-------o-------o
    |                 \
  1 o                  o 2
    |                 /
    o-------o-------o
```

Left Flag

Left Oblong

```
    1       2       3
    o-------o-------o
   /                |
4 o                 o 5
   \                |
    o-------o-------o
    6       7       8
```

```
    o-------o-------o
   /                |
1 o                 o 2
   \                |
    o-------o-------o
```

Both Flag

Oblong

```
    1       2       3
    o-------o-------o
   /                 \
4  o                   o 5
   \                 /
    o-------o-------o
    6       7       8
```

```
    o-------o-------o
   /                 \
1 o                   o 2
   \                 /
    o-------o-------o
```

Ellipse

Sticking

```
1 o---------o 2
```

```
1 o---------o 2
```

Parallelogram

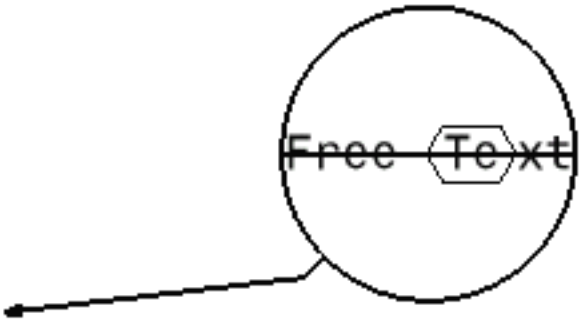
```
    3       4       5
    o-------o-------o
   /               /
 2 o             o 6
  /             /
 o-------o-------o
1        8       7
```

```
    o-------o-------o
   /               /
  o               o
 /               /
o-------o-------o
1               2
```

5. Drag the leader hanged point and see how it behaves depending on whether **Standard Behavior** is selected or not.

6. Select a part of the text, as an example "Te", for this:
   - Double-click on the text to edit it, the Text Editor appears.
   - Select "Te" in the Text Editor or in the drawing.

7. Apply the Both Flag frame to the text.

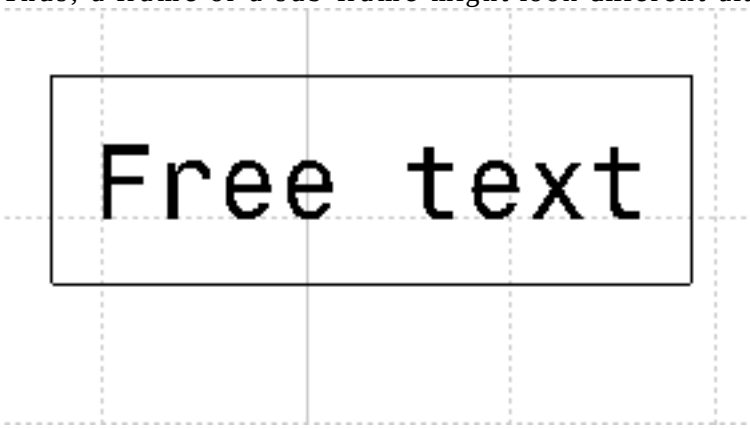


You cannot use the following types of frames as sub-frames: Sticking, Nota, Scored Rectangle, and all types of fixed-size frames.

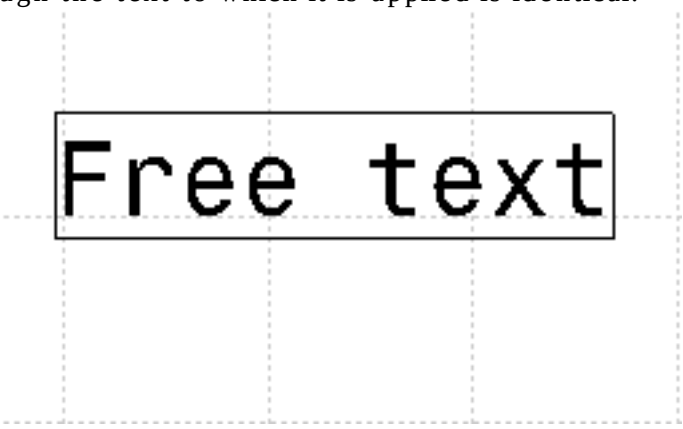
The size of frames depends on:

- whether you use them as frames or sub-frames,
- the height between the characters top and bottom or cap and base,
- margins.

Thus, a frame or a sub-frame might look different although the text to which it is applied is identical.



Example of frame.



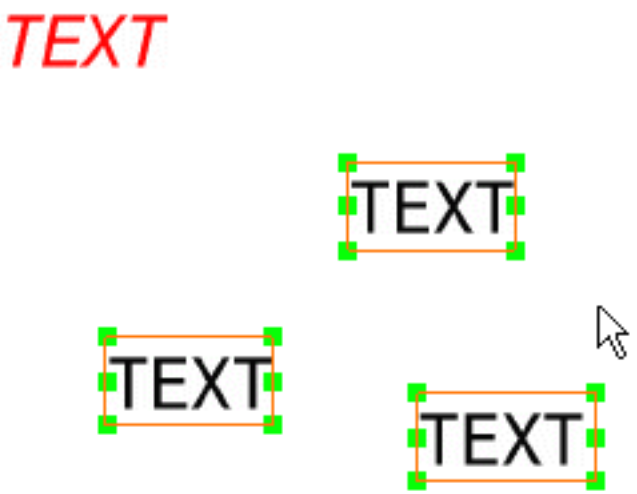
Example of sub-frame.

Copying Graphic Properties

This task shows you how to copy the graphic properties of a text element to existing texts. This is true for any type of Interactive Drafting element. In this task, we will take free text as an example.

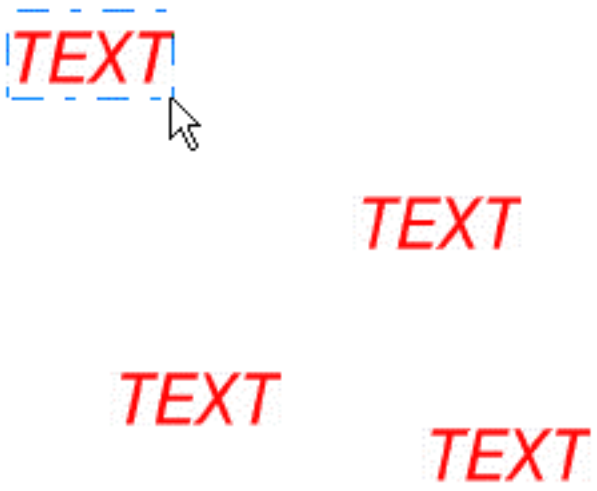
Create free texts.

1. Multi-select the free texts to be modified graphically speaking.



2. Click the Copy Object Format icon from the Graphic Properties toolbar.

3. Select the text to be used as a graphical reference for selected texts.



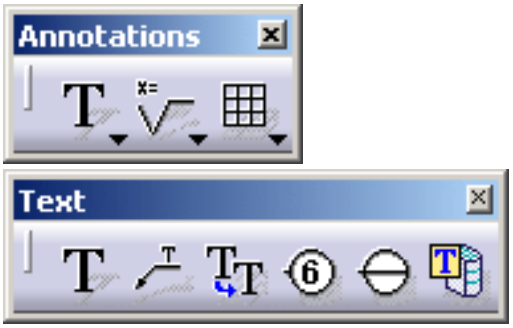
The graphical properties assigned to the text used as a reference are now copied onto the multi-selected free texts to be modified.

Creating a Datum Target

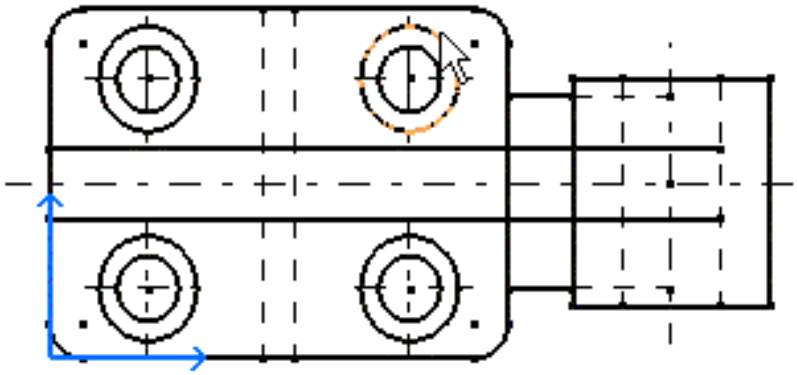
This task will show you how to create a datum target on a right projection view. You can set text properties either before or after you create the datum target.

Open the Brackets_views03.CATDrawing document.

1. Click the **Datum Target** icon from the **Annotations** toolbar (**Text** sub-toolbar).

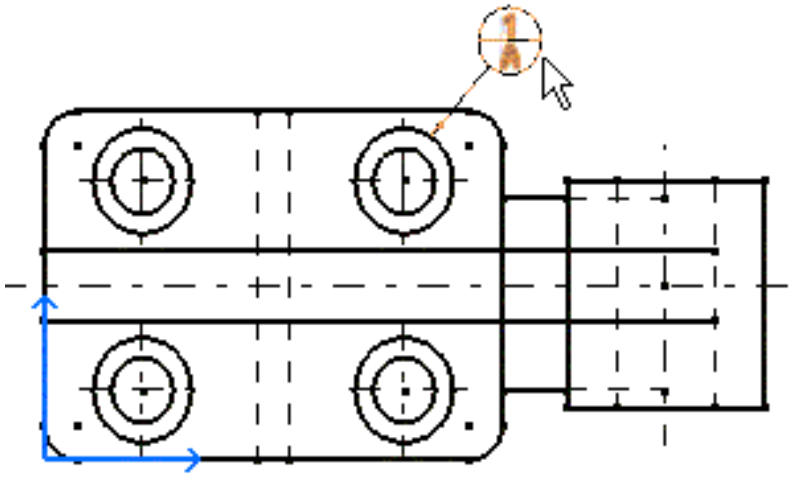


2. Select the attachment point of the datum target leader.
3. Select a point to be used to position the datum target (anchor point).



The **Datum Target Creation** dialog box is displayed.

4. Enter the required values in the fields. For example, 1 and A.



Click the button if you want to specify that the datum target provides information on the diameter of the selected element.

5. Click **OK**.

The datum target is created.

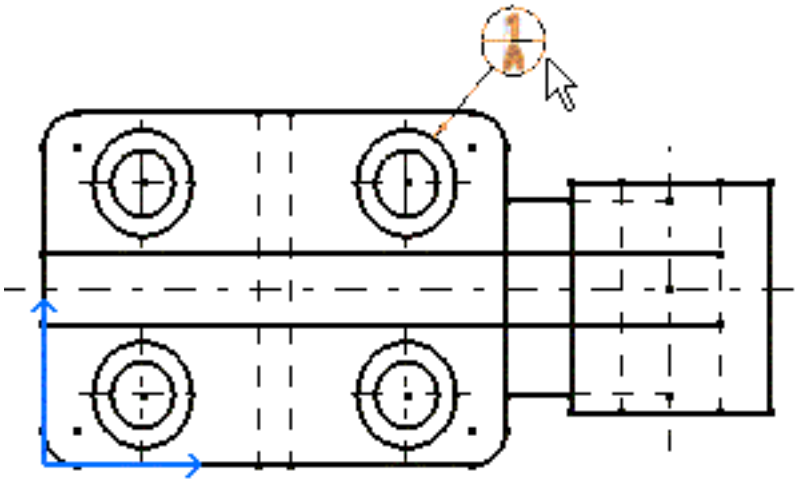
The character string that is edited in the **Datum Target Creation** dialog box is simultaneously previewed on the drawing.

Modifying a Datum Target

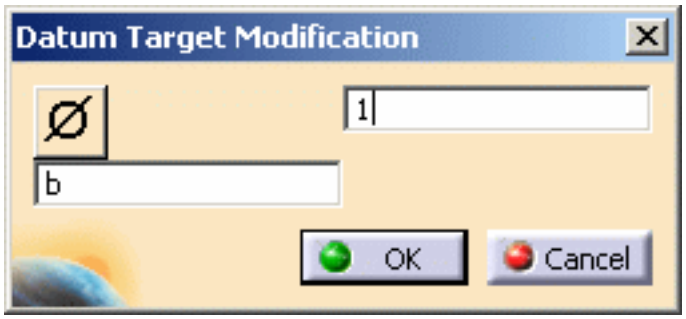
This task shows you how to modify a datum target by editing it.

Open the Brackets_views03.CATDrawing document. Create a datum target.

1. Double-click the datum target you want to modify.



The **Datum Target Modification** dialog box is displayed.

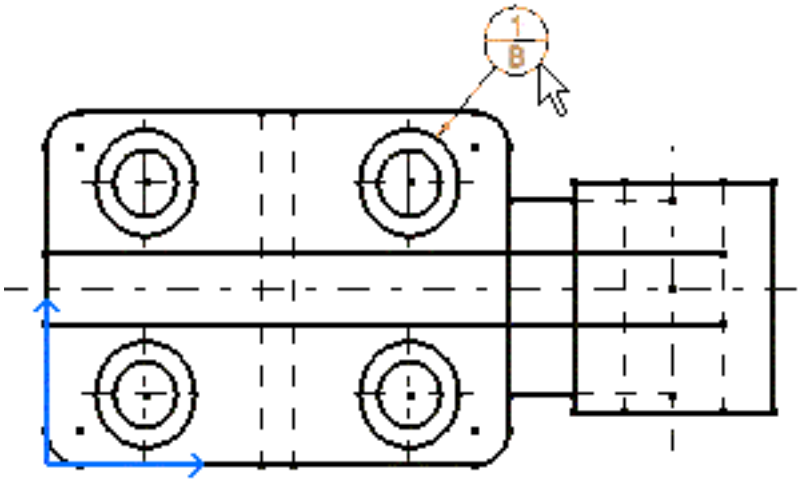


2. Modify any of the datum target values.

   For example, enter B instead of A.

3. Click **OK**.

The datum target is modified.



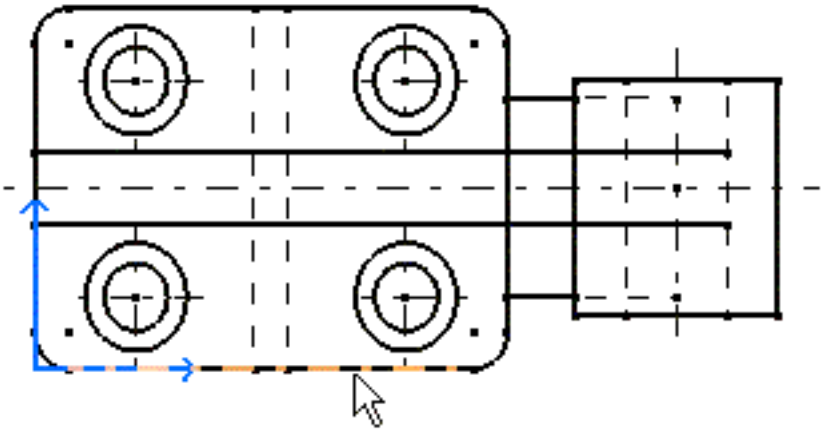
Creating a Balloon

This task will show you how to create a balloon. You can set text properties either before or after you create the text.

Open the Brackets_views03.CATDrawing document.

1. Click the **Balloon** icon from the **Annotations** toolbar (**Text** sub-toolbar).

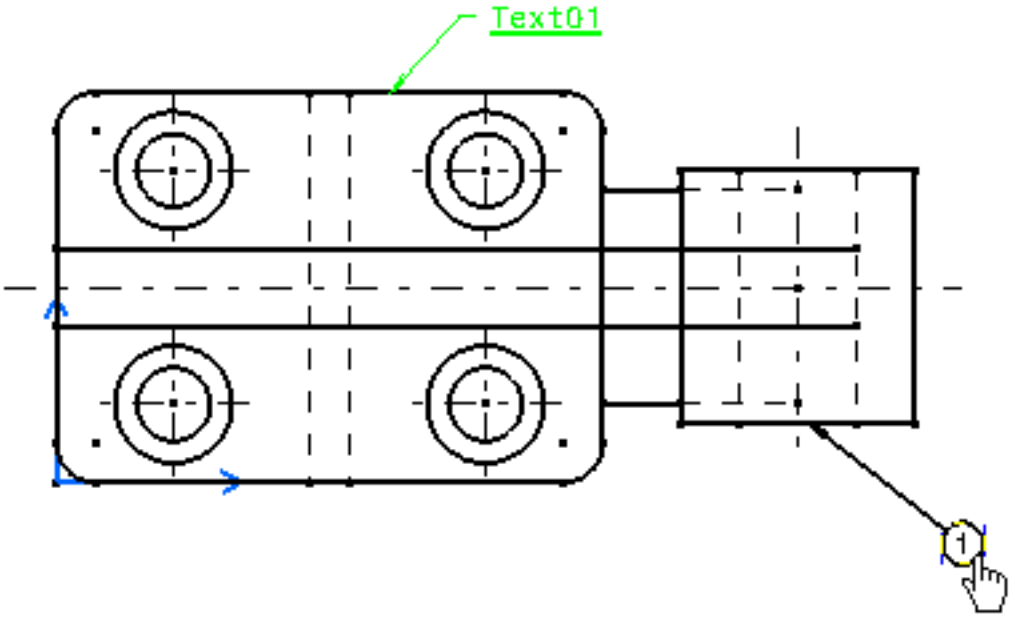
2. Select an element. For example, select the bottom line of the rectangle.
3. Click to define the balloon anchor point.



The **Balloon Creation** dialog box appears, with the value 1 pre-entered in the field.

4. You can enter another string or value as needed. For the purpose of this exercise, leave the pre-entered value as is.

5. Click **OK**.

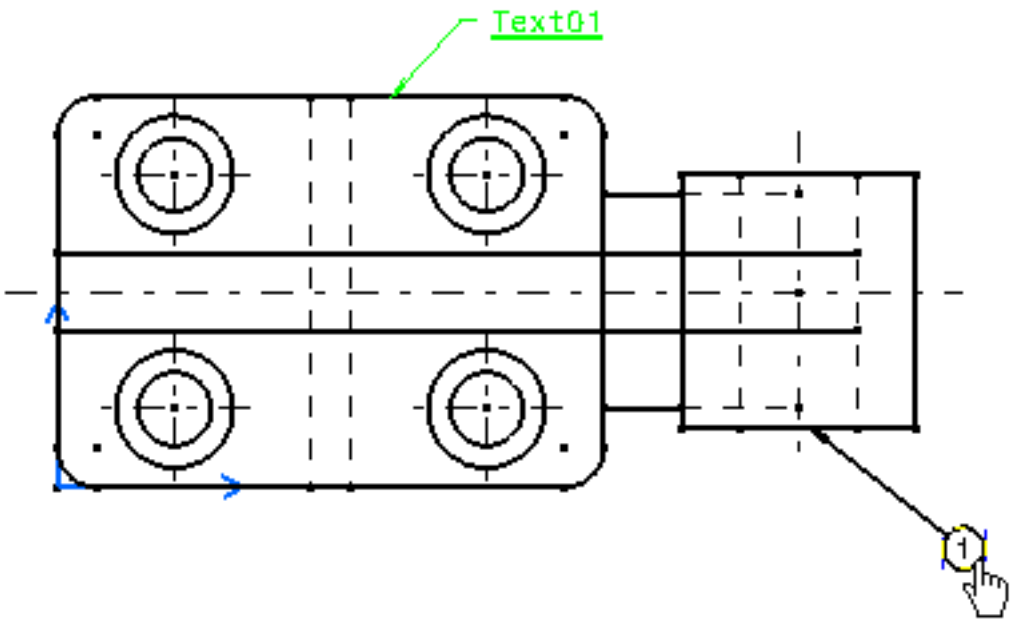


- The value that is edited in the **Balloon Creation** dialog box is simultaneously previewed on the drawing.
- When you create more than one balloon, the value of this balloon is automatically incremented.

Modifying a Balloon

This task shows you how to modify a balloon.

Open the Brackets_views03.CATDrawing document. Create a balloon.



1. Right-click the balloon you want to modify.

2. From the contextual menu, select **Properties**.

3. In the Properties dialog box, click the **Text** tab.

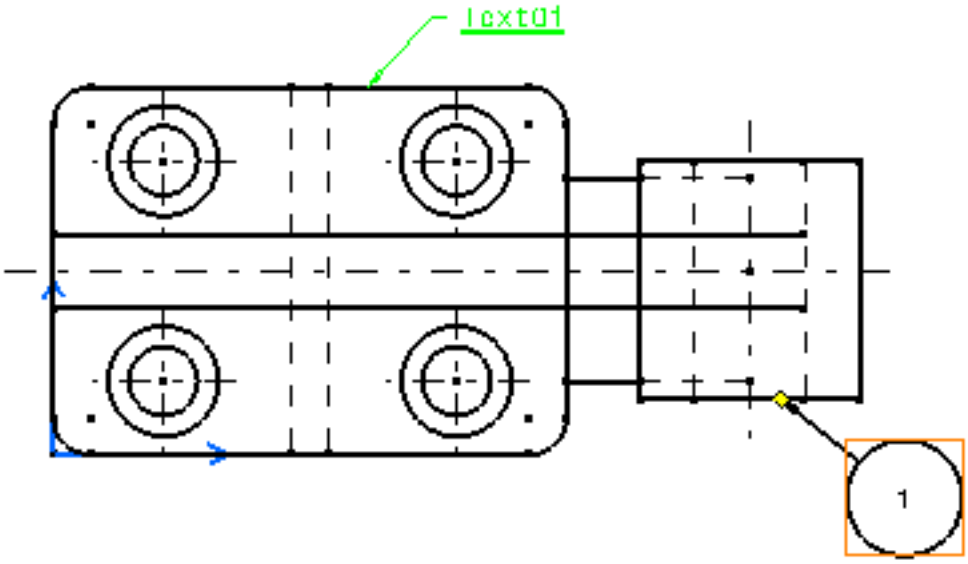
4. You will now define the balloon frame properties from the Frame drop-down list. By default, balloons are assigned a variable-size circle which adapts to the balloon text length. You have other options:

- You can display the balloon without a frame by selecting the None icon .
- You can assign a fixed-size frame to the balloon by selecting the fixed-size Circle icon .

For more information about fixed-sized frames, refer to Adding frames or sub-frames.

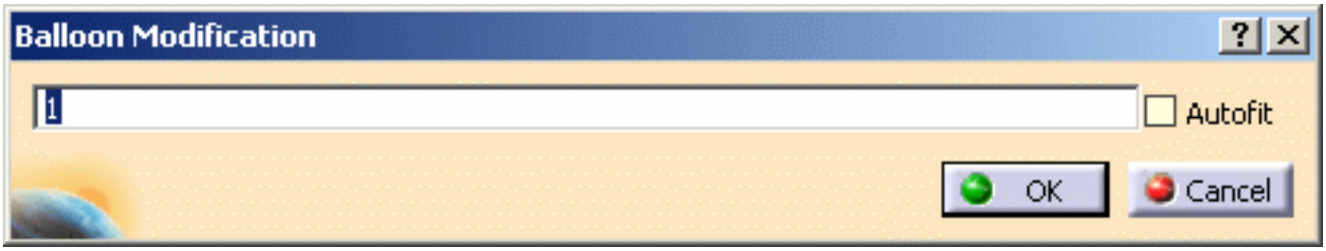
For the purpose of this exercise, select the fixed-size Circle icon .

5. Click OK to validate and close the Properties dialog box. The balloon size is modified.



6. Now, double-click the balloon. The Balloon Modification dialog box is displayed.

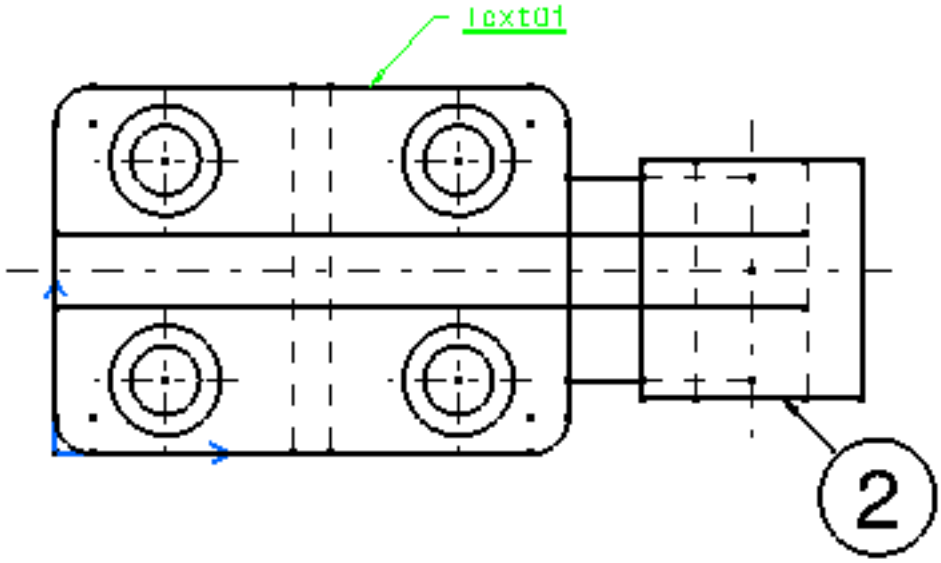
The **Autofit** option is active when the size of the balloon frame is fixed.



7. Modify the balloon value.

8. Select the **Autofit** option to adapt the size of the text to that of the balloon frame.

9. Click OK. The text is enlarged to fit within the balloon frame.



In the case of large texts, the **Autofit** option reduces the text size.

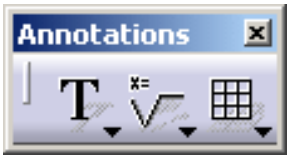
10. You can also modify the anchor point and thereby the position of the balloon.

Creating a Roughness Symbol

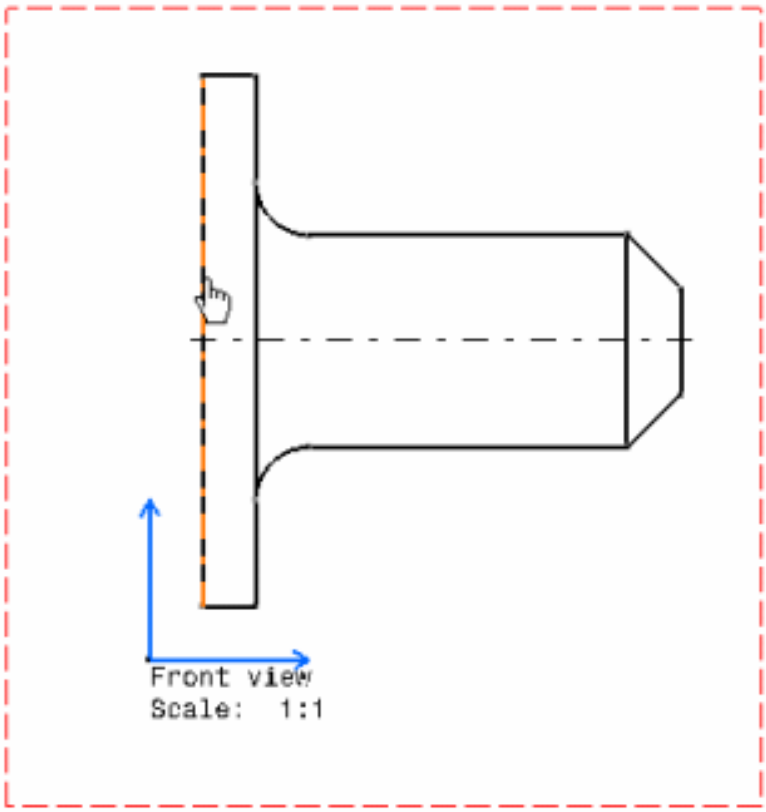
This task will show you how to create a roughness symbol.

Open the Roughness.CATDrawing document.

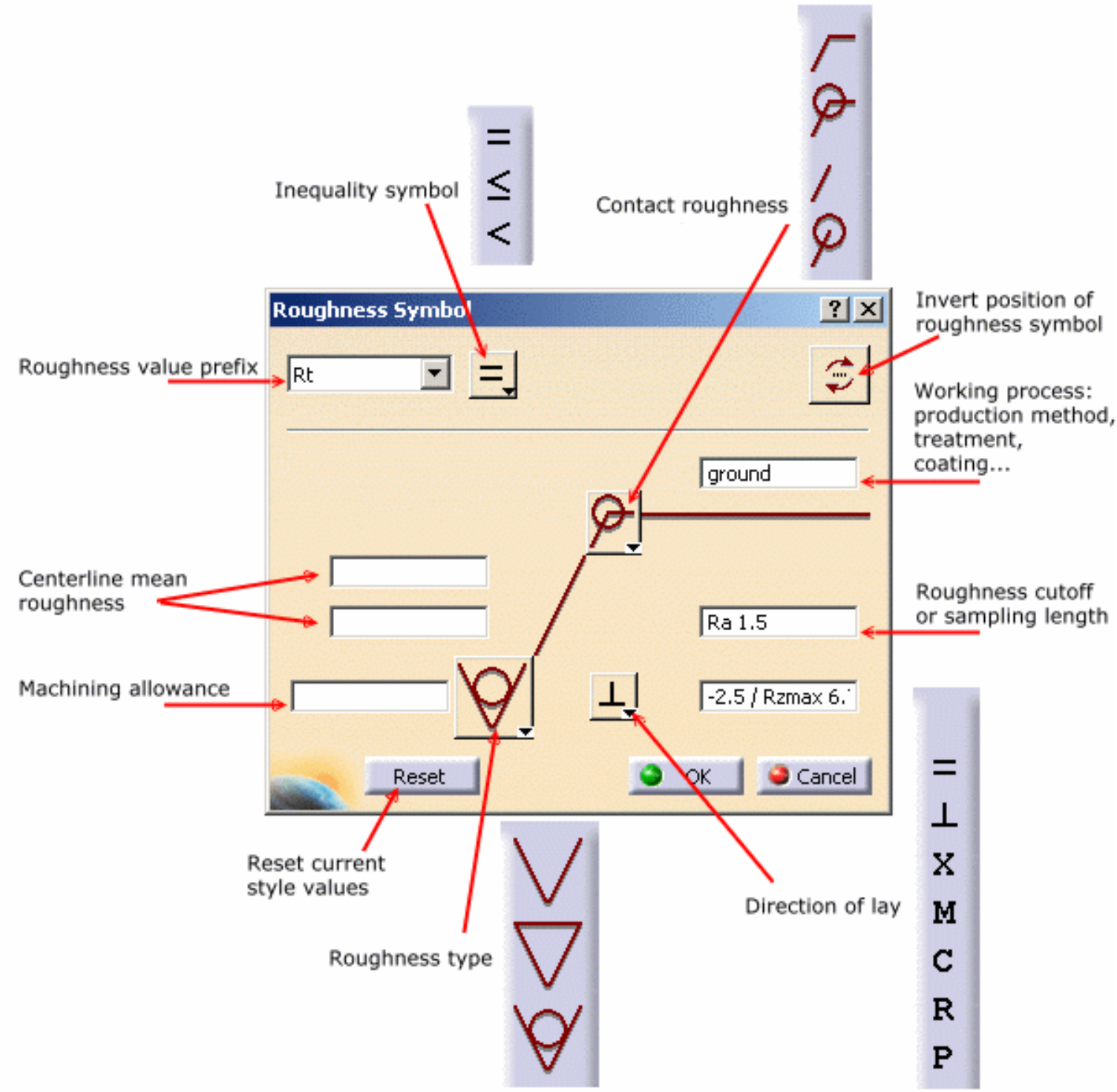
1. Click the Roughness Symbol icon from the Annotations toolbar.



2. Select the attachment point of the roughness symbol. The roughness symbol position and orientation will be associative to this point.



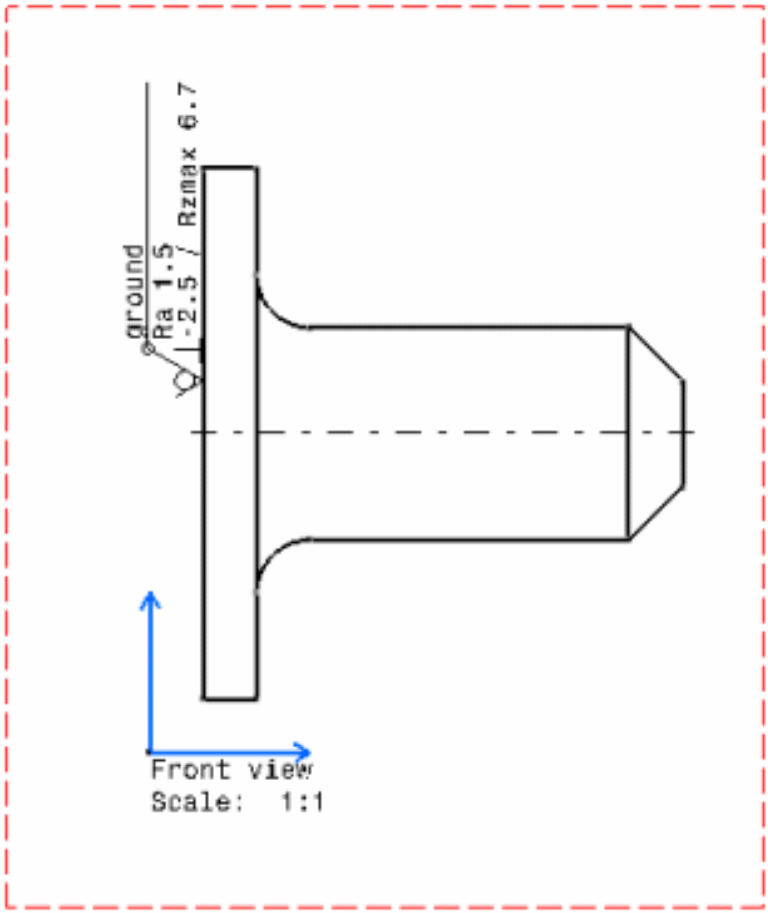
The **Roughness Symbol** dialog box is displayed.
The fields available in the Roughness Symbol dialog box depend on the standard used by the drawing, as defined by the administrator.


Inequality symbol
Contact roughness
Roughness Symbol
Invert position of roughness symbol
Roughness value prefix
Rt
Working process: production method, treatment, coating...
ground
Centerline mean roughness
Roughness cutoff or sampling length
Ra 1.5
Machining allowance
-2.5 / Rzmax 6.
Reset
OK
Cancel
Reset current style values
Direction of lay
Roughness type
X
M
C
R
P

Symbols	Definition
	Surface texture
	Surface texture and all surfaces around
	Basic
	All surfaces around
=	Lay approximately parallel to the line representing the surface
⊥	Lay approximately perpendicular to the line representing the surface
X	Lay angular in both directions
M	Lay multidirectional.
C	Lay approximately circular
R	Lay approximately radial
P	Lay particulate, non-directional, or protuberant
	Basic surface texture
	Material removal by machining is required
	Material removal by machining is prohibited.

3. Enter the required values in the various field(s).

4. Click **OK**. The roughness symbol is created.



5. If needed, modify the roughness symbol position by dragging it to the required location. Note that an extension line may be displayed between the roughness symbol and the element to which it is attached (providing this element is linear), depending on where you drag the roughness symbol.

- By default, there is a 1 millimeter space between the geometry and the extension line, as well as a 1 millimeter space between the end of the extension line and the roughness symbol. Those spaces cannot be customized.
- The roughness symbol default parameters are **1** for thickness and **solid** for line type. They cannot be customized.
- If you have selected the **Use style values to create new objects** option in **Tools -> Options -> Mechanical Design -> Drafting -> Administration** tab, the Roughness Symbol dialog box is pre-filled with custom style values (as defined in the Standards Editor). In this case, Properties toolbars and the Tools Palette are disabled during the creation of the roughness symbol.
  On the other hand, if you have not selected this option, the Roughness Symbol dialog box is pre-filled with the last entered values (if any). In this case, Properties toolbars and the Tools Palette are active during the creation of the of the roughness symbol.
- If you have selected the **Use style values to create new objects** option, you can reset the current style values in the Roughness Symbol Editor dialog box at any time using the **Reset** button.
- At any time, you can modify the roughness symbol. For this, double-click the roughness symbol to be modified and enter the desired modifications in the displayed Roughness Symbol dialog box.
- By default, the roughness symbol's orientation is determined according to the orientation of the line it is associated with. You can modify this orientation using the **Invert** button available in the Roughness Symbol dialog box.
- When this is not already the case, you can link roughness symbol position and orientation to another element, see Making an Existing Annotation Associative.

Creating a Welding Symbol

This task will show you how to create a welding symbol. You can set text properties either before or after you create the text.

Welding symbols

Square butt weld

Singe V butt weld

Single bevel butt weld

Flare V butt weld

Flare bevel butt weld

Single U butt weld

Single J butt weld

Fillet weld

Spot weld

Back weld

Steep-flanked single-bevel butt weld

Steep-flanked single-V weld

Plug weld

Removable backing strip used

Permanent backing strip used

Surfacing weld

V flare weld

Spot weld

Complementary symbols

Weld with flat face

Weld with convex face

Weld with concave face

Flush finished weld

Fillet weld with smooth blended face

Finish symbols

C finish symbol

F finish symbol

G finish symbol

H finish symbol

M finish symbol

R finish symbol

Complementary indications

Field weld

Weld-all-around

Weld text side (up or down)

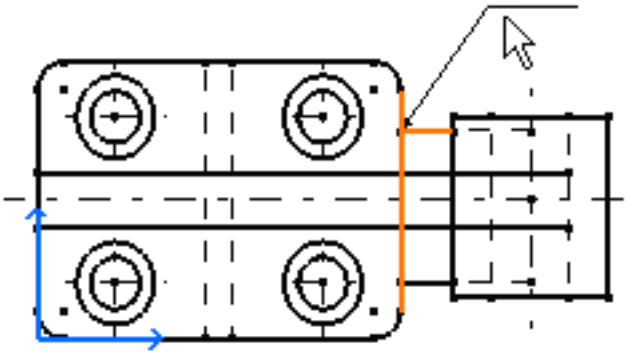
Indent line side (up or down)

Weld tail

Reference

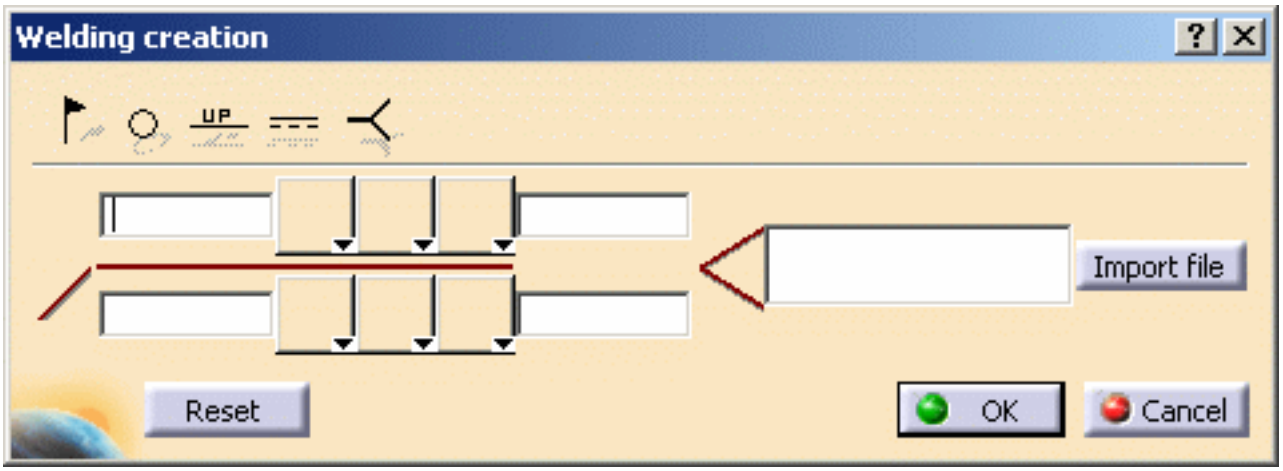
Open the Brackets_views03.CATDrawing document.

1. Click the **Welding Symbol** icon from the **Annotations** toolbar (**Symbols** sub-toolbar).
2. Select an element or click in the free space to position the leader anchor point, and then click to validate. The welding leader will appear.



3. Move the cursor to position the welding symbol and then click at the chosen location.

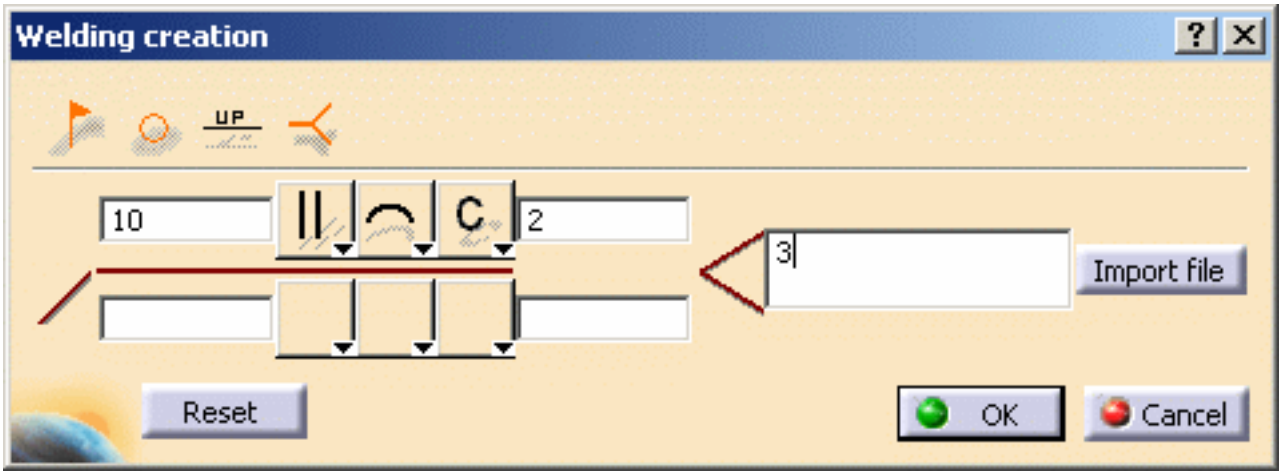
   The **Welding creation** dialog box is displayed.



4. Type the desired values in the upper and/or lower field(s).
5. Click the symbol buttons to choose the welding symbol, complementary symbols and/or finish symbols.

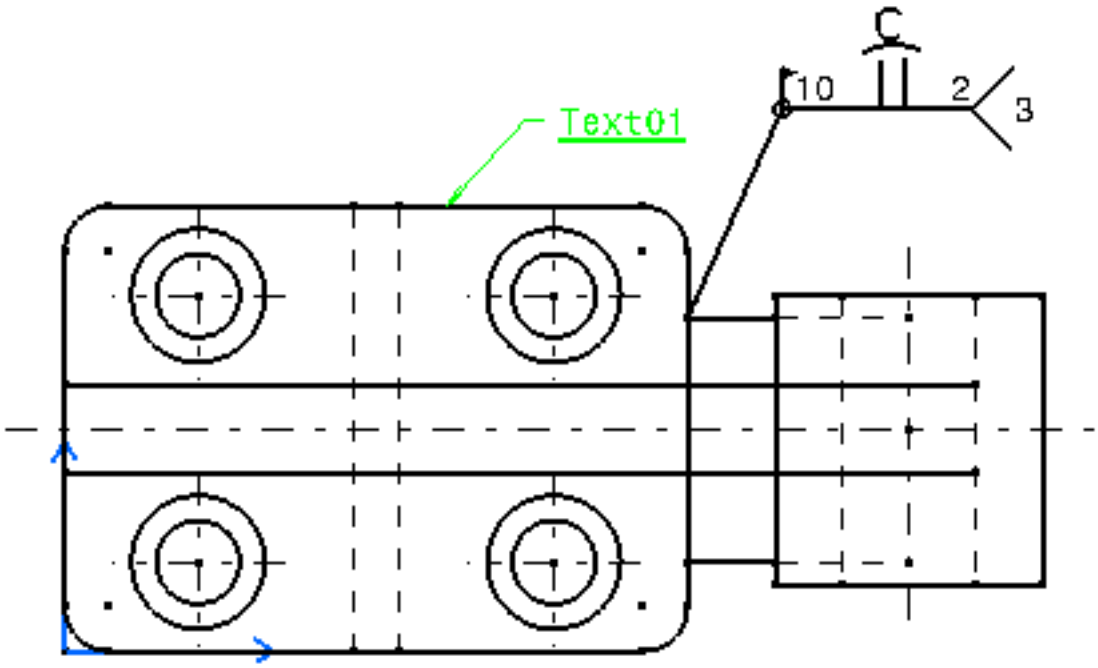
   The welding symbols available depend on your standard.

6. If you want to add complementary indications like a field weld or a weld tail, for example, click the appropriate button.



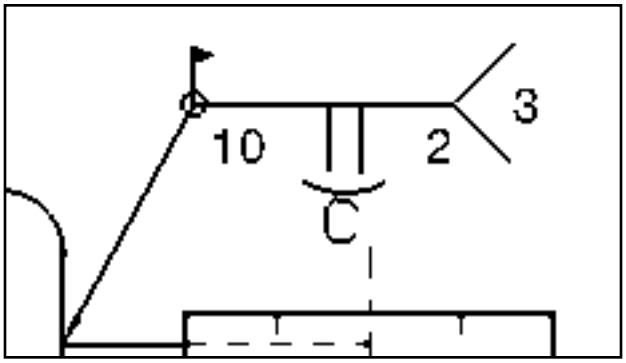
7. Click **OK**.

The welding symbol is created.



8. If needed, modify the welding symbol position by dragging it to the required location.

9. Double-click on the welding symbol to edit it, and change the weld text side for example by clicking the Up/Down switch button.



- If you have selected the **Use style values to create new objects** option in **Tools -> Options -> Mechanical Design -> Drafting -> Administration** tab, the Welding creation dialog box is pre-filled with custom style values (as defined in the Standards Editor). In this case, Properties toolbars and the Tools Palette are disabled during the creation of the welding symbol.
  On the other hand, if you have not selected this option, the Welding creation dialog box is pre-filled with the last entered values (if any). In this case, Properties toolbars and the Tools Palette are active during the creation of the welding symbol.
- You can reset the current style values in the Welding creation dialog box at any time using the **Reset** button.
- You can close the tail (reference) using a rectangle variable-size frame . For more information about adding frames, refer to Adding Frames or Sub-Frames.
- At any time, you can modify the welding symbol. To do this, double-click the welding symbol to be modified and enter the modifications in the displayed dialog box.
- You can import a plain text file (.txt) to use as a reference (specification, process or other) by clicking the Import File button.

Modifying Annotation Positioning

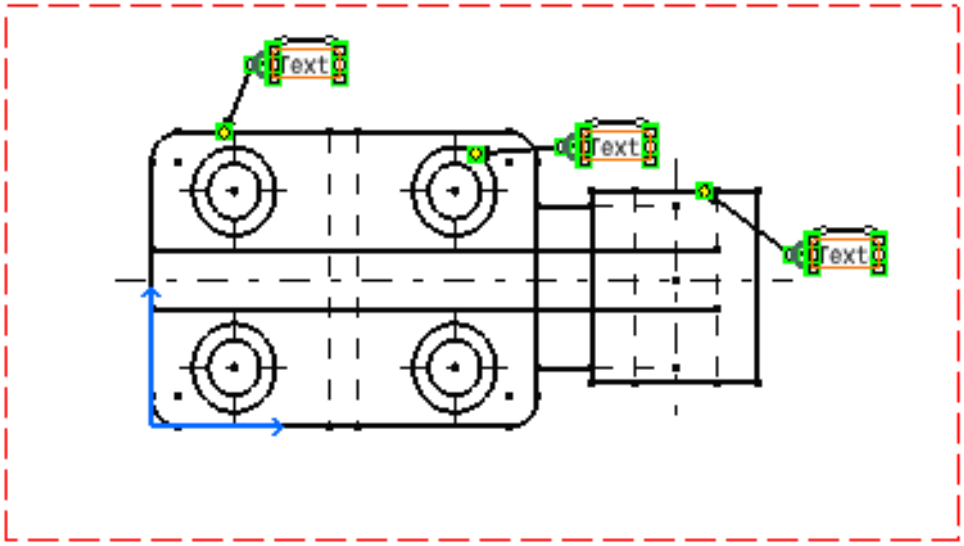
This task will show you how to assign new positioning to existing annotations.
You can also modify the position of the views using the same dialog.

Open the IntDrafting_Annotations_Positioning.CATDrawing document.

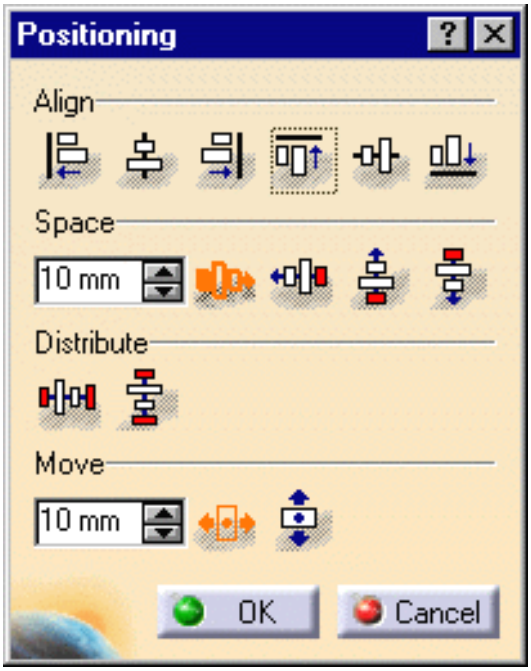
1. Multi-select the annotations to be newly positioned. In this example, multi-select text.

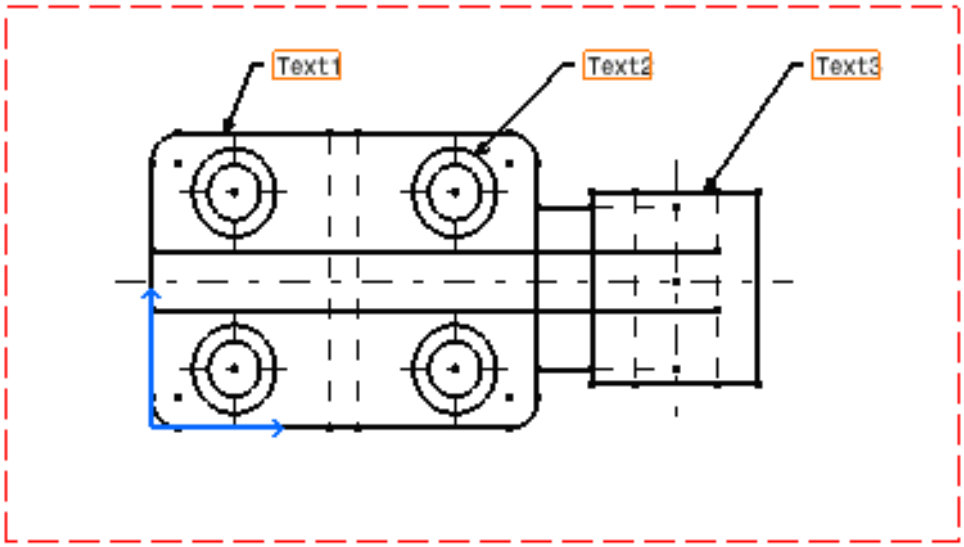
2. Select the **Tools -> Positioning -> Element Positioning** command from the menu bar.

The **Positioning** dialog box appears:



3. Select the **Align** to top option .





Align to the left

The reference text is the text, among the selected texts, that is positioned the most at the left.
The text anchor point is moved to the left (for example, from the bottom center to the bottom left).
The texts are aligned vertically relatively to the reference text origin point (same x abscissa as for the reference text).

Align to the center

The reference text is positioned at the middle of both left and right extremity points.
The text anchor point is moved to the center (for example, from the top left to the top center).
The texts are aligned vertically relatively to the reference text origin point (same x abscissa as for the reference text).

Align to the right

The reference text is the text, among the selected texts, that is positioned the most at the right.
The text anchor point is moved to the right (for example, from the middle center to the middle right).
The texts are aligned vertically relatively to the reference text origin point (same x abscissa as for the reference text).

Align to the top

The reference text is the text, among the selected texts, that is positioned the most at the top.
The text anchor point is moved to the top (for example, from the bottom left to the top left).
The texts are aligned horizontally relatively to the reference text origin point (same y coordinate as for the reference text).

Align to the middle

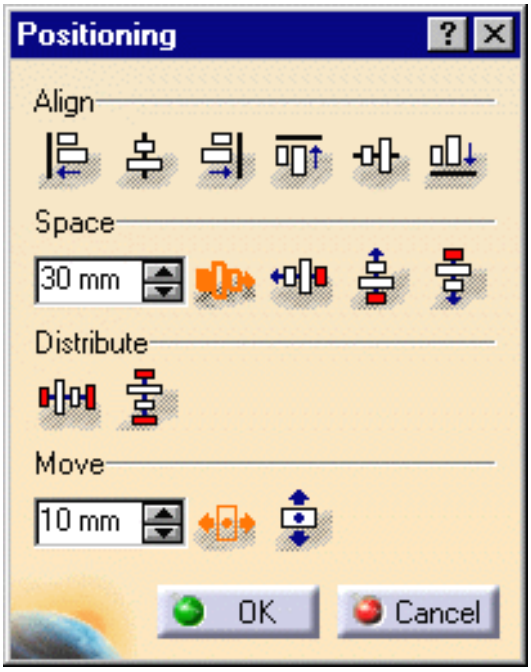
The reference text is positioned at the middle of both top and bottom extremity points.
The selected texts are assigned the middle attribute as text origin (for example, from the top left to the middle left).
The texts are aligned horizontally relatively to the reference text origin point (same y coordinate as for the reference text).

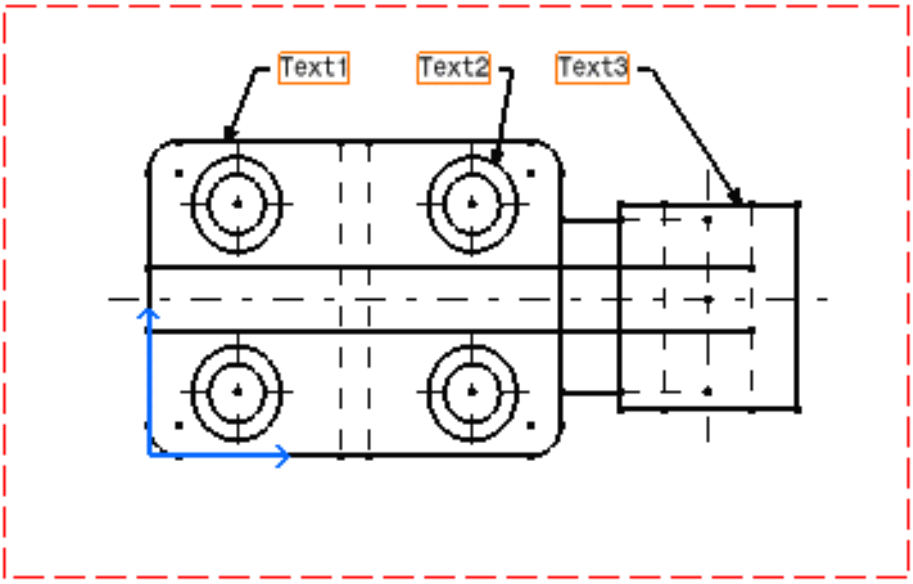
Align to the bottom

The reference text is the text, among the selected texts, that is positioned the most at the bottom.

The text anchor point is moved to the bottom (for example, from the top left to the bottom left).
The texts are aligned horizontally relatively to the reference text origin point (same y coordinate as for the reference text).

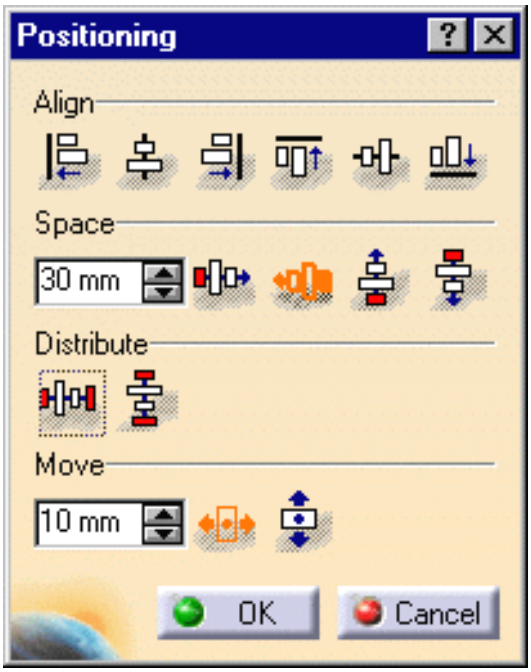
4. Select the **Space** from left to right option and set the **Space** value to **30mm**.

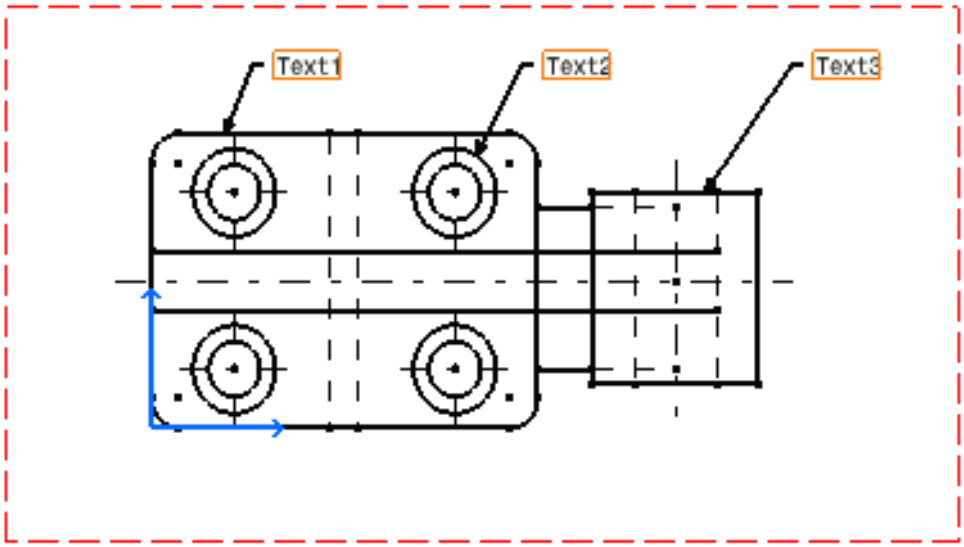




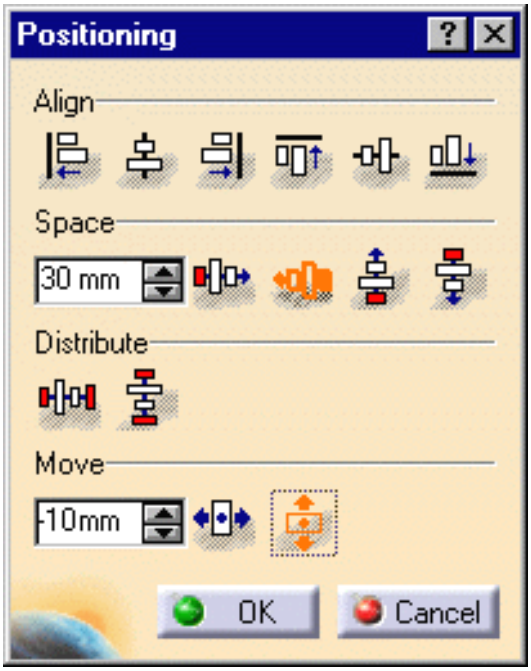
Note that when you select a **Space** option, the modification does not appear similarly on the drawing. This modification only appears when you enter the new **Space** value in the **Positioning** dialog box or when you select a **Space** value.

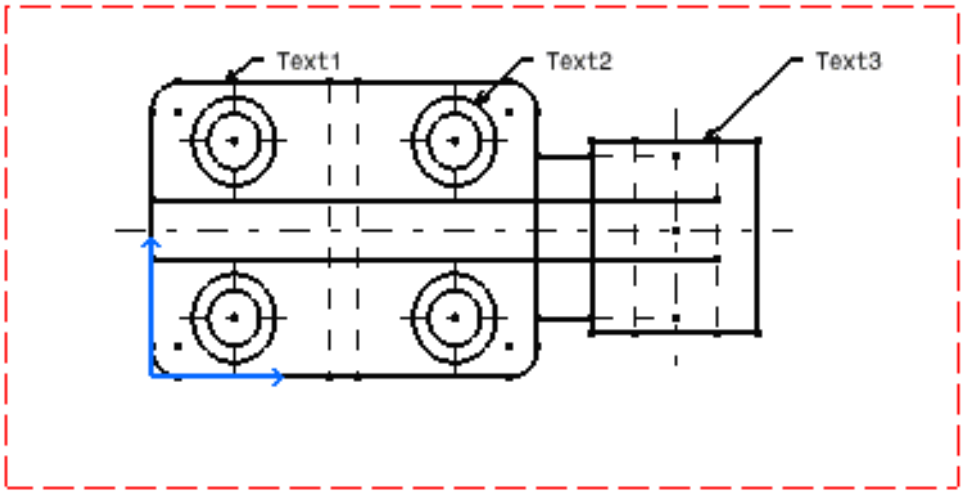
5. Select the **Distribute** horizontally option .





6. Select the **Move** vertically to top option and set the **Move** value to **-10mm**.





Note that when you select a Move option, the modification does not appear similarly on the drawing. This is only the case once you enter the new Move value in the Positioning dialog box or when you select a spacing option.

Creating/Modifying a Table

This task shows you how to create and edit a table by adding text, inserting columns and/or rows, merging cells, inverting lines and/or columns, switching lines and columns, and inserting views. You can also split a table, import a table, and insert a view in a table.

Choose a task:

- creating a table,
- editing and modifying a table,
- splitting a table,
- importing a table,
- inserting a view in table.

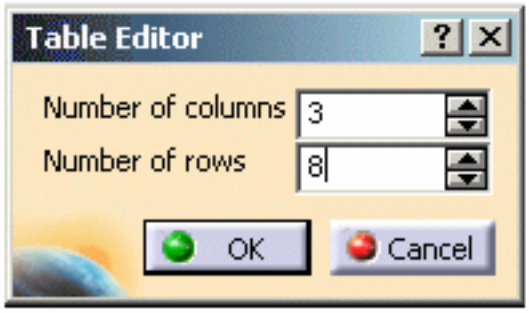
Creating a table

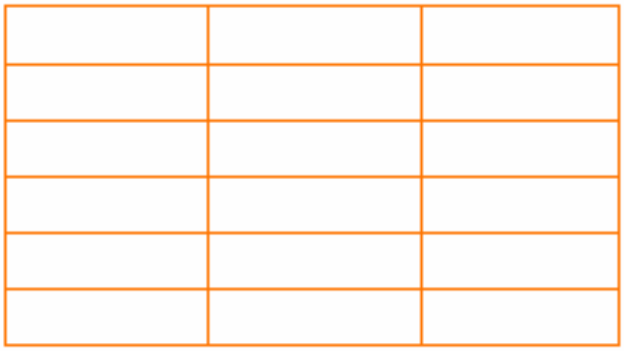
Create a new sheet and a new view.

1. On the **Annotations** toolbar, click **Table** .
2. Click a point in the drawing to specify the table position.

   Tables cannot be associative. Selecting an element in the drawing does not make the table associative to this element.

3. In the Table Editor dialog box that appears, enter the number of columns and rows you want for the table.





- The line height corresponds to the height of a string.
- The line width corresponds to 5 times the height of a string.

4. Click **OK** to validate the table creation.

Editing and modifying a table

Create a table as explained earlier in this task.

1. Select the table and drag it to another position.
2. Double-click the table to switch it to edition mode.

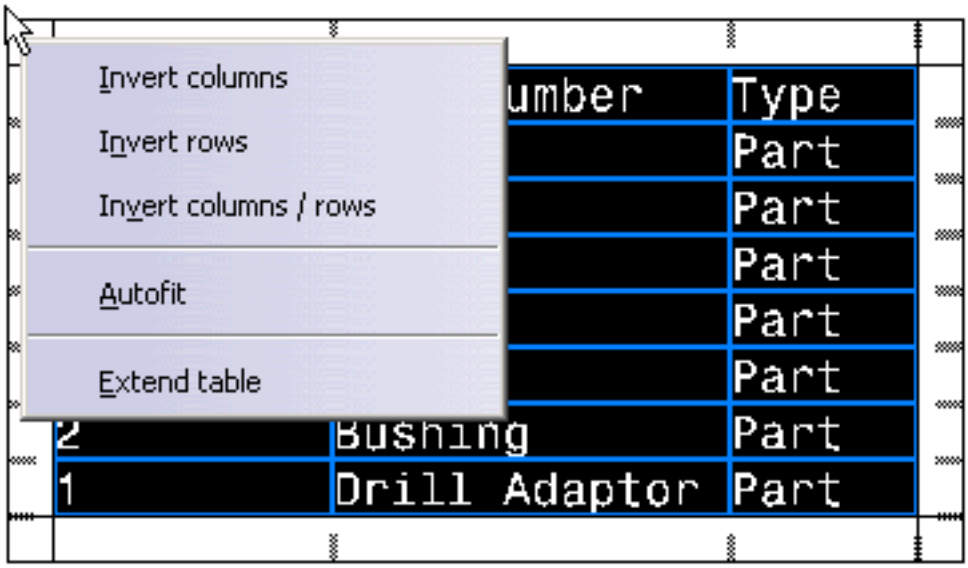
Quantity	Part Number	Type
1	Base	Part
2	Rod	Part
1	Slide	Part
2	Collar	Part
4	Thumb	Part
2	Bushing	Part
1	Drill Adaptor	Part

You cannot move a table while it is in edition mode.

While in edition mode, you can:

- Select a column by clicking above the column when the symbol appears.
- Select a line by clicking on the left-hand side of the row when the symbol appears.
- Leave edition mode by clicking outside the table.

3. To access the general contextual menu, right-click the corner of the frame around the table.



This contextual menu allows you to:

- invert columns.
- invert rows.
- turn rows into columns and columns into rows.
- fit the text in the cells by automatically defining the optimal cell size.
- extend the table by adding columns and/or rows to it.

4. Choose **Invert rows** in the contextual menu. Rows are inverted, i.e., the last row becomes the first one, the first row becomes the last one, etc.

1	Drill Adaptor	part
2	Bushing	part
4	Thumb	part
2	Collar	part
1	Slide	part
2	Rod	part
1	Base	part
Quantity	Part Number	Type

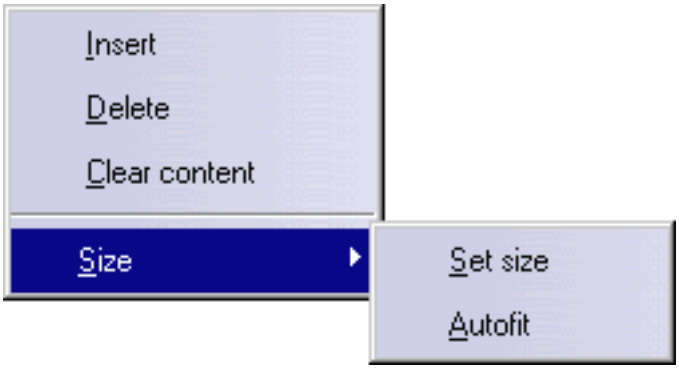
5. Choose **Invert Columns** in the contextual menu. Columns are inverted.

part	Drill Adaptor	1
part	Bushing	2
part	Thumb	4
part	Collar	1
part	Slide	2
part	Rod	2
part	Base	1
Type	Part Number	Quantity

6. Select **Invert Columns / Rows** in the contextual menu. Rows and Columns are inverted:

1	2	4	2	1	2	1	Quantity
Drill Adaptor	Bushing	Thumb	Collar	Slide	Rod	Base	Part Number
part	part	part	part	part	part	part	Type

7. Right-click a column or row to access the related contextual menu, which lets you:
   - Insert a column/row.
   - Delete a column/row.
   - Clear the content of a column/row.
   - Modify the size of a column/row by either:
     - setting a new size for the column/row.
     - "autofitting" the size (the optimal cell size is automatically defined in order for text to fit in it).



The size of a column or row and the text properties depends on the insertion point in the table. If you insert a column/row:
- in the middle of a table, the size and text properties are the same as the preceding column/row.
- at the beginning of the table, the size is the same as the first column/row and the text properties are the same as the current text style.

8. Right-click two cells and choose **Merge**.

You cannot merge a cell that has already been merged.

9. Select the new cell formed by the two cells you have merged and choose **Unmerge** to split them in two cells again.

10. Double-click on the text of a cell. The **Text Editor** dialog box appears: modify the text and click **OK** to validate.

11. To choose vertical and horizontal text alignment, use the Anchor point tool . Align the text of a cell on the right using .
12. Right-click a cell, and select **Properties** from the contextual menu. The properties available are the same as those available for texts. Refer to Editing Text Properties for more information.
13. On the **Font** tab, specify a color, red for example, and click **OK**. The text in the selected cell is now red.

When editing cell properties, note that a number of properties do not apply to the selected cell, but to the table and all its cells.

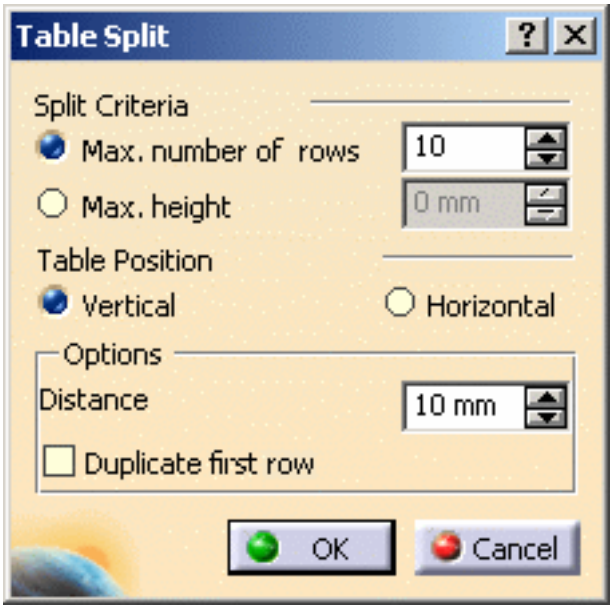
- On the **Text** tab:
  - **X** and **Y** position
  - (Orientation) **Reference**
  - **Orientation**
  - **Blank Background**
- On the **Graphic** tab (**Lines and Curves** section):
  - **Color**
  - **Linetype**
  - **Thickness**

Splitting a table

Open the Split_tables.CATDrawing document. It contains a table that you will split into several tables.

REF.	X	Y	Diameter
A	751.11	424.59	44.72
B	821.11	424.59	20
C	820	480	44.72
D	890	540	44.72
E	640	590	44.72
F	550	590	113.14
G	476.7	441.38	63.25
H	346.98	501.48	63.25
I	400	530	28.28
J	440	550	28.28
K	400	570	20
L	316.7	586.86	63.25
M	400	620	28.28

1. Right-click the table and choose **Split Table**. The Table Split dialog box appears.



It contains the following options:

- **Max. number of rows**: if you want to split the table so that each new table contains a maximum number of rows, select this option and enter the wanted number of rows in the associated field.
- **Max. height**: if you want to split the table so that each new table has a maximum height, select this option and enter the wanted height in the associated field.
- **Vertical**: check this option to create the new tables one below the other.

- **Horizontal**: check this option to position the new tables one next to the other.
- **Distance**: indicate the distance you want between each new table.
- **Duplicate first row**: check this option if you want to duplicate the first row in each new table.

2. Select **Max. number of rows**, and enter 5 in the corresponding field.
3. Select **Vertical**.
4. In the **Distance** field, type 5 mm.
5. Select **Duplicate first line**.
6. Click **OK**. The table is split into several tables, according to the criteria you specified.

REF.	X	Y	Diameter
A	751.11	424.59	44.72
B	821.11	424.59	20
C	820	480	44.72
D	890	540	44.72
E	640	590	44.72

REF.	X	Y	Diameter
F	550	590	113.14
G	476.7	441.38	63.25
H	346.98	501.48	63.25
I	400	530	28.28
J	440	550	28.28

REF.	X	Y	Diameter
K	400	570	20
L	316.7	586.86	63.25
M	400	620	28.28

Importing a table

You can import a table in a drawing (of the .csv type only).

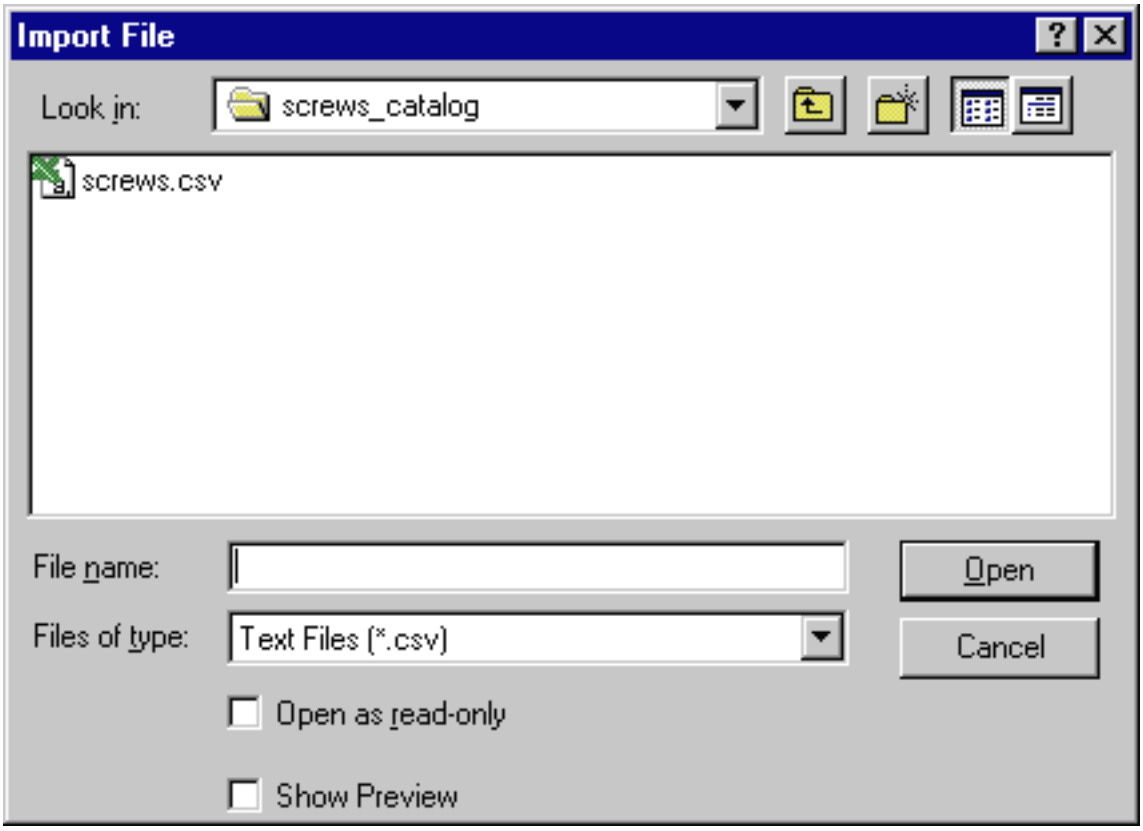
To perform this scenario, you must have a .csv file at your disposal.

Note that this functionality is based on system options. Indeed, in a CSV (Comma Separated Values) file, each line corresponds to a line in the table; within a given line, each cell is separated by a specific character that depends on the decimal separator specified in the system options.

This decimal separator determines the character that will be interpreted as the cell separator:

- if the decimal separator is a dot, then the comma (,) is used as the column separator.
- if the decimal separator is a comma, then the semi-column is used as the column separator.

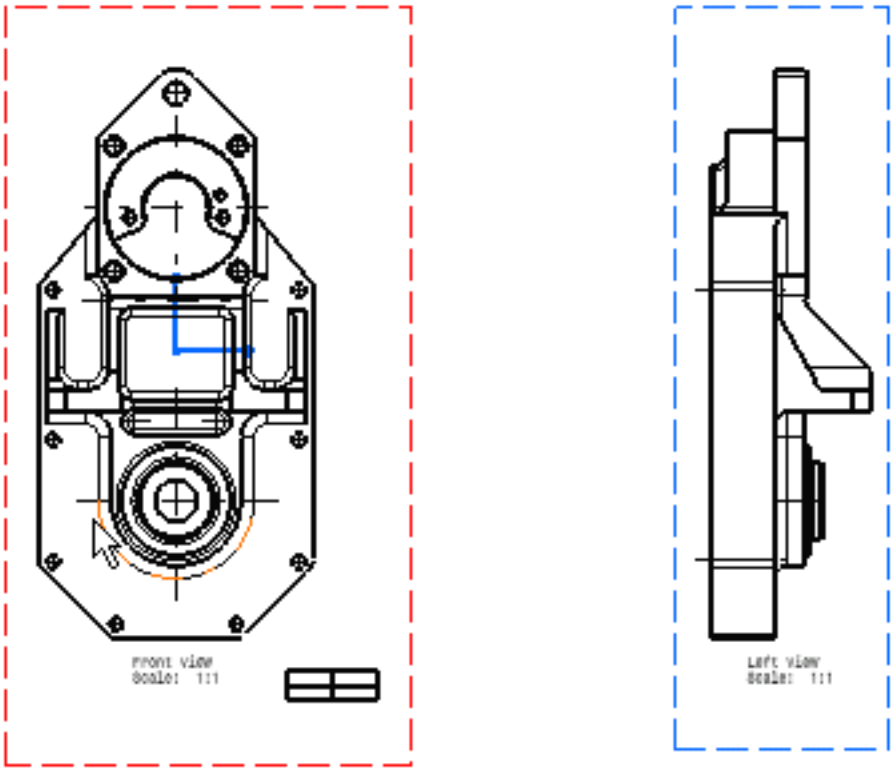
1. Click the **Import Table** icon and select the table you want to import.



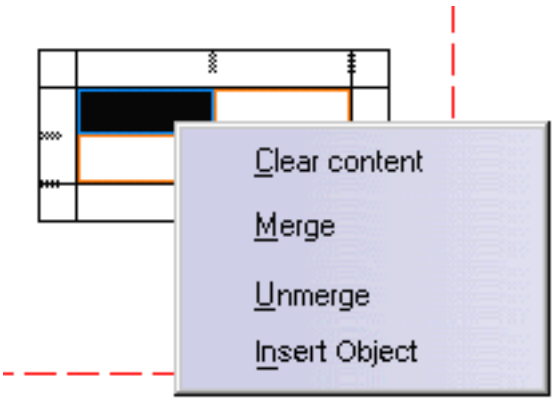
2. Click **Open**. The table is imported into the drawing.

Inserting a view a in table

Open the GenDrafting_part.CATDrawing file. Create a table in the front view.



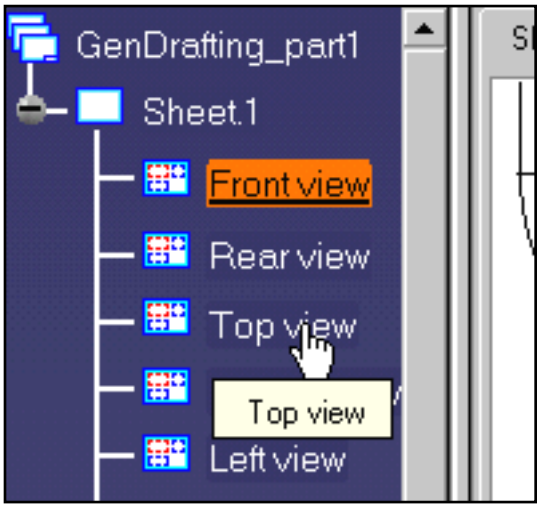
1. Double-click the table to switch it to edition mode.
2. Right-click the cell you want to fill. Select **Insert Object**.



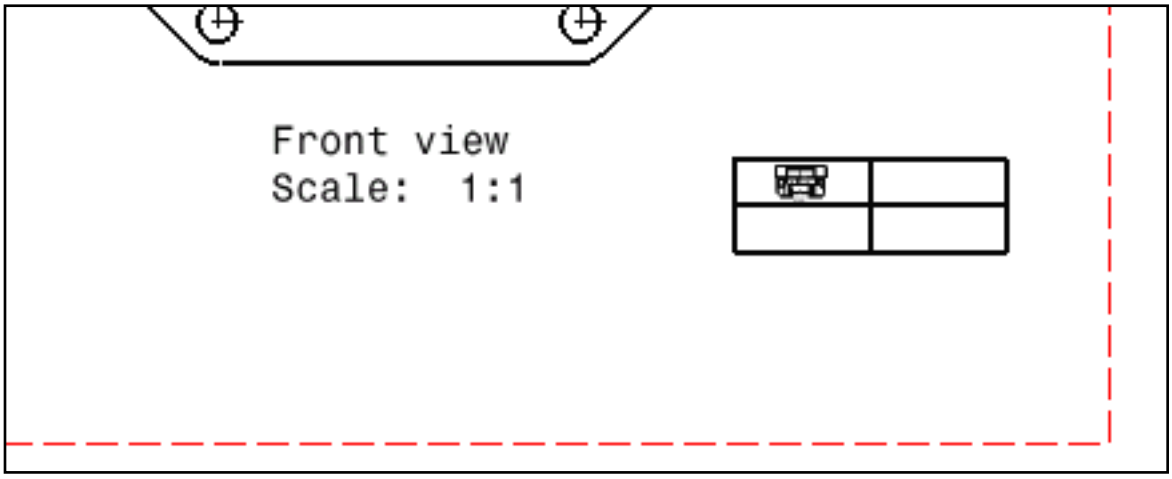
3. Choose the view you want to insert by clicking the view in the drawing or in the tree.

- You cannot select the view containing the table,
- The view must be in the same drawing.
- If you modify the 3D part and update the drawing, the view in the table will be updated as well.

Choose the Top view:



The top view is inserted in the table, and it is resized so as to fit the cell. You can resize the cell if you want to enlarge the view in the table.



Finding And Replacing Text

This task explains first how to locate a string of characters and then how to replace it.

Strings can be found and replaced in the following elements:

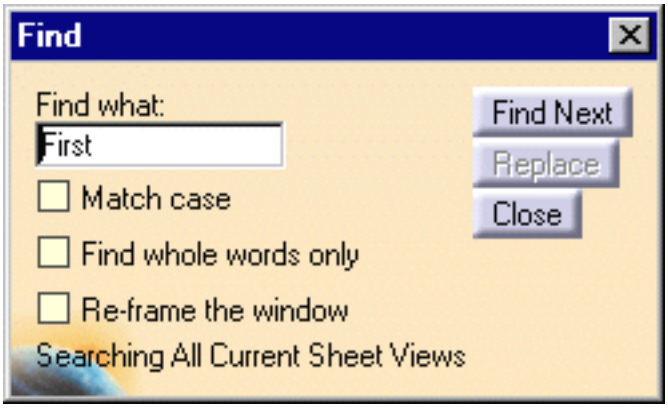
- balloons
- datum features
- datum targets
- dimensions
- texts

Open the IntDrafting_Text_Replace.CATDrawing document.

1. Select the **Edit->Find** item from the menu bar.
The **Find** dialog box appears.

2. Select any of the optional settings. For example, enter **First** as the **Find what** text.

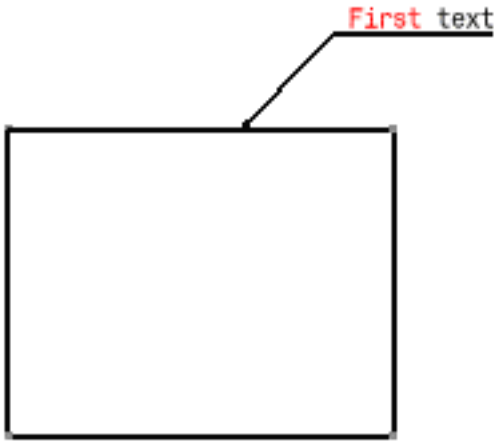
3. Select Find Next.

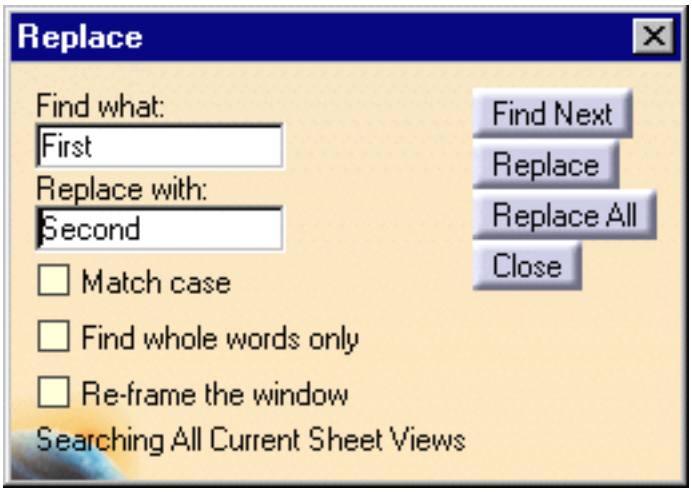


The following message appears in the dialog box: **Searching All Current Sheet Views**. If you previously selected a given number of sheets or elements in the document, the message will be **Searching All Current Elements.**

The first instance found is red colored.

4. If needed, select Find Next to search for other instances.





5. Select Replace.

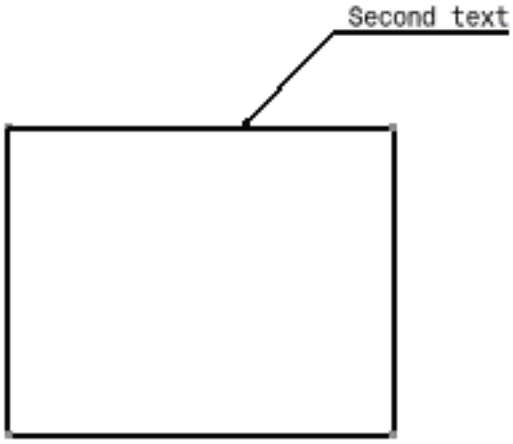
The **Replace** dialog box now appears.

6. Enter the text you want to use as replacement text and select Replace again. For example, enter **Second** as the **Replace with** text.

To replace all instances of the text, select Replace All.

You can also match case, find whole words only or re-frame the window.

7. Select Close.



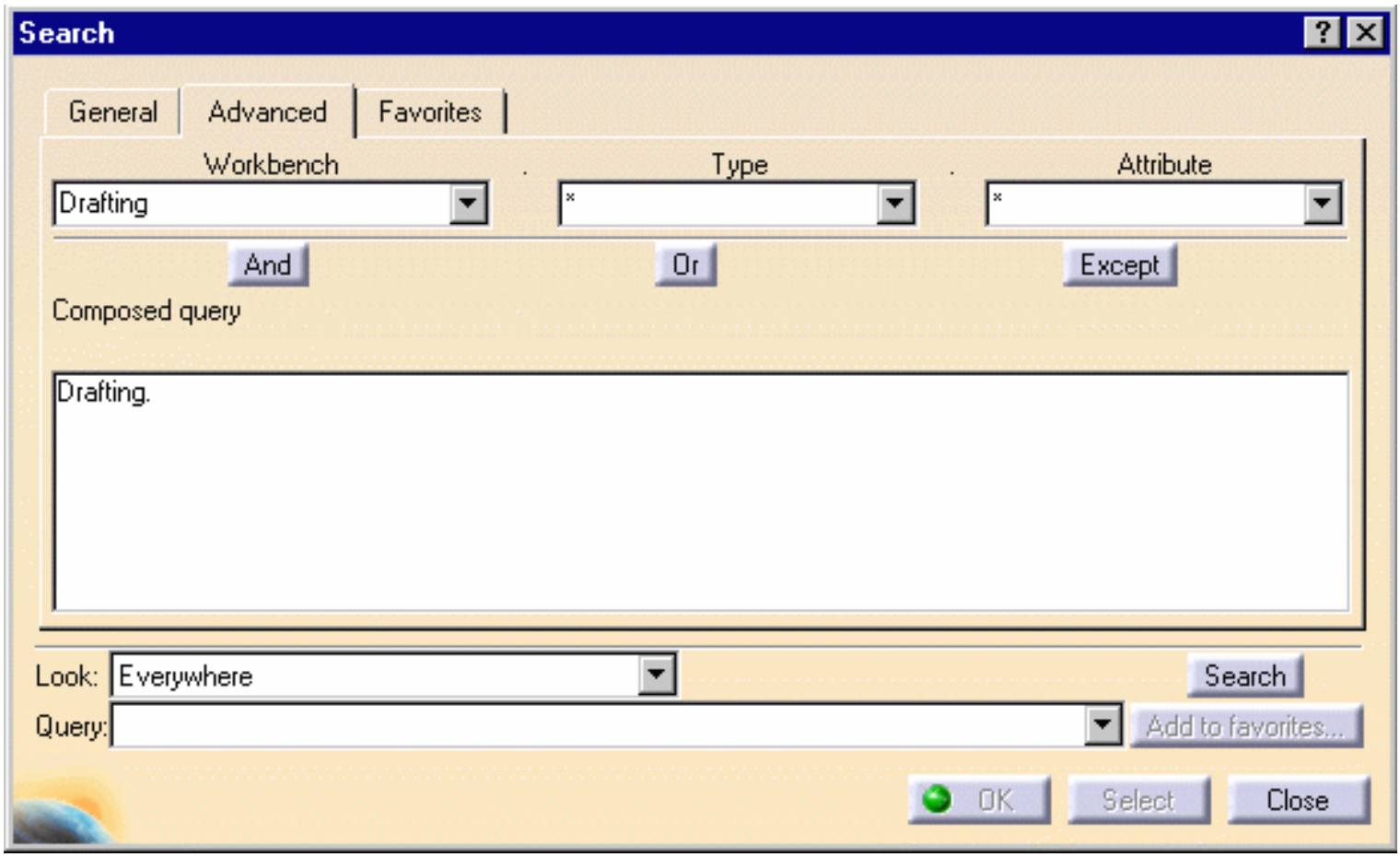
Note that you can directly access the **Replace** dialog box by selecting the **Edit->Replace** item from the menu bar.

Performing an Advanced Search

This task will show you how to use the advanced search command in the Drafting workbench.

First, refer to the *Infrastructure User's Guide* to learn more about advanced search.

1. Select the **Edit->Search...** command then click the Advanced tab:



2. Choose Drafting as the workbench.

Any element type has the following attributes:

- Name: indicate the name of the searched element
- Color: select a color from the color chooser or use the color of an existing element
- Set: a selection set indicating a numeric value with the corresponding unit of measure.

Some elements have additional types:

Type	Additional attributes	Value to select or to key in
Balloon	Part name	name of the searched element
Datum Feature	Reference name	name of the searched element
Datum Target	Reference name	name of the searched element
	Size	size indicated in the searched element
Dimension	Type	type of dimension (angle, diameter, radius, length, etc.) searched
	value	dimension value searched
	Not associative on 3D	Yes/No
	Not updateable	Yes/No
	Fake	Yes/No
	Driving dimension	Yes/No
	True	Yes/No
Geometrical tolerance	value	tolerance value searched
	type	tolerance type (circularity, concentricity, flatness, parallelism, etc.) searched
Text	having attribute links	Yes/No
	text string	text string searched

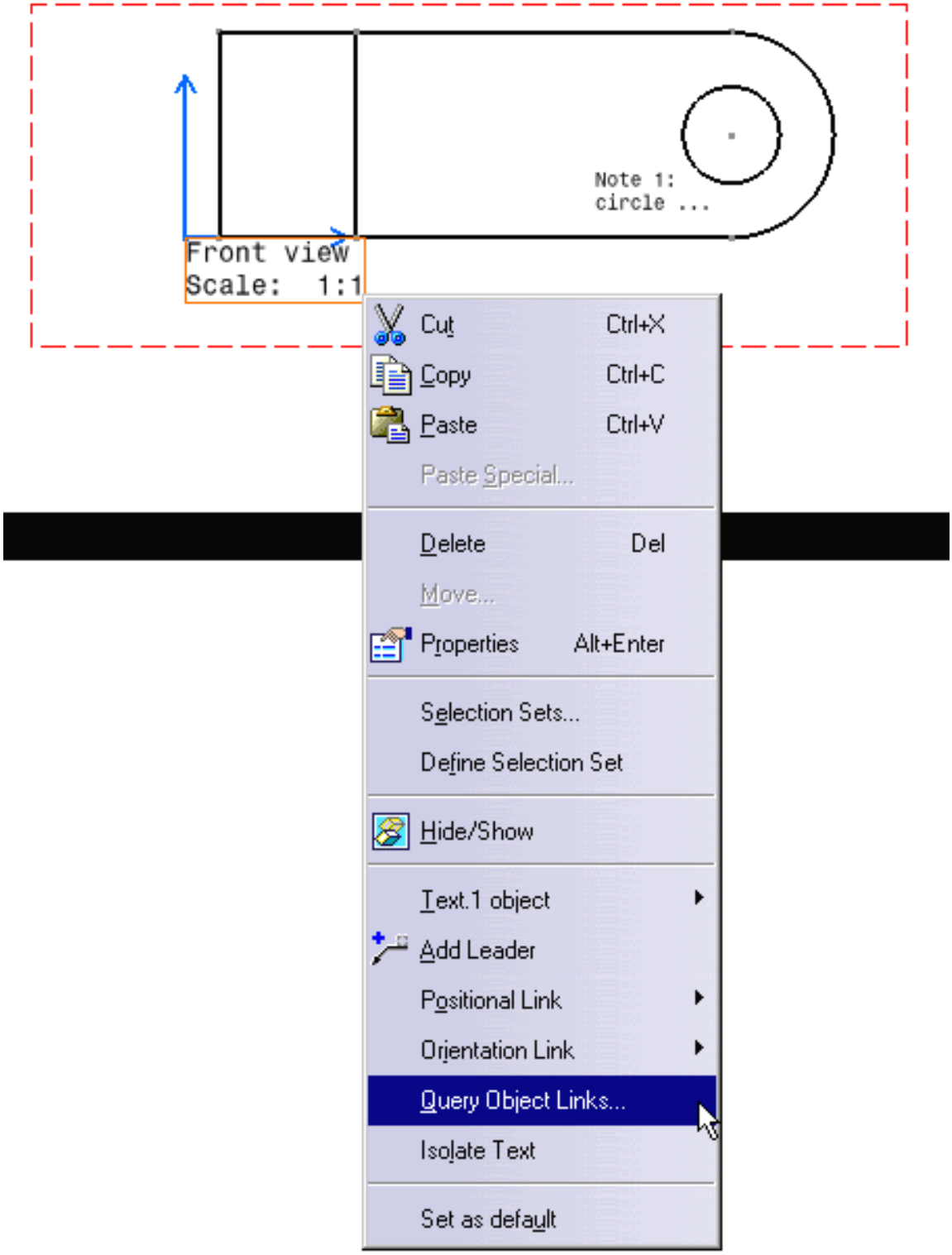
3. Select an operating sign in the first combo box.
4. Select (if there is a combo box) or key in the value you are looking for.

Querying Annotation Links

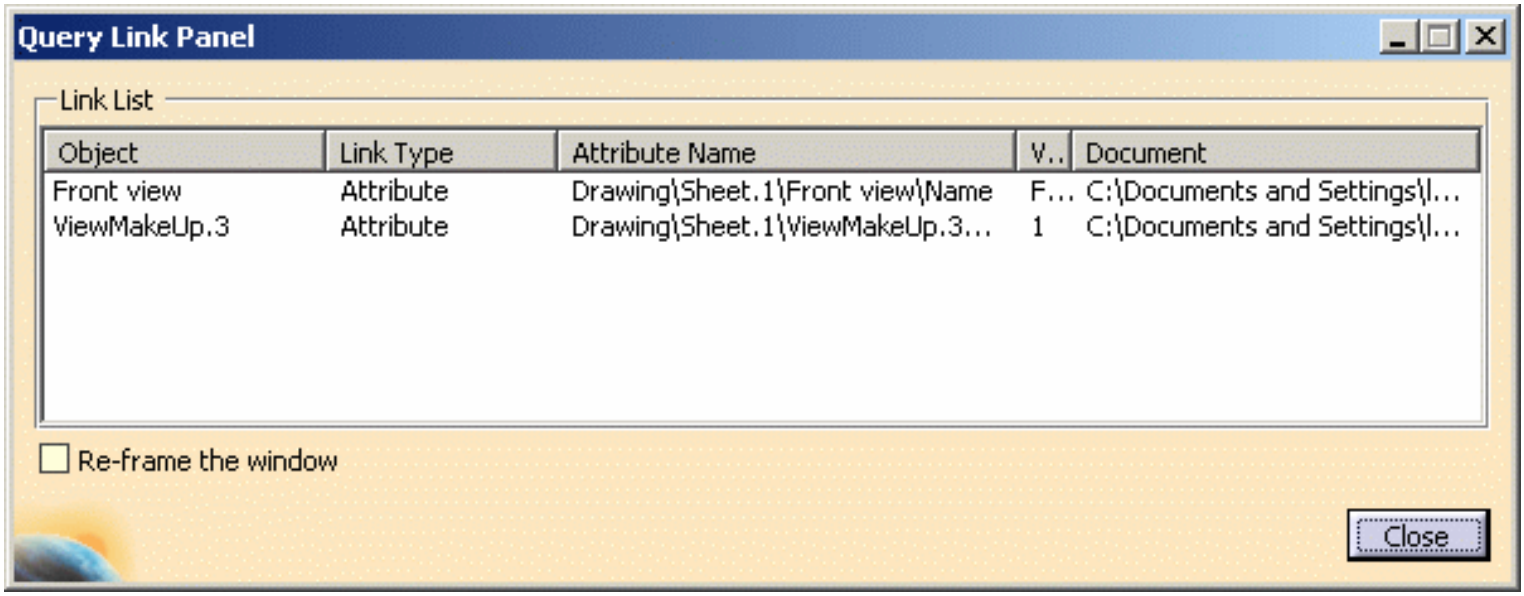
This task explains how to query annotation links in a drawing (this lets you know what object an annotation is linked to) and how to zoom on the linked object.

Open the query_link.CATDrawing document.

1. Right-click on the text "Front view Scale: 1:1" and select **Query Object Links** in the contextual menu.



The Query Link Panel appears:

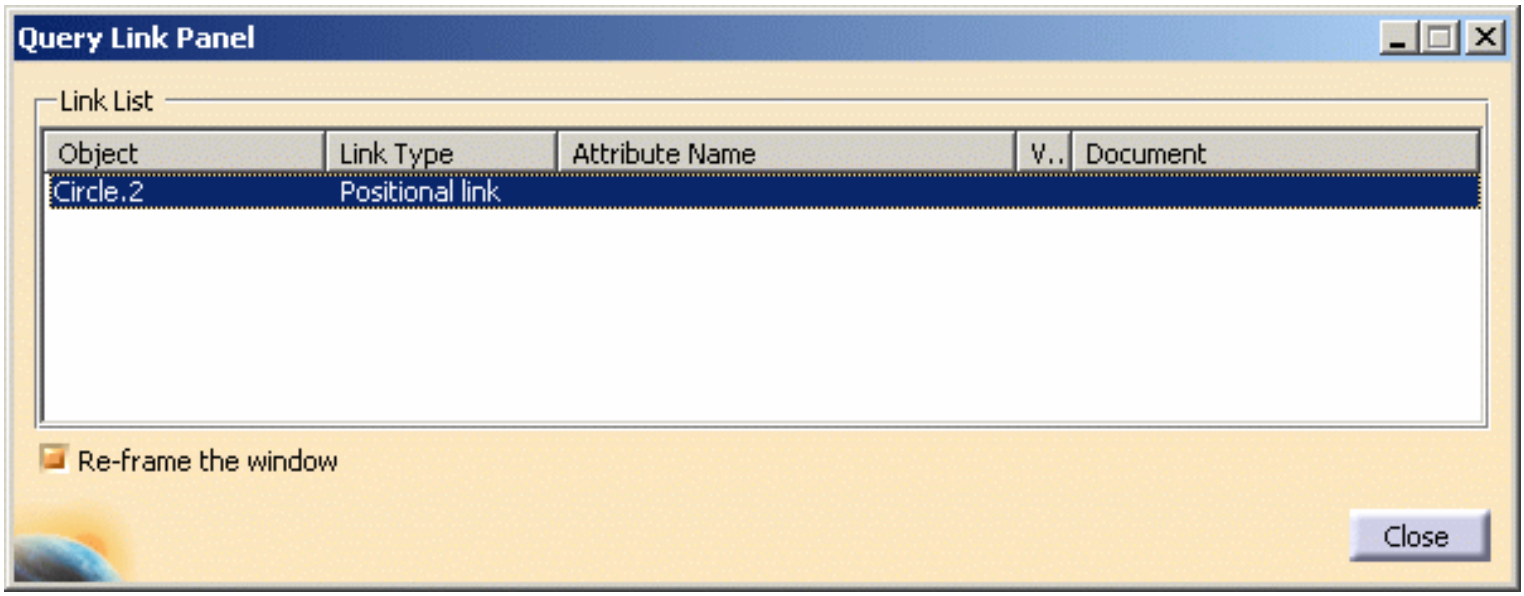


It displays the linked objects name and specifications. In our example, the view name and scale are linked to the front view.

2. Click the **Close** button to close the Query Link Panel.

Zoom on the linked object

1. Right-click on the text "Note 1: Circle..." and select **Query Object Links** in the contextual menu.
2. In the Query Link Panel, check **Re-frame the window** and select the linked object you want to zoom (in this specific case, you can only select Circle.2).



The application zooms on the object to which the text is linked, i.e. the circle.

3. Click the **Close** button to close the Query Link Panel.

Adding Attribute Links to Text

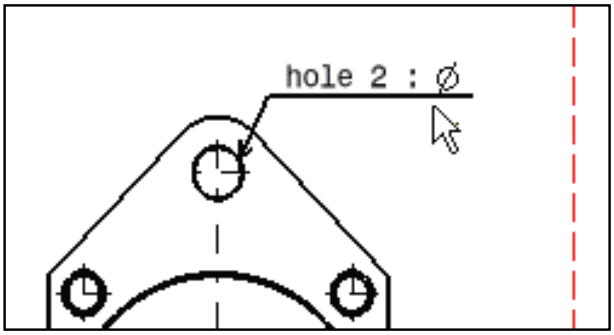
This task shows you how to add one or more attribute links to text. You will learn how to:

- create an attribute link between a hole on the 3D part and the corresponding text in a drawing view
- assign an attribute link to a view

Creating an attribute link between a hole on the 3D part and the corresponding text in a drawing view

Open the GenDrafting_part.CATPart document and the GenDrafting_part_02.CATDrawing document.

1. On the active view, double-click the text to which you want to add a link.

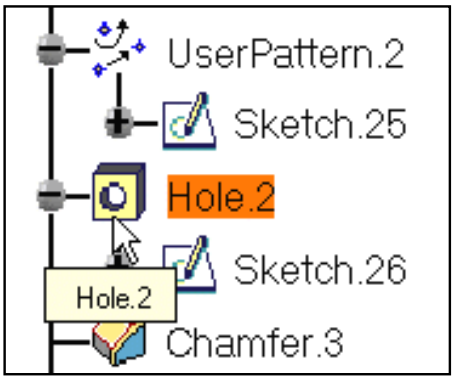


An empty text is created in the drawing. The **Text Editor** dialog box is also displayed. Do not pay attention to this dialog box yet.

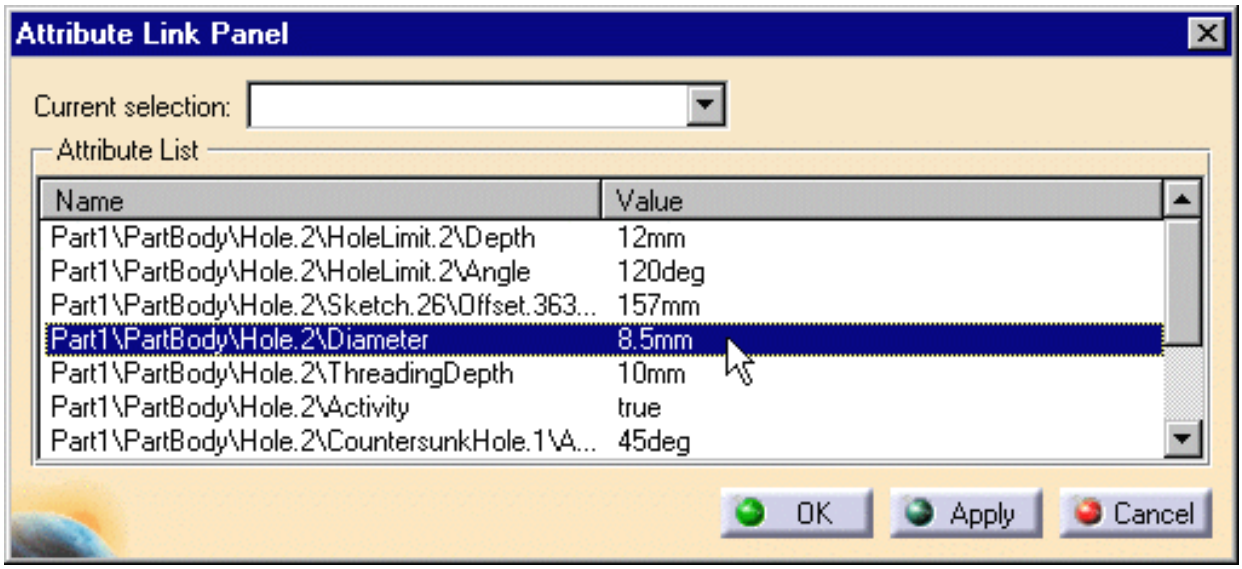
2. In the drawing, right-click the text and select **Attribute Link**.

3. Select the object to which you want the text to be linked, from the specification tree (either from the 3D or from the CATDrawing document).

For example, select **Hole 2** from the CATPart specification tree.

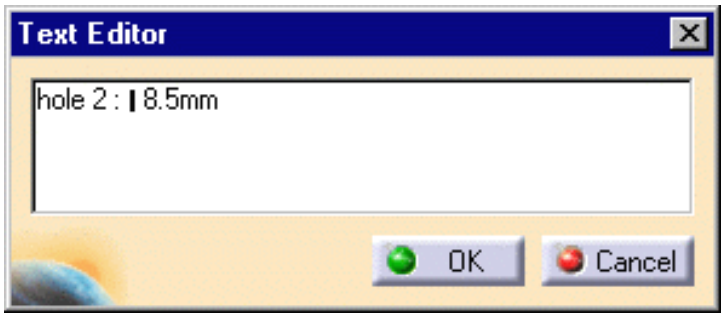


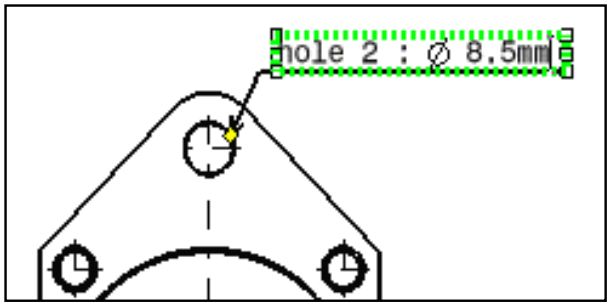
The **Attribute Link Panel** dialog box is displayed in the Drafting window:



4. Select the "**Part1\PartBody\Hole.2\Diameter 8.5mm**" attribute from the list.

   The **8.5mm** value automatically appears both in the Text Editor dialog box and on the CATDrawing.

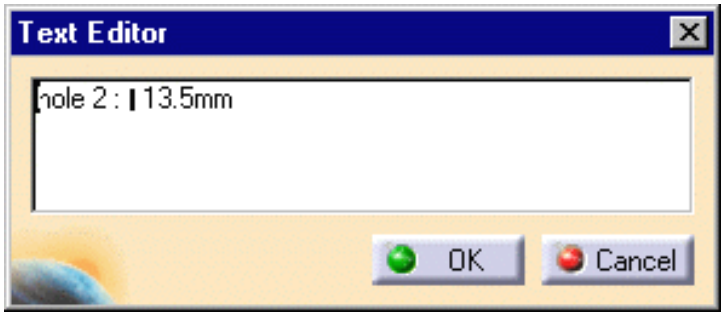


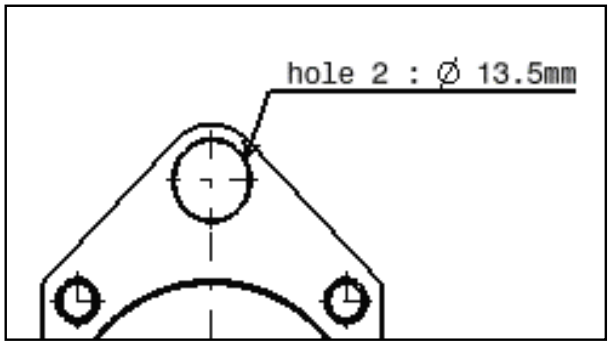


5. Click **OK** to validate and exit the dialog box.

6. Modify the diameter value of **Hole 2** on the CATPart. For example, change it to **13.5mm**.

   The views generated on the CATDrawing and the text attribute value are updated to take this modification into account.





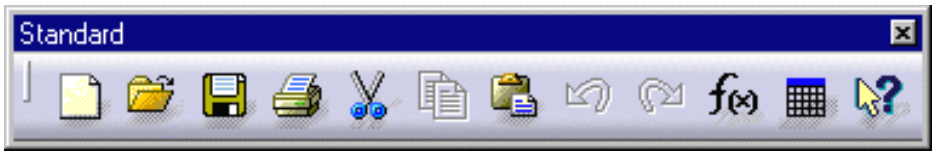
- If the modification is not taken into account, go to **Tools -> Options -> Infrastructure -> Part Infrastructure**, **General** tab and select the **Automatic** update mode.
- At this step, you can perform a query on the link(s) you just created. To do so, right-click the view and select **Query Objects Links**. The **Query Link Panel** dialog box appears, displaying a list of the existing links.
- You can only modify the part of the text that is not of the text attribute type.
  Modifying the text attribute requires isolating it first. To do so, right-click the text attribute and select **Isolate Text**.
- You cannot replace text in attribute links (even though attribute links are displayed and the **Replace** button is available in the Links dialog box, which is available through **Edit -> Links**).

Assigning an attribute link to a view

The GenDrafting_part_02.CATDrawing document is still open from the previous task.

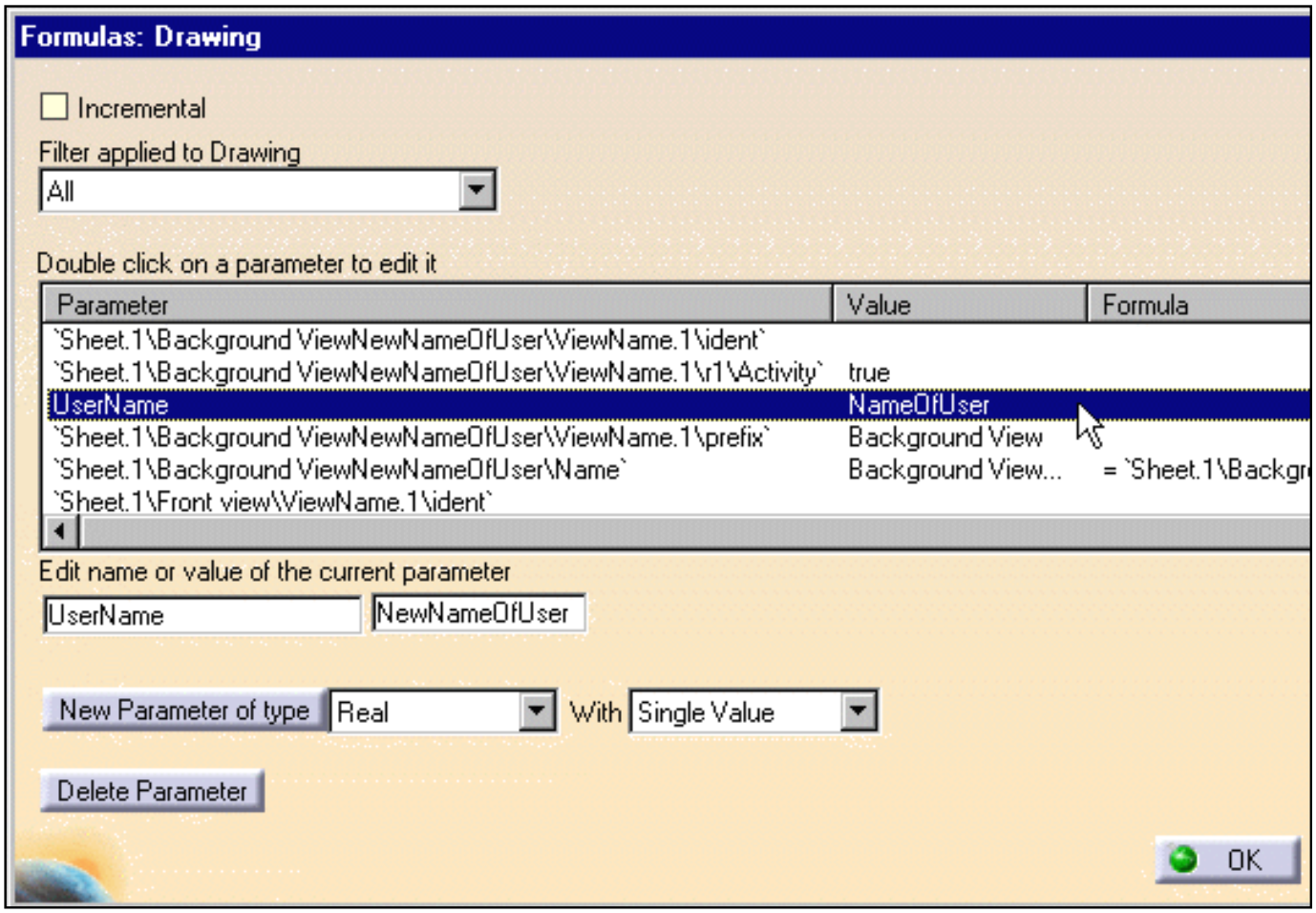
Creating a parameter

1. Click the **Formula** icon from the **Standard** toolbar.



The **Formulas: Drawing** dialog box is displayed.

2. In the drop-down lists next to the **New Parameter of type** button, choose the **String** with **Single Value** type.
3. Click the **New Parameter of type** button.
4. In the **Edit name or value of the current parameter** fields, replace String.1 by UserName in the first field, and enter the value, i.e. NameOfUser in the second field.



5. Click **Apply** and then click **OK**. The parameter is created, and it is displayed in the Drafting specification tree, under the Parameters node.

If the Parameters node in no displayed in the Drafting specification tree, go to **Tools -> Options -> General -> Parameters and Measure**, **Knowledge** tab, and check the **With value** and **With formula** options.

Defining the Text Attribute

6. Click the **Text** icon from the **Annotations** toolbar and click in the free space (under the Front View - Scale:1 text, for example).

An empty text is created in the drawing (in the active view). Also, the **Text Editor** dialog box is displayed. Do not pay attention to this dialog box yet.

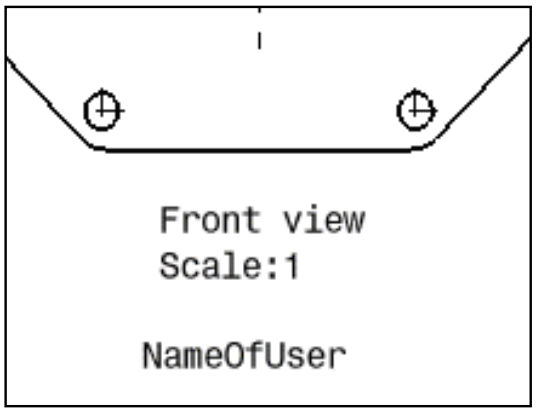
7. In the drawing, right-click on the empty text and select the **Attribute Link** option from the contextual menu.

8. From the specification tree, select the object which you want the text to be linked to. For example, select the CATDrawing document (GenDrafting_part_02 item at the very top of the specification tree). The **Attribute Link Panel** dialog box is displayed.

9. Scroll down the list and select **UserName** in the **Attribute List**.

10. Click **OK**. The value assigned to UserName, i.e. NameOfUser, is displayed in the **Text Editor** dialog box as well as on the drawing view.

11. Click **OK** to close the **Text Editor** dialog box.

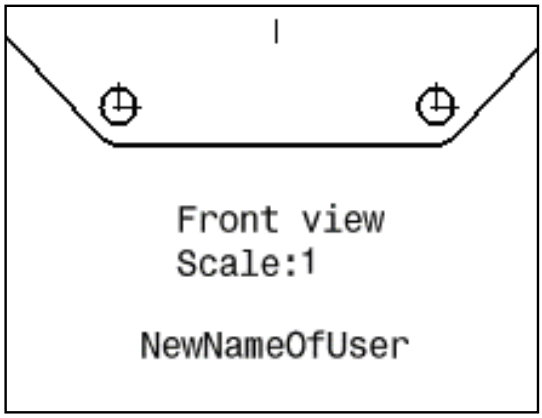


Modifying the Text Attribute

12. Double-click the **UserName** parameter in the specification tree. The **Edit Parameter** dialog box is displayed.

13. Change the parameter name (NameOfUser) to NewNameOfUser.

14. Click **OK.** The parameter name is automatically updated on the view.



Dress-up

The 2D Layout for 3D Design workbench enables you to create:

- 2D dress-up
- associative 3D dress-up
- associative hybrid dress-up between 2D and 3D elements

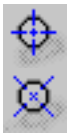
As dress-up commands work as in the Interactive Drafting workbench, most tasks included in this section provide links to the *Interactive Drafting User's Guide*. As such, the information detailed in these tasks is presented in an Interactive Drafting context. You should note that the *Interactive Drafting User's Guide* contains images that correspond to the Interactive Drafting workbench and therefore illustrate dress-up in an environment that is different from the 2D Layout for 3D Design environment (symbols and background color, for example).

Refer to Creating dress-up in a 2D Layout for 3D Design Context which explains what is specific to creating dress-up in the 2D Layout for 3D Design workbench as opposed to doing so in the Interactive Drafting workbench.

Creating dress-up in a 2D Layout for 3D Design context: explains what is specific to creating dress-up in the 2D Layout for 3D Design workbench as opposed to doing so in the Interactive Drafting workbench.

Create center lines (no reference): Apply a center line to one or more circles.

Create center lines (reference): Apply a center line to one or more circles with respect to a reference (linear or circular).


Modify center lines: Modify one or more center lines at one or more ends of this/these center line(s).

Create threads (no reference): Create a thread without a reference.

Create threads (reference): Create a thread with a reference, either circular (circle or point) or linear (line).

Create axis lines: Create an axis line by selecting lines.

Create axis lines and center lines: Create an axis line by selecting lines.

Create an area fill: apply graphical dress-up elements called patterns (these can be hatching, dotting or coloring) on a closed area.

Create arrows: Create an arrow.

Creating Dress-up in a 2D Layout for 3D Design Context

The 2D Layout for 3D Design workbench enables you to create:

- 2D dress-up
- associative 3D dress-up
- associative hybrid dress-up between 2D and 3D elements

As dress-up commands work as in the Interactive Drafting workbench, the tasks included in the Dress-up chapter provide links to the *Interactive Drafting User's Guide*. However, there are a few particularities about creating dress-up in 2D Layout for 3D Design, as opposed to doing so in Drafting, which you will learn in this section.

In this section, you will learn about:

- Selecting elements to annotate
- Dress-up behavior in 2D Layout for 3D Design
  - Available commands
  - General behavior
- Before you begin creating dress-up in 2D Layout for 3D Design

Selecting elements to annotate

Dress-up commands provide a visual feedback indicating whether it is possible to select a given geometrical element. For example, if you draw a circle in the front view, this circle is visualized in the 2D background of the left view as a line. If you activate the center line command, you will not be able to select the line in the left view.

However, you should be aware of the following rule: in a given part layout, it is impossible to add dress-up to another part. You can only create dress-up within a single part layout. For example, in Part.1, it is not possible to add dress-up to an element of Part.2.

When selecting elements to add dress-up to, remember the following points:

- Dress-up can be created in any view, even a non-active one.
- After starting a dress-up command, the view in which you select the first element is the view of creation (that is the view where the dress-up element will be created).
- You can always select an element belonging to the view content.
- Once you have selected the first element, you can only select the other elements in the view of creation.

- You cannot select as the first element a 2D background element.
- You can select an element which belongs to the 3D background of a part layout only if this element belongs to the current layout.
- When multi-selecting elements, all elements must belong to the same view content.
- If you select elements to which dress-up cannot be added before launching a dress-up command, then you will need to reselect elements afterwards.

Dress-up behavior in 2D Layout for 3D Design

Available commands

You can create the following dress-up elements: center line (with or without reference), axis line, thread (with or without reference), axis line and center line, area fill, arrow.

General behavior

You can create dress-up elements:

- in the main view, in the background view, or in a 2D component view (on a layout detail sheet).
- in any visible design view (projection view, auxiliary view, section view/cut) or isometric view of the current sheet, whether or not it is the active view.

To do so, you need to select (and not just point to) 2D elements in the view they belong to. You cannot select elements in the 2D or 3D background.

When creating or modifying dress-up elements, remember the following points:

- You cannot create dress-up elements in a locked view.
- Dress-up elements are re-computed when 2D geometry is modified so as to follow it.
- To modify the overrun of an axis line, a center line or a thread, the layout and sheet must be active, because manipulators are available only in the view where the dress-up element has been created.

Before you begin creating dress-up in 2D Layout for 3D Design

Before you begin creating dress-up in 2D Layout for 3D Design, make sure you are familiar with:

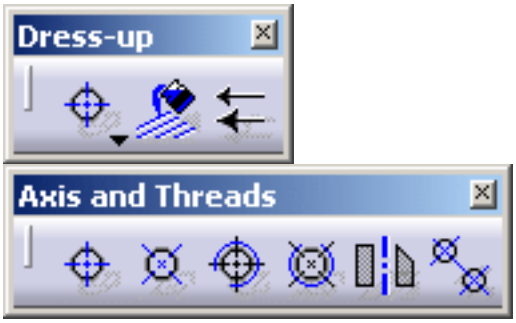
- The Tools toolbar and the Tools Palette.
- SmartPick, an easy-to-use tool designed to assist you when creating dress-up. For more information, refer to the SmartPick task in the *Sketcher User's Guide*.
- Multi-selection. For more information, refer to the Selecting Objects chapter in the *Infrastructure User's Guide*.

Creating Center Lines (No Reference)

This task will show you how to apply a pair of center lines to a circle or an ellipse.

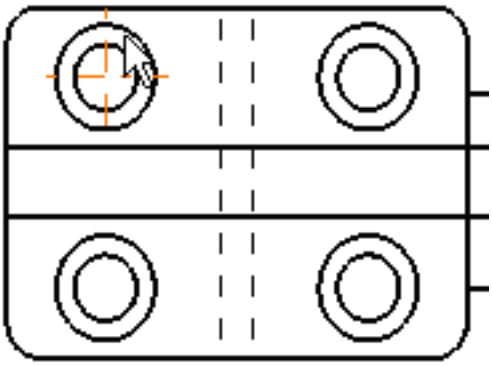
Open the Brackets_views06.CATDrawing document.

1. Click the Center Line icon from the Dressup toolbar (Axis and Threads sub-toolbar).



2. Select a circle.

Center lines are automatically applied to the circle



.

3. Click in the drawing to confirm the creation and select the center lines.
4. Use manipulators to modify center lines size.

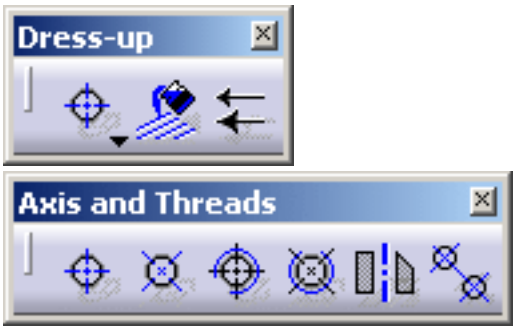
- You can apply this scenario to an ellipse.
- When creating a center line on a generative view, a message will be displayed if the center line cannot be associative to the 3D.

Creating Center Lines (Reference)

This task will show you how to apply a pair of center lines to a circle or an ellipse with respect to a reference (linear or circular).

Open the Brackets_views02.CATDrawing document.

1. Click the Center Line with Reference icon from the Dressup toolbar (Axis and Threads sub-toolbar).

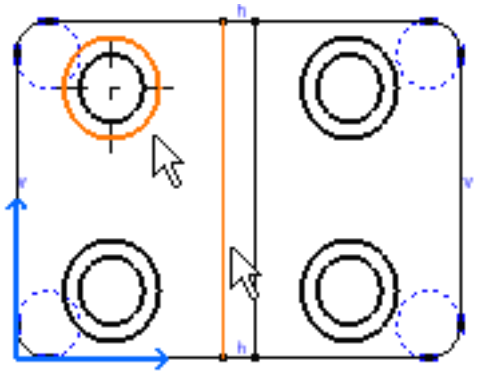


You can multi-select circles before you enter the command to create center lines for all selected circles.

2. Select the circle to be applied a pair of center lines.

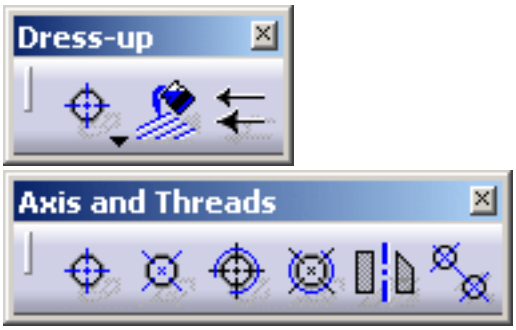
3. Select the reference line.

The center line created is associative with the reference line.



You can create a pair of center lines according to a circular reference (a point or a circle):

4. Click the Center Line with Reference icon from the Dressup toolbar (Axis and Threads sub-toolbar).

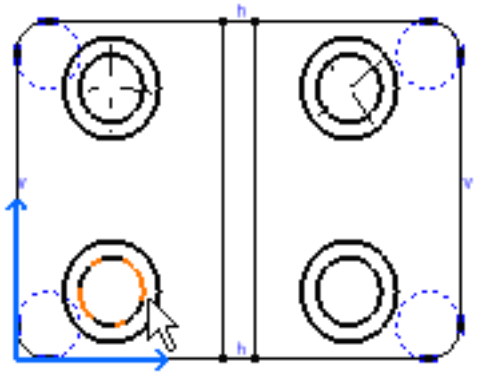


You can multi-select circles before you enter the command and thereby apply center lines to the selected circles.

5. Select the circle to be applied a pair of center lines.

6. Select the reference circle.

The pair of center lines created is associative with the reference circle type element.



You cannot apply this scenario to an ellipse.

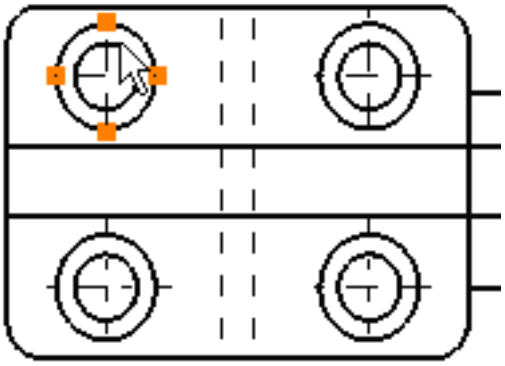
When creating a center line on a generative view, a message will be displayed if the center line cannot be associative to the 3D. In this case, the center line is neither linked to the 3D nor to 2D drawing elements. For example, a non-associative center line with a reference line will not be updated when the reference line is moved.

Modifying Center Lines or Axis Lines

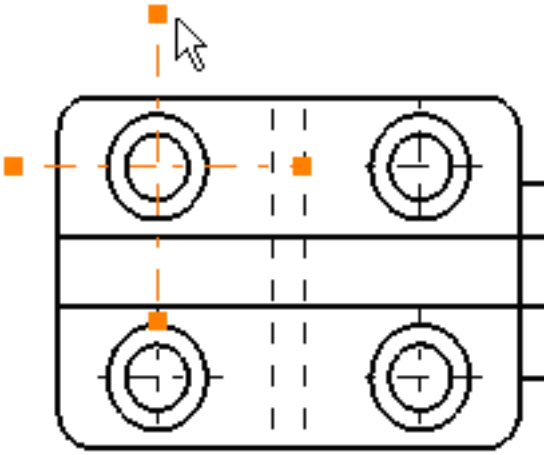
This task will show you how to modify a pair of center lines at one or more end(s) of this/these center lines. You can use the same method to modify axis lines.

Open the Brackets_views08.CATDrawing document.

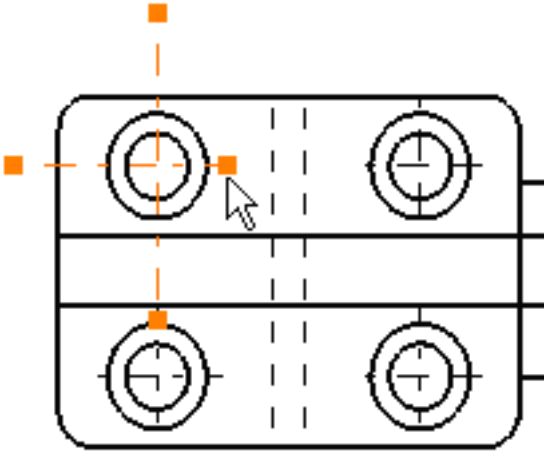
1. Click a center line. End points appear.



2. Select any end point and drag to move all the center line extremities to a new position.



3. Press the Ctrl key while selecting any end point and drag the selected extremity to a new position.



- You can multi-select elements when modifying center lines.
- You can also modify the center line through **Edit** > **Properties** > **Graphic** tab.

Creating Threads (No Reference)

This task will show you how to create a thread without a reference. In this particular case, you will apply a thread to a hole.

Open the Brackets_views06.CATDrawing document.

1. Click the **Thread** icon from the **Dress-up** toolbar (**Axis and Threads** sub-toolbar).

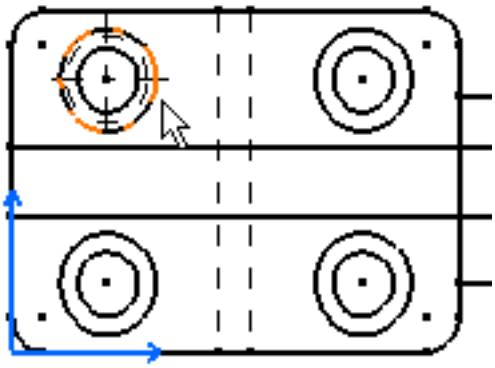
You can also multi-select holes before clicking the Thread icon.

Activating this command displays two options in the Tools Palette which is automatically displayed:

- The **Tap** type option , which is activated by default.
- The **Thread** type option .

2. Select the **Thread** type option .
3. Select the hole (or circle) to which you want to apply the thread.

The thread is created.



4. Select an axis line manipulator and drag it along a direction. Thread axis lines are modified symmetrically.

If you want to move only one axis line, hold on the Ctrl key while you are dragging the manipulator.

If you delete the thread axis line, the external circle is also deleted and vice versa.

- The thread that appears on the hole is assigned a standard radius and representation (compliant with the selected standard).
- When creating a thread on a generative view, a message will be displayed if the thread cannot be associative to the 3D.

Creating Threads (Reference)

This task shows you how to create a thread with a reference, either circular (circle or point) or linear (line). In this particular case, you will apply a thread to a hole with a line as reference.

Open the Brackets_views06.CATDrawing document.

1. Click the **Thread with Reference** icon from the **Dress-up** toolbar (**Axis and Threads** sub-toolbar).

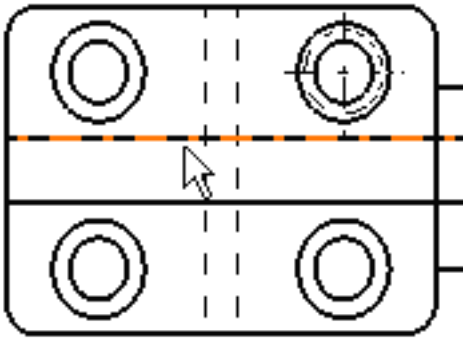
You can also multi-select holes before clicking the Thread icon.

Activating this command displays two options in the Tools Palette which is automatically displayed:

- The **Reference Tap** type option , which is activated by default.
- The **Reference Thread** type option .

2. Select the **Reference Thread** type option .
3. Select the hole (or circle) to which you want to apply the thread.
4. Select a reference line.

The thread is created according to this reference.



5. Select a manipulator and drag it along a direction. Thread axis lines are modified symmetrically.

If you want to move only one axis line, hold on the Ctrl key while you are dragging the manipulator.

When creating a thread on a generative view, a message will be displayed if the center line cannot be associative to the 3D. In this case, the thread is neither linked to the 3D nor to 2D drawing elements. For example, a non-associative thread with a reference line will not be updated when the reference line is moved.

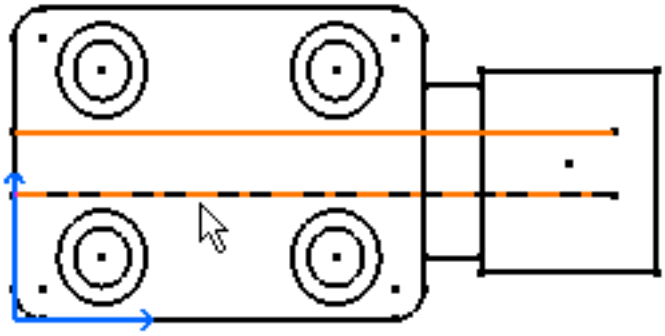
Creating Axis Lines

This task will show you how to create an axis line.

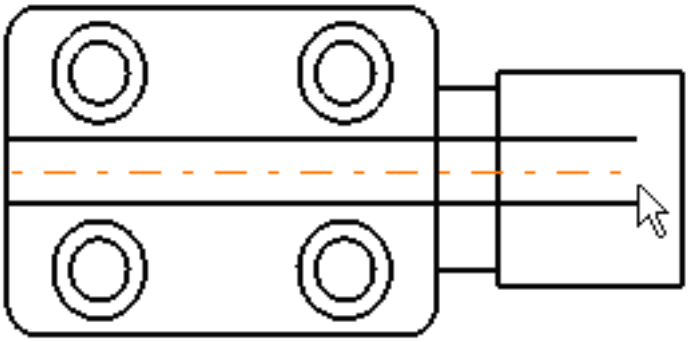
Open the Brackets_views07.CATDrawing document.

1. Click the drawing window, and click the **Axis Line** icon from the **Dress-up** toolbar (**Axis and Threads** sub-toolbar).

2. Select two lines.



The axis line is created.



- If needed, you can select two non-parallel lines that are not co-linear.
- Both in the case of center lines and axis lines, a default overrun is created.
- When creating an axis line in a Generative Drafting context, a message will be displayed if the axis line cannot be associative to the 3D.
- In a Generative Drafting context, you can create axis lines between symbolic fillet edges or fillet representation. Note that these axis lines will not be associative (a message will be displayed).
- In a Generative Drafting context, depending on the type of element selected, the axis line is sometimes created directly after you select a single element. If you are not satisfied with the axis line thus created, you can force the selection of a second element by pressing the Ctrl key prior to making your selection: you will then be able to select the first and then the second element.

If you need to modify an axis line, refer to Modifying Center Lines or Axis Lines. Note that you can multi-select elements when modifying axis lines.

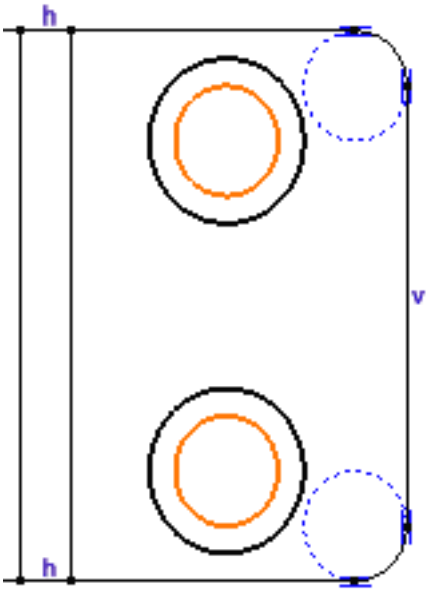
Creating Axis Lines and Center Lines

This task will show you how to create simultaneously axis and center lines on several circles.

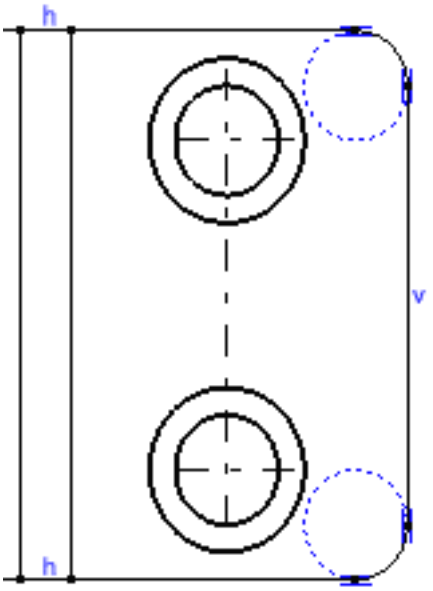
Open the Brackets_views02.CATDrawing document.

1. Click the drawing window, and click the **Axis Line and Center Line** icon from the **Dress-up** toolbar (**Axis and Threads** sub-toolbar).

2. Select two circles.



The axes and center lines are created.



3. Select an axis line manipulator and drag it along a direction. You can notice that thread axis lines are modified symmetrically.

If you want to move only one axis line, hold on the Ctrl key while you are dragging the manipulator.

When creating axes and center lines in a Generative Drafting context, a message will be displayed if axes and center line cannot be associative to the 3D.

Creating an Area Fill

An area fill is a closed area on which you then apply graphical dress-up element called patterns (these can be hatching, dotting or coloring). You can create area fills on the following elements:

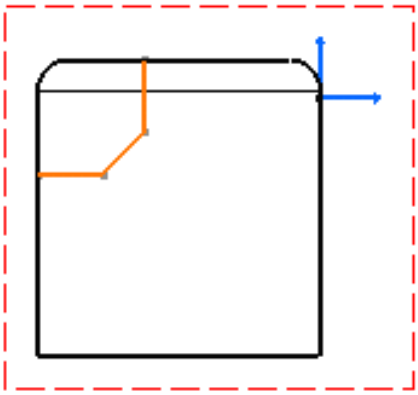
- sketched elements,
- generative elements
- part-sketched, part-generative elements

In this task, you will learn how to create an area fill on a drawing containing a mix of sketched and generative elements.

Open the GenDrafting_Area_Fill.CATDrawing document. This drawing is a generative one.

Define your area fill profile by creating lines so that your drawing looks like the figure shown here. In this example, sketched elements (the ones you create) are selected (they are shown in red), and generative elements are shown in black.

The area fill profile will therefore consist of both sketched and generative elements.

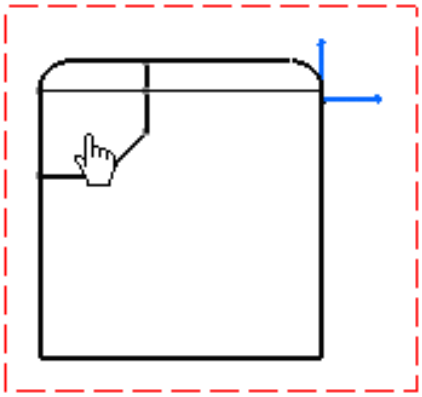


You do not need to activate the view in which you are going to create an area fill.

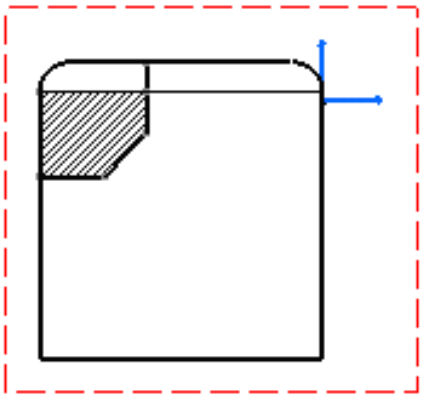
1. Click the **Area Fill** icon in the Dress-Up toolbar.

The **Area Detection** dialog box appears.

2. Click the **Automatic** option (the other option is described in the remarks section below) and then click inside the area for which you just defined the profile, under the line which represents the fillet edge.



The software automatically detects the area to fill based on where you clicked and fills this area with the selected pattern. The Areas to Fill dialog box disappears.



3. Optionally, you can select another pattern for your area fill. To do so:

   a. make sure the area fill is selected and click the down arrow besides the **Pattern** icon in the **Graphic Properties** toolbar.

   b. In the Pattern dialog box, select a pattern for your area fill and click **OK**.

A few remarks

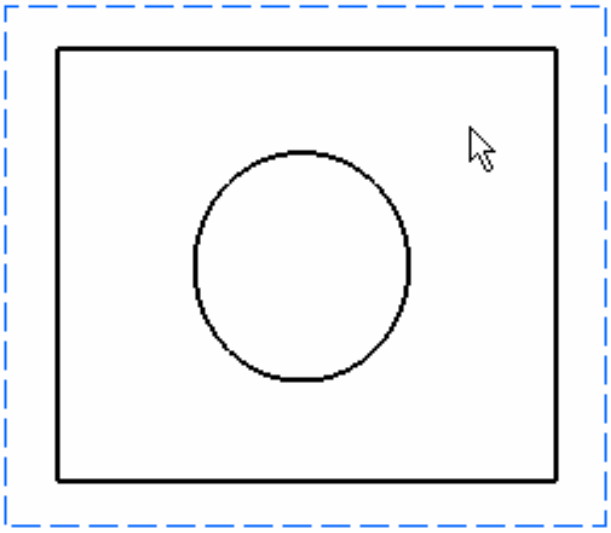
Area to Fill dialog box

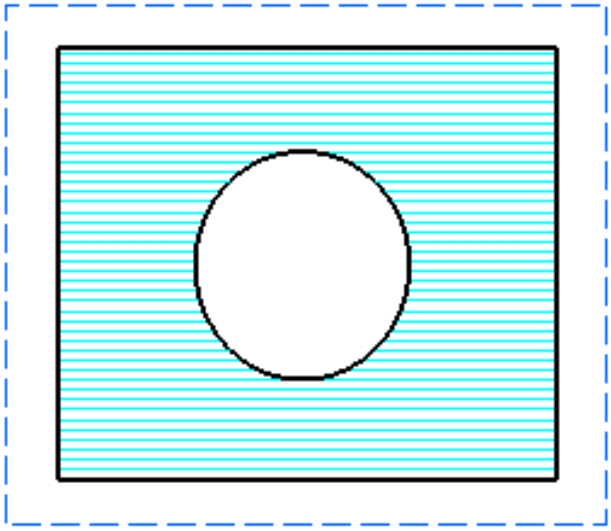
The two options available in the Area to Fill dialog box are described below. You can specify the area you want to fill before or after choosing the option in the Area to Fill dialog box.

For each option, examples illustrate what kind of area fill you will get depending on where you click. Note where the cursor is located on the figures.

- **Automatic** automatically detects the area to fill based on where you click: just click inside the area you want to fill.

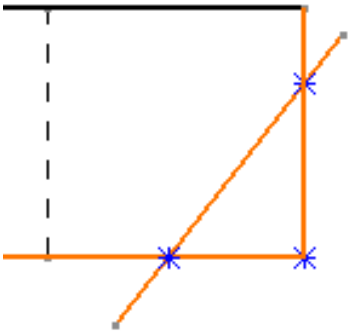
If you click in this area: *You get this area fill:*



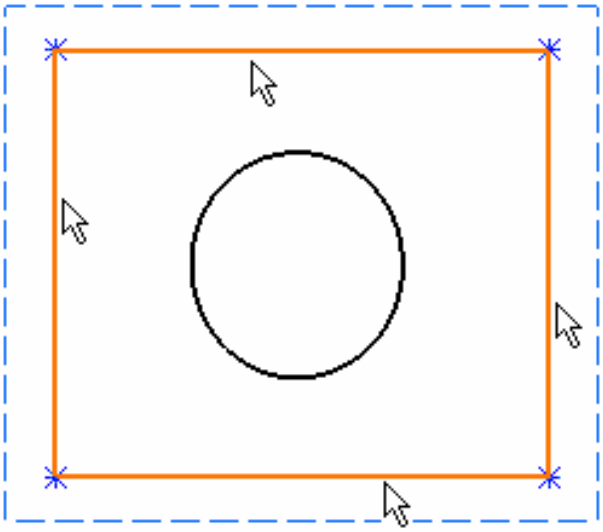


- **With profile selection** lets you specify the area to fill: select **all** the 2D elements that make up the profile of the area you want to fill, and then click inside this area.

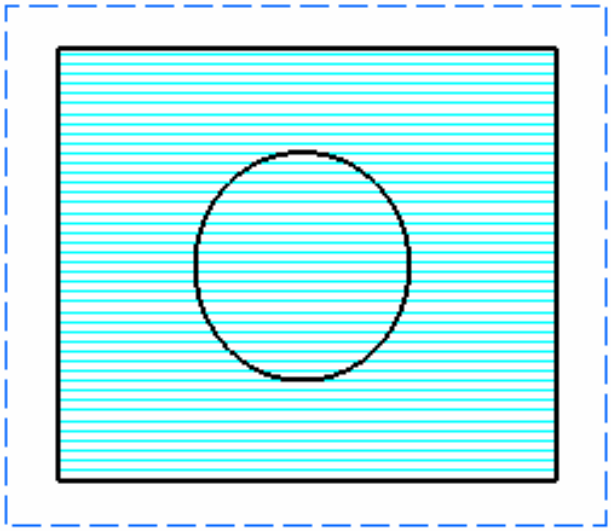
As you select elements on a view, intersection symbols (stars) appear where elements intersect. This enables you to know where the profile is open: in this case, intersection symbols do not appear. As you cannot apply an area fill to an open profile, make sure all elements intersect.

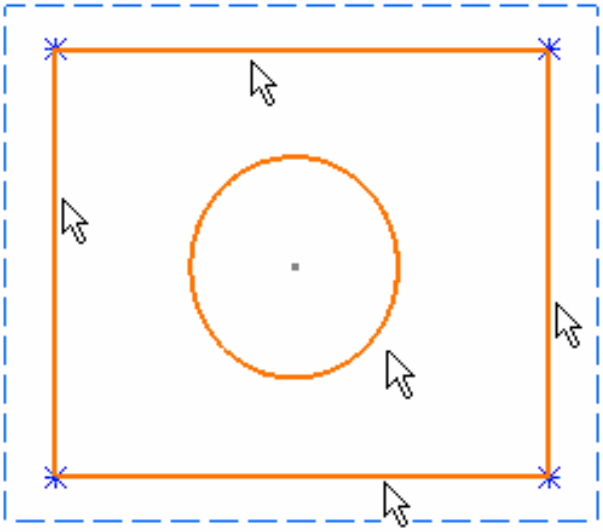


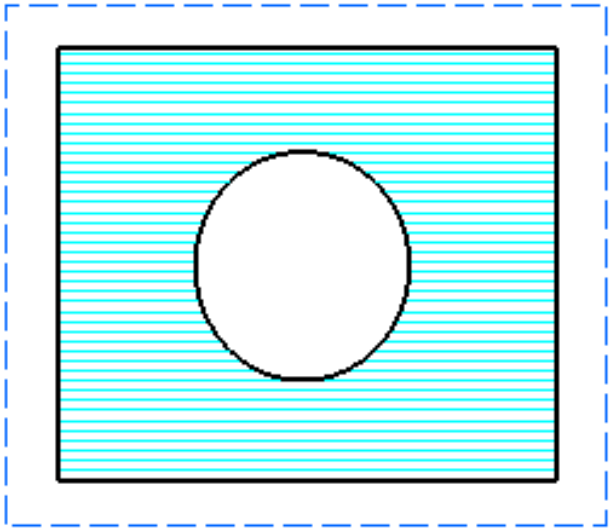
If you select these elements:



You get this area fill:





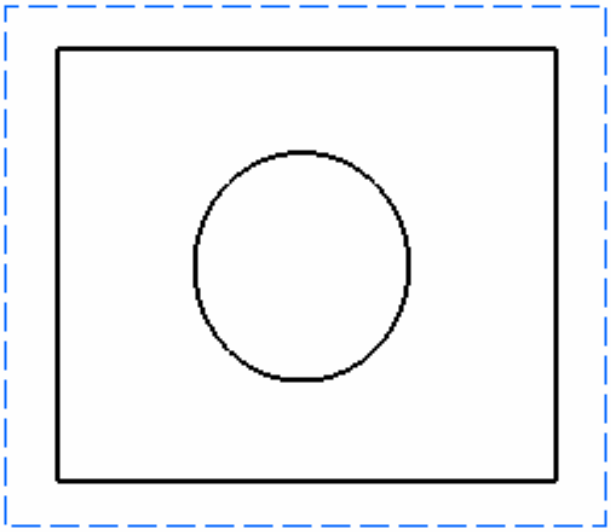


Miscellaneous remarks about applying area fills

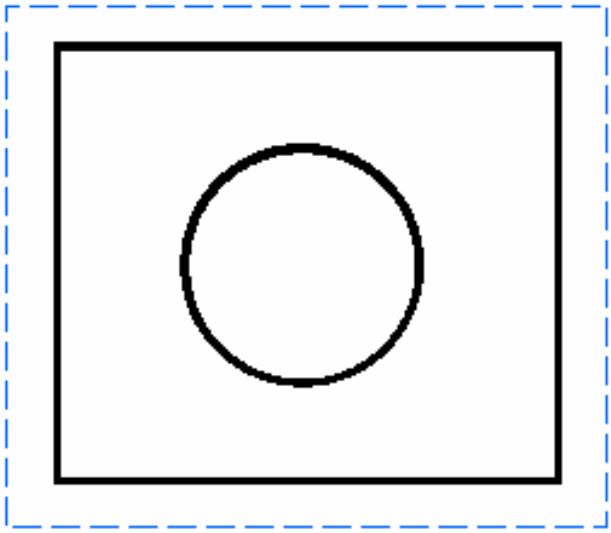
- Whichever option you choose in the Areas to Fill dialog box, make sure the profile you select is closed, i.e. that all elements that make up its profile intersect. An error message will appear if you select a profile which is not closed.
- When you create an area fill on sketched elements, or on part-sketched, part-generative elements, extra sketched elements are added over the generative elements which make up the profile of the area fill. Also, coincidence constraints are created between the original generative elements and the added sketched elements.
- On generative drawings, the area fill is not associative with the 3D part. If you modify the original 3D part and then update the generative drawing, the area fill will not be changed.
- When a view is isolated, any area fill on the view is isolated as well. Consequently, there is no longer any relationship between the area fill and its profile.
- Select elements carefully: the area will be filled according to the elements you selected.
- If you apply modifications to the filled area, the pattern will be modified accordingly.
- In the case of superposed views, the area fill will be created on the active view (provided the active view is one of the superposed views).
- If you create text in a filled area, the background of the text will be blanked as shown here.

- For more information about hatching or dotting patterns, refer to the General remarks about patterns section in Administration Tasks > Setting Standard Parameters and Styles > Setting Standard Parameters > Patterns.

What you have before applying the pattern:



What you get if the pattern cannot be displayed:

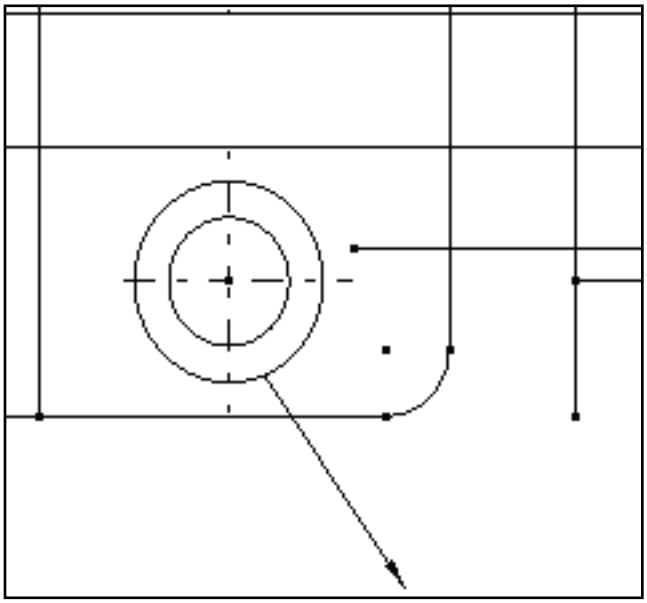


Creating Arrows

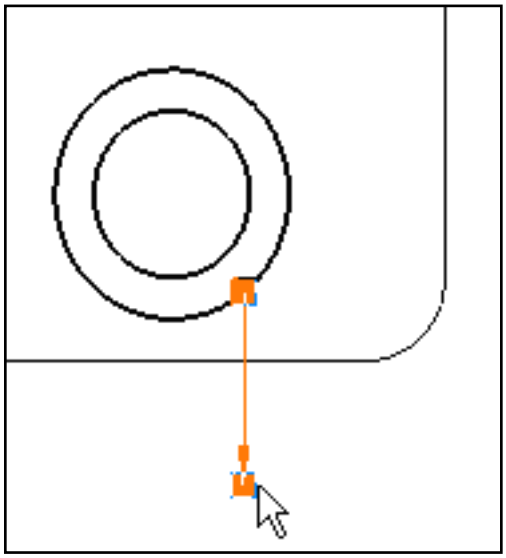
This task will show you how to create an arrow. For the purpose of this exercise, you will use an arrow to illustrate the kind of hole you want to apply to a circle.

Open the Brackets_views06.CATDrawing document.

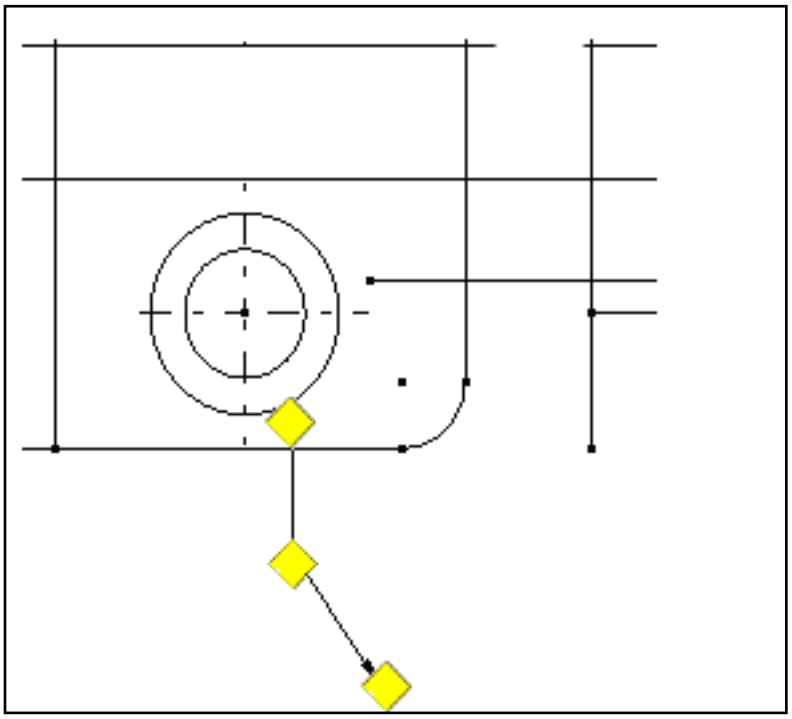
1. Select **Insert -> Dress Up -> Arrow** from the menu bar.
2. Click a point or select an object to define the arrow starting point (the tail). For example, select a circle.
3. Click another point or select another object to define the arrow extremity (the head). The arrow is created.



- The arrow and the selected object are associative.
- To modify the general appearance of the arrow, either click the arrow and then use the Graphic Properties toolbar, or right-click the arrow and then use the Properties dialog box (select **Properties** and click the **Graphic** tab).
- To modify the position of the arrow, click the arrow and use the manipulators to drag it to its new location.

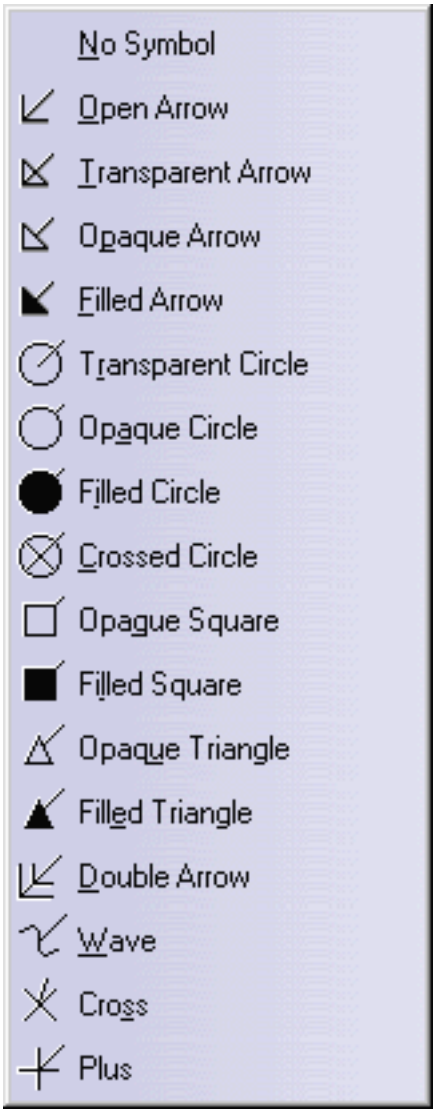


4. You will now add a breakpoint to the arrow. Select the arrow and right-click on a yellow manipulator. A contextual menu appears.
5. Select **Add a Breakpoint**. A breakpoint is added to the arrow; you can drag it to change the arrow path.

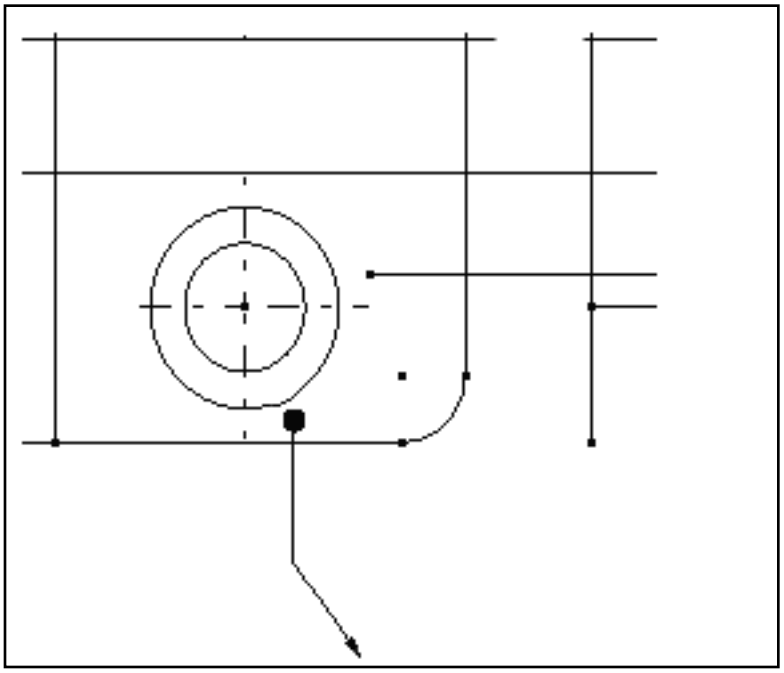


6. You will now choose a symbol for the arrow tail. To do this, right-click on the yellow tail manipulator.

7. In the contextual menu, point to **Symbol Shape** and select a symbol, **Filled Circle** for example.

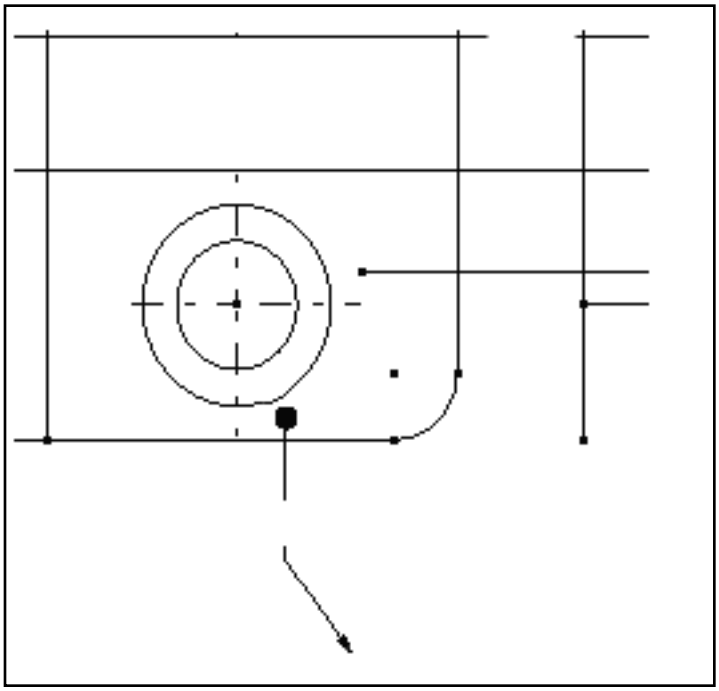


The symbol you choose now appears on the arrow tail. You can also change the symbol used for the arrow head by repeating steps 6 and 7.



8. You will now create an interruption on the arrow tail. Right-click on the yellow tail manipulator again.

9. In the contextual menu, select **Add an Interruption**. An interruption is added to the arrow.



- You cannot add another extremity to an arrow.
- Arrow angle and length are defined by standards. For more information, refer to Dimension Parameters in Administration Tasks > Setting Standard Parameters and Styles > Setting Standard Parameters.

3D Outputs

Create a 3D profile: create a 3D profile on the view support plane, and create a 3D profile on a plane parallel to the view support plane.

Create a 3D plane: create a 3D plane that you can then re-use to create 3D profiles.

Creating a 3D Profile

Within a design view, only part of the geometry is needed for defining 3D shapes: for those elements that do not need to be defined as 3D shapes, a 2D definition is sufficient. 3D profiles enable you to specify the geometry you want to output in 3D. In this task, you will learn how to:

- create a 3D profile on the view support plane
- create a 3D profile on a plane parallel to the view support plane

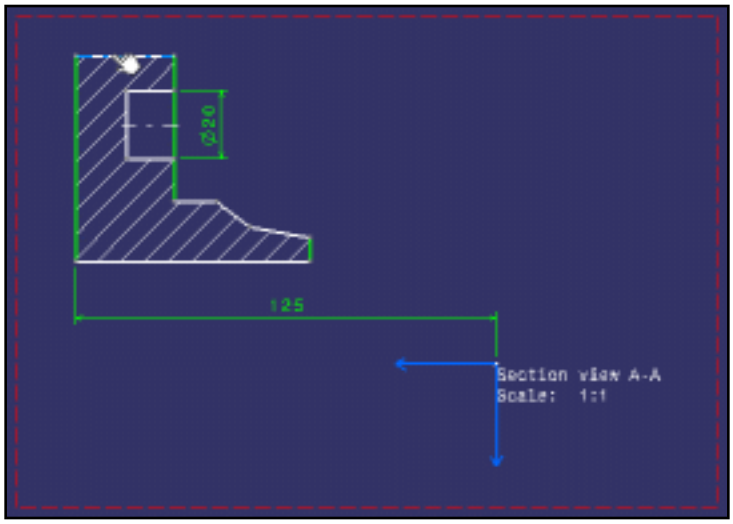
Open the Disk3.CATPart document. Select **Start -> Mechanical Design -> 2D Layout for 3D Design** to open the layout in the 2D window.

Creating a 3D profile on the view support plane

1. Make sure the section view is active. If not, double-click to activate it.

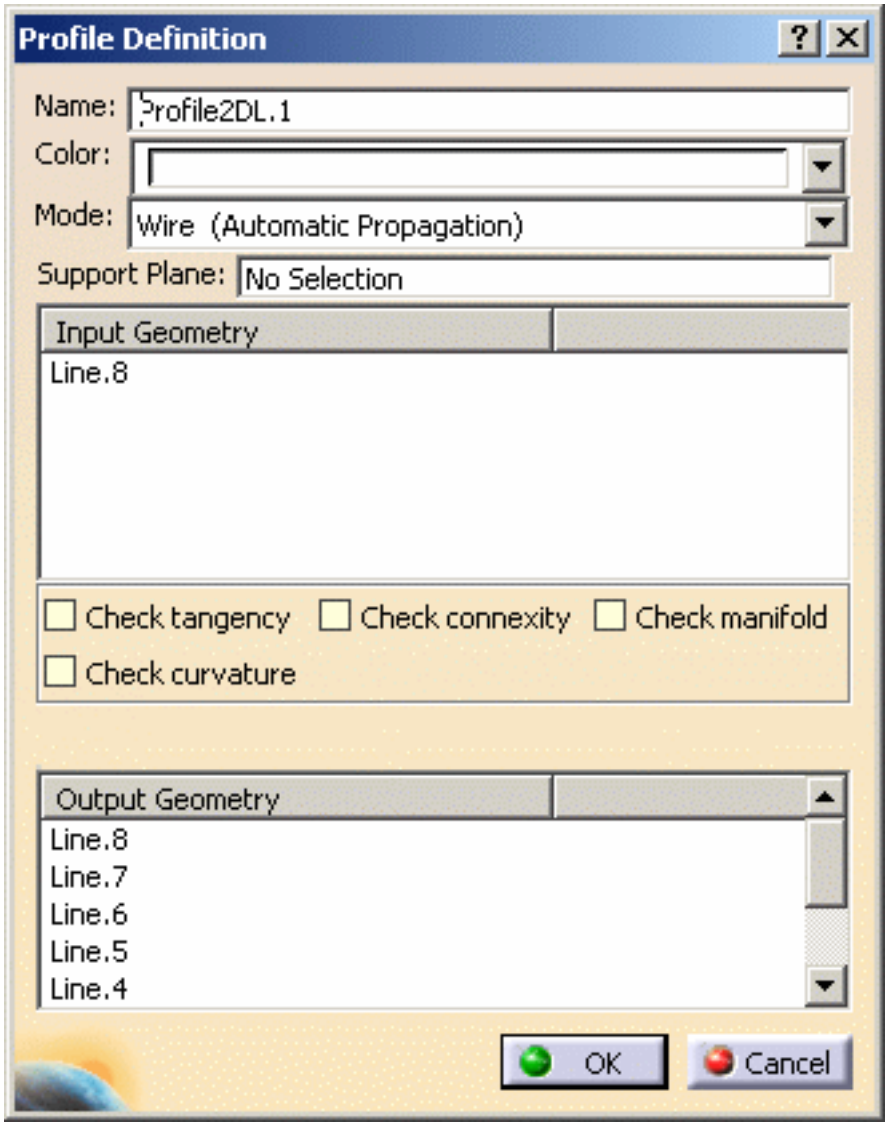
2. Click the **3D Profile** icon in the **3D Geometry** toolbar.

3. Select the line as shown below.



The **Profile Definition** dialog box appears, displaying the name of the 3D profile you are creating in the Name field. The geometry you selected is displayed in the **Input Geometry** list. The resulting geometry (that is all geometrical elements that eventually make up the 3D profile) is displayed in the **Output Geometry** list.

You can select an element from these lists if you want it to be highlighted in the 2D and 3D windows.



4. Enter a name for your 3D profile, Shaft for example.

5. Optionally choose a color for your 3D profile (the color is not applied to the geometry referenced by the profile).

6. Choose a mode from the associated drop-down list.

   - **Point (Explicit Definition)**: you need to select all the points of interest. In that case, the **Input Geometry** and **Output Geometry** fields show the same elements.

   - **Wire (Automatic Propagation)**: after you select a geometrical element, the application detects and selects all connex elements. In that case, the **Input Geometry** and **Output Geometry** fields do not show the same elements.

In certain specific geometrical configurations, an ambiguity may arise, in which case some elements in the profile remain unselected. You can solve the ambiguity by selecting the remaining elements to include in the profile.

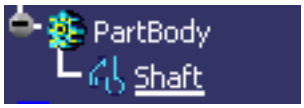
- **Wire (Explicit Definition)**: you need to select all the geometrical elements of interest. In that case, the **Input Geometry** and the **Output Geometry** fields show the same elements.

For the purpose of this scenario, make sure the **Wire (Automatic Propagation)** option is selected from the list.

7. Optionally choose one or several checks to perform. This is to verify that the profile is usable for solid or surface definition.
   - **Check tangency**
   - **Check connexity**
   - **Check manifold**
   - **Check curvature**

   Once checks are performed, warning messages may appear to help you decide whether you can keep your definition as such or if you need to modify it. Note that you can validate the profile definition even if there are some warnings. However, when updating the 3D, you may get an update error (depending on the kind of warning).

8. Click **OK** to validate and close the dialog box. The 3D profile is created, on the same plane as the section view, and it is listed in the specification tree, under the PartBody node.



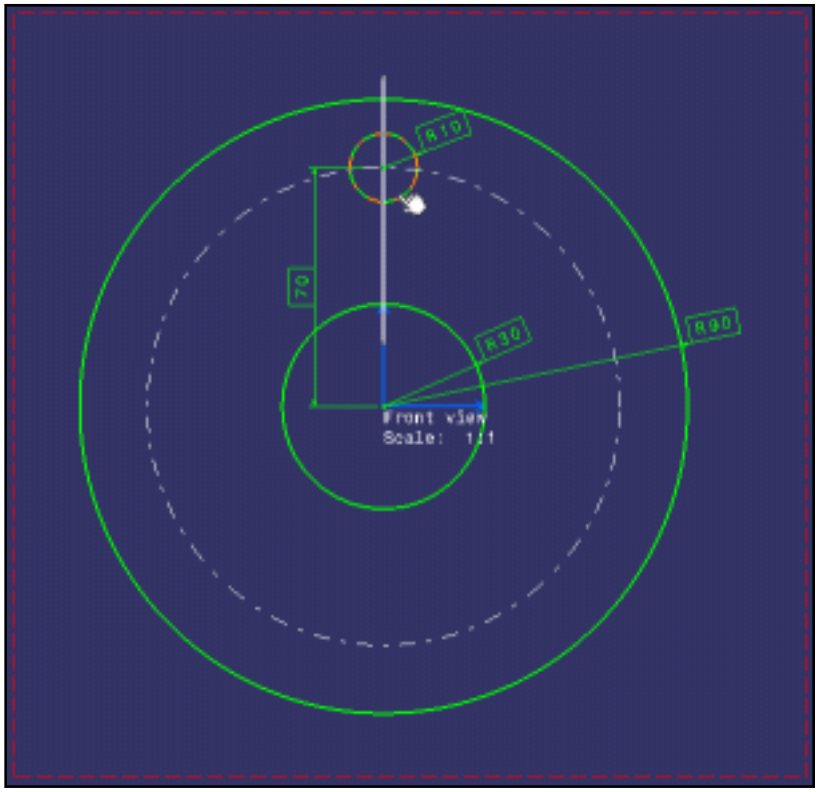
- Of all elements created from 2D geometry in 2D Layout for 3D Design, only 3D profiles and 3D planes belong to the current part body.
- Note that 3D profiles and 3D planes are created under the current part body only when working in a hybrid design environment, that is when the **Enable hybrid design inside part bodies and bodies** option is selected in **Tools -> Options -> Infrastructure -> Part Infrastructure -> Part Document** tab (which is the case by default). Otherwise, when this option is not selected, 3D profiles and 3D planes are created in geometrical sets or ordered geometrical sets.

Creating a 3D profile on a plane parallel to the view support plane

1. Double-click the front view to activate it.

2. Click the **3D Profile** icon in the **3D Geometry** toolbar.

3. Select the R10 circle as shown below.



The **Profile Definition** dialog box is displayed.

4. Choose a support plane. You can either:

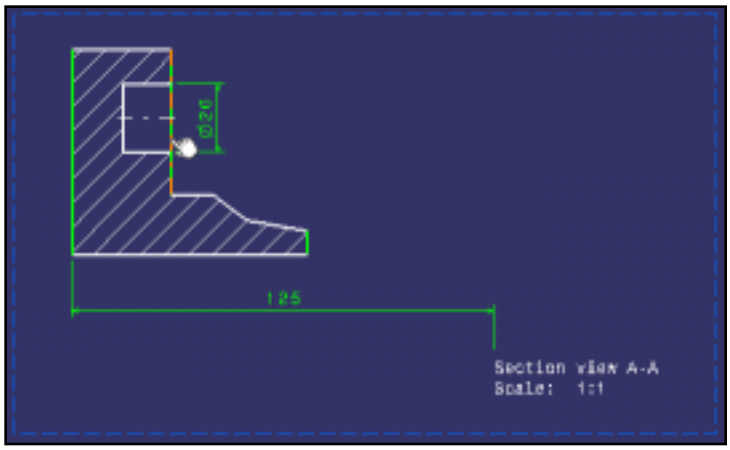
   - select an existing plane, such as the xy, yz or zx plane, the face of a pad, or an existing 3D plane (for more information, refer to Creating a 3D Plane).

   - define a parallel plane on the fly by selecting a line in another layout view (provided the support plane in this view is orthogonal to the support plane you are defining).

For the purpose of our scenario, you will define a plane on the fly. To do this, right-click inside the **Support Plane** field.

5. Select **Create Plane** in the contextual menu which is displayed.

6. Select the line in the section view as shown below.



The 3D plane, Plane2DL.1, is created and it is listed in the specification tree, under the PartBody node.

- You can only create a plane parallel to the support plane of the current view. An error message is displayed when selecting a line that would lead to the creation of a non-parallel plane.
- The 3D plane is associative to the selected line: if the line is modified, the support plane will be recomputed when updating the plane (by exiting the 2D Layout for 3D Design workbench or using the **Update 3D Profile** command) to reflect the modifications.

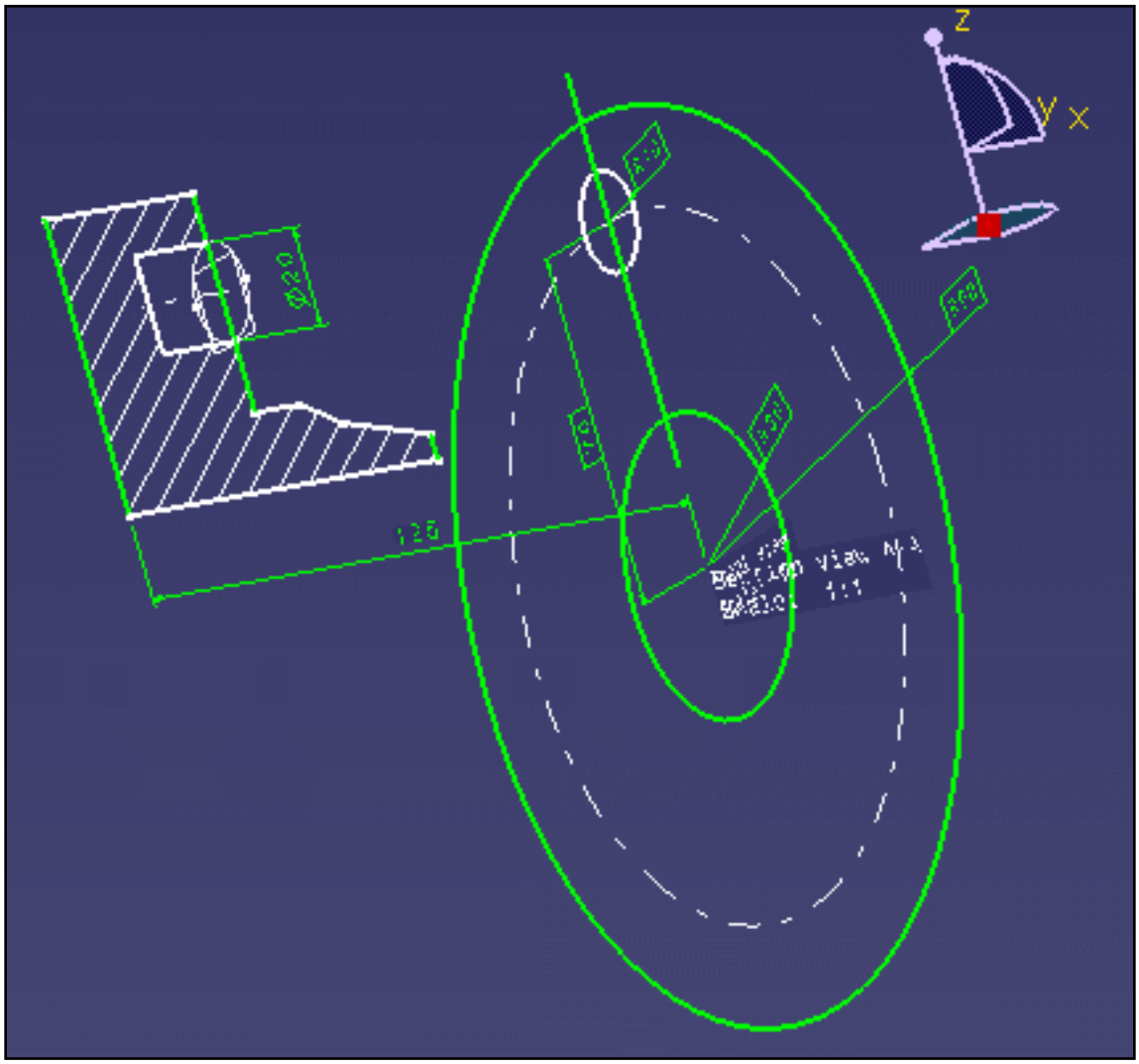
7. In the **Profile Definition** dialog box, enter a name for your 3D profile (Pocket for example).

8. Make sure Plane2DL.1 is selected in the **Support Plane** field.

9. Click **OK** to validate and close the dialog box.

The 3D profile is created, by projecting the circle on the support plane which is parallel to the front view. It is listed in the specification tree under the PartBody node.

Furthermore, the 3D plane and profile are displayed in the 3D window.



More about creating 3D profiles

You can create as many 3D profiles as needed from a design view. 3D profiles can be created on the support plane of the view, as well as on any plane parallel to the view support plane. You can use an existing plane, or define a parallel plane on the fly during the 3D profile creation.

You cannot create 3D profiles for geometry contained in isometric views (because they are not design views).

3D profiles:

- can be created for any 2D geometry contained in a design view, in a part layout (a CATPart document).
- can contain 2D geometry which is already included in other 3D profile or plane (in other words, any 2D geometry can be included in several 3D outputs).
- can be updated independently of the layout. During an update operation, a given 3D profile is only impacted when the 2D geometry is modified.
- have their own graphic properties, independent from the graphic properties of the 2D geometry

which makes up the profile.

- let you expose in a part a set of connected curves or a set of points.
- can be used to create Part Design or Generative Shape Design features.

When creating 3D profiles, remember the following points:

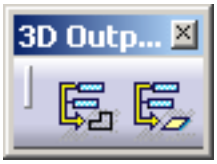
- You can edit a 3D profile by right-clicking it from the specification tree and selecting **Profile definition**. This command is only accessible in the Part Design and Generative Shape Design workbenches.
- Deleting a 3D profile does not delete the original 2D geometry in the layout. A 3D profile can only be deleted from the Part Design and Generative Shape Design workbenches.
- Deleting the 2D geometry used as input when defining a 3D profile in a view (which can only be done from the 2D Layout for 3D Design workbench) prevents the 3D profile from being re-built.
- **Cut**, **Copy** and **Paste** commands are not available for 3D profiles.
- Powercopy is not available for 3D profiles.

Creating a 3D Plane

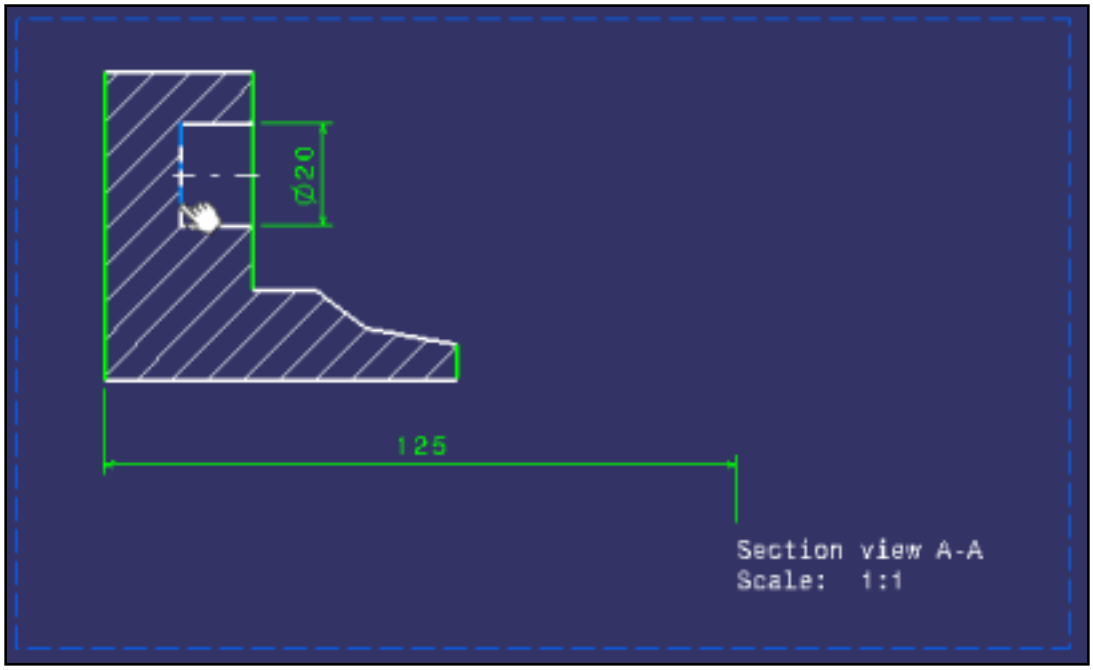
In this task, you will learn how to create a 3D plane.

Open the Disk3.CATPart document. Select **Start -> Mechanical Design -> 2D Layout for 3D Design** to open the layout in the 2D window.

1. Double-click the front view to activate it.
2. Click the **3D Plane** icon in the **3D Geometry** toolbar (**3D Outputs** sub-toolbar).



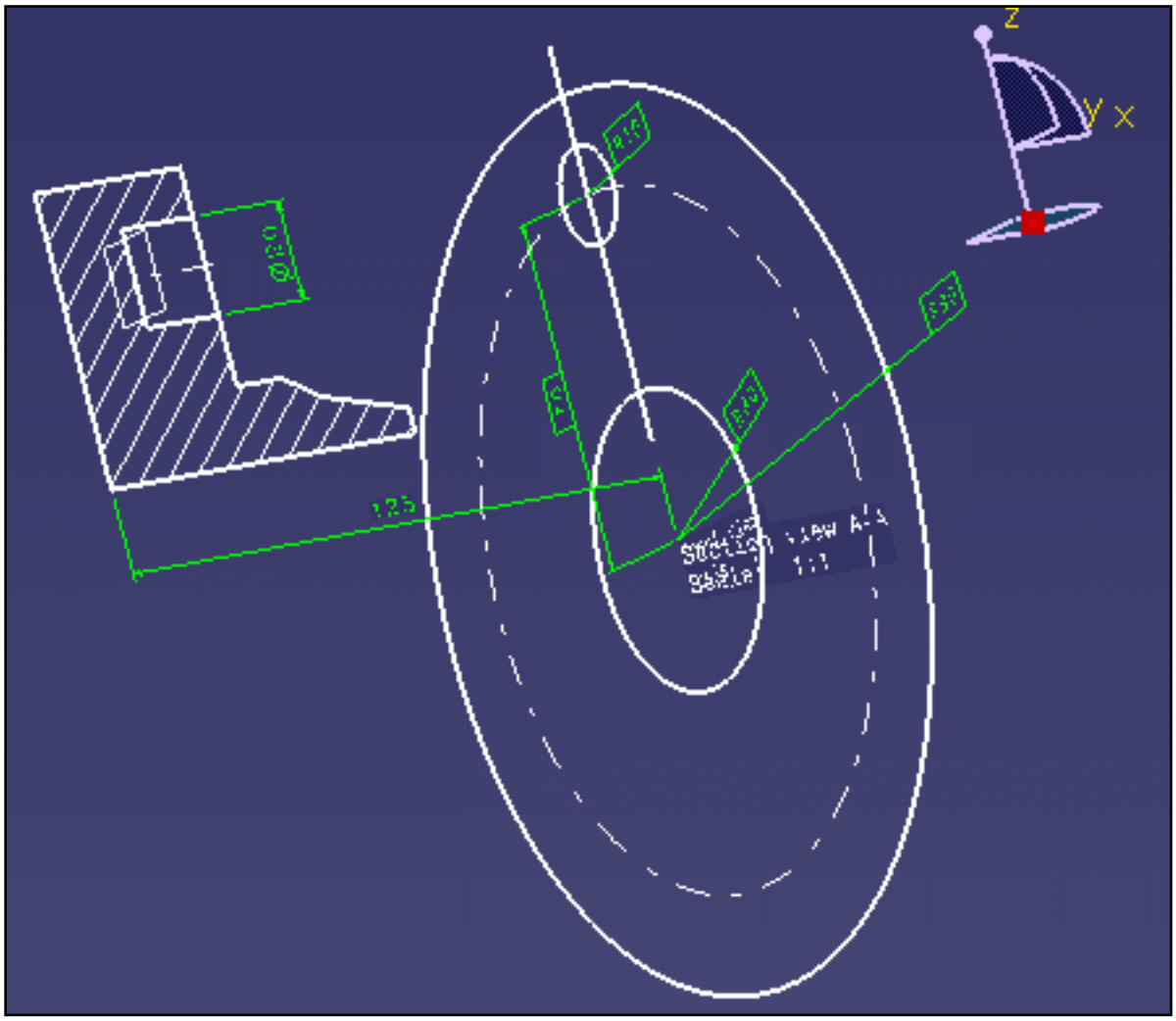
3. Select the line in the section view as shown below.



The 3D plane, Plane2DL.1, is created and it is listed in the specification tree, under the PartBody node.

- Of all elements created from 2D geometry in 2D Layout for 3D Design, only 3D profiles and 3D planes belong to the current part body.
- Note that 3D profiles and 3D planes are created under the current part body only when working in a hybrid design environment, that is when the **Enable hybrid design inside part bodies and bodies** option is selected in **Tools -> Options -> Infrastructure -> Part Infrastructure -> Part Document** tab (which is the case by default). Otherwise, when this option is not selected, 3D profiles and 3D planes are created in geometrical sets or ordered geometrical sets.

The 3D plane is also displayed in the 3D window.



More about creating and using 3D planes

You can create as many 3D planes as needed from a design view. 3D planes can be created:

- from any line contained in a design view, in a part layout (a CATPart document).
- from a line which is already included in other 3D planes or profiles (in other words, any 2D geometry can be included in several 3D outputs).

You cannot create 3D planes in isometric views (because they are not design views).

3D planes can be used to create:

- Part Design or Generative Shape Design features.
- 3D profiles on a plane parallel to the view support plane. For more information, refer to Creating a 3D Profile.

When creating 3D planes, remember the following points:

- Deleting a 3D plane does not delete the original 2D geometry in the layout. A 3D plane can only be deleted from the Part Design and Generative Shape Design workbenches.
- Deleting the 2D geometry used as input when defining a 3D plane in a view (which can only be done from the 2D Layout for 3D Design workbench) prevents the 3D plane from being re-built.
- **Cut**, **Copy** and **Paste** commands are not available for 3D planes.
- Powercopy is not available for 3D planes.
- You can only create a plane parallel to the support plane of the current view. An error message is displayed when selecting a line that would lead to the creation of a non-parallel plane.
- The 3D plane is associative to the line you select when creating it: if the line is modified, the support plane will be recomputed when updating the plane (by exiting the 2D Layout for 3D Design workbench or using the **Update 3D Profile** command) to reflect the modifications.

Use-Edges

Before you begin with use-edges: You should be familiar with important concepts.

Project 3D elements onto the view plane: create geometry in the current design view by projecting a 3D element onto the view plane.

Intersect 3D elements with the view plane: create geometry in the current design view by intersecting a 3D element with the view plane.

Project 3D silhouette edges: create geometry in the current design view by projecting the silhouette edge of a 3D element onto the view plane.

Before You Begin With Use-Edges

What are use-edges?

Use-edges are key in a concurrent engineering process, helping you with your design by letting you re-use existing elements from a part coming from another designer for example. In other words, use-edges help you create geometry from existing 3D shapes as well as from other layouts.

The 2D Layout for 3D Design workbench provides three use-edge functionalities:

- Project 3D elements onto the view plane: lets you create geometry in the current design view by projecting a 3D element onto the view plane.
- Intersect 3D elements with the view plane: lets you create geometry in the current design view by intersecting a 3D element with the view plane.
- Project 3D silhouette edges: lets you create geometry in the current design view by projecting the silhouette edge of a 3D element onto the view plane.

What you need to know about use-edges

Use-edge commands are only available when the current view is a design view. In other words, they are not available in isometric views (because they are not design views).

The input element (the element to be projected or intersected) is any visible geometry which does not directly belong to the current view. This means that you can select geometrical elements contained in the 2D or 3D background of any view, as well as geometrical elements contained in non-current design views, with the exception of geometry in a 2D component instance which cannot be selected.

The table below sums up the types of elements that you can select, depending on where they are located and visualized.

Can I select geometry in...	...when it is visualized in the current view?	...when it is visualized in other views?
...the 3D background...	Yes	Yes
...the 2D background (except 2D Component instances)...	Yes (except geometry defined in the main view)	Yes, (except geometry defined in the current view)
...a design view (except 2D Component instances)...	No	Yes

As shown in the table above, you can select geometry which is not only visible in the current view, but also in a different view.

Differences with use-edges in Sketcher

If you are familiar with the Sketcher workbench, you need to be aware that use-edges in 2D Layout for 3D Design differ from use-edges in Sketcher to a certain extent. The restrictions listed below describe these differences.

- Use-edges are never created associative with their reference element, which means that any modification of the reference geometry is not reflected in the use-edge, even after an update operation.
- As associativity is not supported, use-edge elements are simply datum elements (Point, Line, Circle and so on).
- Use-edges are created with the same color and line thickness as regular elements (they are not displayed using a specific color or line thickness).
- In 2D Layout for 3D Design, elements created in a view are not visualized in the specification tree (geometry, annotation, dimension, dress-up and so on). This is also the case of use-edges: created elements are not visualized in the specification tree.

Projecting 3D Elements onto the View Plane

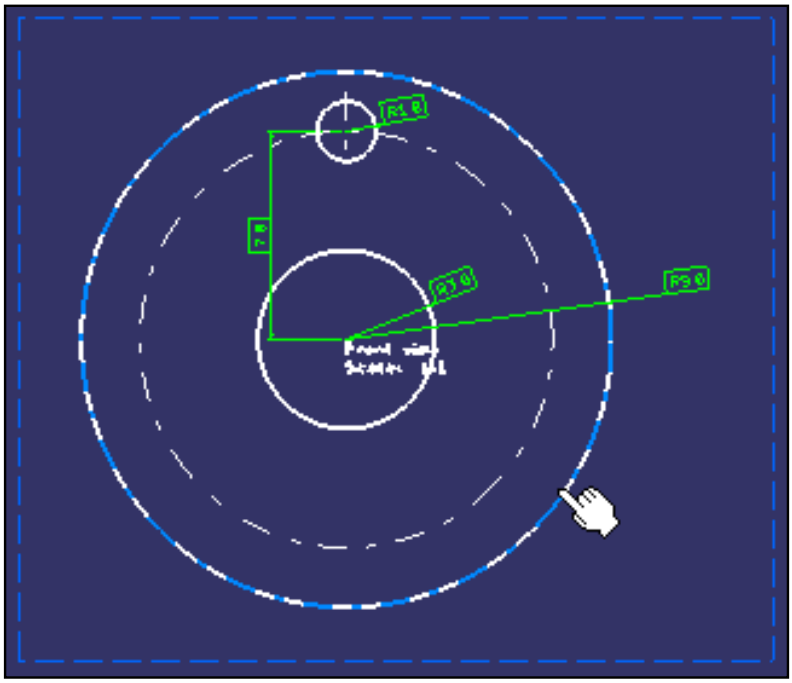
In this task, you will learn how to create geometry in the current design view by projecting a 3D element onto the view plane.

Open the Disk2.CATPart document. Select **Start -> Mechanical Design -> 2D Layout for 3D Design** to open the layout in the 2D window, alongside the 3D window.

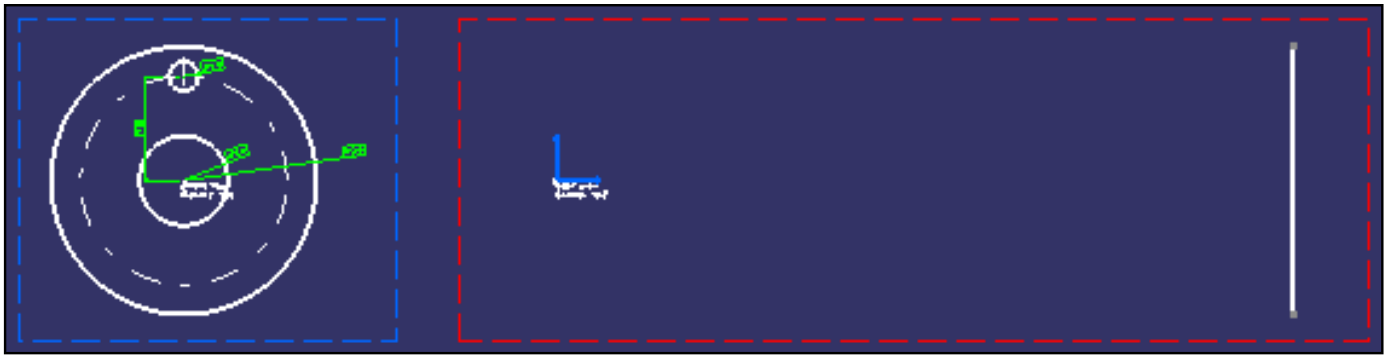
Create a left projection view from the existing front view.

1. Double-click the left view to activate it.
2. Click the **Project 3D Elements** icon in the **3D Geometry** toolbar.

3. Select the element to be projected. For the purpose of our scenario, select the bigger circle in the front view.



The edge is projected onto the left view plane.



More about projecting 3D elements

When projecting 3D elements, remember the following points:

- The element to be projected can be either a 3D face, an edge or a vertex. It cannot be a body.
- Selecting a face results in selecting its boundary edge(s).
- The projected geometry is created in the current view.
- The projected geometry is not necessarily a single element; distinct elements are created corresponding to each selected edge or vertex.
- If the selected element is invalid or if the projection is not properly performed, an error message is displayed.
- Be careful when multi-selecting a mix of 2D and 3D elements to project. Indeed, in this case, 2D elements are projected only if they are located in the view background; in other words, any 2D element located in the foreground of the view will not be taken into account.

Intersecting 3D Elements with the View Plane

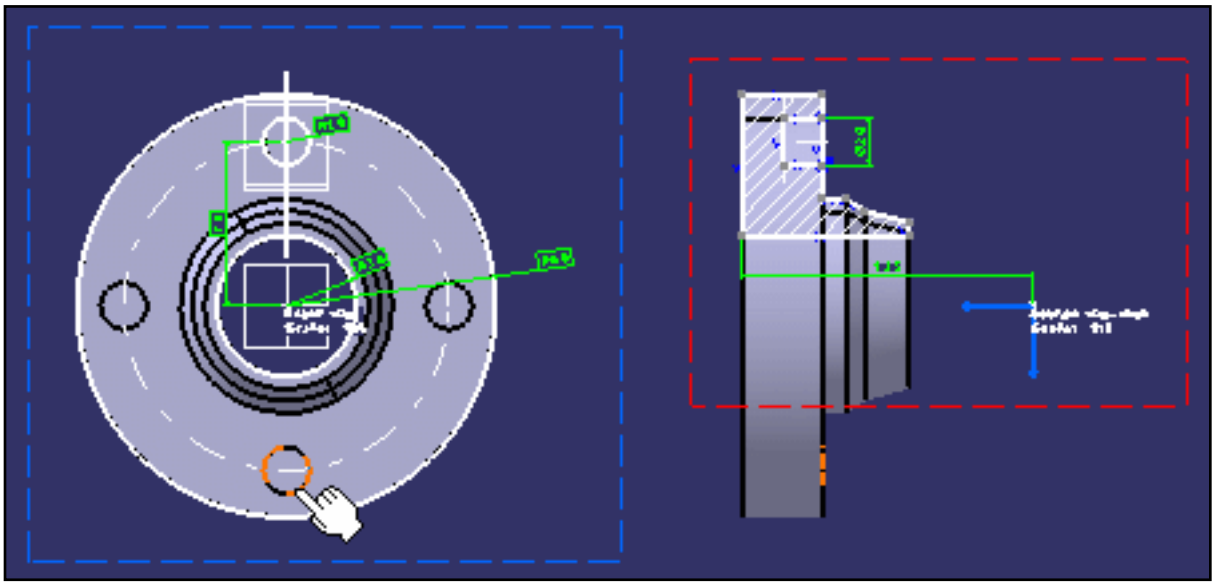
In this task, you will learn how to create geometry in the current design view by intersecting a 3D element with the view plane.

Open the Disk4.CATPart document. Select **Start -> Mechanical Design -> 2D Layout for 3D Design** to open the layout in the 2D window, alongside the 3D window.

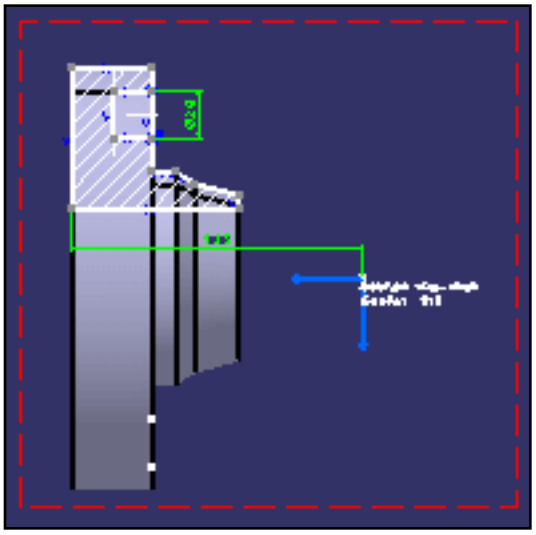
In the **Visualization** toolbar, activate the **Display Backgrounds as Specified for Each View** icons.

1. Double-click the section view to activate it.
2. Click the **Intersect 3D Elements** icon in the **3D Geometry** toolbar (**Use-edge** sub-toolbar).

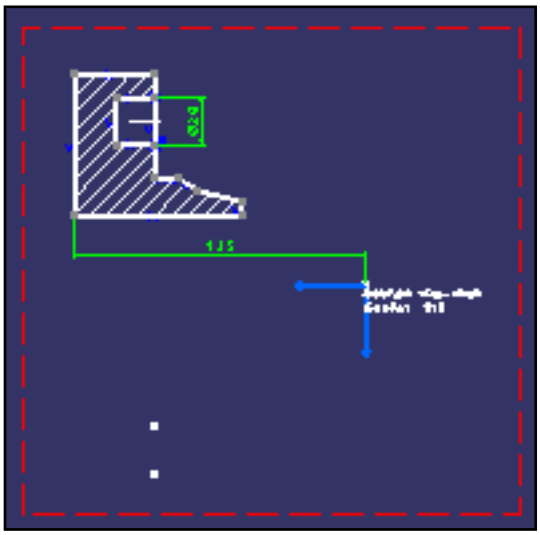
3. Select the element to be intersected. For the purpose of our scenario, select a pocket from the circular pattern in the 3D background of the front view.



The intersected geometry is created onto the section view plane.



4. Optionally, deactivate the **Display Backgrounds as Specified for Each View** icon to view the projected silhouette edge without the 3D background.

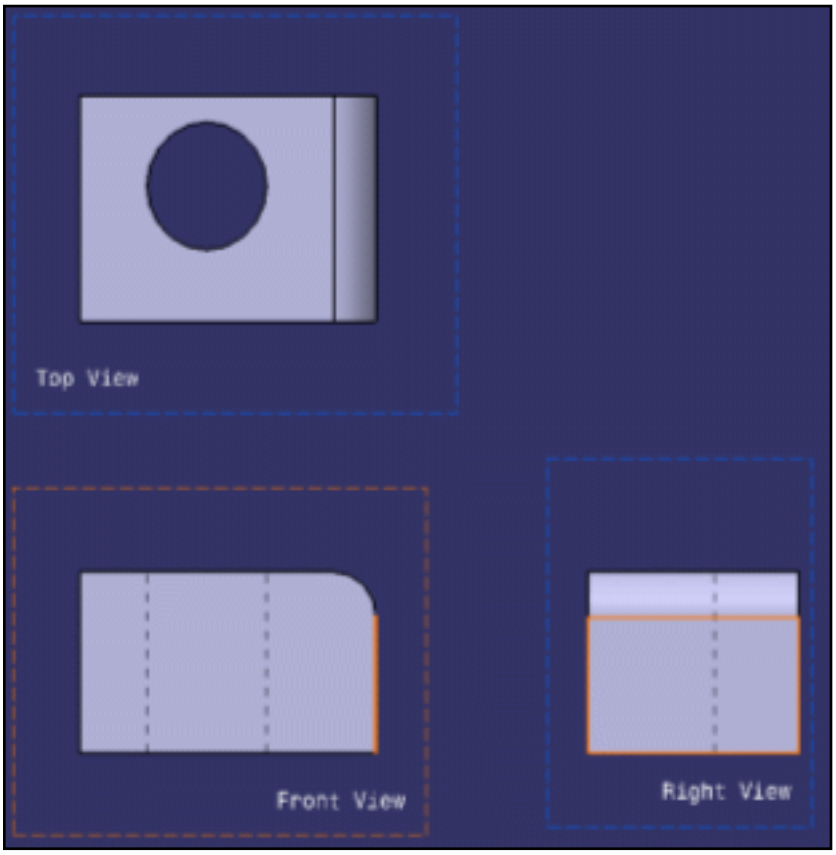


More about intersecting 3D elements

When intersecting 3D elements, remember the following points:

- The element to be intersected can be either a body, a surface/solid feature in a body, a face or an edge.
- Selecting a body or a feature results in selecting its constituent face(s).
- The intersected geometry is created in the current view.
- If the selected element is invalid or if no intersection is found, an error message is displayed.
- Be careful when multi-selecting a mix of 2D and 3D elements to intersect. Indeed, in this case, 2D elements are intersected only if they are located in the view background; in other words, any 2D element located in the foreground of the view will not be taken into account.
- In certain cases, the object to be intersected (often a face) may be perpendicular to the current view, and may be seen as "degenerated", in which case you may have difficulty selecting it.

In such a case, you may select that object from a different view. As shown below, for example, you could select a face in the right view (non-current view) to create an intersection (a line in this case) in the front view (current view).



- When the selected element is a face with planar geometry, the following cases are possible:
  - If the face is not parallel to the view plane and the intersection is found inside the face topology: the result is a limited line, which is identical to the result of the face intersection.
  - If the face is not parallel to the view plane and the intersection is not found inside the face topology: an infinite line is created as the result of the two infinite planes. Note that in this case, the created line is a construction element.
  - If the face is parallel to the view plane: no element is created.

Projecting 3D Silhouette Edges

In this task, you will learn how to create geometry in the current design view by projecting the silhouette edge of a 3D element onto the view plane.

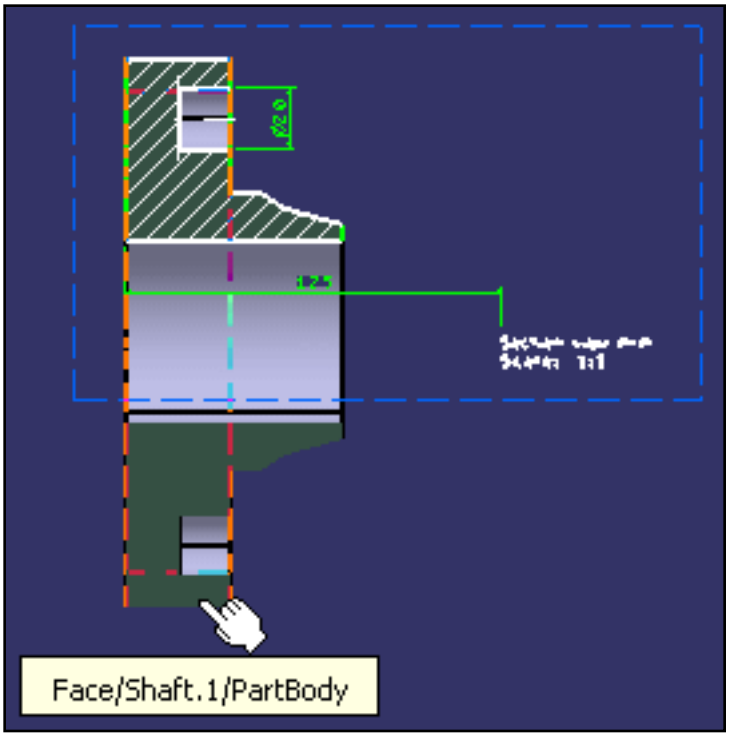
Open the Disk4.CATPart document. Select **Start -> Mechanical Design -> 2D Layout for 3D Design** to open the layout in the 2D window, alongside the 3D window.

Make sure the front view is active. If not, double-click to activate it. Create a right projection view from the existing front view.

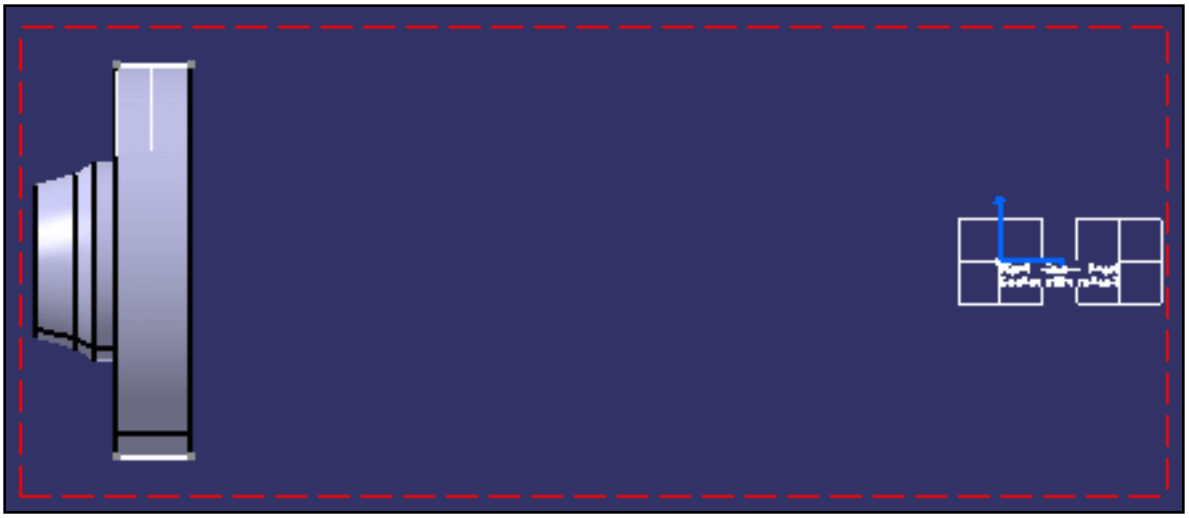
In the **Visualization** toolbar, activate the **Cutting Plane** and the **Display Backgrounds as Specified for Each View** icons.

1. Double-click the right view to activate it.
2. Click the **Project 3D Silhouette Edges** icon in the **3D Geometry** toolbar (**Use-edge** sub-toolbar).

3. In the 3D background of the section view, select the canonical surface to be projected.



The silhouette edge is projected onto the right view plane.



4. Optionally, deactivate the **Display Backgrounds as Specified for Each View** icon to view the projected silhouette edge without the 3D background.

More about projecting 3D silhouette edges

When projecting 3D silhouette edges, remember the following points:

- You can only project a silhouette edge from a canonical surface (cylinder, sphere, torus) whose axis is parallel to the view plane.
- The projected geometry is created in the current view.
- If the selected element is invalid, an error message is displayed.

Integration with the Drafting Workbench

Export a drawing view to a layout: create, from a selected view in a drawing, a layout view containing annotations, dress-up elements and geometry.

Create drawings and drawing views from a layout: create a drawing from a layout, and create a drawing view from a layout design view or isometric view.

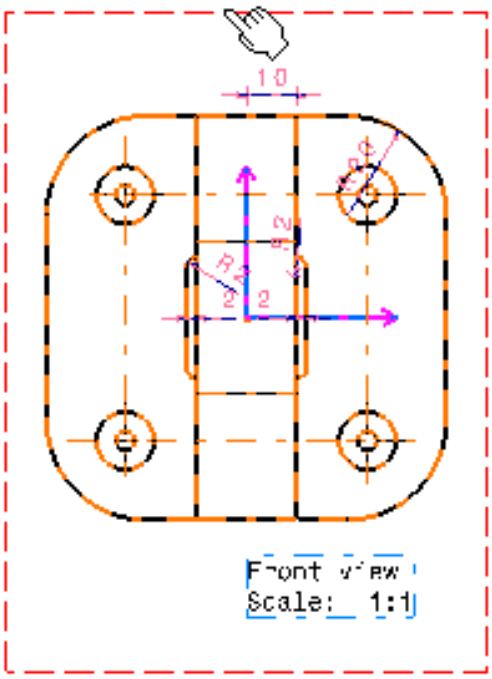
Exporting a Drawing View to a Layout

In this task, you will learn how to create, from a selected view in a drawing, a layout view containing geometry, annotations, dimensions, dress-up elements and so on.

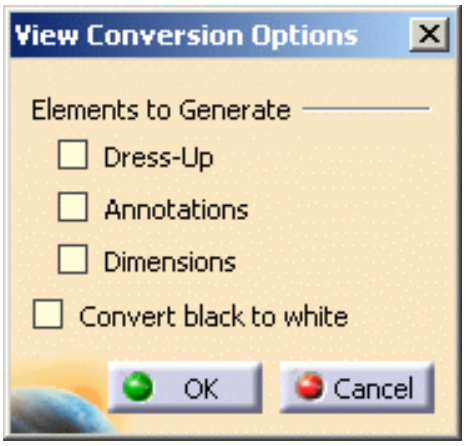
Open the Disk.CATPart document. Double-click Sheet.1 in the specification tree to open the empty layout in the 2D window.

Open the Views.CATDrawing document. This drawing contains front, left and top views.

1. From the drawing window, select **Tools -> Export to Layout View**.
2. Select a view, either from the specification tree or from the drawing sheet. For example, select the front view.



The **View Conversion Options** dialog box is displayed.



3. Optionally select the **Dress-Up**, **Annotations** and **Dimensions** options to generate these elements in

the layout.

4. Select the **Convert black to white** option, to generate elements (dress-up, annotations, dimensions and geometry) using the white color in the layout if they are black in the drawing: this will allow you to see them better.

   If the original drawing elements are in a color other than black, this color is maintained for the generated 3D elements.

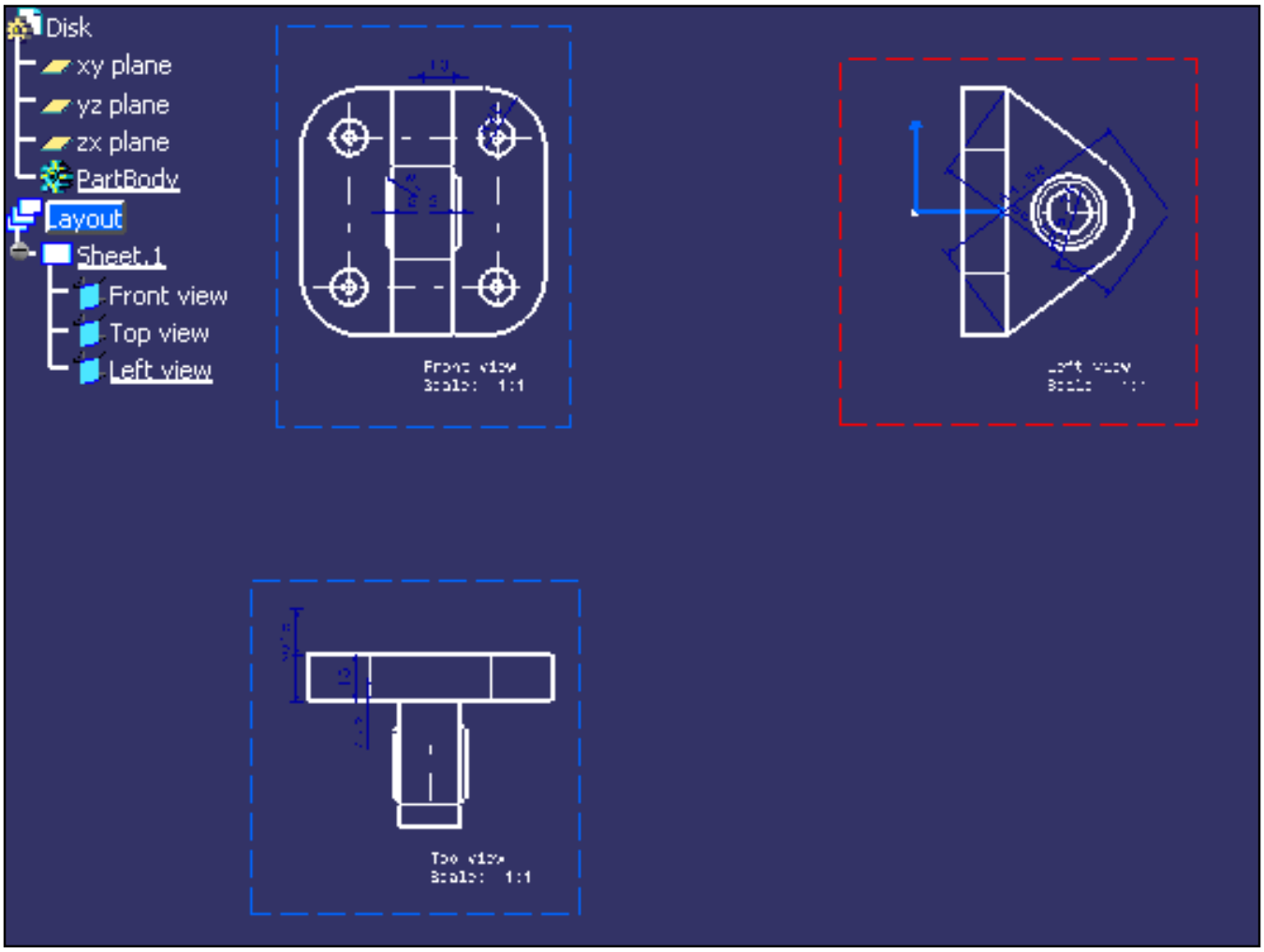
5. Click **OK**.

6. In the layout window, select the empty sheet either from the specification tree or from the geometry area.

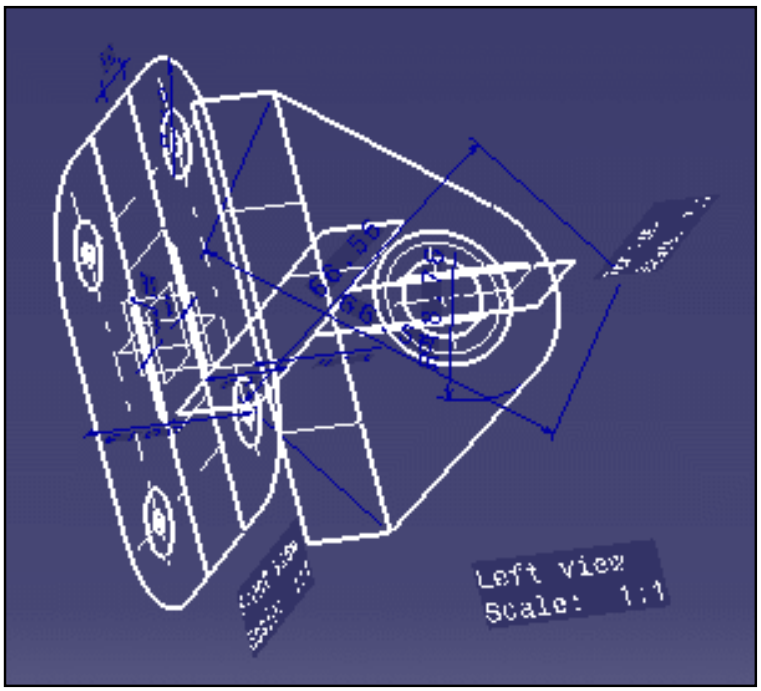
   The front view is created in the layout sheet and listed in the specification tree.

7. Repeat steps 1 to 6 to export the top and left views from the drawing to the layout.

   All views are created on the layout sheet (with their dress-up, annotations and dimensions if you generated them) and they are listed in the specification tree.



If you now switch to the part window, you can see how the layout is previewed.



More about exporting a drawing view to a layout

When exporting a drawing view to a layout, remember the following points:

- You cannot export generative views (generated from the 3D via the Generative Drafting workbench) to a layout sheet unless you isolate them first. In other words, you can only export interactive views or isolated generated views.
- Whatever their original type, all exported views are created as auxiliary views in the drawing, even though they keep the same name. For example, a left view in a layout sheet will still be called left view in the drawing but will be an auxiliary type view.
- You can only export to a layout sheet the drawing view types listed below:
  - Front, Left, Right, Top, Bottom, Rear and Auxiliary View
  - Isometric View (no geometrical element will be visualized in the 3D window)
  - Isolated Section View and isolated Section Cut.

  You cannot export any other view type than those listed above, and you cannot export views in detail sheets.
- The position in 3D space of layout views thus generated is implicitly defined by the drawing view.
- You cannot export a drawing view to a layout detail sheet.
- When a 2D component instance contained in a drawing view to export uses a 2D component reference from a catalog, the instance created in the layout will be linked to the original reference in the catalog, not to the 2D component instance in the drawing. In this case, if the 2D component instance in the drawing view is not synchronized with its catalog reference when exporting the view, then the instance in the layout will differ from that of the original drawing.
- You cannot export to a layout sheet any image contained in the drawing. Drawing views containing images will be exported without these images.

- The generated layout elements (dimensions, annotations and dress-up) are represented according to the standard used by this layout, independently of the standard used by the drawing. In other words, some elements may be visualized differently in the layout and in the drawing if they use two different standards.
- The graphic properties applied to dimensions, annotations and dress-up are preserved at export (with the exception of the color which can be set to white if the original drawing element is black and you select the **Convert black to white** option when exporting).
- The graphic properties applied to geometry (color, pickable, Low/High Int, line type, line thickness) are preserved at export (with the exception of the color which can be set to white if the original drawing geometry is black and you select the **Convert black to white** option when exporting).
- Dimensions and annotations remain associative to geometry (positional and orientation links) at export.

Creating Drawings and Drawing Views from a Layout

In this task, you will learn how to:

- create a drawing from a layout
- create a drawing view from a layout design view or isometric view

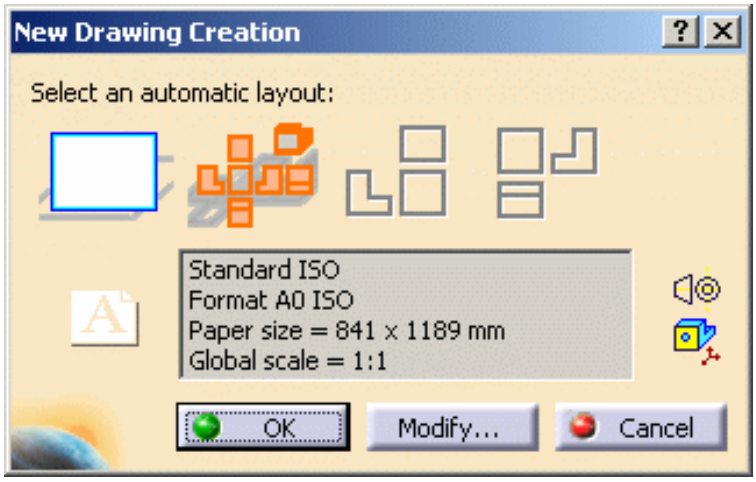
- Open the Disk4.CATPart document. Double-click Sheet.1 in the specification tree to open the layout in the 2D window.
- Go to **Tools -> Options -> Mechanical Design -> Drafting -> View** tab and make sure:
  - **Exact view** is selected as the view generation mode in the **View Generation** area (other view generation modes - CGR, Approximate and Raster - are not supported),
  - **Generate 2D Geometry** is selected in the **View from 3D** area (this specifies that 2D geometry should be generated).
- Still on the **View** tab, select **Project 3D wireframe** in the **Geometry Generation** area to make sure wireframe geometry is generated.
- Make sure you start the scenarios below from the 2D Layout for 3D Design window.

Creating a drawing from a layout

1. Select **Start -> Mechanical Design -> Drafting**.

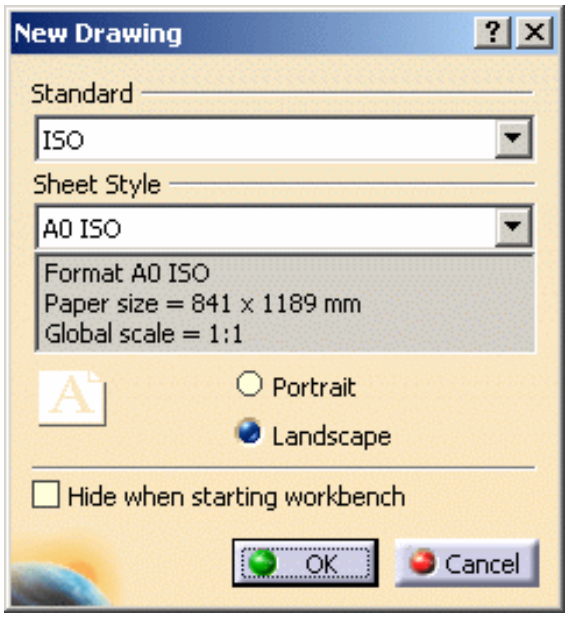
   The **New Drawing Creation** dialog box is displayed, offering the following drawing creation options:

   - **Empty sheet**: creates a new drawing with an empty sheet.
   - **All views**: creates a drawing from the current layout.



2. Click the **Modify** button.

   The **New Drawing** dialog box is displayed, allowing you to specify the standard, sheet style and orientation you want for the drawing. The sheet style defines among other things the sheet format, scale and orientation.



3. Make sure the ISO sheet style is selected: since the standard used for the layout is ISO_3D, you need to select a compatible standard.

4. Make sure the **A0 ISO** sheet style is selected, as well as the **Landscape** orientation.

5. If you do not want the **New Drawing** dialog box to appear the next time you create a drawing via the **Start** menu, select the **Hide when starting workbench** option.

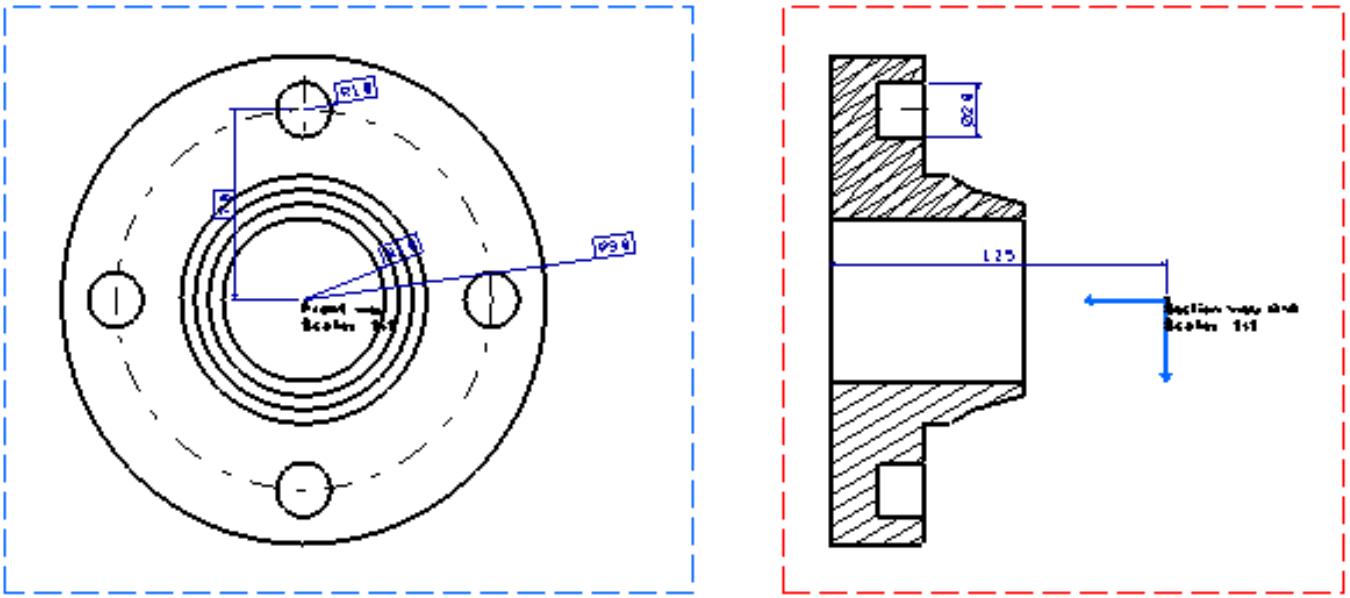
   In this case, the last selected standard, sheet style and orientation will be used by default when creating a drawing. You will always be able to reactivate this dialog box by unselecting the **Hide when starting workbench** option available through **Tools -> Options -> Mechanical Design -> Drafting -> General** tab.

6. Click **OK**.

7. Back in the **New Drawing Creation** dialog box, make sure **All views** is selected.

8. Click **OK**. When the drawing is created, the Drafting workbench is activated.

The created drawing is displayed with a front and section view as in the layout. Notice that elements which are white in the layout are converted to black in the drawing.



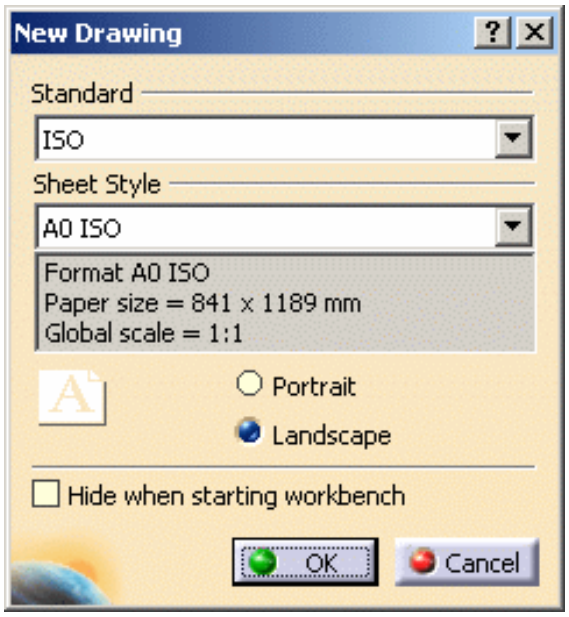
The created views are listed in the drawing specification tree.

Do not forget to save the drawing using **File -> Save As**.

Creating a drawing view from a layout design view or isometric view

1. Click the **New** icon or select **File -> New**. The New dialog box is displayed.
2. Select **Drawing** from the **List of Types** field, and click **OK.**

   The **New Drawing** dialog box is displayed, allowing you to specify the standard, sheet style and orientation you want for the drawing. The sheet style defines among other things the sheet format, scale and orientation.



3. Make sure the ISO sheet style is selected: since the standard used for the layout is ISO_3D, you need to select a compatible standard.

4. Make sure the **A0 ISO** sheet style is selected, as well as the **Landscape** orientation.

5. If you do not want the **New Drawing** dialog box to appear the next time you create a drawing via the **Start** menu, select the **Hide when starting workbench** option.

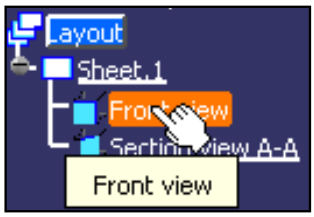
6. Click **OK**. A new drawing is created with an empty sheet.

7. Optionally tile the layout and the drawing windows vertically.

8. Click the **View From 3D** icon from the **Views** toolbar (**Projections** sub-toolbar).

9. Activate the 2D Layout for 3D Design window.

10. Select a view, either from the specification tree or from the geometry area. For example, select Front View from the specification tree.

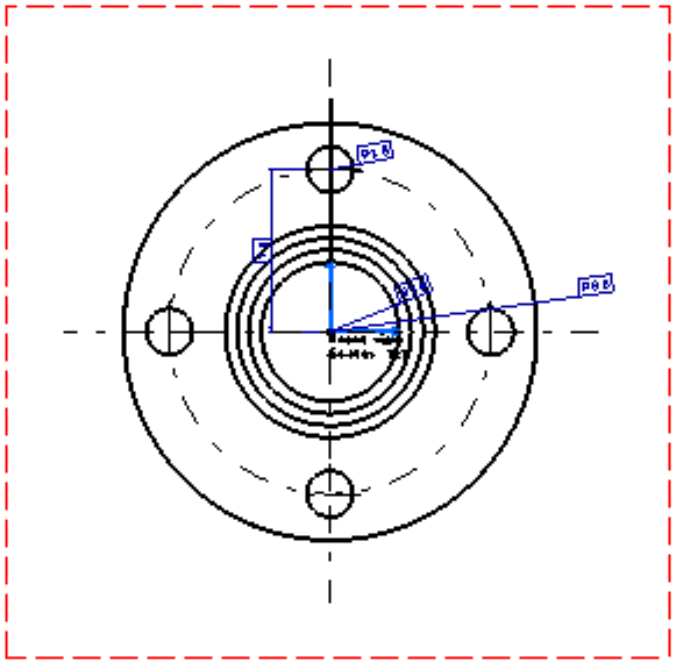


You could also select the view from the specification tree in the 3D part window.

The Drafting window is automatically activated. No preview is available.

11. Click on the drawing sheet at the location where you want to create the new view.

The front view is created in the drawing.



It is listed in the drawing specification tree.

12. Repeat steps 8 to 11 if you want to create an additional view (the section view) in the drawing.

Do not forget to save the drawing using **File -> Save As**.

More about creating drawings and drawing views from a layout

Creating a drawing and drawing views from a layout lets you create classical drawings for production needs, enabling you to exchange strictly necessary data with third-parties without sharing the 3D model.

When creating a drawing or a drawing view from a layout, remember the following points:

- The standard used for the drawing must be compatible with the standard used for the layout (for example, JIS for the drawing and JIS_3D for the layout).
- Once created, drawing views have the same type and the same name as their original views.
- Only design views and isometric views can be generated from a layout to a drawing.
  Main views and background views cannot be generated. However their content is copied when creating a full drawing.
- 2D component references in layout detail sheets are not generated in drawings. Likewise, detail sheets are not taken into account during the generation.
- 2D component instances are not generated as such in drawing views: it is their sub-elements which are generated as 2D geometry.

- Construction geometry and geometry which is placed in No Show space are not generated in drawing views.
- Dimensions that are generated directly from the 3D are isolated. Such dimensions are displayed by default using the dark blue color when the Analysis Display Mode is activated in **Tools -> Options -> Mechanical Design -> Drafting -> Dimension** tab.
- Dimensions added to generated 2D geometry (in drawing views) are not associative. Such dimensions are displayed by default using the grey color when the Analysis Display Mode is activated in **Tools -> Options -> Mechanical Design -> Drafting -> Dimension** tab.
- Interactive geometry cannot be constrained to generated 2D geometry. Therefore, when adding interactive geometry, geometrical constraints are detected but not created, and constraint creation commands do not authorize the selection of generated 2D geometry.
- Associative position and associative orientation do not work on generated 2D geometry and generated annotations (in drawing views). Therefore, selecting such items is impossible when creating annotations.
- The generation of 2D geometry (wireframe and 2D points) is optional. You can specify whether you want to generate 2D geometry using the **View from 3D -> Generate 2D Geometry** option available via **Tools -> Options -> Mechanical Design -> Drafting -> View** tab (this option is also accessible in the drawing via the generated view's properties: **Properties -> View from 3D -> Generate 2D Geometry**).
  Once this option on, you need to select **Project 3D wireframe** and/or **Project 3D points**, which are available via **Tools -> Options -> Mechanical Design -> Drafting -> View** tab to make sure 2D geometry and/or 2D points are generated.
- Layout elements which are hidden are not generated: this avoids overloading .CATDrawing documents with elements in No Show space.
- The created drawing is associative to the layout, which means that if you modify the layout (if you add or delete annotations, dimensions or dress-up, or if you add or delete geometry for example), the drawing will usually appear as being not up-to-date and you can update it (you can update all views or a selection of views in the drawing). There is an exception: if you simply modify geometry (change the coordinates, for example) or graphic properties, the drawing will not appear as being not up-to-date.

  The Update icon is active in the Update toolbar to indicate a drawing or a sheet which is not up-to-date and needs to be updated (this can be all views in the sheet or some of them only). Update symbols also appear in the specification tree to indicate drawings and sheets containing views that need to be updated.

  - You can update all views in the active sheet by clicking the Update icon .
  - You can update all views in a given sheet (or in a selection of sheets), by selecting and right-clicking the sheet(s) and then choosing **Update Selection**. You can also use the same method for a drawing: this will update all sheets (and therefore all views) in the drawing.
  - You can update a selection of views by selecting and right-clicking the view(s) you want to update and choosing **Update Selection** from the contextual menu. Only the items you select are updated.

  Update symbols remain in the specification tree for the items that have not been updated, so you always know which items are up-to-date and which are not.
- Any operation (change the color, delete, move, for example) performed on drafting items generated from a layout view content is lost after an update. However, you will not be prevented from modifying these items.

Creating View Filters

In this task, you will learn how to create view filters, which enable you to modify the visualization of elements in a layout design view.

Elements visualized in the 2D and 3D backgrounds of a given layout view come from other views in the layout or from other bodies of the part. As a result, the representation of a view can easily become overloaded: thanks to filters, you can visualize only those elements that are relevant for the design.

There are two types of view filters:

- Display filters define a list of elements to display and possibly overload while the remainder of the background is hidden.
- Mask filters define a list of elements to overload while the remainder of the background is displayed in the same way as defined in 3D.

Open the Valve.CATPart document. Double-click Sheet.1 in the specification tree to open the layout in the 2D window.

In the **Visualization** toolbar, make sure that the **Display Backgrounds as Specified for Each View** icon is active.

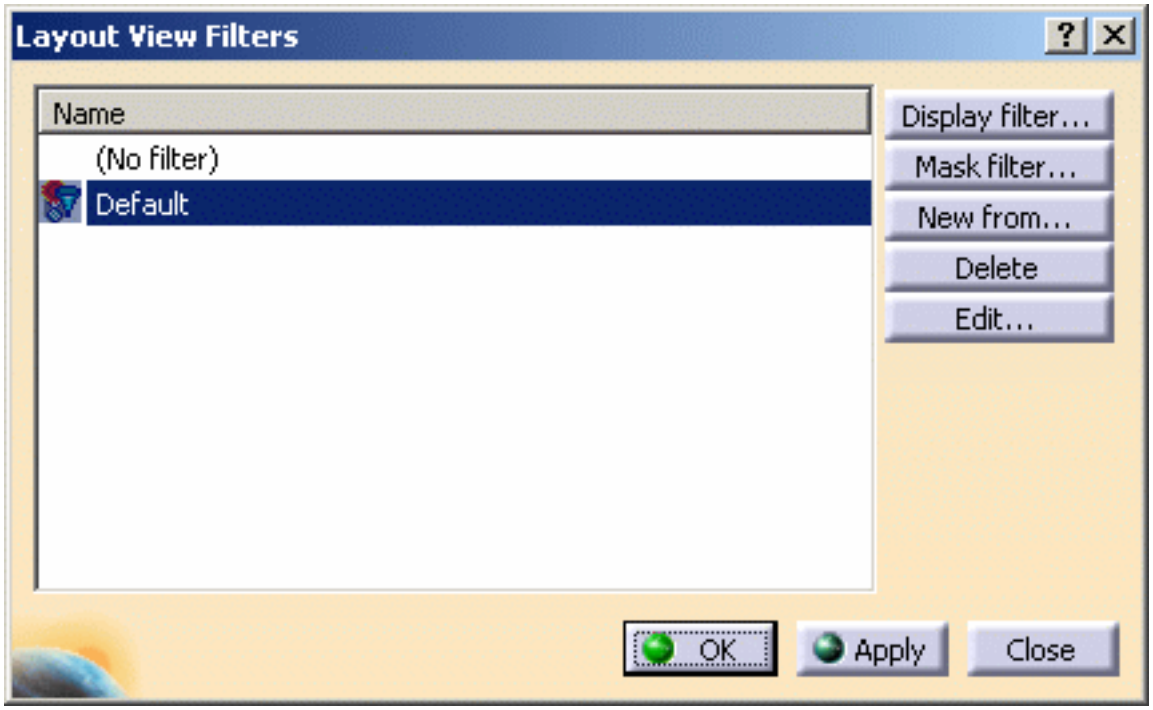
1. From the layout window, right-click the Front view from the geometry area or from the specification tree, and select **Filter -> Layout View Filters...**. The Layout View Filters dialog box is displayed, enabling you to create filters.

   - You can also:
     - select **Filter -> Edit Filter: Default** to start editing directly the filter that is applied to the view by default (or whatever filter is applied to the current view).
     - multi-select views to create a filter that will apply to several views.
     - launch the Layout View Filters dialog box directly (without right-clicking a view first) using **Tools -> Layout View Filters...**

     Whichever method you choose, any filter can be applied to any view in a layout.

   - To apply an existing filter to a specific view, use one of the following methods:
     - right-click the view(s) and select **Filter -> Layout View Filters...**. From the Layout View Filters dialog box, select the chosen filter and click **Apply**.
     - click the view(s) and select **Edit -> Properties**. On the **Visualization** tab, select the chosen filter from the **Filter** list and click **OK**. Refer to Editing View Properties for more information.

   - To edit a filter name in the Layout View Filters dialog box, select the filter name and then click it.

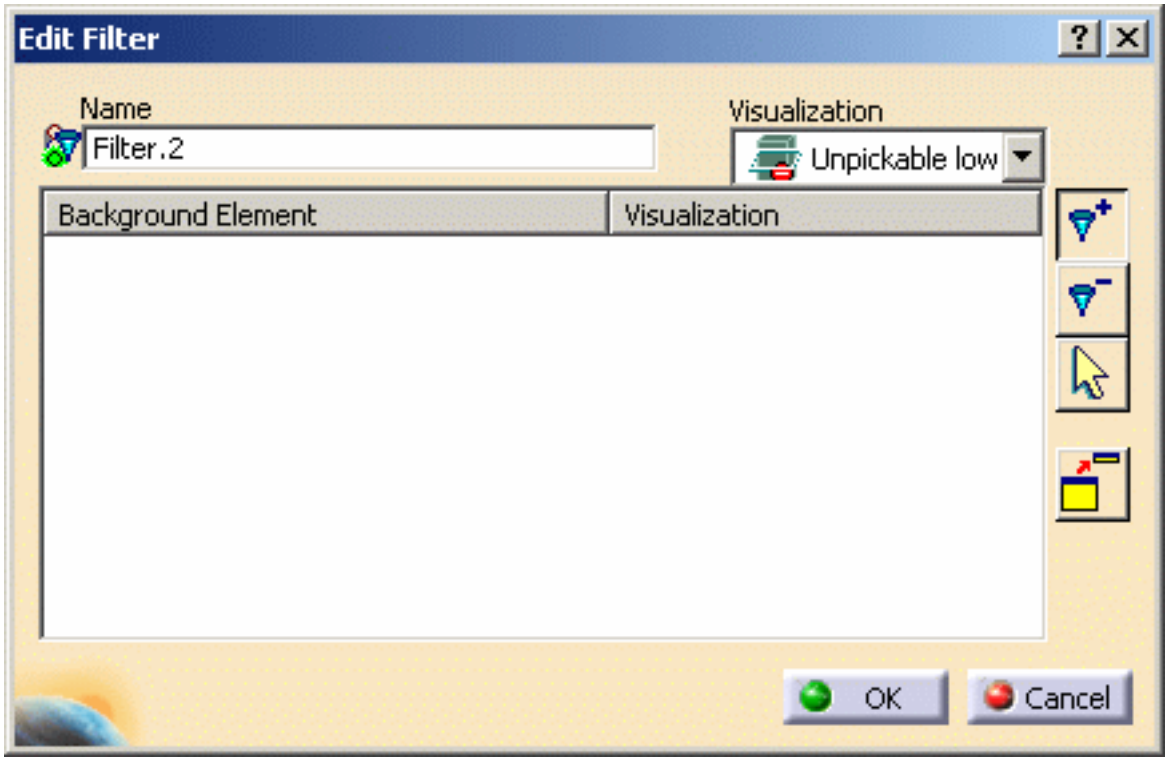


The Layout View Filters dialog box contains the following buttons:

- **Display filter...**: creates a new display filter.
- **Mask filter...**: creates a new mask filter.
- **New from...**: creates a new filter of the same type (Display or Mask) as the selected filter.

  Once created, this filter is not associative with the original filter.
- **Delete**: deletes the selected filter(s).
- **Edit...**: lets you edit the characteristics of the selected filter through the Edit Filter dialog box.
- **Apply**: applies the selected filter to the selected view without exiting the dialog box.
- **OK**: applies the selected filters and exits the dialog box.
- **Close**: exits the dialog box without validating your changes.

2. Click the **Display filter...** button. The Edit Filter dialog box is displayed, and a display filter named Filter.2 is created. You can edit this name if you want.



The Edit Filter dialog box contains the following fields and buttons:

- **Name**: lets you edit the name of the filter.
- **Visualization**: lets you choose the visualization type that should be applied to the selected elements.

  The various visualization types are:

  - **Standard**: shows both the 2D and 3D backgrounds.
  - **Unpickable**: prevents selecting all elements in both the 2D and 3D backgrounds, even though you can see them. You can just handle 2D elements which belong to the current view.
  - **Low-intensified**: dims all elements in both the 2D and 3D backgrounds.
  - **Unpickable low-intensified**: dims all elements in both the 2D and 3D backgrounds. Additionally, although you can see these elements, you cannot select them. You can just handle 2D elements in the current view.

- **Add mode** : adds to the list the elements you select in the specification tree or in the geometry area, and filters them according to the visualization type selected in the **Visualization** list.

- **Remove mode**: removes from the list the elements you select in the specification tree or in the geometry area.

- **Select mode**: lets you select elements and change their visualization type in the **Visualization** list.

- **Switches to Tools Palette**: transforms the Edit Filter dialog box into a **Tools Palette** containing the same commands as the dialog box.

In the **Tools Palette**:

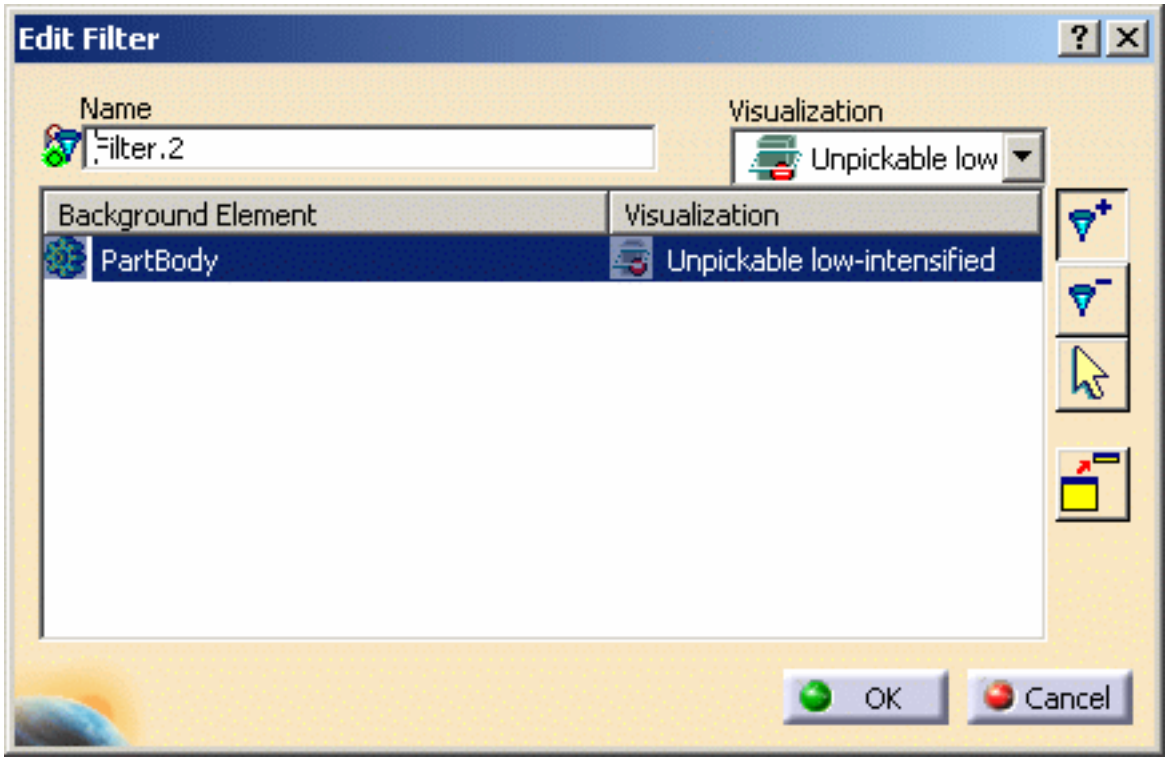
  - the **Switches to dialog box** icon transforms the **Tools Palette** into the Edit Filter dialog box.

  - the **Validates command** icon validates your modifications and closes the **Tools Palette**.

3. For the purpose of this scenario, you will create a filter that prevents PartBody from being selected using its 3D background and that dims it.

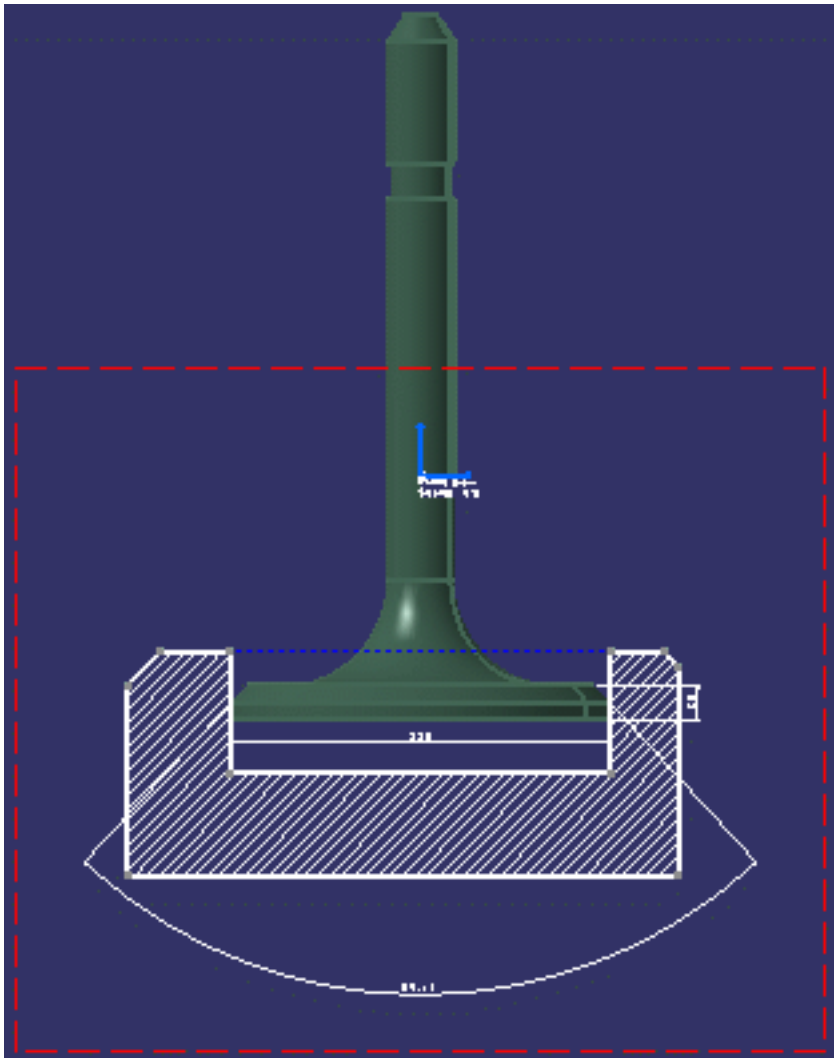
   Select the **Unpickable low-intensified** visualization type and make sure the **Add mode** is active.

4. Select PartBody in the specification tree. PartBody is now added to the filter and the **Unpickable low-intensified** visualization type is applied to it.



5. Click **OK** to validate and exit the dialog box.

6. Back in the Layout View Filters dialog box (where your new filter, Filter.2, is now listed and selected), click **OK**. The dialog box is closed and the newly created filter is applied to the front view. Note that you cannot select the PartBody via the 3D background and that it is displayed using a dimmed color.



About view filters

Filterable elements

You can filter the following elements:

- Part Design: xy plane, yz plane, zx plane, PartBody, Body at the first level, Geometrical Set (under the root part or contained in another Geometrical Set), Ordered Geometrical Set of first level, User Defined Feature (only contained in a Geometrical Set)
- Functional Tolerancing & Annotation: Dimension, Annotation, Note, Geometrical Tolerance, Datum, Capture Callout, Deviation, Distance Between two points, Constructed geometry, View, Annotations Set.
- Generative Shape Design: all elements contained in a Geometrical Set (under the root part or contained in another Geometrical Set), and only those.
- 2D Layout for 3D Design: design views belonging to the same layout.

Non-filterable elements

In general, you cannot filter any element that is not listed as being filterable (in the preceding paragraph). Among these, you cannot filter the following elements:

- Bodies or Ordered Geometrical Sets which are not at the first level.
- Elements contained in a PartBody or in an Ordered Geometrical Set.
- Part features, CAA features, Volumes, 3D Measure elements.
- Bodies that are the object of a boolean operation.
- Design views belonging to another layout, isometric views belonging to the same or another layout.

More about view filters

View filters can be created for the following view types:

- Projection views (front, top, bottom, left, right and rear)
- Isometric views
- Section views, section cuts and auxiliary views

When creating view filters, remember the following points:

- You cannot change the filter type (Display or Mask) once it has been created.
- Filters do not change the graphic properties of elements in the filtered view, they simply overload them. This means that filters cannot show elements that are hidden in the 3D model, nor can they make pickable elements which are not pickable in the 3D model.
- More generally, filters conform to the standard behavior of elements. Refer to the specific User's Guide for more information.
- When pasting to another layout a view to which view filters are applied, the filter is not pasted (therefore, the view is not filtered).
- When editing a multi-instantiated part layout in the context of a product, only the edited instance is affected by the filter, and only the sub-elements of the part reference can be filtered.
- Filtered elements may be impacted by the result of an operation (such as activating/deactivating the elements, changing its Hide, Low Intensity or No Pick property, cutting, deleting, etc.). Most operations are automatically supported and you do not need to perform an update.
  However, after having selected a feature or a Geometrical Set in the context of a **Change Geometrical Set** operation, you need to perform a manual update (using the Local Update command which is available when right-clicking the layout, sheet or view).
- You can launch **Edit -> Selection Sets** and **Edit -> Search** while the Edit Filter dialog box is running. However, only filterable elements are actually selected. You can also use the power input mode to launch a search query. For more information on selection sets, search operations and the power input mode, refer to the Infrastructure User's Guide.

About the Default filter

A Default filter exists in any layout and may be applied automatically to all newly created views (refer to About newly created views below for more information). This filter specifies only that the view should be hidden from the 2D background (the **Invisible** option for **2D background views** is set to Yes for each existing view): this means that in each view, by default, you only see the view content and the 3D background.

The purpose of this Default filter is to avoid overloading the layout with unnecessary data. Indeed, it is only in certain specific cases that you will want the 2D background to be displayed (when creating section views for example). You can always delete this filter, or apply another filter to any view in the layout.

About newly created views

New views are created according to the options selected in **Tools -> Options -> Mechanical Design -> 2D Layout for 3D Design -> View Creation** tab **-> Filter** category. Refer to View Creation for more information.

Printing a Layout

This task deals with the following subjects:

- Printing a Whole Layout Sheet
- Printing an Area of a Layout Sheet
- More About Printing
  - What is printed?
  - Which print mode (raster or vector) is used?
  - For more information

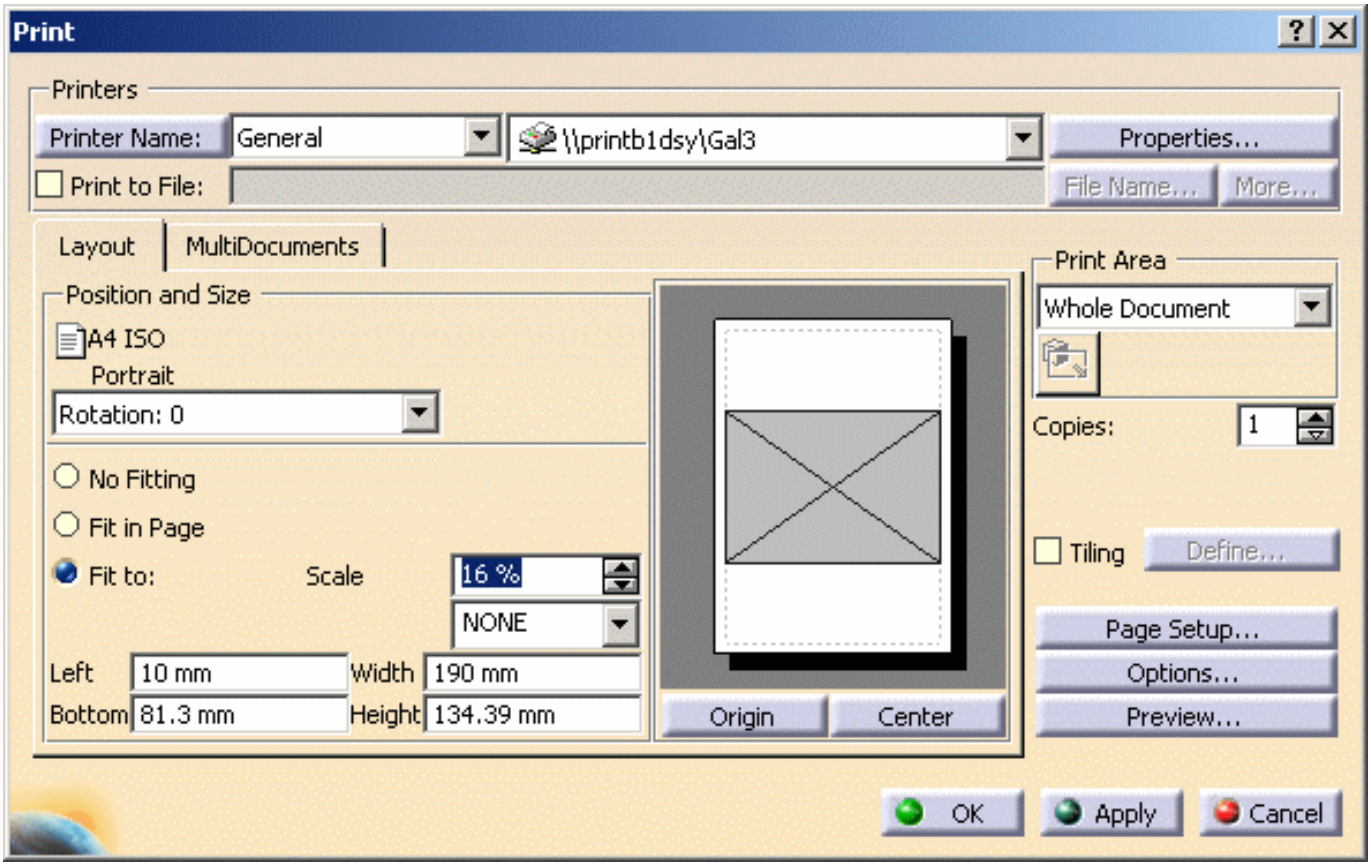
Open the Disk4.CATPart document. Double-click Sheet.1 in the specification tree to open the layout in the 2D window.

Printing a Whole Layout Sheet

Note that you may also print several sheets if a layout contains several of them.

1. Select **File -> Print**.

   The **Print** dialog box is displayed.



2. Choose your print options as required:

- The **Printers** area lets you choose the printer you want to use or specify whether you want to print to a file.
- The **Layout** tab lets you define the sheet orientation, position and size.
- The **MultiDocuments** tab lets you specify additional choices if the current document contains several sheets. Refer to Printing Multi-Documents in the *Infrastructure User's Guide* for more information.
- The **Print Area** area lets you define whether you want to print:
  - the entire sheet: **Whole Document**.
  - the sheet as seen on screen: **Display**.
  - the area selected using the **Selection** button . Refer to Printing an Area of a Layout Sheet below for more information.
  - the print area previously defined for the sheet: **Document area**. This print area is defined (and activated) in the sheet properties. Refer to Editing Sheet Properties for more information.

    Note that the **Document area** option appears only if you activated the print area in the sheet properties prior to accessing the **Print** dialog box.
- The **Copies** field lets you specify the number of copies to print.
- The **Tiling** option lets you tile the sheet and print it on several pages.
- The **Page Setup...** button lets you define the page setup.
- The **Options...** button lets you define additional options.
- The **Preview...** button lets you preview the document to be printed.

3. Click **OK** to print the sheet and close the **Print** dialog box.

Printing an Area of a Layout Sheet

1. Select **File -> Print**.

   The **Print** dialog box is displayed.

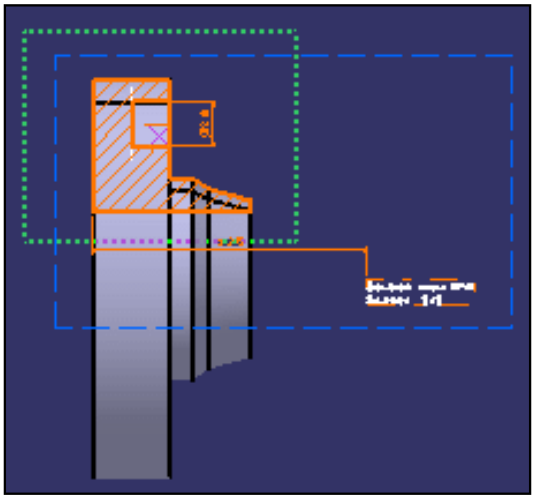
2. Choose your print options as required.

3. Choose **Selection** in **Print Area**.

   This activates the selection mode button and allows you to select the area to print.

4. Click the selection mode button .

5. Drag the cursor on the layout to define the print area.



6. Click **OK** to print the sheet and close the **Print** dialog box.

More About Printing

What is printed?

The print functionality usually prints exactly what is visible in the layout window (or, when printing multi-documents, what would be visible in the layout window if the printed sheet was the active one).

However, what is printed sometimes depends on the print mode (raster or vector). For example:

- View filters and the layout view background are always taken into account, independently of the print mode.
- The cutting plane, clipping frame, back-clipping plane are taken into account only when printing in raster mode.

In the case of the clipping frame, the frame itself is not printed.

For more information, refer to the next paragraph Which print mode (raster or vector) is used?.

Which print mode (raster or vector) is used?

Depending on which print **Rendering Mode** option you choose (from the **Print** dialog box: click the **Options...** button and then the **Various** tab), and on which visualization features are activated or not in the layout, print will be performed using a combination of raster and vector elements. The table below summarizes the possible scenarios.

In the table below:

- *2D elements* means the content of a layout view (2D geometry, dimensions, annotations, dress-up, 2D components) as well as its 2D background.
- *3D elements* means the 3D background of a layout view (including 3D wireframe, which belongs to the 3D background).
- *Basic visualization* means that the **Display all elements using Z-buffer** check box (in **Tools -> Options -> General -> Visualization** tab) is cleared and that the cutting plane, clipping frame, back-clipping plane are not activated.
- *Advanced visualization* means that either:
  - the **Display all elements using Z-buffer** check box is selected,
  - or the cutting plane, clipping frame, and/or back-clipping plane are activated. However, note that these features are not taken into account in vector mode, and that print is performed as if they were not activated.

	Then with basic visualization...		And with advanced visualization...
If the print **Rendering Mode** option is...	2D elements are printed in...	3D elements are printed in...	2D and 3D elements are printed in...
Default	Vector mode	Raster mode	Raster mode
Wireframe, Hidden Line Removal, Shading with Triangles	Vector mode	Vector mode	Vector mode*
Dynamic Hidden Line	Vector mode	Vector mode	Not supported

* NB: the cutting plane, clipping frame, and/or back-clipping plane are not taken into account in vector mode. If you need to print in vector mode while using advanced visualization, you can create a drawing from your layout and print this drawing. This will allow you to control in detail the vector representation options.

For more information

For detailed information about printing, refer to the the Printing Documents chapter in the *Infrastructure User's Guide*. Make sure you refer to Customizing Print Settings Before Printing Your Documents. The Printing Multi-Documents task should also be helpful.

Properties

This section descibes how to quickly access and edit information on layout sheets, views, 2D geometry, dress-up elements, annotations and dimensions in the Properties dialog box, which is available via the **Edit -> Properties** contextual command.

Note that the properties of most objects (dimensions, annotations, dress-up and so on) are edited exactly as in the Drafting workbench. For this reason, most tasks in this section of the documentation provide links to the *Interactive Drafting User's Guide*. As such, the information detailed in these tasks is presented in a Drafting context. You should note that the *Interactive Drafting User's Guide* contains images that correspond to the Drafting workbench and therefore illustrate geometry in an environment that is different from the 2D Layout for 3D Design environment (symbols and background color, for example).

Tasks documented in the *2D Layout for 3D Design User's Guide:*

Edit sheet properties: Access and edit sheet properties.

Edit view properties: Access and edit view properties.

Tasks documented in the *Interactive Drafting User's Guide:*

Edit 2D geometry feature properties: Access and edit information on 2D geometry features (name and stamp).

Edit graphic properties: Access and edit graphic properties.

Edit pattern properties: Access and edit pattern properties.

Edit annotation font properties: Access and edit annotation font properties.

Edit text properties: Access and modify text color, position and/or orientation.

Edit picture properties: Access and modify picture position, size, scale and compression.

Edit dimension text properties: Access and edit dimension text properties.

Edit dimension font properties: Access and edit dimension font properties.

Edit dimension value properties: Access and edit dimension value properties.

Edit dimension tolerance properties: Access and edit dimension tolerance properties.

Edit dimension extension line properties: Access and edit dimension extension line properties.

Edit dimension line properties: Access and edit information on dimension line properties.

Edit dimension system properties: Access and edit information on dimension system properties.

Edit 2D component instance properties: Access and edit 2D component instance properties.

Editing Sheet Properties

This task explains how to edit layout sheet properties.

Open the Disk3.CATPart document. Double-click Sheet.1 in the specification tree to open the layout in the 2D window.

1. Click Sheet.1 in the specification tree.
2. Select **Edit -> Properties**.

   The Properties dialog box appears, displaying the following tabs: **3D Visualization** and **Sheet**.
3. Make sure the **3D Visualization** tab is displayed. It contains the following property:

   ☐ Do not visualize in 3D

   ## Do not visualize in 3D

   Select this check box if you do not want any element on the sheet (views, geometry, annotations and so on) to be visible in a 3D workbench (such as Part Design).

   In this case, sheet elements will not be visible in Show or No Show space.

   Note that this property is independent of the **Hide / Show** option available in the sheet's contextual menu, which controls the visualization of the sheet both in 2D and 3D.
4. Click the **Sheet** tab. It contains a number of properties, some of which are unavailable in 2D Layout for 3D Design:
   - General properties
   - Format properties
   - Projection Method
   - Generative Views Positioning Mode
   - Print Area

General properties

Name

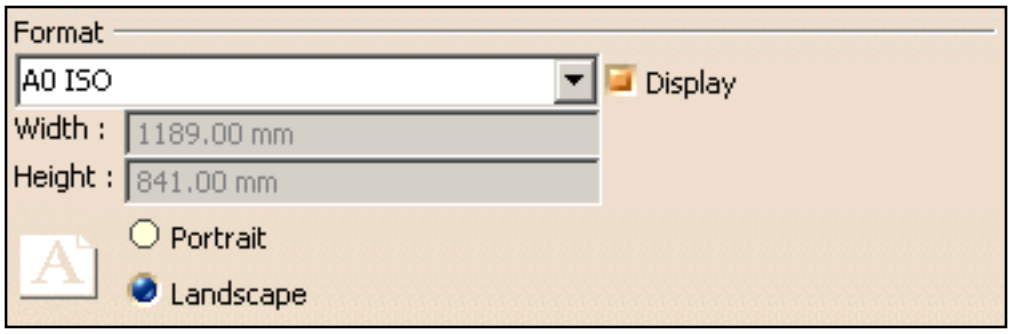
Enter a name for the sheet.

Scale

Specify the scale (the scaling factor) to be applied to all views in the sheet.

This implies that the scale of all existing views is multiplied by that of the sheet (for example, if existing views already have a scale of 1/10, and if you change the sheet scale to 1/10, then existing views will now have a scale of 1/100).

Format properties



Format

This list contains the format names defined by the administrator in the Standards Editor. For more details, see Sheet Format Definition in the Administration Tasks chapter.

You may also create your own user-defined formats, defined locally for a given layout. To create your own format, proceed as follows:

1. Type a name for the format in the **Format** field.
   The name of the newly created format must be different from those in the user-defined and standard lists of formats. If not, a warning message is displayed to inform you the format name you specified is not valid.
2. Use the tab key to access the **Width** and **Height** fields and set their values.

Display

Display the frame representing the format of the sheet.

Width

Width of the selected format. This field is editable in the case of user-defined formats only.

Height

Height of the selected format. This field is editable in the case of user-defined formats only.

Portrait / Landscape

Orientation of the selected format. Available only if the selected format allows you to modify the orientation type. For more information, refer to Sheet Format Definition.

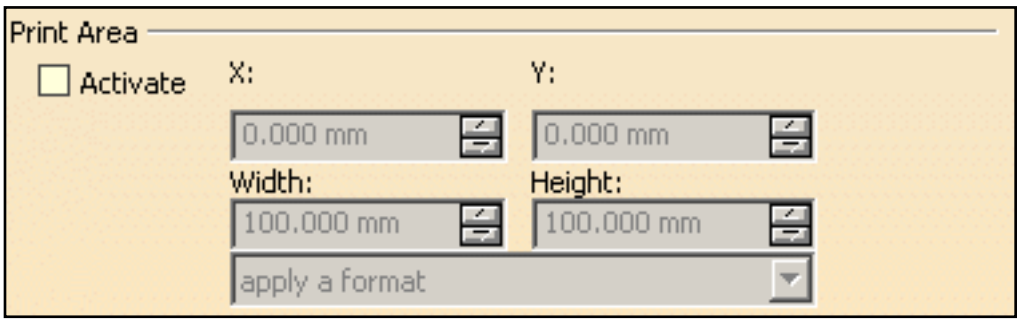
Projection Method

These properties are not available in 2D Layout for 3D Design.

Generative Views Positioning Mode

These properties are not available in 2D Layout for 3D Design.

Print area



Activate

Select this check box to specify that only a specific area of the sheet should be printed. Doing this will activate the associated fields so that you can define the print area.

In addition, you must select **Document area** option as the **Print area** in the Print dialog box in order for the print area to be printed. If you do not select the **Document area** option, the whole document will be printed. Refer to Printing a Layout for more information.

X

Specify the X coordinate of the lower left-hand corner of the print area.

Y

Specify the Y coordinate of the lower left-hand corner of the print area.

Width

Specify the width of the print area.

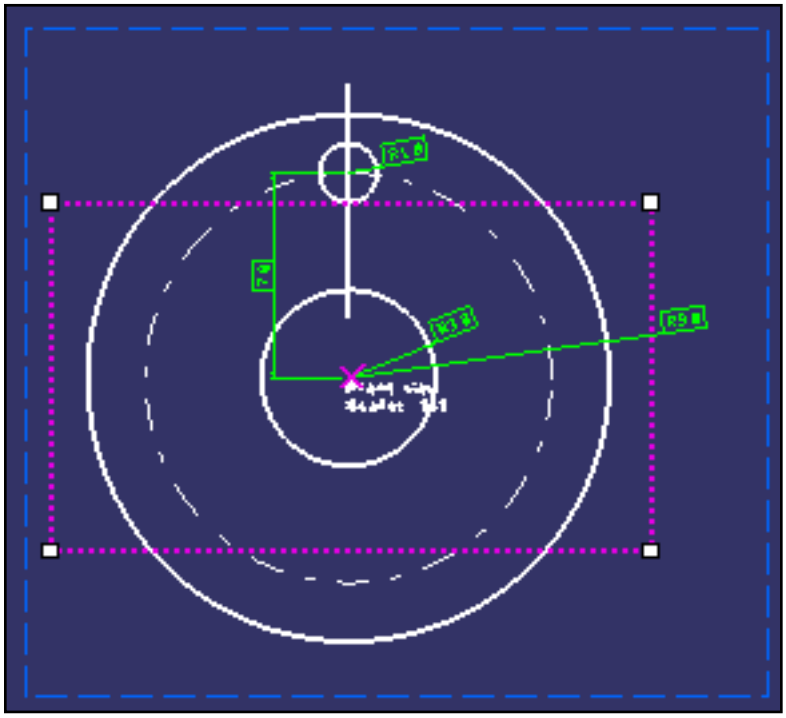
Height

Specify the height of the print area.

Format

Select a format if you want to define the print area using the width and height specified for that format.

A specific contextual command lets you visualize the print area (providing it is activated), so as to re-position or re-dimension it for example. To do so, either right-click the sheet item in the specification tree and select **Sheet.X object -> Visualize Print Area**, or activate the sheet and select **Edit -> Sheet.X object -> Visualize Print Area.** This zooms onto the print area, which is outlined as a purple dashed box, with an X cross at its center.



Use the manipulators at the corners of the box to re-dimension the print area. Drag the dashed box or the central X cross to re-position the print area.

You can then exit the print area visualization mode by pressing the **Escape** key or by clicking elsewhere in the drawing. You can check the sheet properties to make sure that the coordinates, width or height have been updated.

5. Change the sheet properties as desired, then click **OK** to validate.

Editing View Properties

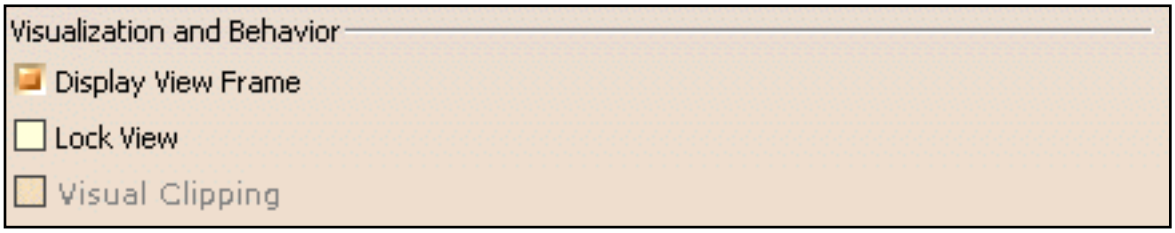
This task explains how to edit view properties.

Open the Disk3.CATPart document. Double-click Sheet.1 in the specification tree to open the layout in the 2D window.

1. Click the front view.
2. Select **Edit -> Properties**.

   The Properties dialog box appears, displaying the following tabs: **View**, **Visualization** and **Graphic**.
3. Make sure the **View** tab is displayed. It contains a number of properties, some of which are unavailable in 2D Layout for 3D Design:
   - Visualization and behavior
   - Scale and Orientation
   - Dress-up
   - View name
   - Generation mode

Visualization and behavior



Display View Frame

Shows/hides the view frame.

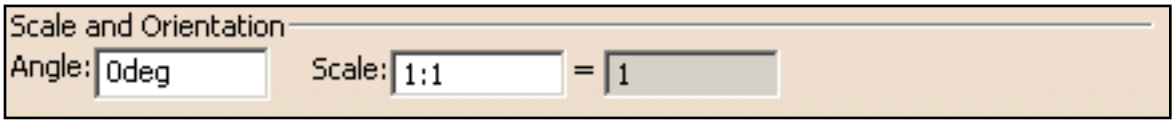
Lock View

Locks the view so that it cannot be modified.

Visual Clipping

This property is not available in 2D Layout for 3D Design.

Scale and Orientation



Angle

Defines the angle between the view and the sheet.

Scale

Defines the scale of the view.

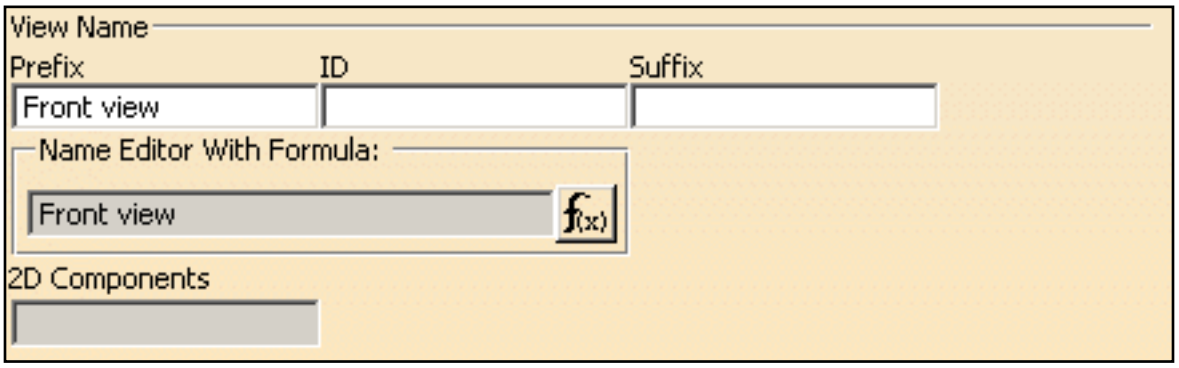
=

Displays the decimal value with respect to the scale. This field is read-only.

Dress-up

These properties are not available in 2D Layout for 3D Design.

View Name



Prefix / ID / Suffix

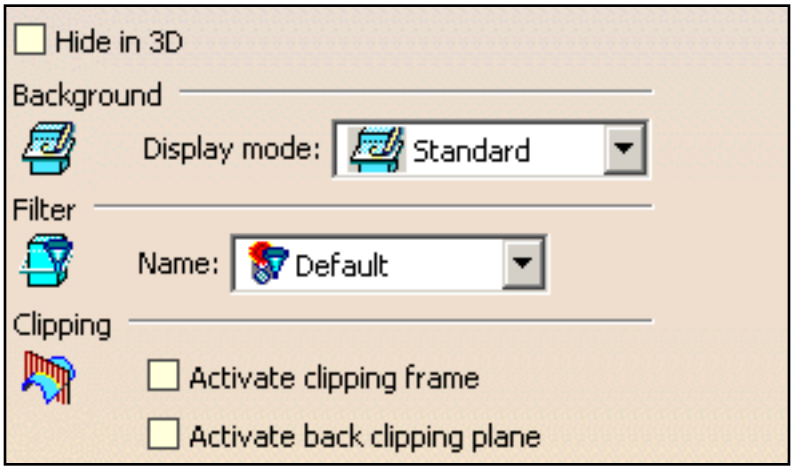
Allows you to modify the name of the view (or 2D component reference when pertinent), and to enter a prefix, an ID or a suffix. You can also create a formula for the view name. For more information, refer to the *Knowledge Advisor User's Guide*.

Generation mode

These properties are not available in 2D Layout for 3D Design.

4. Click the **Visualization** tab. It contains a number of properties:

- Hide in 3D
- Background
- Filter
- Clipping



Hide in 3D

Select this check box if you do not want any element on the sheet (views, geometry, annotations, and so on) to be visible in the view's 3D background and in a 3D workbench (such as Part Design).

In this case, sheet elements will not be visible in Show or No Show space.

Background

Display Mode

Select a display mode to specify how the view's 2D and 3D backgrounds should be handled:

- **Standard**: shows both the 2D and 3D backgrounds.
- **Invisible**: hides both the 2D background (the 3D representation of 2D elements which do not belong to the current view, but to other views) and the 3D background (the representation of all 3D elements, including edges, faces and 3D wireframe).
- **Unpickable**: prevents selecting all elements in both the 2D and 3D backgrounds, even though you can see them. You can just handle 2D elements which belong to the current view.
- **Low-intensified**: dims all elements in both the 2D and 3D backgrounds.
- **Unpickable low-intensified**: dims all elements in both the 2D and 3D backgrounds. Additionally, although you can see these elements, you cannot select them. You can just handle 2D elements in the current view.

Refer to Managing the Layout View Background for more information.

Filter

Name

Select an existing filter to apply to the view. Refer to Creating View Filters for more information.

Clipping

Activate clipping frame

Check this box to clip the 3D background of the view using a frame. Refer to Using the Clipping Frame for more information.

Activate back-clipping plane

Check this box to clip the 3D background of the view using a back-clipping plane. Refer to Using the Back-Clipping Plane for more information.

5. Click the **Graphic** tab. It contains a number of properties:

Shown

This property is not available in 2D Layout for 3D Design.

Pickable

Select this box if you want the view to always be selectable.

Layers

This property is not available in 2D Layout for 3D Design.

Rendering Style

This property is not available in 2D Layout for 3D Design.

LowInt

Select this box to apply the low-intensity color to the selected view.

Refer to Displaying and Editing Graphic Properties in the *Infrastructure User's Guide* for more information about these properties.

6. Change the view properties as wanted, and then click **OK** to validate.

Editing 2D Geometry Feature Properties

This task shows you how to access and edit information on 2D geometry features (name and stamp).

Open the Brackets_views03.CATDrawing document.

1. Select a 2D element on the CATDrawing you opened.
2. Select **Edit->Properties** and click the **Feature Properties** tab.

You can also right click the 2D element and then select **Properties** from the displayed contextual menu.

3. If needed, click the **More** button.
4. Enter a new name for the element in the field. The information displayed concerns the creation of the elements.
5. Click the **Graphic** Tab. A number of properties are available. For more information, refer to Editing 2D Element Graphic Properties.
6. Click **OK** to validate and exit.

Editing Graphic Properties

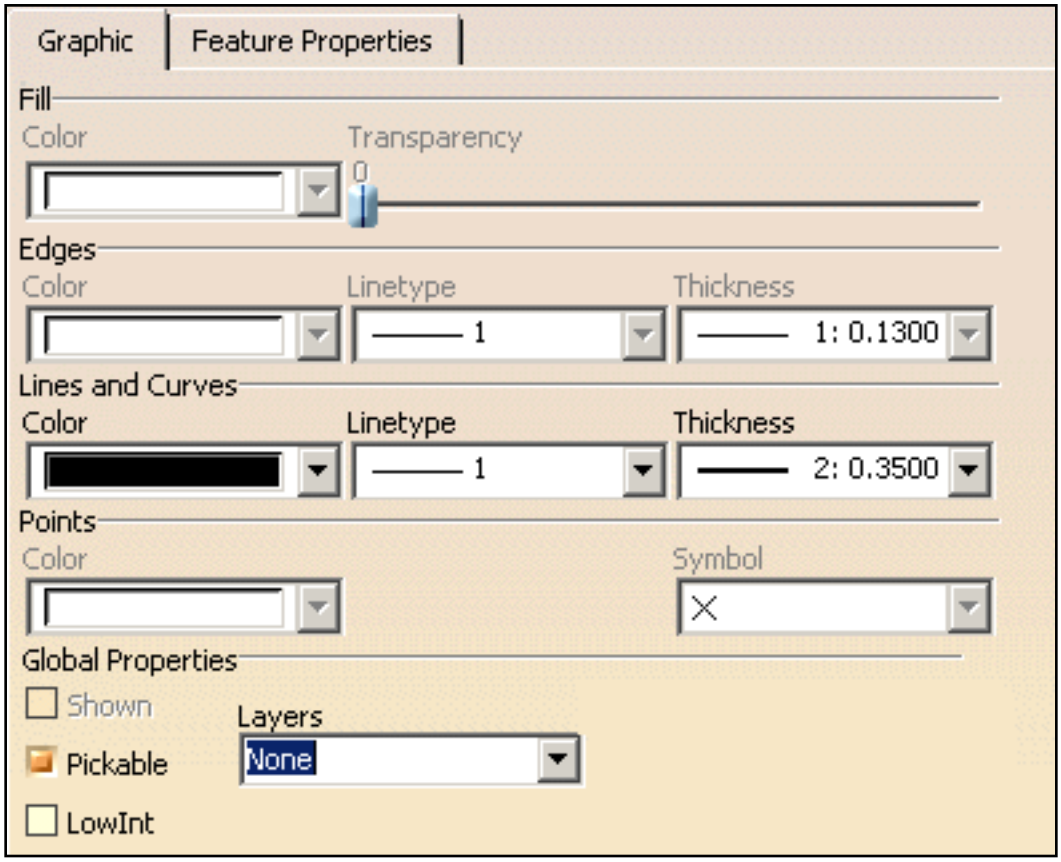
This task explains how to access and edit graphic properties for elements, such as geometry, annotations, etc.

Open the Brackets_views03.CATDrawing document. For the purpose of this scenario, you will be editing the graphic properties of a 2D geometrical element.

1. Select a 2D element on the CATDrawing you opened.
2. Select **Edit -> Properties** and click the **Graphic** tab.

You can also right click the 2D element and then select **Properties** from the displayed contextual menu.

3. If needed, click the **More** button.
4. If needed, modify the available properties. Depending on the element you selected, not all properties will be available.



- **Fill**:
  - you can color the selected element and set the filling transparency.

- **Edges**:
  - you can define the color, linetype (dotted, dashed, etc.) and thickness that will be used for edges. See Graphic Properties Toolbar.

- **Lines and Curves**:
  - you can define the color, linetype (dotted, dashed, etc.) and thickness that will be used for lines and curves. See Graphic Properties Toolbar.

- **Points**:
  - you can define the color and the symbol that will be used for points.

- **Global Properties**:
  - you can choose if the element will be shown or not (check/uncheck **Shown** option)
  - you can activate or deactivate **Pickable** mode. If you uncheck it, geometry will not be selectable anymore. See Pick/No Pick mode.
  - you can choose to display the selected element using a lower intensity.
  - you can choose a layer for the selected geometry.

In some cases, changing the color of an element provides unexpected results, as the color of related elements will be changed as well.
For example, if you change the color of a table frame, the font color of the table's text will also be changed. In this case, a workaround is to change the font color after having edited the graphic properties.

5. Click **OK**.

For more information on graphic properties, refer to the *Infrastructure User's guide*.

Pick/No Pick mode

When you create elements using the No Pick mode (**Pickable** option unchecked),

- If you want to make one or several elements pickable back again, perform as follows:

  1. Select **Edit -> Search** from the menu bar and select the element(s) to be modified from the Search dialog box.
  2. Select **Edit -> Properties** from the menu bar and check the **Pickable** option from the Properties dialog box.

- If you want to make all the elements on a sheet or in a view pickable back again, perform as follows:

  1. Click the sheet or the view(s) to be applied the Pick mode from the specification tree.
  2. Select **Force Pick Mode** from the contextual menu.

Graphic Properties Toolbar

You can also modify graphic properties using the **Graphic Properties** toolbar.

The **Graphic Properties** toolbar lets you modify the following graphical options:

- the line color
- the line thickness
- the linetype
- the symbol to be used for points
- a layer for the selected geometry
- copying objects (**Copy Object Format** icon)
- the pattern (**Pattern** icon). This option display the Pattern Chooser dialog box, from which you can select a pattern.

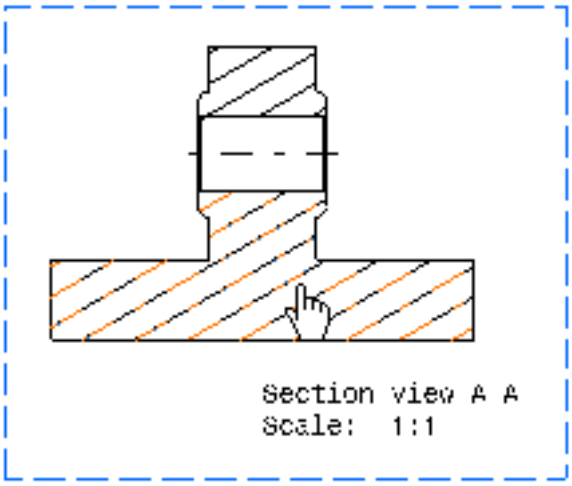
Care when you assign graphic attributes to a line (for example, make it thick and red).
When you turn this red thick line into a construction line (from the contextual menu: Object.Line -> Definition..., Construction line option in the Line Definition dialog box), the line will become a dotted gray line. Even though you then decide to make it a standard line back again (by un-checking the Construction line option), the line will have lost its "red" and "thickness" attributes and will be assigned its original attributes.

Editing Pattern Properties

This task explains how to access and edit pattern properties. Patterns are used for area fills or, in a Generative Drafting context, when cutting through material in section views/cuts or breakout views, for example.

Open the GenDrafting_Edit_Pattern_Properties.CATDrawing document.

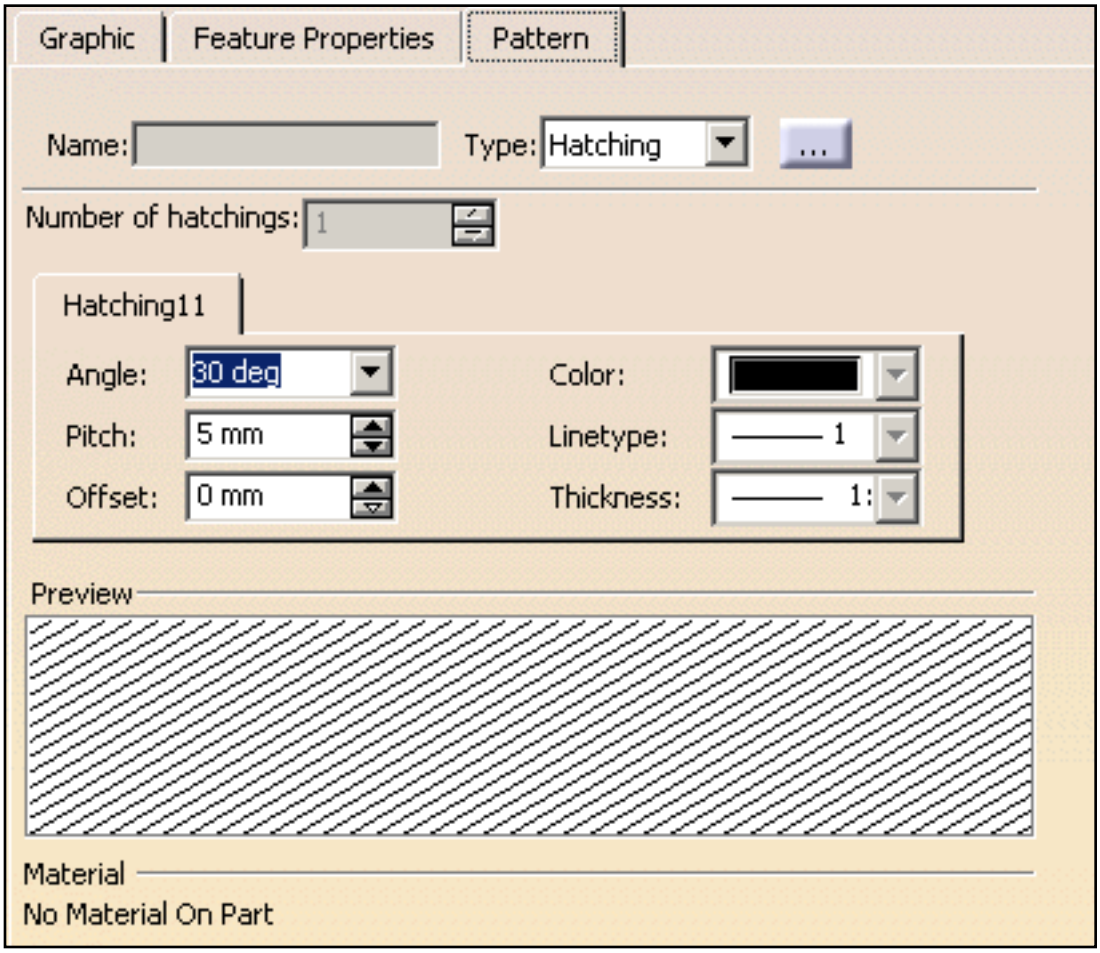
1. Select the pattern be modified. For the purpose of our scenario, select the hatching pattern in the Section view.



2. Select **Edit-> Properties**.

   You can also right-click the pattern and then select **Properties** from the displayed contextual menu.

3. In the Properties dialog box that appears, click the **Pattern** tab.



4. To define your own pattern, choose a pattern type from the **Type** drop-down list:

   - Hatching
   - Dotting
   - Coloring
   - Image

   If you want to choose from the various patterns available, you can also click the **[...]** button. This will display the pattern chooser, from which you can make your selection.

5. Select your options as required.

   - The options available in the dialog box depend on the type of pattern you selected, as well as on the standard used by the drawing.
   - When editing the properties of a pattern associated with a part material, the software offers its own selection of patterns, and not the patterns defined in the standard.

Hatching

- **Number of hatchings**: Defines the number of different hatchings to use in this pattern. A tab will be created for each hatching, to let you define each one individually.
  This option is unavailable with the current drawing standard.
- **Angle**: For each hatching this pattern, specifies the angle value in degrees.
- **Pitch**: For each hatching in this pattern, specifies the pitch in millimeters.
- **Offset**: For each hatching in this pattern, specifies the offset in millimeters.
- **Color**: For each hatching in this pattern, specifies the color.
  This option is unavailable with the current drawing standard.
- **Linetype**: For each hatching in this pattern, specifies the linetype.
  This option is unavailable with the current drawing standard.
- **Thickness**: For each hatching in this pattern, specifies the linetype thickness.
  This option is unavailable with the current drawing standard.
- **Preview**: Lets you preview the resulting hatching pattern.

The **Color**, **Linetype** and **Thickness** options can be modified, provided the **EditAvailability** parameter is set to Yes under the Pattern node in the Standards editor. For more information, refer to Pattern Definition.

Dotting

- **Pitch**: Specifies the dotting pitch in millimeters.
- **Color**: Specifies the dotting color.
- **Zigzag**: Specifies whether dotting should zigzag.
- **Preview**: Lets you preview the resulting dotting pattern.

Coloring

- **Color**: Specifies the color.
- **Preview**: Lets you preview the resulting coloring pattern.

Image

- **Browse** button: Lets you select the image to use for this pattern.
  This option is unavailable with the current drawing standard. You can only use the images defined by the administrator. These images are available from the pattern chooser (click the **[...]** button).
- **Angle**: Specifies the angle value in degrees.
- **Scale**: Specifies the scale.
- **Preview**: Lets you preview the original image (not the result after modifying the angle and scale).

6. Click **OK** to validate and exit.

- You can also modify pattern properties using the **Pattern** icon on the **Graphic Properties** toolbar. This option displays the Pattern Chooser dialog box, from which you can select a pattern.
- Patterns are defined in the standards. For more information, refer to Pattern Definition.

Editing Annotation Font Properties

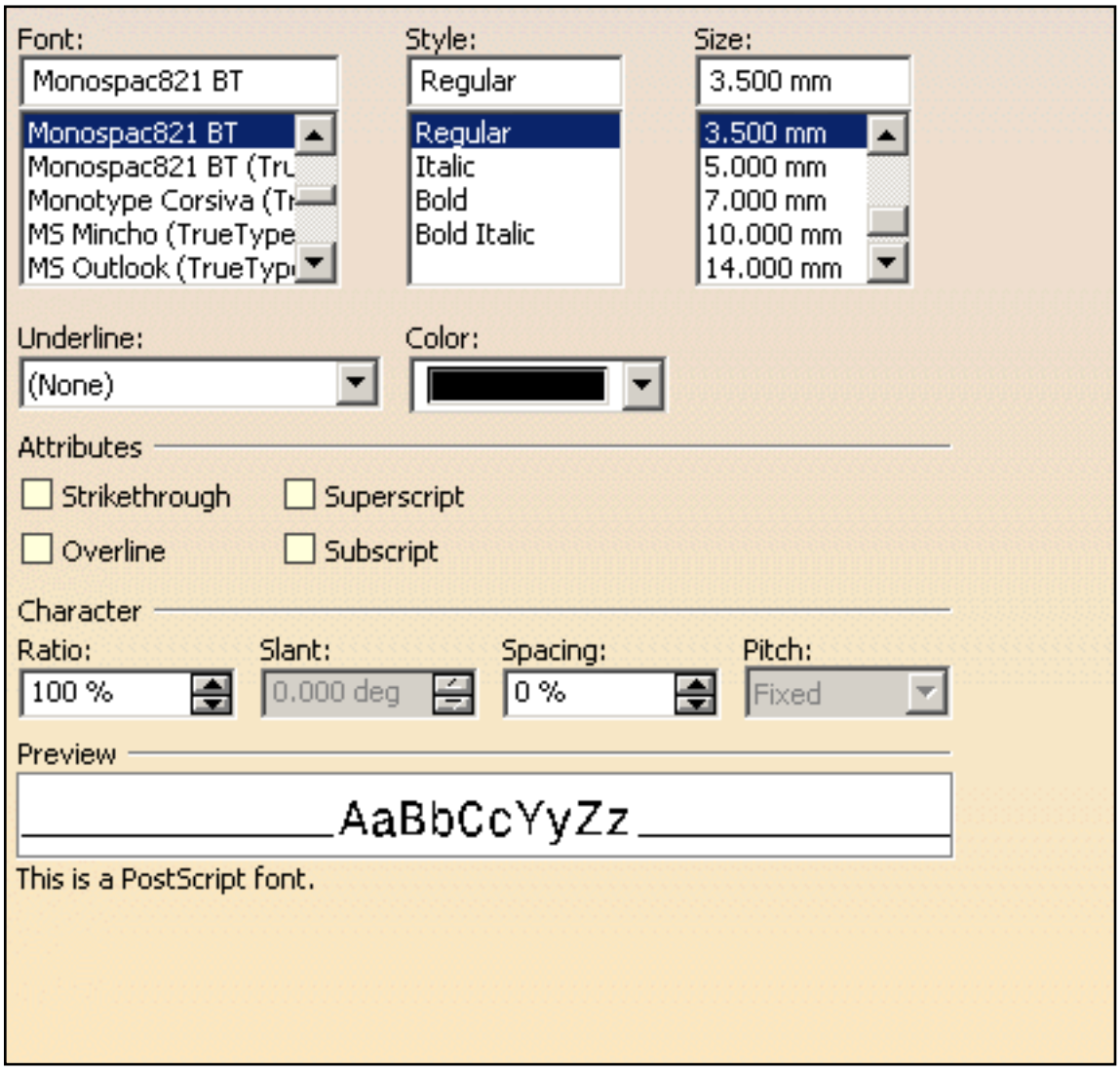
This task explains how to access and edit annotation font properties.

Open the Brackets_views03.CATDrawing document. Create a free text, for example.

1. Double-click the text to switch it to edit mode.
2. Select the whole text (you can also select only part of the text) and then select **Edit -> Properties**.

You can also right-click the selected text and then choose **Properties** from the contextual menu.

3. In the **Properties** dialog box that appears, click the **Font** tab. The associated panel is displayed.



- **Font**, **Style**, **Size, Underline** and **Color**: choose the font, size, style and color of the text, and underline it.
- **Attributes**: draw a line through (**Strikethrough**) or above (**Overline**) the selected text, and make it superscript or subscript.

You can either underline or overline a text, but you cannot do both.

When you are using a font stroke for annotations, the character's thickness is set to 1 for regular style and 3 for bold style.

You can customize standard files in order to remove this parameter from the thickness' combo box so that it cannot be applied to annotations' characters.

- **Character**:
  - **Ratio**: modify character width.
  - **Slant**: modify character slant (for italic text, slant=15 deg).
  - **Spacing**: change the spacing between characters.
  - **Pitch**: set a fixed or a variable pitch. As an example, create the free text "Tools" and apply the font ROM1.

Fixed Pitch

Variable Pitch

The **Slant** and **Pitch** options are available only for stroke fonts.

The pitch of some stroke fonts cannot be modified. In that case, the **Pitch** combo list is disabled.

In case you use characters in some fonts that have no or very little spacing (i.e. i or l), you should not set the spacing to 0 mm, otherwise they would look as if they are superimposed and only one character would seem to be displayed in your annotation.

Clicking the More button displays extra options, if any are available.

4. Modify the available options as required.

5. Click **OK** to validate and exit the dialog box.

For more information on font properties, refer to the *Infrastructure User's Guide*.

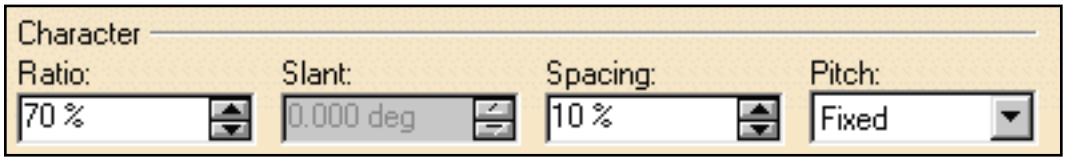
Changing Character Ratio and Spacing

In this task, you will learn how to change the character ratio and spacing of a portion of text, but it is also possible to change these for a whole text.

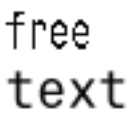
Create a free text.

1. Double-click the text to switch it to edit mode.
2. Select a portion of text and right-click it.

3. Click **Properties** in the menu that appears. The **Properties** dialog box appears.
4. Click the **Font** tab.
5. In the Character area, increase or decrease the value in the **Ratio** field to change the character ratio.



6. Modify the value in the **Spacing** field to change the character spacing.
7. Click **OK** to validate your changes. The text is updated.



Making Text Superscript or Subscript

In this task, you will learn how to make a text superscript, how to make a text subscript, and how to specify their position.

Create a free text.

1. Double-click the text to switch it to edit mode.

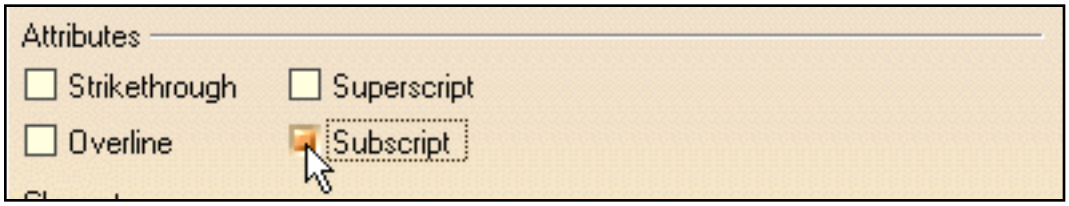
2. Type a text, "subscript" for example, after the text you created previously.

3. Select the piece of text you just typed and right-click it.

4. Click **Properties** in the menu that appears. The **Properties** dialog box appears.

5. Click the **Font** tab.

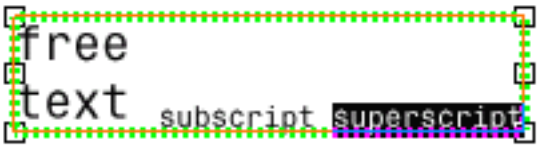
6. In the Attributes area, select the **Subscript** check box.



7. Click OK to validate your changes. The selected text is made subscript.

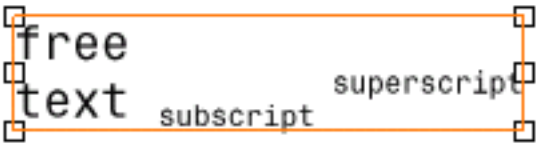
8. Now type another text, "superscript" for example, after the existing text. For the moment, the new text takes on the properties of the subscript text in front of it.

9. Select the piece of text you just typed and right-click it.



10. Repeat steps 4 and 5.

11. In the Attributes area, select the **Superscript** check box (instead of **Subscript**) and click OK. The selected text is made superscript.



12. For the purpose of this exercise, you will now align the subscript and superscript texts and set their offset and size. To do this, select the whole text and right-click it.

The offset defines the vertical position of the superscript or subscript text from the baseline of the text. The size defines the height of the superscript or subscript text. Both values are expressed as a percentage of the font size.

13. Click **Properties** in the menu that appears.

14. In the Properties dialog box, click the **Text** tab.

15. In the Options area, select the **Back Field** check box to align the texts.

16. Increase or decrease the values for the superscript and subscript texts in the **Offset** and **Size** fields to set the offset and size.

17. Click OK to validate. The subscript and superscript texts are now aligned and set as defined.

This functionality does not always work when the text is wrapped.

Editing Text Properties

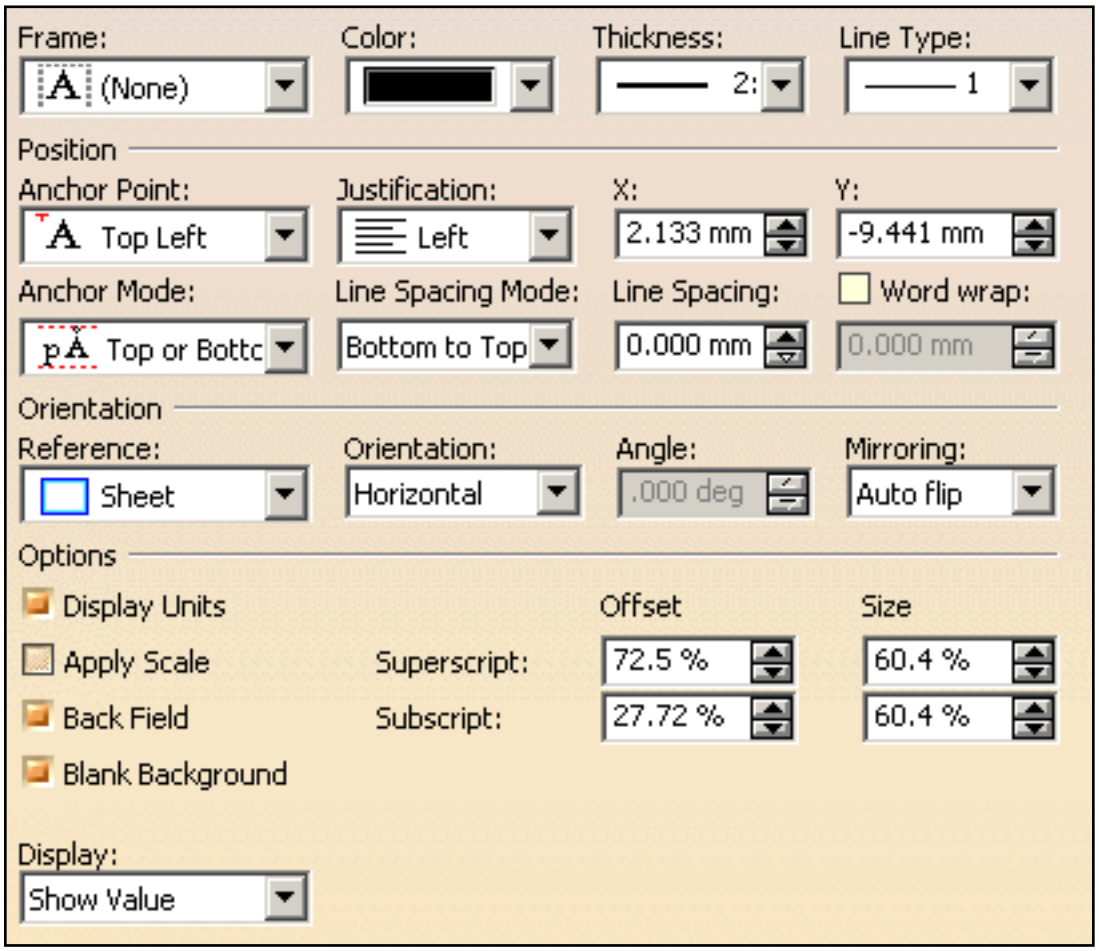
This task explains how to access and modify text color, position and/or orientation. You will also learn how to specify the text display mode.

Open the Brackets_views02.CATDrawing document. Create an annotation such as a free text, for example.

1. Select the annotation you just created. (For the purpose of this exercise, you select a free text, but you could also select any other type of annotation.)
2. Select **Edit-> Properties**.

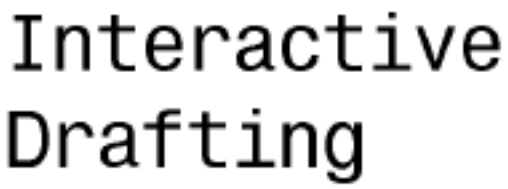
You can also right-click on this dimension and then choose **Properties** from the contextual menu.

3. Click the **Text** tab. The associated panel is displayed.

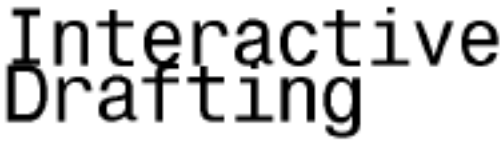


- **Frame**: you can choose a frame type for the selected text that is to say rectangle, triangle, circle, etc. You can specify the color, line thickness and line type for the frame in the associated fields.
- **Position**:
  - **Anchor Point**: you can change the text position in relation to the anchor point.
  - **Justification**: you can specify a justification for the text: left, center or right.
  - **X**, **Y**: you can modify anchor point coordinates.
  - **Anchor Mode**: it allows you to position the anchor line to the character Top and Bottom or to the character Cap or Base.

- **Line Spacing Mode**: you can choose the spacing mode between to line of characters. As an example, create the following free text:



Now, select **base to cap** option in the combo box. The spacing between the two lines will be between the base of first line characters and cap of second line characters:



- **Line spacing**: you can increase or decrease the spacing between two lines of characters.
- **Word wrap**: allows you to wrap the text in a width you specify.

When you create a free text, the anchor point is the point you click in the free space to define a location for the free text.

- **Orientation**: specify a text orientation.
  - **Reference**: choose **Sheet** to use the sheet as the reference for the text orientation, or **View / 2D Component** to use the view or 2D component as the reference for the text orientation.
  - **Orientation**: the text is oriented according to the chosen reference; choose **Horizontal** to position it horizontally, **Vertical** to position it vertically or **Fixed Angle** to position it using a fixed angle.
  - **Angle**: if you choose **Fixed Angle** for **Orientation**, you can define the orientation angle according to the chosen reference.
  - **Mirroring**: specify whether you want to mirror the selected text, and what kind of mirroring, or if the text should flip automatically in such a way that it will always be in a readable position.

- **Options**:
  - **Display Units**: in a text containing parameters with units, displays these units.
  - **Apply scale**: applies the scale of the view or of the 2D reference component to the display of the text or to the value of a dimension.

    If you want to use as symbols 2D components with text, activate both the **Apply Scale** property and the **Create with a constant size** setting (in **Tools -> Options -> Mechanical Design -> Drafting -> Annotation and Dress-up** tab): the size of both the 2D component and its text will then be independent from the view scale.

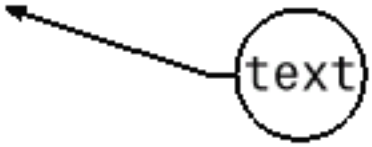
  - **Back Field**: aligns superscript and subscript texts above one another.
  - **Blank Background**: specifies that the text background should be blanked when the text is displayed over a pattern or over a picture.
  - **Superscript**: increase or decrease the values for the superscript texts.
    The **Offset** parameter specifies the distance of the superscript text from the base line according to the font size of the text.
    The **Size** parameter specifies the size of the superscript text according to the font size of the text.
  - **Subscript**: increase or decrease the values for the subscript texts.
    The **Offset** parameter specifies the distance of the subscript text from the base line according to the font size of the text.
    The **Size** parameter specifies the size of the subscript text according to the font size of the text.
  - **Display**: specifies a display mode for the text: **Show Value, Show Box** or **Hide Value**. Refer to Specifying the Text Display Mode below for more details.

4. Modify the available options as required.

5. Click **OK** to validate and exit the dialog box.

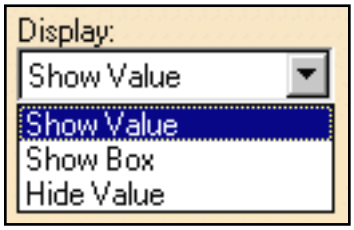
Specifying the Text Display Mode

In this task, you will learn how to specify the display mode for the text. For the purpose of this exercise, you will use a text with a leader and a frame, but this feature is also available with text only, as well as with dimension texts.
Create a text with a leader and a frame.

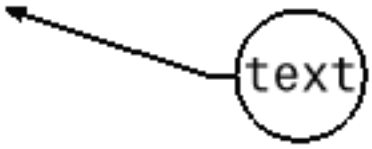


1. Select the text and right-click it.
2. Click **Properties** in the menu that appears. The Properties dialog box appears.
3. Click the **Text** tab.
4. In the Options area, choose the display mode you want for your text from the **Display** list.

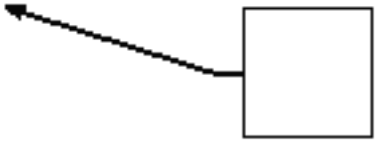
You have the following options:



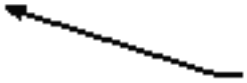
- **Show Value**: displays the text, and (when applicable) its leader and its frame. This option is selected by default.



- **Show Box**: replaces the text and (when applicable) its frame by a rectangular box and displays its leader.



- **Hide Value**: hides the text and (when applicable) its frame but (when applicable) displays its leader.

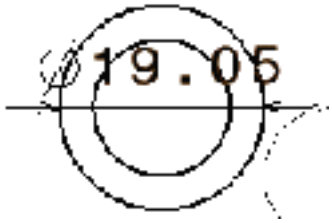


5. Click OK to validate. The text is now displayed using the mode you set.

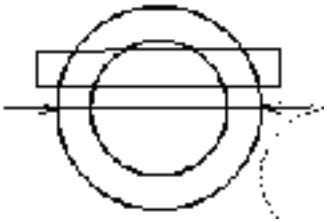
If you select Hide Value as the display mode for a text with no leader, the text will not be visible at all on your drawing. You can find all hidden texts in a drawing using advanced Search options. To do this, choose **Edit -> Search**, click the **Advanced** tab. Select **Drafting** from the **Workbench** list, **Text** from the **Type** list, **Display** from the **Attributes** list. In the dialog box that appears, select **=** and **Hide Value** and then click **OK**. Click the **Search** icon. All hidden texts are listed.

In the case of dimensions, the display modes are as shown below:

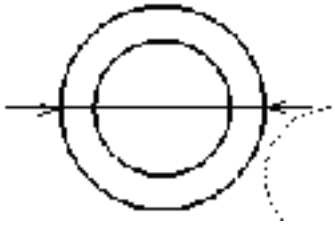
- **Show Value**: displays the dimension and its leader. This option is selected by default.



- **Show Box**: replaces the dimension by a rectangular box and displays its leader.



- **Hide Value**: hides the dimension but displays its leader.



Editing Picture Properties

This task explains how to access and edit raster or vector picture properties.

- Define a new sheet.
- Save the logo.tif document on your computer (to do this, right-click on "logo.tif" and choose Save Target As in the contextual menu).

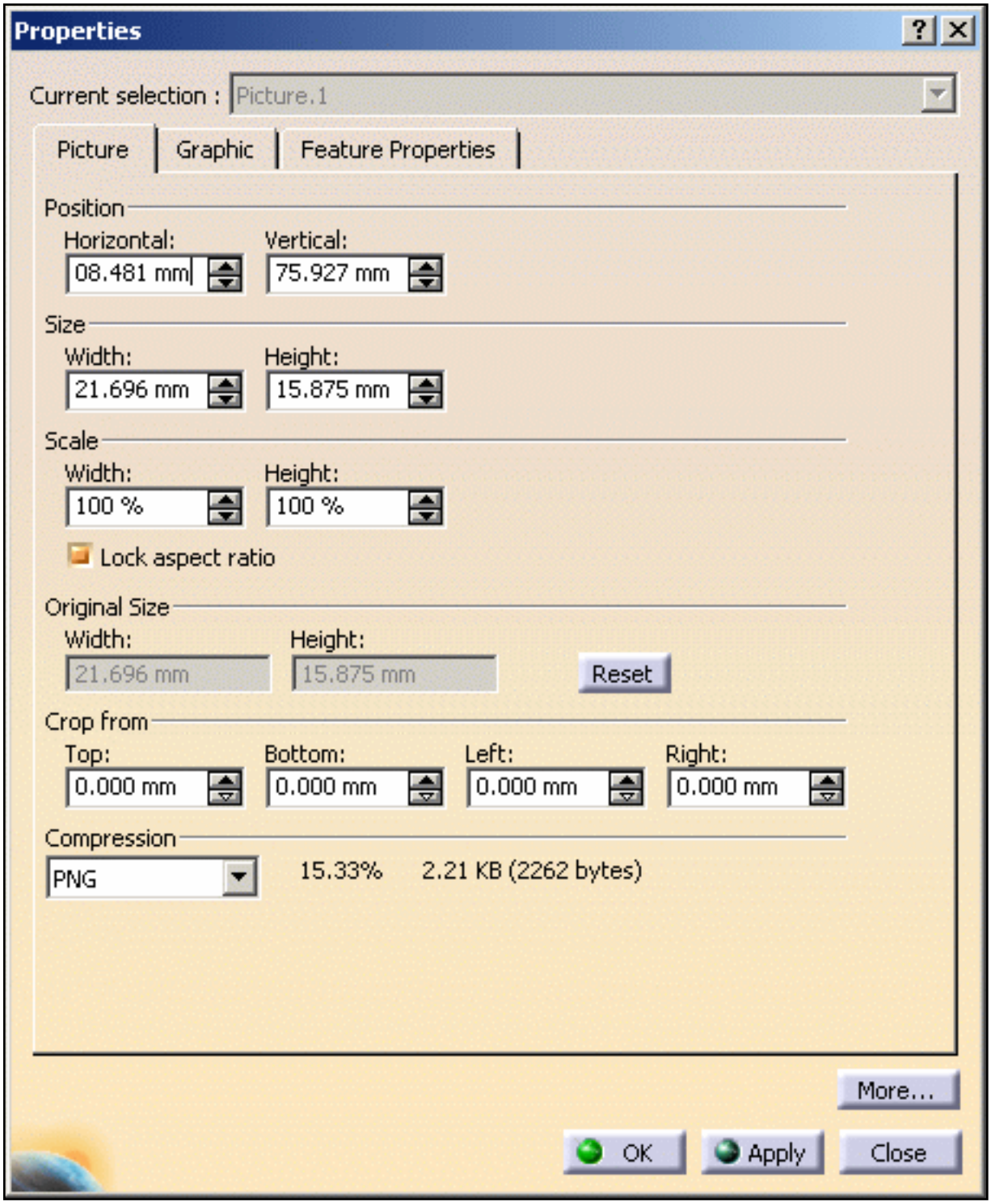
1. Select the **Insert -> Picture** command.

   A dialog box appears, allowing you to browse your disk.

2. Select the file "logo.tif" you have previously imported. The image is imported in your drawing.

3. Right-click on the picture and select Properties in the contextual menu.

   The Picture tab of the Properties dialog box is displayed.



You can modify the properties of the picture and click on **Apply** to preview the changes.

Position

- **Horizontal**: defines the horizontal positioning on the sheet, taking the bottom left corner of the image into account.
- **Vertical**: defines the vertical positioning on the sheet, taking the bottom left corner of the image into account.

Size

- **Width**: defines the width in millimeters of the picture.
- **Height**: defines the height in millimeters of the picture.

When you modify the size, the scale is automatically modified at the same time.

Scale

- **Width**: defines the width in percentage of the picture.
- **Height**: defines the height in percentage of the picture.
- **Lock aspect ratio:** enables to keep the ratio between width and height. If it is not checked, the picture might look distorted when modifying only one feature.

Original Size

- **Width**: defines the original width of the picture in millimeters.
- **Height**: defines the original height of the picture in millimeters.
- **Reset**: resets the original size of the picture if one feature was modified.

Crop from

- **Top**: defines the top part of the picture to be hidden.
- **Bottom**: defines the bottom part of the picture to be hidden.
- **Left**: defines the left part of the picture to be hidden.
- **Right**: defines the right part of the picture to be hidden.

Compression

You can compress a picture to a given format, available in the combo box.

- **None**: the picture keeps its original size.
- **PNG**: the picture is compressed without loss.

The PNG format is used by default when inserting a picture whatever its original format.

- **JPEG**: the picture is compressed in JPEG format, that is with loss.
- Picture compression properties: you are given the compression rate in percentage and the size of the picture in bytes according to the chosen format once it is saved on your hard drive. The compression rate and picture size vary according to your picture.

  In our example, the compression properties for the logo picture are as follows:

Format/Properties	Compression rate	Picture size
None	100.00%	14.44 KB (14783 bytes)
PNG	15.33%	2.21 KB (2262 bytes)
JPEG	15.09%	2.18 KB (2228 bytes)

In case you insert a monochrome picture, it is automatically set to the CCITTG3 format, which is a unique format without loss.

Should you need to edit the picture, refer to the Editing Raster Images section.

Editing Dimension Text Properties

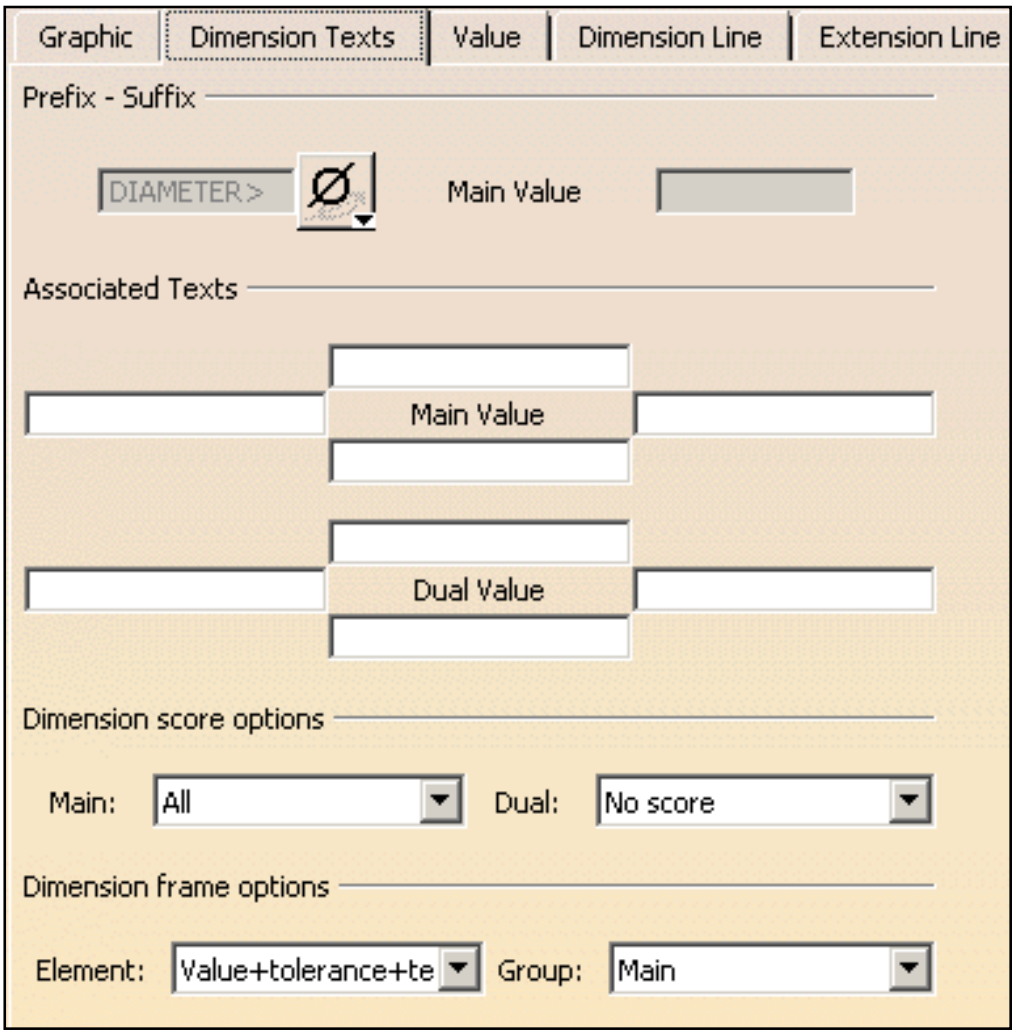
This task explains how to access and, if needed, edit dimension text properties.

Open the Brackets_views02.CATDrawing document. Create a diameter dimension, for example.

1. Select a dimension (whatever the type) on the CATDrawing you opened.
2. Select **Edit -> Properties** and click the **Dimension Texts** tab.

You can also right click the current element and then select the Properties command from the displayed contextual menu.

3. If needed, modify the available options.



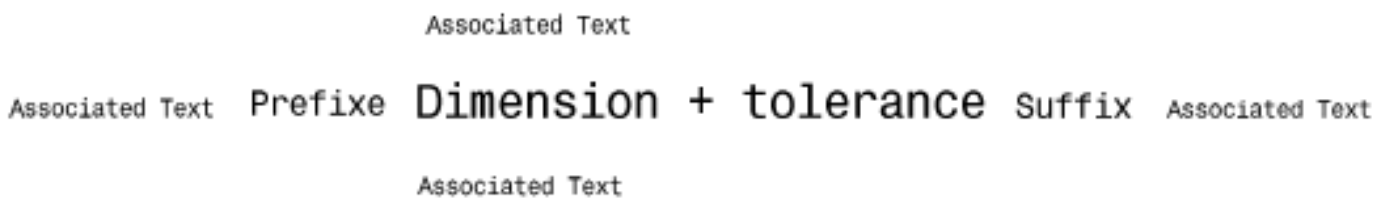
- **Prefix - Suffix**: you can insert either a symbol or a text before the dimension text or a text after the dimension text.

You cannot insert a prefix and a suffix.

If you want to remove the symbol before the dimension text, click the **Insert Symbol** icon and, from the list of symbols that appears, select this symbol:

- **Associated Texts**: you can insert texts before, after, below and above the main and the dual value.

Dimension texts positioning:



- **Dimension score options**: you can choose to score only the value, all dimension texts or not to score (for Main Value and/or Dual Value).
- **Dimension frame options**: you can choose to include in the frame Value+tolerance+texts or Value+tolerance or Value for Main Value, Dual Value or both.

5. Click **OK** to validate and exit the dialog box.

Editing Dimension Font Properties

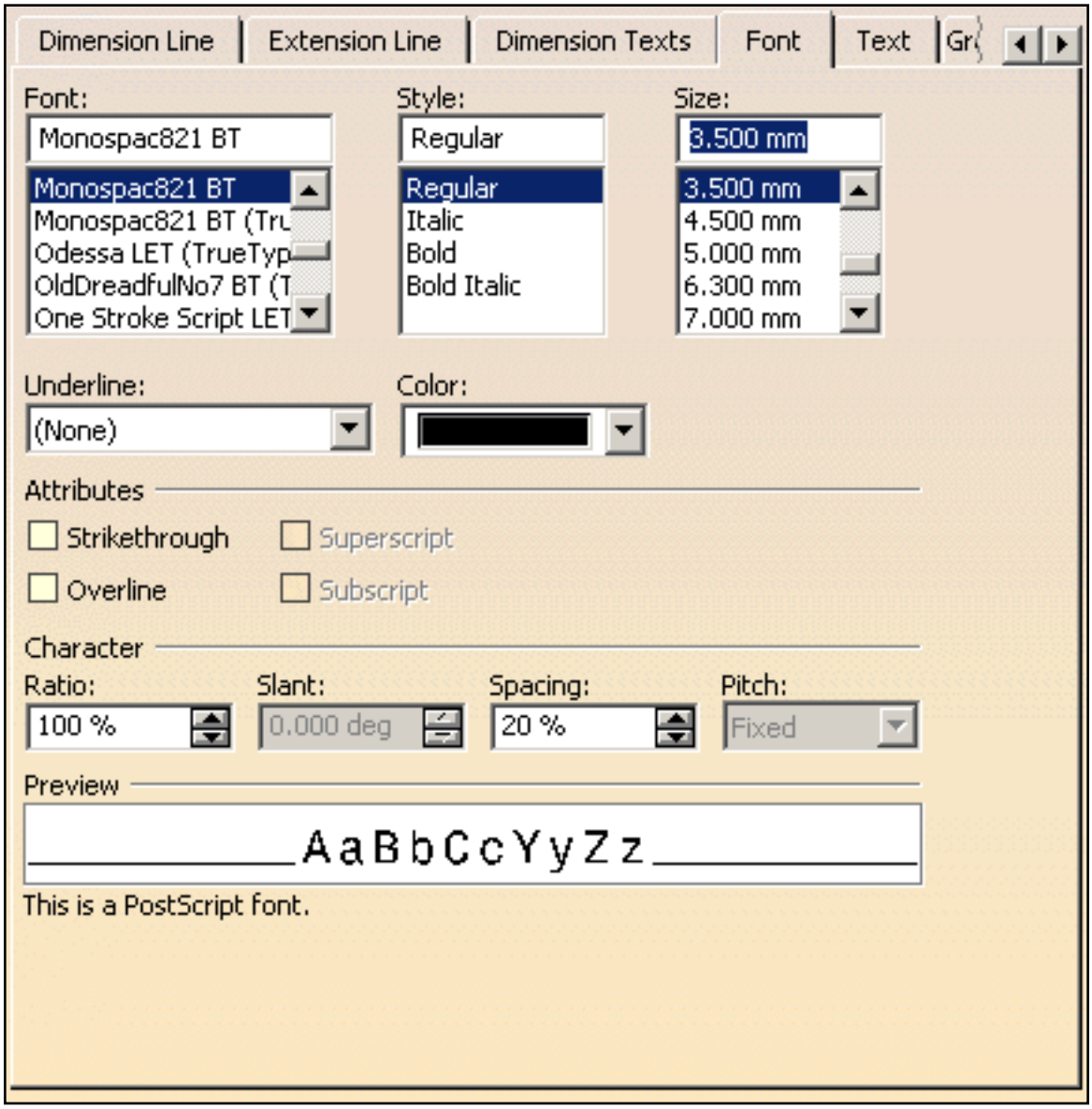
This task explains how to access and edit dimension font properties.

Open the Brackets_views03.CATDrawing document. Create a dimension (of whatever type).

1. Select the dimension.
2. Select **Edit -> Properties**.

You can also right-click the dimension and then select **Properties** from the displayed contextual menu.

3. In the **Properties** dialog box that appears, click the **Font** tab. The associated panel is displayed.



- **Font**: choose the font.
- **Style**: choose the font style.
- **Size**: choose the font size.
- **Underline**: underline the dimension text.
- **Color**: choose the font color.
- **Strikethrough**: draw a line through the dimension text.
- **Overline**: draw a line above the dimension text.

You can either underline or overline a text, but you cannot do both.

- **Ratio**: modify the character width.
- **Spacing**: change the spacing between characters.

4. Modify the available properties as required.

5. Click **OK** to validate and exit the dialog box.

For more information on font properties, refer to the *Infrastructure User's Guide*.

Editing Dimension Value Properties

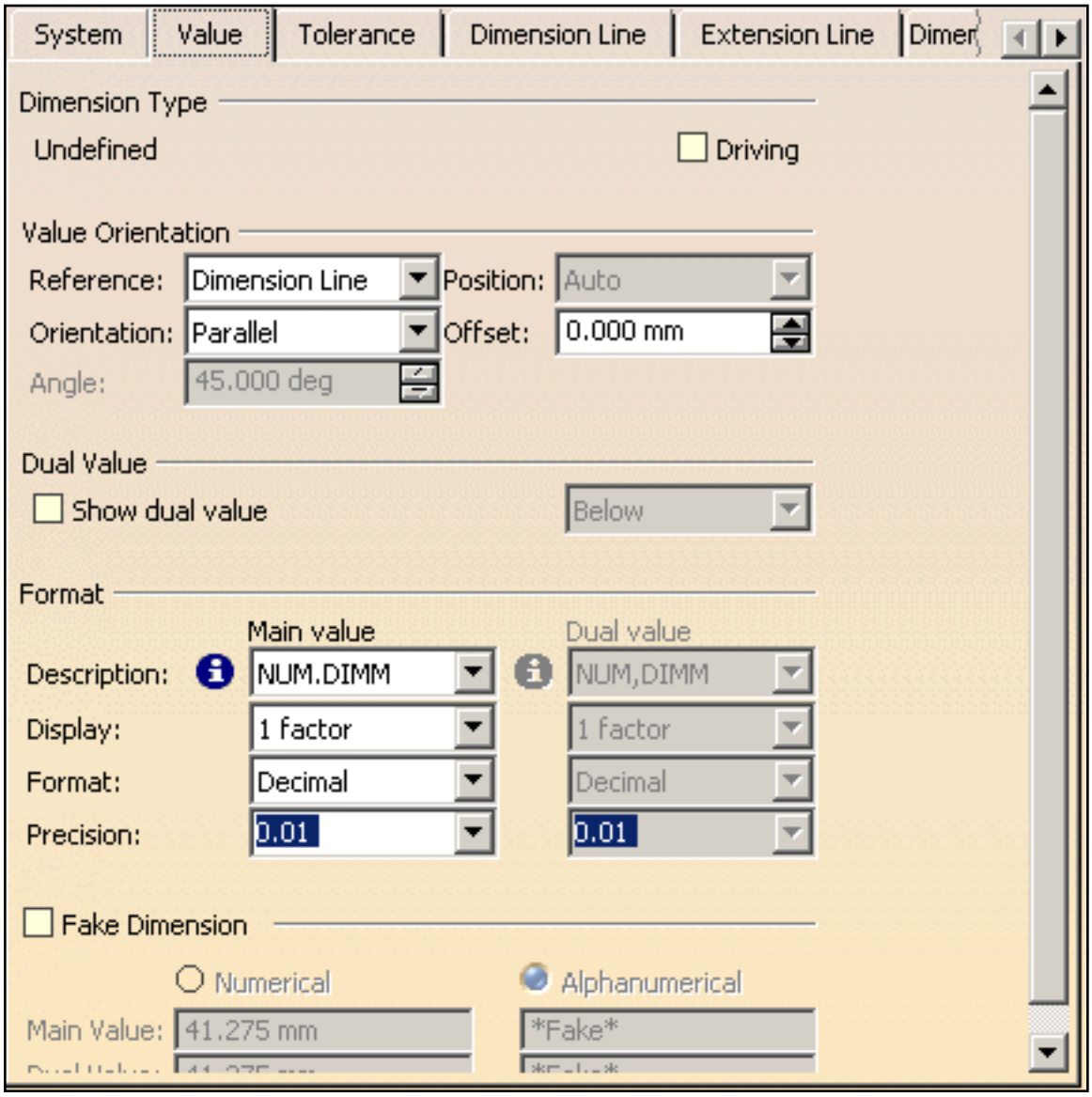
This task explains how to access and edit dimension value properties.

Open the Brackets_views03.CATDrawing document. Create a diameter dimension, for example.

1. Select a dimension (whatever the type) on the CATDrawing you opened.
2. Select **Edit -> Properties** and click the **Value** tab.

You can also right-click the dimension and then select **Properties** from the displayed contextual menu.

3. If needed, modify the available options.



Dimension Type: check **Driving** if you want projected dimensions to drive geometry.
If you want to key in a value for the driving dimension, you must close **Properties** dialog box, double-click the dimension in the drawing, check Drive geometry and key in a value.

Value Orientation: you can choose:

- the value orientation reference (**Screen**, **View** or **Dimension Line**),
- the value orientation (**Parallel**, **Perpendicular** or **Fixed Angle**),
- the orientation angle if **Fixed Angle** is selected in orientation,
- the value position (**Auto**, **Inside** or **Outside**),
- the value offset in relation to the dimension line.

Dual Value: you can show dual value by checking **Show dual value** and choosing its location: **Below, Fractional** or **Side-by-Side**.

Format: you can set Main value and Dual value format.

- **Description**: select a type of format.
- **Display**: choose to display one, two or three factors.
- **Format**: choose fractional or decimal format.
- **Precision**: select the value precision.

For chamfers, you can set **Description**, **Display** and **Format** in the chamfer tab.

Fake Dimension: check this option to display fake dimensions, you can choose to display numerical or alphanumerical fake dimensions.
Texts for numerical fake dimensions are restricted to six characters.
If you need to insert a text containing more than six characters, simply use the alphanumerical fake dimension.

5. Click **OK** to validate and exit the dialog box.

Editing Dimension Tolerance Properties

This task explains how to access and edit dimension tolerance properties.

There are different types of tolerances:

- Numerical tolerances

  38.1 +0.150 -0.100

- Alphanumerical tolerances

  38.1 H6

- Combined tolerances (an alphanumerical value and two numerical values): ISOCOMB tolerance.

  38.1 H6 (+0.20 +0.20)

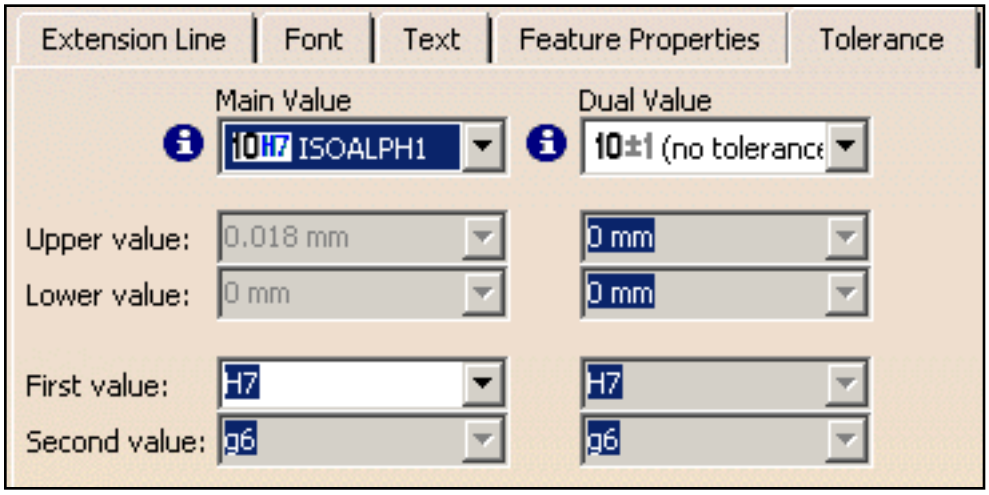
Open the Brackets_views02.CATDrawing document. Create a diameter dimension, for example.

1. Select the diameter dimension.
2. Select **Edit-> Properties** and click the **Tolerance** tab.

You can also right-click the dimension and then select **Properties** from the displayed contextual menu.

3. You can associate a tolerance to the selected dimension. In this example, choose ISOALPH1 in the **Main Value** field.

The **First value** field is enabled and displays an alphanumerical value. The corresponding numerical equivalents are displayed in the **Upper value** and **Lower value** fields. (These equivalents are defined by standards.)



4. Assign the desired tolerance to this dimension by selecting another alphanumerical value. In this example, select H9 in the **First value** field. The corresponding numerical equivalents are automatically displayed.

5. In some cases, you may wish to display another tolerance. In this case, select a tolerance type in the **Dual Value** field.

- If you choose the same tolerance type for main and for dual value, then the values for this tolerance will also be the same.
- For a full description of the tolerance type selected in the **Main Value** and **Dual Value** fields, click the information (i) icon in front of each field.

6. Click **OK** to validate and exit the dialog box.

- For dimensions with alphanumerical tolerances, you can display the corresponding numerical equivalents in the drawing, simply by selecting the dimension and placing the cursor over the tolerance in the drawing. The numerical equivalents are displayed in a tooltip.
- For dimensions with tolerance js and JS, there is no correspondence between the numerical and alpha numerical value. The numerical value displayed is +-0 or the previous numerical value applied to the dimension.

Editing Dimension Extension Line Properties

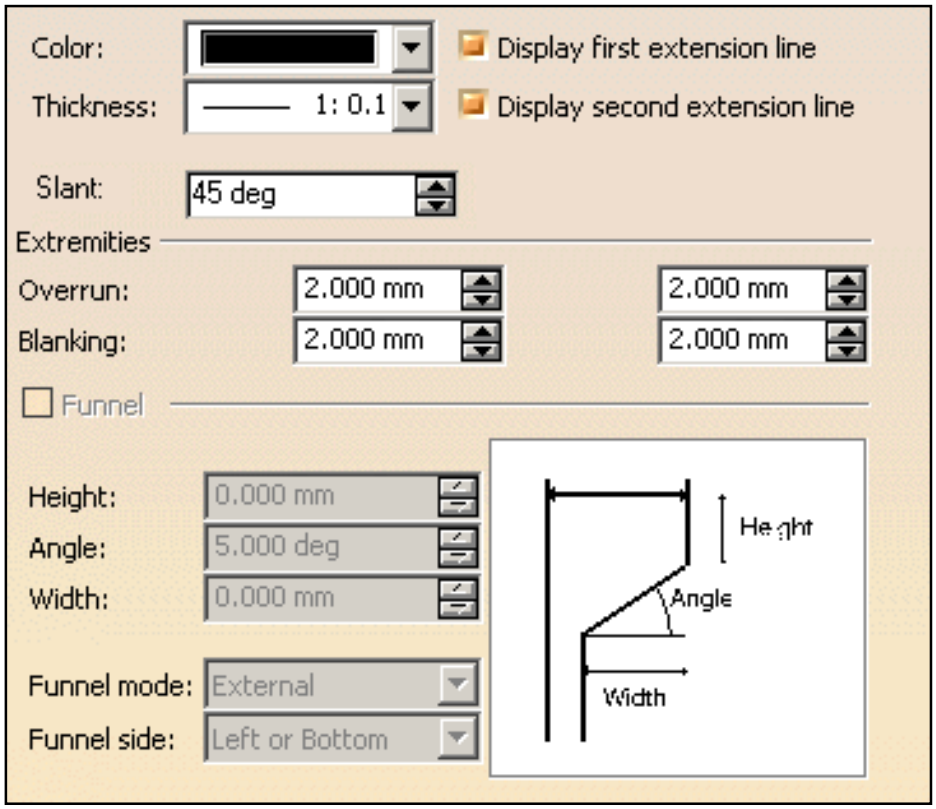
This task explains how to access and, if needed, edit dimension extension line properties.

Open the Brackets_views02.CATDrawing document. Create a diameter dimension, for example.

1. Select the dimension you created (whatever the type).
2. Select **Edit -> Properties**.

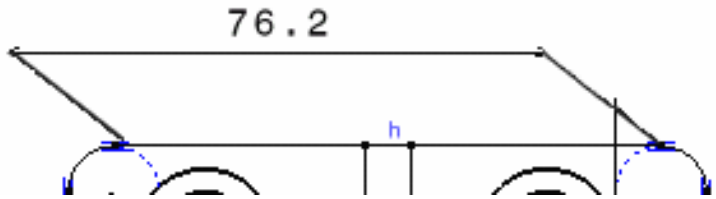
You can also right-click on this dimension and then choose **Properties** from the contextual menu.

3. In the Properties dialog box that appears, click the **Extension Line** tab.



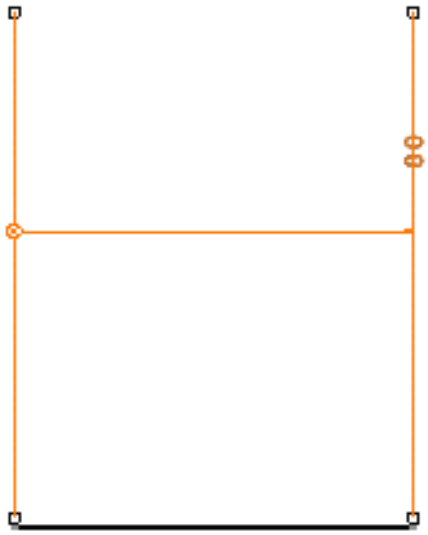
- **Color**: choose a color for the extension line.
- **Thickness**: specify the thickness for the extension line.
- **Display first extension line**: check to display or uncheck to hide the first extension line, when applicable.
- **Display second extension line**: check to display or uncheck to hide the second extension line, when applicable.
- **Slant**: set the slant angle for the extension line. This angle is set between 90 degrees and -90 degrees **excluded**, the default angle being 0 degree.

This functionality works only on linear dimension line and the line linking extension line anchor points (blanking excluded) has to be parallel to the dimension line, as shown below.

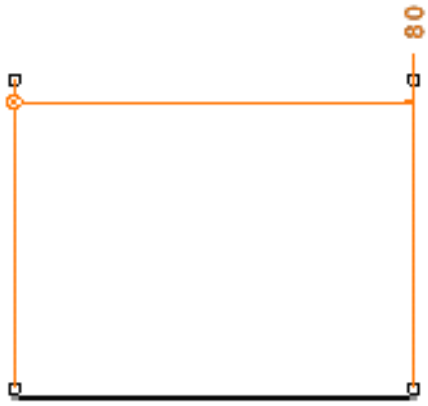


- **Extremities**: it allows you to increase or decrease extension line **Overrun** and **Blanking**.

Overrun is the overrun minimum value. As an example, for a cumulated dimension (for ISO Standard):



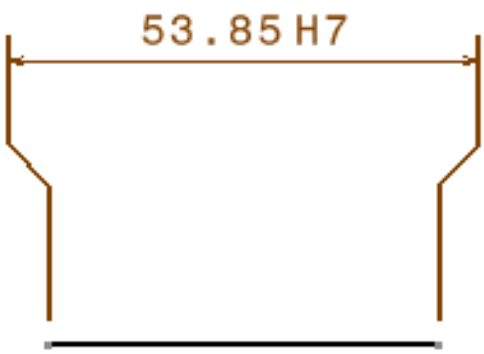
You can increase the overrun size



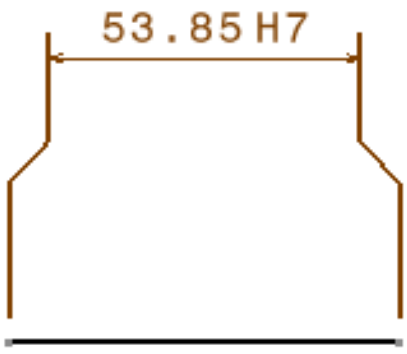
You cannot decrease it below the minimum value

To set extension line length and text position for cumulated dimensions, use the CUMLExtMode dimension parameter in the standards.

- **Funnel**: to insert a funnel, you must check this option. You can configure the funnel:
  - the **Height**,
  - the **Angle**,
  - the **Width**,
  - the **funnel mode**: external or internal



External Funnel Mode



Internal Funnel Mode

- the **Funnel side** allows you to apply a funnel only on one extension line (**Left or Bottom**, **Right or Top**) or both of them (**Both Sides**).

You cannot create interruptions on funneled dimension lines.

4. Modify the available options as required.

5. Click **OK** to validate and exit the dialog box.

Editing Dimension Line Properties

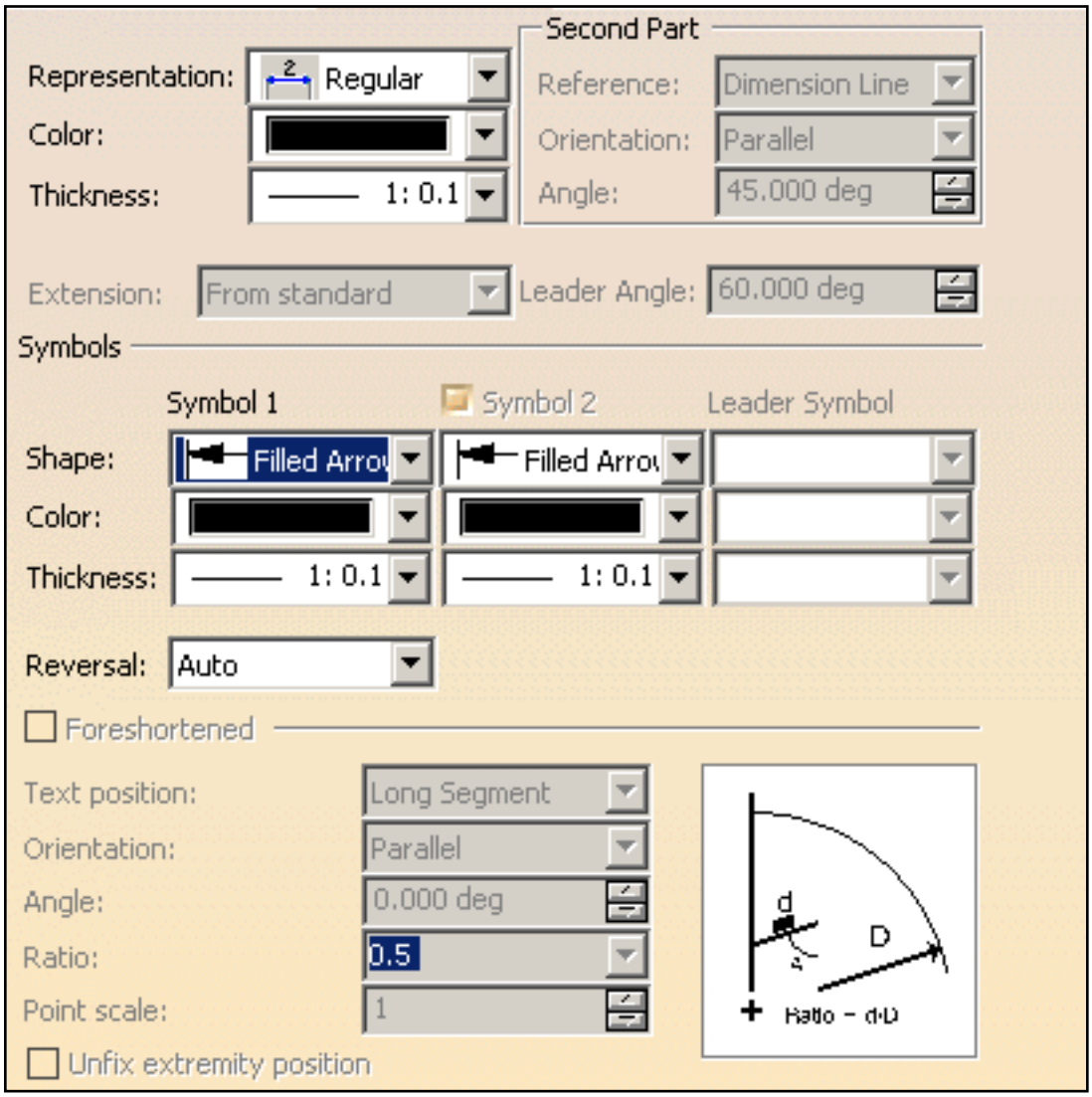
This task explains how to access and, if needed, edit dimension line properties.

Open the Brackets_views02.CATDrawing document. Create a dimension.

1. Select the dimension you just created (whatever the type).
2. Select **Edit-> Properties**.

You can also right-click on this dimension and then choose **Properties** from the contextual menu.

3. In the Properties dialog box that appears, click the **Dimension Line** tab. The associated panel is displayed. Not all fields are active: their activation depends on your choice of options.



Representation

Specify how you want the dimension line represented: **Regular, Two Parts, Leader one Part, Leader two Parts**.

Color

Choose a color for the dimension line.

Thickness

Specify the thickness for the dimension line.

Second part

If you chose Two parts or Leader two Parts for the representation, you need to provide information about the second leader part:

- the **Reference** for positioning the second part of the dimension line,
- the **Orientation** for the secondary part of the dimension line in relation to its reference,
- the **Angle** for the secondary part of the dimension line in relation to its reference (if you selected **Dimension Line** in the **Orientation** field and **Fixed Angle** in the **Reference** field).

Extension

Choose an extension type for your dimension line.

Leader Angle

Specify the angle you want for the extension line.

Symbols

Choose the properties you want to apply to **Symbol 1**, **Symbol 2** (you may need to check this box to specify you want to the dimension to display two symbols), and **Leader Symbol** (if you chose to represent the dimension line with a leader).

- **Shape**: you can choose the dimension line shape (open arrow, outlined circle, plus, etc.).
- **Color**: you can choose the symbols color.
- **Thickness**: you can define the symbol thickness.
- **Reversal**: you can set the position of the symbols (inside or outside) in relation to the extension line.

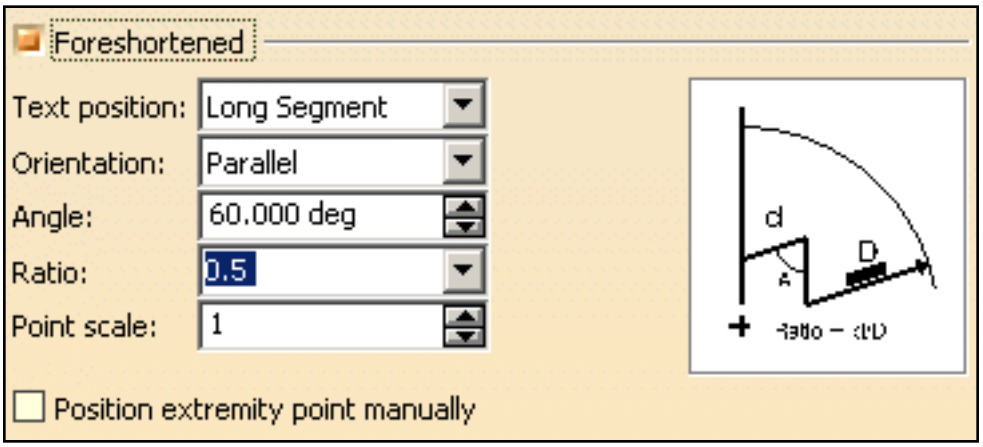
In the case of two-symbols dimensions, you can specify a different position for each symbol (i.e. symbol 1 inside and symbol 2 outside, or vice-versa).

You can also do this interactively using the **Ctrl** key.

You can apply different kinds of modifications between arrow symbol 1 and symbol 2 on the condition the drawing was created from version 5 release 5 on.

Foreshortened

For radius dimensions, you can activate the **Foreshortened** option.



It allows you to transform a radius dimension line into a foreshortened radius dimension line. You can then choose from the following options:

- **Text position**: specify whether the text should be positioned on the long segment or on the short segment of the dimension.
- **Orientation**: define the orientation of the text associated to the dimension line (parallel or convergent).
- **Angle**: specify the angle value.
- **Ratio**: specify the ratio for the short segment and the long segment of the foreshortened dimension.
- **Point scale**: specify the point scale value.
- **Unfix extremity position**: check this box to unfix the extremity point of the foreshortened dimension line. You will then be able to move the extremity point using a yellow manipulator.

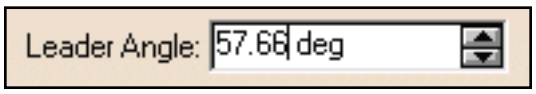
For foreshortened radius dimensions, you can define the appearance of the extremity point by making sure the **Symbol 2** box in the Symbols area is checked, and then choosing the appropriate options.

Clicking the **More** button displays extra options, if any are available.

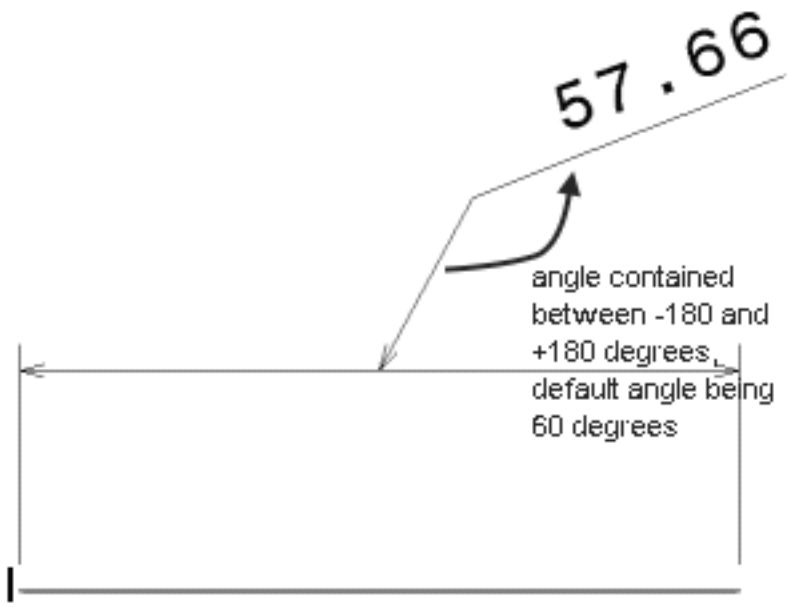
4. Modify the available options as required.

For example, from the **Representation** drop-down list, choose **Leader two Parts**.

5. In the **Leader Angle** field, specify the angle you want between the two parts of the leader.



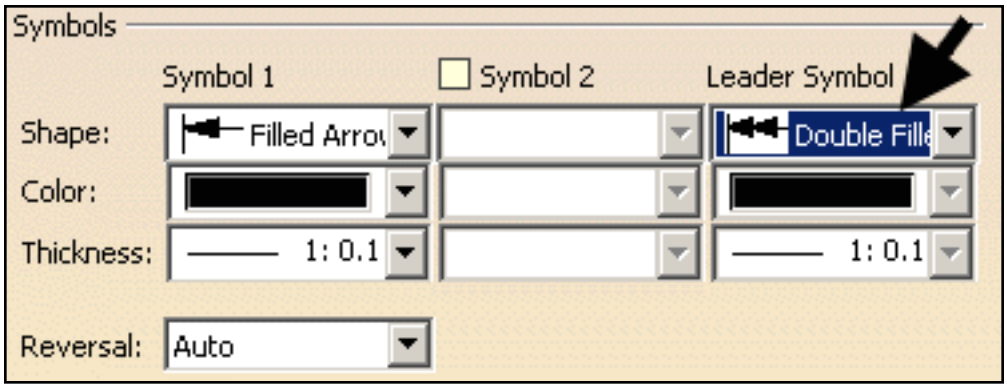
This angle is applied to the first segment:



You can also drive the second segment from the options in the **Second Part** area: it can be horizontal, vertical, parallel, perpendicular, fixed angle with screen, view, or dimension horizontal and vertical.

6. Change the Leader symbol in **Symbols-> Shape**.

Choose Double Filled Arrow, for example.



7. Transform this two parts leader into a one part leader: from the **Representation** drop-down list, choose **Leader one Part.**

8. Click **OK** to validate and exit the dialog box.

Editing Dimension System Properties

This task explains how to access and edit dimension system properties.

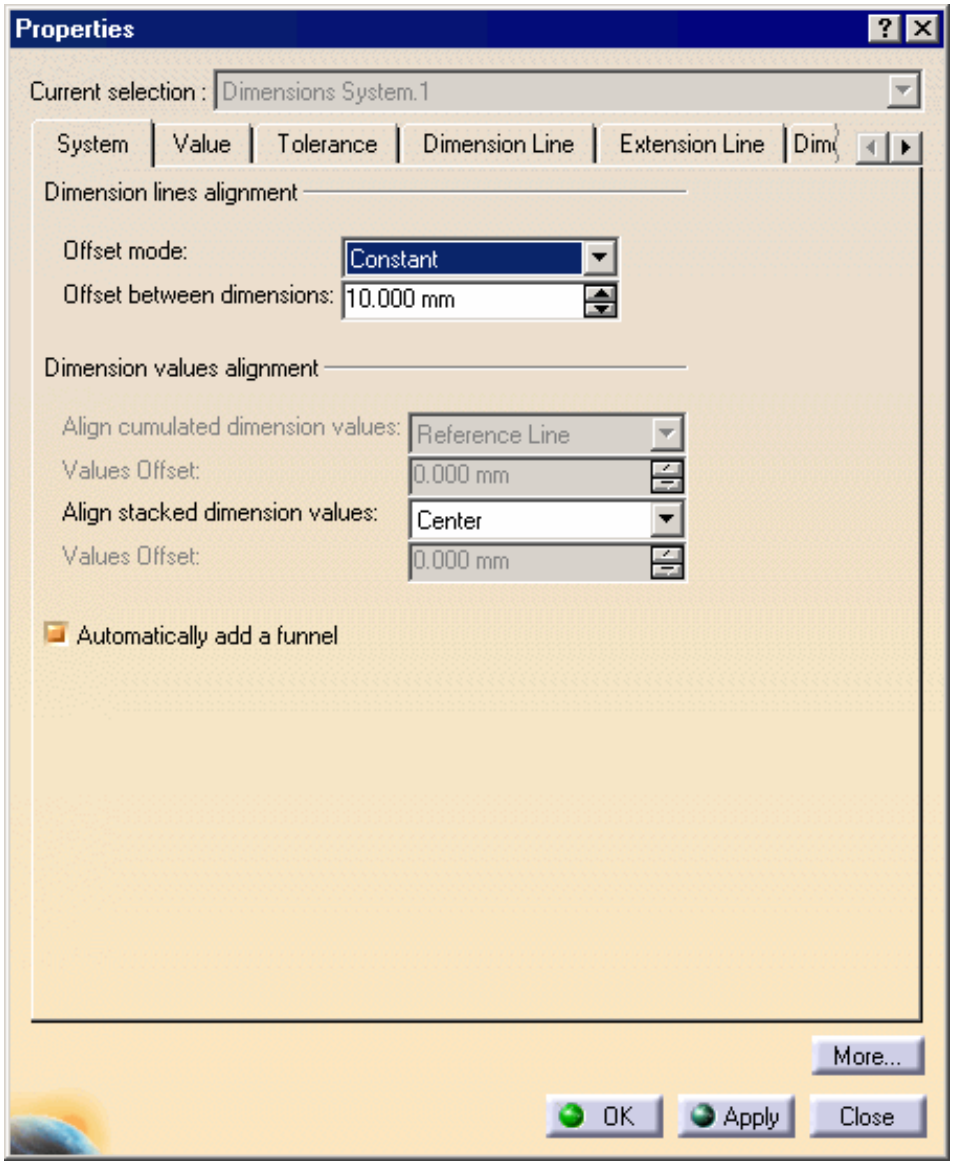
Note that chained dimension systems do not have specific properties.

1. Right-click on any stacked or cumulated dimension system you created, and select **Properties**.

   You can also select a stacked or cumulated dimension system and select **Edit -> Properties**.

2. Click on the **System** tab.

3. If needed, modify the available options.



Dimension lines alignment

Offset mode

Select an offset mode. Note that the options available depend on the selected dimensions system type.

- **Constant**: the offset between dimensions of a system remains constant and equal to the value defined in the **Offset between dimensions** field. Available for stacked dimension systems.

The offset between dimensions remains constant to the scale of the view as well. Consequently, if you modify the scale of a view and perform an alignment on the dimension system, the offset between dimensions is also modified so as to remain constant to the view's scale.

- **Free**: the dimensions of a system can be moved independently. Available for stacked dimension systems and cumulated dimension systems.

Offset between dimensions

Enter the value of the offset between dimensions.

Dimension values alignment

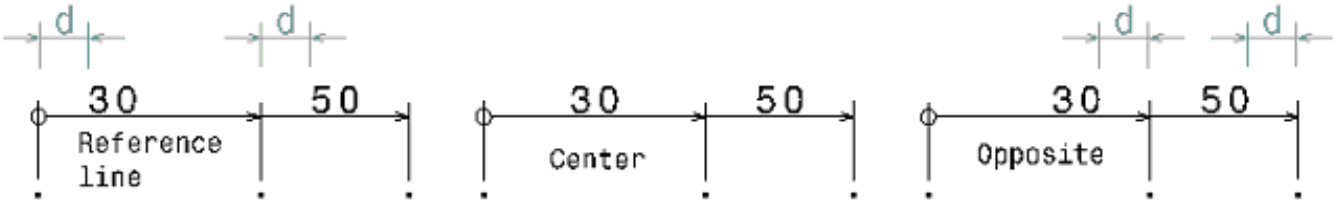
Chained dimension systems can only be centered.

Align cumulated dimension values

Select an option to specify how dimension values should be aligned for cumulated dimensions systems.

- **Reference line**
- **Center**
- **Opposite**
- **From standard**

In the image below, **d** is the **Values Offset** properties:

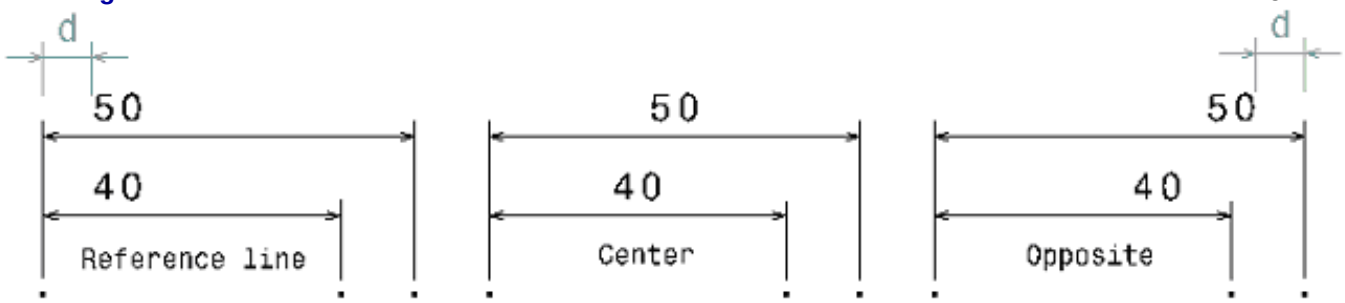


Align stacked dimension values

Select an option to specify how dimension values should be aligned for stacked dimensions systems.

- **Reference line**
- **Center**
- **Opposite**

In the image below, **d** is the **Values Offset** properties:



Values Offset

Specify the distance between the alignment reference and the dimension value.

In the standards, the Cumulate Dimension: Value Orientation Reference parameter lets you specify whether cumulated dimensions are set with their value oriented along extension lines or dimension lines.

The cumulated dimension behavior then differs depending on what you chose as the value orientation reference:

- extension lines: the Cumulate Dimensions: Parameters applying only if the value orientation reference is Extension Line standard parameters are used. Only some dimension system styles are used.
- dimension lines: some Cumulate Dimensions: Parameters applying only if value orientation reference is Dimension Line standard parameters are used. However, the value horizontal positioning is not defined in the standards (the corresponding parameters are deprecated), but by the following dimension system styles: Aligned cumulated dimension values and Values Offset.

Automatically add a funnel

Select this check box if you want funnels to be automatically added to cumulated or stacked systems whenever a dimension value line-up is performed.

If you do not select this check box, you can add funnels manually via **Edit -> Properties** or when creating the dimension system.

4. Click **OK** to validate and exit the dialog box.

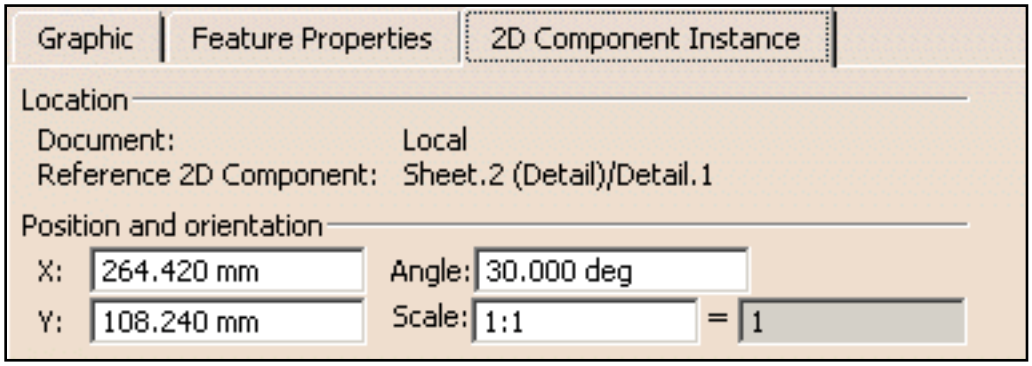
Editing 2D Component Instance Properties

This task explains how to access and, if needed, edit information on instantiated 2D component properties.

Open the Position_Component03.CATDrawing document.

1. Right-click on the 2D component instance to access the contextual menu.
2. Select **Properties** and click the **2D Component Instance** tab. You can modify the 2D component instance position and orientation:

You can also select the instance, go to **Edit -> Properties** and click the **2D Component Instance** tab.

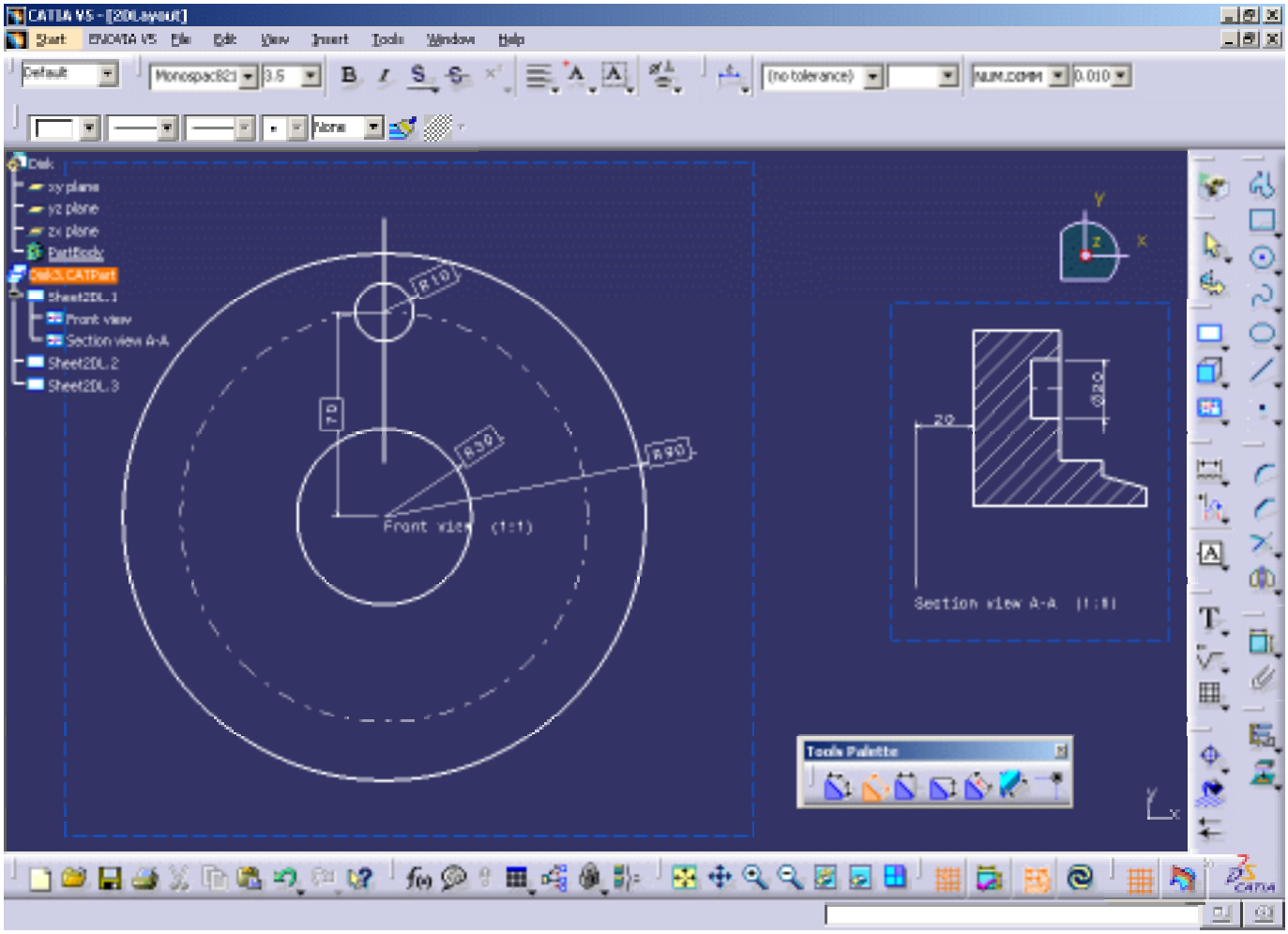


- **Location**:
  It allows you to access the instance location and the origin of the 2D component it was instantiated from.
- **Position and orientation**:
  you can modify detail instantiated 2D component coordinates, angle with horizontal reference axis and scale.

3. Click **OK** to validate and exit the dialog box.

Workbench Description

This section contains the list of the icons and menus which are specific to 2D Layout for 3D Design workbench. These commands are documented in more detail in other parts of the guide.



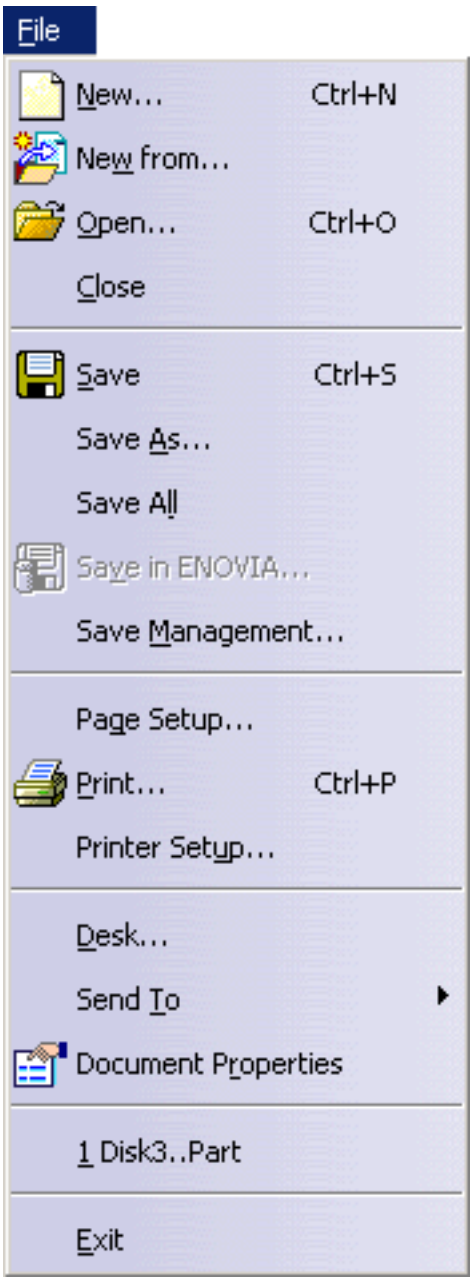
2D Layout for 3D Design Menu Bar
2D Layout for 3D Design Toolbars

2D Layout for 3D Design Menu Bar

This chapter describes the various menus, submenus and commands specific to the 2D Layout for 3D Design workbench. General menu commands are described in the *Infrastructure User's Guide*.

Start	File	Edit	View	Insert	Tools	Windows	Help

File



For...	See...
New	Creating a Layout
Page Setup	Modifying a Sheet
Print	Printing a Layout

Edit

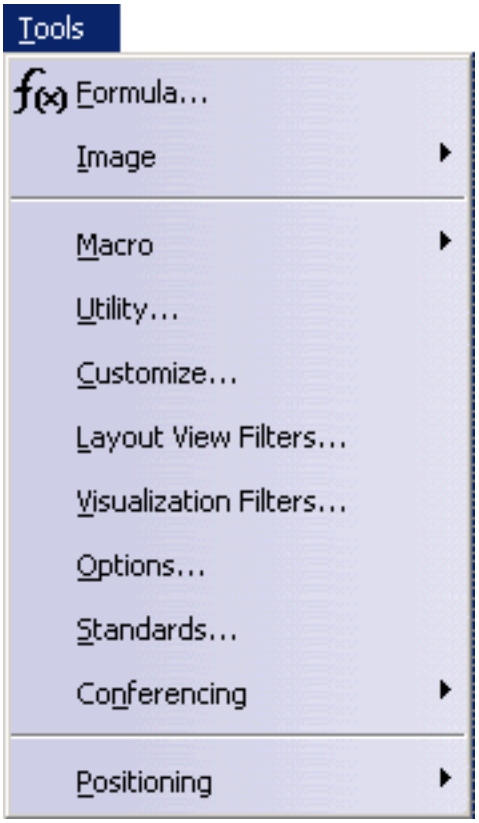
For...	See...
Background	Editing a Sheet and/or its Background

Insert

For...	See...
Layout	Layout Sheets, View Creation, 2D Components
Dimensioning	Dimensioning
Annotations	Annotations
Dress-up	Dress-up
Geometry creation	2D Geometry

Geometry modification	2D Geometry Modification
Constraint	Constraints
3D Geometry	3D Outputs, Use-edges

Tools



For...	See...
Layout View Filters	Creating View Filters
Visualization Filters	Visualization filters are not available in 2D Layout for 3D Design
Options	Customizing Settings
Standards	Administration Tasks
Positioning	Lining Up Dimensions in the *Interactive Drafting User's Guide*

2D Layout for 3D Design Toolbars

This section describes the various toolbars of the 2D Layout for 3D Design workbench. The toolbars are located around the workbench in the default set-up, except for the Tools palette which appears only when specific tools are available for a given command.

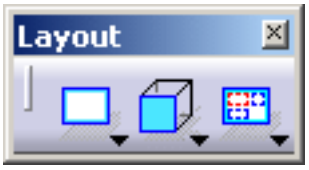
A number of toolbars (and commands) available in the 2D Layout for 3D Design workbench are actually documented in the *Interactive Drafting User's Guide* as well as in the *Sketcher User's Guide*. These toolbars are also listed below, but you may be directed to another documentation for specific commands.

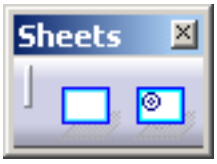
Toolbars documented in the *2D Layout for 3D Design User's Guide*	**Purpose**
3D Geometry	Create 3D profiles and 3D planes; project and intersect 3D elements with the view plane as well as project 3D silhouette edges
Layout	Create sheets, views, 2D components, as well as frame and title blocks
Tools	Activate a number of helpful tools
Tools Palette	Use specific options or value fields available for a given command
Visualization	Activate a number of visualization-related options
Toolbars documented in other *User's Guides*	**Purpose**
Annotations	Add annotations to existing views by creating them
Constraint	Create different types of constraints, and fix elements together
Dimensioning	Create all types of dimensions needed for your layout
Dimension Properties	Modify dimension properties
Dress-Up	Add dress-up elements on the layout
Geometry Creation	Create geometry
Geometry Modification	Transform existing 2D elements and add constraints to elements on the layout
Graphic Properties	Modify the graphic properties of all types of features
Style	Set the style that will be used to create a new object
Text Properties	Modify text properties

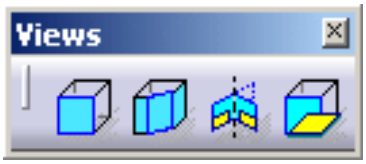
Layout

The sub-toolbars and icons available in the Layout toolbar depend on whether you are working in the main view or background view. For more information about the main and background views, refer to Editing a Sheet and/or its Background.

Layout toolbar in the main view







See Adding a Sheet to a Layout

See Creating a 2D Component

See Creating a Projection View

See Creating a Section/Auxiliary View

See Creating a Section From Two Planes

See Creating a View From Another Element

See Creating a 2D Component Reference

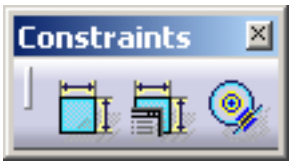
See Instantiating a 2D Component

Layout toolbar in the background view

See Creating a Frame and Title Block (documented in the *Interactive Drafting User's Guide*)

See Instantiating a 2D Component

Constraint



Note that the Constraint commands listed below are documented either in the *Interactive Drafting User's Guide* or in the *Sketcher User's Guide*.

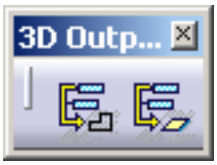
See Creating Quick Constraints (documented in *Interactive Drafting User's Guide*)

See Fixing Elements Together (documented in *Sketcher User's Guide*)

See Creating Constraints via a Dialog Box (documented in *Interactive Drafting User's Guide*)

See Creating a Contact Constraint (documented in *Sketcher User's Guide*)

3D Geometry



See Creating a 3D Profile

See Creating a 3D Plane

See Projecting 3D Elements onto the View Plane

See Intersecting 3D Elements with the View Plane

See Projecting 3D Silhouette Edges

Tools

See Layout Tools

Visualization

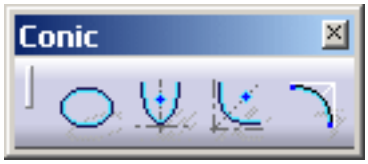
See Layout Tools

Tools Palette

The options or fields available in the Tools Palette depend on the selected command. Only a few examples are provided here.

See Layout Tools

Geometry Creation



Note that all Geometry Creation commands listed below are documented in the *Sketcher User's Guide*.

See Profiles

See Rectangles

See Oriented Rectangles

See Parallelograms

See Elongated Holes

See Cylindrical Elongated Holes

See Keyhole Profiles

See Hexagons

See Centered Rectangle

See Centered Parallelogram

See Circles

See Three Point Circle

See Circles Using Coordinates

See Tri-Tangent Circle

See Three Point Arc

See Three Point Arc with Limits

See Arcs

See Splines

See Connect

See Ellipses

See Parabola by Focus

See Hyperbola by Focus

See Conic

See Lines

See Infinite Lines

See Bi-Tangent Lines

See Bisecting Lines

See Lines Normal to Curves

See Points

See Points using Coordinates

See Equidistant Points

See Intersection Points

See Projection Points

Geometry Modification

Note that all Geometry Modification commands listed below are documented in the *Sketcher User's Guide*.

See Modifying a Corner

See Modifying a Chamfer

See Trimming Elements

See Breaking Elements

See Breaking and Trimming Elements

See Closing Elements

See Complementing an Arc (Circle or Ellipse)

See Creating Mirrored Elements

See Moving Element by Symmetry

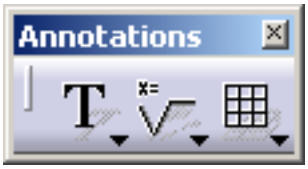
See Translating Elements

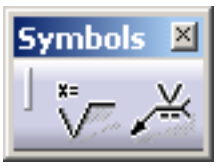
See Rotating Elements

See Scaling Elements

See Offsetting Elements

Annotations





Note that all Annotation commands listed below are documented in the *Interactive Drafting User's Guide*.

See Creating Text

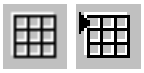
See Creating Text with Leader

See Creating a Balloon

See Creating a Datum Target

See Creating a Roughness Symbol

See Creating a Welding Symbol

 See Creating/Modifying a Table

Dress-Up

Note that all Dress-up commands listed below are documented in the *Interactive Drafting User's Guide*.

See Creating a Center Line and Modifying a Center Line or an Axis Line

See Creating a Center Line with Reference and Modifying a Center Line or an Axis Line

See Creating a Thread

See Creating a Thread with Reference

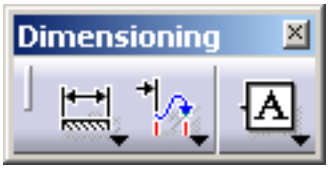
See Creating an Axis Line and Modifying a Center Line or an Axis Line

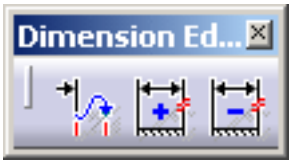
See Creating Axis Lines and Center Lines

See Creating an Area Fill

See Creating an Arrow

Dimensioning





Note that all Dimensioning commands listed below are documented in the *Interactive Drafting User's Guide*.

See Creating a Dimension

See Creating a Chained Dimension

See Creating a Cumulated Dimension

See Creating a Stacked Dimension

See Creating Explicit Dimensions

See Creating Explicit Dimensions

See Creating Explicit Dimensions

See Creating Explicit Dimensions

See Creating a Chamfer Dimension

See Creating Thread Dimensions

See Creating a Coordinate Dimension

See Re-routing Dimensions

See Interrupting Extension Lines

See Interrupting Extension Lines

See Creating a Datum Feature

See Creating a Geometrical Tolerance

Dimension Properties

See Before You Begin - Dimensions (documented in *Interactive Drafting User's Guide*)

Text Properties

See Setting Text Properties (documented in *Interactive Drafting User's Guide*)

Graphic Properties

See Editing 2D Element Graphic Properties (documented in *Interactive Drafting User's Guide*)

Note that layer functionalities are not available in 2D Layout for 3D Design.

Style

See Using Standard-Defined Styles (documented in *Interactive Drafting User's Guide*)

Customizing

The 2D Layout for 3D Design workbench is customized via Drafting settings. This section explains how to customize settings and toolbars.

Customizing Settings
Customizing Toolbars

Customizing Settings

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

1. Select **Tools -> Options**. The Options dialog box is displayed.
2. Choose the **Mechanical Design** category in the left-hand box.
3. Select the **2D Layout for 3D Design** sub-category. The View Creation tab that appears lets you set options that will be used when creating views.

4. Set options in this tab according to your needs.
5. Select the **Drafting** sub-category. Various tabs are displayed.

- General lets you set general settings to be used in the Generative Drafting workbench.
- Layout lets you customize options that will be used when creating views or when adding sheets.
- View lets you customize geometry, dress-up and view generation options that will be used when generating views.
- Geometry lets you customize options that will be used when creating 2D geometry, whether using autodetection (or SmartPick) or not, or still adding constraints to this geometry.
- Dimension lets you customize options that will be used when creating or re-positioning dimensions.
- Manipulators lets you visualize options that will be used for manipulators whenever creating or modifying dimensions.
- Annotation and Dress-Up lets you customize options that will be used when creating annotations.
- Administration lets you customize options for the management of drawings.

Note that the settings in this sub-category apply to 2D Layout for 3D Design as well as to the Drafting workbenches. Some settings may not apply to 2D Layout for 3D Design.

6. Set options in these tabs according to your needs.
7. Click **OK** in the Options dialog box when done.

View Creation

This page deals with the following categories of options in the **View Creation** tab:

- View Visualization
- View Background
- Filter
- Clipping

View Visualization

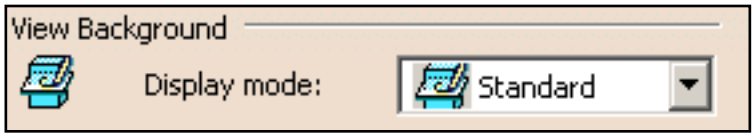
Hide in 3D

Select this check-box if you want newly created views to be hidden from the 3D window.

You can show or hide the view in the 3D window at any time by right-clicking it in the 2D window and selecting **Visualization -> Show in 3D** or **Visualization -> Hide in 3D** respectively.

By default, this check-box is not selected.

View Background



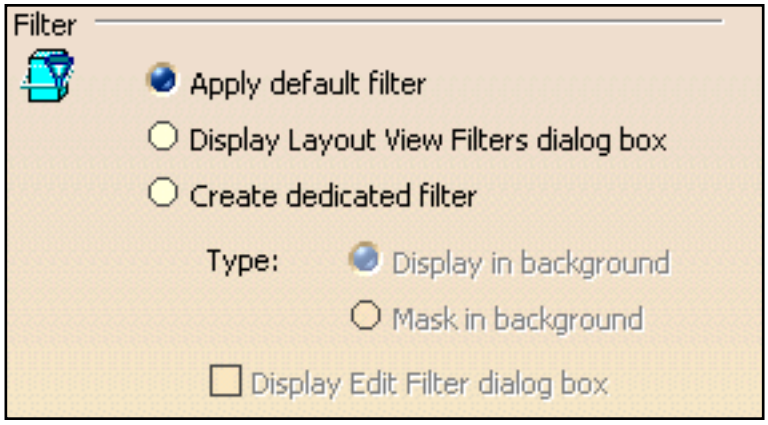
Display mode

From the drop-down list, select a display mode to specify how the 2D and 3D backgrounds should be visualized for newly created views:

- **Standard**: shows both the 2D and 3D backgrounds.
- **Invisible**: hides both the 2D background (the 3D representation of 2D elements which do not belong to the current view, but to other views) and the 3D background (the representation of all 3D elements, including edges, faces and 3D wireframe).
- **Unpickable**: prevents selecting elements in both the 2D and 3D backgrounds, even though you can see them. You can just handle 2D elements which belong to the current view.
- **Low-intensified**: dims all elements in both the 2D and 3D backgrounds.
- **Unpickable low-intensified**: dims all elements in both the 2D and 3D backgrounds. Additionally, although you can see these elements, you cannot select them. You can just handle 2D elements in the current view.

By default, the **Standard** display mode is selected.

Filter



For more information on filters, refer to Creating View Filters.

Apply default filter

Select this option if you want each newly created view to be filtered by the first layout filter in the list of view filters (available in the Layout View Filters dialog box). By default, this filter is the Default filter (a Mask filter created at the same time as the layout).

By default, this option is selected.

Display Layout View Filters dialog box

Select this option if you want the Layout View Filters dialog box to be displayed each time you create a new view.

By default, this option is not selected.

Create dedicated filters

Select this option if you want a new filter to be created and applied to each newly created view.

Selecting this option makes the options below it available.

By default, this option is not selected.

Display in background

This option is available when the **Created dedicated filters** option is selected.

Select this option if you want the new filter that is created for each new view to be of the Display type.

By default, this option is selected.

Mask in background

This option is available when the **Created dedicated filters** option is selected.

Select this option if you want the new filter that is created for each new view to be of the Mask type.

By default, this option is not selected.

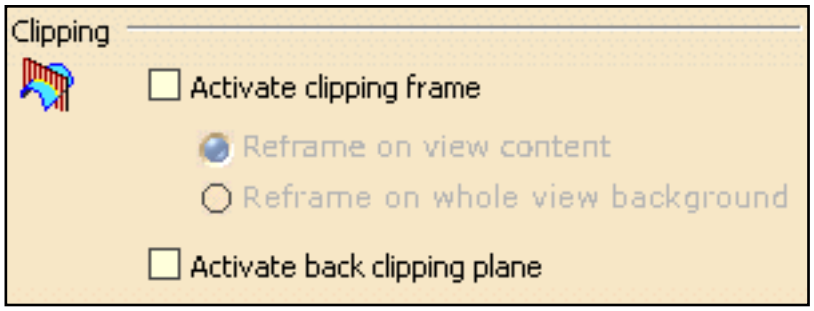
Display Edit Filter dialog box

This option is available when the **Created dedicated filters** option is selected.

Select this option if you want to launch the edition of the new filter automatically.

By default, this option is not selected.

Clipping



Activate clipping frame

Select this check-box if you want to clip the 2D and 3D backgrounds of newly created views using a rectangular frame. Refer to Using the Clipping Frame for more information.

By default, this check-box is not selected.

Reframe on view content

This option is available when the **Activate clipping frame** option is selected.

Select this option if you want the clipping frame to be reframed on the view content. Thus reframed, the clipping frame is slightly larger than the layout view frame.

By default, this option is selected.

Reframe on whole view background

This option is available when the **Activate clipping frame** option is selected.

Select this option if you want the clipping frame to be reframed on the whole view background. Thus reframed, the clipping frame shows the whole 2D and 3D background.

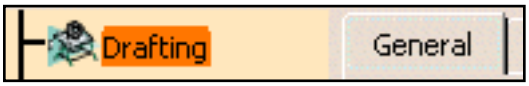
By default, this option is not selected.

Activate back clipping plane

Select this check-box if you want to clip the 2D and 3D backgrounds of newly created views using a back clipping plane. Refer to Using the Back Clipping Plane for more information.

By default, this check-box is not selected.

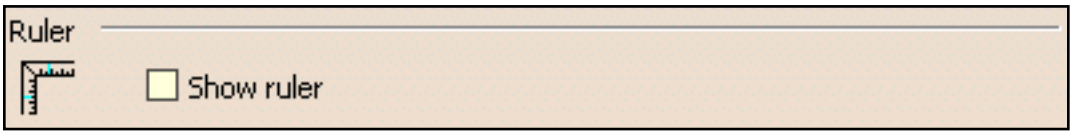
General



This page deals with the following categories of options in the **General** tab:

- Ruler
- Grid
- Rotation
- Colors
- Tree
- View Axis
- Start Workbench

Ruler

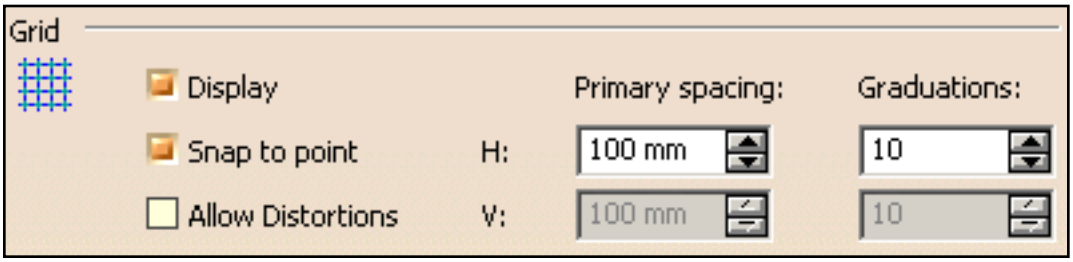


Show ruler

Select this option to display the ruler in your sheet. It means you visualize the cursor coordinates as you are drawing.

By default, this option is not selected.

Grid



Display

Select this option to display the grid in your session. Note that this option is also available via the **Tools** toolbar.

By default, this option is selected.

Snap to point

Select this option if you want geometry (as well as 2D components) to begin or end on the various intersection points of the grid. Note that this option is also available via the **Tools** toolbar.

By default, this option is selected.

From V5 R15 onwards, this option no longer applies to annotations.
Now, only the **Snap by default (Shift toggles)** setting is used to specify whether snapping should be activated by default for annotations. For more information, refer to Annotation and Dress-up.

Allow Distortions

Select this option to apply different graduations and spacing between **H** and **V**.

By default, this option is selected.

Primary spacing: H & V

The **Primary spacing** field lets you define the spacing between the major lines of the grid. To define your grid, enter the values of your choice in the **H** and/or **V** fields.

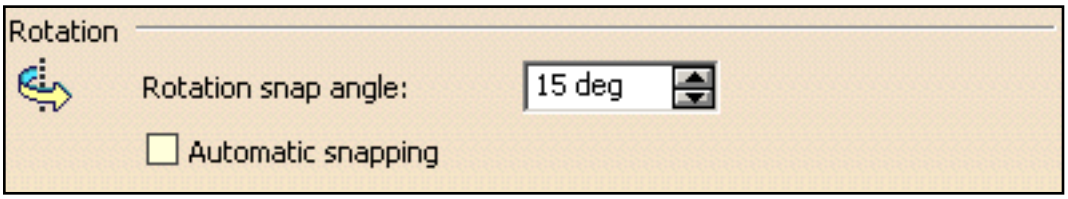
By default, primary spacing is set to 100 mm.

Graduations / H & V

The **Graduations** field lets you set the number of graduations between the major lines of the grid, which actually consists in defining a secondary grid. To define your grid, enter the values of your choice in the **H** and/or **V** fields.

By default, graduations are set to 10.

Rotation



Rotation Snap Angle

Specify the angle that should be used when rotating text elements (text, frame, or leader) using snapping. In other words, this option defines the snapping value used when rotating an element using the **Select** or **Rotate** commands.

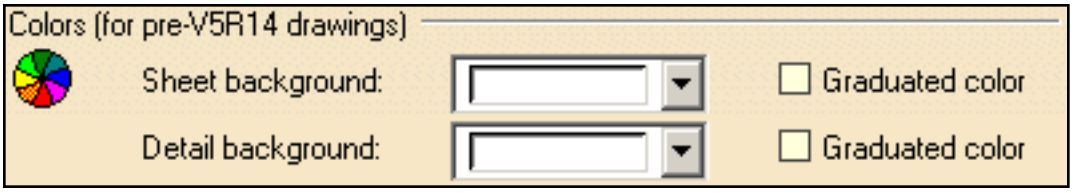
By default, rotation snap angle is set to 15 deg.

Automatic Snapping

Select this option to use snapping automatically when rotating an element.

By default, this option is not selected.

Colors (for pre-V5R14 drawings)



These options apply to drawings created with versions prior to V5R14. For drawings created with V5R14 onwards, sheet and detail backgrounds colors are defined in the standard file, under the node **Sheet colors** of **General** node.

Sheet background

Choose the color that will be used for the sheet background.

By default, the color is white.

Detail background

Choose the color that will be used for the background of 2D components.

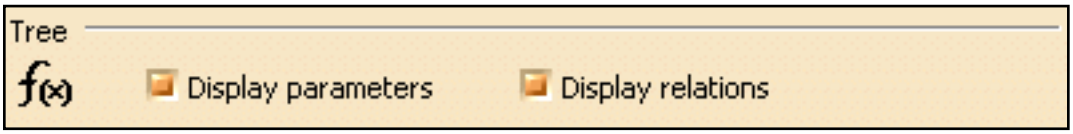
By default, the color is white.

Graduated color

If you want the sheet background and/or the detail (i.e. 2D component) background to be graduated, select the associated box.

By default, these options are not selected.

Tree



Display parameters

Select this option to display in the specification tree the formula parameters used in the drawing.

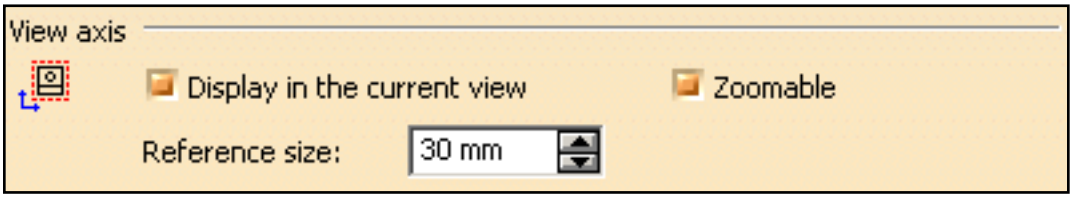
By default, this option is selected.

Display relations

Select this option to display in the specification tree the relation parameters used in the drawing.

By default, this option is selected.

View axis



Display in the current view

Select this option if you want the view axis to be displayed when you activate a view.

By default, this option is selected.

Zoomable

Select this option if you want to be able to zoom view axes (as you can do with geometry).

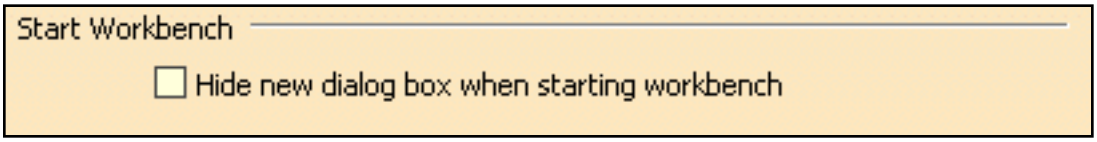
By default, this option is selected.

Reference size

Enter the size that you want to use as a reference to display view axes size.

By default, the reference size is 30 mm.

Start Workbench



Hide new dialog box when starting workbench

Select this option if you want to hide the New Drawing dialog box (in the case of Drafting) and the New Layout dialog box (in the case of 2D Layout for 3D Design) when you start the workbench.

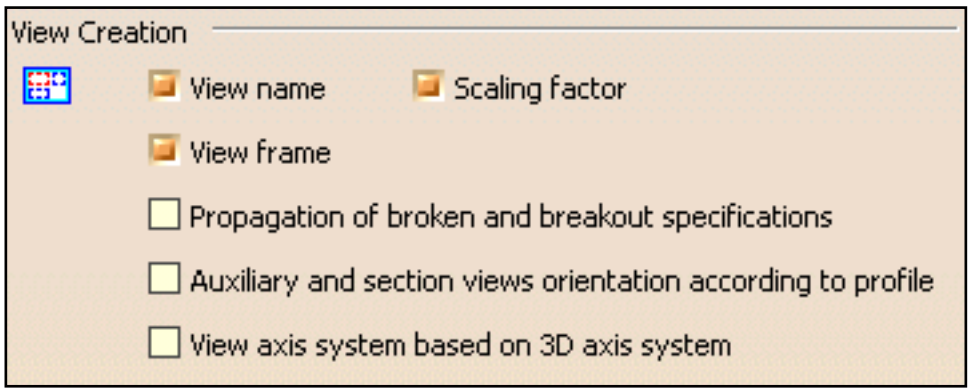
By default, this option is not selected.

Layout

This page deals with the following categories of options in the **Layout** tab:

- View Creation
- New Sheet
- Background View
- Section / Projection Callout

View Creation



View name

Select this option if you want the view name to be created automatically when creating views.

By default, this option is selected.

Scaling factor

Select this option if you want the scaling factor to be created automatically when creating views.

By default, this option is selected.

View frame

Select this option if you want the view frame to be created automatically when creating views.

By default, this option is selected.

Propagation of broken and breakout specifications

Select this option if you want broken and breakout specifications to be reproduced.

By default, this option is not selected.

Auxiliary and section views orientation according to profile

Select this option if you want auxiliary and section views to be oriented according to the profile. In this case, the X axis will be parallel to the profile.

Be aware that this setting is a default value, i.e. the setting used when creating the view is stored by the view and cannot be edited via the view's properties (available via Edit > Properties).

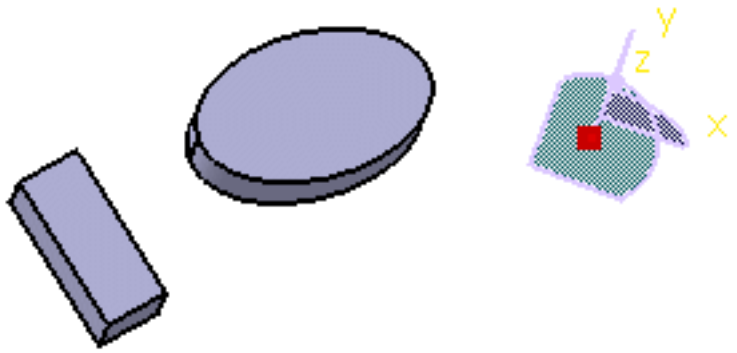
This setting drives the angle of the view in the sheet. So if the auxiliary/section view profile is modified, the view rotation angle will be modified accordingly.

By default, this option is not selected.

View axis system based on 3D axis system

Select this option if you want the axis system of the generated view to be based on the axis system of the 3D part. This enables you to create views with the same orientation if, when creating two views in the same projection plane by selecting two different faces, the axis systems which are specific to these faces are different.

For example, take the following part:



With the **View axis system based on 3D axis system** option not checked, the view orientation will be different depending on the element selected in the 3D when creating the view:

View orientation when a face of the rectangular pad is selected

View orientation when a face of the elliptic pad is selected

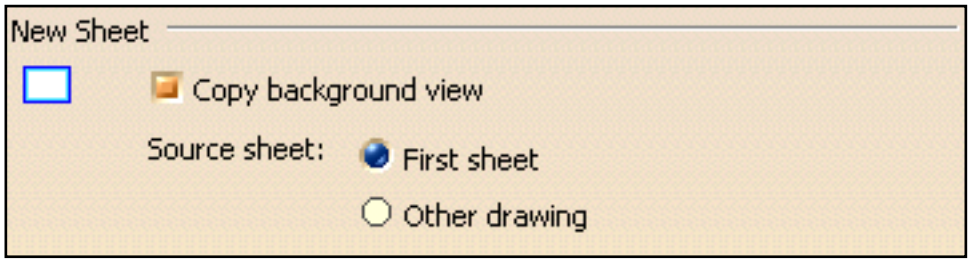
View orientation when the absolute XY plane is selected

With the **View axis system based on 3D axis system** option checked, the view orientation will always be the same, no matter what element is selected in the 3D when creating the view:

View orientation when either a face of the rectangular pad, a face of the elliptic pad, or the absolute XY plane is selected

By default, this option is not selected.

New sheet



Copy background view

Select this option if you want a background view to be copied into newly created sheets.

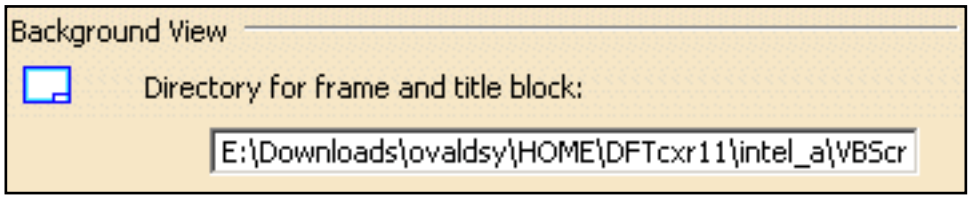
By default, this option is selected.

Source sheet

Specify whether you want the source sheet for the background view to be the first sheet of the current drawing, or a sheet from another drawing by selecting the appropriate option.

By default, the option is **First sheet**.

Background view

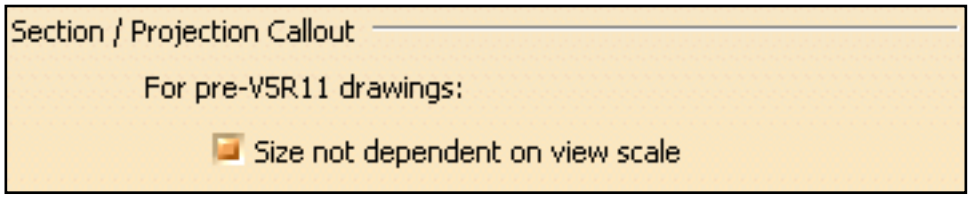


Directory for frame and title block

You can specify the path to the directory containing the frame and title block macros.

By default, the path is install_root/VBScript/FrameTitleBlock.

Section/Projection Callout



Size not dependent on view scale

Select this option if you do not want the size of projection and section callout elements to be dependent on the view scale. This option will apply to newly created callouts, i.e. selecting this option will not have any impact on existing callouts.

Note that this option only applies to drawings created with versions prior to V5 R11 (i.e. versions up to V5 R10).

By default, this option is selected.

View

This page deals with the following categories of options in the **View** tab:

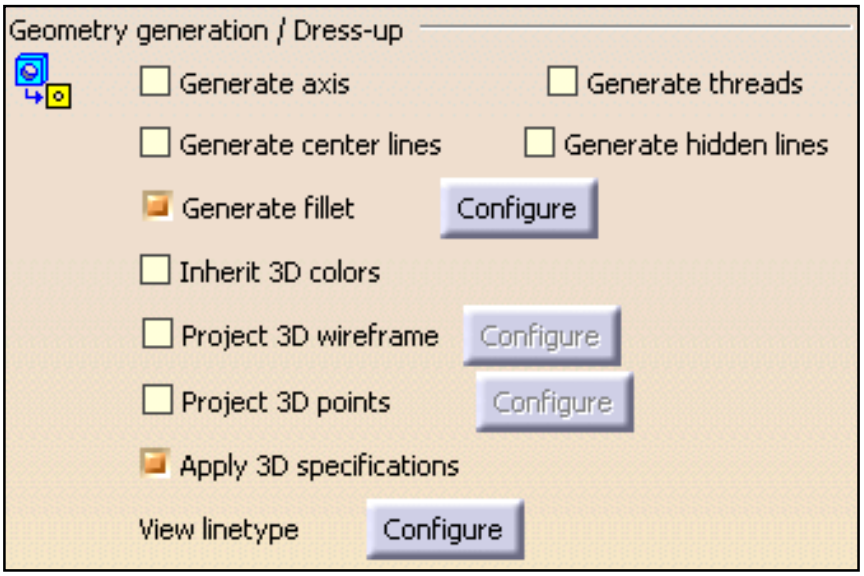
- Geometry generation / Dress-up
- View generation
- Clipping view
- View from 3D

These options do not apply to the Interactive Drafting workbench.

These options are applied when creating new views. Once the view has been created, you can modify its properties through the **Properties** dialog box: from the contextual menu of a given view, select **Properties**, click the **View** tab and then select the desired options. For more information, refer to Editing View Properties.

Geometry generation / Dress-up

This category of options lets you specify what kind of geometry and dress-up elements you want to generate when generating views.



Generate axis

Select this option to generate axis lines.

By default, this option is not selected.

Generate threads

Select this option to generate threads.

By default, this option is not selected.

Generate center lines

Select this option to generate center lines.

By default, this option is not selected.

Generate hidden lines

Select this option to generate hidden lines.

By default, this option is not selected.

Generate fillets

Select this option to generate fillets.

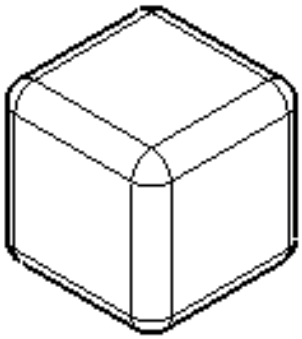
By default, this option is selected.

Additionally, click the **Configure** button to configure fillet generation. You can choose to generate either of the types of fillets described below.

By default, **Boundaries** is selected.

Boundaries

Thin lines, representing the mathematical limits of the fillets.

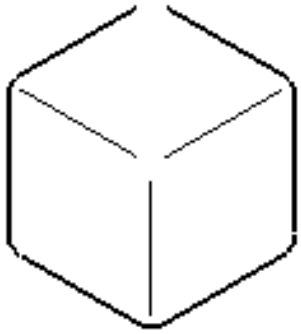


Boundaries will not be projected if they correspond to two faces which are continuous in curvature. They will be projected only if they correspond to a smooth edge which is situated between two faces whose curvature radii vary.

This mode will be used automatically to represent a connection between two faces which are not joined by a fillet, no matter what option you select.

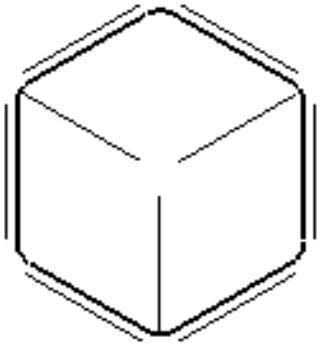
Symbolic

Original edges, projected in a direction that is normal to each corresponding surface.



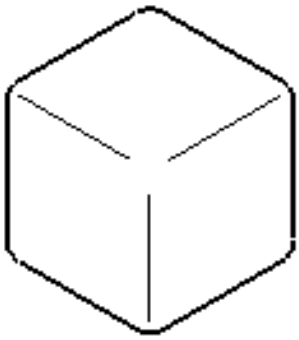
Approximated Original Edges

Original edges, at the intersection of the two surfaces joined by the fillet.



Projected Original Edges

Original edges, projected on fillet surfaces in the direction of the view projection.



This projection mode is equivalent to the CATIA V4 fillet projection mode.

The following restrictions apply to **Symbolic**, **Approximated Original Edges** and **Projected Original Edges**:

- Dimensions on such fillets are not associative.
- Such fillets cannot inherit 3D colors (see below). Likewise, when using generative view styles, such fillets cannot inherit the 3DInheritance view dress-up parameters (defined in **Tools -> Standard -> generativeparameters -> *.XML** file, **Drafting -> ViewDressup -> 3DInheritance**).

Inherit 3D colors

Select this option if you want the colors of a part to be automatically generated onto the views.

In the case of white parts, the views generated with this option selected will be white, and will therefore not be displayed properly.

By default, this option is not selected.

Project 3D Wireframe

Select this option to visualize both the wireframe and the geometry on generated views.

By default, this option is not selected.

Additionally, click the **Configure** button to configure the 3D wireframe projection mode. You can choose whether projected 3D wireframe *can be hidden* (3D wireframe geometry will follow standard removal of hidden lines) or *is always visible* (3D wireframe geometry will not follow standard removal of hidden lines and will always be visible). Whichever option you choose, note that linear elements will not be projected as points when they are perpendicular to the projection plane.

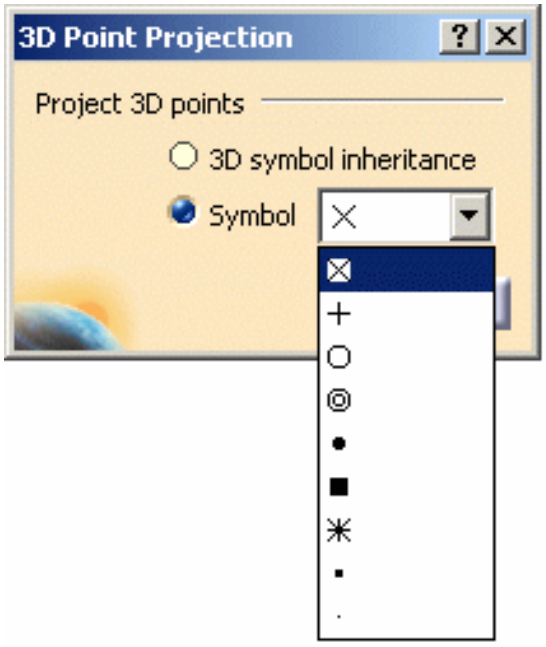
By default, **Can be hidden** is selected.

Project 3D Points

Select this option to project points from 3D (no construction element).

By default, this option is not selected.

Additionally, click the **Configure** button to select the type of points visualized in the projected drawing. In the 3D Point Projection dialog box, you can choose between keeping the symbols that are used in the 3D or using a new symbol.



By default, the **X** symbol is selected.

Apply 3D specification

Select this option to specify whether, in an assembly, the properties assigned to given parts (also called components) will be applied in the view.

The following 3D specifications may be defined for components in the Product Structure workbench:

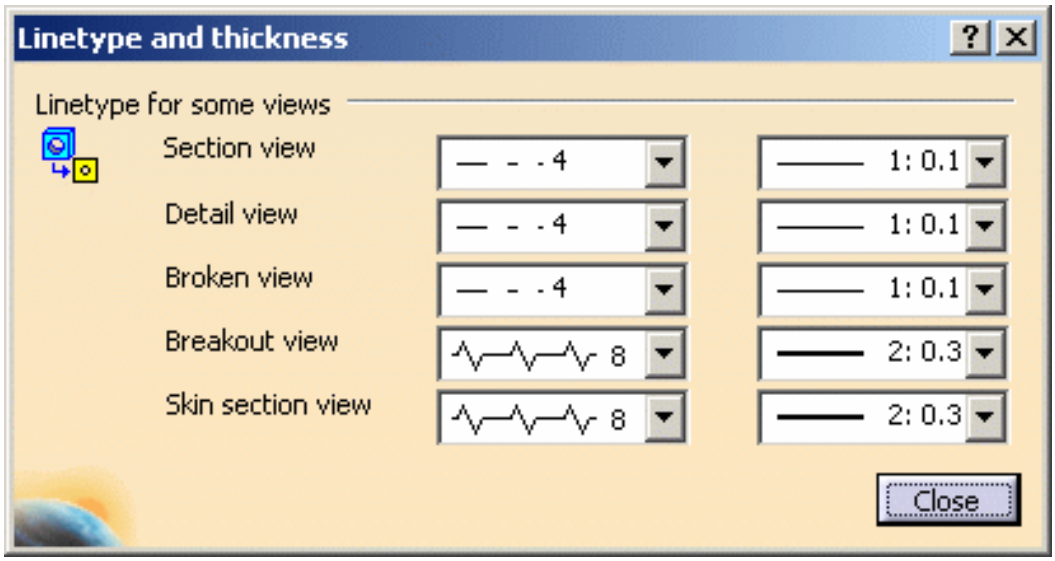
- The component will, or will not, be cut when projected in section views (Do not cut in section views).
- The component will, or will not, be projected in views (Do not use when projecting).
- The component will, or will not, be represented with hidden lines (Represented with hidden lines).

For more information, refer to Modifying Component Properties in the *Product Structure User's Guide*.

By default, this option is selected.

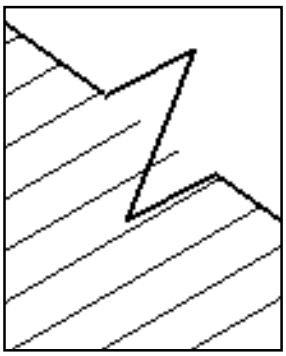
View Linetype

Click the **Configure** button to configure linetypes and thicknesses for specific types of views: section view, detail view, broken view, breakout view, skin section view (in the case of wireframes and surfaces). In the Linetype and thickness dialog box, select the line type and the thickness you want for each type of view, from the associated fields. Click **Close** when you are done.

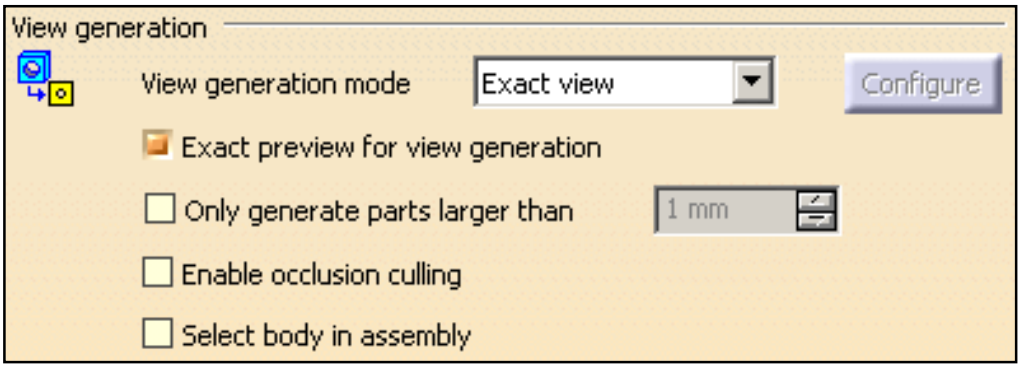


By default, linetypes and thicknesses are configured as shown above.

If you choose the zigzag linetype (linetype #8), note that this linetype is just a graphical dress-up of the view. This means that if one line is relimited on the breakout line, then it will be relimited on the theoretical line as shown here, and not on the visualized zigzag line.



View generation



View generation mode

From this list, select how you want to generate views.

By default, **Exact view** is selected.

A short description of the various view generation modes is provided below. For a detailed description (including the advantages and restrictions pertaining to each mode), refer to View Creation > About the View Generation Modes.

Exact view

Generates exact views from the Design mode, i.e. views for which the geometry is available. The exact generation mode will be the best option in most cases.

CGR

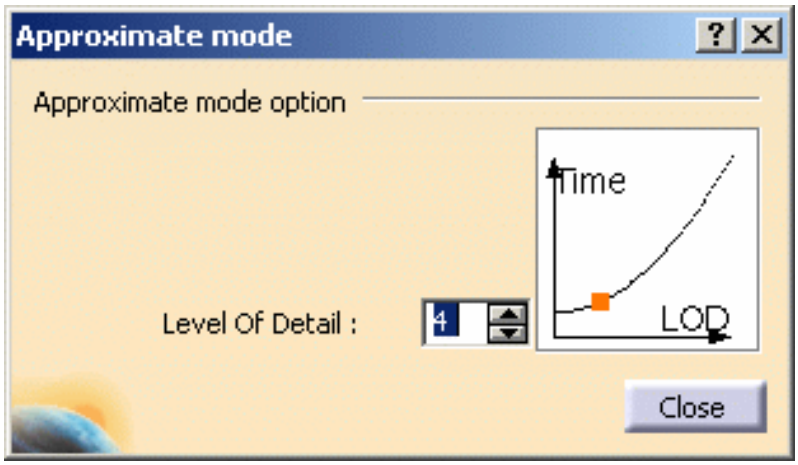
Generates views using the CGR format (CATIA Graphical Representation). CGR corresponds to a data format containing a graphical representation of the geometry only, which is available with the Visualization mode (as opposed to the exact geometry, which is available with the Design mode). With CGR, only the external appearance of the component is used and displayed; the geometry is not available. The corresponding .cgr file, if it exists, is inserted from the cache system.

Approximate

Generates views in Approximate mode. Although Approximate views are not as high in precision and quality as exact views, this generation mode dramatically reduces memory consumption. Performances may also be improved, depending on how you fine-tune precision. Therefore, the Approximate mode is particularly well-adapted to sophisticated products or assemblies involving large amounts of data.

You can fine-tune the generation options according to your needs. Click the **Configure** button. The **Approximate mode** dialog box is displayed, allowing you to set the precision (that is the level of detail - LOD) with respect to the performances (that is generation time - Time).

The level of detail corresponds to the precision with which the application determines which edges are hidden and which are not. As a result, decreasing this precision may lead to smaller geometry being visible whenever it should not be, and vice-versa. The higher the precision, the lower the performances. In any case, memory consumption will not be impacted.



Proceed as follows:

a. Specify the level of detail in the dedicated field, or move the cursor to set the precision in the graphic area on the right.

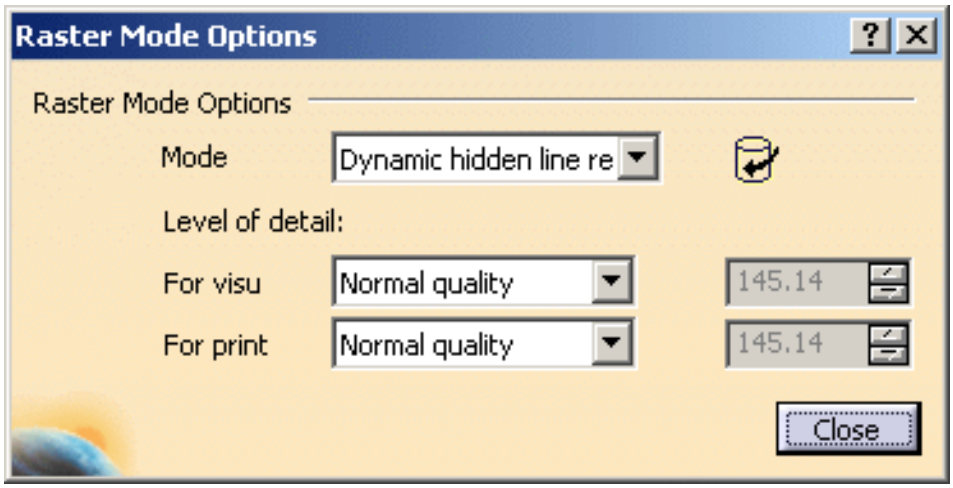
b. Click **Close** when you are done.

Raster

Generates views as images. This enables you to quickly generate overall views for large products or assemblies, regardless of drawing quality. Such views are associative to the 3D geometry and can be updated when the part or product changes.

To optimize disk space and memory consumption, it is recommended that you do not select the **Inherit 3D colors** option when generating raster views.

You can configure the raster mode options according to your needs. Click the **Configure** button. The **Raster Mode Options** dialog box is displayed.



Proceed as follows:

a. From the **Mode** list, select the mode that you want to use: Dynamic Hidden Line Removal, Shading, Shading with edges. These modes are equivalent to the 3D rendering styles. For more information, refer to Using Rendering Styles in the *Infrastructure User's Guide*.

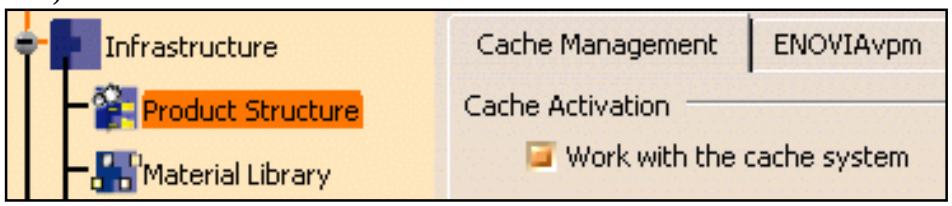
If you want the colors of a part to be used when generating Raster views using the Shading or Shading with edges mode, remember to select the **Inherit 3D Colors** option. Otherwise, the view will be generated using shades of grey.

b. Now, set the level of detail (i.e. the definition, in dpi) that will respectively be used to visualize and to print the drawing. You can choose between three pre-defined modes (Low quality, Normal quality and High quality) and a custom mode (Customize). If you choose to customize the definition yourself, set the dpi for visualization and for print in the appropriate fields.

The level of detail applies to the scale of the view. In some cases (when the view would print with a considerable height or width), there may be too many pixels to generate the view. In this case, the view will be displayed as a red cross-mark. If this happens, try to reduce the scale of the view and/or the level of detail.

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

To further improve performance when generating Raster or CGR views, we recommend you work in Visualization mode: to do this, in the Options dialog box, go to **Infrastructure -> Product Structure -> Cache Management** tab and select **Work with the cache system**. (For more information, see Customizing Cache Settings in the *Infrastructure User's Guide* and Visualization mode in the *Product Structure User's Guide*.)



By default, this option is not selected.

Exact preview for view generation

Make sure this check box is selected if you want an exact preview when generating views. Note that exact previews show exactly what will be projected.
In this case, the part or product will be loaded in Design mode when previewing the view to generate, even if you are working in Visualization mode.

Clear this check box to get a quick preview of the 3D document when generating views. Note that quick previews do not necessarily show what will be projected, but show exactly what is visualized in 3D.
In this case, a part or product open in Visualization mode will not be loaded in Design mode for the preview, which optimizes memory consumption.

By default, this option is selected.

Only generate parts larger than

To specify that you only want to generate parts which are larger than a certain size, select this option and indicate the appropriate size by providing a value in millimeters in the appropriate field.

By default, this option is not selected.

Enable occlusion culling

Select this option if you want to save memory when generating exact views from an assembly which is loaded in Visualization mode (i.e. when the **Work with the cache system** option is active). This will load only the parts which will be seen in the resulting view (instead of loading all of them, which is the case by default), which optimizes memory consumption and CPU usage.
To ensure the efficiency of this option, make sure that the **Exact preview for view generation** option is not selected.
In the case of an assembly which is loaded in Design mode, or in the case of a part, the **Enable occlusion culling** option will help increase performance by reducing CPU usage.

Keep the following restrictions in mind when selecting the **Enable occlusion culling** option:

- The **Project 3D points** option will be disabled.
- The only option available for **Project 3D wireframe** is **Can be hidden**.

- If you choose to project 3D wireframe, you will need to make sure that your wireframe elements have been taken into account when the CGR data was created: this is the case if you activated the **Save lineic elements in cgr** option from **Tools -> Options -> Infrastructure -> Product Structure -> Cgr Management** before the creation of CGR data (i.e. before you launched the part or product in Visualization mode). If not, you need to activate the **Save lineic elements in cgr** option and then re-create the CGR data. To do this:

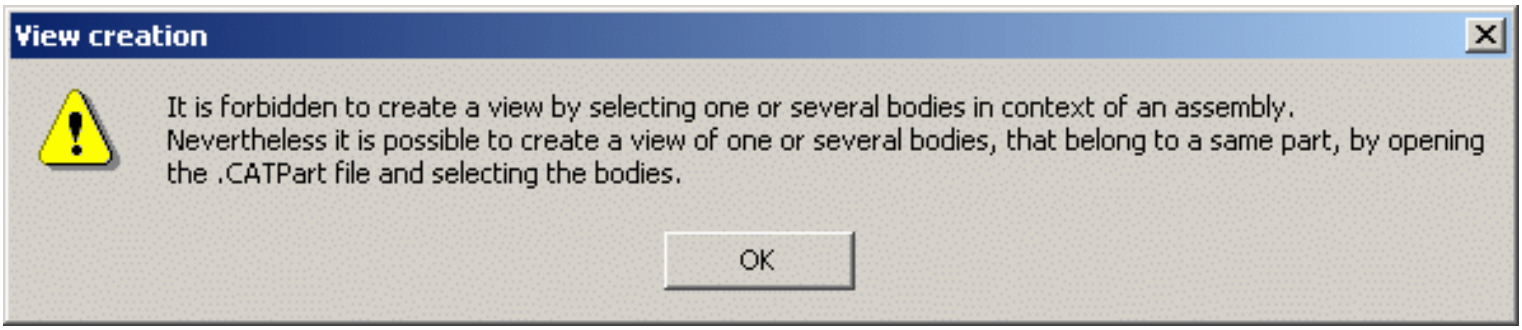
  1. Close all open parts and products and exit the application.
  2. Delete your CGR data from the cache. (The cache location is specified in **Tools -> Options -> Infrastructure -> Product Structure -> Cache Management** tab, **Path to the local cache** field.)
  3. Re-open the product in Visualization mode.

By default, this option is not selected.

Select body in assembly

Select this option if you want to be allowed to create a view selecting one or several bodies in an assembly.

By default, the box is not checked and the following error message is displayed if you try to generate a view from a body.



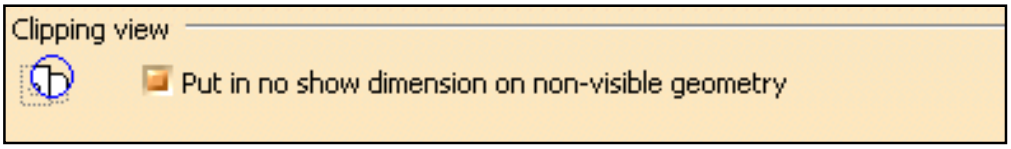
Once **Select body in assembly** is checked, a warning is displayed when creating the view as you are strongly advised not to use this option.

Actually, generating a view on a body from a .CATProduct prevents many features from working properly:

- Positioning of the different parts in the assembly is not taken into account in the resulting view and parts might be superimposed,
- Changes such as rotation or translation in the assembly are not taken into account,
- Modification of an instance properties such as visibility or colors are not taken into account,
- **Overload properties** is disabled as it is linked to the assembly's properties,
- Creation of balloons is not possible,
- **Edit / Links** option references only two parts.

Moreover, multi-selecting a body in two different parts modifies the behavior of the **Modify links** and **Modify Projection Plane** according to the order of selection, since the .CATPart of the first selected body will be used as reference document and not the .CATProduct.

Clipping view



Put in no show dimension on non-visible geometry

Select this check box if you want to put automatically in no show mode the dimensions which are attached to geometry which is not visible in clipping views.

If you do not select this check box, dimensions are displayed in clipping views using the colors defined in **Tools -> Options -> Mechanical Design -> Drafting -> Dimension** tab, **Analysis display mode** area. For more information, refer to Analysis display mode.

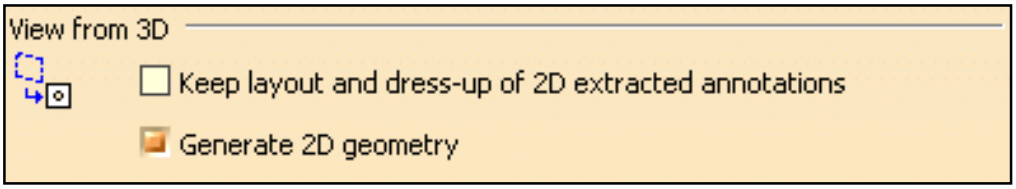
The color used depends on whether the dimension is interactive (that is, created manually) or generated automatically:

- interactive dimensions are displayed using the color defined for dimensions on non-visible geometry (light blue by default).
- generated dimensions are displayed using the color defined for dimensions generated from 3D constraints (light green by default).

Dimensions are put in no show mode only if none of the geometrical elements to which the dimension is attached are visible. If an element is visible while the other is not, the dimension is displayed using the color defined by the analysis display mode as specified above.

By default, this option is selected.

View From 3D



Keep layout and dress-up of 2D extracted annotations

Select this option if you want the layout and dress-up applied to 2D extracted annotations to be preserved when updating a view from 3D (Functional Tolerancing and Annotation workbench).

- 2D dress-up modifications are kept when updating design changes from 3D.
- Associativity of the annotations or their leader with the 3D geometry is taken into account.
- Associativity between annotations is taken into account.

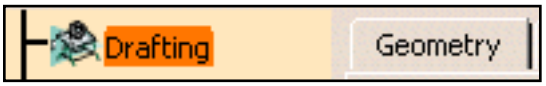
By default, this option is not selected.

Generate 2D geometry

Select this option if you want to generate 2D geometry (i.e. wireframe and 2D points) when generating a view from 3D from a layout view (2D Layout for 3D Design workbench).

By default, this option is selected.

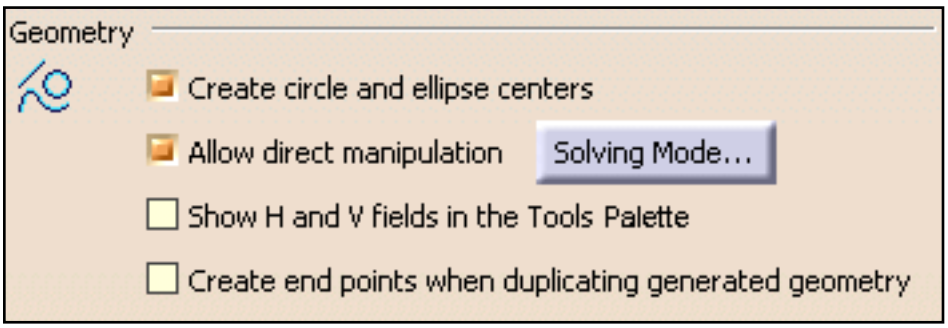
Geometry



This page deals with the following categories of options in the **Geometry** tab:

- Geometry
- Constraints creation
- Constraints Display
- Colors

Geometry



Create circle and ellipse centers

Specify whether you want to create centers when creating circles or ellipses. Clear this check box if you do not need to create circle and ellipse centers.

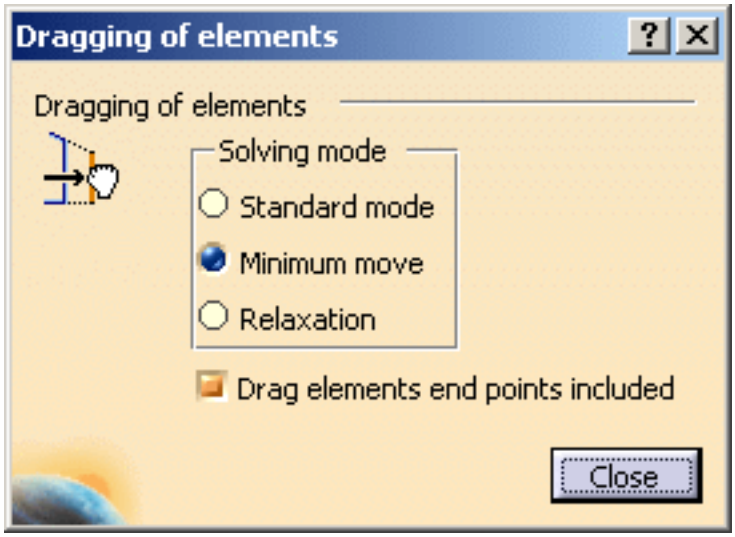
By default, this option is selected.

Allow direct manipulation

Select this check box to be able to move geometry using the mouse. When moving geometry, you can move either the minimum number of elements, the maximum number of elements, or still the minimum number by modifying the shape of elements, if needed.

By default, this option is selected.

Additionally, click the **Solving mode...** button to configure manipulation.



The dialog box that appears offers the following options as regards the solving mode:

Standard mode
You move as many elements as possible and also respect existing constraints.

Minimum move
You move as few elements as possible and also respect existing constraints.

Relaxation
You move elements by re-distributing them over the sketch, globally speaking. This method solves element moving by minimizing energy cost.

Furthermore, you can choose to drag elements along with their end points by selecting **Drag elements end points included**.

By default, **Minimum move** and **Drag elements end points included** are selected.

Show H and V fields in the Tools Palette

Select this check box to show the H and V fields in the Tools Palette when creating 2D geometry or when offsetting elements. Leaving the option unselected enables you to directly enter the value corresponding to the type of element you are creating: for example, the length when creating a line, the radius when creating a circle or the offset value when offsetting elements.

When a command (such as the Point creation command) does not have any parameters other than H and V, then these two fields will remain in the Tools Palette, whether you select this option or not.

By default, this option is not selected.

Create end points when duplicating generated geometry

Select this check box if you want to create end points when duplicating geometrical elements that were generated from the 3D.

By default, this option is not selected.

Constraints creation

Create detected and feature-based constraints

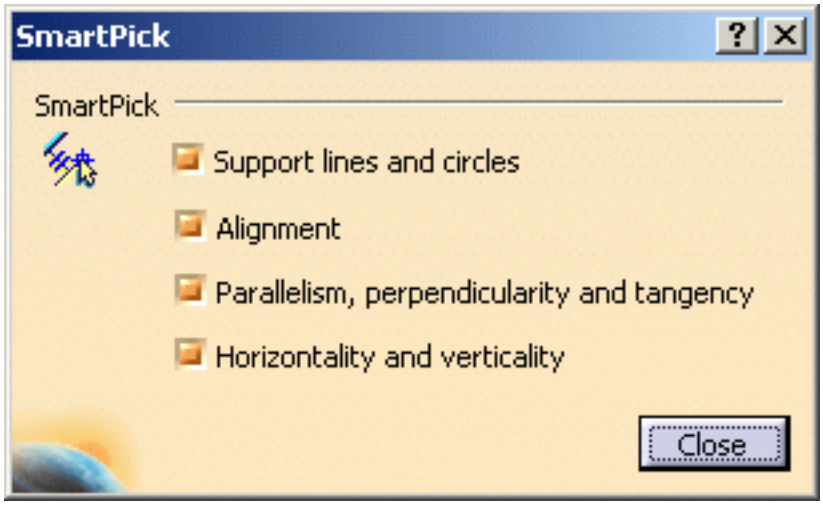
Select this check box if you want the geometrical or dimensional constraints detected by the SmartPick tool to be created.

This option is also available in the Tools toolbar through **Create detected constraints** . If you leave this check box unselected, **Create detected constraints** will be inactive by default. You will be able to activate it at any time.

By default, this option is not selected.

SmartPick... *(button)*

As you create more and more elements, SmartPick detects multiple directions and positions, and more and more relationships with existing elements. This may lead to confusion due to the rapid highlighting of several different detection possibilities as you point the cursor at different elements in rapid succession. Consequently, you can decide to filter out undesired detections by clicking the **SmartPick...** button.



The SmartPick dialog box provides these options:

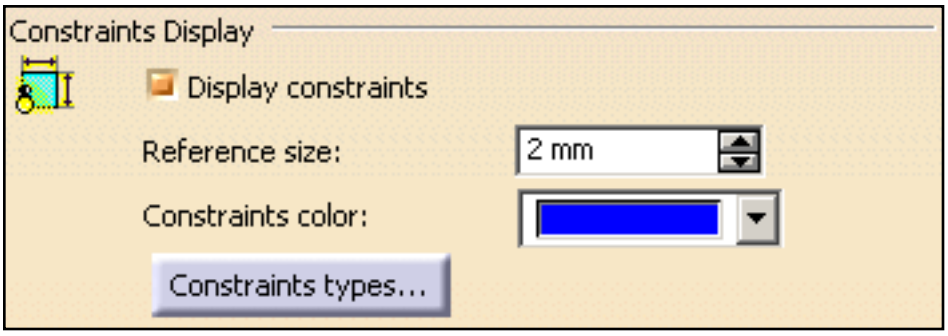
- **Support lines and circles**
- **Alignment**
- **Parallelism, perpendicularity and tangency**
- **Horizontality and verticality**

Clear the check boxes corresponding to the elements you do not wish to detect when sketching.

Clearing all check boxes in the SmartPick dialog box disables SmartPick completely. This will be particularly useful when your screen is full of elements: in this case, disabling SmartPick lets you concentrate only on the geometry.

By default, all options are selected.

Constraints Display



Display constraints

Select this check box to visualize the logical constraints specific to the elements. Note that if the **Display constraints** option is cleared, the other options in this category are not available.

By default, this option is selected.

Reference size

Specify the size that will be used as a reference to display constraints symbols. Changing this reference size will modify the size of all constraints representations.

By default, the reference size is 2 mm.

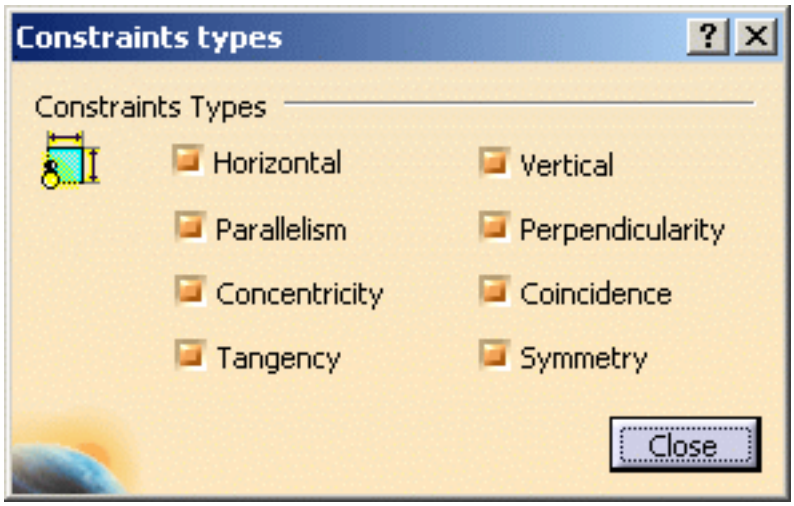
Constraints color

Choose the color that will be used to display constraints.

By default, the color for constraints is blue.

Constraints types... *(button)*

Click this button to define which types of constraints you will visualize as you create the geometry.



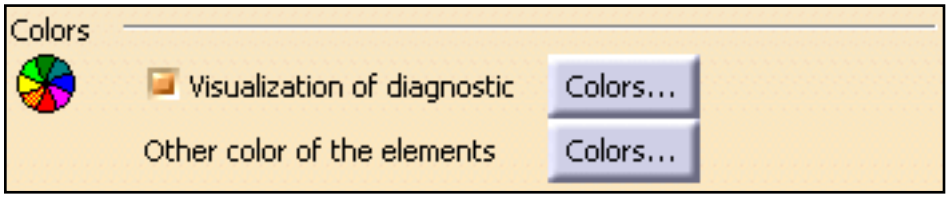
The Constraints types dialog box provides these options:

- **Horizontal**
- **Vertical**
- **Parallelism**
- **Perpendicularity**
- **Concentricity**
- **Coincidence**
- **Tangency**
- **Symmetry**

Clear the check boxes corresponding to the types of constraints you do not want to visualize as you create the geometry.

By default, all options are selected.

Colors



Two types of colors may be applied to sketched elements. These two types of colors correspond to colors illustrating:

- Graphical properties
  Colors that can be modified. These colors can therefore be modified using the **Tools->Options** dialog box.

OR

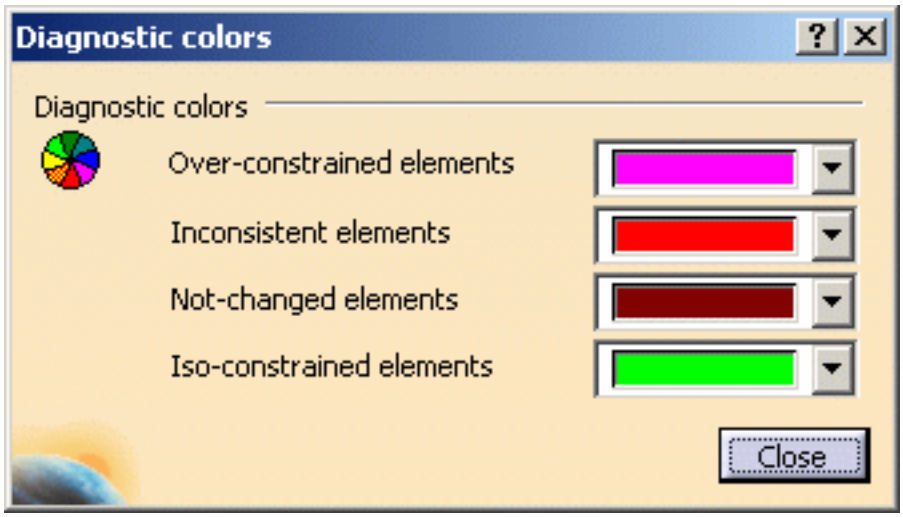
- Constraint diagnosis
  Colors that represent constraint diagnoses are colors that are imposed to elements whatever the graphical properties previously assigned to these elements and in accordance with given diagnoses. As a result, as soon as the diagnosis is solved, the element is assigned the color as defined in the **Tools -> Options** dialog box.

Visualization of diagnosis

Select this option if you want over-constrained, inconsistent, not-changed or iso-constrained elements to be identified using specific colors.

By default, this option is selected.

Then, click the **Colors...** button to configure these colors.



In the dialog box that appears, you can configure colors for the following types of elements:

Over-constrained elements
The dimensioning scheme is over-constrained: too many dimensions were applied to the geometry.

Inconsistent elements
At least one dimension value needs to be changed. This is also the case when elements are under-constrained and the system proposes defaults that do not lead to a solution.

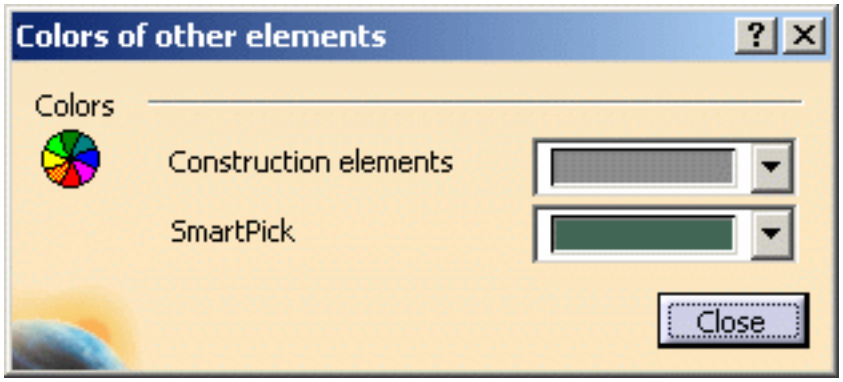
Not-changed elements
Some geometrical elements are over-defined or not-consistent. As a result, geometry that depend(s) on the problematic area will not be recalculated.

Iso-constrained elements
All the relevant dimensions are satisfied. The geometry is fixed and cannot be moved from its geometrical support.

By default, colors are configured as shown above.

Other color of the elements

Click the **Colors...** button to configure the colors of other elements.



In the dialog box that appears, you can configure colors for the following types of elements:

Construction elements
Specifies the color used for construction elements.

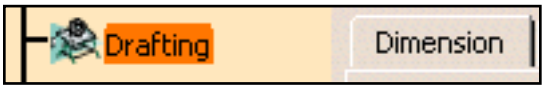
A construction element is an element that is internal to, and only visualized by, the sketch. This element is used as positioning reference. It is not used for creating solid primitives.

SmartPick
Specifies the color used for SmartPick elements and symbols.

By default, colors are configured as shown above.

When opening a drawing, colors are not recomputed. Colors will not be displayed until you create another element or move the geometry.

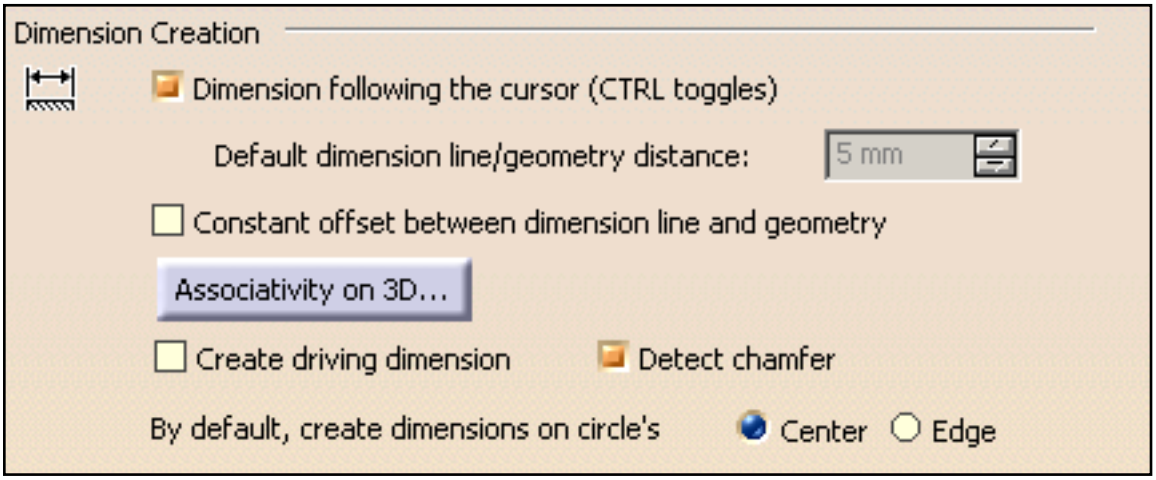
Dimension



This page deals with the following categories of options in the **Dimension** tab:

- Dimension Creation
- Move
- Line-Up
- Analysis Display Mode

Dimension Creation



Dimension following the mouse (ctrl toggles)

Select this option to specify that the dimension line should be positioned according to the cursor, following it dynamically during the creation process.

By default, this option is selected.

Default dimension line/geometry distance

If you position the dimension according to the cursor, you can define the value at which the dimension is created. If you create associativity between the dimension and the geometry, you can define the value at which the dimension will remain positioned.

By default, this option is set to 5 mm.

Constant offset between dimension line and geometry

Select this option to specify that the distance between the created dimension and the geometry should remain the same when moving the geometry.

By default, this option is not selected.

Associativity on 3D... *(button)*

If you click the button, the Dimensions associativity on 3D dialog box appears. These options are used in a Generative Drafting context, and specify how associativity between dimensions and the 3D part or product is handled.

When they are associative, dimensions are linked to the 3D part or product. As a result, when you update the drawing, the dimension is automatically re-computed. When dimensions are not associative, you need to re-create them after performing an update.

Associativity on 3D

When creating dimensions in a Generative Drafting context, specify whether and how dimensions should be associative with the 3D part/product:

- **Only create non-associative dimensions**: dimensions will never be associative with the 3D.
- **Allow non-associative dimensions**: dimensions are associative with the 3D whenever possible, but dimensions that cannot be associative with the 3D are also created.
- **Never create non-associative dimensions**: only dimensions that can be associative with the 3D are created.

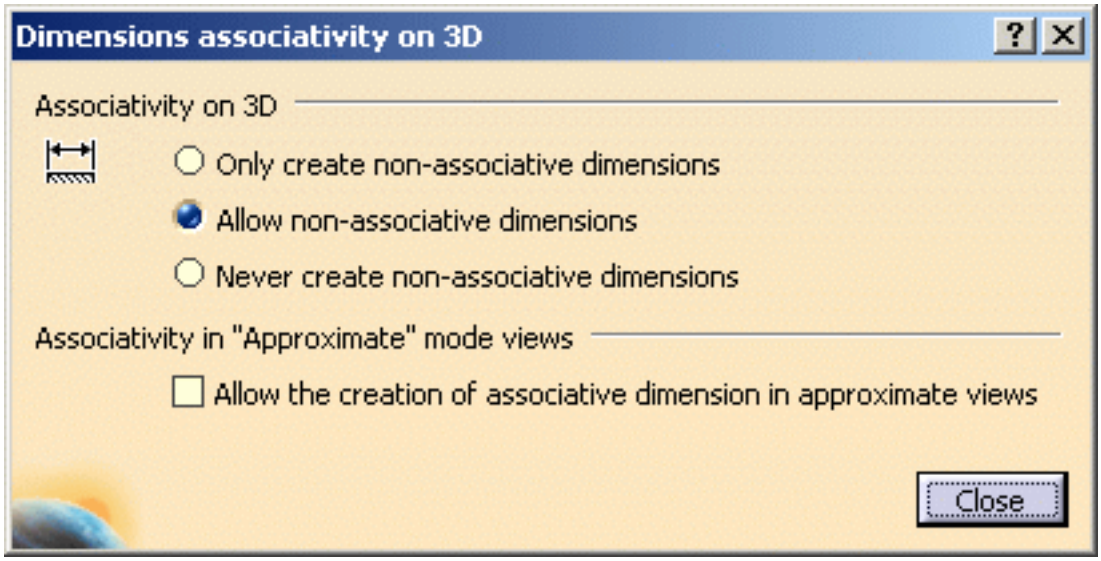
By default, **Allow non-associative dimensions** is selected.

Associativity in "Approximate" mode views

Select the **Allow the creation of associative dimension in approximate views** option if you want the option chosen in the Associativity on 3D section above to be applied to dimensions in Approximate views (i.e. views generated using the Approximate view generation mode). This option only applies to the following dimension creation commands: Dimensions, Chained Dimensions, Cumulated Dimensions, Stacked Dimensions, Distance/Length Dimensions, Angle Dimensions, Radius Dimensions, Diameter Dimensions, Chamfer Dimensions, Dimension Systems. It does not apply to other dimension types, nor to driving dimensions.

If this option is not selected, dimensions will not be associative in Approximate views (as was the case prior to V5R15), which minimizes memory requirements.

By default, this option is not selected.



Create driving dimensions

Select this option if you want newly created dimensions to drive the geometry.

A new field will appear in the Tools Palette during the creation process, allowing you to enter the driving dimension value.

By default, this option is not selected.

Detect chamfer

Select this option to activate chamfer detection: this will lets you create chamfer dimensions in a single click.
As chamfer detection may slow performance down, you may want to deactivate this option for large products or assemblies.

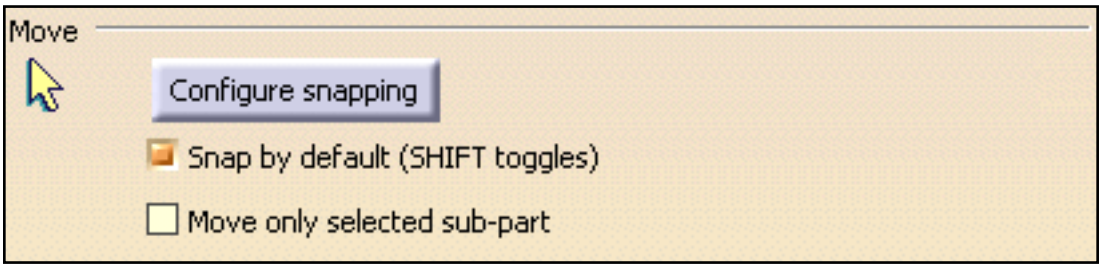
By default, this option is selected.

By default, create dimensions on circle's

Specify whether the dimension you will create between a circle and another element should be on the circle center or on the circle edge.

By default, the **Center** option is selected.

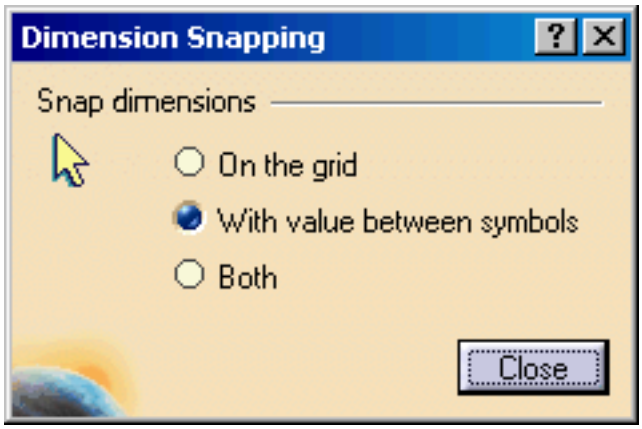
Move



Configure snapping

Click the **Configure snapping** button to define how snapping should be performed. In the dialog box that appears, specify whether the dimension should be snapped on the grid, whether the dimension value should be located at its default position between symbols (it will work only if the cursor is between the symbols), or both.

By default, **Value between symbols** is selected.



Snap by default (SHIFT toggles)

Select this option to activate snapping by default when creating or handling dimensions. Pressing the Shift key will enable you to temporarily deactivate snapping. Note that snapping is performed according to the options set in the **Configure snapping** dialog box.

With this option unselected, snapping is not active. In this case, pressing the Shift key will enable you to temporarily activate it.

By default, this option is selected.

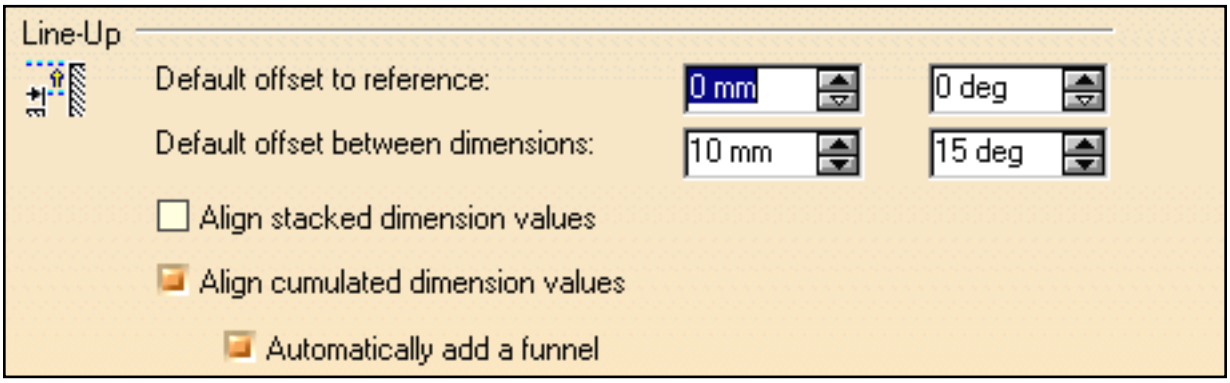
Before V5 R15, this option used to be called Activate Snapping (SHIFT toggles).

Move only selected sub-part

Select this option if you want to move only a dimension sub-part (text, line, etc.).

By default, this option is not selected.

Line-Up



You can organize dimensions into a system with a linear offset. The offset will align the dimensions to each other as well as the smallest dimension to the reference element.

Default offset to reference

This allows you to set the offset between the smallest dimension and the reference element.

By default, the offset is 0 mm and 0 deg.

Default offset between dimensions

This allows you to set the offset between dimensions.

By default, the offset is 10 mm and 15 deg.

Align stacked dimension values

Lets you align all the values of a group of stacked dimensions on the value of the smallest dimension of the group.

By default, this option is not selected.

Align cumulated dimension values

Lets you align all the values of a group of cumulated dimensions on the value of the smallest dimension of the group.

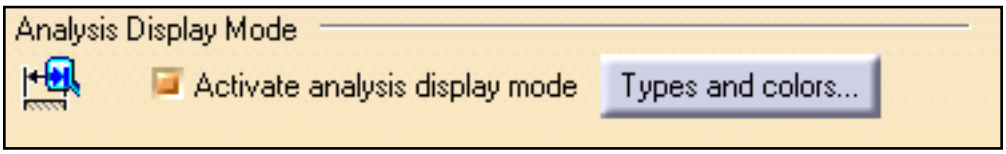
By default, this option is selected.

Automatically add a funnel

Whenever the value of a cumulated dimension requires a funnel to be displayed correctly, lets you have one added automatically.

By default, this option is selected.

Analysis Display Mode

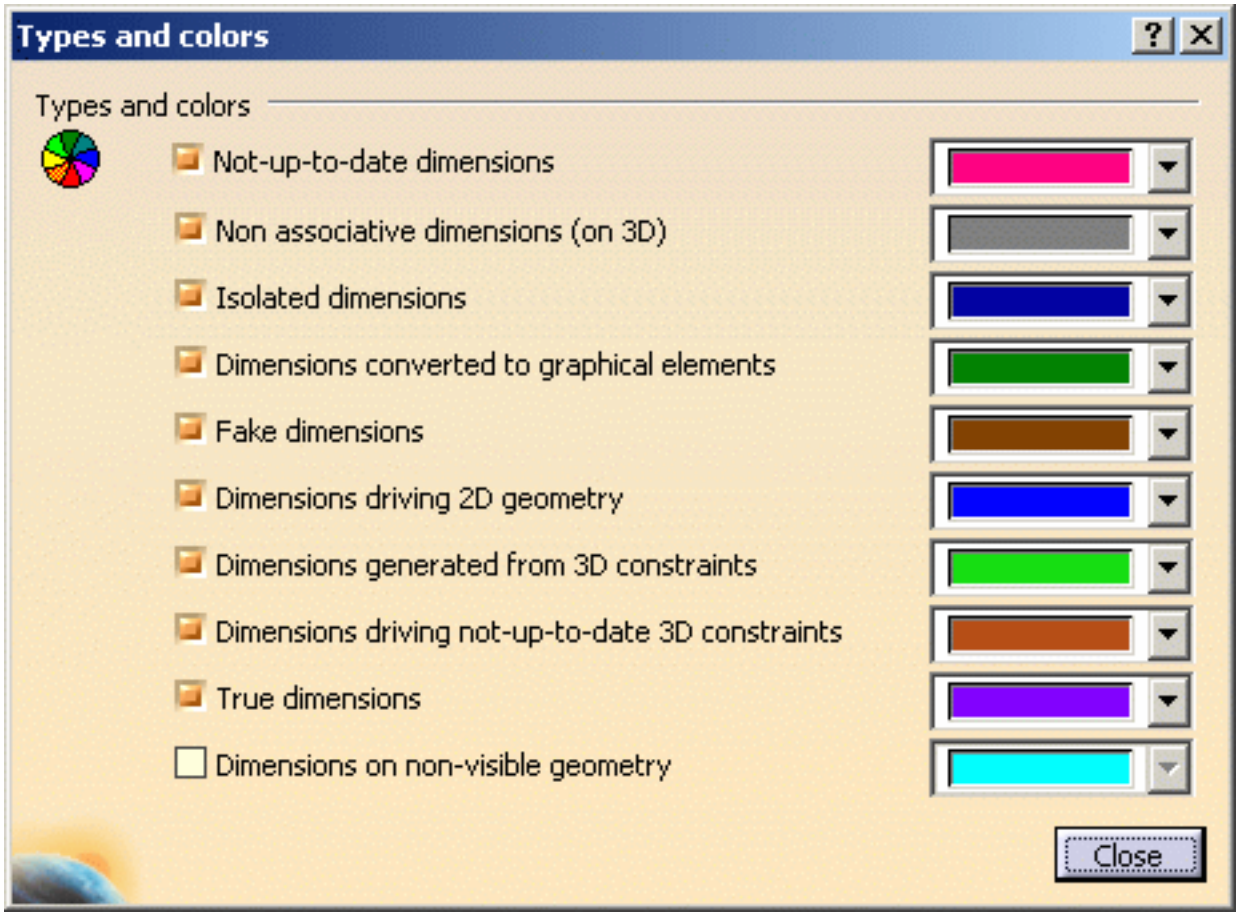


Activate analysis display mode

Select this option to display dimensions using different colors according to their status (not-up-to-date, isolated, fake, etc.).

By default, this option is selected.

Additionally, click the **Types and colors** button to customize the colors that will be used. The Types and colors of dimensions dialog box lets you assign the desired color(s) to the selected dimension types. You will then be able to visualize the different types of dimensions using their assigned colors.



By default, colors are configured as shown above.

Manipulators

This page deals with the following categories of options in the **Layout** tab:

- Manipulators
- Dimension Manipulators

Manipulators

These settings can be used for any type of manipulator (texts, leaders, center lines, dimensions and so forth).

Reference size

Specify the reference size that should be used for manipulators. In the case of texts, for example, this reference size corresponds to the diameter of the rotation manipulators.

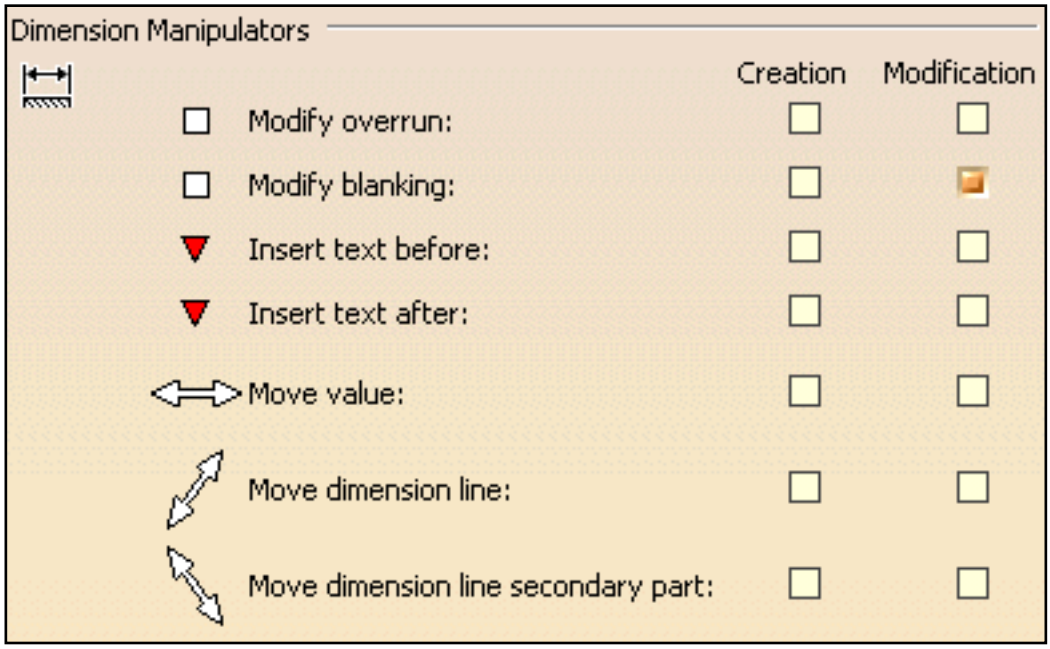
By default, the reference size is set to 2mm.

Zoomable

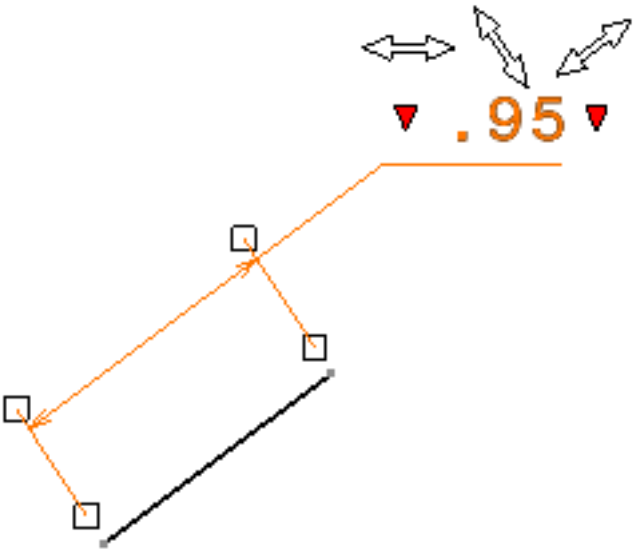
Check this box if you want to make manipulators zoomable.

By default, this option is selected.

Dimension Manipulators



These options let you define which manipulators you will visualize and therefore use when creating and/or modifying dimensions:



Modify overrun

If you drag select one overrun manipulator, both overrun extension lines are modified. To modify only the selected overrun extension line, use the **Ctrl** key. You can also double-click on the manipulator and enter the new value in the dialog box that appears.

By default, the **Creation** and **Modification** options are not selected.

Modify blanking

If you drag select one blanking manipulator, both blanking are modified. To modify only the selected

blanking, use the **Ctrl** key. You can also double-click on the manipulator and enter the new value in the dialog box that appears.

By default, the **Creation** option is not selected, and the **Modification** option is.

Insert text before

Allow inserting a text before, without using the Properties dialog box. For this, you will click on the manipulator and enter the new text in the dialog box that appears.

By default, the **Creation** and **Modification** options are not selected.

Insert text after

Allows inserting a text after, without using the Properties dialog box. For this, you will click on the manipulator and enter the new text in the dialog box that appears.

By default, the **Creation** and **Modification** options are not selected.

Move value

Lets you move only the dimension value.

By default, the **Creation** and **Modification** options are not selected.

Move dimension line

Lets you move only the dimension line by dragging it to the new location.

By default, the **Creation** and **Modification** options are not selected.

Move dimension line secondary part

Lets you move only the dimension line secondary part by dragging it to the new location.

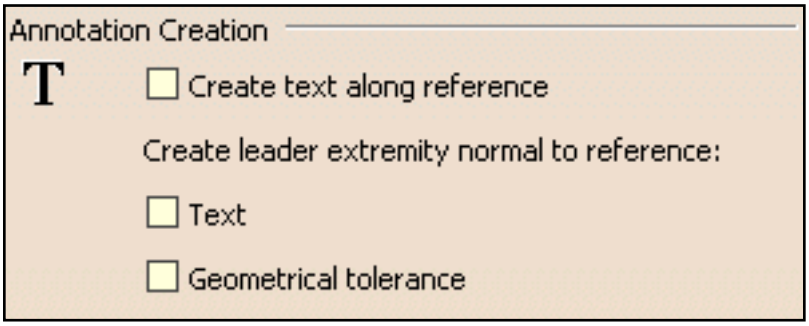
By default, the **Creation** and **Modification** options are not selected.

Annotation and Dress-Up

This page deals with the following categories of options in the **Layout** tab:

- Annotation Creation
- Move
- 2D Component Creation
- Balloon Creation
- Table

Annotation Creation



In order for these options to be taken into account, the **Snap by default (SHIFT toggles)** box must be selected. Note that the option selected in the **Snapping Configuration for Annotations** dialog box will be taken into account. See the Move section.

These options are taken into account only when creating annotations, not when adding a reference line, for example.

Create text along reference

Select this option if you want to create annotation texts along a reference direction. For example, if you select a line when creating a text, the text will be oriented parallel to the line.

By default, this option is not selected.

Text

Select this option if you want to create the extremity of text leaders normal to a reference direction. For example, if you select a line when creating a text with leader, the leader will be normal to the line.

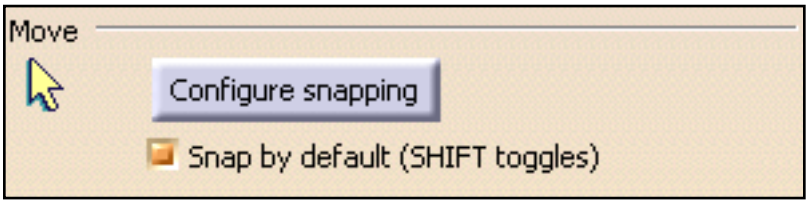
By default, this option is not selected.

Geometrical tolerance

Select this option if you want to create the extremity of geometrical tolerance leaders normal to a reference direction. For example, if you select a line when creating a geometrical tolerance, the leader will be normal to the line.

By default, this option is not selected.

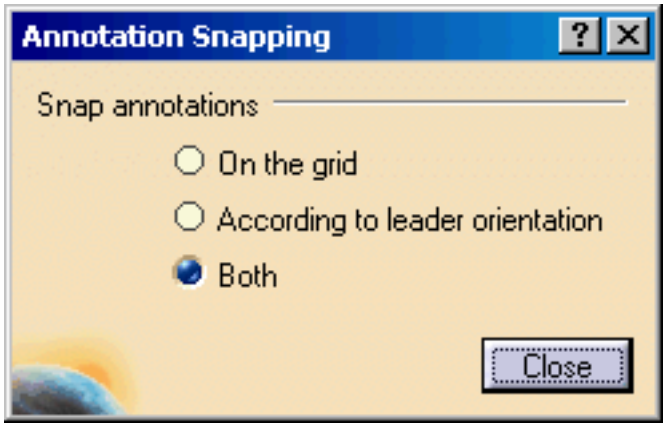
Move



Configure snapping

Click the **Configure snapping** button to define how snapping should be performed. In the dialog box that appears, specify whether you want annotations to be snapped on the grid, according to the leader orientation, or both. This will apply to the annotations selected in the Annotation Creation area.

By default, **Both** is selected.



Snap by default (SHIFT toggles)

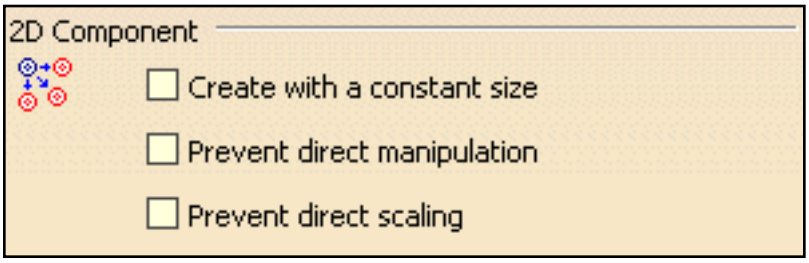
Select this option to activate snapping by default when creating or handling annotations. Pressing the Shift key will enable you to temporarily deactivate snapping. Note that snapping is performed according to the options set in the **Snapping Configuration for Annotations** dialog box.

With this option unselected, snapping is not active. In this case, pressing the Shift key will enable you to temporarily activate it.

By default, this option is selected.

- From V5 R15 onwards, the Snap to Point option on the General tab no longer applies to annotations. Only this option is taken into account.
- Before V5 R15, this option used to be called Activate Snapping (SHIFT toggles). It has been renamed for clarity reasons.

2D Component Creation



Create with a constant size

Select this option if you want all 2D component instances to have the same size when you create them, no matter what the view scale is.

This lets you create 2D component instances whose size is independent from the view scale so that they always look the same. You can use them as symbols, for example.

If you want to use as symbols 2D components with text, activate both the **Create with a constant size** setting and the **Apply Scale** property for the text (in **Edit -> Properties**): the size of both the 2D component and its text will then be independent from the view scale.

By default, this option is not selected.

Prevent direct manipulation

Select this option to make it impossible to move 2D components using the mouse.

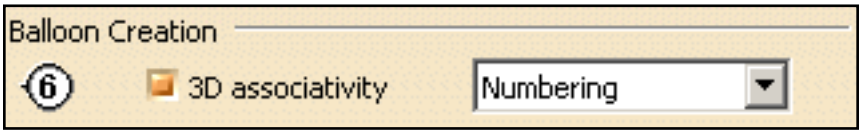
By default, this option is not selected.

Prevent direct scaling

Select this option to make it impossible to scale 2D components using the mouse.

By default, this option is not selected.

Balloon Creation



You can specify what kind of balloons you want to create (using the Balloon command from the Annotation toolbar) or to generate (using the Generate Balloons command from the Generation toolbar).

3D associativity

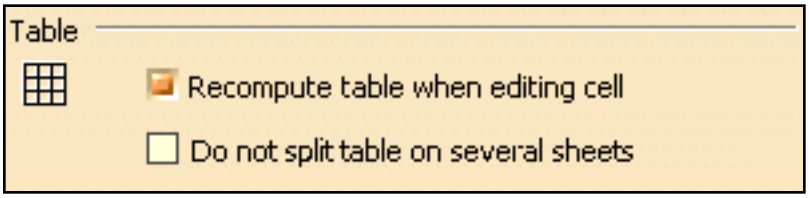
Select this option to indicate that you want to associate balloons with information from the 3D.

By default, this option is selected.

Additionally, select from the list the kind of balloons you want to create or generate: the numbering of parts within an assembly (default option), the instance name or the part number.

By default, **Numbering** is selected.

Table



Recompute table when editing cell

Select this option if you want a table to be resized while you are editing a cell. If it is not selected, the table is resized after closing the Text Editor.

By default, this option is selected.

Do not split table on several sheets

This option is available only when certain licenses are installed.

With certain licenses, tables are split automatically on several sheets. Select this option if you want to keep tables on a single sheet.

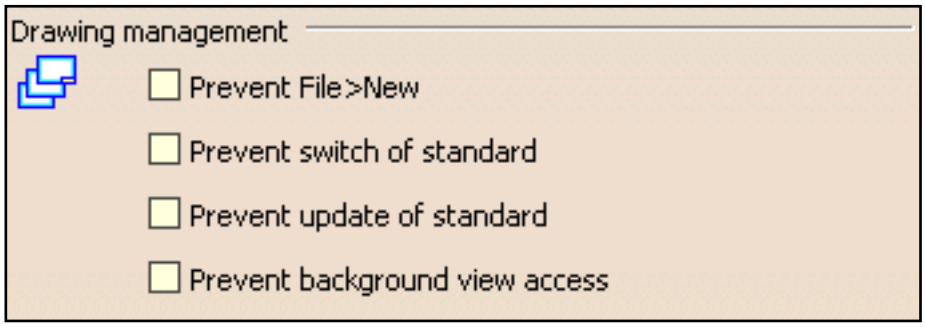
By default, this option is not selected.

Administration

This page deals with the following categories of options in the **Administration** tab:

- Drawing management
- Style
- Generative view style
- Dress-up

Drawing management



Prevent File>New

Select this option to make it impossible to create drawings using the **File -> New** command. All drawings will be created using the **File -> New From...** command instead.

By default, this option is not selected.

Prevent switch of standard

Select this option to make it impossible to change standards, i.e. to use a standard other than the one currently defined in the Page Setup dialog box.

By default, this option is not selected.

Prevent update of standard

Select this option to make it impossible to update standards for the current document in the Page Setup dialog box.

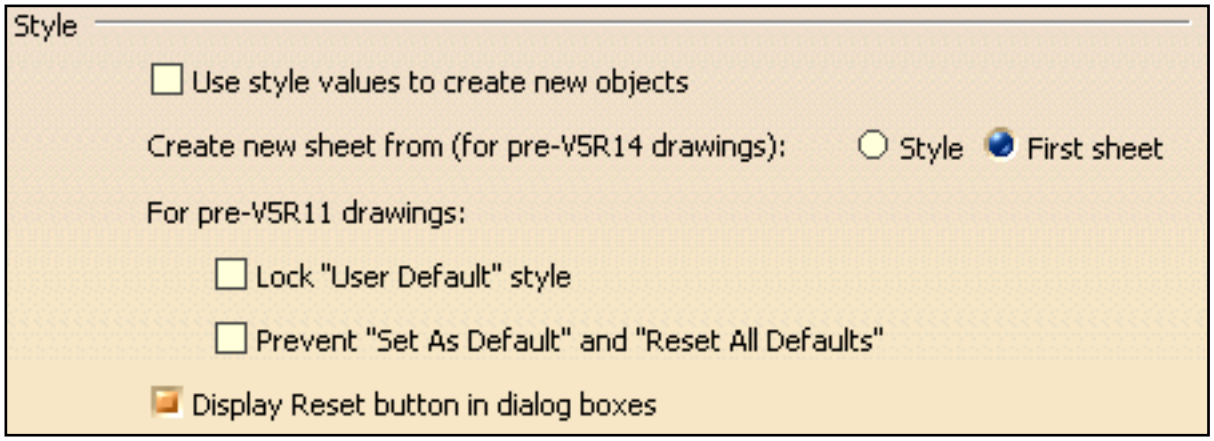
By default, this option is not selected.

Prevent background view access

Select this option to make it impossible to access the background view.

By default, this option is not selected.

Style



Use style values to create new objects

Select this option if you want dialog boxes, Properties toolbars and the Tools Palette to be pre-filled with custom style values (as defined in the Standards Editor) when creating new annotations. In this case, Properties toolbars and the Tools Palette will be disabled during the creation of the annotation.
If you leave this box unchecked, annotation dialog boxes, Properties toolbars and the Tools Palette will be pre-filled with the last entered values (except for Texts, Texts with leader, Balloons and Datum features). In this case, Properties toolbars and the Tools Palette will be active during the creation of the annotation.

If you select this option, you will be able to reset the current style values in dialog boxes at any time using the **Reset** button unless it is disabled.

By default, this option is not selected.

Create new sheet from (for pre-V5R14 drawings)

This option lets you specify if the properties used for creating new sheets should be those defined in the standards or those defined in the first sheet of a drawing. These properties are the scale and the projection method (first or third angle).

Select **Style** if you want the sheet to use the style defined in the standards (in **Tools -> Standards -> Drafting -> [StandardName] -> Styles -> Sheet**).

Select **First sheet** if you want the sheet to use the properties defined in the first sheet of a drawing. For example, you can use this option if you use an existing drawing to create a new one (i.e. when you want the new drawing to have the same properties as the existing drawing).

This option applies only to drawings created with versions prior to V5R14.

By default, **First sheet** is selected.

Lock "User Default" Style

Select this option to make it compulsory to use User Defaults (i.e., user-defined values set as default). The **Styles** drop-down list will be set to **Only User Defaults** and will be inactive so that **Original Defaults** or **User Defaults** cannot be selected.

This option applies only to drawings created with versions up to V5 R10 whose standard has NOT been updated or changed in V5 R11 and later.

By default, this option is not selected.

Prevent "Set As Default" and "Reset All Defaults"

Select this option to use the current defaults and to make it impossible to create, change and reset user defaults (i.e. user-defined values). This disables the **Set as Default** and the **Reset All Defaults** commands.

This option applies only to drawings created with versions up to V5 R10 whose standard has NOT been updated or changed in V5 R11 and later.

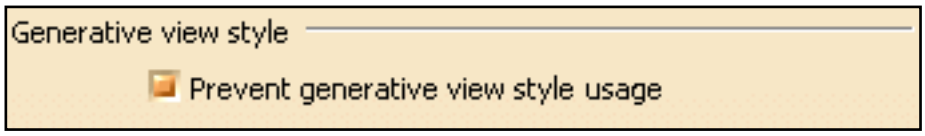
By default, this option is not selected.

Display Reset button in dialog boxes

Select this option to display the **Reset** button in dialog boxes. Deselecting this option hides the **Reset** button in dialog boxes and disables the **Reset** functionality.

By default, this option is selected.

Generative view style



Prevent generative view style usage

This option is used for the Generative Drafting workbench only.

Select this check-box if you do not want to use generative view styles when creating or updating views. In this case, you will not be able to select a generative view style after having selected a view creation command, which means that the Generative View Style toolbar will not be displayed. (In the case of advanced front views, it is the Generative view style list in the View Parameters dialog box which will not be displayed).

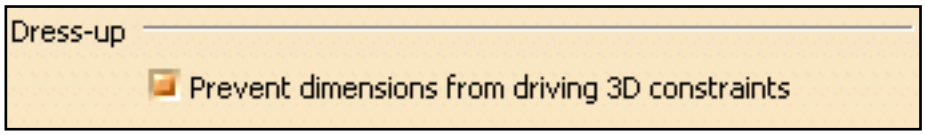
Note that this option also drives view generation parameters when updating views, whether the view was created using generative view styles or not. In this case, there are two possibilities:

- If you created the view using generative view styles (i.e. with this check-box cleared) and then select this check-box, then, at the next update, the view properties (available via **Edit -> Properties**) will be used for all overloaded parameters (an asterisk * appears in the Properties dialog box in front of the parameters which are considered as being overloaded) and the view settings (available via **Tools -> Options -> Mechanical Design -> Drafting - > View** tab) will be used for the others.
- If you created the view without generative view styles (i.e. with this check-box selected) and then clear this check-box, then, at the next update, all parameters available in the view properties (available via **Edit -> Properties**) will be considered as being overloaded and will be used. To make sure that generative view styles are used in this case, click the **Reset to style values** button in the Properties dialog box.

For more information, refer to the Generative View Styles chapter in the *Generative Drafting User's Guide*.

By default, this option is selected.

Dress-up



Prevent dimensions from driving 3D constraints

Select this option to make it impossible to modify a 3D constraint via a 2D dimension that was generated from it.

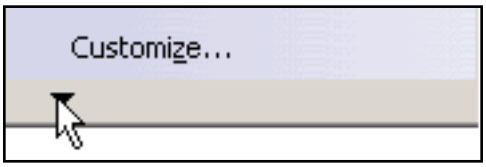
By default, this option is selected.

Customizing Toolbars

You can customize the appearance of some fields in the following properties toolbars: Styles, Graphic Properties, Text Properties and Dimension Properties.

1. Right-click the toolbar field you want to customize. A contextual menu is displayed.

2. If necessary, scroll down this contextual menu to display the toolbar customization options.



The customization options that you can apply to the selected field are displayed.

The options available depend on the selected field. For more information on what options are available for each field, see the table below.

3. Click the option you want. Depending on the option you selected, the corresponding dialog box appears.

4. Enter the appropriate value in the dialog box.

- **Set text width**: sets the width used to display the field in the toolbar, in number of characters to be displayed (based on 'W').
- **Set list width**: sets the width used to display the drop-down list, in number of characters to be displayed (based on 'W').
- **Set list height**: sets the height used to display the list, in number of lines to be displayed (up and down arrows will make it possible to scroll within the list).
- **Icons display**: defines whether icons should be displayed in this field, or only in the list, when the list is collapsed.
- **Precision**: sets the precision used to display a numerical value in this field, in number of digits after the separator.

5. Click **OK** to validate.

The table below indicates which fields you can customize in each toolbar, along with what you can customize for each field.

	Set text width	Set list width	Set list height	Icons display	Set precision
Style toolbar					
Style	Yes	Yes	Yes	Yes	No
Text Properties toolbar					
Font Name	Yes	Yes	Yes	Yes	No
Font Size	Yes	Yes	Yes	No	Yes
Dimension Properties toolbar					
Tolerance Description	Yes	Yes	Yes	Yes	No
Tolerance	Yes	Yes	Yes	No	No
Numerical Display Description	Yes	Yes	Yes	No	No
Precision	Yes	Yes	Yes	No	Yes
Graphic Properties toolbar					
Color	Yes	No	No	No	No
Thickness	Yes	No	Yes	No	No
Linetype	Yes	No	Yes	No	No
Point type	Yes	No	Yes	No	No

Administration Tasks

In the 2D Layout for 3D Design workbench, administrators can manage and customize standards (international or company standards), in the Standards Editor. It lets administrators set the standards used for such elements as dress-up, dimensions, annotations as well as set the styles that will be used as defaults for element properties in the 2D Layout for 3D Design workbench.

A number of standards are provided with the application (in .xml files):

- Some are specifically intended for Drafting, such as JIS, ISO and ANSI.
- Some are specifically intended for 2D Layout for 3D Design (as well as for the Functional Tolerancing and Annotation workbench), based on the Drafting standards: these specific standards are suffixed with _3D (such as JIS_3D or ISO_3D). In these specific standards, some properties such as the colors have been customized for optimized display.
  For 2D Layout for 3D Design, we recommend that you customize these specific xxx_3D standards, or that you create your own standards based on these specific standards.

Since all standards are either Drafting, or Drafting-based standards, administration tasks are actually documented in the *Interactive Drafting User's Guide*.

As a 2D Layout for 3D Design administrator, an important element you need to define is the view box, which gathers all the data needed to fully define the layout of a view set in the 2D window, as well as the position of each view in the 3D space. For specific information regarding the view box, refer to Layout Views Customization (still in the Administration Tasks chapter in the *Interactive Drafting User's Guide)*.

The information you will find in this section is listed below:

Before You Begin
Administering Standards and Generative View Styles
Upgrading Standard Files from Previous Releases
Setting Standard Parameters and Styles

Before You Begin

About Drafting Standards

When users modify the properties of an element in the Interactive Drafting workbench, the modifications are only applied to the selected element, in the current drawing. Standard files let administrators set the properties of an element so that they will be applied to all elements of the same type in a drawing, as well as in all drawings which use a given standard.

A standard file is an XML file which makes it possible to customize globally, for a CATDrawing, the appearance and behavior of drafting elements.

With standard files, administrators can:

- set standard styles that will be used as default values when creating new elements, i.e.:
  - define sheet styles
  - define geometry styles
  - define annotation styles
  - define dimension styles
  - define dress-up and dress-up symbols styles
  - define callout styles
- set standard parameters, i.e.:
  - control the user interface, with general parameters to restrict the values of some element properties
  - customize dimensions
  - customize annotations
  - customize dress-up elements
  - define new dimension tolerance formats
  - define new dimension value formats
  - control pre-defined formats for tolerance and dimension values
  - control view generation parameters
  - customize fixed-size frames
  - customize line thickness
  - customize patterns

Once defined, a format is applied to elements as a property.

The format of the standard file has been changed from V5 R9 onwards. If you were using a customized CATDrwStandard file on a previous release (up to V5 R8), you need to upgrade the standard file to the new XML format.

Management of Drafting Standards

Standalone drawings

When users create a CATDrawing document (**File -> New**), they specify the standard that will be associated with this document. The values of the parameters in the specified standard file are then copied into the CATDrawing document. Each drawing contains a copy of the standard and is therefore standalone. This makes it possible for users, projects, or companies to exchange CATDrawing documents without needing to send the standard file along.

Administrator-controlled access and modification

The administrator defines and controls the location of the standard files as well as the ability to define new standards, or to modify existing standards. For example, the administrator can define a single standard, and prevent users from modifying it.

Seven standard files available by default

By default, seven standard files are provided: four for each of the international standards available when creating a new CATDrawing file, and three for use with other workbenches. These files are located in **install_root/resources/standard/drafting**.

- Some are specifically intended for Drafting:
  - ISO.xml
  - ANSI.xml
  - JIS.xml
  - ASME.xml

- Some are specifically intended for the 2D Layout for 3D Design and Functional Tolerancing and Annotation workbenches. They are based on the Drafting standards and are suffixed with _3D. In these specific standards, the colors, for example, have been customized for optimized display.
  - ASME_3D.xml
  - ISO_3D.xml
  - JIS_3D.xml

Administrators can add as many standard files as needed. Refer to Administering Standards for more information.

Editing the standard file

The standard files can be edited using an interactive editor. This editor provides an easy-to-use graphic interface to let administrators customize the parameters included in the standard files. For information on how to customize these parameters, refer to Setting Standard Parameters.

The interactive editor is available in **Tools -> Standards**. (It is the same editor with which you can customize generative view styles). For more information on how to use this editor, refer to the Customizing Standards chapter in the *Infrastructure User's Guide*.

Make sure you use the Standards editor available in **Tools -> Standards** when modifying and customizing the XML standard files. Using other editors (such as text editors) may alter the consistency of the standard file, and may make the standards XML files unusable.

Switching to another standard

When several standards are defined, users can switch a drawing to another standard. Refer to Switching to Another Standard.

Updating the standard of a drawing

When a standard file is modified, users need to explicitly update the drawings which use this standard. Note that only standard parameters are affected by this update, not styles. Refer to Updating the Standard of a Drawing.

Administering Standards and Generative View Styles

This task documents the administration of both standards (Interactive Drafting workbench) and generative view styles (Generative Drafting workbench), as the procedure is basically the same whether you are administering standards or generative view styles. When applicable, differences will be notified. The examples provided in this task specifically deal with the administration of standard files.

For more information on customizing and administering generative view styles, refer to the Administration Tasks chapter in the *Generative Drafting User's Guide*.

Location of standard or generative view style files

The location of standard files or generative view style files is defined by two environment variables which can be set during installation or modified afterwards:

Variable name	Description
CATCollectionStandard	Path and name of the directory (or directories) which contains: • the **drafting** sub-directories (which themselves contain the customized drafting standards). **It is in these drafting sub-directories that you should add the drafting standards customized for a company, project or user.** • the **generativeparameters** sub-directories (which themselves contain the customized generative view styles). **It is in these generativeparameters sub-directories that you should add the generative view styles customized for a company, project or user.**
CATDefaultCollectionStandard	Path and name of the directory (or directories) which contains: • the **drafting** sub-directories (which themselves contain the predefined drafting standards delivered by Dassault Systemes). • the **generativeparameters** sub-directories (which themselves contain the predefined generative view styles delivered by Dassault Systemes). The default location for this directory (set during the installation process) is the installation directory **install_root\resources\standard**.

Setting the location of standard files

Refer to the Administration Tasks chapter in the *Generative Drafting User's Guide* for specific information on how to set the location of generative view style files.

There are two possibilities:

- If you want to place all customized drafting standards in a custom directory, named **mydirectory** for example, you need to proceed as follows:

  1. Create a directory named as you like (**mydirectory**, for example).
  2. Create a sub-directory under this directory, which needs to be named **drafting**.
  3. Place the XML files containing your customized drafting standards in **mydirectory\drafting**.

- If you have not yet customized your XML standard files, then proceed as follows:

  1. Create a directory named as you like (**mydirectory**, for example).
  2. Create a sub-directory under this directory, which needs to be named **drafting**.
  3. Set the CATCollectionStandard variable to **mydirectory.** After you have customized the XML standard files, the standard editor will then save them in **mydirectory\drafting**.

- If the CATDefaultCollectionStandard and the CATCollectionStandard variables both contain an identically-named standard, it is always the standard found in CATCollectionStandard which will be used.
- If two directories referenced by the CATCollectionStandard and/or CATDefaultCollectionStandard variables contain identically-named standard files, it is always the standard in the directory listed first which will be used.

Customizing and defining standards or generative view styles

To edit and save standard files or generative view style files in **Tools -> Standards**, you must be running the V5 session in administrator mode (-admin).

The recommended method for customizing standard files or generative view style files is the following:

1. You need to work in administrator mode. To do this, proceed as follows:

   a. Set up the CATReferenceSettingPath variable.

   b. Start a V5 session using the **-admin** option.

   For more information, refer to the Managing Environments chapter in the *Infrastructure Installation Guide*.

2. Set up the CATCollectionStandard environment variable as explained above.

   If none of the conditions are respected, a warning message will appear to let you know that you will neither be able to modify nor save the XML files.

3. Modify the Drafting standards or the generative view styles as appropriate.

4. Use the **Save As** or the **OK** button to store your modifications.

5. To exit, use the **Cancel** button.

Once the standard files or the generative view style files have been customized and saved, they can be used in a V5 session in normal mode.

Availability of standard switch and update

Note: The information provided below does not apply to generative view styles.

Using the settings available in **Tools -> Options -> Mechanical Design -> Drafting -> Administration**, administrators can forbid or allow users to:

- switch a drawing to another standard (via **File -> Page Setup**),
- update the standard used by a drawing (via **File -> Page Setup**).

Moreover, administrators can lock these settings so that other users running a session with the same environment inherit those settings and cannot change them. This feature is described in the Locking Settings section, in the *Infrastructure Installation User's Guide*.

Upgrading Standard Files from Previous Releases

Depending on your needs, you can:

- Upgrade CATDrwStandard files (i.e. standard files customized in releases up to and including V5R8) to the current level for XML standard files
- Upgrade XML standard files from previous releases (i.e. XML standard files customized in releases starting from V5R9) to the current level for XML standard files

Upgrading CATDrwStandard Files to the Current XML Standard Files

Up to and including V5R8, the standard file defining standard XXX was a file named XXX.CATDrwStandard, located in **install_root/reffiles/Drafting.**

In V5R9, the format of the standard file was changed to XML. The standard file defining standard XXX is now a file named XXX.xml, located in **install_root/resources/standard/drafting.**

If you have customized or defined a CATDrwStandard file, and wish to re-use this customization in the current release, you need to convert your CATDrwStandard file into a XML file. There are 2 ways of doing this:

Manual upgrade

If the degree of customization of the standard file is small, you can start from one of the 4 pre-defined standard files (ISO, ANSI, JIS or ASME), and modify it using the standards editor (**Tools -> Standards**). You will need to modify the parameter values, and add the styles that you had defined in the CATDrwStandard file.

Automatic upgrade

A batch utility is provided in order to automatically generate the XXX.xml file starting from a XXX.CATDrwStandard file. All the customization done on the CATDrwStandard file will be reproduced in the XML file, and all styles defined in CATDrwStandard file will be added.

The utility will also add to the XML file the new standard parameters (with default values), as well as the new pre-defined styles.

- If you want to convert a single CATDrwStandard to the current XML format, use:
  CATAnnStandardTools MIGRATE XXX [dir]
- If you want to convert all CATDrwStandard files to the current XML format, use:
  CATAnnStandardTools MIGRATE_ALL [dir]

For more information on using these commands on Windows and on Unix, see below.

The tasks below will show you how to use the standard automatic upgrade tool on Windows and on Unix.

Using the standard automatic upgrade tool on Windows

1. Open an MS-DOS Window.
2. Change to the folder in which you installed the product.

   The default folder is \intel_a\code\\bin

3. You have two options:

- To generate XML files for all the CATDrwStandard files located in reffiles\Drafting, enter this command:

  CATAnnStandardTools MIGRATE_ALL [dir]

  where [dir] is an optional directory in which to write the resulting XML files. Local directory is the default.

- To generate the XML file corresponding to one single standard, enter this command:

  CATAnnStandardTools MIGRATE XXX [dir]

  where XXX is the name of the standard you want to convert (ISO, ANSI...) and [dir] is an optional directory in which to write the resulting XML file. Local directory is the default.

Using the standard automatic upgrade tool on UNIX

1. Open a shell command window.
2. Change to the directory in which you installed the product.

   The default directory is /OS_a/code/command/

   where OS_a is:

- aix_a
- hpux_a
- irix_a

- solaris_a

3. You have two options:

- To generate XML files for all the CATDrwStandard files located in reffiles\Drafting, enter this command:

  ./catstart -run "CATAnnStandardTools MIGRATE_ALL [dir]"

  where [dir] is an optional directory in which to write the resulting XML files. Local directory is the default.

- To generate the XML file corresponding to one single standard, enter this command:

  ./catstart -run "CATAnnStandardTools MIGRATE XXX [dir]"

  where XXX is the name of the standard you want to convert (ISO, ANSI...) and [dir] is an optional directory in which to write the resulting XML files. Local directory is the default.

Upgrading XML Standard Files from Previous Releases to the Current Level

The XML standard file has evolved in each release since V5R9. New standard parameters have been added, some have been modified, and new functionalities (such as styles) have been introduced.

If you have customized or defined an XML standard file in a previous release (i.e. a release starting from V5R9), and wish to re-use this customization in the current level, you need to upgrade your XML file. There are 2 ways of doing this:

Manual upgrade

If the degree of customization of the standard file is small, you can start from one of the 4 pre-defined standard files (ISO, ANSI, JIS or ASME), and modify it using the standards editor (**Tools -> Standards**). You will need to modify the parameter values and customize new parameters and/or styles.

Automatic upgrade

A batch utility is provided in order to automatically generate the current XML file starting from an XML file from a previous release. All the customization done on the starting file will be reproduced in the upgraded XML file.

The utility will also add the new parameters and styles introduced in the current release (with default values) in the XML file.

- If you want to upgrade a single XML file to the current version, use:
  CATAnnStandardTools UPGRADE XXX [dir]
- If you want to upgrade all XML files to the current version, use:
  CATAnnStandardTools UPGRADE_ALL [dir]

For more information on using these commands on Windows and on Unix, see below.

The tasks below will show you how to use the standard automatic upgrade tool on Windows and on Unix.

Using the standard automatic upgrade tool on Windows

1. Open an MS-DOS Window.
2. Change to the folder in which you installed the product.

   The default folder is \intel_a\\code\\bin

3. You have two options:

- To upgrade standard files for all the XML files located in

  install_root\resources\standard\drafting, enter this command:

  CATAnnStandardTools UPGRADE_ALL [dir]

  where [dir] is an optional directory in which to write the resulting XML files. Local

  directory is the default.

- To upgrade the XML file corresponding to one single standard, enter this command:

  CATAnnStandardTools UPGRADE XXX [dir]

  where XXX is the name of the standard you want to convert (ISO, ANSI, MY_ISO...) and [dir] is an optional directory in which to write the resulting XML file. Local directory is the default.

The batch will first search the standard file in the directory defined by the exported variable CATCollectionStandard (e.g. set CATCollectionStandard=e:\tmp), and then, if not found, in the following directory: install_root\resources\standard\drafting.

Using the standard automatic upgrade tool on UNIX

1. Open a shell command window.
2. Change to the directory in which you installed the product.

   The default directory is /OS_a/code/command/

   where OS_a is:

- aix_a
- hpux_a
- irix_a
- solaris_a

3. You have two options:

- To upgrade standard files for all the XML files located in install_root\resources\standard\drafting, enter this command:

  ```
  ./catstart -run "CATAnnStandardTools UPGRADE_ALL [dir]"
  ```

  where [dir] is an optional directory in which to write the resulting XML files. Local directory is the default.

- To upgrade the XML file corresponding to one single standard, enter this command:

  ```
  ./catstart -run "CATAnnStandardTools UPGRADE XXX [dir]"
  ```

  where XXX is the name of the standard you want to convert (ISO, ANSI...) and [dir] is an optional directory in which to write the resulting XML files. Local directory is the default.

The batch will first search the standard file in the directory defined by the exported variable CATCollectionStandard (e.g. export CATCollectionStandard=d/tmp), and then, if not found, in the following directory: install_root\resources\standard\drafting.

Setting Standard Parameters and Styles

The Interactive Drafting workbench lets administrators set and create standard parameters and standard styles.

Before you Begin: You should be familiar with important concepts: structure of the standards, general syntax for the standard editor values, DBCS restriction.
Setting Standard Parameters: Set standard parameters and create standard formats.
Setting Standard Styles: Set standard styles that will be used as default values when creating new elements.

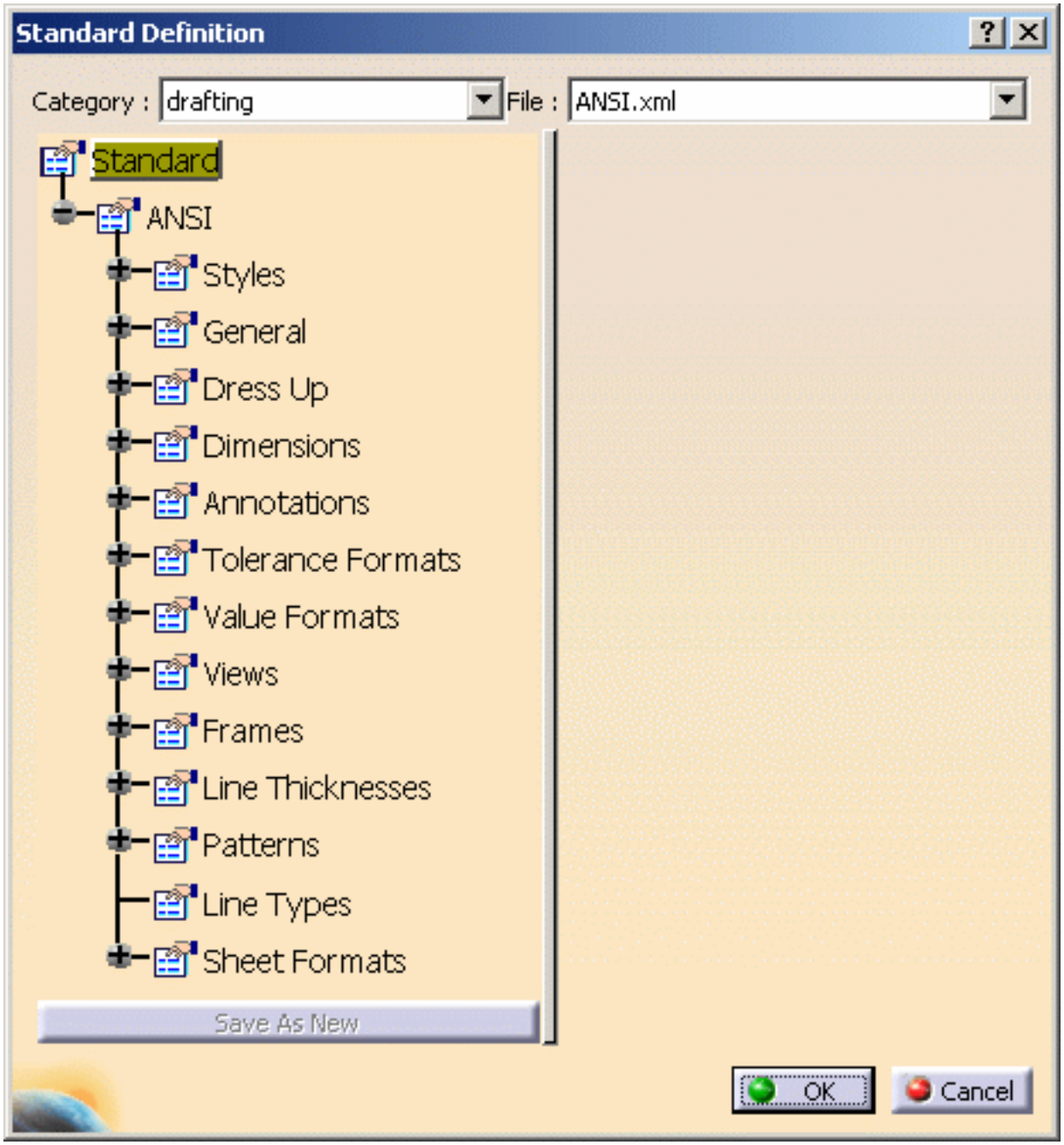
Before You Begin

Structure of the Standard

Standards are defined by the administrator.

A drafting standard file is structured as a tree, as it appears in the Standards Editor (available via **Tools -> Standards**). It contains several main sections, each dealing with a specific aspect of drafting customization:

- Styles
- General parameters
- Dress-up parameters
- Dimension parameters
  - Company-defined dimension tolerance formats
  - Company-defined dimension value display formats
  - Pre-defined formats for tolerance and dimension values
- Annotation parameters
- Company-defined view generation
- Company-defined frame formats
- Company-defined line thickness
- Company-defined patterns
- Company-defined linetypes
- Company-defined sheet formats



General Syntax for the Standard Editor Values

Fractions and operations

The standards editor can handle basic numerical operations to help you enter the values for the parameters. You can enter your value as a set of operations, and let the program compute the result when you validate the field.

For example, for each parameter of the "real" type, you can specify the value using a fraction:
NDFact_1 = 1/60.

You can also use units or trigonometric functions in your operations:
NDSepPos_1 = 1 in + 1 mm + cos(0.12)

Special characters

For each parameter of the "string" type, you can enter special characters using the following keywords:

- [DEGREE] will be displayed as deg
- [MINUTE] will be displayed as '
- [SECOND] will be displayed as "

A special character can be used alone or combined with other characters (the special character only counts as 1 character):

NDSepar_1 , [DEGREE]

or

NDSepar_1 , " in [DEGREE]"

DBCS Restriction

Double-byte character sets are not supported when creating new standard parameters and styles (i.e. corresponding to new nodes in the standards editor tree).

However, double-byte character sets are supported when setting values for standard parameter and styles.

For More Information

- Refer to About Standard Parameters for more information on how to customize standard parameters, how to define new standard formats, and on the general syntax for the standard editor values.
- Refer to About Styles for more information on setting default values for elements using styles, and on how to customize standard styles.

Setting Standard Parameters

The Interactive Drafting workbench lets administrators set standard parameters and create standard formats.

About Standard parameters: Learn more about the management of standard parameters.

General parameters: Customize the parameters that let you control and restrict the values that are available for some element properties.

Dress-up parameters: Customize the parameters that deal with the appearance of dress-up elements, such as markup arrows.

Dimension parameters: Customize the parameters that deal with the appearance of annotation and dimension elements.

Annotation Parameters: Customize the parameters that deal with the position of text leaders.

Tolerance Formats: Customize the dimension tolerance formats, which are user-defined formats to be applied to dimension tolerances.

Value Formats: Customize the dimension value formats, which are user-defined formats to be applied to dimension values.

Pre-defined Formats for Tolerance and Dimension Values: Customize the pre-defined formats for tolerance and dimension values.

Pre-defined Styles Definitions: Customize the pre-defined non-modifiable styles and their definition, which you can use as a reference when defining new formats.

View Generation Definition: Define view generation, i.e. customize settings that should be applied when generating views.

Frame Definition Parameters: Define customizable fixed-size frames. A frame is a property which can be applied to texts as well as certain types of annotations and dress up elements.

Line Thickness Definition: Define line thickness. Line thickness is a property which can be applied to, and drives the representation of, almost all elements in a drawing, such as lines, curves, dimension lines, etc.

Pattern Definition: Define patterns. Patterns are used for area fills or when generating section views/cuts or breakout views.

Linetype Definition: Define linetypes. Linetypes can be applied to, and drive the representation of, almost all elements in a drawing, such as lines, curves, dimension lines, etc.

Sheet Format Definition: Define sheet format. Sheets contain a view that supports the geometry when creating a drawing.

Layout Views Customization - for the 2D Layout for 3D Design workbench: Define the 2D Layout for 3D Design view box, which gathers all the data needed to fully define the layout of a view set in the 2D window, as well as the position of each view in the 3D space.

About Standard Parameters

Customizing Standard Parameters

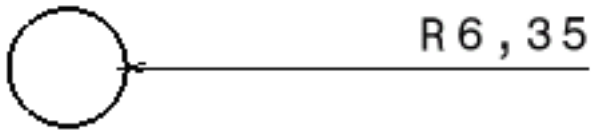
In this scenario, administrators will learn how to customize standard parameters using an example.

This scenario provides an example of dimension customization, but the procedure is the same when customizing other standard parameters (dimensions, annotations, dress-up elements, etc.) The procedure differs when customizing styles. For more information, refer to About Styles.

With the pre-defined ISO standard, a radius dimension extension lines reaches the center of the circle. You will modify the extension line so that it does not reach the center of the circle.

Select **Tools -> Standards** to launch the standards editor. Choose the **Drafting** category, and then open the ISO.xml file from the drop-down list.

1. Select the **Dimensions** node in the editor.
2. Modify the **DIMLRadiusExtReachCenter** parameter value from Yes to No.
3. Set the **DIMLRadiusExtLength** parameter value to 2.
4. Click **OK** to save your modifications and exit the standards editor.
5. Create an ISO drawing using the **File - > New** command.
6. Create a circle, and add a radius dimension to it. The dimension extension line does not reach the center, as it would have with the pre-defined ISO standard.

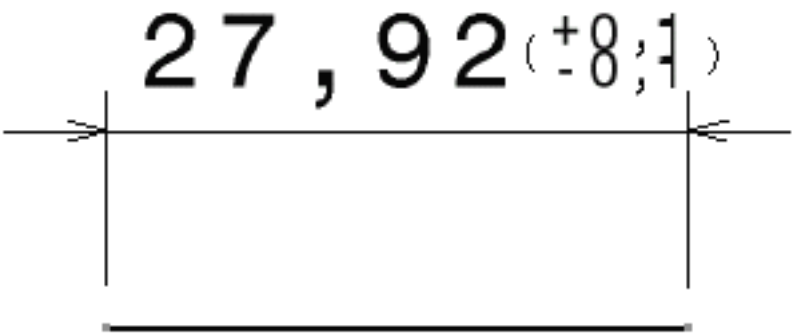


Defining a New Format

In this scenario, administrators will learn how to create a new format using an example.

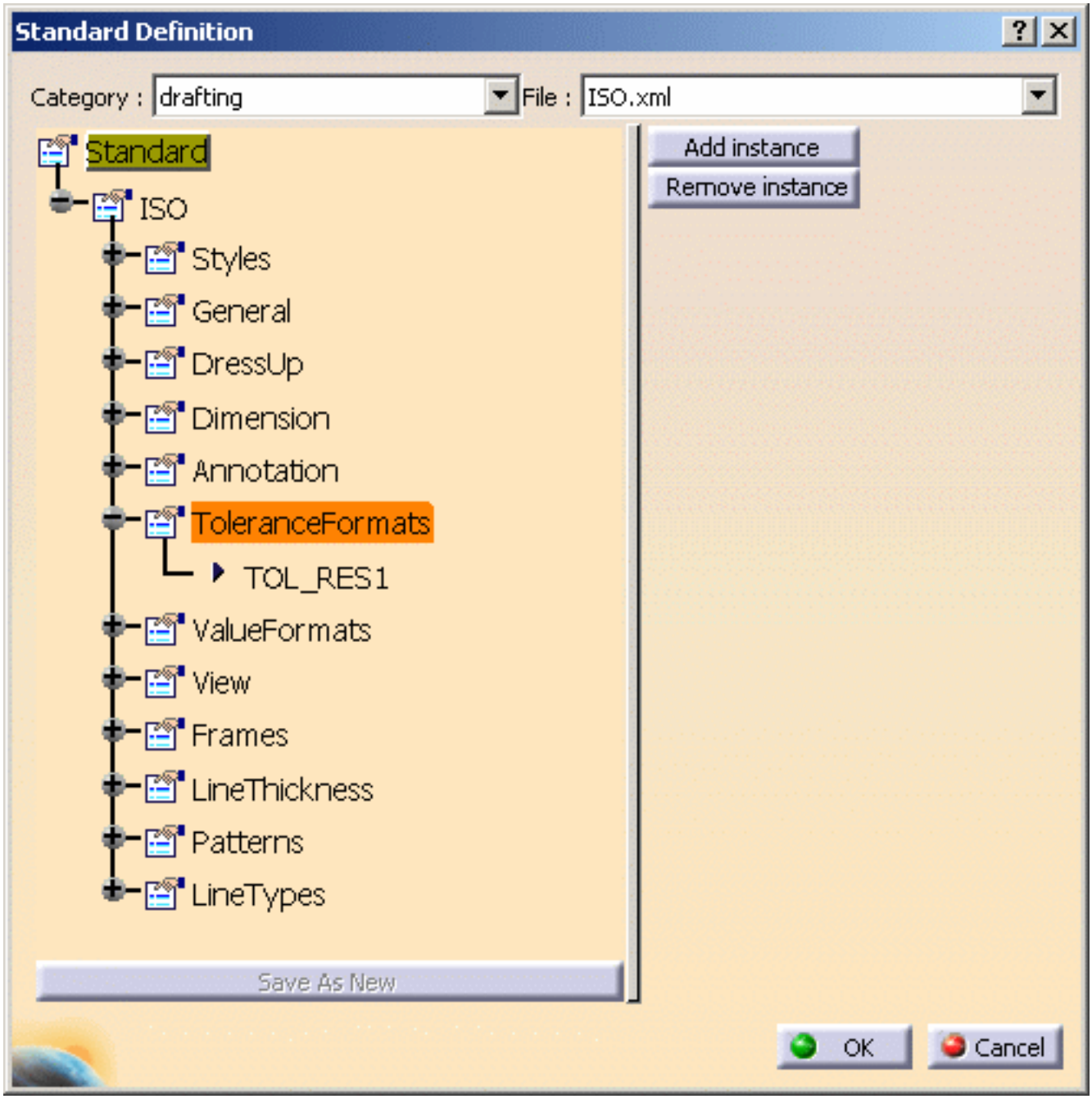
This scenario shows how to create a dimension tolerance format as an example, but the procedure is the same for other formats (dimension value, line thickness, etc.). Specific differences are indicated in the course of this scenario.

You want to create this new dimension tolerance format, with superimposed tolerance values and parenthesis as separators.



Select **Tools -> Standards** to launch the standards editor. Choose the **Drafting** category, and then open the ISO.xml file from the drop-down list.

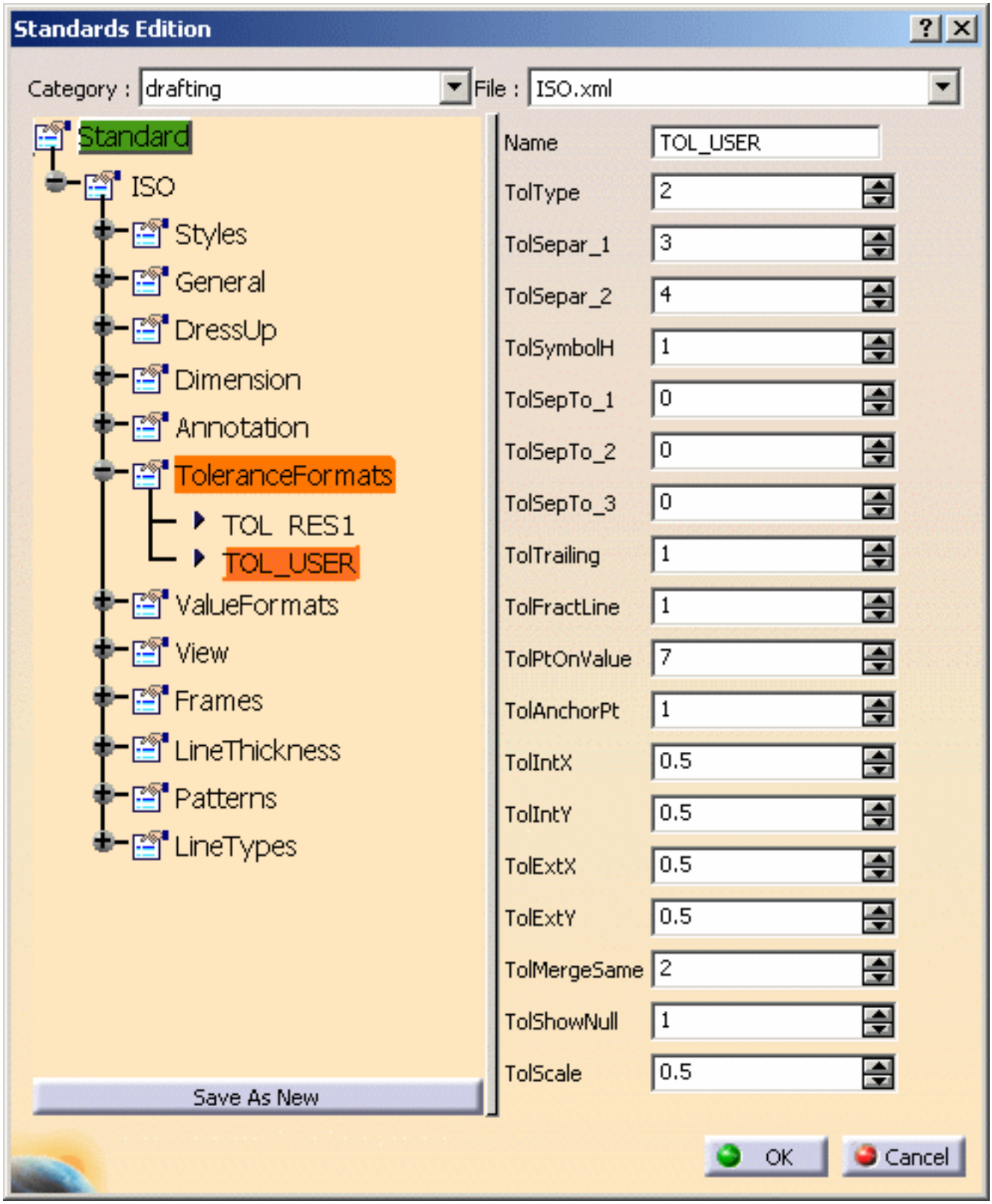
1. Select the **Tolerance Formats** node in the editor.



2. Click on the **Add Instance** button. A format called TOLXXX is created.

3. Rename this format TOL_USER.

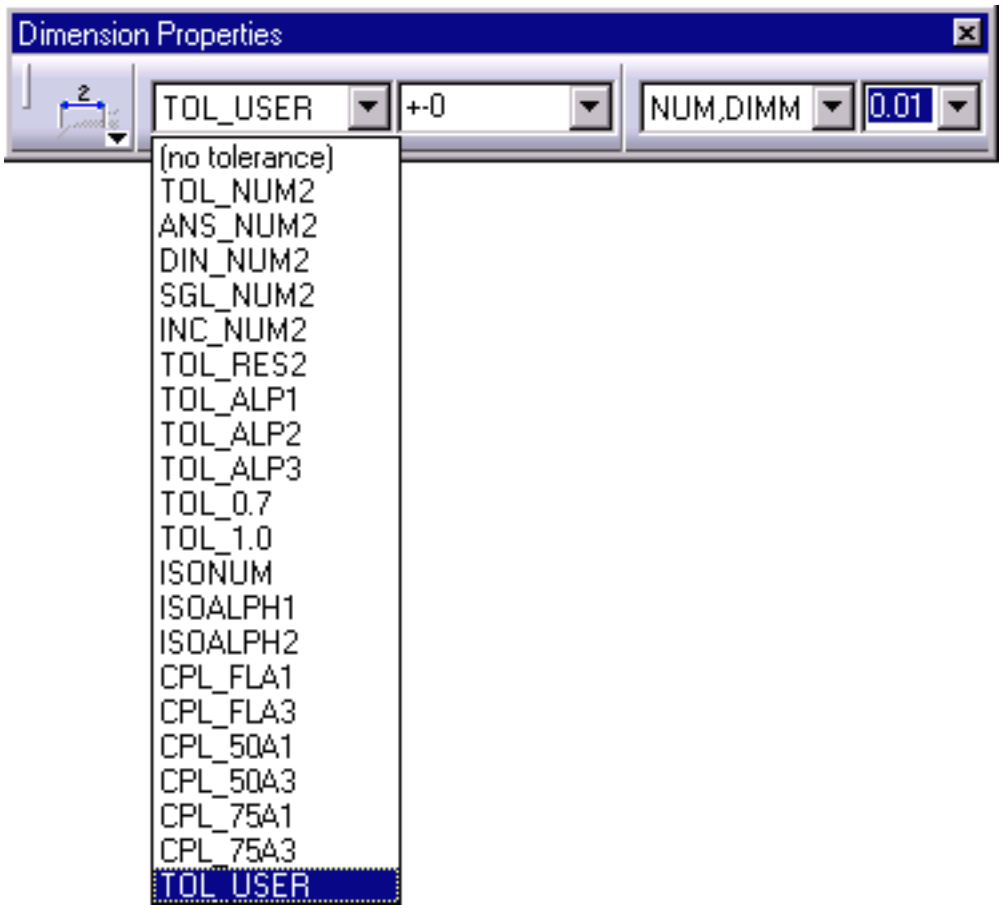
4. Customize the values as follows:



- Make sure you set every parameter.
- You must use the symbol used by the computer system to set a parameter to a real value ("," or ".").
  Note: you can use a fraction to set a parameter.

5. Click **OK** to save the ISO.xml file and exit the standards editor.

6. Create a new ISO drawing. The new tolerance style will appear in the tolerance combo box.



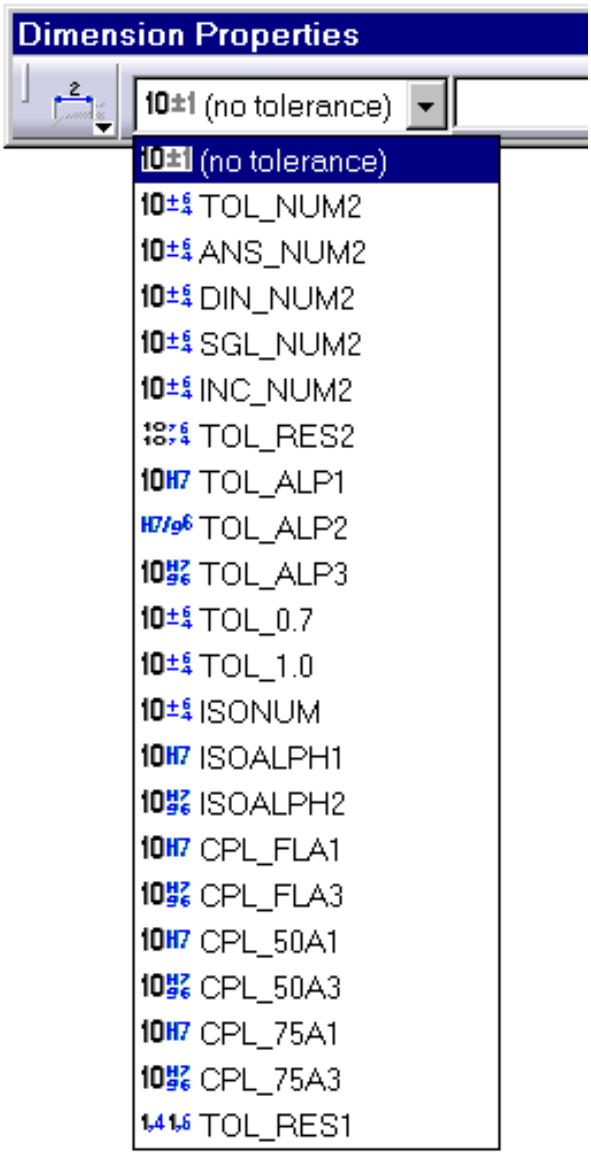
General Parameters

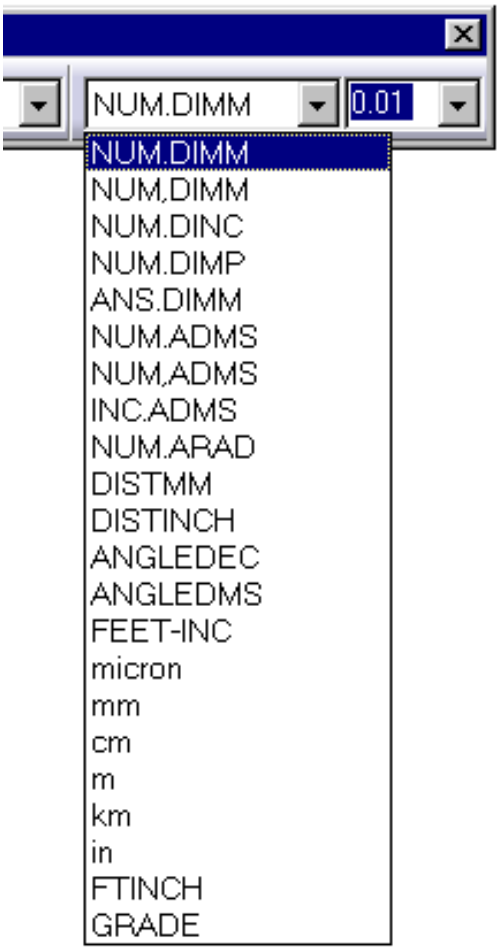
This section deals with general parameters. These let you control and restrict the values that are available for some element properties, by controlling the values in the Properties toolbar or in the element properties.

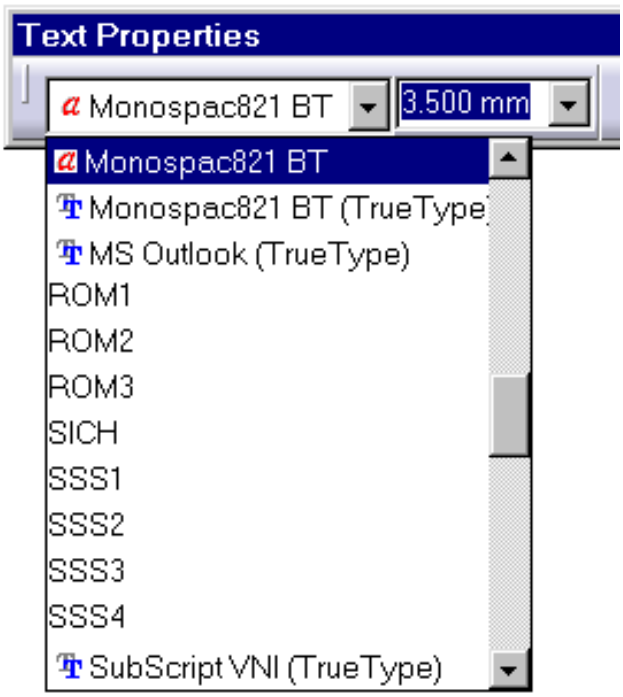
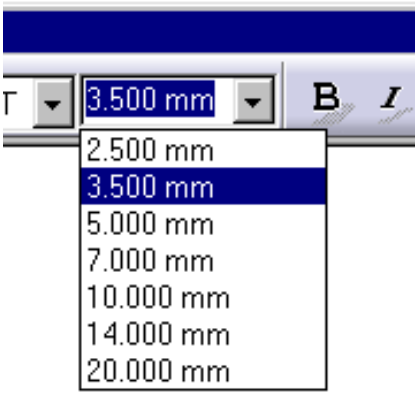
Changing these values will not have an impact on already existing elements, since they control the user interface and not directly the drafting elements.

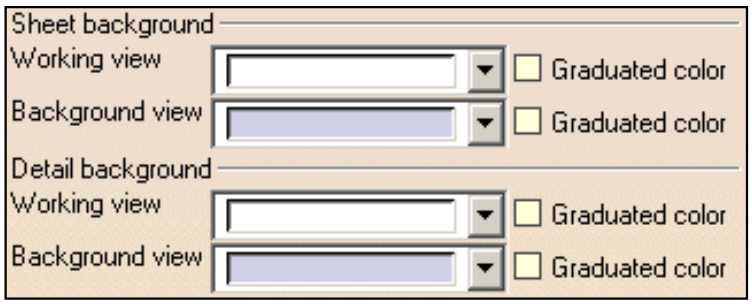
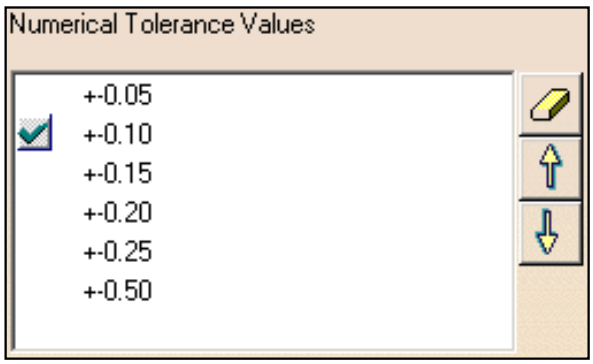
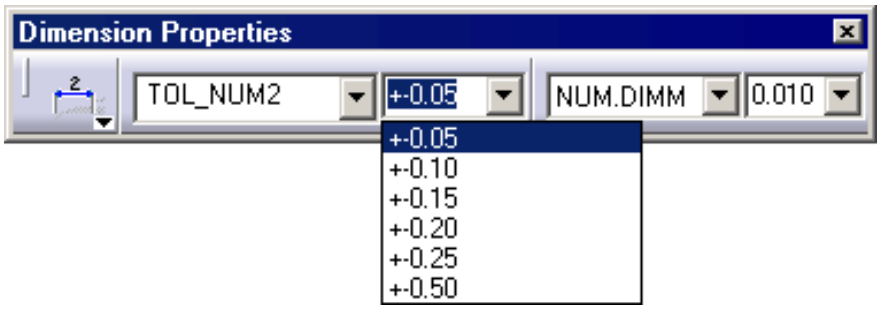
Defining General Parameters

General parameters are located in the **General** node of the Standards editor, available via **Tools -> Standards.**

Parameter Name	Description	Value
AllowedToleranceFormats	Lists tolerance styles allowed on dimensions. Only the listed styles will be displayed and available to users through the **Dimension Properties** toolbar or via **Edit -> Properties**.  	List of strings empty list = all defined tolerance styles are available

DefaultToleranceFormat	**Deprecated** Now managed in Dimension Styles	-
AllowedNumericalFormats	Lists value display styles allowed on dimensions. Only the listed styles will be available to users through the **Dimension Properties** toolbar or via **Edit -> Properties.**  	Strings: list of Value Display styles, **spelled exactly as they appear in the Dimension Properties toolbar or in Edit -> Properties** empty list = all Value Display styles are available
DefaultNumericalFormatLength	**Deprecated** Now managed in Dimension Styles	-
DefaultNumericalFormatAngle	**Deprecated** Now managed in Dimension Styles	-
AllowedTextFonts	Lists allowed text fonts. Only the listed fonts will be available to users in the text **Text Properties** toolbar or via **Edit -> Properties.**	Strings: list of font names,

		**spelled exactly as they appear in the Text Properties toolbar or in Edit -> Properties** blank = all installed fonts will be available
DefaultTextFont	**Deprecated** Now managed in Annotation Styles	-
AllowedTextFontSizes	Lists allowed text font sizes (in mm). Only the listed sizes will be available to users in the **Text Properties** toolbar or via **Edit -> Properties.** 	List of values in mm
DefaultTextFontSize	**Deprecated** Now managed in Annotation Styles	-
StandardUpdateMigration	Specifies whether some annotations (roughness symbol, geometrical tolerance and balloon) should be migrated when updating the standard of a drawing. Setting this parameter to Yes is recommended if you want to benefit, when applicable, from any new functionality implemented for such annotations since they were created on a previous release. Annotation customization will not be lost.	Yes/No

Sheet Colors	Specifies the default color for: • The **Sheet background**: ○ The **Working view** color, graduated or not. ○ The **Background view** color, graduated or not. • The **Detail background**: ○ The **Working view** color, graduated or not. ○ The **Background view** color, graduated or not. 	
Tolerance Values	Specifies the default value for: • The **Numerical Tolerance Values.** • The **Alphanumerical Tolerance Values.** • The **Bi-Alphanumerical Tolerance Values.** • The **Multiple Tolerance Values.**  You can modify, add, remove or organize tolerance values. The check symbol indicates the default tolerance value. 	

Dress-Up parameters

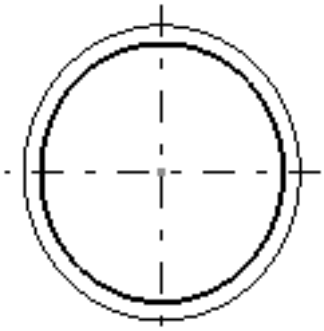
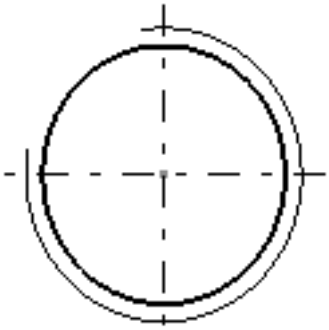
This section deals with dress-up parameters. These let you define the appearance of dress-up elements, such as markup arrows and threads.

Defining Dress-Up Parameters

Dress-up parameters are located in the **DressUp** node of the Standards editor, available via **Tools -> Standards:**

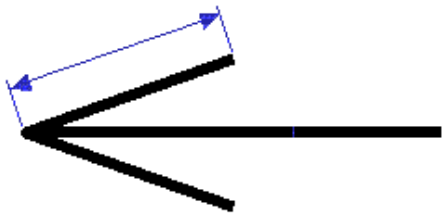
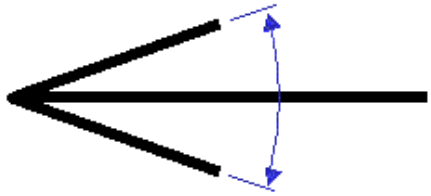
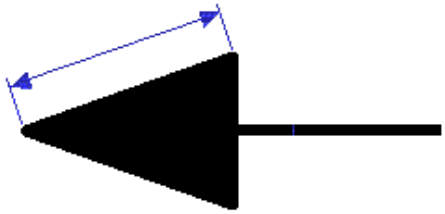
- Thread
- Symbols

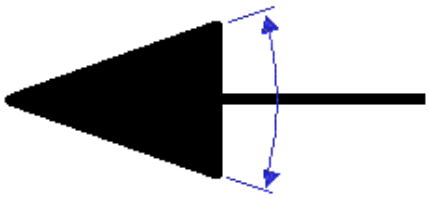
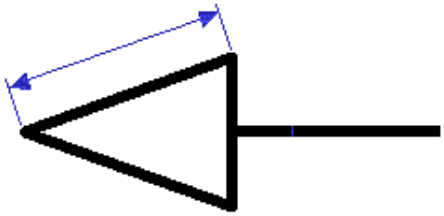
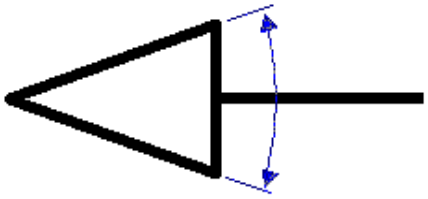
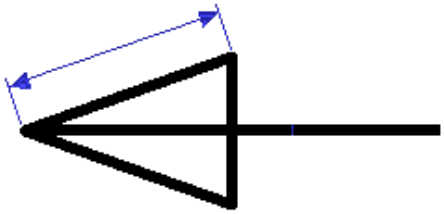
Thread

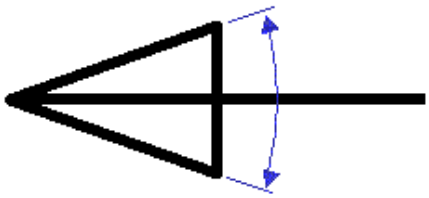
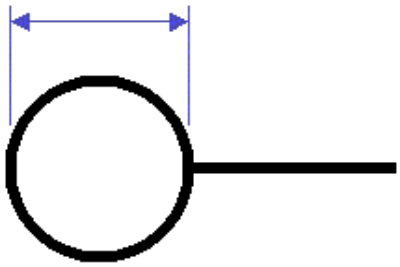
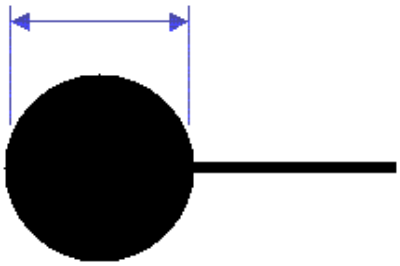
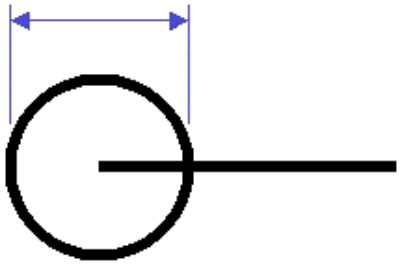
Parameter Name	Description	Value
ThreadRepresentation	Specifies how threads should be represented.	Circle 
		ArcCircle 

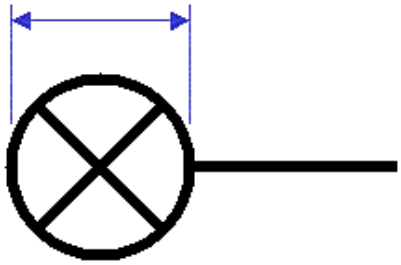
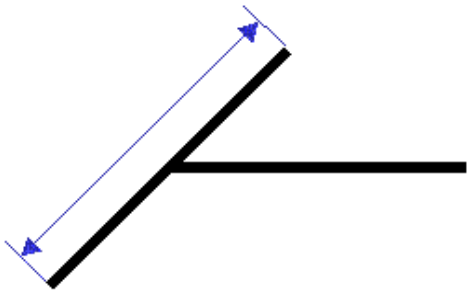
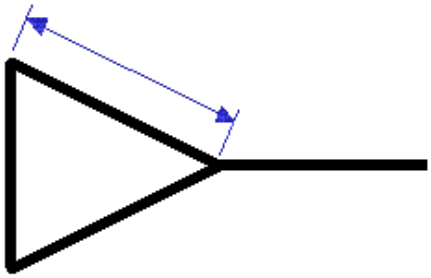
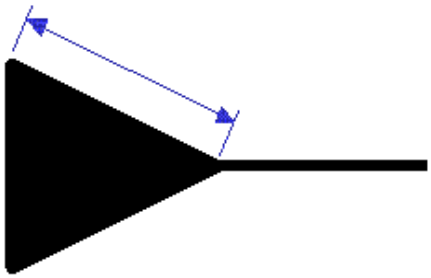
Symbols

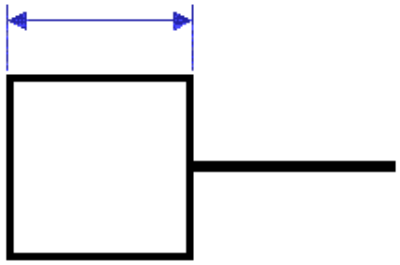
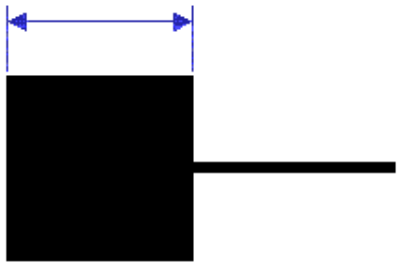
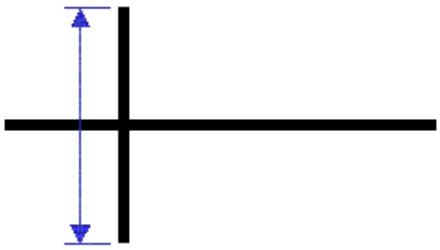
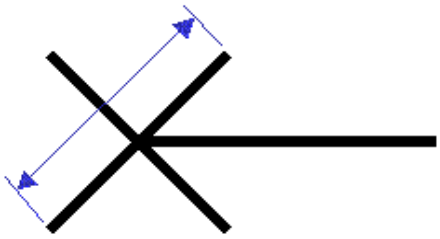
Note that symbol parameters apply only to arrows, and not to leaders (annotation leaders, dimension leaders, etc.). If you want to modify dimension and annotation leader symbols, refer to the Dimension Parameters > Dimension and Annotation Leader Symbols section.

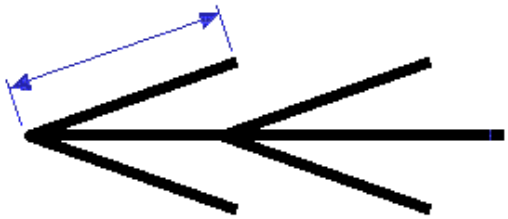
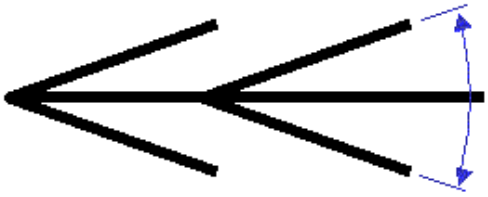
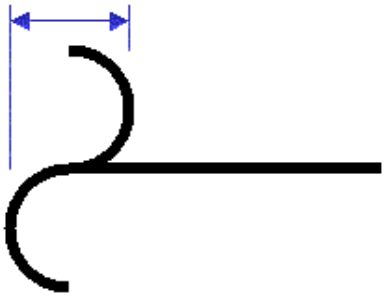
Parameter Name	Description	Value
Open Arrow > Length	Defines simple arrow length 	mm
Open Arrow > Angle	Defines simple arrow angle 	Degrees
Filled Arrow > Length	Defines filled arrow length 	mm

Filled Arrow > Angle	Defines filled arrow angle 	Degrees
Outlined Arrow > Length	Defines closed arrow length 	mm
Outlined Arrow > Angle	Defines closed arrow angle 	Degrees
Transparent Arrow > Length	Defines symetric arrow length 	mm

Transparent Arrow > Angle	Defines symetric arrow angle 	Degrees
Outlined Circle > Diameter	Defines circle size 	mm
Filled Circle > Diameter	Defines filled circle size 	mm
Transparent Circle > Diameter	Defines symetric (crossed) circle size 	mm

Crossed Circle > Diameter	Defines cross circle size 	mm
Slash > Length	Defines slash size 	mm
Outlined Triangle > Length	Defines triangle size 	mm
Filled Triangle > Length	Defines filled triangle length 	mm

Outlined Square > Length	Defines white filled square length 	mm
Filled Square > Length	Defines black filled square length 	mm
Plus > Length	Defines plus length 	mm
Cross > Length	Defines cross length 	mm

Double Open Arrow > Length	Defines double arrow length 	mm
Double Open Arrow > Angle	Defines double arrow angle 	Degrees
Wave > Diameter	Defines wave arrow size 	mm

Dimension Parameters

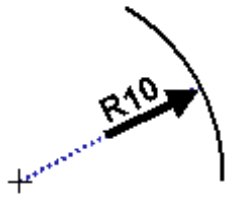
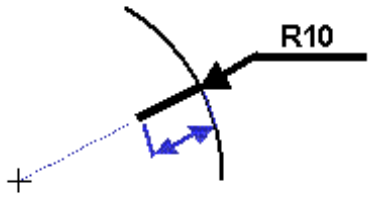
The dimension parameters are located in the **Dimension** node of the standard file. They deal with the appearance of annotation and dimension elements.

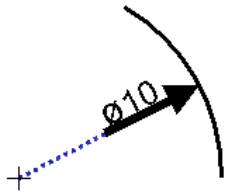
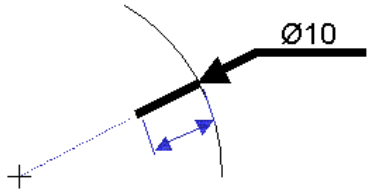
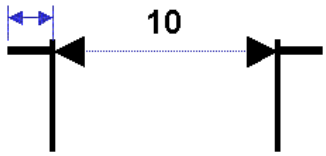
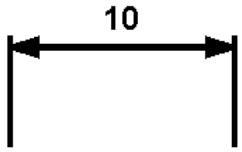
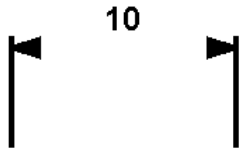
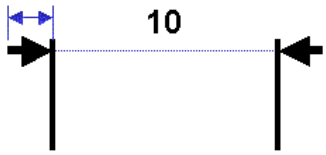
- Dimension and Annotation Parameters
- Dimension and Annotation Leader Symbols
- Dimension Value
- Chamfer Dimension Parameters
- Half Dimensions
- Dimension Associated Texts
- Annotations
- Fake dimensions
- Dual Dimensions
- Cumulate Dimensions (Ordinate Dimensions): General Parameters
- Cumulate Dimensions: Parameters applying only if value orientation reference is Dimension Line (Cumulate Dimension: Value Orientation Reference = 1)
- Cumulate Dimensions: Parameters applying only if the value orientation reference is Extension Line (Cumulate Dimension: Value Orientation Reference = 2)
- Curvilinear Length Symbol
- Intersection Point

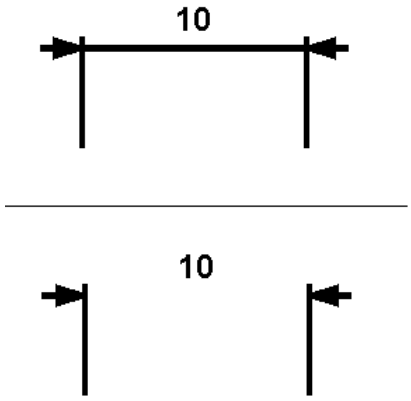
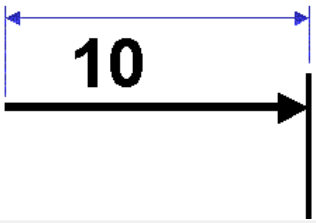
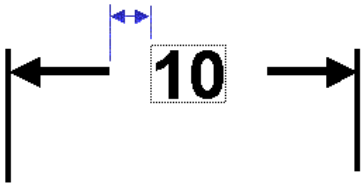
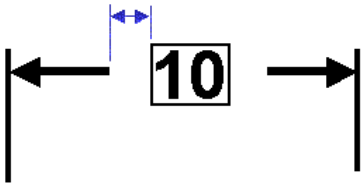
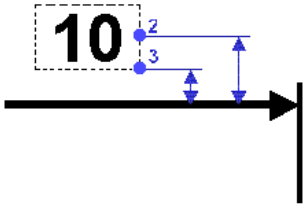
These parameters are global, which means that changing their value will have an impact on all elements in the drawing.

This section lists all the parameters which were contained in CATDrwStandard files up to V5 R9.

Dimension and Annotation Parameters

Parameter	Value	Description
International standard	[ISO/ANSI/JIS]	Each user-defined standard is based on one of 3 international standards: ISO, ANSI, JIS. This sets some basic parameters.
Dimension Line: Extension on radius dimensions (value inside circle), Reach center	[Yes/No] Yes = till center No = till value	
Dimension Line: Extension on radius dimensions (value outside circle), Reach center	[Yes/No] Yes = till center No = constant overrun	
Dimension Line: Extension on radius dimensions (value outside circle), Overrun length	(mm)	
Dimension Line: Extension on one-symbol diameter dimensions (value inside circle), Reach center	[Yes/No] Yes = till center No = till value	

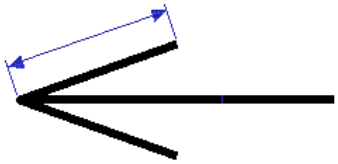
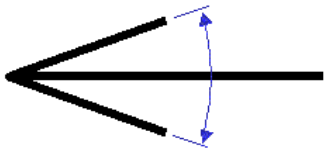
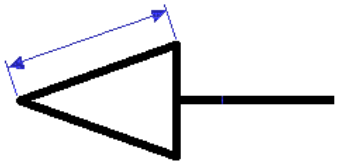
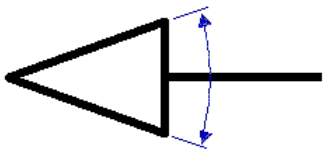
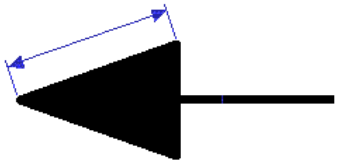
Dimension Line: Extension on one-symbol diameter dimensions (value inside circle), Overrun length	(mm)	
Dimension Line: Extension on one-symbol diameter dimensions (value outside circle), Reach center	[Yes/No] Yes = till center No = constant overrun	
Dimension Line: Extension on one-symbol diameter dimensions (value outside circle), Overrun length	(mm)	
Dimension Line: Display and extent (for non-flipped symbols), Overrun length	(mm)	
Dimension Line: Display and extent (for non-flipped symbols), Show	[Yes/No] Yes = displayed No = not displayed	 
Dimension Line: Display and extent (for flipped symbols), Overrun length	(mm)	

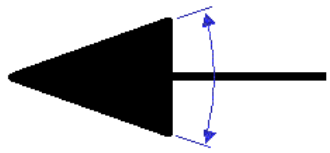
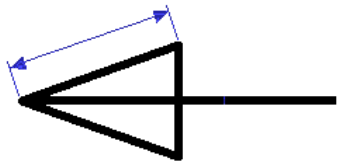
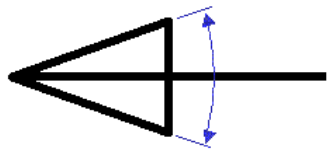
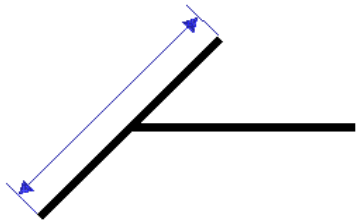
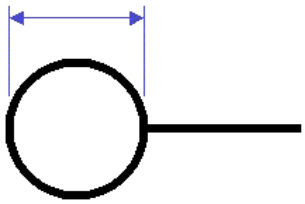
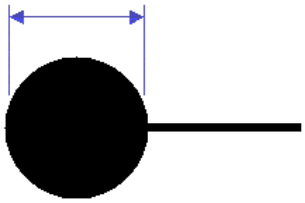
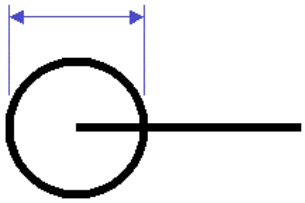
Dimension Line: Display and extent (for flipped symbols), Show	[Yes/No] Yes = displayed No = not displayed	
Dimension Line: Length for one-symbol dimensions (distance and angle), Underline	[2/1] 2 = Length relative to value 1 = Constant length	The dimension line may either have a given length, or automatically adjust to reach the dimension value.
Dimension Line: Length for one-symbol dimensions (distance and angle), Constant length	(mm)	if Dimension Line: Length for one-symbol dimensions (distance and angle), Underline = 1 
Dimension Line: Gap around unframed value	(mm)	
Dimension Line: Gap around framed value	(mm)	
DIMTYPos	-	**Deprecated**
Dimension Value: Vertical justification	[2 / 3] 2 = center 3 = bottom	

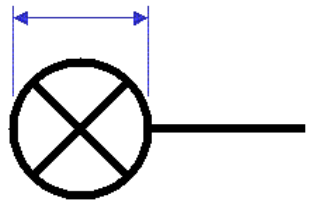
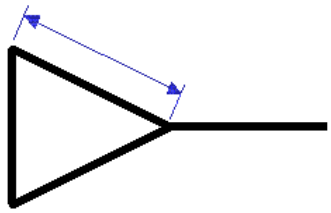
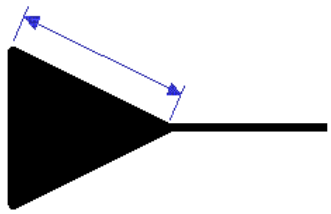
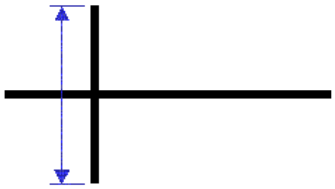
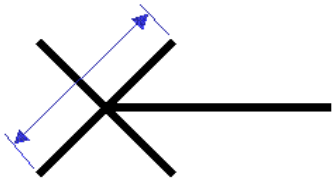
Dimension tolerance: Multi-tolerance with associative numerical value	[No/Yes] No = not associative Yes = associative	Specifies whether the numerical definition of a multi-tolerance is associative to the dimension value (in which case it is automatically updated when the dimension value is changed).

Dimension and Annotation Leader Symbols

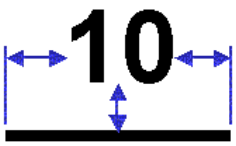
Note that dimension and annotation leader symbols do not apply to arrows. If you want to modify symbol parameters for arrows, refer to the Dress-up Parameters > Symbols section.

Parameter	Value	Description
Dimension and Annotation Leader Symbols: Open arrow size	(mm)	
Dimension and Annotation Leader Symbols: Open arrow angle	(degrees)	
Dimension and Annotation Leader Symbols: Outlined arrow size	(mm)	
Dimension and Annotation Leader Symbols: Outlined arrow angle	(degrees)	
Dimension and Annotation Leader Symbols: Filled arrow size	(mm)	

Dimension and Annotation Leader Symbols: Filled arrow angle	(degrees)	
Dimension and Annotation Leader Symbols: Transparent arrow size	(mm)	
Dimension and Annotation Leader Symbols: Transparent arrow angle	(degrees)	
Dimension and Annotation Leader Symbols: Slash size	(mm)	
Dimension and Annotation Leader Symbols: Outlined circle size	(mm)	
Dimension and Annotation Leader Symbols: Filled circle size	(mm)	
Dimension and Annotation Leader Symbols: Transparent circle size	(mm)	

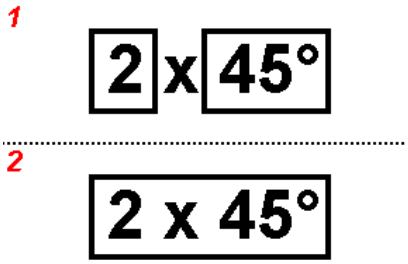
Dimension and Annotation Leader Symbols: Crossed circle size	(mm)	
Dimension and Annotation Leader Symbols: Outlined triangle size	(mm)	
Dimension and Annotation Leader Symbols: Filled triangle size	(mm)	
Dimension and Annotation Leader Symbols: Plus size	(mm)	
Dimension and Annotation Leader Symbols: Cross size	(mm)	
Dimension and Annotation Leader Symbols: Symbol reversal limit	(mm)	

Dimension Value

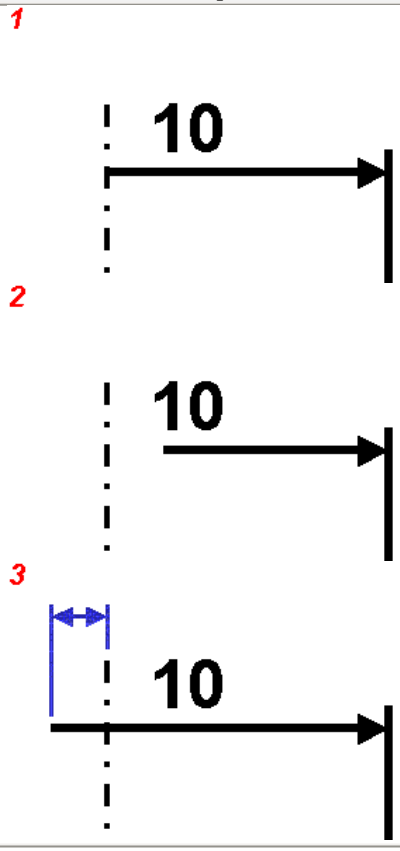
Parameter	Value	Description
Dimension Value: Underlining size, Left tail	(mm)	
Dimension Value: Underlining size, Right tail		
Dimension Value: Underlining size, Vertical space		

Chamfer Dimension Parameters

Parameter	Value	Description

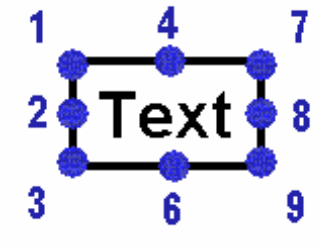
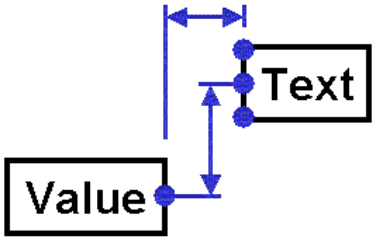
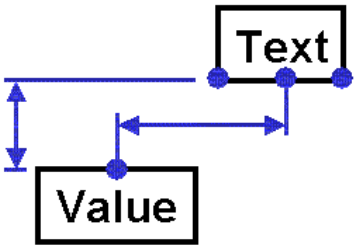
Chamfer Dimension: Separator font height	(mm)	
Chamfer Dimension: Value framing	[1 / 2] 1 = separately 2 = as a whole	

Half-Dimensions

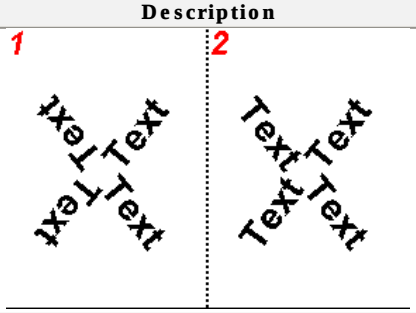
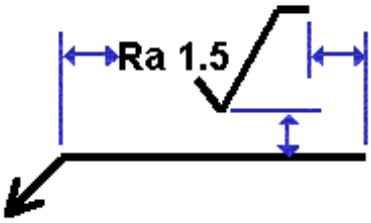
Parameter	Value	Description
Half-Dimension: Dimension line extent (with value inside), Overrun mode	[1 / 2 / 3] 1 = till Axis 2 = under value 3 = over axis	
Half-Dimension: Dimension line extent (with value inside), Overrun length	(mm)	if Half-Dimension: Dimension line extent (with value inside), Overrun mode = 3

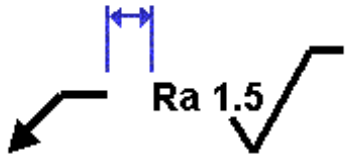
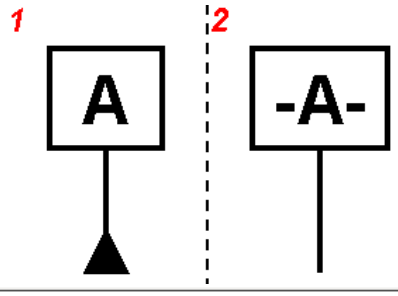
Dimension Associated Texts

Parameter	Value	Description
Dimension Associated Texts: Reference for positioning Before text	7 = top 8 = center 9 = bottom	reference for positioning

Dimension Associated Texts: Reference for positioning After text	1 = top 2 = center 3 = bottom	 for Before/After/Insert texts
Dimension Associated Texts: Reference for positioning Insert text	1 = top 2 = center 3 = bottom	
Dimension Associated Texts: Reference for positioning Upper text	3 = left 6 = center 9 = right	 for Upper/Lower texts
Dimension Associated Texts: Reference for positioning Lower text	1 = left 4 = center 7 = right	
Dimension Associated Texts: Horizontal offset of Before text	(mm)	
Dimension Associated Texts: Vertical offset of Before text		
Dimension Associated Texts: Horizontal offset of After text		
Dimension Associated Texts: Vertical offset of After text		
Dimension Associated Texts: Horizontal offset of Insert text		
Dimension Associated Texts: Vertical offset of Insert text		
Dimension Associated Texts: Horizontal offset of Upper text		
Dimension Associated Texts: Vertical offset of Upper text		
Dimension Associated Texts: Horizontal offset of Lower text		
Dimension Associated Texts: Vertical offset of Lower text		

Annotations

Parameter	Value	Description
Annotation: Text angle	[No / Yes] No = 0 to 360 degrees Yes = -90 to 90 degrees	
Annotation: Text leader size, Leader side	(mm)	**Warning:** this parameter is used only for roughness symbols created before V5R12. 
Annotation: Text leader size, Opposite leader side	(mm)	
Annotation: Text leader size, Vertical space	(mm)	

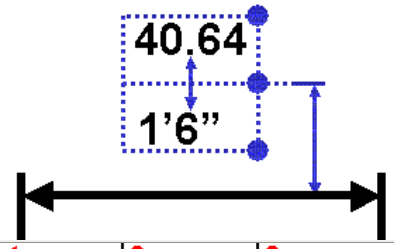
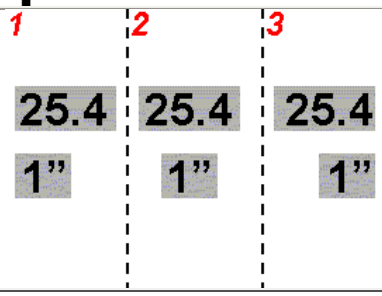
Annotation: Text leader size, Leader gap	(mm)	
Annotation: Text thickness	(mm)	For compatibility with V4. For dimensions created before R14 and annotations created before R13 SP4: • non-bold fonts take this parameter into account. • bold fonts are assigned a thickness of 0,7 mm (to preserve the previous behavior). Complex text and roughness annotations do not take this parameter into account. **Deprecated** for dimensions created since R14 and annotations created since R13 SP4. For these annotations and dimensions: • bold fonts are assigned thickness index 3. • non-bold fonts are assigned thickness index 1.
Annotation: Datum feature leader representation mode	[1 / 2] 1 = Normal 2 = Flag	ANSI parent standard only 

Fake dimensions

Parameter	Value	Description
Fake Dimension: Underline, Tail length	(mm)	If Fake Dimension: Value display mode = 1
Fake Dimension: Underline, Vertical offset		

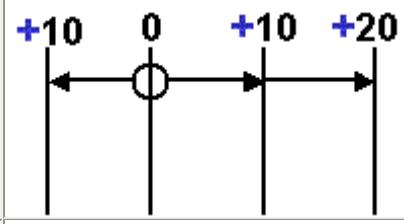
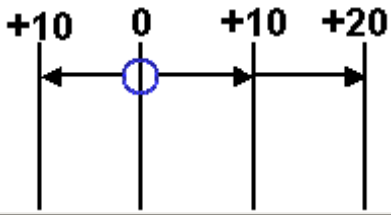
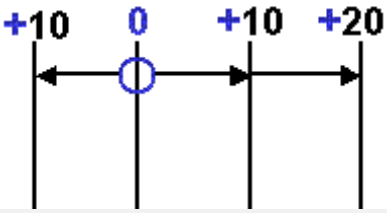
Dual Dimensions

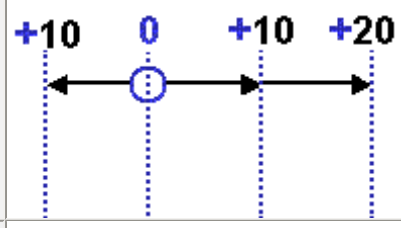
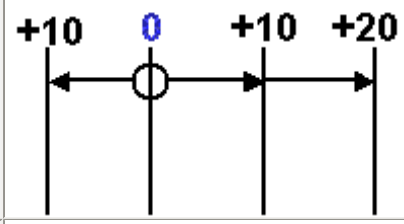
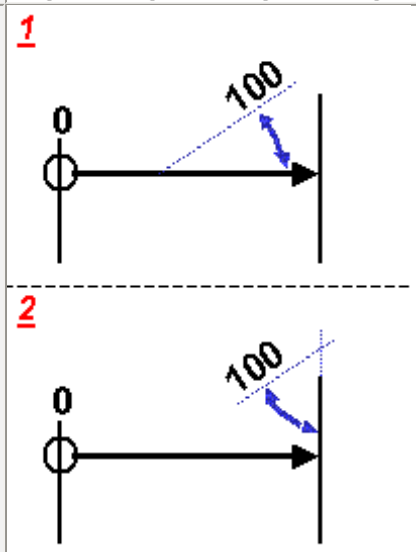
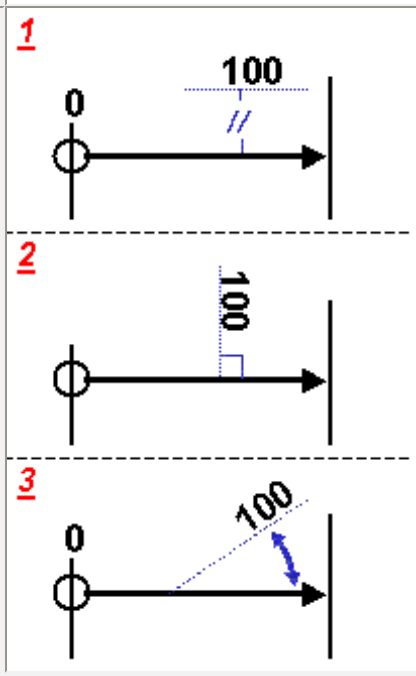
Parameter	Value	Description

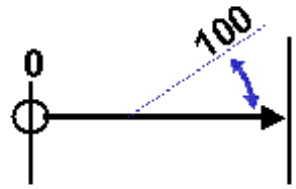
Dual Dimension: Side-by-side dual display mode, Separator height	(mm)	
Dual Dimension: Values above-one-another display mode, Above offset	(mm)	
Dual Dimension: Values above-one-another display mode, Above space	(mm)	
Dual Dimension: Values above-one-another display mode, Position reference	[1 / 2 / 3] 1 = top 2 = center 3 = bottom	
Dual Dimension: Values above-one-another display mode, Justification	[1 / 2 / 3] 1 = left 2 = center 3 = right	

Cumulate Dimensions (Ordinate Dimensions): General Parameters

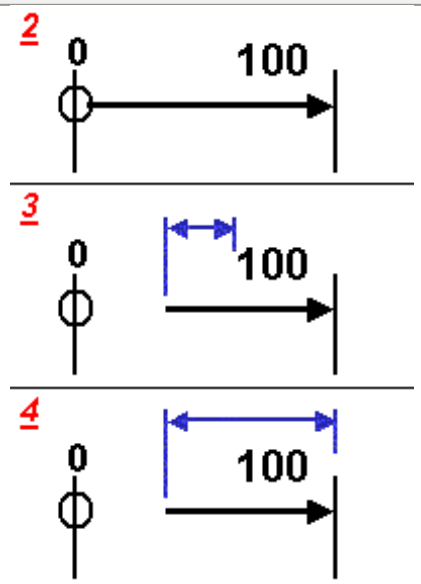
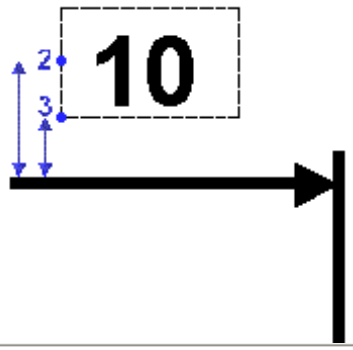
Table 1

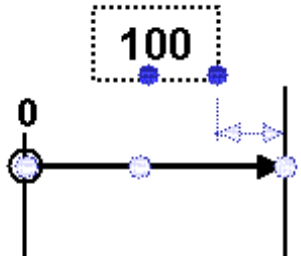
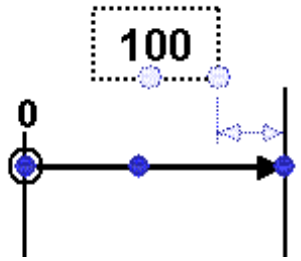
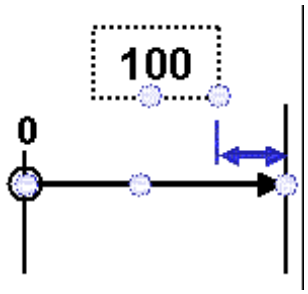
Parameter	Value	Description
Cumulate Dimension: Sign display	[1 / 3] 1 = no sign 3 = positive sign on all values	
Cumulate Dimension: Origin symbol shape	[0 / ... / 13] 0 = none 1-13 = refer to the dimension line symbols table	
Cumulate Dimension: Origin symbol scale	(real)	

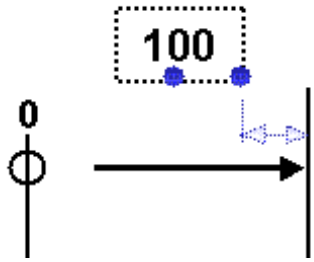
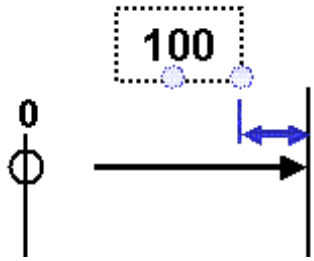
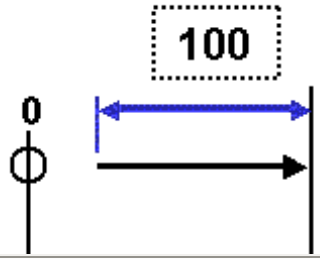
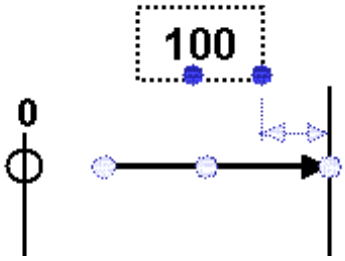
Cumulate Dimension: Extension line display	[Yes / No] Yes = display No = no display	
Cumulate Dimension: Display of origin zero	[Yes / No] Yes = display No = no display	
Cumulate Dimension: Value orientation reference	[1 / 2] 1 = dimension line 2 = extension line	
Cumulate Dimension: Value orientation	[1 / 2 / 3] 1 = Parallel to Reference (specified by Cumulate Dimension: Value orientation reference) 2 = Perpendicular to Reference (specified by Cumulate Dimension: Value orientation reference) 3 = Angle to reference	

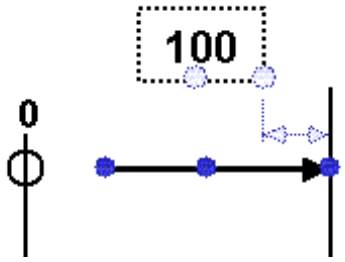
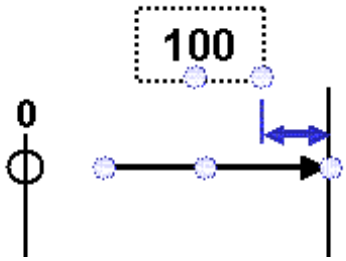
Cumulate Dimension: Value orientation angle	(degrees)	if Cumulate Dimension: Value orientation=3 

Cumulate Dimensions: Parameters applying only if value orientation reference is "Dimension Line" (Cumulate Dimension: Value orientation reference = 1)

Table 2		
Cumulate Dimension: Dimension line length mode	[2 / 3 / 4] 2 = Dimension Line to origin 3 = Length is relative to value text 4 = Length is constant	
Cumulate Dimension: Value vertical positioning, Justification	[1 / 2] 1 = Edge 2 = Center	
CUMLTxtDecalY	-	**Deprecated** Now managed by the **Value** > **OffsetY** parameter available for each Dimension Style.
If Dimension Line goes to origin (Cumulate Dimension: Dimension line length mode = 2):		

Cumulate Dimension: Value horizontal positioning, Justification	[1 / 2] 1 = Edge 2 = Center	For dimensions created before R14:  For dimensions created in R14 and R15: see the note below. For dimensions created from R16 onwards: this parameter is used if the dimension values alignment mode for cumulated dimension systems is defined as "From standard" in Dimension System Styles.
Cumulate Dimension: Value positioning reference	[1 / 2 / 3] 1 = Extension line 2 = Dim line center 3 = Origin	For dimensions created before R14:  For dimensions created in R14 and R15: see the note below. For dimensions created from R16 onwards: this parameter is used if the dimension values alignment mode for cumulated dimension systems is defined as "From standard" in Dimension System Styles.
Cumulate Dimension: Value H/V positioning, Offset from orientation reference	(mm)	For dimensions created before R14:  For dimensions created in R14 and R15: see the note below. For dimensions created from R16 onwards: this parameter is used if the dimension values alignment mode for cumulated dimension systems is defined as "From standard" in Dimension System Styles.
If Dimension Line is relative to value (Cumulate Dimension: Dimension line length mode = 3):		

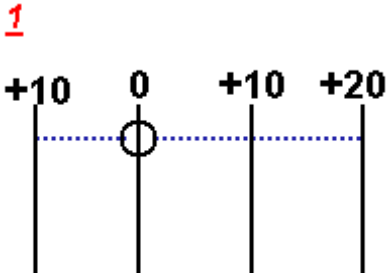
Cumulate Dimension: Value horizontal positioning, Justification	[1 / 2] 1 = Edge 2 = Center	For dimensions created before R14:  For dimensions created in R14 and R15: see the note below. For dimensions created from R16 onwards: this parameter is used if the dimension values alignment mode for cumulated dimension systems is defined as "From standard" in Dimension System Styles.
Cumulate Dimension: Value H/V positioning, Offset from orientation reference	(mm)	For dimensions created before R14:  For dimensions created in R14 and R15: see the note below. For dimensions created from R16 onwards: this parameter is used if the dimension values alignment mode for cumulated dimension systems is defined as "From standard" in Dimension System Styles.
If Dimension Line has a constant length (Cumulate Dimension: Dimension line representation = 4):		
Cumulate Dimension: Dimension/Extension line length	(mm)	
Cumulate Dimension: Value horizontal positioning, Justification	[1 / 2] 1 = Edge 2 = Center	For dimensions created before R14:  For dimensions created in R14 and R15: see the note below. For dimensions created from R16 onwards: this parameter is used if the dimension values alignment mode for cumulated dimension systems is defined as "From standard" in Dimension System Styles.

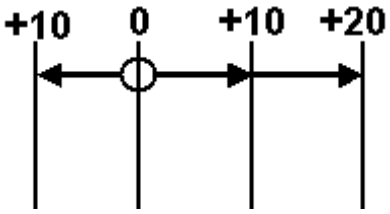
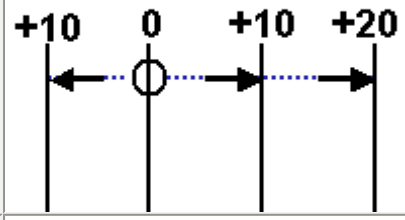
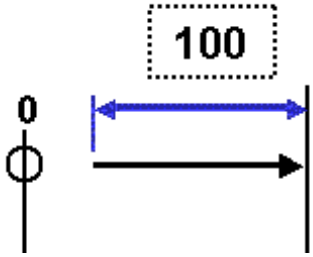
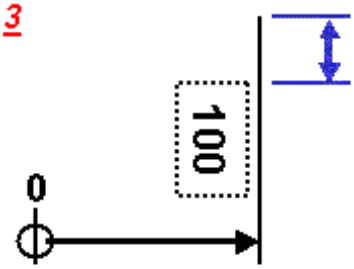
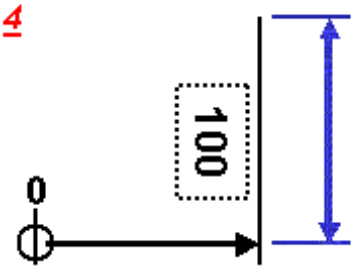
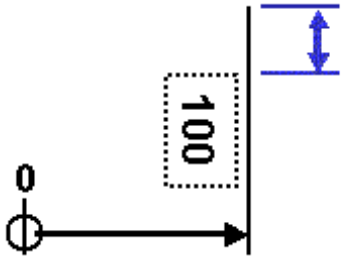
Cumulate Dimension: Value positioning reference	[1 / 2 / 3] 1 = Extension line 2 = Dim line center 3 = Origin	For dimensions created before R14:  For dimensions created in R14 and R15: see the note below. For dimensions created from R16 onwards: this parameter is used if the dimension values alignment mode for cumulated dimension systems is defined as "From standard" in Dimension System Styles.
Cumulate Dimension: Value H/V positioning, Offset from orientation reference	(mm)	For dimensions created before R14:  For dimensions created in R14 and R15: see the note below. For dimensions created from R16 onwards: this parameter is used if the dimension values alignment mode for cumulated dimension systems is defined as "From standard" in Dimension System Styles.

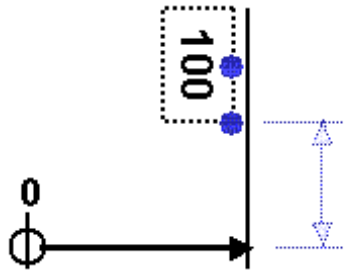
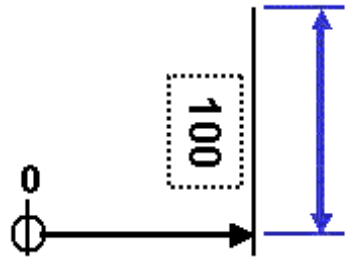
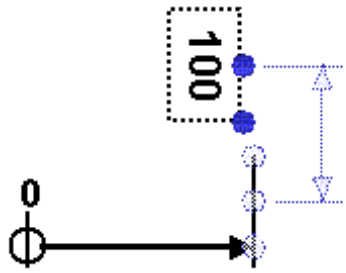
A number of parameters applying only if value orientation reference is "Dimension Line" were deprecated for dimensions created in R14 and R15. In this case, cumulate dimensions were created within dimension systems and if the dimension value orientation reference was the dimension line, then the value horizontal positioning was defined by the following Dimension System Styles: Aligned cumulated dimension values and Values Offset.

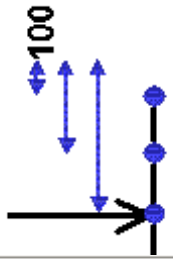
Cumulate Dimensions: Parameters applying only if the value orientation reference is "Extension Line" (Cumulate Dimension: Value orientation reference = 2)

Table 3

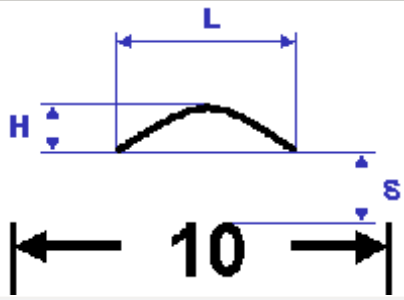
Cumulate Dimension: Dimension line representation	[1 / 2 / 4] 1 = no display 2 = full display 4 = partial length	

		2  4 
Cumulate Dimension: Dimension line length	(mm)	if Dimension Line has a partial length (Cumulate Dimension: Dimension line representation=4) 
Cumulate Dimension: Extension line length mode	[3 / 4] 3 = relative to text box 4 = constant	3  4 
If extension line is relative to value text (Cumulate Dimension: Extension line length mode = 3):		
Cumulate Dimension: Extension line overrun	(mm)	

Cumulate Dimension: Value vertical positioning, Justification 2	[1 / 2] 1 = Edge 2 = Center	
Cumulate Dimension: Value HV positioning, offset from orientation reference	(mm)	**Deprecated** Now managed by the **Value > OffsetX** parameter available for each Dimension Style
Cumulate Dimension: Value horizontal positioning, Justification	[1 / 2] 1 = Edge 2 = Center	
CUMLExtLTxtHPos	-	**Deprecated** Now managed by the **Value > OffsetY** parameter available for each Dimension Style
If extension line is constant (Cumulate Dimension: Extension line length mode = 4):		
Cumulate Dimension: Dimension/Extension line length	(mm)	
Cumulate Dimension: Value vertical positioning, Justification 2	[1 / 2] 1 = Edge 2 = Center	

Cumulate Dimension: Value positioning reference	[1 / 2 / 3] 1 = Dimension line 2 = Middle of extension line 3 = Extension line end point (opposite to dimension line)	
Cumulate Dimension: Value HV positioning, offset from orientation reference	(mm)	**Deprecated** Now managed by the **Value > OffsetX** parameter available for each Dimension Style.
Cumulate Dimension: Value horizontal positioning, Justification	[1 / 2] 1 = Edge 2 = Center	
CUMLExtLTxtHPos	(mm)	**Deprecated** Now managed by the **Value > OffsetY** parameter available for each Dimension Style.

Curvilinear Length Symbol

Option	Description
	
Display Symbol	Specifies whether the curvilinear length symbol should be displayed.
Height	Indicates the height (in mm) of the curvilinear length symbol.
Spacing	Indicates the spacing (in mm) between the curvilinear length symbol and the dimension value.
Underline value	Specifies whether the dimension value should be underlined.
Length	Indicates the length (in mm) of the curvilinear length symbol.
Minimum Length	Indicates the minimum length (in mm) of the curvilinear length symbol.
Minimum Length	Indicates the maximum length (in mm) of the curvilinear length symbol.

Intersection Point

Option	Description
Print intersection points	Specifies whether the intersection point should be printed. If you leave this option unchecked, then the intersection point will be a construction point and its style will be the default construction point style as defined in the Styles > Point > Default section of the standard. If you check this option, then the intersection point will not be a construction point and its style can be chosen among the various point styles defined in the Styles > Point section of the standard.
Point style	Indicates the style that should be used to represent the point (as defined in the Styles > Point section of the standard).
Show construction lines	Specifies whether construction lines should be displayed.
Print construction lines	Specifies whether construction lines should be printed. This option is available when the Show construction lines option is checked.
Line style	Specifies the style that should be used to represent the construction line (as defined in the LineTypes section of the standard).

Dimension Tolerance Formats

Format Definitions

This section deals with dimension tolerance descriptions, which are user-defined formats to be applied to dimension tolerances.

To create a new tolerance format, you must use the Standards editor. Select the **Tolerance Format** type in the standards editor, and then click the **Add Instance** button to add a new instance of a format. This will create a sample format definition that you will then customize to suit your needs, by modifying one or several values of the parameters defining the format.

Once defined, a format can be applied to dimensions just as any dimension attribute, either via **Edit -> Properties**, or using the Dimension Properties toolbar.

Dimension Tolerance Formats

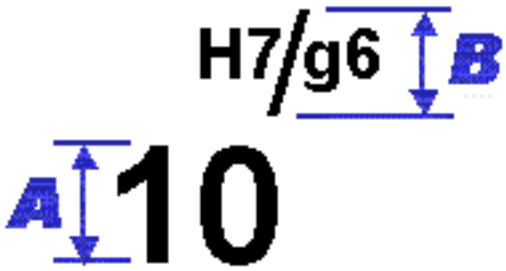
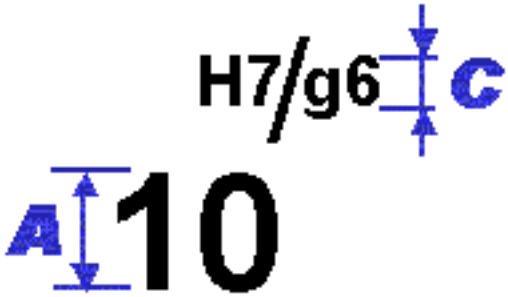
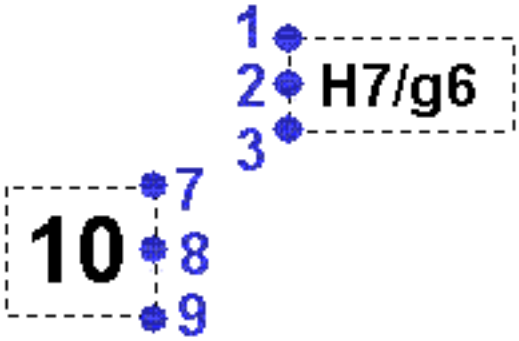
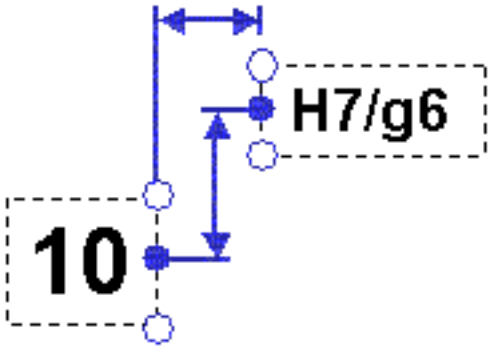
These parameters are located in the **Tolerance formats** node of the standard file.

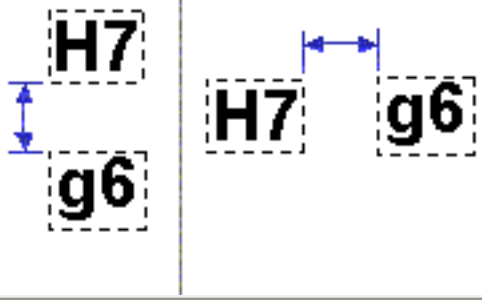
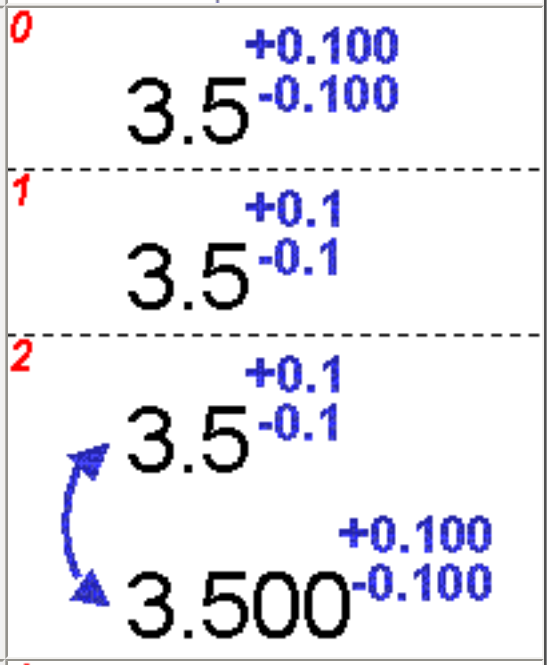
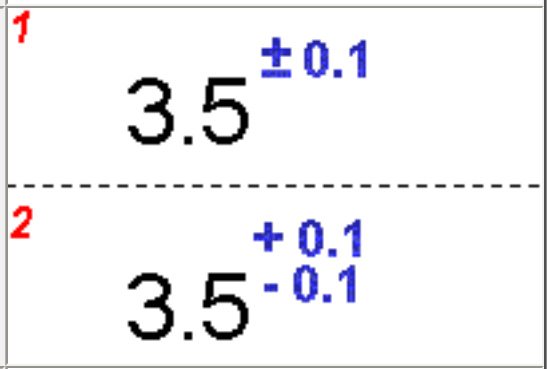
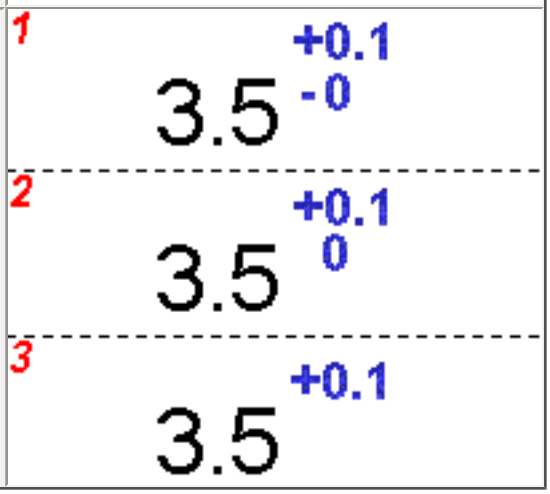
The tolerance format parameters drive the representation of a dimension tolerance, and include parameters such as:

- Type of tolerance (numerical/alphanumerical)
- Separator between values
- position relatively to dimension value
- font size for tolerance
- trailing zeros display for numerical type
- and so forth.

Parameter	Parameter Name	Value	Description
Tolerance Format Name	TolName	(8 char string)	User-defined name that will be used as the description identifier.

Tolerance Format Type	Toltype	[1 / 2 / 3 / 4 / 5 / 6 / 7] 1 = Numerical side by side 2 = Numerical super-imposed 3 = Resolved Numerical side by side 4 = Resolved numerical super-imposed 5 =Alphanumerical Single Value 6 = Alphanumerical side by side 7 = Alphanumerical super-imposed	1 10 +0.1 / -0.1 2 10 +0.1 -0.1 3 +10.1 / -9.9 4 +10.1 - 9.9 5 10 H8 6 10 H8 f7 7 10 H8 / f7
Separators for super-imposed tolerances	*Before* TolSepar_1 *After* TolSepar_2	[0...18] separator number as described in the Separator Character Table	(+0.1 -0.1)
Separators for side-by-side tolerances	*Before* TolSepTo_1 *Between* TolSepTo_2 *After* TolSepTo_3	[0...18] separator number as described in the Separator Character Table	(+0.1 / -0.1)

Fraction line on super-imposed tolerances	TolFractLine	[2/1] 2 = Fraction line 1 = No fraction line	
Separator Character Size (Ratio between Separator Character and Value Text font sizes)	TolSymbolH	(real) = separator height / value height (= B / A)	
Tolerance Size (Ratio between Tolerance Text and Value Text font sizes)	TolScale	(real) = tolerance height / value height (= C / A)	
Tolerance Position Anchor Point (for offset computing)	TolPtOnValue	[7 / 8 / 9] 7 =Top 8 = Middle 9 = Bottom	
	TolAnchorPt	[1 / 2 / 3] 1 =Top 2 = Middle 3 = Bottom	
Offset between dimension value and tolerance	TolExtX	(mm)	
	TolExtY		

Offset between the 2 tolerance values	TolIntX TolIntY	(mm)	
Display of tolerance trailing zeros	TolTrailing	[0 / 1 / 2] 0 = Display (number of digits specified in the value precision) 1 = No Display 2 = Same "display" mode as the dimension value	
Display of identical Tolerance Values (for numerical tolerances only)	TolMergeSame	[1 / 2] 1 = Display common value 2 = Display separate values	
Display of null Tolerance Values (for numerical tolerances only)	TolShowNull	[1 / 2 / 3] 1 = Display null value with sign 2 = Display null value without sign 3 = No Display of null value	

Separator Character Table

This table lists the characters that can be used as separators before, between or after the tolerance values.

Separators	
Symbol #	**Character**
00	(none)
01	/
02	:
03	(
04	)
05	"
06	,
07	<
08	>
09	X
10	*
11	.
12	;
13	+
14	[
15	]
16	-
17	_
18	(space)

Dimension Value Formats

Format Definitions

This section deals with dimension value descriptions, which are user-defined formats to be applied to dimension values.

To create a new dimension value display format, you must use the Standards editor. Select the **Value Formats** type in the standards editor, and then click the **Add Instance** button to add a new instance of a format. This will create a sample format definition that you will then customize to suit your needs, by modifying one or several values of the parameters defining the format.

Once defined, a format can be applied to dimensions just as any dimension attribute, either via **Edit -> Properties**, or using the Dimension Properties toolbar.

Dimension Value Display Formats

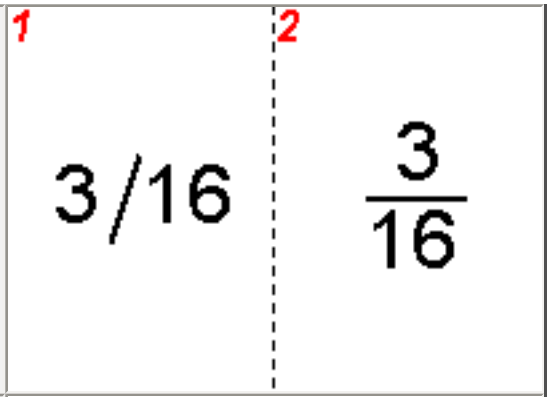
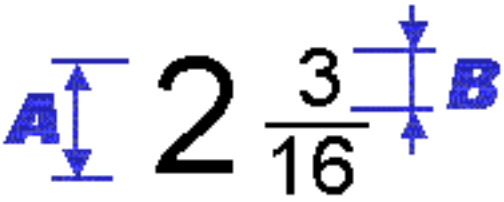
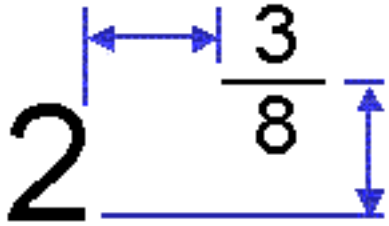
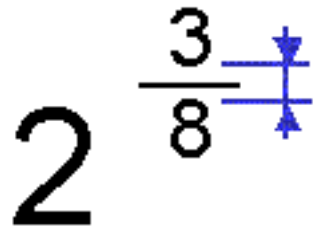
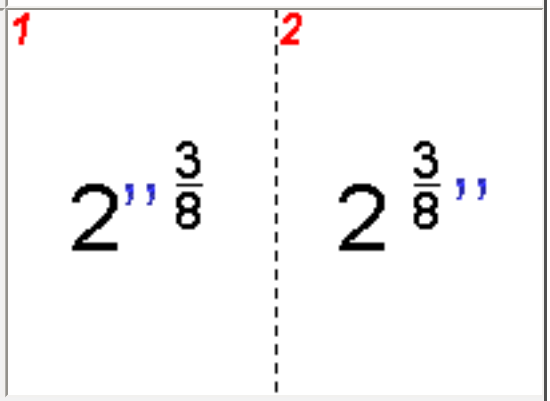
These parameters are located in the **Value Formats** node of the standard file.

The dimension value display style parameters drive the representation of a dimension value, and include parameters such as:

- multiplying factor
- separators for thousands
- position relatively to dimension line
- display of fractional values
- trailing zeros display
- and so forth.

Parameter	Parameter Name	Value	Description
Value Format Name	NDName	(8 char string)	User-defined name that will be used as the description identifier
Value Magnitude (type)	NDType	[1 / 2] 1 = length (for length/distance/radius/diameter dimensions) 2 = Angle (angle dimensions)	
Value Units	NDUnit	[1 / 2 / 3 / 4 / 5] 1 = mm 2 = inch 3 = radian 4 = degree 5 = grade	Unit used to display the dimension value

Global Multiplying Factor	NDGlobFact	(real)	The dimension measured value is multiplied by this factor prior to being displayed. For example, to display a distance in kilometers with units set to mm (NDUnit=1), use: NDGlobFact = 0.000001
Separator Characters for Decimal and Thousands	*Display of separator for Thousands* NDExise	[1 / 2] 1 = No display of separator 2 = Display of separator	1.000,25
	Decimal Separator NDSepNum *Thousands Separator* NDSep1000	[0...18] separator number as described in the Separator Character Table	
Display of Trailing Zeros	NDFinZer	[1 / 2] 1 = No display of trailing zeros 2 = Display of trailing zeros (number of digits specified in the value precision)	1 60,25 cm 2 60,250 cm
Fractional Rest Justification	NDAlignFrac	Not yet implemented	1 $\frac{3}{16}$ 2 $\frac{3}{16}$ 3 $\frac{3}{16}$

Fractional Rest Display Mode	NDTypFrac	[1 / 2] 1 = Side by side 2 = Super-imposed	
Fractional Rest Height Ratio	NDResScl	(real) = Unit height / value height (= B / A)	
Fractional Rest Positioning Offsets (the horizontal offset also applies to decimal rests)	NDRestX NDRestY	(real) This value is a ratio to the character height	
Offset between Fractional Rest Numerator and Denominator	NDOperY	(real) This value is a ratio to the character height	
Position of Last Term Unit	NDSepDen	[1 / 2] 1 = Before fractional rest 2 = After fractional rest	

Number of Terms in the Value	NDFact	[1...3]	1 160,25cm 2 1m 60,25cm 3 1m 60cm 2,5mm

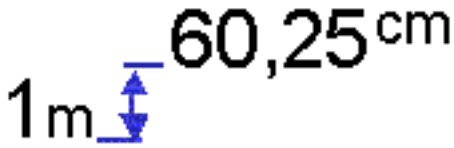
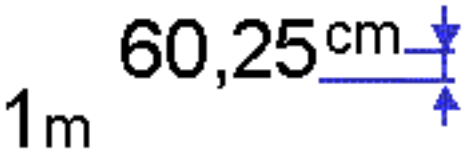
Definition of each of the value terms

A value can be made of up to three terms plus a rest. All of the following parameters, suffixed by the term number, apply to each of the possible 3 terms.
The numbering of the terms goes from right to left, #1 being the right-most term.

1m 60cm 2,5mm
3 2 1

Parameter	Parameter Name	Value	Description
Display of Null Terms	NDNulFac_1	[1 / 2] 1 = No display of zeros 2 = Display of zeros	1 60,25cm 2 0m 60,25cm

Display of Leading Zeros in Last Factor	NDNulFac_2	[1 / 2] 1 = No display of zeros 2 = Display of zeros	1 0,6025m 2 ,6025m
DEPRECATED	NDNulFac_3	-	-
Display of Null Terms	NDNulOther	[1 / 2] 1 = No display of zeros 2 = Display of zeros	1 2m 60,25cm 2 2m 60,250cm
Term Multiplying Factor	NDFact_1 NDFact_2 NDFact_3	(real)	The term measured value is multiplied by this factor prior to being displayed (the global multiplying factor is also used). All 3 values must have increasing and distinct values. Example: to display a value with a term in centimeters and a term in millimeters, with NDUnit=mm and NDGlobFact=1, set NDFact_1 = 1 NDFact_2 = 10
Term Unit Suffix	NDSepar_1 NDSepar_2 NDSepar_3	(16 char string)	1 m 60 cm 2,5 mm

Term Unit Height Ratio	NDSepScl_1 NDSepScl_2 NDSepScl_3	(real) = Unit height / value height (= B / A)	
Term Vertical Positioning Offset (relatively to the left-most term)	NDValPos_1 NDValPos_2 NDValPos_3	(mm)	
Term Unit Vertical Positioning Offset (relatively to its term)	NDSepPos_1 NDSepPos_2 NDSepPos_3	(mm)	

Pre-defined Formats for Tolerance and Dimension Values

Some basic formats are provided by default for dimension tolerance and value display. Some of these pre-defined formats can be modified while others cannot. All pre-defined formats can be de-activated (i.e. taken out of the list of available styles).

Modifiable formats

They appear in the default standard files provided by Dassault Systemes, just as any company defined style would appear. They can be modified or deleted using the Standards Editor, or de-activated (i.e. taken out of the list of available styles) using the Allowed* parameters described in the General Parameters section.

For Tolerance styles	For Value Display styles
TOL_RES1	micron
	mm
	cm
	m
	km
	in
	ftinch
	grade

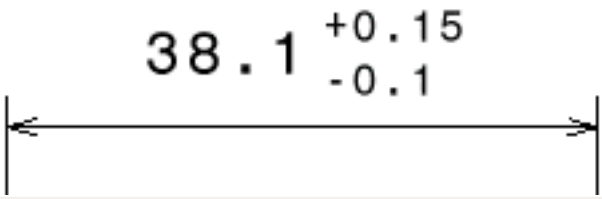
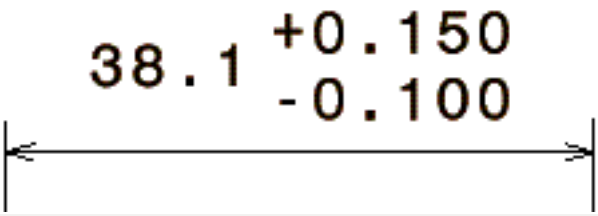
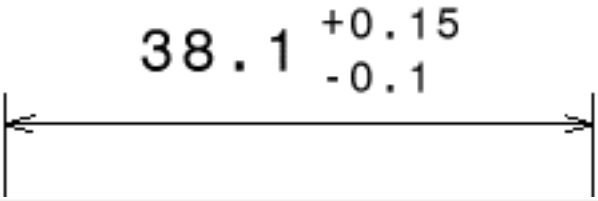
Non-modifiable formats

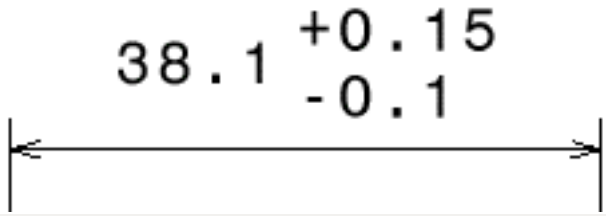
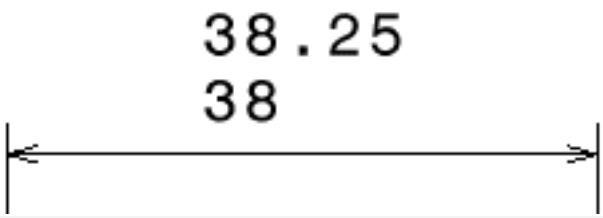
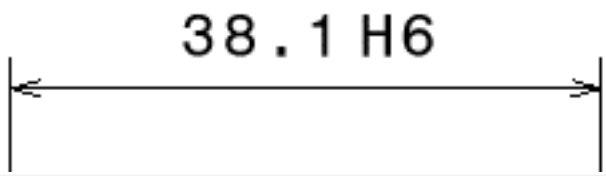
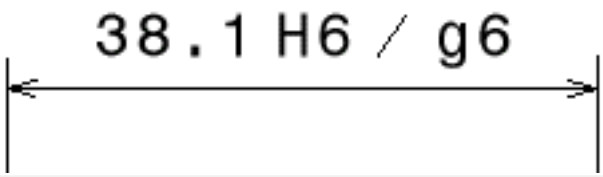
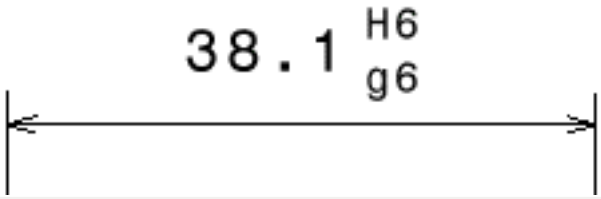
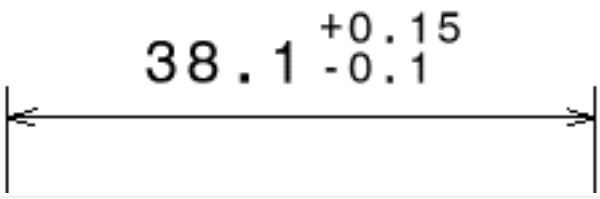
They are not defined in the standard file, but in the code itself. They cannot be modified, but can be de-activated (i.e. taken out of the list of available styles) using the Allowed* parameters described in the General Parameters section. All styles provided up to V5R8 are of this type.

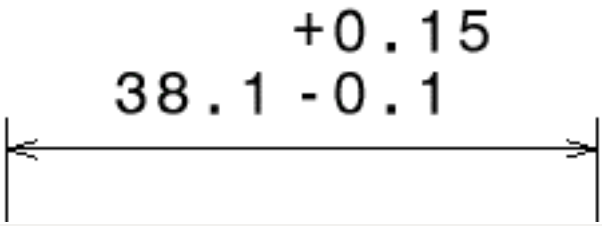
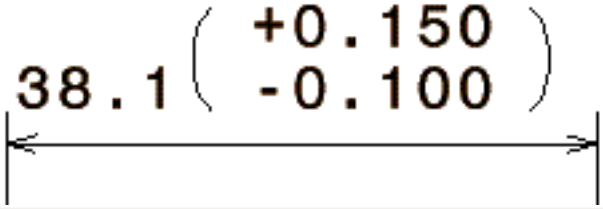
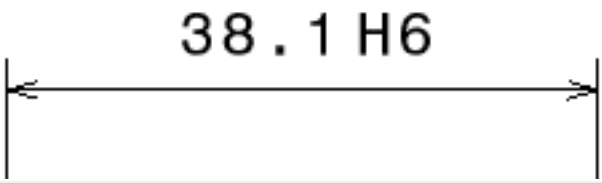
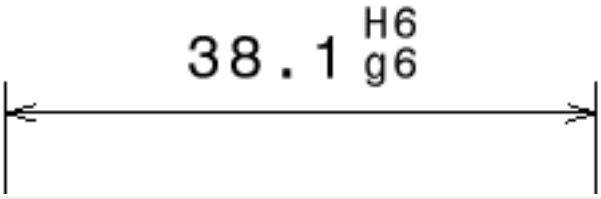
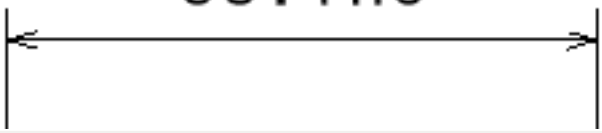
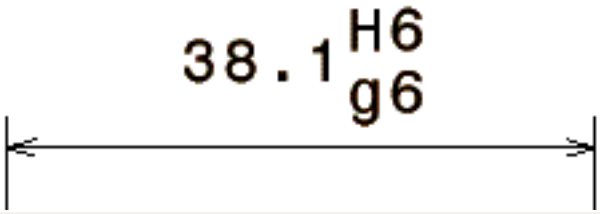
For Tolerance styles	For Value Display styles
TOL_NUM2	NUM.DIMM
ANS_NUM2	NUM,DIMM
DIN_NUM2	NUM.DINC
SGL_NUM2	NUM.DIMP
INC_NUM2	ANS.DIMM
TOL_RES2	DISTMM
TOL_ALP1	DISTINCH
TOL_ALP2	FEET-INC
TOL_ALP3	NUM.ADMS
TOL_0.7	NUM,ADMS
TOL_1.0	INC.ADMS
ISONUM	NUM.ARAD
ISOALPH1	ANGLEDEC
ISOALPH2	ANGLEDMS
CPL_FLA1	
CPL_FLA3	
CPL_50A1	
CPL_50A3	
CPL_75A1	
CPL_75A3	

The following tables list these non-modifiable styles, along with an example of the result when applied on a dimension. The right-most column contains a link to the style definition, from which you can derive new formats, simply by copying all or part of their definition.

Tolerance Formats

Name	Display	Description	Link to the style definition
TOL_NUM2		Numerical superimposed (small)	Click here
ANS_NUM2		Numerical superimposed with trailing zeros (large)	Click here
DIN_NUM2		Numerical superimposed (small)	Click here
SGL_NUM2		Numerical superimposed with trailing zeros and parentheses (small)	Click here

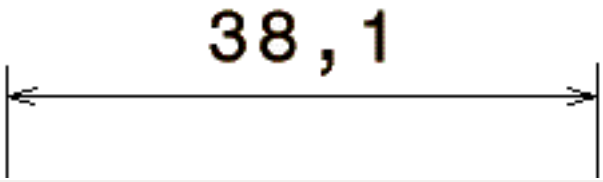
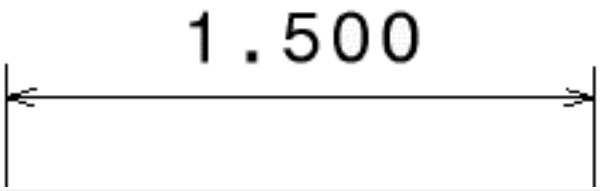
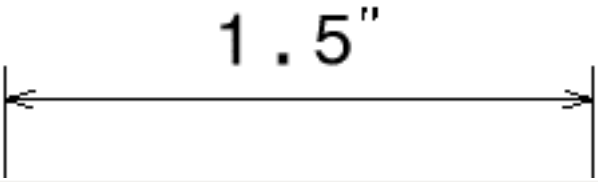
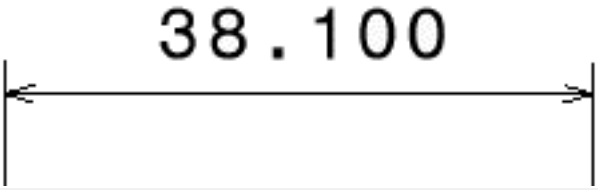
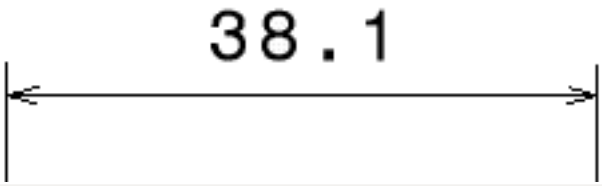
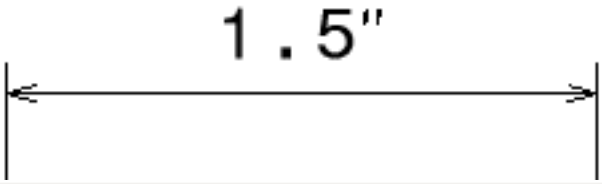
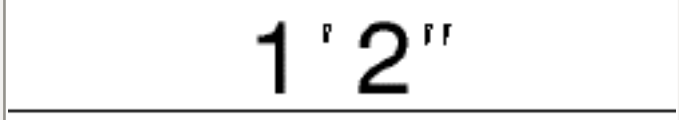
INC_NUM2		Numerical superimposed (large)	Click here
TOL_RES2		Numerical resolved	Click here
TOL_ALP1		Alphanumerical single value (large)	Click here
TOL_ALP2		Alphanumerical double value side-by-side (large)	Click here
TOL_ALP3		Alphanumerical double value superimposed (small)	Click here
TOL_0.7		Numerical superimposed (small)	Click here

TOL_1.0		Numerical superimposed (small)	Click here
ISONUM		Numerical superimposed with trailing zeros and parentheses (large)	Click here
ISOALPH1		Alphanumerical single value (large)	Click here
ISOALPH2		Alphanumerical double value superimposed (small)	Click here
CPL_FLA1		Alphanumerical single value (large)	Click here
CPL_FLA3		Alphanumerical double value superimposed (large)	Click here

CPL_50A1	38.1 H6	Alphanumerical single value (small)	Click here
CPL_50A3	38.1 H6 g6	Alphanumerical double value superimposed (small)	Click here
CPL_75A1	38.1 H6	Alphanumerical single value (medium)	Click here
CPL_75A3	38.1 H6 g6	Alphanumerical double value superimposed (medium)	Click here

Value Display Formats

Name	Display	Description	Link to the style definition
NUM.DIMM	38.1	Millimeters with dot	Click here

NUM,DIMM		Millimeters with comma	Click here
NUM.DINC		inches with trailing zeros	Click here
NUM.DIMP		inches with unit display	Click here
ANS.DIMM		Millimeters with trailing zeros	Click here
DISTMM		Millimeters with dot	Click here
DISTINC		inches with unit display	Click here
FEET-INC		feet and inch with unit display	Click here

NUM.ADMS	20°29'45.6"	Degrees/minutes/seconds with dot	Click here
NUM,ADMS	20°29'45,6"	Degrees/minutes/seconds with comma	Click here
INC.ADMS	20°29'45.600"	Degrees/minutes/seconds with dot and trailing zeros	Click here
NUM.ARAD	0.358	radians	Click here
ANGLEDEC	20.496°	Degrees with decimal format	Click here
ANGLEDMS	20°29'45.6"	Degrees/minutes/seconds with dot	Click here

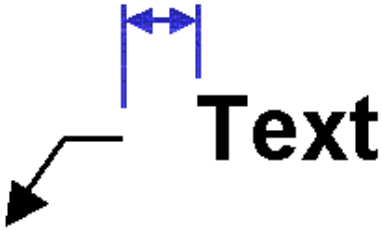
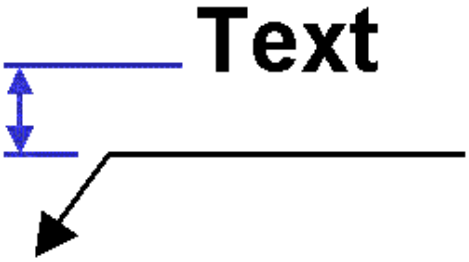
Annotation Parameters

The annotation parameters are located in the **Annotation** node of the standard editor. They deal with the position of text leaders.

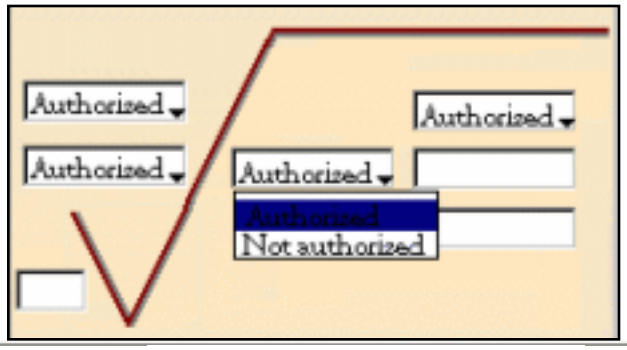
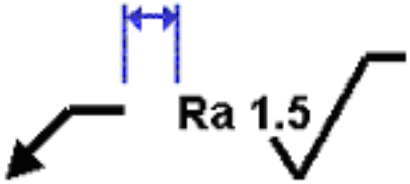
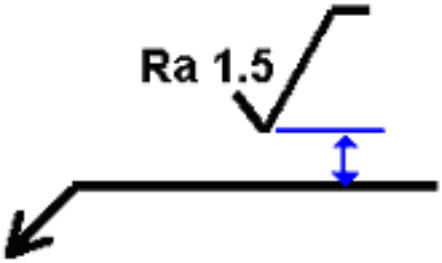
Note: The parameters which allow you to customize annotation leader symbols are located in the **Dimension** node of the standard editor. The parameters located in the **DressUp** node let you customize the appearance of dress-up elements, such as markup arrows and threads.

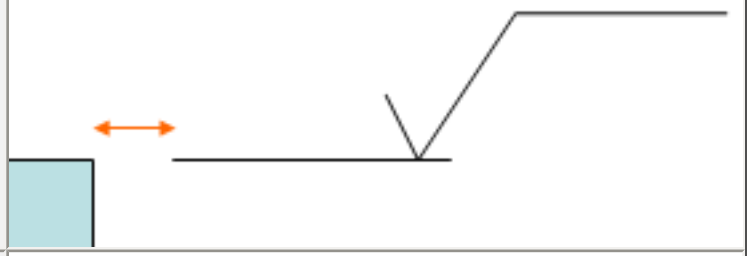
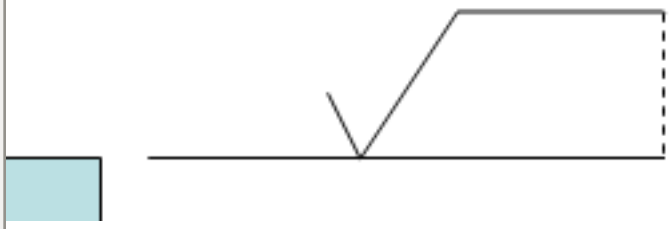
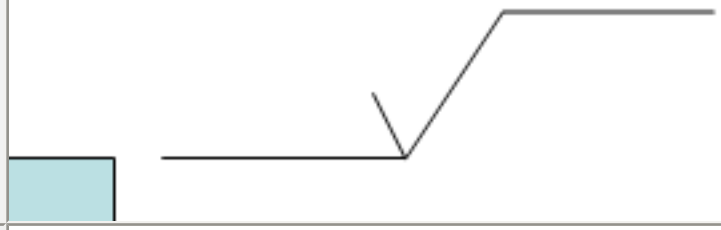
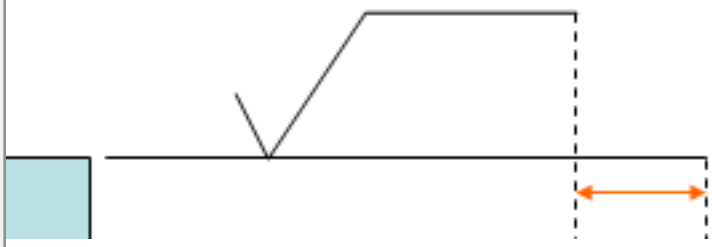
These parameters depend on a given parent standard.

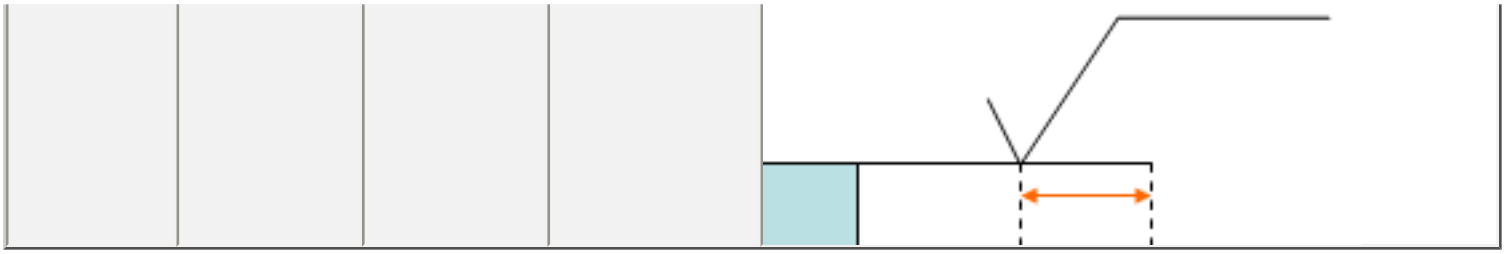
Annotation Texts

Parameter Name	Parameter	Parent standard	Value	Description
Text > Leader Gap	Horizontal offset between the text and the leader extremity	ANSI and ASME only	(mm)	
Text > Leader Vertical Space	Vertical offset between the bottom of the text and the horizontal part of the leader	ISO and JIS only	(mm)	

Roughness Symbols

Parameter Name	Parameter	Parent standard	Value	Description
Roughness > Layout	Layout of the roughness symbol	All standards	Authorized / Not authorized	Specifies whether a given field should be displayed (Authorized) or hidden (Not authorized) in the Roughness Symbol dialog box 
Roughness > Leader Gap	Horizontal offset between the roughness and the leader extremity	ANSI and ASME	(mm)	
Roughness > Leader Vertical Space	Vertical offset between the bottom of the roughness and the horizontal part of the leader	ISO and JIS only	(mm)	
Roughness > Extension Line > Thickness > Behavior	Behavior of the extension line thickness	All standards	Fixed / Variable with symbol	Specifies whether the thickness of the extension line should be fixed (using the value specified by the Line Thickness parameter below) or variable (using the graphic property of the thickness symbol itself).
Roughness > Extension Line > Thickness > Line Thickness	Extension line thickness index	All standards	Integer	Specifies the line thickness index (as defined in the LineThickness node of the current standard) that should be used to represent the extension line. This parameter is taken into account only if the Behavior parameter is set to Fixed.

Roughness > Extension Line > Gap to surface	Extension line gap to surface	All standards	(mm)	Specifies the gap between the extension line and the pointed object. 
Roughness > Extension Line > Length according to symbol	Extension line length defined according to symbol or not	All standards	Yes / No	Specifies whether the extension line length should be defined according to the roughness symbol length. With value set to Yes:  With value set to No: 
Roughness > Extension Line > Overrun	Extension line overrun	All standards	(mm)	Specifies the overrun of the extension line. When Length according to symbol parameter is set to Yes:  When Length according to symbol parameter is set to No:



Frame Definition

This section deals with fixed-size frame definition. A frame is a property which can be applied to texts as well as certain types of annotations and dress up elements.

Defining Frames

Fixed-size frame definitions are located in the **Frame** node of the Standards editor, available via **Tools -> Standards**. They specify the geometrical definition of fixed-size frames (as opposed to variable-size frame).

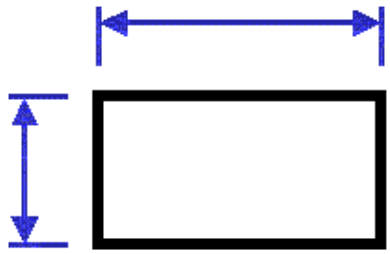
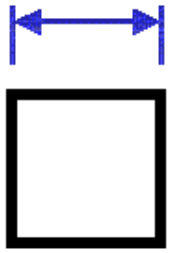
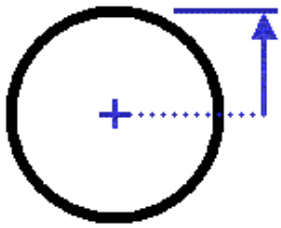
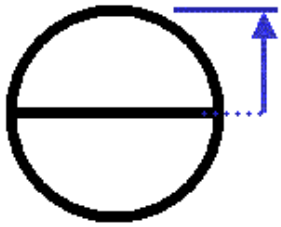
Frame definitions available in the Standards editor are pre-defined, and their number is fixed. You cannot add additional instances of frame definitions.

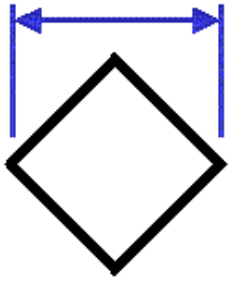
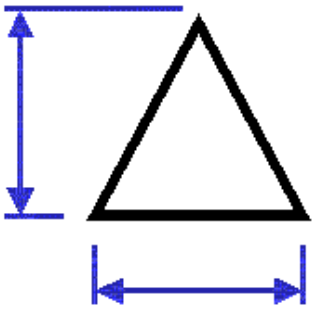
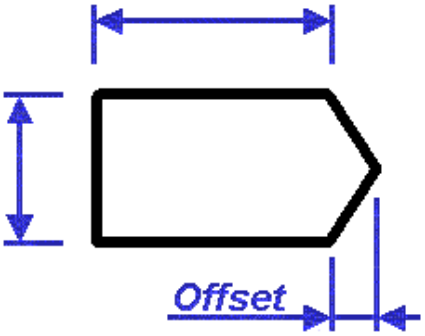
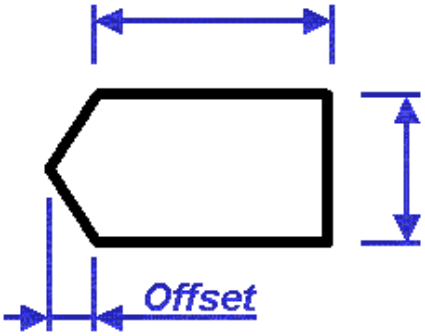
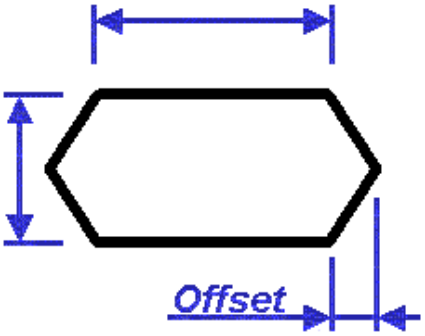
You can customize these definitions to suit your needs, by modifying one or several values of the parameters defining the style. Once defined, a fixed-size frame can be applied to any element which supports it, either via **Edit -> Properties**, or using the Text Properties toolbar.

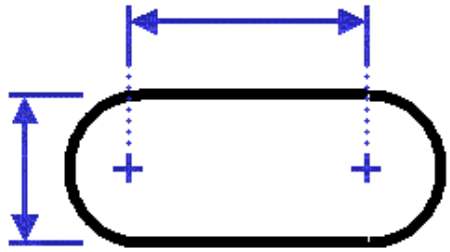
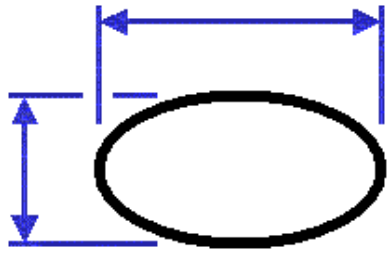
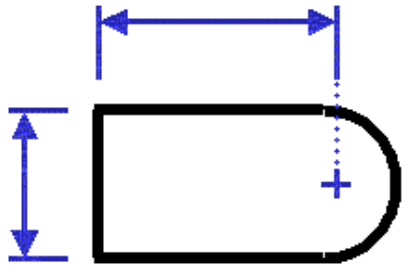
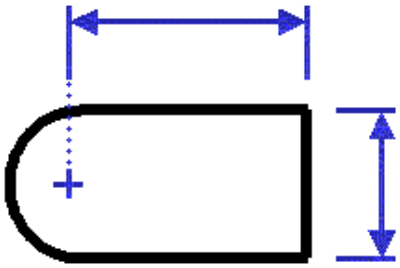
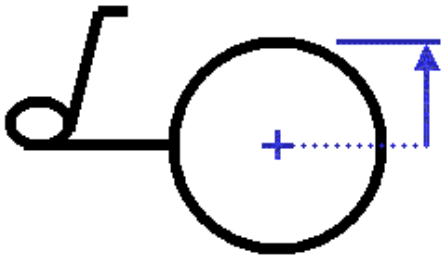
The fixed-size frame definitions include the following parameters:

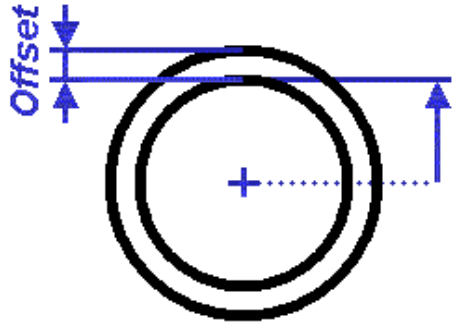
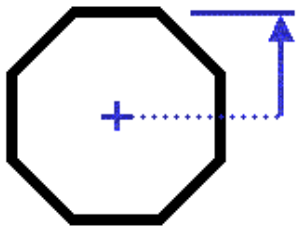
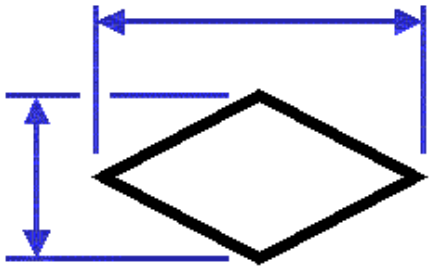
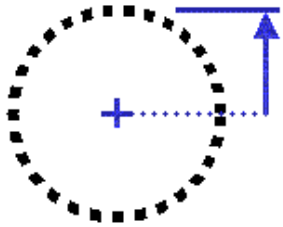
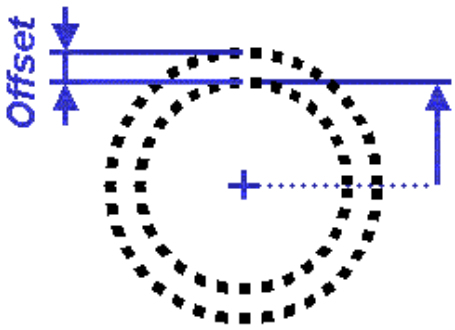
- Name: identifies the frame - DO NOT EDIT
- Type: defines the geometrical type of the frame - DO NOT EDIT
- Behavior - DO NOT EDIT
- Length
- Height
- Radius
- Offset
- Vertical Margin - NOT YET IMPLEMENTED
- Horizontal Margin - NOT YET IMPLEMENTED

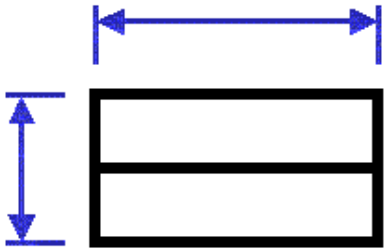
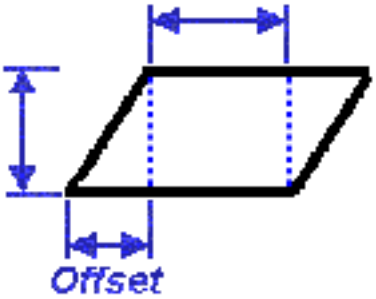
- The Name, Type and Behavior parameters MUST NOT BE EDITED, and are listed for information and compliance purposes only.
- The Vertical Margin and Horizontal Margin parameters are not implemented yet, and are listed for compliance purposes only.
- For each frame definition, all parameters are listed. However, depending on the frame type, not all parameters are used to define the frame, but only some of them.

Parameter	Length	Height	Radius	Offset	Description
Rectangle	Yes	Yes	N/A	N/A	
Square	Yes	N/A	N/A	N/A	
Circle	N/A	N/A	Yes	N/A	
ScoredCircle	N/A	N/A	Yes	N/A	

Diamond	Yes	N/A	N/A	N/A	
Triangle	Yes	Yes	N/A	N/A	
RightFlag	Yes	Yes	N/A	Yes	
LeftFlag	Yes	Yes	N/A	Yes	
BothFlag	Yes	Yes	N/A	Yes	

Oblong	Yes	Yes	N/A	N/A	
Ellipse	Yes	Yes	N/A	N/A	
RightOblong	Yes	Yes	N/A	N/A	
LeftOblong	Yes	Yes	N/A	N/A	
Sticking	N/A	N/A	Yes	N/A	

Set	N/A	N/A	Yes	Yes	
Fixed support	N/A	N/A	Yes	N/A	
Nota	Yes	Yes	N/A	N/A	
SymPart	N/A	N/A	Yes	N/A	
SymSet	N/A	N/A	Yes	Yes	

ScoredRectangle	Yes	Yes	N/A	N/A	
Parallelogram	Yes	Yes	N/A	Yes	

View Generation Definition

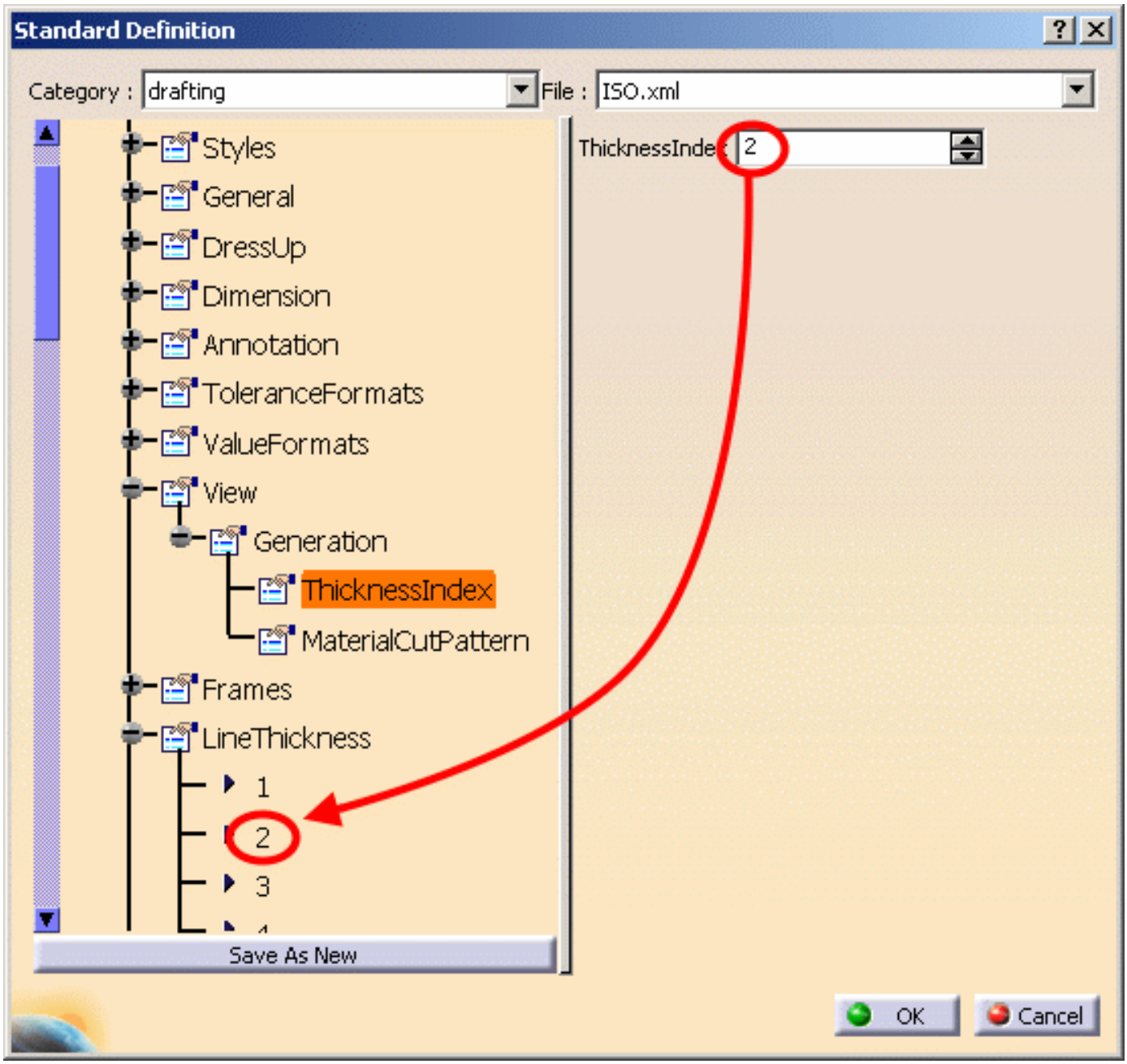
This section deals with view generation definition. This lets you customize settings that should be applied when generating views in a Generative Drafting context.

Defining View Generation Parameters

The view generation definition parameters are located in the **View -> Generation** node of the Standards editor, available via **Tools -> Standards**.

There are two parameters:

- **ThicknessIndex**: this parameter lets you customize the line thickness for geometry which is automatically generated in views (this includes all geometry except fillet edges).
  Specify the number of the line thickness definition parameter, as specified in the **Line Thickness** node of the Standards editor. For more information, refer to Line Thickness Definition.



- **MaterialCutPattern**: this parameter is used when generating views from parts which use a material to which a specific pattern is associated.
  - Select **Material** to use the pattern associated to a given material (instead of the patterns defined in the standards), even if this pattern is not defined in the standards.
  - Select **Standard** to use standard patterns only, instead of the pattern associated to a given material. Refer to Pattern Definition for more information on defining standard patterns.

Line Thickness Definition

This section deals with line thickness definition. Line thickness is a property which can be applied to, and drives the representation of, almost all elements in a drawing, such as lines, curves, dimension lines, etc. (Line thickness cannot be applied to fonts and points).

- In releases up to V5 R9 SP2, line thickness used to be defined in **Tools -> Options -> General -> Display -> Thickness & Font** for the Drafting workbench as well as for other workbenches. For Drafting, line thickness is now defined in standards. Therefore, line thickness in drawings does not depend on the options defined in **Tools -> Options**, but on what is defined in the standards.
- When opening a drawing created with releases up to V5 R9 SP2 (i.e. a drawing which does not contain its own line thickness parameters), the line thickness options defined in **Tools -> Options** will be used. You can upgrade a CATDrawing document to this new standard format at any time, by performing the following operations in **File -> Page Setup**:
  - changing the standard to another standard (ISO -> ANSI for instance)
  - updating the current standard to the new format.

The information below is specifically intended for administrators customizing standards for the 2D Layout for 3D Design workbench:

- When displaying line thicknesses in the 2D Layout for 3D Design window: the line thicknesses which are used are those defined in standards.
  This is the case of all elements displayed in the 2D or 3D background of a view in the 2D Layout for 3D Design window, whether they are:
  - layout elements (geometry, annotations and so on),
  - 3D wireframe elements (lines, points and so on).
- When displaying line thicknesses in the 3D window (Part Design, for example): the line thicknesses which are used are those defined in the settings (through **Tools -> Options -> General -> Display -> Thickness & Font**).
  This is the case of all layout elements (geometry, annotations and so on), when displayed in the 3D window.
- When printing a layout: the line thicknesses which are used are those defined in standards.
- When editing part layouts in the context of a product: the line thicknesses which are used are those defined in the standards of the current layout (even when visualizing elements which belong to another layout).
  For example, take the following scenario, where:
  - the layout of Part.1 uses ISO_3D
  - the layout of Part.2 uses JIS_3D
  - the line thickness definitions are different in ISO_3D and JIS_3D.

  In this case, when visualizing Part.2 in the background of the Part.1 layout, Part.2 will be displayed using the line thickness definitions of Part.1.

Therefore, if you want the visualization of elements to be homogeneous in the 2D and 3D windows, you need to make sure that line thickness definitions in the standards match line thickness definitions in the settings.

Defining Line Thickness

Line thickness definitions are located in the **Line Thickness** node of the Standards editor, available via **Tools -> Standards**.

There are 55 line thickness definitions in the Standards editor. You cannot add additional instances of line thickness definitions. Out of these 55 definitions,

- line thickness definitions ranging from 1 to 8 are pre-defined with different parameters for each, and available.
- line thickness definitions ranging from 9 to 55 are pre-defined with the same parameters for all, and unavailable.

You can customize these definitions to suit your needs, by modifying one or several values of the parameters defining the style. Once defined, a thickness can be applied to any element which supports it, either via **Edit -> Properties**, or using the Graphic Properties toolbar.

Parameter	Description
Availability	Indicates whether this definition is available for users to choose from: 1 = available 0 = unavailable.
Pixels	Specifies the size in pixels, with a maximum of 16; reflects the result displayed on screen.
mm	Specifies the size in millimeters; corresponds to the printed version.

The Availability parameter specifies whether or not a given line thickness should be available in the thickness list for users to choose from, when creating or editing elements. Users will only be able to assign "available" line thickness definitions to these elements. However, existing element properties in drawings will not be affected: if an existing element is assigned a line thickness which is flagged as "unavailable" in the Standards editor, then this line thickness will be used for this element but it will not be available in the thickness list, so that users cannot apply it to other elements.

Pre-defined Styles Definition

This section lists pre-defined non-modifiable styles along with their definition. You can use these styles as a reference when defining new formats, simply by copying all or part of their definition.

```
TolName= TOL_NUM2
TolType= 2
TolSepar_1= 0
TolSepar_2= 0
TolSymbolH= 1.0
TolSepTo_1= 0
TolSepTo_2= 0
TolSepTo_3= 0
TolTrailing= 2
TolFractLine= 1
TolPtOnValue= 8
TolAnchorPt= 2
TolIntX= 0.0
TolIntY= 0.6
TolExtX= 0.6
TolExtY= 0.0
TolMergeSame= 1
TolShowNull= 2
TolScale= 0.7
```

```
TolName= ANS_NUM2
TolType= 2
TolSepar_1= 0
TolSepar_2= 0
TolSymbolH= 1.0
TolSepTo_1= 0
TolSepTo_2= 0
TolSepTo_3= 0
TolTrailing= 0
TolFractLine= 1
TolPtOnValue= 8
TolAnchorPt= 2
TolIntX= 0.0
TolIntY= 0.6
TolExtX= 0.6
TolExtY= 0.0
TolMergeSame= 1
TolShowNull= 2
TolScale= 1.0
```

```
TolName= DIN_NUM2
TolType= 2
TolSepar_1= 0
TolSepar_2= 0
TolSymbolH= 1.0
```

```
TolSepTo_1= 0
TolSepTo_2= 0
TolSepTo_3= 0
TolTrailing= 2
TolFractLine= 1
TolPtOnValue= 8
TolAnchorPt= 2
TolIntX= 0.0
TolIntY= 0.6
TolExtX= 0.6
TolExtY= 0.0
TolMergeSame= 1
TolShowNull= 3
TolScale= 0.7


TolName= SGL_NUM2
TolType= 2
TolSepar_1= 3
TolSepar_2= 4
TolSymbolH= 2.0
TolSepTo_1= 0
TolSepTo_2= 0
TolSepTo_3= 0
TolTrailing= 0
TolFractLine= 1
TolPtOnValue= 8
TolAnchorPt= 2
TolIntX= 0.0
TolIntY= 0.6
TolExtX= 0.6
TolExtY= 0.0
TolMergeSame= 1
TolShowNull= 2
TolScale= 0.7


TolName= INC_NUM2
TolType= 2
TolSepar_1= 0
TolSepar_2= 0
TolSymbolH= 1.0
TolSepTo_1= 0
TolSepTo_2= 0
TolSepTo_3= 0
TolTrailing= 2
TolFractLine= 1
TolPtOnValue= 8
TolAnchorPt= 2
TolIntX= 0.0
TolIntY= 0.6
TolExtX= 0.6
TolExtY= 0.0
TolMergeSame= 1
TolShowNull= 1
TolScale= 1.0
```

```
TolName= TOL_RES2
TolType= 4
TolSepar_1= 0
TolSepar_2= 0
TolSymbolH= 1.0
TolSepTo_1= 0
TolSepTo_2= 0
TolSepTo_3= 0
TolTrailing= 2
TolFractLine= 1
TolPtOnValue= 9
TolAnchorPt= 3
TolIntX= 0.0
TolIntY= 0.6
TolExtX= 0.0
TolExtY= 0.0
TolMergeSame= 1
TolShowNull= 2
TolScale= 1.0


TolName= TOL_ALP1
TolType= 5
TolSepar_1= 0
TolSepar_2= 0
TolSymbolH= 1.0
TolSepTo_1= 0
TolSepTo_2= 0
TolSepTo_3= 0
TolTrailing= 0
TolFractLine= 0
TolPtOnValue= 9
TolAnchorPt= 3
TolIntX= 0.0
TolIntY= 0.0
TolExtX= 0.6
TolExtY= 0.0
TolMergeSame= 0
TolShowNull= 0
TolScale= 1.0


TolName= TOL_ALP2
TolType= 6
TolSepar_1= 0
TolSepar_2= 0
TolSymbolH= 1.0
TolSepTo_1= 0
TolSepTo_2= 1
TolSepTo_3= 0
TolTrailing= 0
TolFractLine= 0
TolPtOnValue= 9
TolAnchorPt= 3
TolIntX= 0.6
```

```
TolIntY= 0.0
TolExtX= 0.6
TolExtY= 0.0
TolMergeSame= 0
TolShowNull= 0
TolScale= 1.0


TolName= TOL_ALP3
TolType= 7
TolSepar_1= 0
TolSepar_2= 0
TolSymbolH= 1.0
TolSepTo_1= 0
TolSepTo_2= 0
TolSepTo_3= 0
TolTrailing= 0
TolFractLine= 1
TolPtOnValue= 8
TolAnchorPt= 2
TolIntX= 0.0
TolIntY= 0.6
TolExtX= 0.6
TolExtY= 0.0
TolMergeSame= 0
TolShowNull= 0
TolScale= 0.7


TolName= TOL_0.7
TolType= 2
TolSepar_1= 0
TolSepar_2= 0
TolSymbolH= 1.0
TolSepTo_1= 0
TolSepTo_2= 0
TolSepTo_3= 0
TolTrailing= 2
TolFractLine= 1
TolPtOnValue= 9
TolAnchorPt= 3
TolIntX= 0.0
TolIntY= 0.250000
TolExtX= 0.5
TolExtY= 0.0
TolMergeSame= 1
TolShowNull= 3
TolScale= 0.715000


TolName= TOL_1.0
TolType= 2
TolSepar_1= 0
TolSepar_2= 0
TolSymbolH= 1.0
TolSepTo_1= 0
```

```
TolSepTo_2= 0
TolSepTo_3= 0
TolTrailing= 2
TolFractLine= 1
TolPtOnValue= 9
TolAnchorPt= 3
TolIntX= 0.0
TolIntY= 0.5
TolExtX= 0.5
TolExtY= 0.0
TolMergeSame= 1
TolShowNull= 2
TolScale= 1.0


TolName= ISONUM
TolType= 2
TolSepar_1= 3
TolSepar_2= 4
TolSymbolH= 2.5
TolSepTo_1= 0
TolSepTo_2= 0
TolSepTo_3= 0
TolTrailing= 0
TolFractLine= 1
TolPtOnValue= 9
TolAnchorPt= 3
TolIntX= 0.0
TolIntY= 0.5
TolExtX= -0.5
TolExtY= 0.0
TolMergeSame= 2
TolShowNull= 2
TolScale= 1.0


TolName= ISOALPH1
TolType= 5
TolSepar_1= 0
TolSepar_2= 0
TolSymbolH= 1.0
TolSepTo_1= 0
TolSepTo_2= 0
TolSepTo_3= 0
TolTrailing= 0
TolFractLine= 0
TolPtOnValue= 9
TolAnchorPt= 3
TolIntX= 0.0
TolIntY= 0.5
TolExtX= 0.5
TolExtY= 0.0
TolMergeSame= 0
TolShowNull= 0
TolScale= 1.0
```

```
TolName= ISOALPH2
TolType= 7
TolSepar_1= 0
TolSepar_2= 0
TolSymbolH= 1.0
TolSepTo_1= 0
TolSepTo_2= 0
TolSepTo_3= 0
TolTrailing= 0
TolFractLine= 1
TolPtOnValue= 9
TolAnchorPt= 3
TolIntX= 0.0
TolIntY= 0.250000
TolExtX= 0.5
TolExtY= 0.0
TolMergeSame= 0
TolShowNull= 0
TolScale= 0.715000


TolName= CPL_FLA1
TolType= 5
TolSepar_1= 0
TolSepar_2= 0
TolSymbolH= 25.4
TolSepTo_1= 0
TolSepTo_2= 0
TolSepTo_3= 0
TolTrailing= 0
TolFractLine= 0
TolPtOnValue= 9
TolAnchorPt= 3
TolIntX= 0.0
TolIntY= 0.0
TolExtX= 0.285714
TolExtY= 0.0
TolMergeSame= 0
TolShowNull= 0
TolScale= 1.0


TolName= CPL_FLA3
TolType= 7
TolSepar_1= 0
TolSepar_2= 0
TolSymbolH= 1.0
TolSepTo_1= 0
TolSepTo_2= 0
TolSepTo_3= 0
TolTrailing= 0
TolFractLine= 1
TolPtOnValue= 8
TolAnchorPt= 2
TolIntX= 0.0
TolIntY= 0.5
```

```
TolExtX= 0.285714
TolExtY= 0.0
TolMergeSame= 0
TolShowNull= 0
TolScale= 1.0


TolName= CPL_50A1
TolType= 5
TolSepar_1= 0
TolSepar_2= 0
TolSymbolH= 25.4
TolSepTo_1= 0
TolSepTo_2= 0
TolSepTo_3= 0
TolTrailing= 0
TolFractLine= 0
TolPtOnValue= 9
TolAnchorPt= 3
TolIntX= 0.0
TolIntY= 0.0
TolExtX= 0.214286
TolExtY= 0.250000
TolMergeSame= 0
TolShowNull= 0
TolScale= 0.5


TolName= CPL_50A3
TolType= 7
TolSepar_1= 0
TolSepar_2= 0
TolSymbolH= 1.0
TolSepTo_1= 0
TolSepTo_2= 0
TolSepTo_3= 0
TolTrailing= 0
TolFractLine= 1
TolPtOnValue= 8
TolAnchorPt= 2
TolIntX= 0.0
TolIntY= 0.250000
TolExtX= 0.214286
TolExtY= 0.0
TolMergeSame= 0
TolShowNull= 0
TolScale= 0.5


TolName= CPL_75A1
TolType= 5
TolSepar_1= 0
TolSepar_2= 0
TolSymbolH= 25.4
TolSepTo_1= 0
TolSepTo_2= 0
```

```
TolSepTo_3= 0
TolTrailing= 0
TolFractLine= 0
TolPtOnValue= 9
TolAnchorPt= 3
TolIntX= 0.0
TolIntY= 0.0
TolExtX= 0.250000
TolExtY= 0.125000
TolMergeSame= 0
TolShowNull= 0
TolScale= 0.750000


TolName= CPL_75A3
TolType= 7
TolSepar_1= 0
TolSepar_2= 0
TolSymbolH= 25.4
TolSepTo_1= 0
TolSepTo_2= 0
TolSepTo_3= 0
TolTrailing= 0
TolFractLine= 1
TolPtOnValue= 8
TolAnchorPt= 2
TolIntX= 0.0
TolIntY= 0.375000
TolExtX= 0.250000
TolExtY= 0.0
TolMergeSame= 0
TolShowNull= 0
TolScale= 0.750000


NDName= NUM.DIMM
NDType= 1
NDUnit= 1
NDGlobFact= 1.0
NDNulFac_1= 1
NDNulFac_2= 2
NDExise= 1
NDSep1000= 0
NDFact_1= 1.0
NDFact_2= 0.0
NDFact_3= 0.0
NDValPos_1= 0.0
NDValPos_2= 0.0
NDValPos_3= 0.0
NDSepar_1=
NDSepar_2=
NDSepar_3=
NDSepScl_1= 1.0
NDSepScl_2= 0.0
NDSepScl_3= 0.0
NDSepPos_1= 0.0
```

```
NDSepPos_2= 0.0
NDSepPos_3= 0.0
NDRestY= 0.0
NDFinZer= 1
NDSepNum= 11
NDTypFrac= 2
NDSepDen= 2
NDOperY= 0.5
NDNulOther= 1
NDResScl= 1.0
NDFact= 1
NDRestX= 0.5


NDName= NUM,DIMM
NDType= 1
NDUnit= 1
NDGlobFact= 1.0
NDNulFac_1= 1
NDNulFac_2= 2
NDExise= 1
NDSep1000= 0
NDFact_1= 1.0
NDFact_2= 0.0
NDFact_3= 0.0
NDValPos_1= 0.0
NDValPos_2= 0.0
NDValPos_3= 0.0
NDSepar_1=
NDSepar_2=
NDSepar_3=
NDSepScl_1= 1.0
NDSepScl_2= 0.0
NDSepScl_3= 0.0
NDSepPos_1= 0.0
NDSepPos_2= 0.0
NDSepPos_3= 0.0
NDRestY= 0.0
NDFinZer= 1
NDSepNum= 6
NDTypFrac= 2
NDSepDen= 2
NDOperY= 0.5
NDNulOther= 1
NDResScl= 1.0
NDFact= 1
NDRestX= 0.5


NDName= NUM.DINC
NDType= 1
NDUnit= 2
NDGlobFact= 1.0
NDNulFac_1= 1
NDNulFac_2= 1
NDExise= 1
NDSep1000= 0
```

```
NDFact_1= 1.0
NDFact_2= 0.0
NDFact_3= 0.0
NDValPos_1= 0.0
NDValPos_2= 0.0
NDValPos_3= 0.0
NDSepar_1=
NDSepar_2=
NDSepar_3=
NDSepScl_1= 1.0
NDSepScl_2= 0.0
NDSepScl_3= 0.0
NDSepPos_1= 0.0
NDSepPos_2= 0.0
NDSepPos_3= 0.0
NDRestY= 0.0
NDFinZer= 2
NDSepNum= 11
NDTypFrac= 2
NDSepDen= 2
NDOperY= 0.5
NDNulOther= 2
NDResScl= 1.0
NDFact= 1
NDRestX= 0.5


NDName= NUM.DIMP
NDType= 1
NDUnit= 2
NDGlobFact= 1.0
NDNulFac_1= 1
NDNulFac_2= 2
NDExise= 1
NDSep1000= 0
NDFact_1= 1.0
NDFact_2= 12.0
NDFact_3= 0.0
NDValPos_1= 0.0
NDValPos_2= 0.0
NDValPos_3= 0.0
NDSepar_1= "
NDSepar_2= '
NDSepar_3=
NDSepScl_1= 1.0
NDSepScl_2= 1.0
NDSepScl_3= 0.0
NDSepPos_1= 0.2
NDSepPos_2= 0.2
NDSepPos_3= 0.0
NDRestY= 0.0
NDFinZer= 1
NDSepNum= 11
NDTypFrac= 2
NDSepDen= 2
NDOperY= 0.5
NDNulOther= 2
```

```
NDResScl= 1.0
NDFact= 2
NDRestX= 0.5


NDName= ANS.DIMM
NDType= 1
NDUnit= 1
NDGlobFact= 1.0
NDNulFac_1= 1
NDNulFac_2= 2
NDExise= 1
NDSep1000= 0
NDFact_1= 1.0
NDFact_2= 0.0
NDFact_3= 0.0
NDValPos_1= 0.0
NDValPos_2= 0.0
NDValPos_3= 0.0
NDSepar_1=
NDSepar_2=
NDSepar_3=
NDSepScl_1= 1.0
NDSepScl_2= 0.0
NDSepScl_3= 0.0
NDSepPos_1= 0.0
NDSepPos_2= 0.0
NDSepPos_3= 0.0
NDRestY= 0.0
NDFinZer= 2
NDSepNum= 11
NDTypFrac= 2
NDSepDen= 2
NDOperY= 0.5
NDNulOther= 2
NDResScl= 1.0
NDFact= 1
NDRestX= 0.5


NDName= DISTMM
NDType= 1
NDUnit= 1
NDGlobFact= 1.0
NDNulFac_1= 2
NDNulFac_2= 2
NDExise= 1
NDSep1000= 0
NDFact_1= 1.0
NDFact_2= 0.0
NDFact_3= 0.0
NDValPos_1= 0.0
NDValPos_2= 0.0
NDValPos_3= 0.0
NDSepar_1=
NDSepar_2=
NDSepar_3=
```

```
NDSepScl_1= 1.0
NDSepScl_2= 0.0
NDSepScl_3= 0.0
NDSepPos_1= 0.0
NDSepPos_2= 0.0
NDSepPos_3= 0.0
NDRestY= 0.0
NDFinZer= 1
NDSepNum= 11
NDTypFrac= 2
NDSepDen= 2
NDOperY= 0.5
NDNulOther= 1
NDResScl= 1.0
NDFact= 1
NDRestX= 0.5


NDName= DISTINC
NDType= 1
NDUnit= 2
NDGlobFact= 1.0
NDNulFac_1= 1
NDNulFac_2= 2
NDExise= 1
NDSep1000= 0
NDFact_1= 1.0
NDFact_2= 0.0
NDFact_3= 0.0
NDValPos_1= 0.0
NDValPos_2= 0.0
NDValPos_3= 0.0
NDSepar_1= "
NDSepar_2=
NDSepar_3=
NDSepScl_1= 1.0
NDSepScl_2= 0.0
NDSepScl_3= 0.0
NDSepPos_1= 0.0
NDSepPos_2= 0.0
NDSepPos_3= 0.0
NDRestY= 0.0
NDFinZer= 1
NDSepNum= 11
NDTypFrac= 2
NDSepDen= 2
NDOperY= 0.5
NDNulOther= 1
NDResScl= 1.0
NDFact= 1
NDRestX= 0.5


NDName= FEET-INC
NDType= 1
NDUnit= 2
NDGlobFact= 1.0
```

```
NDNulFac_1= 1
NDNulFac_2= 1
NDExise= 1
NDSep1000= 0
NDFact_1= 1.0
NDFact_2= 12.0
NDFact_3= 0.0
NDValPos_1= 0.0
NDValPos_2= 0.0
NDValPos_3= 0.0
NDSepar_1= "
NDSepar_2= '
NDSepar_3=
NDSepScl_1= 1.0
NDSepScl_2= 1.0
NDSepScl_3= 0.0
NDSepPos_1= 0.0
NDSepPos_2= 0.0
NDSepPos_3= 0.0
NDRestY= 0.0
NDFinZer= 1
NDSepNum= 11
NDTypFrac= 1
NDSepDen= 2
NDOperY= 0.5
NDNulOther= 1
NDResScl= 1.0
NDFact= 2
NDRestX= 0.5


NDName= NUM.ADMS
NDType= 2
NDUnit= 4
NDGlobFact= 1.0
NDNulFac_1= 2
NDNulFac_2= 2
NDExise= 1
NDSep1000= 0
NDFact_1= 1/3600
NDFact_2= 1/60
NDFact_3= 1.0
NDValPos_1= 0.0
NDValPos_2= 0.0
NDValPos_3= 0.0
NDSepar_1= "
NDSepar_2= '
NDSepar_3= deg
NDSepScl_1= 1.0
NDSepScl_2= 1.0
NDSepScl_3= 1.0
NDSepPos_1= 0.2
NDSepPos_2= 0.2
NDSepPos_3= 0.2
NDRestY= 0.0
NDFinZer= 1
NDSepNum= 11
```

```
NDTypFrac= 2
NDSepDen= 2
NDOperY= 0.5
NDNulOther= 1
NDResScl= 1.0
NDFact= 3
NDRestX= 0.5


NDName= NUM,ADMS
NDType= 2
NDUnit= 4
NDGlobFact= 1.0
NDNulFac_1= 1
NDNulFac_2= 2
NDExise= 1
NDSep1000= 0
NDFact_1= 1/3600
NDFact_2= 1/60
NDFact_3= 1.0
NDValPos_1= 0.0
NDValPos_2= 0.0
NDValPos_3= 0.0
NDSepar_1= "
NDSepar_2= '
NDSepar_3= deg
NDSepScl_1= 1.0
NDSepScl_2= 1.0
NDSepScl_3= 1.0
NDSepPos_1= 0.2
NDSepPos_2= 0.2
NDSepPos_3= 0.2
NDRestY= 0.0
NDFinZer= 1
NDSepNum= 6
NDTypFrac= 2
NDSepDen= 2
NDOperY= 0.5
NDNulOther= 1
NDResScl= 1.0
NDFact= 3
NDRestX= 0.5


NDName= INC.ADMS
NDType= 2
NDUnit= 4
NDGlobFact= 1.0
NDNulFac_1= 1
NDNulFac_2= 1
NDExise= 1
NDSep1000= 0
NDFact_1= 1/3600
NDFact_2= 1/60
NDFact_3= 1.0
NDValPos_1= 0.0
NDValPos_2= 0.0
```

```
NDValPos_3= 0.0
NDSepar_1= "
NDSepar_2= '
NDSepar_3= deg
NDSepScl_1= 1.0
NDSepScl_2= 1.0
NDSepScl_3= 1.0
NDSepPos_1= 0.2
NDSepPos_2= 0.2
NDSepPos_3= 0.2
NDRestY= 0.0
NDFinZer= 2
NDSepNum= 11
NDTypFrac= 2
NDSepDen= 2
NDOperY= 0.5
NDNulOther= 2
NDResScl= 1.0
NDFact= 3
NDRestX= 0.5


NDName= NUM.ARAD
NDType= 2
NDUnit= 3
NDGlobFact= 1.0
NDNulFac_1= 2
NDNulFac_2= 2
NDExise= 1
NDSep1000= 0
NDFact_1= 1.0
NDFact_2= 0.0
NDFact_3= 0.0
NDValPos_1= 0.0
NDValPos_2= 0.0
NDValPos_3= 0.0
NDSepar_1=
NDSepar_2=
NDSepar_3=
NDSepScl_1= 1.0
NDSepScl_2= 0.0
NDSepScl_3= 0.0
NDSepPos_1= 0.0
NDSepPos_2= 0.0
NDSepPos_3= 0.0
NDRestY= 0.0
NDFinZer= 1
NDSepNum= 11
NDTypFrac= 2
NDSepDen= 2
NDOperY= 0.5
NDNulOther= 1
NDResScl= 1.0
NDFact= 1
NDRestX= 0.5
```

```
NDName= ANGLEDEC
NDType= 2
NDUnit= 4
NDGlobFact= 1.0
NDNulFac_1= 2
NDNulFac_2= 2
NDExise= 1
NDSep1000= 0
NDFact_1= 1.0
NDFact_2= 0.0
NDFact_3= 0.0
NDValPos_1= 0.0
NDValPos_2= 0.0
NDValPos_3= 0.0
NDSepar_1= deg
NDSepar_2=
NDSepar_3=
NDSepScl_1= 1.0
NDSepScl_2= 0.0
NDSepScl_3= 0.0
NDSepPos_1= 0.0
NDSepPos_2= 0.0
NDSepPos_3= 0.0
NDRestY= 0.0
NDFinZer= 1
NDSepNum= 11
NDTypFrac= 2
NDSepDen= 2
NDOperY= 0.5
NDNulOther= 1
NDResScl= 1.0
NDFact= 1
NDRestX= 0.5


NDName= ANGLEDMS
NDType= 2
NDUnit= 4
NDGlobFact= 1.0
NDNulFac_1= 2
NDNulFac_2= 2
NDExise= 1
NDSep1000= 0
NDFact_1= 1/3600
NDFact_2= 1/60
NDFact_3= 1.0
NDValPos_1= 0.0
NDValPos_2= 0.0
NDValPos_3= 0.0
NDSepar_1= "
NDSepar_2= '
NDSepar_3= deg
NDSepScl_1= 1.0
NDSepScl_2= 1.0
NDSepScl_3= 1.0
NDSepPos_1= 0.0
```

```
NDSepPos_2= 0.0
NDSepPos_3= 0.0
NDRestY= 0.0
NDFinZer= 1
NDSepNum= 11
NDTypFrac= 2
NDSepDen= 2
NDOperY= 0.5
NDNulOther= 1
NDResScl= 1.0
NDFact= 3
NDRestX= 0.5
```

Pattern Definition

This section deals with pattern definition. Patterns are used for area fills or in a Generative Drafting context when cutting through material in section views/cuts or breakout views, for example.

Defining Patterns

Pattern definitions are located in the **Patterns** node of the Standards editor, available via **Tools -> Standards**.

There are a number of pre-defined pattern definitions available in the Standards editor. You can customize these definitions to suit your needs, by modifying one or several values of the parameters defining the pattern.

You can also add additional instances of pattern definitions. To create a new pattern definition, you must use the Standards editor. Select the **Patterns** type in the standards editor, and then click the **Add Instance** button to add a new pattern instance. This will create a sample pattern definition that you will then customize to suit your needs, by modifying one or several values of the parameters defining the pattern.

Once defined or customized, a pattern can be applied to area fills (either via **Edit -> Properties**, or using the Graphic Properties toolbar), or it can be used when cutting through material in generative section views/cuts or breakout views, for example.

General remarks about patterns

- If no pattern is defined in the standard XML file (i.e. if all instances of pattern definitions are removed from the standards editor), the software will automatically use its own selection of patterns. In this case, you will be able to edit all the properties of these patterns via **Edit -> Properties** or the Graphic Properties toolbar.
- You need to define at least one pattern for each type of pattern (hatching, dotting, coloring, image) if you want this specific pattern type to be available from **Edit -> Properties** or from the Graphic Properties toolbar.
- With hatching or dotting patterns, the spacing between each hatch or dot is sometimes larger than the area to fill. This makes it impossible to display the pattern properly. In such a case, the area fill contour is made bold and is turned into the same color as the pattern color. This enables you to identify items with area fills even if the pattern is not visible.
- If you modify the standard of a drawing which already includes patterns (in area fills, sections or breakouts, for example), existing patterns will not changed, even when updating the drawing. However, if you create new area fills or if you create section views/cuts or breakout views from new parts in this drawing, the newly-defined patterns will be used.

General remarks about generated patterns (Generative Drafting context)

- Hatching patterns are a combination of the angle and hatching lists available in the standard XML file: newly created patterns are based on the first hatching standard referencing successively the values of the angle list, then the second hatching standard referencing successively the values of the angle list, and so on.
- A new hatching pattern is created for each newly cut part or body. Patterns are associated to bodies (their attributes are kept each time the view is updated) but are they are not persistent. This means

that if a part becomes not cut after a view update, the associated pattern is definitively lost. If this part is cut again, a new pattern will be created.

- Some parts use a material to which a specific pattern is associated. In this case:
  - You can either choose to use the pattern associated to this material (instead of the patterns defined in the standards) even if this pattern is not defined in the standards, or you can choose to use standard patterns only, instead of the pattern associated to this material. Refer to View Generation Definition for more information.
  - The angle list available in the standard XML file is used to define the pattern. However, note the following exception to this rule: if the material pattern angle equals 0, this is considered as a strong material specification. In this case, the angle for the generated pattern is kept at 0.
  - If you modify the pattern associated with the part material, the generated pattern is not modified even if you update the view.
  - When editing the properties of a pattern associated with a part material, a specific button is available in the Properties dialog box: **Reset using Part Material Pattern**. When the original properties of the part material are modified, this button is activated to let you revert to the original part material pattern. For example, if you replace or modify a pattern associated with a part material by a pattern from the standard, then the generated pattern is considered as overloaded, and will not be updated if you modify the properties of the part material. In this case, you can always revert to the original part material pattern using the **Reset using Part Material Pattern** button.
    Also use this button if you modify the part material pattern and want to apply these modifications to the generated patterns.
    If you reset a pattern using this button and do not modify it using other properties available in the Properties dialog box (no overload), and if you then modify the part material pattern, then this pattern will be updated accordingly.

- When editing the properties of a pattern associated with a part material (via **Edit -> Properties** or the Graphic Properties toolbar), the software offers its own selection of patterns, and not the patterns defined in the standard XML file.

Specific remarks about Image patterns

- It is recommended that you place images that you want to use for the Image pattern in either one (or in both) of the directories set by the following variables:
  - CATStartupPath (images used by materials)
  - CATGraphicPath (images and icons).
  Make sure you do not remove the files which are present in these directories when placing your images there. For more information on the CATStartupPath and on the CATGraphicPath variables, refer to the Infrastructure Installation Guide.

- Make sure that images referenced in the standard XML file are present on the computer of each user who will use this standard.

- Once an Image pattern is used in a drawing, the corresponding file is embedded inside it. You do not need to export the image files used in a given drawing.

- Images used as patterns must be bitmap images (not vector).

Parameter	Parameter Name	Description
EditAvailability	EditAvailability	Specifies whether users can modify all or only some pattern properties via **Edit -> Properties**. If Yes, all pattern properties can be modified in the Properties dialog box. If No: only some pattern properties will be available from the Properties dialog box, namely: - Hatching: users can only modify the angle, the pitch and the offset, and choose a new pattern. - Dotting: users can modify all properties. - Coloring: users can modify all properties. - Image: users can only use the images defined in the standards. They can modify the image angle and scale.
AngleList	Angle	Specifies preferred angle values that will be used when creating section views/cuts or breakout views. These values will be available via **Edit -> Properties.**
hatching	Name	Specifies the name of this pattern.
	Type	Shows the current pattern type, in this case, hatching.
	Number of Hatching	Defines the number of different hatchings to use in this pattern. A tab will be created for each hatching, to let you define each one individually.
	Angle	For each hatching used in this pattern, specifies the angle value in degrees.
	Pitch	For each hatching used in this pattern, specifies the pitch in millimeters.
	Offset	For each hatching used in this pattern, specifies the offset in millimeters.
	Color	For each hatching used in this pattern, specifies the color.
	Linetype	For each hatching used in this pattern, specifies the linetype.
	Thickness	For each hatching used in this pattern, specifies the linetype thickness.
	Preview	Lets you preview the resulting hatching pattern.
dotting	Name	Specifies the name of this pattern.
	Type	Shows the current pattern type, in this case, dotting.
	Pitch	Specifies the dotting pitch in millimeters.
	Color	Specifies the dotting color.
	Zigzag	Specifies whether dotting should zigzag.
	Preview	Lets you preview the resulting dotting pattern.
coloring	Name	Specifies the name of this pattern.
	Type	Shows the current pattern type, in this case, coloring.
	Color	Specifies the color.
	Preview	Lets you preview the resulting coloring pattern.
image	Name	Specifies the name of this pattern.
	Type	Shows the current pattern type, in this case, image.
	Browse button	Lets you select the image to use for this pattern. Refer to Specific remarks about Image patterns for more information.
	Angle	Specifies the angle value in degrees.

	Scale	Specifies the scale.
	Preview	Lets you preview the original image (not the result after modifying the angle and scale).

Line Type Definition

This section deals with line type definition. Line types can be applied to, and drive the representation of, almost all elements in a drawing, such as lines, curves, dimension lines, etc. (Line types cannot be applied to fonts and points).

- In releases before V5 R11, line types used to be defined in **Tools -> Options -> General -> Display -> Line Types** for the Drafting workbench as well as for other workbenches. For Drafting, line types are now defined in standards. Therefore, line types in drawings do not depend on the options defined in **Tools -> Options**, but on what is defined in the standards.
- When opening a drawing created with releases before V5 R11 (i.e. a drawing which does not contain its own line type parameters), the line type options defined in **Tools -> Options** will be used. You can upgrade a CATDrawing document to this new standard format at any time, by performing the following operations in **File -> Page Setup**:
  - changing the standard to another standard (ISO -> ANSI for instance)
  - updating the current standard to the new format.

The information below is specifically intended for administrators customizing standards for the 2D Layout for 3D Design workbench:

- When displaying line types in the 2D Layout for 3D Design window: the line types which are used are those defined in standards.
  This is the case of all elements displayed in the 2D or 3D background of a view in the 2D Layout for 3D Design window, whether they are:
  - layout elements (geometry, annotations and so on),
  - 3D wireframe elements (lines, points and so on).
- When displaying line types in the 3D window (Part Design, for example): the line types which are used are those defined in the settings (through **Tools -> Options -> General -> Display -> Line Types**).
  This is the case of all layout elements (geometry, annotations and so on), when displayed in the 3D window.
- When printing a layout: the line types which are used are those defined in standards.
- When editing part layouts in the context of a product: the line types which are used are those defined in the standards of the current layout (even when visualizing elements which belong to another layout).
  For example, take the following scenario, where:
  - the layout of Part.1 uses ISO_3D
  - the layout of Part.2 uses JIS_3D
  - the line type definitions are different in ISO_3D and JIS_3D.

  In this case, when visualizing Part.2 in the background of the Part.1 layout, Part.2 will be displayed using the line type definitions of Part.1.

Therefore, if you want the visualization of elements to be homogeneous in the 2D and 3D windows, you need to make sure that line type definitions in the standards match line type definitions in the settings.

Defining Line Types

Line type definitions are located in the **Line Types** node of the Standards editor, available via **Tools -> Standards**. Line types can either be mono-dimensional, i.e. defined by a sequence of non-continuous segments, or bi-dimensional, i.e. defined by a polyline. Once defined, a line type can be applied to any element which supports it, either via **Edit -> Properties**, or using the Graphic Properties toolbar.

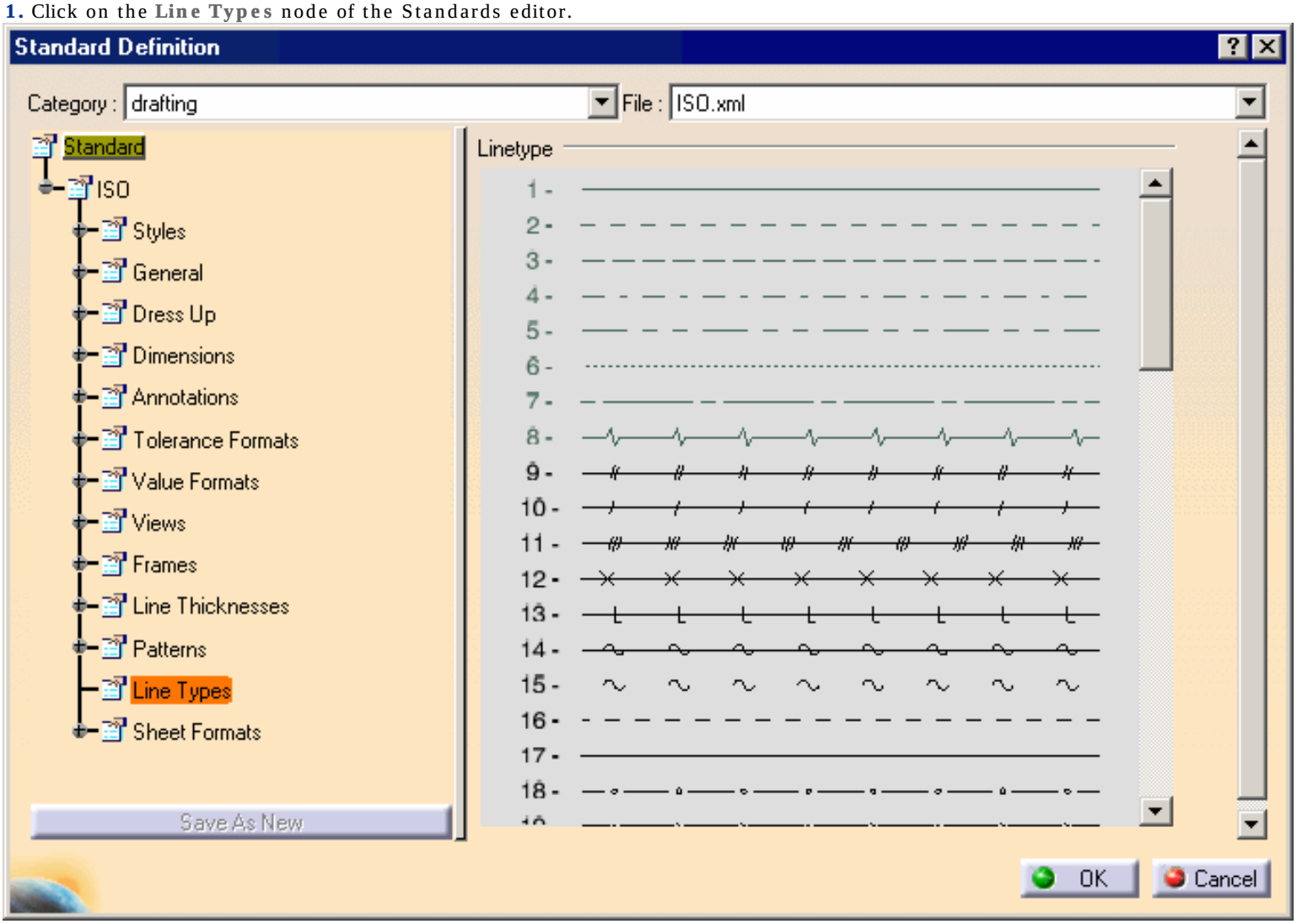
There are 63 line type definitions in the Standards editor. You cannot add additional instances of line type definitions. Out of these 63 definitions,

- line type definitions ranging from 1 to 8 are pre-defined with different parameters for each and cannot be customized.
- line type definitions ranging from 9 to 19 are pre-defined with different parameters for each and can be customized.
- line type definitions ranging from 20 to 63 are not pre-defined and can be customized.

Note that it is impossible to assign line types greater than line type 7 to spline geometry, and greater than line type 8 to circle geometry.

You can customize the definitions of line types ranging from 9 to 63. To do this, proceed as follows:

1. Click on the **Line Types** node of the Standards editor.



2. In the right-hand panel, double-click on the line type you want to define. The line type editor appears for you to set the line type properties. For more information on using the line type editor, refer to Line Type in the *Infrastructure User's Guide*.

3. For each line type definition, you can also specify whether or not a given line type should be available in the line types list for users to choose from. In the right-hand panel, double-click on the number of the line type you want to make unavailable. Perform the same operation to make an unavailable line type available.

Users will only be able to assign "available" line type definitions when creating or editing elements. However, existing element properties in drawings will not be affected: if an existing element is assigned a line type which is flagged as "unavailable" in the Standards editor, then this line type will be used for this element but it will not be available in the line types list, so that users cannot apply it to other elements.

Sheet Format Definition

This section deals with sheet format definition.

Defining Sheet Formats

Sheet format definitions are located in the **Sheet Formats** node of the Standards editor, available via **Tools -> Standards**, according to specified normalized standards such as ISO, ASME, etc.

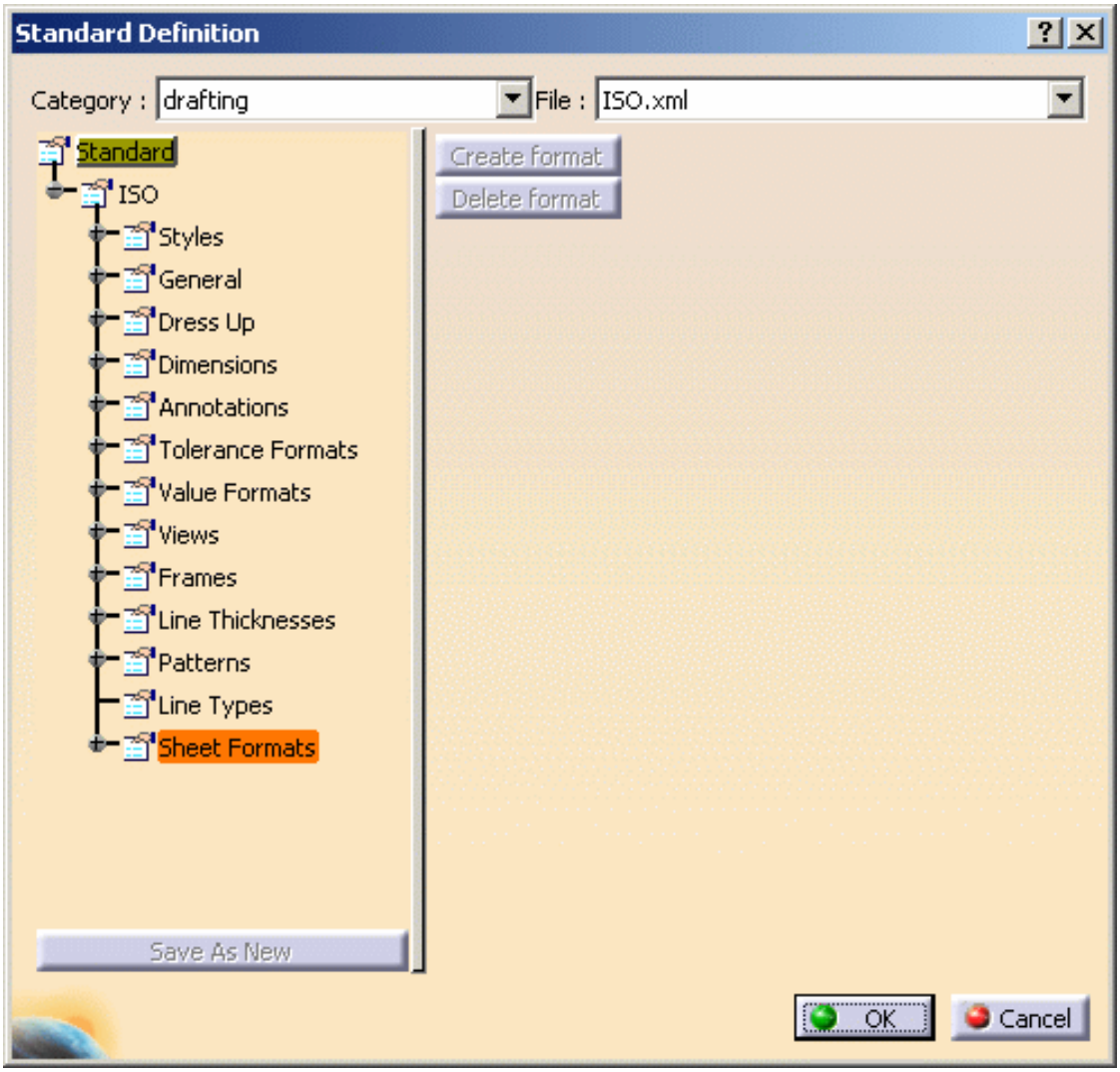
The list of available sheet formats can be extended, reduced or modified by the administrator.

For a specific sheet format, you can:

- Change its name.
- Modify the orientation width.
- Modify the orientation height.
- Modify the orientation type:
  - PortraitByDefault: Portrait will be the default orientation, but Landscape will be available.
  - LandscapeByDefault: Landscape will be the default orientation, but Portrait will be available.
  - PortraitOnly: Portrait will be the only available orientation. Landscape will not be available.
  - LandscapeOnly: Landscape will be the only available orientation. Portrait will not be available.

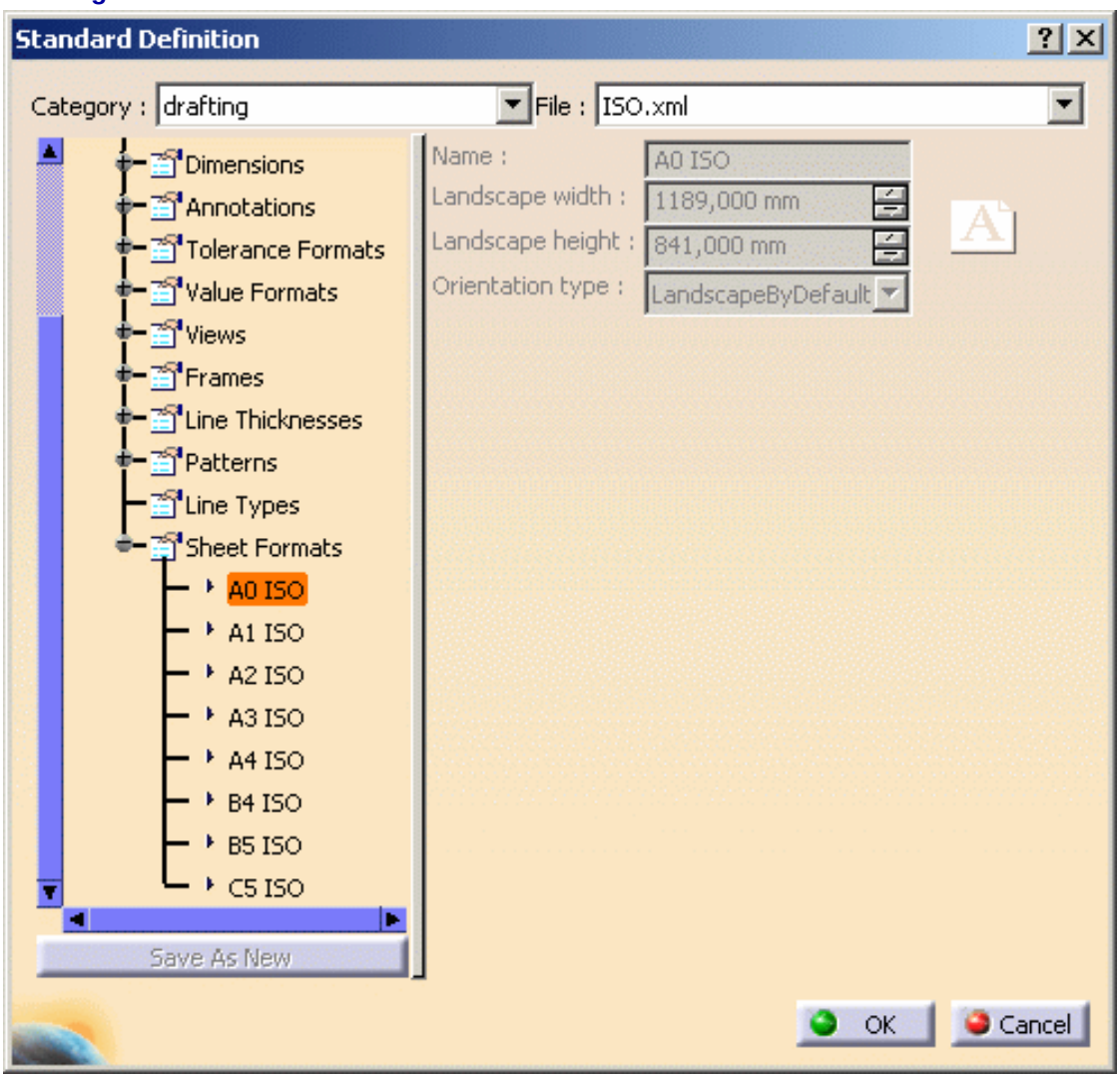
A sheet format is referenced by a sheet style. For more information, refer to Sheet Styles.

1. Click on the **Sheet Formats** node of the Standards editor. You can create or delete a sheet format from this node only.



Note that sheet formats are not listed in alphabetical order. However, in the **New Drawing** or **Page Setup** dialog box, they will be listed alphabetically.

2. For instance, click on the **A0 ISO** node and modify the desired parameters as needed.


Standard Definition
Category : drafting
File : ISO.xml
Dimensions
Annotations
Tolerance Formats
Value Formats
Views
Frames
Line Thicknesses
Patterns
Line Types
Sheet Formats
A0 ISO
A1 ISO
A2 ISO
A3 ISO
A4 ISO
B4 ISO
B5 ISO
C5 ISO
Save As New
Name : A0 ISO
Landscape width : 1189,000 mm
Landscape height : 841,000 mm
Orientation type : LandscapeByDefault
OK
Cancel

Layout Views Customization

This section deals with the customization of layout views. It is specifically intended for administrators of the 2D Layout for 3D Design workbench.

Defining Layout Views

Defining layout views is an important step in the layout design process. Designers need to have a general idea of the overall dimensions of their design, as well as of its position in space.

When customizing layout views, administrators actually define the 2D Layout for 3D Design view box, which gathers all the data needed to fully define the layout of a view set in the 2D window, as well as the position of each view in the 3D space.

This data is made up of:

- Primary view type
- 3D axis
- View box anchor point
- 3D axis origin
- View box overall dimensions

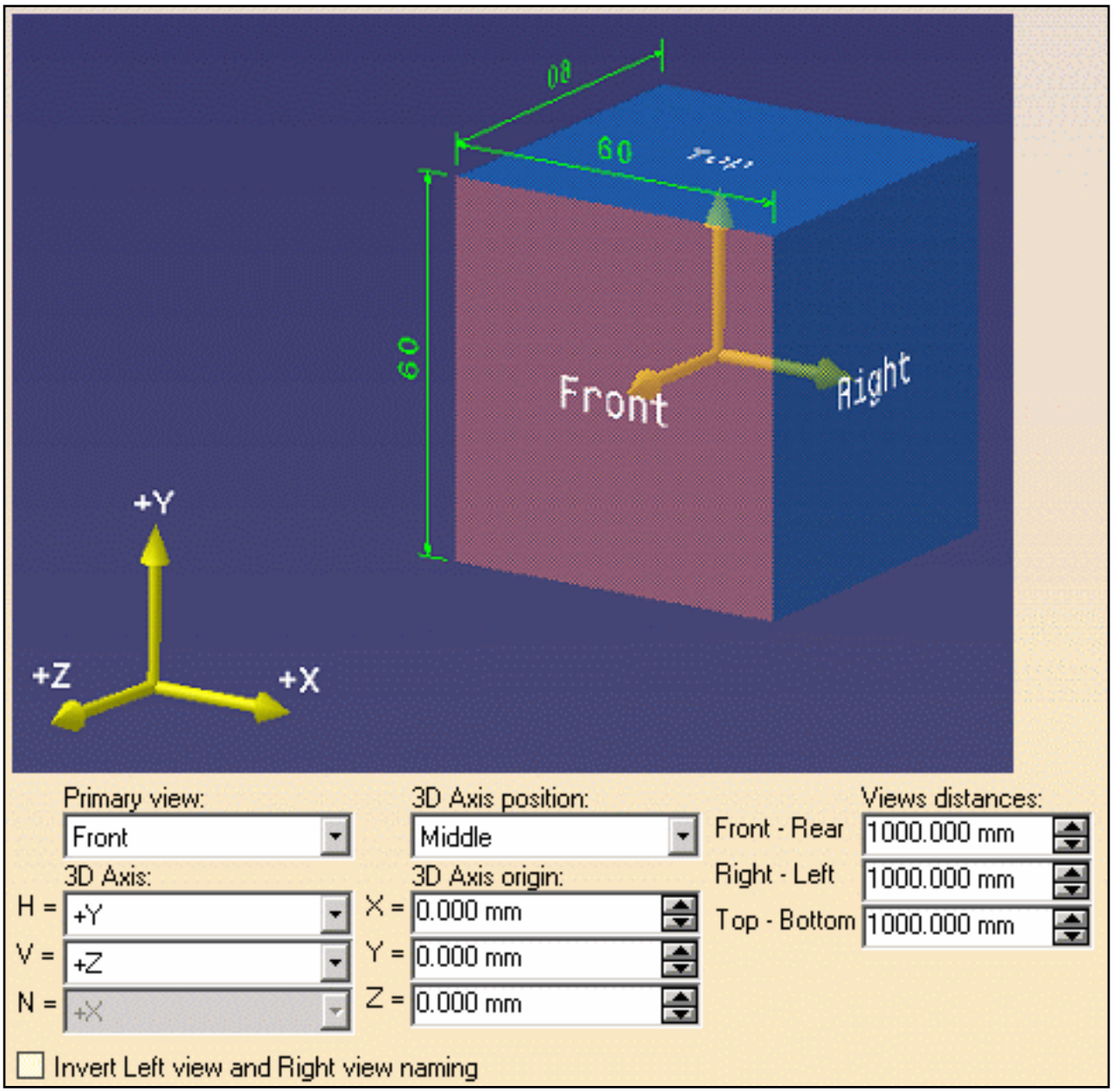
A single, default, view box is defined for each standard (for example, there is a single view box for the ISO_3D standard). The standard definition is retrieved when creating the first view of a view set: the view box definition is associated to each view set. Therefore, an update of the standard does not impact existing views.

Administrators will probably want to customize layout views for the ASME_3D, ISO_3D and JIS_3D standards, which have been specifically customized for the 2D Layout for 3D Design workbench (the colors, for example, have been customized for optimized display).

On the other hand, it does not make much sense to customize layout views for the ISO, JIS, ANSI and ASME standards, which are purely 2D-based, Drafting standards.

The view box is defined in the **Layout Views Customization** node of the Standards Editor, available via **Tools -> Standards**.

1. Click on the **Layout Views Customization** node of the Standards editor. The editor is displayed.



2. Define the primary view type (Front, Right, Rear, Left, Top or Bottom) from the **Primary view** drop-down list.

3. Define the 3D axis from the **3D Axis**: **H** (horizontal), **V** (vertical) and **N** (normal) drop-down lists: this specifies which plane the primary view lies on and which orientation has the outgoing normal.

   For example, defining a front view on the xy plane and +z normal means that the rear view is located below the front view, along the z axis. On the opposite, defining a front view on the xy

plane and -z normal means that the rear view is above the front view, along the z axis.

4. Define the view box anchor point from the **3D Axis position** drop-down list.

5. Define the 3D axis origin from the **3D Axis origin**: **X**, **Y** and **Z** drop-down lists.

6. Define the view box overall dimensions from the **Views distances**: **Front-Rear** (distance from the front to the rear of the box), **Right-Left** (distance from the right hand-side to the left hand-side of the box) and **Top-Bottom** (distance from the top to the bottom of the box) drop-down lists.

7. Select **Invert Left view and Right view naming** if you want the Right view to be called Left view, and vice-versa. This only inverts the name (not the type) of the views.

Setting Standard Styles

The Interactive Drafting workbench lets administrators set standard styles that will be used as default values when creating new elements.

About styles: Learn more about style management.

Sheet styles: Define the styles that will be used by default when creating sheets.

Geometry styles: Define the styles that will be used by default when creating geometry.

Annotation styles: Define the styles that will be used by default when creating annotations.

Dimension styles: Define the styles that will be used by default when creating dimensions.

Dimension System Styles: Define the styles that will be used by default when creating dimension systems.

Dress-up and dress-up symbols styles: Define the styles that will be used by default when using dress-up elements or dress-up symbols.

View callout styles: Define the styles that will be used by default when using callouts.

About Styles

Setting default values for elements using styles

Styles enable administrators to set the default values that will be applied to all properties of such elements as sheets, geometry, annotations, dimensions, dress-up and dress-up symbols, callouts, etc.

The default values are defined and stored in the standard XML file, where a set of new parameters are defined, one parameter for each element property whose default value can be set.

Default values are applied to elements as they are created. After creation, the user can modify element values as required.

If you modify styles in the standard itself and then update the standard file used by the drawing, the elements which have already been created will NOT be modified (i.e. their default values will remain as previously). Updating the standard will only have an impact on the next elements to be created.

Styles replace the former management of default values (which was performed using the Set as Default / Use Default functionalities), for drawings:

- created with version V5 R11 and later
- created with versions up to V5 R10, whose standard has been updated in V5 R11

For drawings created with versions up to V5 R10 and NOT updated, default values still use the Set as Default / Use Default functionalities. For more information, refer to Setting Properties As Default and Using Properties Set as Default.

Defining default values

The default values for all element properties are stored in a specific Styles section of the standard XML file, and are defined by the administrator.

Styles are defined for all Drafting element types. By default, one style named Default is predefined in the standard files for each type of element. In this Default style, all element properties are pre-defined, enabling the administrator to set the value for this property. This Default style cannot be renamed.

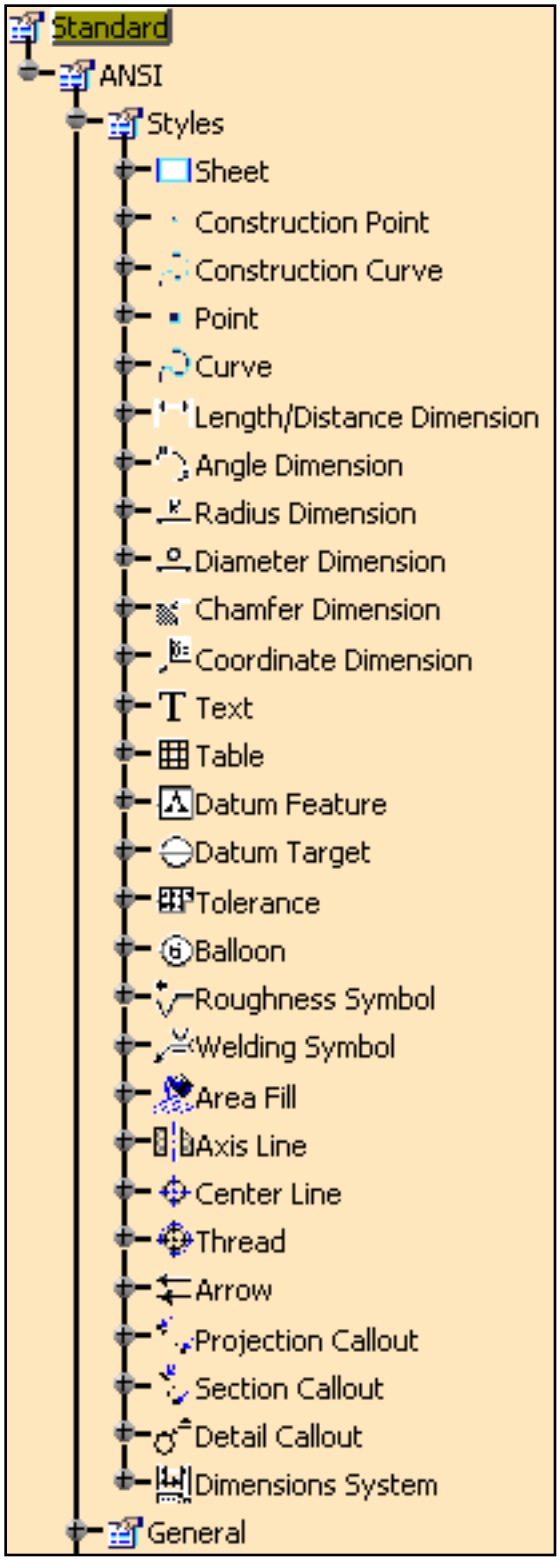
You can create your own styles (based on existing styles), as well as delete styles (providing there remains at least one style for each type of element).

Using default values

When creating a Drafting element, default values are automatically used. So, when users select a command that creates a specific type of element, the Style toolbar displays the current style for this type:

- If only one style is defined for this type of element, then this style is displayed in the toolbar.
- If several styles are defined for this type of element, then a style is defined as the current style and is displayed in the toolbar. Users can use the toolbar to select another style of the same type before creating the element.

The toolbar reflects the value of the style, but users can always modify the value of specific elements.



Re-applying a style to an object

When a Drafting element is selected, the Style toolbar displays the list of the styles that can be applied to it. If the user selects one of these, this style is re-applied to the element. This enables users to reset to its default values an element whose properties have been modified.

Customizing Styles

In this scenario, administrators will learn how to customize styles.

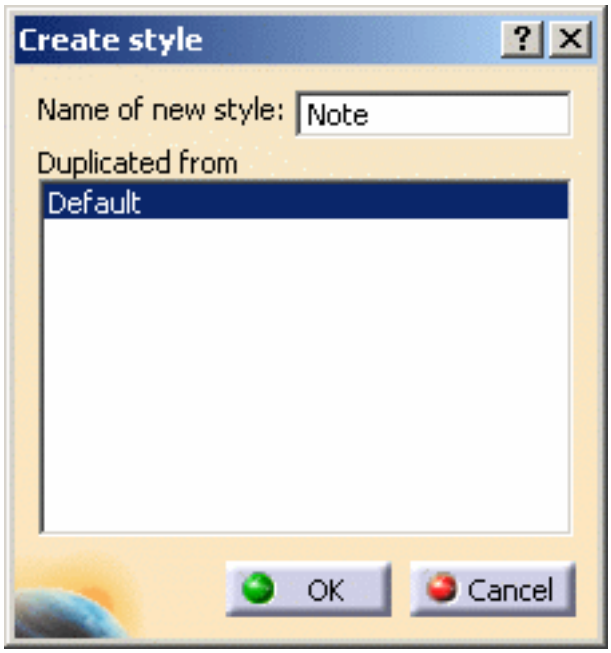
This scenario provides an example of style customization. The procedure differs when customizing standard parameters (dimensions, annotations, dress-up elements, etc.). For more information, refer to About Standard Parameters.

Note that a new style is always based on an existing style.

You want to create a new text style that you will use for adding notes. You want to use the Verdana font, and you want a frame around the text. You then want to delete the Default style.

Select **Tools -> Standards** to launch the standards editor. Choose the **Drafting** category, and then open the ISO.xml file from the drop-down list.

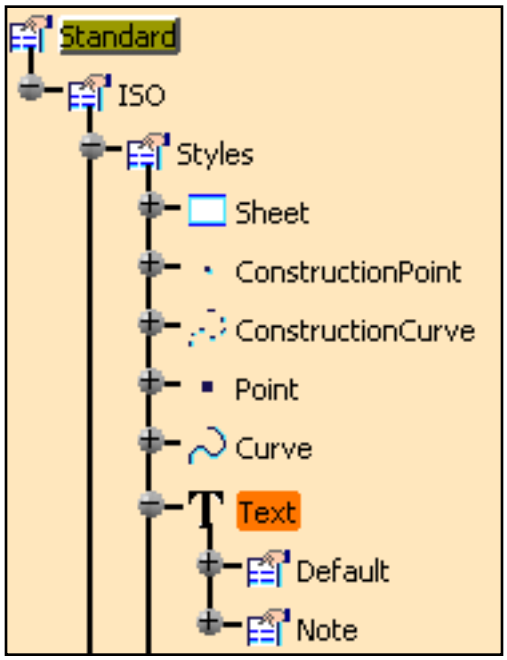
1. Expand the **Styles** node in the editor.
2. Select the **Text** node.
3. Click on the **Create style** button in the right-hand pane. The Create style dialog box is displayed.
4. Type the name of the new style in the appropriate field.



The **Duplicated from** list is used when several styles exist for a given type of element to specify which existing style the new style should be based on. In our example, only the Default style exists. Therefore, the new style will be created based on this Default style.

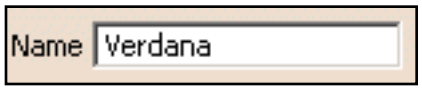
You cannot create styles containing characters such as < > . / : ; " ' \ | as well as spaces at the beginning and/or at the end of the style's name.

5. Click **OK**. A new style called Note is added under the Text node in the editor.



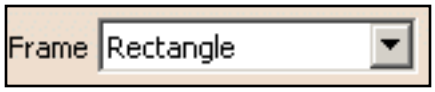
6. Expand the **Note -> Font** node in the editor, and then select the **Name** node.

7. Type Verdana in the **Name** field in the right-hand pane.



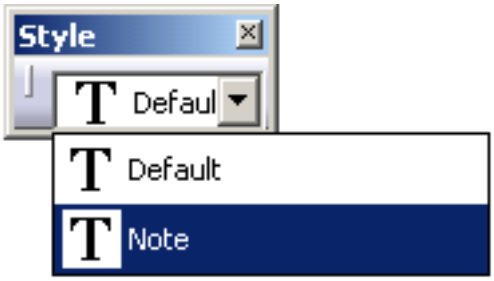
8. Expand the **Text** node in the editor, and then select the **Frame** node.

9. Choose Rectangle from the **Frame** drop-down list in the right-hand pane.



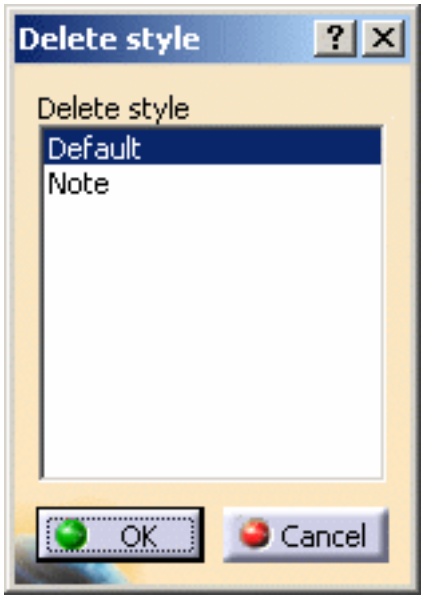
10. Click **OK** to save your modifications and exit the standards editor.

11. Now, start creating a new text in a sheet. In the Style toolbar, you can notice that two styles are now available: Default and Note.

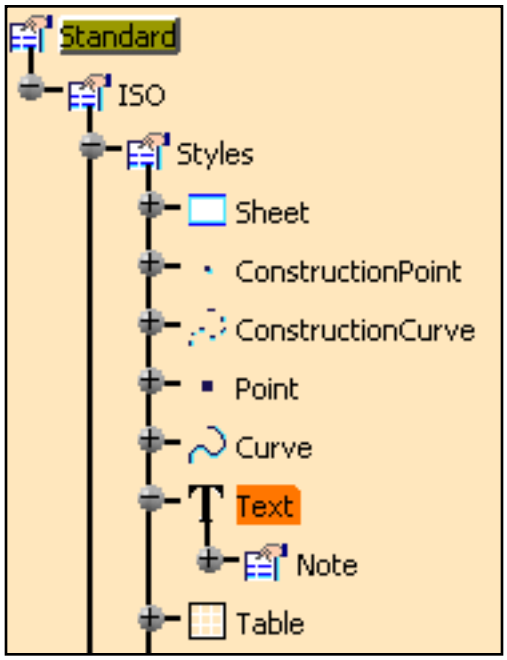


11. Choose the Note style, click on the sheet to indicate where you want to position the note, type your note in the text editor and then click **OK**. The note is creating using the values you specified.

12. You will now delete the Default style. To do this, launch the standards editor again.
13. Expand the **Styles** node and then select the **Text** node.
14. Click on the **Delete style** button in the right-hand pane. The Delete style dialog box is displayed.



15. Select Default as the style that you want to delete, and click **OK**. The Default style is deleted from the Text node in the editor.



Geometry Styles

This section deals with geometry styles. These let you define the default values that will be used when creating geometry.

Defining Geometry Styles

Geometry styles are located in the following nodes of the Standards editor, available via **Tools -> Standards:**

- Styles -> ConstructionPoint
- Styles -> ConstructionCurve
- Styles -> Point
- Styles -> Curve

By default, a style called Default is available for each geometry style.

All the parameters associated to a given geometry style are listed in a dedicated table. The Description column provides a description of each parameter.

All parameters are taken into account both at creation time (i.e. when creating a geometrical element), and at modification time (i.e. when reapplying a style to a geometrical element).

ConstructionPoint Style

Parameter Name	Description
Color	Specifies the color that should be used to represent construction points.
PointType	Specifies the type (e.g., cross, dot, etc.) that should be used to represent construction points.

ConstructionCurve Style

Parameter Name	Description
Color	Specifies the color that should be used to represent construction curves.
LineType	Specifies the number of the linetype (as defined in the LineTypes node of the current standard) that should be used to represent construction curves.
Thickness	Specifies the line thickness index (as defined in the LineThickness node of the current standard) that should be used to represent construction curves.
ControlPoints > Color	Specifies the color that should be used to represent control points in construction curves.
ControlPoints > PointType	Specifies the type (e.g., cross, dot, etc.) that should be used to represent control points in construction curves.

Point Style

Parameter Name	Description
Color	Specifies the color that should be used to represent points.
PointType	Specifies the type (e.g., cross, dot, etc.) that should be used to represent points.

Curve Style

Parameter Name	Description
Color	Specifies the color that should be used to represent curves.
LineType	Specifies the number of the linetype (as defined in the LineTypes node of the current standard) that should be used to represent curves.
Thickness	Specifies the line thickness index (as defined in the LineThickness node of the current standard) that should be used to represent curves.
ControlPoints > Color	Specifies the color that should be used to represent control points in curves.
ControlPoints > PointType	Specifies the type (e.g., cross, dot, etc.) that should be used to represent control points in curves.

Annotation Styles

This section deals with annotation styles. These let you define the default values that will be used when creating annotations.

Defining Annotation Styles

Annotation styles are located in the following nodes of the Standards editor, available via **Tools -> Standards:**

- Styles -> Text
- Styles -> Table
- Styles -> DatumFeature
- Styles -> DatumTarget
- Styles -> Tolerance
- Styles -> Balloon
- Styles -> RoughnessSymbol
- Styles -> WeldingSymbol

By default, a style called Default is available for each geometry style.

All the parameters associated to a given annotation style are listed in a dedicated table.
The Description column provides a description of each parameter.
Certain parameters are only taken into account at creation time (i.e. when creating the annotation), and not at modification time (i.e. when reapplying a style to an annotation): the Applies at modification column indicates whether this parameter is taken into account at modification time.

Text Styles

Parameter Name	Description	Applies at modification
Font > Name	Specifies the name of the font that should be used for texts. If no font name is specified, the system's default font will be used. Make sure that the text font specified here belongs to the list of allowed text fonts, as defined in the **General > AllowedTextFonts** node of the Standards editor.	Yes
Font > Bold	Indicates whether or not texts should be displayed in bold.	Yes

Font > Italic	Indicates whether or not texts should be displayed in italic.	Yes
Font > Size	Indicates the font size that should be used for texts.	Yes
Font > Underline	Indicates whether or not texts should be underlined.	Yes
Font > Color	Specifies the color that should be used to display texts.	Yes
Font > Ratio	Specifies the ratio that should be used to display texts.	Yes
Font > Slant	Specifies the slant that should be used to display texts.	Yes
Font > Spacing	Specifies the spacing that should be used to display texts.	Yes
Font > Pitch	Specifies the pitch (fixed or variable) that should be used to display texts.	Yes
Font > Strikethrough	Indicates whether or not strikethrough should be used for texts.	Yes
Font > Overline	Indicates whether or not texts should be overlined.	Yes
Text > Frame	Specifies the type of frame that should be used to represent texts. Note that fixed-size frames are defined in the Frame node of the current standard. Accepted value: any	Yes
Text > AnchorPoint	Specifies the text position in relation to the anchor point (e.g., top left, middle left, etc.).	Yes
Text > AnchorLine	Specifies the text position in relation to the anchor line (e.g., top or bottom, cap or base, etc.).	Yes
Text > LineSpacing	Specifies the spacing that should be used between two lines of text.	Yes
Text > LineSpacingMode	Specifies the spacing mode between two lines of text (e.g., bottom to top, base to cap, etc.).	Yes
Text > Justification	Specifies a justification for the text.	Yes
Text > WordWrap	Specifies a width to wrap the text.	Yes
Text > OrientationReference	Specifies whether the sheet, or whether the view or 2D component should be used as the reference for the text orientation.	Yes
Text > Angle	Specifies the text orientation angle according to the chosen reference.	Yes

Text > Mirroring	Specifies whether a symmetry, and which one, should be applied to the text.	Yes
Text > Backfield	Specifies whether or not superscript and subscript texts should be aligned above one another.	Yes
Text > SuperscriptOffset	Specifies the offset value (as a percentage of the font height) for superscript texts.	Yes
Text > SuperscriptSize	Specifies the size (as a percentage of the font height) of superscript texts.	Yes
Text > SubscriptOffset	Specifies the offset value (as a percentage of the font height) for subscript texts.	Yes
Text > SubscriptSize	Specifies the size (as a percentage of the font height) of subscript texts.	Yes
Text > DisplayUnit	Specifies whether or not texts should be displayed.	Yes
Text > ApplyScale	Specifies whether or not the scale of the view or of the 2D reference component scale should be applied to the display of the text.	Yes
Text > Blanking	Indicates whether or not blanking should be used.	Yes
Graphic > Thickness	Specifies the line thickness index (as defined in the LineThickness node of the current standard) that should be used to represent text frames and leaders.	Yes
Graphic > Linetype	Specifies the number of the linetype (as defined in the LineTypes node of the current standard) that should be used to represent text frames and leaders.	Yes
Graphic > Color	Specifies the color that should be used to represent text frames and leaders.	Yes
Leader > Symbol	Specifies the symbol (e.g., simple arrow, circle, etc.) that should be used for text leaders. If you choose the Automatic option, a default symbol will be used, depending on the standard type: • Filled arrow for ANSI / ASME • Simple arrow (a.k.a. Open arrow) for ISO / JIS	Yes
Leader > Delta	Specifies the length of the first leader segment (i.e. the segment which is located before the first leader breakpoint).	Yes

Leader > AnchorPoint	Specifies the leader position in relation to the anchor point of the text frame. When the Leader > StandardBehavior parameter is set to Yes: - 0 positions the leader automatically on the closest anchor point. - 1 positions the leader on the left-hand anchor point. - 2 positions the leader on the right-hand anchor point. When the Leader > StandardBehavior parameter is set to No: - 0 positions the leader automatically on the closest anchor point. - 1 to 8 position the leader on a specific anchor point. Open the TextLeaderAnchorPoints.CATDrawing document to know the value you should assign to the Leader > AnchorPoint parameter, depending on where you want to position the leader in relation to the anchor point, and on the type of frame used.	Yes
Leader > StandardBehavior	Specifies whether or not the position of text leaders can be different than left or right.	Yes

Table Styles

Parameter Name	Description	Applies at modification
Cells > Font > Name	Specifies the name of the font that should be used for text in table cells. If no font name is specified, the system's default font will be used. Make sure that the text font specified here belongs to the list of allowed text fonts, as defined in the **General > AllowedTextFonts** node of the Standards editor.	Yes
Cells > Font > Bold	Indicates whether or not texts in table cells should be displayed in bold.	Yes
Cells > Font > Italic	Indicates whether or not texts in table cells should be displayed in italic.	Yes
Cells > Font > Size	Indicates the font size that should be used for texts in table cells.	Yes

Cells > Font > Underline	Indicates whether or not texts in table cells should be underlined.	Yes
Cells > Font > Color	Specifies the color that should be used to display texts in table cells.	Yes
Cells > Font > Ratio	Specifies the ratio that should be used to display texts in table cells.	Yes
Cells > Font > Slant	Specifies the slant that should be used to display texts in table cells.	Yes
Cells > Font > Spacing	Specifies the spacing that should be used to display texts in table cells.	Yes
Cells > Font > Pitch	Specifies the pitch (fixed or variable) that should be used to display texts in table cells.	Yes
Cells > Font > Strikethrough	Indicates whether or not strikethrough should be used for texts in table cells.	Yes
Cells > Font > Overline	Indicates whether or not texts in table cells should be overlined.	Yes
Cells > LineSpacing	Specifies the spacing that should be used between two lines of text in table cells.	Yes
Cells > LineSpacingMode	Specifies the spacing mode between two lines of text in table cells (e.g., bottom to top, base to cap, etc.).	Yes
Cells > Backfield	Specifies whether or not superscript and subscript texts in table cells should be aligned above one another.	Yes
Cells > SuperscriptOffset	Specifies the offset value (as a percentage of the font height) for superscript texts in table cells .	Yes
Cells > SuperscriptSize	Specifies the size of superscript texts (as a percentage of the font height) in table cells .	Yes
Cells > SubscriptOffset	Specifies the offset value for subscript texts (as a percentage of the font height) in table cells .	Yes
Cells > SubscriptSize	Specifies the size of subscript texts (as a percentage of the font height) in table cells.	Yes
Cells > DisplayUnit	Specifies whether or not texts in table cells should be displayed.	Yes
Cells > HorizontalAlignment	Specifies the horizontal alignment for the contents of table cells.	Yes
Cells > VerticalAlignment	Specifies the vertical alignment for the contents of table cells.	Yes

Cells > HorizontalMargin	Specifies the horizontal space between the contents and the inside border of a cell.	Yes
Cells > VerticalMargin	Specifies the vertical space between the contents and the inside border of a cell.	Yes
AnchorPoint	Specifies the position of the table in relation to the anchor point (e.g., top left, middle left, etc.).	Yes
OrientationReference	Specifies whether the sheet, or whether the view or 2D component should be used as the reference for the orientation of the table.	Yes
Angle	Specifies the orientation angle of the table according to the chosen reference.	Yes
Graphic > Thickness	Specifies the line thickness index (as defined in the LineThickness node of the current standard) that should be used to represent table frames and leaders.	Yes
Graphic > Linetype	Specifies the number of the linetype (as defined in the LineTypes node of the current standard) that should be used to represent table frames and leaders.	Yes
Graphic > Color	Specifies the color that should be used to represent table frames and leaders.	Yes
Leader > Symbol	Specifies the symbol (e.g., simple arrow, circle, etc.) that should be used for table leaders. If you choose the Automatic option, a default symbol will be used, depending on the standard type: • Filled arrow for ANSI / ASME • Open arrow for ISO / JIS	Yes
Leader > Delta	Specifies the value of the delta that should be applied between a table and its leader.	Yes
Blanking	Indicates whether or not blanking should be used.	Yes

DatumFeature Styles

Parameter Name	Description	Applies at modification
OrientationReference	Specifies whether the sheet, or whether the view or 2D component should be used as the reference for the orientation of datum features.	Yes
Angle	Specifies the orientation angle of datum features according to the chosen reference.	Yes
AnchorPoint	Specifies the position of datum features in relation to the anchor point (e.g., top left, middle left, etc.).	Yes
Display	Choose the display mode you want for the datum feature: - Show value: displays the datum feature, its leader and its frame. - Show box: replaces the datum feature and its frame by a rectangular box and displays its leader. - Hide value: hides the datum feature and its frame but displays its leader.	Yes
Leader > Symbol	Specifies the symbol (e.g., simple arrow, circle, etc.) that should be used for datum feature leaders. If you choose the Automatic option, a default symbol will be used, depending on the standard type: • None (a.k.a. No symbol) for ANSI • Filled Triangle for ASME • Blanked triangle (a.k.a. Triangle) for ISO / JIS	Yes
Font > Name	Specifies the name of the font that should be used for datum feature texts. If no font name is specified, the system's default font will be used. Make sure that the text font specified here belongs to the list of allowed text fonts, as defined in the **General > AllowedTextFonts** node of the Standards editor.	Yes
Font > Bold	Indicates whether or not datum feature texts should be displayed in bold.	Yes
Font > Italic	Indicates whether or not datum feature texts should be displayed in italic.	Yes
Font > Size	Indicates the font size that should be used for datum feature texts.	Yes

Font > ApplyScale	Indicates whether or not datum feature texts should be scaled according to the view or the 2D reference component's scale.	Yes
Graphic > Thickness	Specifies the line thickness index (as defined in the LineThickness node of the current standard) that should be used to represent datum feature frames and leaders.	Yes
Graphic > Linetype	Specifies the number of the linetype (as defined in the LineTypes node of the current standard) that should be used to represent datum feature frames and leaders.	Yes
Graphic > Color	Specifies the color that should be used to represent datum feature frames, leaders and texts.	Yes

DatumTarget Styles

Parameter Name	Description	Applies at modification
Diameter	Indicates whether or not the surface is plane on a disk.	Yes
OrientationReference	Specifies whether the sheet, or whether the view or 2D component should be used as the reference for the orientation of datum targets.	Yes
Angle	Specifies the orientation angle of datum targets according to the chosen reference.	Yes
AnchorPoint	Specifies the position of datum targets in relation to the anchor point (e.g., top left, middle left, etc.).	Yes
Display	Choose the display mode you want for the datum target: - Show value: displays the datum target, its leader and its frame. - Show box: replaces the datum target and its frame by a rectangular box and displays its leader. - Hide value: hides the datum target and its frame but displays its leader.	Yes
Font > Name	Specifies the name of the font that should be used for datum target texts. If no font name is specified, the system's default font will be used. Make sure that the text font specified here belongs to the list of allowed text fonts, as defined in the **General > AllowedTextFonts** node of the Standards editor.	Yes

Font > Bold	Indicates whether or not datum target texts should be displayed in bold.	Yes
Font > Italic	Indicates whether or not datum target texts should be displayed in italic.	Yes
Font > Size	Indicates the font size that should be used for datum target texts.	Yes
Font > ApplyScale	Indicates whether or not datum target texts should be scaled according to the view or the 2D reference component's scale.	Yes
Graphic > Thickness	Specifies the line thickness index (as defined in the LineThickness node of the current standard) that should be used to represent datum target frames and leaders.	Yes
Graphic > Linetype	Specifies the number of the linetype (as defined in the LineTypes node of the current standard) that should be used to represent datum target frames and leaders.	Yes
Graphic > Color	Specifies the color that should be used to represent datum target frames, leaders and texts.	Yes
Leader > Symbol	Specifies the symbol (e.g., simple arrow, circle, etc.) that should be used for datum feature leaders. If you choose the Automatic option, a default symbol will be used, depending on the standard type: • None (a.k.a. No symbol) for ANSI • Closed arrow (a.k.a. Unfilled arrow) for ASME • Open arrow for ISO / JIS	

Tolerance Styles

Parameter Name	Description	Applies at modification
Type	Specifies the tolerance type (e.g. straightness, circularity, etc.)	No
Value	Specifies the tolerance value.	No
UpperText	Specifies the tolerance upper text.	No
LowerText	Specifies the tolerance lower text.	No

Font > Name	Specifies the name of the font that should be used for tolerances. If no font name is specified, the system's default font will be used. Make sure that the text font specified here belongs to the list of allowed text fonts, as defined in the **General > AllowedTextFonts** node of the Standards editor.	Yes
Font > Bold	Indicates whether or not tolerances should be displayed in bold.	Yes
Font > Italic	Indicates whether or not tolerances should be displayed in italic.	Yes
Font > Size	Indicates the font size that should be used for tolerances.	Yes
Font > Ratio	Specifies the ratio that should be used to display tolerances.	Yes
Font > Spacing	Specifies the spacing that should be used to display tolerances.	Yes
Text > AnchorPoint	Specifies the tolerance text position in relation to the anchor point (e.g., top left, middle left, etc.).	Yes
Text > OrientationReference	Specifies whether the sheet, or whether the view or 2D component should be used as the reference for the tolerance text orientation.	Yes
Text > Angle	Specifies the tolerance text orientation angle according to the chosen reference.	Yes
Text > Blanking	Indicates whether or not blanking should be used.	Yes
Text > Mirroring	Specifies whether a symmetry, and which one, should be applied to the tolerance.	Yes
Text > Blanking	Indicates whether or not blanking should be used.	Yes
Graphic > Thickness	Specifies the line thickness index (as defined in the LineThickness node of the current standard) that should be used to represent text frames and leaders.	Yes
Graphic > Linetype	Specifies the number of the linetype (as defined in the LineTypes node of the current standard) that should be used to represent tolerance frames and leaders.	Yes

Graphic > Color	Specifies the color that should be used to represent tolerance frames and leaders.	Yes
Leader > Symbol	Specifies the symbol (e.g., simple arrow, circle, etc.) that should be used for tolerance leaders. If you choose the Automatic option, a default symbol will be used, depending on the standard type and on whether the leader is associated to an element or not: • If the leader is associated to an element: ○ Filled arrow for ANSI / ASME ○ Open arrow for ISO / JIS • If the leader is not associated to an element: ○ Symmetric circle (a.k.a. Unfilled circle) for ANSI / ASME ○ Filled circle for ISO / JIS	Yes
Leader > Delta	Specifies the value of the delta that should be applied between a tolerance and its leader.	Yes
Leader > AnchorPoint	Specifies the leader position in relation to the anchor point of the tolerance frame. - 0 positions the leader automatically on the closest anchor point. - 1 positions the leader on the middle-left anchor point. - 2 positions the leader on the middle-right anchor point.	Yes

Balloon Styles

Parameter Name	Description	Applies at modification
Font > Name	Specifies the name of the font that should be used for balloons. If no font name is specified, the system's default font will be used. Make sure that the text font specified here belongs to the list of allowed text fonts, as defined in the **General > AllowedTextFonts** node of the Standards editor.	Yes

Font > Bold	Indicates whether or not balloons should be displayed in bold.	Yes
Font > Italic	Indicates whether or not balloons should be displayed in italic.	Yes
Font > Size	Indicates the font size that should be used for balloons.	Yes
Font > Underline	Indicates whether or not balloon fonts should be underlined.	Yes
Font > Ratio	Specifies the ratio that should be used to display balloons.	Yes
Font > Slant	Specifies the slant that should be used to display balloons.	Yes
Font > Spacing	Specifies the spacing that should be used to display balloons.	Yes
Font > Pitch	Specifies the pitch (fixed or variable) that should be used to display balloons.	Yes
Font > Strikethrough	Indicates whether or not strikethrough should be used in balloons.	Yes
Font > Overline	Indicates whether or not overline should be used in balloons.	Yes
Text > Frame	Specifies the type of frame that should be used to represent balloons. Note that fixed-size frames are defined in the Frame node of the current standard. Accepted values: none, circle, fixed-size circle	Yes
Text > AnchorPoint	Specifies the balloon position in relation to the anchor point (e.g., top left, middle left, etc.).	Yes
Text > AnchorLine	Specifies the balloon position in relation to the anchor line (e.g., top or bottom, cap or base, etc.).	Yes
Text > OrientationReference	Specifies whether the sheet, or whether the view or 2D component should be used as the reference for the balloon orientation.	Yes
Text > Angle	Specifies the balloon orientation angle according to the chosen reference.	Yes
Text > Mirroring	Specifies whether a symmetry, and which one, should be applied to the balloon.	Yes

Text > ApplyScale	Specifies whether or not the scale of the view or of the 2D reference component scale should be applied to the display of the balloon.	Yes
Text > Display	Choose the display mode you want for the balloon: - Show value: displays the balloon, its leader and its frame. - Show box: replaces the balloon and its frame by a rectangular box and displays its leader. - Hide value: hides the balloon and its frame but displays its leader.	Yes
Text > Blanking	Indicates whether or not blanking should be used.	Yes
Graphic > Thickness	Specifies the line thickness index (as defined in the LineThickness node of the current standard) that should be used to represent balloon frames and leaders.	Yes
Graphic > Linetype	Specifies the number of the linetype (as defined in the LineTypes node of the current standard) that should be used to represent balloon frames and leaders.	Yes
Graphic > Color	Specifies the color that should be used to represent balloon frames, leaders and texts.	Yes

Leader > Symbol	Specifies the symbol (e.g., simple arrow, circle, etc.) that should be used for balloon leaders.
	
	If you choose the Automatic option, a default symbol will be used, depending on the standard type and on whether the leader is associated to an element or not:
	
	• If the leader is associated to an element:
	  ○ Closed arrow (a.k.a. Unfilled arrow) for ANSI / ASME
	  ○ Open arrow for ISO / JIS
	
	• If the leader is not associated to an element:
	  ○ Blanked circle for ANSI / ASME
	  ○ Filled circle for ISO / JIS
		Yes

RoughnessSymbol Styles

Parameter Name	Description	Applies at modification
Values > Type	Specifies the roughness symbol type (e.g. basic, machining required, machining prohibited)	No
Values > Mode	Specifies the roughness surface pattern (e.g. M, C, orthogonal, etc.)	No
Values > SpecificationLine	Indicates whether or not the specification line should be displayed.	No
Values > AllAround	Indicates whether or not an all-around symbol should be added.	No
Values > FirstRequirement	Specifies the first requirement.	No
Values > SecondRequirement	Specifies the second requirement.	No
Values > OtherRequirement	Specifies another requirement.	No
Values > ProductionMethod	Specifies the production method.	No
Values > MachiningAllowance	Specifies the machining allowance.	No
Values > CutOff	Specifies the cutoff value.	No
Values > Max	Specifies the maximum value.	No
Values > Min	Specifies the minimum value.	No
Text > AnchorPoint	Specifies the position of roughness symbol texts in relation to the anchor point (e.g., top left, middle left, etc.).	Yes
Text > AnchorLine	Specifies the position of roughness symbol texts in relation to the anchor line (e.g., top or bottom, cap or base, etc.).	Yes
Text > Blanking	Indicates whether or not blanking should be used.	Yes
Graphic > Color	Specifies the color that should be used to represent roughness symbols.	Yes
Graphic > Linetype	Specifies the number of the linetype (as defined in the LineTypes node of the current standard) that should be used to represent roughness symbol leaders.	Yes
Graphic > Thickness	Specifies the line thickness index (as defined in the LineThickness node of the current standard) that should be used to represent roughness symbol leaders.	Yes

Font > Name	Specifies the name of the font that should be used for roughness symbols. If no font name is specified, the system's default font will be used. Make sure that the text font specified here belongs to the list of allowed text fonts, as defined in the **General > AllowedTextFonts** node of the Standards editor.	Yes
Font > Bold	Indicates whether or not roughness symbol fonts should be displayed in bold.	Yes
Font > Italic	Indicates whether or not roughness symbol fonts should be displayed in italic.	Yes
Font > Size	Indicates the font size that should be used for roughness symbols.	Yes
Font > Ratio	Specifies the ratio that should be used to display roughness symbols.	Yes
Font > Spacing	Specifies the spacing that should be used to display roughness symbols.	Yes

Leader > Symbol	Specifies the symbol (e.g., simple arrow, circle, etc.) that should be used for roughness symbol leaders.
	
	If you choose the Automatic option, a default symbol will be used, depending on the standard type and on whether the leader is associated to an element or not:
	
	• If the leader is associated to an element:
	  ○ Closed arrow (a.k.a. Unfilled arrow) for ANSI / ASME
	  ○ Open arrow for ISO / JIS
	
	• If the leader is not associated to an element:
	  ○ Symmetric circle (a.k.a. Unfilled circle) for ANSI / ASME
	  ○ Filled circle for ISO / JIS
		Yes

Leader > Delta	Specifies the value of the delta that should be applied between a roughness symbol and its leader.	Yes

Leader > AnchorPoint	Specifies the leader position in relation to the anchor point of the roughness symbol. - 0 positions the leader automatically on the closest anchor point. - 1 positions the leader on the middle-left anchor point. - 2 positions the leader on the middle-right anchor point.	Yes

WeldingSymbol Styles

Parameter Name	Description	Applies at modification
Length1	Specifies the length of the first welding symbol.	No
Size1	Specifies the size of the first welding symbol.	No
WeldingType1	Specifies the type (e.g. SquareWelding, UGrooveWelding, etc.) of the first welding symbol.	No
SurfaceShape1	Specifies the surface shape (e.g. flat, convex, etc.) of the first welding symbol.	No
FinishingMethod1	Specifies the finishing method (e.g. grinding, hammering, etc.) of the first welding symbol.	No
Length2	Specifies the length of the second welding symbol.	No
Size2	Specifies the size of the second welding symbol.	No
WeldingType2	Specifies the type (e.g. SquareWelding, UGrooveWelding, etc.) of the second welding symbol.	No
SurfaceShape2	Specifies the surface shape (e.g. flat, convex, etc.) of the second welding symbol.	No
FinishingMethod2	Specifies the finishing method (e.g. grinding, hammering, etc.) of the second welding symbol.	No
Reference	Specifies the reference of the welding symbol.	No
FieldWeld	Indicates whether or not a field weld should be added.	No
AllAround	Indicates whether or not a weld-all-around symbol should be added.	No

ReferenceLine	Indicates whether the reference line should be up or down. *This parameter works only for the ISO standard.*	No
Text > Frame	Specifies the type of frame that should be used to represent welding symbols. Note that fixed-size frames are defined in the Frame node of the current standard. Accepted values: none, rectangle	Yes
Text > OrientationReference	Specifies whether the sheet, or whether the view or 2D component should be used as the reference for the welding symbol orientation.	Yes
Text > Angle	Specifies the welding symbol orientation angle according to the chosen reference.	Yes
Text > LineSpacing	Specifies the spacing that should be used between two lines of text.	Yes
Text > LineSpacingMode	Specifies the spacing mode between two lines of text (e.g., bottom to top, base to cap, etc.).	Yes
Font > Name	Specifies the name of the font that should be used for welding symbols. If no font name is specified, the system's default font will be used. Make sure that the text font specified here belongs to the list of allowed text fonts, as defined in the **General > AllowedTextFonts** node of the Standards editor.	Yes
Font > Bold	Indicates whether or not welding symbols should be displayed in bold.	Yes
Font > Italic	Indicates whether or not welding symbols should be displayed in italic.	Yes
Font > Size	Indicates the font size that should be used for welding symbols.	Yes
Font > Ratio	Specifies the ratio that should be used to display welding symbols.	Yes
Font > Spacing	Specifies the spacing that should be used to display welding symbols.	Yes
Graphic > Thickness	Specifies the line thickness index (as defined in the LineThickness node of the current standard) that should be used to represent welding symbol frames and leaders.	Yes

Graphic > Linetype	Specifies the number of the linetype (as defined in the LineTypes node of the current standard) that should be used to represent welding symbol frames and leaders.	Yes
Graphic > Color	Specifies the color that should be used to represent welding symbol frames and leaders.	Yes
Leader > Symbol	Specifies the symbol (e.g., simple arrow, circle, etc.) that should be used for welding symbol leaders. If you choose the Automatic option, filled arrow will be used by default.	Yes

Dimension Styles

This section deals with dimension styles. These let you define the default values that will be used when creating different types of dimensions.

Defining Dimension Styles

Dimension styles are located in the following nodes of the Standards editor, available via **Tools -> Standards:**

- Styles -> DistanceLengthDimension
- Styles -> AngleDimension
- Styles -> RadiusDimension
- Styles -> DiameterDimension
- Styles -> ChamferDimension
- Styles -> CoordinateDimension

By default, a style called Default is available for each dimension style.

All parameters are taken into account both at creation time (i.e. when creating a dimension), and at modification time (i.e. when reapplying a style to a dimension).

DistanceLengthDimension Styles

Parameter Name	Description
Value > OrientationReference	Specifies whether the screen, the view, the dimension line or the extension line should be used as the reference for the distance length dimension value orientation.
Value > Angle	Specifies the distance length dimension value orientation angle according to the chosen reference.
Value > Position	Specifies the distance length dimension value position.

Value > OffsetX	Specifies the horizontal offset value for distance length dimension values: • if Value > OrientationReference is set to Screen, the horizontal direction is defined by the X axis of the screen. • if Value > OrientationReference is set to View, the horizontal direction is defined by the X axis of the view. • if Value > OrientationReference is set to Dimension Line, the horizontal direction is parallel to the dimension line. • if Value > OrientationReference is set to Extension Line, the horizontal direction is parallel to the dimension extension line.
Value > OffsetY	Specifies the vertical offset value for distance length dimension values: • if Value > OrientationReference is set to Screen, the horizontal direction is defined by the Y axis of the screen. • if Value > OrientationReference is set to View, the horizontal direction is defined by the Y axis of the view. • if Value > OrientationReference is set to Dimension Line, the horizontal direction is normal to the dimension line. • if Value > OrientationReference is set to Extension Line, the horizontal direction is normal to the dimension extension line.
Value > Display	Choose the display mode you want for the distance length dimension: - Show value: displays the dimension, its leader and its frame. - Show box: replaces the dimension and its frame by a rectangular box and displays its leader. - Hide value: hides the dimension and its frame but displays its leader.
Driving	Indicates whether or not distance length dimensions should be driving dimensions.
DualValueDisplay	Specifies whether there will be a dual value display for the distance length dimension, and, if any, what kind (e.g. fractional, side-by-side, etc.).

ValueDisplayFormat > MainValue > Name	Specifies the name of the main value display format. Make sure that the display format specified here belongs to the list of Value Display styles allowed on dimensions, as defined in the **General > AllowedNumericalFormats** node of the Standards editor.
ValueDisplayFormat > MainValue > DisplayedFactorNumber	Specifies the number of factors displayed for the main value, according to the number of factors available for the value display format used by this style.
ValueDisplayFormat > MainValue > PrecisionMode	Specifies whether the precision mode for the main value will be decimal or fractional.
ValueDisplayFormat > MainValue > Precision	Specifies the precision for the main value.
ValueDisplayFormat > DualValue > Name	Specifies the name of the dual value display format, if any. Make sure that the display format specified here belongs to the list of Value Display styles allowed on dimensions, as defined in the **General > AllowedNumericalFormats** node of the Standards editor.
ValueDisplayFormat > DualValue > DisplayedFactorNumber	Specifies the number of factors displayed for the dual value (if any), according to the number of factors available for the value display format used by this style.
ValueDisplayFormat > DualValue > PrecisionMode	Specifies whether the precision mode for the dual value (if any) will be decimal or fractional.
ValueDisplayFormat > DualValue > Precision	Specifies the precision for the dual value, if any.
Fake > Mode	Indicates whether distance length dimensions will be fake dimensions, and, if yes, of what type (e.g. numerical or alphanumerical).
Fake > MainValue	Specifies the fake main value for distance length dimensions.
Fake > DualValue	Specifies the fake dual value for distance length dimensions.
Tolerance > MainValue > Format	Specifies the tolerance main value format for distance length dimensions.
Tolerance > MainValue > AlphanumericalValue1	Specifies the first alphanumerical value for the tolerance main value.
Tolerance > MainValue > AlphanumericalValue2	Specifies the second alphanumerical value for the tolerance main value.

Tolerance > MainValue > NumericalLower	Specifies the lower numerical for the tolerance main value.
Tolerance > MainValue > NumericalUpper	Specifies the upper numerical for the tolerance main value.
Tolerance > DualValue > Format	Specifies the tolerance dual value format for distance length dimensions.
Tolerance > DualValue > AlphanumericalValue1	Specifies the first alphanumerical value for the tolerance dual value.
Tolerance > DualValue > AlphanumericalValue2	Specifies the second alphanumerical value for the tolerance dual value.
Tolerance > DualValue > NumericalLower	Specifies the lower numerical for the tolerance dual value.
Tolerance > DualValue > NumericalUpper	Specifies the upper numerical for the tolerance dual value.
DimensionLine > Representation	Specifies how the dimension line should be represented (e.g. regular, one-part leader, etc.)
DimensionLine > Color	Specifies the color that should be used to display dimension lines.
DimensionLine > Thickness	Specifies the line thickness index (as defined in the LineThickness node of the current standard) that should be used to represent dimension lines.
DimensionLine > SecondPartReference	In the case of Two parts or Leader two parts for the representation, specifies the Reference for positioning the second part of the dimension line.
DimensionLine > SecondPartAngle	In the case of Two parts or Leader two parts for the representation, specifies the angle for the second part of the dimension line in relation to its reference.
DimensionLine > LeaderAngle	Specifies the angle for the dimension line leader.
Symbols > Symbol1 > Type	Specifies the type of the first symbol (e.g. arrow, filled circle, etc.) that should be used for distance length dimensions. If you choose the Automatic option, simple arrow will be used by default.
Symbols > Symbol1 > Color	Specifies the color of the first symbol.
Symbols > Symbol1 > Thickness	Specifies the line thickness index (as defined in the LineThickness node of the current standard) of the first symbol.
Symbols > Symbol2 > Type	Specifies the type of the second symbol (e.g. arrow, filled circle, etc.) that should be used for distance length dimensions. If you choose the Automatic option, simple arrow will be used by default.

Symbols > Symbol2 > Color	Specifies the color of the second symbol.
Symbols > Symbol2 > Thickness	Specifies the line thickness index (as defined in the LineThickness node of the current standard) of the second symbol.
Symbols > SymbolMode	Specifies the symbol mode (e.g. inside, outside, etc.).
ExtensionLine > Color	Specifies the color of the distance length dimension extension line.
ExtensionLine > Thickness	Specifies the line thickness index (as defined in the LineThickness node of the current standard) of the distance length dimension extension line.
ExtensionLine > SlantAngle	Specifies the slant angle for the extension line. This angle is contained between 90 degrees and -90 degrees excluded, the default angle being 0 degree.
ExtensionLine > Left > Hide	Indicates whether or not the left extension line should be hidden.
ExtensionLine > Left > Overrun	Specifies the overrun for the left extension line.
ExtensionLine > Left > Blanking	Specifies the blanking for the left extension line.
ExtensionLine > Right > Hide	Indicates whether or not the right extension line should be hidden.
ExtensionLine > Right > Overrun	Specifies the overrun for the right extension line.
ExtensionLine > Right > Blanking	Specifies the blanking for the right extension line.
ExtensionLine > Funnel > Display	Indicates whether or not the extension line should be displayed as a funnel.
ExtensionLine > Funnel > Height	Specifies the funnel height.
ExtensionLine > Funnel > Width	Specifies the funnel width.
ExtensionLine > Funnel > Angle	Specifies the funnel angle.
ExtensionLine > Funnel > Mode	Specifies the funnel mode (external or internal).
ExtensionLine > Funnel > Side	Specifies whether the funnel should be applied on the left or bottom, on the right or top, or on both sides.
AssociatedTexts > MainValue > Prefix	Specifies the prefix for the main value of the associated text.
AssociatedTexts > MainValue > Suffix	Specifies the suffix for the main value of the associated text.
AssociatedTexts > MainValue > Before	Specifies the text that should be displayed before the main value of the associated text.

AssociatedTexts > MainValue > After	Specifies the text that should be displayed after the main value of the associated text.
AssociatedTexts > MainValue > Upper	Specifies the text that should be displayed above the main value of the associated text.
AssociatedTexts > MainValue > Lower	Specifies the text that should be displayed below the main value of the associated text.
AssociatedTexts > DualValue > Before	Specifies the text that should be displayed before the dual value of the associated text.
AssociatedTexts > DualValue > After	Specifies the text that should be displayed after the dual value of the associated text.
AssociatedTexts > DualValue > Upper	Specifies the text that should be displayed above the dual value of the associated text.
AssociatedTexts > DualValue > Lower	Specifies the text that should be displayed below the dual value of the associated text.
Framing > FramedSubpart	Specifies whether the frame subpart should display the value, the value and tolerance, etc.
Framing > FramedGroup	Specifies whether the framed group should display the main value, the dual value, both values separately, etc.
Framing > Frame	Specifies the type of frame that should be used for distance length dimensions. Note that fixed-size frames are defined in the Frame node of the current standard. Accepted values: none, circle, fixed-size scored circle, fixed-size diamond, fixed-size square, rectangle, oblong, right flag, fixed-size triangle
Framing > MainValueScoring	Specifies the main value scoring.
Framing > DualValueScoring	Specifies the dual value scoring.
Font > Name	Specifies the name of the font that should be used for distance length dimension texts. If no font name is specified, the system's default font will be used.
Font > Bold	Indicates whether or not distance length dimension texts should be displayed in bold.
Font > Italic	Indicates whether or not distance length dimension texts should be displayed in italic.
Font > Size	Indicates the font size that should be used for distance length dimension texts.
Font > Color	Specifies the color that should be used to display distance length dimension texts.

Font > Underline	Indicates whether or not distance length dimension texts should be underlined (in this case, overline cannot be applied).
Font > Strikethrough	Indicates whether or not strikethrough should be used for distance length dimension texts.
Font > Overline	Indicates whether or not distance length dimension texts should be overlined (in this case, underline cannot be applied).
Font > Ratio	Specifies the character width.
Font > Spacing	Specifies the spacing between characters.
Font > ApplyScale	Indicates whether or not distance length dimension texts should be scaled according to the view or the 2D reference component's scale.

AngleDimension Styles

Parameter Name	Description
Value > OrientationReference	Specifies whether the screen, the view, the dimension line or the extension line should be used as the reference for the angle dimension value orientation.
Value > Angle	Specifies the angle dimension value orientation angle according to the chosen reference.
Value > Position	Specifies the angle dimension value position.
Value > OffsetX	Specifies the horizontal offset value for angle dimension values: • if Value > OrientationReference is set to Screen, the horizontal direction is defined by the X axis of the screen. • if Value > OrientationReference is set to View, the horizontal direction is defined by the X axis of the view. • if Value > OrientationReference is set to Dimension Line, the horizontal direction is parallel to the dimension line. • if Value > OrientationReference is set to Extension Line, the horizontal direction is parallel to the dimension extension line.

Value > OffsetY	Specifies the vertical offset value for angle dimension values: • if Value > OrientationReference is set to Screen, the horizontal direction is defined by the Y axis of the screen. • if Value > OrientationReference is set to View, the horizontal direction is defined by the Y axis of the view. • if Value > OrientationReference is set to Dimension Line, the horizontal direction is normal to the dimension line. • if Value > OrientationReference is set to Extension Line, the horizontal direction is normal to the dimension extension line.
Value > Display	Choose the display mode you want for the angle dimension: - Show value: displays the dimension, its leader and its frame. - Show box: replaces the dimension and its frame by a rectangular box and displays its leader. - Hide value: hides the dimension and its frame but displays its leader.
Driving	Indicates whether or not angle dimensions should be driving dimensions.
DualValueDisplay	Specifies whether there will be a dual value display for the angle dimension, and, if any, what kind (e.g. fractional, side-by-side, etc.).
ValueDisplayFormat > MainValue > Name	Specifies the name of the main value display format. Make sure that the display format specified here belongs to the list of Value Display styles allowed on dimensions, as defined in the **General > AllowedNumericalFormats** node of the Standards editor.
ValueDisplayFormat > MainValue > DisplayedFactorNumber	Specifies the number of factors displayed for the main value, according to the number of factors available for the value display format used by this style.
ValueDisplayFormat > MainValue > PrecisionMode	Specifies whether the precision mode for the main value will be decimal or fractional.
ValueDisplayFormat > MainValue > Precision	Specifies the precision for the main value.

ValueDisplayFormat > DualValue > Name	Specifies the name of the dual value display format, if any. Make sure that the display format specified here belongs to the list of Value Display styles allowed on dimensions, as defined in the **General > AllowedNumericalFormats** node of the Standards editor.
ValueDisplayFormat > DualValue > DisplayedFactorNumber	Specifies the number of factors displayed for the dual value (if any), according to the number of factors available for the value display format used by this style.
ValueDisplayFormat > DualValue > PrecisionMode	Specifies whether the precision mode for the dual value (if any) will be decimal or fractional.
ValueDisplayFormat > DualValue > Precision	Specifies the precision for the dual value, if any.
Fake > Mode	Indicates whether angle dimensions will be fake dimensions, and, if yes, of what type (e.g. numerical or alphanumerical).
Fake > MainValue	Specifies the fake main value for angle dimensions.
Fake > DualValue	Specifies the fake dual value for angle dimensions.
Tolerance > MainValue > Format	Specifies the tolerance main value format for angle dimensions.
Tolerance > MainValue > AlphanumericalValue1	Specifies the first alphanumerical value for the tolerance main value.
Tolerance > MainValue > AlphanumericalValue2	Specifies the second alphanumerical value for the tolerance main value.
Tolerance > MainValue > NumericalLower	Specifies the lower numerical for the tolerance main value.
Tolerance > MainValue > NumericalUpper	Specifies the upper numerical for the tolerance main value.
Tolerance > DualValue > Format	Specifies the tolerance dual value format for angle dimensions.
Tolerance > DualValue > AlphanumericalValue1	Specifies the first alphanumerical value for the tolerance dual value.
Tolerance > DualValue > AlphanumericalValue2	Specifies the second alphanumerical value for the tolerance dual value.
Tolerance > DualValue > NumericalLower	Specifies the lower numerical for the tolerance dual value.
Tolerance > DualValue > NumericalUpper	Specifies the upper numerical for the tolerance dual value.

DimensionLine > Representation	Specifies how the dimension line should be represented (e.g. regular, one-part leader, etc.)
DimensionLine > Color	Specifies the color that should be used to display dimension lines.
DimensionLine > Thickness	Specifies the line thickness index (as defined in the LineThickness node of the current standard) that should be used to represent dimension lines.
DimensionLine > SecondPartReference	In the case of Two parts or Leader two parts for the representation, specifies the Reference for positioning the second part of the dimension line.
DimensionLine > SecondPartAngle	In the case of Two parts or Leader two parts for the representation, specifies the angle for the second part of the dimension line in relation to its reference.
DimensionLine > LeaderAngle	Specifies the angle for the dimension line leader.
Symbols > Symbol1 > Type	Specifies the type of the first symbol (e.g. arrow, filled circle, etc.) that should be used for angle dimensions. If you choose the Automatic option, simple arrow will be used by default.
Symbols > Symbol1 > Color	Specifies the color of the first symbol.
Symbols > Symbol1 > Thickness	Specifies the line thickness index (as defined in the LineThickness node of the current standard) of the first symbol.
Symbols > Symbol2 > Type	Specifies the type of the second symbol (e.g. arrow, filled circle, etc.) that should be used for angle dimensions. If you choose the Automatic option, simple arrow will be used by default.
Symbols > Symbol2 > Color	Specifies the color of the second symbol.
Symbols > Symbol2 > Thickness	Specifies the line thickness index (as defined in the LineThickness node of the current standard) of the second symbol.
Symbols > SymbolMode	Specifies the symbol mode (e.g. inside, outside, etc.).
ExtensionLine > Color	Specifies the color of the angle dimension extension line.
ExtensionLine > Thickness	Specifies the line thickness index (as defined in the LineThickness node of the current standard) of the angle dimension extension line.
ExtensionLine > SlantAngle	Specifies the slant angle for the extension line. This angle is contained between 90 degrees and -90 degrees excluded, the default angle being 0 degree.
ExtensionLine > Left > Hide	Indicates whether or not the left extension line should be hidden.

ExtensionLine > Left > Overrun	Specifies the overrun for the left extension line.
ExtensionLine > Left > Blanking	Specifies the blanking for the left extension line.
ExtensionLine > Right > Hide	Indicates whether or not the right extension line should be hidden.
ExtensionLine > Right > Overrun	Specifies the overrun for the right extension line.
ExtensionLine > Right > Blanking	Specifies the blanking for the right extension line.
AssociatedTexts > MainValue > Prefix	Specifies the prefix for the main value of the associated text.
AssociatedTexts > MainValue > Suffix	Specifies the suffix for the main value of the associated text.
AssociatedTexts > MainValue > Before	Specifies the text that should be displayed before the main value of the associated text.
AssociatedTexts > MainValue > After	Specifies the text that should be displayed after the main value of the associated text.
AssociatedTexts > MainValue > Upper	Specifies the text that should be displayed above the main value of the associated text.
AssociatedTexts > MainValue > Lower	Specifies the text that should be displayed below the main value of the associated text.
AssociatedTexts > DualValue > Before	Specifies the text that should be displayed before the dual value of the associated text.
AssociatedTexts > DualValue > After	Specifies the text that should be displayed after the dual value of the associated text.
AssociatedTexts > DualValue > Upper	Specifies the text that should be displayed above the dual value of the associated text.
AssociatedTexts > DualValue > Lower	Specifies the text that should be displayed below the dual value of the associated text.
Framing > FramedSubpart	Specifies whether the frame subpart should display the value, the value and tolerance, etc.
Framing > FramedGroup	Specifies whether the framed group should display the main value, the dual value, both values separately, etc.
Framing > Frame	Specifies the type of frame that should be used for angle dimensions. Note that fixed-size frames are defined in the Frame node of the current standard. Accepted values: none, circle, fixed-size scored circle, fixed-size diamond, fixed-size square, rectangle, oblong, right flag, fixed-size triangle
Framing > MainValueScoring	Specifies the main value scoring.

Framing > DualValueScoring	Specifies the dual value scoring.
Font > Name	Specifies the name of the font that should be used for angle dimension texts. If no font name is specified, the system's default font will be used.
Font > Bold	Indicates whether or not angle dimension texts should be displayed in bold.
Font > Italic	Indicates whether or not angle dimension texts should be displayed in italic.
Font > Size	Indicates the font size that should be used for angle dimension texts.
Font > Color	Specifies the color that should be used to display angle dimension texts.
Font > Underline	Indicates whether or not angle dimension texts should be underlined (in this case, overline cannot be applied).
Font > Strikethrough	Indicates whether or not strikethrough should be used for angle dimension texts.
Font > Overline	Indicates whether or not angle dimension texts should be overlined (in this case, underline cannot be applied).
Font > Ratio	Specifies the character width.
Font > Spacing	Specifies the spacing between characters.
Font > ApplyScale	Indicates whether or not angle dimension texts should be scaled according to the view or the 2D reference component's scale.

RadiusDimension Styles

Parameter Name	Description
Value > OrientationReference	Specifies whether the screen, the view or the dimension line should be used as the reference for the radius dimension value orientation.
Value > Angle	Specifies the radius dimension value orientation angle according to the chosen reference.
Value > Position	Specifies the radius dimension value position.

Value > OffsetX	Specifies the horizontal offset value for radius dimension values: • if Value > OrientationReference is set to Screen, the horizontal direction is defined by the X axis of the screen. • if Value > OrientationReference is set to View, the horizontal direction is defined by the X axis of the view. • if Value > OrientationReference is set to Dimension Line, the horizontal direction is parallel to the dimension line.
Value > OffsetY	Specifies the vertical offset value for radius dimension values: • if Value > OrientationReference is set to Screen, the horizontal direction is defined by the Y axis of the screen. • if Value > OrientationReference is set to View, the horizontal direction is defined by the Y axis of the view. • if Value > OrientationReference is set to Dimension Line, the horizontal direction is normal to the dimension line.
Value > Display	Choose the display mode you want for the radius dimension: - Show value: displays the dimension, its leader and its frame. - Show box: replaces the dimension and its frame by a rectangular box and displays its leader. - Hide value: hides the dimension and its frame but displays its leader.
Driving	Indicates whether or not radius dimensions should be driving dimensions.
DualValueDisplay	Specifies whether there will be a dual value display for the radius dimension, and, if any, what kind (e.g. fractional, side-by-side, etc.).
ValueDisplayFormat > MainValue > Name	Specifies the name of the main value display format. Make sure that the display format specified here belongs to the list of Value Display styles allowed on dimensions, as defined in the **General > AllowedNumericalFormats** node of the Standards editor.

ValueDisplayFormat > MainValue > DisplayedFactorNumber	Specifies the number of factors displayed for the main value, according to the number of factors available for the value display format used by this style.
ValueDisplayFormat > MainValue > PrecisionMode	Specifies whether the precision mode for the main value will be decimal or fractional.
ValueDisplayFormat > MainValue > Precision	Specifies the precision for the main value.
ValueDisplayFormat > DualValue > Name	Specifies the name of the dual value display format, if any. Make sure that the display format specified here belongs to the list of Value Display styles allowed on dimensions, as defined in the **General > AllowedNumericalFormats** node of the Standards editor.
ValueDisplayFormat > DualValue > DisplayedFactorNumber	Specifies the number of factors displayed for the dual value (if any), according to the number of factors available for the value display format used by this style.
ValueDisplayFormat > DualValue > PrecisionMode	Specifies whether the precision mode for the dual value (if any) will be decimal or fractional.
ValueDisplayFormat > DualValue > Precision	Specifies the precision for the dual value, if any.
Fake > Mode	Indicates whether radius dimensions will be fake dimensions, and, if yes, of what type (e.g. numerical or alphanumerical).
Fake > MainValue	Specifies the fake main value for radius dimensions.
Fake > DualValue	Specifies the fake dual value for radius dimensions.
Tolerance > MainValue > Format	Specifies the tolerance main value format for radius dimensions.
Tolerance > MainValue > AlphanumericalValue1	Specifies the first alphanumerical value for the tolerance main value.
Tolerance > MainValue > AlphanumericalValue2	Specifies the second alphanumerical value for the tolerance main value.
Tolerance > MainValue > NumericalLower	Specifies the lower numerical for the tolerance main value.
Tolerance > MainValue > NumericalUpper	Specifies the upper numerical for the tolerance main value.
Tolerance > DualValue > Format	Specifies the tolerance dual value format for radius dimensions.
Tolerance > DualValue > AlphanumericalValue1	Specifies the first alphanumerical value for the tolerance dual value.

Tolerance > DualValue > AlphanumericalValue2	Specifies the second alphanumerical value for the tolerance dual value.
Tolerance > DualValue > NumericalLower	Specifies the lower numerical for the tolerance dual value.
Tolerance > DualValue > NumericalUpper	Specifies the upper numerical for the tolerance dual value.
DimensionLine > Representation	Specifies how the dimension line should be represented (e.g. regular, one-part leader, etc.)
DimensionLine > Color	Specifies the color that should be used to display dimension lines.
DimensionLine > Thickness	Specifies the line thickness index (as defined in the LineThickness node of the current standard) that should be used to represent dimension lines.
DimensionLine > SecondPartReference	In the case of Two parts or Leader two parts for the representation, specifies the Reference for positioning the second part of the dimension line.
DimensionLine > SecondPartAngle	In the case of Two parts or Leader two parts for the representation, specifies the angle for the second part of the dimension line in relation to its reference.
DimensionLine > LeaderAngle	Specifies the angle for the dimension line leader.
DimensionLine > TillCenter	Indicates whether or not the dimension line should reach the center.
DimensionLine > ExtensionFromStandard	Indicates whether or not the extension of the dimension line should be driven by the standards. If set to Yes, then the DimensionLine > TillCenter parameter is not taken into account; in this case, the extension is performed as defined by the DIMLRadiusIntReachCenter and the DIMLRadiusExtReachCenter standard parameters. Refer to Dimension Parameters for more information.
Symbols > Symbol1 > Type	Specifies the type of the first symbol (e.g. arrow, filled circle, etc.) that should be used for radius dimensions. If you choose the Automatic option, simple arrow will be used by default.
Symbols > Symbol1 > Color	Specifies the color of the first symbol.
Symbols > Symbol1 > Thickness	Specifies the line thickness index (as defined in the LineThickness node of the current standard) of the first symbol.

Symbols > Symbol2 > Type	Specifies the type of the second symbol (e.g. arrow, filled circle, etc.) that should be used for radius dimensions. If you choose the Automatic option, simple arrow will be used by default.
Symbols > Symbol2 > Color	Specifies the color of the second symbol.
Symbols > Symbol2 > Thickness	Specifies the line thickness index (as defined in the LineThickness node of the current standard) of the second symbol.
Symbols > SymbolMode	Specifies the symbol mode (e.g. inside, outside, etc.).
Foreshortened > IsForeshortened	Indicates whether or not the radius dimension should be foreshortened.
Foreshortened > ValuePosition	Specifies whether the foreshortened dimension value should be positioned on the long segment or on the short segment.
Foreshortened > Orientation	Specifies whether the foreshortened dimension orientation should be parallel or convergent.
Foreshortened > Angle	Specifies the foreshortened dimension angle.
Foreshortened > Ratio	Specifies the foreshortened dimension ratio.
Foreshortened > SymbolScale	Specifies the scale that should be used for the foreshortened dimension symbol.
Foreshortened > MoveEndPoint	Indicates whether or not the foreshortened dimension end point can be moved.
ExtensionLine > Color	Specifies the color of the radius dimension extension line.
ExtensionLine > Thickness	Specifies the line thickness index (as defined in the LineThickness node of the current standard) of the radius dimension extension line.
ExtensionLine > SlantAngle	Specifies the slant angle for the extension line. This angle is contained between 90 degrees and -90 degrees excluded, the default angle being 0 degree.
ExtensionLine > Left > Hide	Indicates whether or not the left extension line should be hidden.
ExtensionLine > Left > Overrun	Specifies the overrun for the left extension line.
ExtensionLine > Left > Blanking	Specifies the blanking for the left extension line.
ExtensionLine > Right > Hide	Indicates whether or not the right extension line should be hidden.
ExtensionLine > Right > Overrun	Specifies the overrun for the right extension line.

ExtensionLine > Right > Blanking	Specifies the blanking for the right extension line.
ExtensionLine > Funnel > Display	Indicates whether or not the extension line should be displayed as a funnel.
ExtensionLine > Funnel > Height	Specifies the funnel height.
ExtensionLine > Funnel > Width	Specifies the funnel width.
ExtensionLine > Funnel > Angle	Specifies the funnel angle.
ExtensionLine > Funnel > Mode	Specifies the funnel mode (external or internal).
ExtensionLine > Funnel > Side	Specifies whether the funnel should be applied on the left or bottom, on the right or top, or on both sides.
AssociatedTexts > MainValue > Prefix	Specifies the prefix for the main value of the associated text.
AssociatedTexts > MainValue > Suffix	Specifies the suffix for the main value of the associated text.
AssociatedTexts > MainValue > Before	Specifies the text that should be displayed before the main value of the associated text.
AssociatedTexts > MainValue > After	Specifies the text that should be displayed after the main value of the associated text.
AssociatedTexts > MainValue > Upper	Specifies the text that should be displayed above the main value of the associated text.
AssociatedTexts > MainValue > Lower	Specifies the text that should be displayed below the main value of the associated text.
AssociatedTexts > DualValue > Before	Specifies the text that should be displayed before the dual value of the associated text.
AssociatedTexts > DualValue > After	Specifies the text that should be displayed after the dual value of the associated text.
AssociatedTexts > DualValue > Upper	Specifies the text that should be displayed above the dual value of the associated text.
AssociatedTexts > DualValue > Lower	Specifies the text that should be displayed below the dual value of the associated text.
Framing > FramedSubpart	Specifies whether the frame subpart should display the value, the value and tolerance, etc.
Framing > FramedGroup	Specifies whether the framed group should display the main value, the dual value, both values separately, etc.

Framing > Frame	Specifies the type of frame that should be used for radius dimensions. Note that fixed-size frames are defined in the Frame node of the current standard. Accepted values: none, circle, fixed-size scored circle, fixed-size diamond, fixed-size square, rectangle, oblong, right flag, fixed-size triangle
Framing > MainValueScoring	Specifies the main value scoring.
Framing > DualValueScoring	Specifies the dual value scoring.
Font > Name	Specifies the name of the font that should be used for radius dimension texts. If no font name is specified, the system's default font will be used.
Font > Bold	Indicates whether or not radius dimension texts should be displayed in bold.
Font > Italic	Indicates whether or not radius dimension texts should be displayed in italic.
Font > Size	Indicates the font size that should be used for radius dimension texts.
Font > Color	Specifies the color that should be used to display radius dimension texts.
Font > Underline	Indicates whether or not radius dimension texts should be underlined (in this case, overline cannot be applied).
Font > Strikethrough	Indicates whether or not strikethrough should be used for radius dimension texts.
Font > Overline	Indicates whether or not radius dimension texts should be overlined (in this case, underline cannot be applied).
Font > Ratio	Specifies the character width.
Font > Spacing	Specifies the spacing between characters.
Font > ApplyScale	Indicates whether or not radius dimension texts should be scaled according to the view or the 2D reference component's scale.

DiameterDimension Styles

Parameter Name	Description
Value > OrientationReference	Specifies whether the screen, the view or the dimension line should be used as the reference for the diameter dimension value orientation.
Value > Angle	Specifies the diameter dimension value orientation angle according to the chosen reference.
Value > Position	Specifies the diameter dimension value position.
Value > OffsetX	Specifies the horizontal offset value for diameter dimension values: • if Value > OrientationReference is set to Screen, the horizontal direction is defined by the X axis of the screen. • if Value > OrientationReference is set to View, the horizontal direction is defined by the X axis of the view. • if Value > OrientationReference is set to Dimension Line, the horizontal direction is parallel to the dimension line.
Value > OffsetY	Specifies the vertical offset value for diameter dimension values: • if Value > OrientationReference is set to Screen, the horizontal direction is defined by the Y axis of the screen. • if Value > OrientationReference is set to View, the horizontal direction is defined by the Y axis of the view. • if Value > OrientationReference is set to Dimension Line, the horizontal direction is normal to the dimension line.
Value > Display	Choose the display mode you want for the diameter dimension: - Show value: displays the dimension, its leader and its frame. - Show box: replaces the dimension and its frame by a rectangular box and displays its leader. - Hide value: hides the dimension and its frame but displays its leader.
Driving	Indicates whether or not diameter dimensions should be driving dimensions.

DualValueDisplay	Specifies whether there will be a dual value display for the diameter dimension, and, if any, what kind (e.g. fractional, side-by-side, etc.).
ValueDisplayFormat > MainValue > Name	Specifies the name of the main value display format. Make sure that the display format specified here belongs to the list of Value Display styles allowed on dimensions, as defined in the **General > AllowedNumericalFormats** node of the Standards editor.
ValueDisplayFormat > MainValue > DisplayedFactorNumber	Specifies the number of factors displayed for the main value, according to the number of factors available for the value display format used by this style.
ValueDisplayFormat > MainValue > PrecisionMode	Specifies whether the precision mode for the main value will be decimal or fractional.
ValueDisplayFormat > MainValue > Precision	Specifies the precision for the main value.
ValueDisplayFormat > DualValue > Name	Specifies the name of the dual value display format, if any. Make sure that the display format specified here belongs to the list of Value Display styles allowed on dimensions, as defined in the **General > AllowedNumericalFormats** node of the Standards editor.
ValueDisplayFormat > DualValue > DisplayedFactorNumber	Specifies the number of factors displayed for the dual value (if any), according to the number of factors available for the value display format used by this style.
ValueDisplayFormat > DualValue > PrecisionMode	Specifies whether the precision mode for the dual value (if any) will be decimal or fractional.
ValueDisplayFormat > DualValue > Precision	Specifies the precision for the dual value, if any.
Fake > Mode	Indicates whether diameter dimensions will be fake dimensions, and, if yes, of what type (e.g. numerical or alphanumerical).
Fake > MainValue	Specifies the fake main value for diameter dimensions.
Fake > DualValue	Specifies the fake dual value for diameter dimensions.
Tolerance > MainValue > Format	Specifies the tolerance main value format for diameter dimensions.
Tolerance > MainValue > AlphanumericalValue1	Specifies the first alphanumerical value for the tolerance main value.

Tolerance > MainValue > AlphanumericalValue2	Specifies the second alphanumerical value for the tolerance main value.
Tolerance > MainValue > NumericalLower	Specifies the lower numerical for the tolerance main value.
Tolerance > MainValue > NumericalUpper	Specifies the upper numerical for the tolerance main value.
Tolerance > DualValue > Format	Specifies the tolerance dual value format for diameter dimensions.
Tolerance > DualValue > AlphanumericalValue1	Specifies the first alphanumerical value for the tolerance dual value.
Tolerance > DualValue > AlphanumericalValue2	Specifies the second alphanumerical value for the tolerance dual value.
Tolerance > DualValue > NumericalLower	Specifies the lower numerical for the tolerance dual value.
Tolerance > DualValue > NumericalUpper	Specifies the upper numerical for the tolerance dual value.
DimensionLine > Representation	Specifies how the dimension line should be represented (e.g. regular, one-part leader, etc.)
DimensionLine > Color	Specifies the color that should be used to display dimension lines.
DimensionLine > Thickness	Specifies the line thickness index (as defined in the LineThickness node of the current standard) that should be used to represent dimension lines.
DimensionLine > SecondPartReference	In the case of Two parts or Leader two parts for the representation, specifies the Reference for positioning the second part of the dimension line.
DimensionLine > SecondPartAngle	In the case of Two parts or Leader two parts for the representation, specifies the angle for the second part of the dimension line in relation to its reference.
DimensionLine > LeaderAngle	Specifies the angle for the dimension line leader.
DimensionLine > TillCenter	Indicates whether or not the dimension line should reach the center.
DimensionLine > ExtensionFromStandard	Indicates whether or not the extension of the dimension line should be driven by the standards. If set to Yes, then the DimensionLine > TillCenter parameter is not taken into account; in this case, the extension is performed as defined by the DIMLDiameterIntReachCenter and the DIMLDiameterExtReachCenter standard parameters. Refer to Dimension Parameters for more information.

Symbols > Symbol1 > Type	Specifies the type of the first symbol (e.g. arrow, filled circle, etc.) that should be used for diameter dimensions. If you choose the Automatic option, simple arrow will be used by default.
Symbols > Symbol1 > Color	Specifies the color of the first symbol.
Symbols > Symbol1 > Thickness	Specifies the line thickness index (as defined in the LineThickness node of the current standard) of the first symbol.
Symbols > Symbol2 > Type	Specifies the type of the second symbol (e.g. arrow, filled circle, etc.) that should be used for diameter dimensions. If you choose the Automatic option, simple arrow will be used by default.
Symbols > Symbol2 > Color	Specifies the color of the second symbol.
Symbols > Symbol2 > Thickness	Specifies the line thickness index (as defined in the LineThickness node of the current standard) of the second symbol.
Symbols > SymbolMode	Specifies the symbol mode (e.g. inside, outside, etc.).
ExtensionLine > Color	Specifies the color of the diameter dimension extension line.
ExtensionLine > Thickness	Specifies the line thickness index (as defined in the LineThickness node of the current standard) of the diameter dimension extension line.
ExtensionLine > SlantAngle	Specifies the slant angle for the extension line. This angle is contained between 90 degrees and -90 degrees excluded, the default angle being 0 degree.
ExtensionLine > Left > Hide	Indicates whether or not the left extension line should be hidden.
ExtensionLine > Left > Overrun	Specifies the overrun for the left extension line.
ExtensionLine > Left > Blanking	Specifies the blanking for the left extension line.
ExtensionLine > Right > Hide	Indicates whether or not the right extension line should be hidden.
ExtensionLine > Right > Overrun	Specifies the overrun for the right extension line.
ExtensionLine > Right > Blanking	Specifies the blanking for the right extension line.
ExtensionLine > Funnel > Display	Indicates whether or not the extension line should be displayed as a funnel.
ExtensionLine > Funnel > Height	Specifies the funnel height.
ExtensionLine > Funnel > Width	Specifies the funnel width.

ExtensionLine > Funnel > Angle	Specifies the funnel angle.
ExtensionLine > Funnel > Mode	Specifies the funnel mode (external or internal).
ExtensionLine > Funnel > Side	Specifies whether the funnel should be applied on the left or bottom, on the right or top, or on both sides.
AssociatedTexts > MainValue > Prefix	Specifies the prefix for the main value of the associated text.
AssociatedTexts > MainValue > Suffix	Specifies the suffix for the main value of the associated text.
AssociatedTexts > MainValue > Before	Specifies the text that should be displayed before the main value of the associated text.
AssociatedTexts > MainValue > After	Specifies the text that should be displayed after the main value of the associated text.
AssociatedTexts > MainValue > Upper	Specifies the text that should be displayed above the main value of the associated text.
AssociatedTexts > MainValue > Lower	Specifies the text that should be displayed below the main value of the associated text.
AssociatedTexts > DualValue > Before	Specifies the text that should be displayed before the dual value of the associated text.
AssociatedTexts > DualValue > After	Specifies the text that should be displayed after the dual value of the associated text.
AssociatedTexts > DualValue > Upper	Specifies the text that should be displayed above the dual value of the associated text.
AssociatedTexts > DualValue > Lower	Specifies the text that should be displayed below the dual value of the associated text.
Framing > FramedSubpart	Specifies whether the frame subpart should display the value, the value and tolerance, etc.
Framing > FramedGroup	Specifies whether the framed group should display the main value, the dual value, both values separately, etc.
Framing > Frame	Specifies the type of frame that should be used for diameter dimensions. Note that fixed-size frames are defined in the Frame node of the current standard. Accepted values: none, circle, fixed-size scored circle, fixed-size diamond, fixed-size square, rectangle, oblong, right flag, fixed-size triangle
Framing > MainValueScoring	Specifies the main value scoring.
Framing > DualValueScoring	Specifies the dual value scoring.

Font > Name	Specifies the name of the font that should be used for diameter dimension texts. If no font name is specified, the system's default font will be used.
Font > Bold	Indicates whether or not diameter dimension texts should be displayed in bold.
Font > Italic	Indicates whether or not diameter dimension texts should be displayed in italic.
Font > Size	Indicates the font size that should be used for diameter dimension texts.
Font > Color	Specifies the color that should be used to display diameter dimension texts.
Font > Underline	Indicates whether or not diameter dimension texts should be underlined (in this case, overline cannot be applied).
Font > Strikethrough	Indicates whether or not strikethrough should be used for diameter dimension texts.
Font > Overline	Indicates whether or not diameter dimension texts should be overlined (in this case, underline cannot be applied).
Font > Ratio	Specifies the character width.
Font > Spacing	Specifies the spacing between characters.
Font > ApplyScale	Indicates whether or not diameter dimension texts should be scaled according to the view or the 2D reference component's scale.
OneSymbolDiameter	Indicates whether or not diameter dimensions should be displayed as one-symbol dimensions (as opposed to two-symbols).

ChamferDimension Styles

Parameter Name	Description
Value > OrientationReference	Specifies whether the screen, the view or the dimension line should be used as the reference for the chamfer dimension value orientation.
Value > Angle	Specifies the chamfer dimension value orientation angle according to the chosen reference.
Value > Position	Specifies the chamfer dimension value position.
Value > OffsetX	Specifies the horizontal offset value for chamfer dimension values: • if Value > OrientationReference is set to Screen, the horizontal direction is defined by the X axis of the screen. • if Value > OrientationReference is set to View, the horizontal direction is defined by the X axis of the view. • if Value > OrientationReference is set to Dimension Line, the horizontal direction is parallel to the dimension line.
Value > OffsetY	Specifies the vertical offset value for chamfer dimension values: • if Value > OrientationReference is set to Screen, the horizontal direction is defined by the Y axis of the screen. • if Value > OrientationReference is set to View, the horizontal direction is defined by the Y axis of the view. • if Value > OrientationReference is set to Dimension Line, the horizontal direction is normal to the dimension line.
Value > Display	Choose the display mode you want for the chamfer dimension: - Show value: displays the dimension, its leader and its frame. - Show box: replaces the dimension and its frame by a rectangular box and displays its leader. - Hide value: hides the dimension and its frame but displays its leader.
Driving	Not available for chamfer dimensions.

DualValueDisplay	Specifies whether there will be a dual value display for the chamfer dimension, and, if any, what kind (e.g. fractional, side-by-side, etc.).
ValueDisplayFormat > MainValue > Name	Specifies the name of the main value display format. Make sure that the display format specified here belongs to the list of Value Display styles allowed on dimensions, as defined in the **General > AllowedNumericalFormats** node of the Standards editor.
ValueDisplayFormat > MainValue > DisplayedFactorNumber	Specifies the number of factors displayed for the main value, according to the number of factors available for the value display format used by this style.
ValueDisplayFormat > MainValue > PrecisionMode	Specifies whether the precision mode for the main value will be decimal or fractional.
ValueDisplayFormat > MainValue > Precision	Specifies the precision for the main value.
ValueDisplayFormat > DualValue > Name	Specifies the name of the dual value display format, if any. Make sure that the display format specified here belongs to the list of Value Display styles allowed on dimensions, as defined in the **General > AllowedNumericalFormats** node of the Standards editor.
ValueDisplayFormat > DualValue > DisplayedFactorNumber	Specifies the number of factors displayed for the dual value (if any), according to the number of factors available for the value display format used by this style.
ValueDisplayFormat > DualValue > PrecisionMode	Specifies whether the precision mode for the dual value (if any) will be decimal or fractional.
ValueDisplayFormat > DualValue > Precision	Specifies the precision for the dual value, if any.
Fake > Mode	Indicates whether chamfer dimensions will be fake dimensions, and, if yes, of what type (e.g. numerical or alphanumerical).
Fake > MainValue	Specifies the fake main value for chamfer dimensions.
Fake > DualValue	Specifies the fake dual value for chamfer dimensions.
Tolerance > MainValue > Format	Specifies the tolerance main value format for chamfer dimensions.

Tolerance > MainValue > AlphanumericalValue1	Specifies the first alphanumerical value for the tolerance main value.
Tolerance > MainValue > AlphanumericalValue2	Specifies the second alphanumerical value for the tolerance main value.
Tolerance > MainValue > NumericalLower	Specifies the lower numerical for the tolerance main value.
Tolerance > MainValue > NumericalUpper	Specifies the upper numerical for the tolerance main value.
Tolerance > DualValue > Format	Specifies the tolerance dual value format for chamfer dimensions.
Tolerance > DualValue > AlphanumericalValue1	Specifies the first alphanumerical value for the tolerance dual value.
Tolerance > DualValue > AlphanumericalValue2	Specifies the second alphanumerical value for the tolerance dual value.
Tolerance > DualValue > NumericalLower	Specifies the lower numerical for the tolerance dual value.
Tolerance > DualValue > NumericalUpper	Specifies the upper numerical for the tolerance dual value.
DimensionLine > Representation	Specifies how the dimension line should be represented (e.g. regular, one-part leader, etc.)
DimensionLine > Color	Specifies the color that should be used to display dimension lines.
DimensionLine > Thickness	Specifies the line thickness index (as defined in the LineThickness node of the current standard) that should be used to represent dimension lines.
DimensionLine > SecondPartReference	In the case of Two parts or Leader two parts for the representation, specifies the Reference for positioning the second part of the dimension line.
DimensionLine > SecondPartAngle	In the case of Two parts or Leader two parts for the representation, specifies the angle for the second part of the dimension line in relation to its reference.
DimensionLine > LeaderAngle	Specifies the angle for the dimension line leader.
Symbols > Symbol1 > Type	Specifies the type of the first symbol (e.g. arrow, filled circle, etc.) that should be used for chamfer dimensions. If you choose the Automatic option, simple arrow will be used by default.
Symbols > Symbol1 > Color	Specifies the color of the first symbol.

Symbols > Symbol1 > Thickness	Specifies the line thickness index (as defined in the LineThickness node of the current standard) of the first symbol.
Symbols > Symbol2 > Type	Specifies the type of the second symbol (e.g. arrow, filled circle, etc.) that should be used for chamfer dimensions. If you choose the Automatic option, simple arrow will be used by default.
Symbols > Symbol2 > Color	Specifies the color of the second symbol.
Symbols > Symbol2 > Thickness	Specifies the line thickness index (as defined in the LineThickness node of the current standard) of the second symbol.
Symbols > SymbolMode	Specifies the symbol mode (e.g. inside, outside, etc.).
ExtensionLine > Color	Specifies the color of the chamfer dimension extension line.
ExtensionLine > Thickness	Specifies the line thickness index (as defined in the LineThickness node of the current standard) of the chamfer dimension extension line.
ExtensionLine > SlantAngle	Specifies the slant angle for the extension line. This angle is contained between 90 degrees and -90 degrees excluded, the default angle being 0 degree.
ExtensionLine > Left > Hide	Indicates whether or not the left extension line should be hidden.
ExtensionLine > Left > Overrun	Specifies the overrun for the left extension line.
ExtensionLine > Left > Blanking	Specifies the blanking for the left extension line.
ExtensionLine > Right > Hide	Indicates whether or not the right extension line should be hidden.
ExtensionLine > Right > Overrun	Specifies the overrun for the right extension line.
ExtensionLine > Right > Blanking	Specifies the blanking for the right extension line.
ExtensionLine > Funnel > Display	Indicates whether or not the extension line should be displayed as a funnel.
ExtensionLine > Funnel > Height	Specifies the funnel height.
ExtensionLine > Funnel > Width	Specifies the funnel width.
ExtensionLine > Funnel > Angle	Specifies the funnel angle.
ExtensionLine > Funnel > Mode	Specifies the funnel mode (external or internal).

ExtensionLine > Funnel > Side	Specifies whether the funnel should be applied on the left or bottom, on the right or top, or on both sides.
AssociatedTexts > MainValue > Prefix	Specifies the prefix for the main value of the associated text.
AssociatedTexts > MainValue > Suffix	Specifies the suffix for the main value of the associated text.
AssociatedTexts > MainValue > Before	Specifies the text that should be displayed before the main value of the associated text.
AssociatedTexts > MainValue > After	Specifies the text that should be displayed after the main value of the associated text.
AssociatedTexts > MainValue > Upper	Specifies the text that should be displayed above the main value of the associated text.
AssociatedTexts > MainValue > Lower	Specifies the text that should be displayed below the main value of the associated text.
AssociatedTexts > DualValue > Before	Specifies the text that should be displayed before the dual value of the associated text.
AssociatedTexts > DualValue > After	Specifies the text that should be displayed after the dual value of the associated text.
AssociatedTexts > DualValue > Upper	Specifies the text that should be displayed above the dual value of the associated text.
AssociatedTexts > DualValue > Lower	Specifies the text that should be displayed below the dual value of the associated text.
Framing > FramedSubpart	Specifies whether the frame subpart should display the value, the value and tolerance, etc.
Framing > FramedGroup	Specifies whether the framed group should display the main value, the dual value, both values separately, etc.
Framing > Frame	Specifies the type of frame that should be used for chamfer dimensions. Note that fixed-size frames are defined in the Frame node of the current standard. Accepted values: none, circle, fixed-size scored circle, fixed-size diamond, fixed-size square, rectangle, oblong, right flag, fixed-size triangle
Framing > MainValueScoring	Specifies the main value scoring.
Framing > DualValueScoring	Specifies the dual value scoring.

Font > Name	Specifies the name of the font that should be used for chamfer dimension texts. If no font name is specified, the system's default font will be used.
Font > Bold	Indicates whether or not chamfer dimension texts should be displayed in bold.
Font > Italic	Indicates whether or not chamfer dimension texts should be displayed in italic.
Font > Size	Indicates the font size that should be used for chamfer dimension texts.
Font > Color	Specifies the color that should be used to display chamfer dimension texts.
Font > Underline	Indicates whether or not chamfer dimension texts should be underlined (in this case, overline cannot be applied).
Font > Strikethrough	Indicates whether or not strikethrough should be used for chamfer dimension texts.
Font > Overline	Indicates whether or not chamfer dimension texts should be overlined (in this case, underline cannot be applied).
Font > Ratio	Specifies the character width.
Font > Spacing	Specifies the spacing between characters.
Font > ApplyScale	Indicates whether or not chamfer dimension texts should be scaled according to the view or the 2D reference component's scale.
Chamfer > DisplayType	Specifies whether chamfer dimensions should be displayed with one extension line or with two extension lines.
Chamfer > ValueFormat	Specifies the value format for the chamfer dimension (e.g. length/length, angle/length, etc.)
Chamfer > SecondaryValueDisplayFormat > MainValue > Name	Specifies the name of the secondary value display format for the main value.
Chamfer > SecondaryValueDisplayFormat > MainValue > DisplayedFactorNumber	Specifies the number of factors displayed for the main value, according to the number of factors available for the value display format used by this style.
Chamfer > SecondaryValueDisplayFormat > DualValue > Name	Specifies the name of the secondary value display format for the dual value.

Chamfer > SecondaryValueDisplayFormat > DualValue > DisplayedFactorNumber	Specifies the number of factors displayed for the dual value (if any), according to the number of factors available for the value display format used by this style.

CoordinateDimension Styles

Parameter Name	Description
OrientationReference	Specifies whether the screen, the view or the dimension line should be used as the reference for the coordinate dimension orientation.
Angle	Specifies the coordinate dimension orientation angle according to the chosen reference.
AnchorPoint	Specifies the coordinate dimension position in relation to the anchor point (e.g., top left, middle left, etc.).
Display	Choose the display mode you want for the coordinate dimension: - Show value: displays the dimension, its leader and its frame. - Show box: replaces the dimension and its frame by a rectangular box and displays its leader. - Hide value: hides the dimension and its frame but displays its leader.
Font > Name	Specifies the name of the font that should be used for coordinate dimension texts. If no font name is specified, the system's default font will be used.
Font > Bold	Indicates whether or not coordinate dimension texts should be displayed in bold.
Font > Italic	Indicates whether or not coordinate dimension texts should be displayed in italic.
Font > Size	Indicates the font size that should be used for coordinate dimension texts.
Font > ApplyScale	Indicates whether or not coordinate dimension texts should be scaled according to the view or the 2D reference component's scale.
Font > Underline	Indicates whether or not coordinate dimension texts should be underlined (in this case, overline cannot be applied).
Font > Strikethrough	Indicates whether or not strikethrough should be used for coordinate dimension texts.
Font > Overline	Indicates whether or not coordinate dimension texts should be overlined (in this case, underline cannot be applied).
Graphic > Color	Specifies the color that should be used to represent coordinate dimension texts frames and leaders.

Leader > Symbol	Specifies the symbol (e.g. arrow, filled circle, etc.) that should be used for coordinate dimension leaders.
	
	If you choose the Automatic option, a default symbol will be used, depending on the standard type and on whether the leader is associated to an element or not:
	
	• If the leader is associated to an element:
	  ○ Symmetric arrow (a.k.a. Unfilled arrow) for ANSI / ASME
	  ○ Simple arrow for ISO / JIS
	
	• If the leader is not associated to an element:
	  ○ Symmetric circle (a.k.a. Unfilled circle) for ANSI / ASME
	  ○ Filled circle for ISO / JIS

DisplayUnit	Indicates whether or not units for coordinate dimension texts should be displayed.

Dress-up and Dress-up Symbols Styles

This section deals with dress-up and dress-up symbols styles. These let you define the default values that will be used when applying dress-up to elements.

Defining Dress-up and Dress-up Symbols Styles

Dress-up and dress-up symbols styles are located in the following nodes of the Standards editor, available via **Tools -> Standards:**

- Styles -> AreaFill
- Styles -> AxisLine
- Styles -> CenterLine
- Styles -> Thread
- Styles -> Arrow

By default, a style called Default is available for each dress-up/dress-up symbol style.

All the parameters associated to a given dress-up or dress-up symbol style are listed in a dedicated table. The Description column provides a description of each parameter.

All parameters are taken into account both at creation time (i.e. when creating the dress-up element or dress-up symbol), and at modification time (i.e. when reapplying a style to a dress-up element or dress-up symbol).

Area Fill Style

Parameter Name	Description
Pattern	Specifies the name of the pattern (as defined in the Patterns node of the current standard) that should be used for area fills.

Axis Line Style

Parameter Name	Description
Graphic > Color	Specifies the color that should be used to represent axis lines.
Graphic > LineType	Specifies the number of the linetype (as defined in the LineTypes node of the current standard) that should be used to represent axis lines.
Graphic > Thickness	Specifies the line thickness index (as defined in the LineThickness node of the current standard) that should be used to represent axis lines.
OverRunAuto	Indicates whether or not the overrun between the element and its axis line is computed automatically. When set to Yes, this parameter overrides any value set for OverRunLength, and the overrun makes up 10% of the axis length.
OverRunLength	When OverRunAuto is set to No, specifies the length of the overrun between the element and its axis line.

Center Line Style

Parameter Name	Description
Graphic > Color	Specifies the color that should be used to represent center lines.
Graphic > LineType	Specifies the number of the linetype (as defined in the LineTypes node of the current standard) that should be used to represent center lines.
Graphic > Thickness	Specifies the line thickness index (as defined in the LineThickness node of the current standard) that should be used to represent center lines.
OverRunAuto	Indicates whether or not the overrun between the element and its center line is computed automatically. When set to Yes, this parameter overrides any value set for OverRunLength, and the overrun makes up 30% of the radius.
OverRunLength	When OverRunAuto is set to No, specifies the length of the overrun between the element and its center line.

Thread Style

Parameter Name	Description
Graphic > Color	Specifies the color that should be used to represent threads.
Graphic > LineType	Specifies the number of the linetype (as defined in the LineTypes node of the current standard) that should be used to represent threads.
Graphic > Thickness	Specifies the line thickness index (as defined in the LineThickness node of the current standard) that should be used to represent threads.
OverRunAuto	Indicates whether or not the overrun between the element and its thread is computed automatically. When set to Yes, this parameter overrides any value set for OverRunLength, and the overrun makes up 30% of the radius.
OverRunLength	When OverRunAuto is set to No, specifies the length of the overrun between the element and its thread.

Thread styles only apply to threads viewed along their axis, whether in Interactive or Generative views. As a result, thread styles do not apply to such views as section views for example.

Arrow Style

Parameter Name	Description
Graphic > Color	Specifies the color that should be used to represent arrows.
Graphic > LineType	Specifies the number of the linetype (as defined in the LineTypes node of the current standard) that should be used to represent arrows.
Graphic > Thickness	Specifies the line thickness index (as defined in the LineThickness node of the current standard) that should be used to represent arrows.
HeadSymbol	Specifies the symbol (e.g., simple arrow, circle, etc.) that should be used for arrow heads. If you choose the Automatic option, a default symbol will be used, depending on the standard type: • Filled arrow for ANSI / ASME • Open arrow for ISO / JIS
TailSymbol	Specifies the symbol (e.g., simple arrow, circle, etc.) that should be used for arrow tails. If you choose the Automatic option, by default, no symbol will be used.

View Callout Styles

This section deals with view callout styles. These let you define the default values that will be used for projection, section or detail view callouts.

Defining View Callout Styles

View callout styles are located in the following nodes of the Standards editor, available via **Tools -> Standards:**

- Styles -> ProjectionCallout
- Styles -> SectionCallout
- Styles -> DetailCallout

By default, a style called Default is available for each view callout style.

ProjectionCallout Styles

Parameter Name	Description
ViewScaleDependant	Indicates whether or not projection view callouts should be dependent on the view scale.
Type	Indicates the type of callout (e.g., lines and arrows, lines, corners and arrows, etc.) that should be used to represent section view callouts.
Profile > LineType	Specifies the number of the linetype (as defined in the LineTypes node of the current standard) that should be used to represent projection view callout profiles. Available values range from 1 to 7.
Profile > Thickness	Specifies the line thickness index (as defined in the LineThickness node of the current standard) that should be used to represent projection view callout profiles. Available values range from 1 to 8.
Corners > Thickness	Specifies the line thickness index (as defined in the LineThickness node of the current standard) that should be used to represent projection view callout corners. Available values range from 1 to 8.
Corners > Length	Specifies the length of projection view callout corners. Available values range from 10E-3 to 10E3mm.
Attachment	Indicates whether callout arrows are attached by the head or the tail of projection view callout arrows.
Arrows > Length	Specifies the length of projection view callout arrows. Available values range from 10E-3 to 10E3mm.

Arrows > Head > Angle	Specifies the angle used for projection view callout arrow heads. Available values range from 5deg to 175deg.
Arrows > Head > Type	Specifies the type used for projection view callout arrow heads (e.g. filled arrow, blanked arrow, closed arrow or simple arrow).
Arrows > Head > Length	Specifies the length of projection view callout arrow heads. Available values range from 10E-3 to 10E3mm.
Color	Specifies the color that should be used to display projection view callouts.
Text > Font > Name	Specifies the name of the font that should be used for projection view callouts.
Text > Font > Bold	Indicates whether or not projection view callouts texts should be displayed in bold.
Text > Font > Italic	Indicates whether or not projection view callouts texts should be displayed in italic.
Text > Font > Size	Indicates the font size that should be used for projection view callouts texts.
Text > Font > Underline	Indicates whether or not projection view callouts texts should be underlined.
Text > Font > Color	Specifies the color that should be used to display projection view callouts texts.
Text > Font > Ratio	Specifies the ratio that should be used to display projection view callouts texts.
Text > Font > Slant	Specifies the slant that should be used to display projection view callouts texts.
Text > Font > Spacing	Specifies the spacing that should be used to display projection view callouts texts.
Text > Font > Pitch	Specifies the pitch (fixed or variable) that should be used to display projection view callouts texts.
Text > Font > Strikethrough	Indicates whether or not strikethrough should be used for projection view callouts texts.
Text > Font > Overline	Indicates whether or not projection view callouts texts should be overlined.
Text > Frame	Specifies the type of frame (e.g., rectangle, square, fixed-size rectangle, fixed-size square, etc.) that should be used to represent projection view callouts. Note that fixed-size frames are defined in the Frame node of the current standard.

SectionCallout Styles

Parameter Name	Description
ViewScaleDependant	Indicates whether or not section view callouts should be dependent on the view scale.
Type	Indicates the type of callout (e.g., lines and arrows, lines, corners and arrows, etc.) that should be used to represent section view callouts.
Profile > LineType	Specifies the number of the linetype (as defined in the LineTypes node of the current standard) that should be used to represent section view callout profiles. Available values range from 1 to 7.
Profile > Thickness	Specifies the line thickness index (as defined in the LineThickness node of the current standard) that should be used to represent section view callout profiles. Available values range from 1 to 8.
Corners > Thickness	Specifies the line thickness index (as defined in the LineThickness node of the current standard) that should be used to represent section view callout corners. Available values range from 1 to 8.
Corners > Length	Specifies the length of section view callout corners. Available values range from 10E-3 to 10E3mm.
Attachment	Indicates whether callout arrows are attached by the head or the tail of section view callout arrows.
Arrows > Length	Specifies the length of section view callout arrows. Available values range from 10E-3 to 10E3mm.
Arrows > Head > Angle	Specifies the angle used for section view callout arrow heads. Available values range from 5deg to 175deg.
Arrows > Head > Type	Specifies the type used for section view callout arrow heads (e.g. filled arrow, blanked arrow, closed arrow or simple arrow).
Arrows > Head > Length	Specifies the length of section view callout arrow heads. Available values range from 10E-3 to 10E3mm.
Color	Specifies the color that should be used to display section view callouts.
Text > Font > Name	Specifies the name of the font that should be used for section view callouts.
Text > Font > Bold	Indicates whether or not section view callouts texts should be displayed in bold.
Text > Font > Italic	Indicates whether or not section view callouts texts should be displayed in italic.
Text > Font > Size	Indicates the font size that should be used for section view callouts texts.
Text > Font > Underline	Indicates whether or not section view callouts texts should be underlined.

Text > Font > Color	Specifies the color that should be used to display section view callouts texts.
Text > Font > Ratio	Specifies the ratio that should be used to display section view callouts texts.
Text > Font > Slant	Specifies the slant that should be used to display section view callouts texts.
Text > Font > Spacing	Specifies the spacing that should be used to display section view callouts texts.
Text > Font > Pitch	Specifies the pitch (fixed or variable) that should be used to display section view callouts texts.
Text > Font > Strikethrough	Indicates whether or not strikethrough should be used for section view callouts texts.
Text > Font > Overline	Indicates whether or not section view callouts texts should be overlined.
Text > Frame	Specifies the type of frame (e.g., rectangle, square, fixed-size rectangle, fixed-size square, etc.) that should be used to represent section view callouts. Note that fixed-size frames are defined in the Frame node of the current standard.

DetailCallout Styles

Parameter Name	Description
Type	Indicates the type of callout (e.g., leader text, circle, etc.) that should be used to represent detail view callouts.
LineType	Specifies the number of the linetype (as defined in the LineTypes node of the current standard) that should be used to represent detail view callouts. Available values range from 1 to 7.
Thickness	Specifies the line thickness index (as defined in the LineThickness node of the current standard) that should be used to represent detail view callouts. Available values range from 1 to 8.
Arrows > Head > [all parameters]	**NOT IMPLEMENTED YET**
Text > Font > Name	Specifies the name of the font that should be used for detail view callouts.
Text > Font > Bold	Indicates whether or not detail view callouts texts should be displayed in bold.
Text > Font > Italic	Indicates whether or not detail view callouts texts should be displayed in italic.

Text > Font > Size	Indicates the font size that should be used for detail view callouts texts.
Text > Font > Underline	Indicates whether or not detail view callouts texts should be underlined.
Text > Font > Color	Specifies the color that should be used to display detail view callouts texts.
Text > Font > Ratio	Specifies the ratio that should be used to display detail view callouts texts.
Text > Font > Slant	Specifies the slant that should be used to display detail view callouts texts.
Text > Font > Spacing	Specifies the spacing that should be used to display detail view callouts texts.
Text > Font > Pitch	Specifies the pitch (fixed or variable) that should be used to display detail view callouts texts.
Text > Font > Strikethrough	Indicates whether or not strikethrough should be used for detail view callouts texts.
Text > Font > Overline	Indicates whether or not detail view callouts texts should be overlined.
Text > Frame	Specifies the type of frame (e.g., rectangle, square, fixed-size rectangle, fixed-size square, etc.) that should be used to represent detail view callouts. Note that fixed-size frames are defined in the Frame node of the current standard.

Sheet Styles

This section deals with sheet styles. These let you define the default values that will be used when creating sheets.

Defining Sheet Styles

Sheet styles are located in the **Styles -> Sheet** node of the Standards editor, available via **Tools -> Standards**.

By default, a sheet style called Default is available.

All the parameters associated to a given sheet style are listed in the table below. The Description column provides a description of each parameter.

Parameter Name	Description
GlobalScale	Real number that specifies the global scale that should be applied to the sheet. For example, if you want a global scale of 1:2, you should enter *0.5* and if you want a global scale of 1:1, you should enter *1*.
ProjectionMethod	Specifies whether projection views should be created using the first angle standard, or the third angle standard. Choose a projection method from the list.
Format	Specifies the sheet format defined in the Standard editor and the corresponding paper size that should be applied to the sheet.
Orientation	Specifies the orientation that should be applied to the sheet, that is portrait or landscape.
DisplayFormat	Specifies whether the frame representing the format of the sheet is displayed.
GenViewsPosMode	Specifies whether generative views should be positioned according to the center of gravity of the 3D geometry (which ensures that the center of gravity of the 3D geometry remains at a fixed position at update), or according to the 3D axis system (which ensures that the projection of the 3D axis remains at a fixed position at update, even if the center of gravity of the 3D geometry has changed).

Dimension System Styles

This section deals with dimension system styles. These let you define the default values that will be used when creating different types of dimension systems.

Defining Dimension System Styles

Dimension system styles are located in the following nodes of the Standards editor, available via **Tools -> Standards:**

- Styles -> Dimensions System

By default, a style called Default is available for each geometry style.

All parameters are taken into account both at creation time (i.e. when creating a dimension system), and at modification time (i.e. when reapplying a style to a dimension system).

Dimensions System Styles

Parameter Name	Description
Offset mode	Specifies the dimension lines alignment mode for stacked dimension systems: • Constant: the offset between dimensions of a system remain constant and equal to the value defined in the **Offset between dimensions** field. • Free: the dimensions of a system can be moved independently.
Offset between dimensions	Specifies the distance between two consecutive dimension lines for stacked dimension systems.
Aligned cumulated dimension values	Specifies the dimension values alignment mode for cumulated dimension systems: • Reference line (value horizontal justification is edge) • Center (value horizontal justification is center) • Opposite (value horizontal justification is edge) • From standard
Values Offset	Specifies the distance between the alignment reference and the dimension value for cumulated dimension systems.

Aligned stacked dimension values	Specifies the dimension values alignment mode for stacked dimension systems: • Reference line • Center • Opposite
Values Offset	Specifies the distance between the alignment reference and the dimension value for stacked dimension systems.

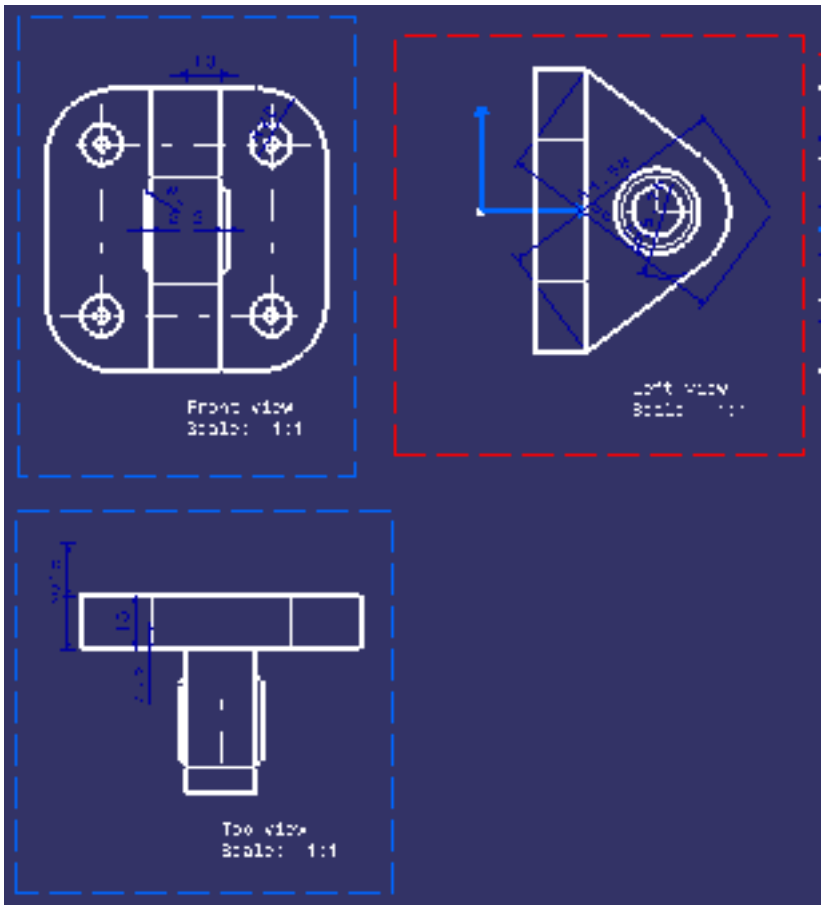
Glossary

9 B D F I L M P S U V

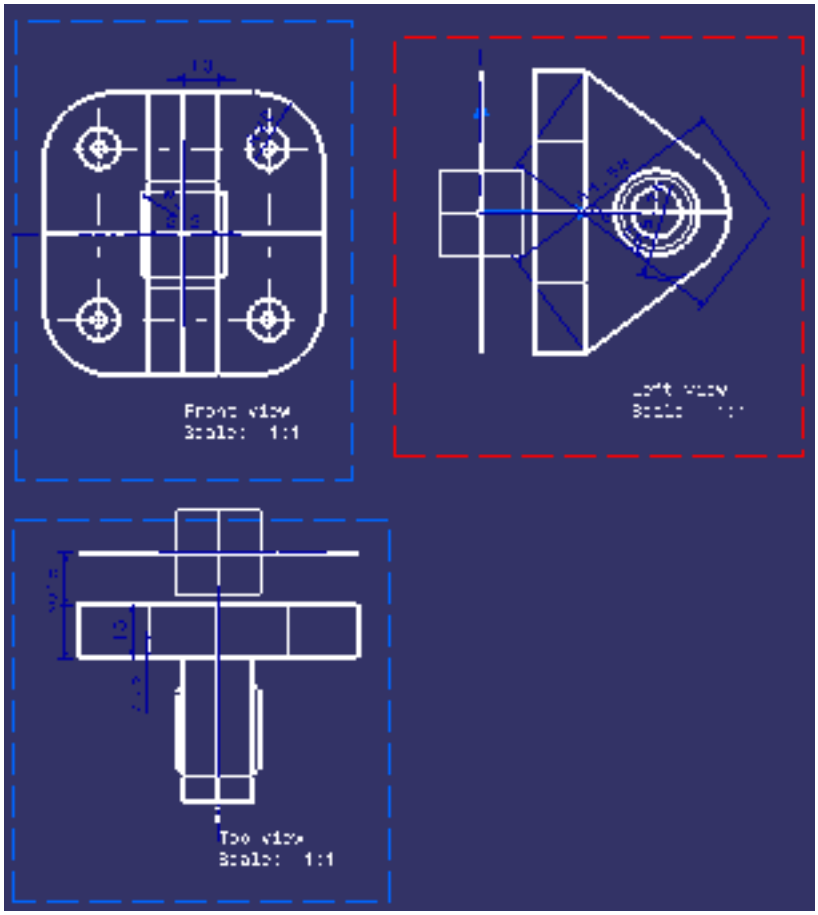
Numerics

2D background The visualization - in transparency - in the 2D window (through a design view) of the 3D representation of 2D elements which do not belong to the current view, but to other views.

Below is an example of views with no 2D background:



Below is an example of the same views with a 2D background:

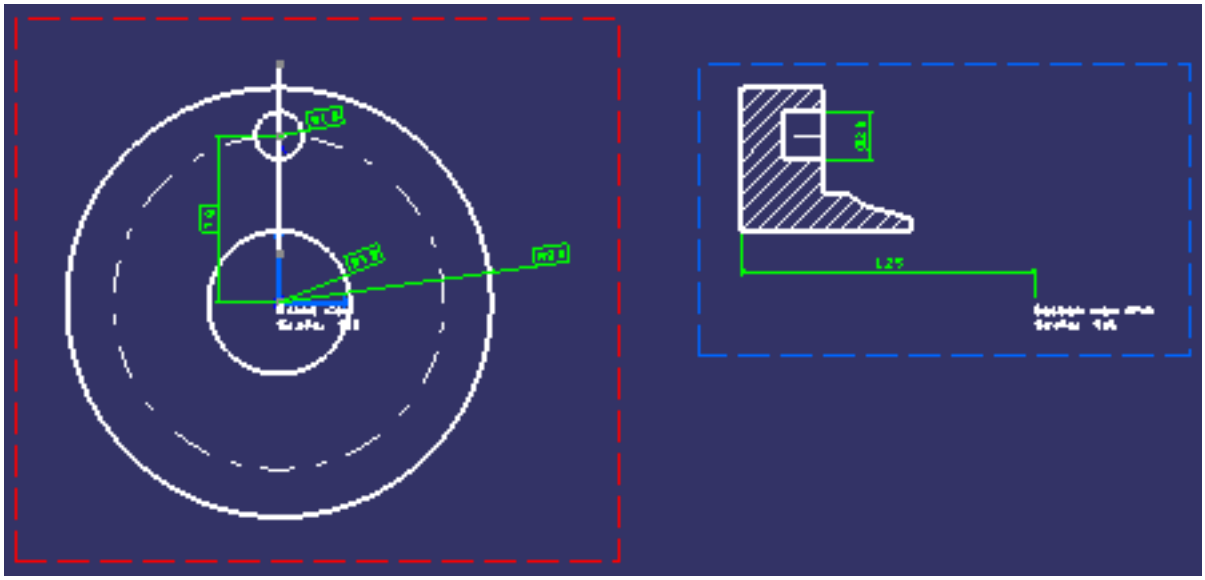


2D component A re-usable set of geometry and annotations, stored in a detail sheet or a catalog. A 2D component can be instantiated in any view.

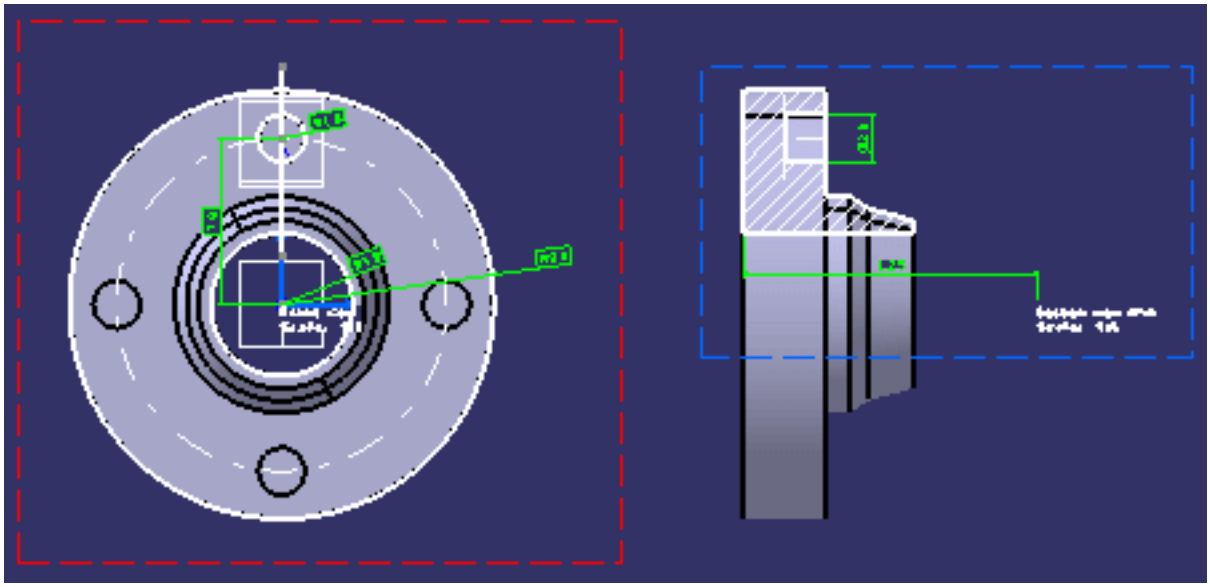
2D window A standard window with a specification tree and a 3D viewer that cannot be rotated. It is in this window that you create your layout, that is the views containing 2D geometry, dimensions, annotations and so on.

3D background The visualization - in transparency - in the 2D window (through a design view or an isometric view) of all 3D elements. This includes edges, faces and 3D wireframe.

Below is an example of views with no 3D background:



Below is an example of the same views with a 3D background:



3D window A standard window with a specification tree and a 3D viewer that can be rotated. This window contains the 3D representation of a layout.

3D workbench A workbench dedicated to the 3D edition of part or product documents, such as the Part Design or Product Structure workbench.

B

background view In a sheet, the background view is dedicated to frames and title blocks. See also main view.

D

design view A view in which you design and whose content can be visualized and output in 3D context. Design views have a specific position in 3D space. The 2D background and the 3D background can be seen through design views.

Design views can be of the following types: Front, Left, Right, Rear, Top, Bottom, Auxiliary, Section and Section Cut. See also Isometric view.

detail sheet A sheet that is used as an intermediate catalog for positioning 2D reference elements (geometry) that will be instantiated afterwards. A layout detail sheet contains a main view and a background view.

display filter A view filter which defines a list of elements to display and possibly overload while the remainder of the background is hidden. See also Mask filter.

dress-up view A view visualized only in the 2D Layout for 3D Design workbench. No background (whether 2D or 3D) can be seen through such a view.

Dress-up views can be of the following types: main, background and views in detail sheets.

F

filter

See View filter.

I

isometric view

A type of view visualized only in the 2D Layout for 3D Design workbench (not in a 3D window). However, the 3D background (as well as the 2D background) can be seen through isometric views.

Note that isometric views are not design views. For this reason, a number of functionalities available for design views (such as creating use-edges, 3D profiles or 3D planes) are not available for isometric views. See also Design view.

L

layout

A layout is represented in 2 different ways:

- a 3D visualization, based on the plane definition associated to each layout design view. This visualization is available in 3D workbenches, and displayed in the 3D window (the 3D document edition window).
- a 2D visualization, representing the model from different viewpoints (called layout views), positioned within a sheet, with respect to a standardized projection method (first or third angle projection). This visualization is available in the 2D Layout for 3D Design workbench, and displayed in the 2D window.

M

main view

In a sheet, the main view (also called working view) supports the geometry directly created in the sheet. See also background view.

mask filter

A view filter which defines a list of elements to overload while the remainder of the background is displayed in the same way as defined in 3D. See also Display filter.

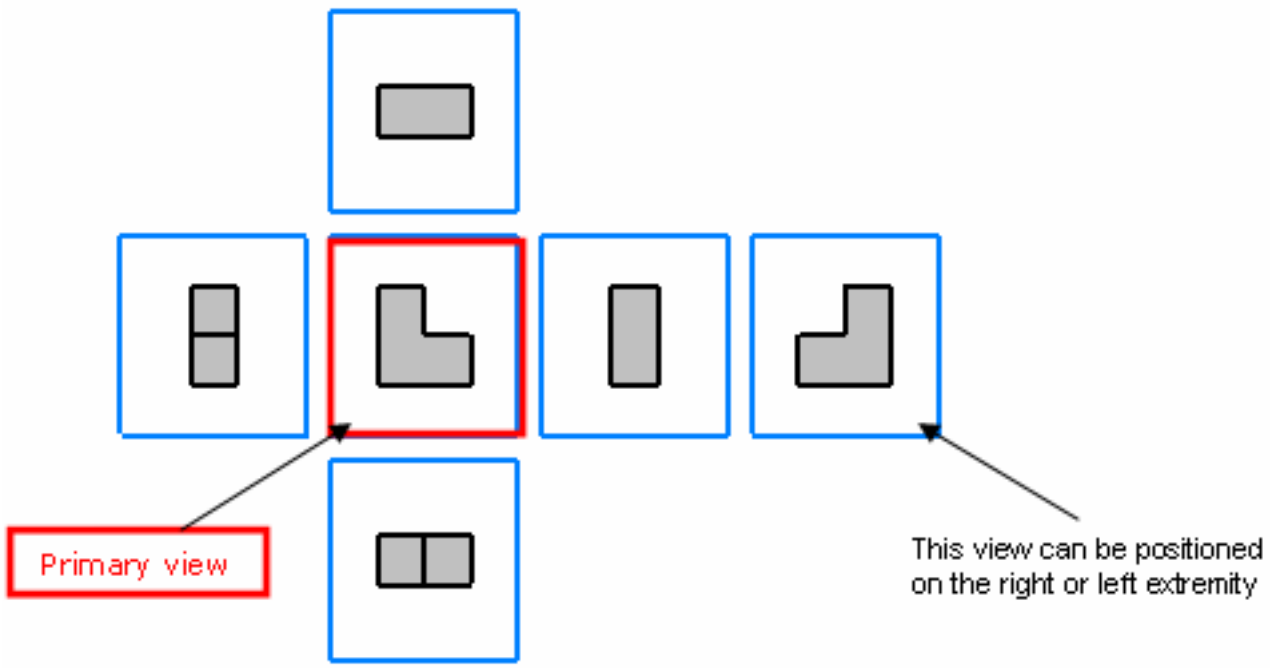
P

part layout

A layout which is stored in a part document.

primary view

A primary view is the first basic view chosen to describe an object because it best represents the shape of this object. The primary view also determines the projection views' position in the sheet.



projection view

A view conceived to be drawn or projected onto planes known as projection planes.

S

sheet

A set of views. In the 2D Layout for 3D Design workbench, a sheet contains a main view and a background view. It can also contain design views and isometric views.

U

use-edge

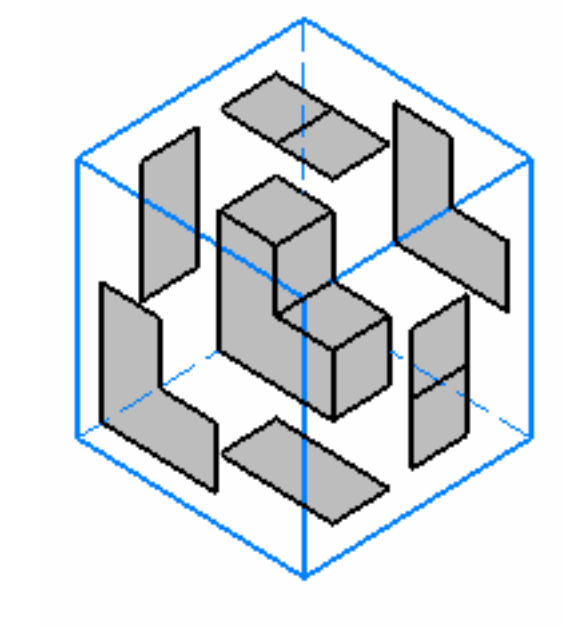
A datum element that results from one of the following operations: projection of a 3D element, intersection of a 3D element or projection of 3D silhouette edges. Note that use-edges in the 2D Layout for 3D Design workbench are not associative to the reference element.

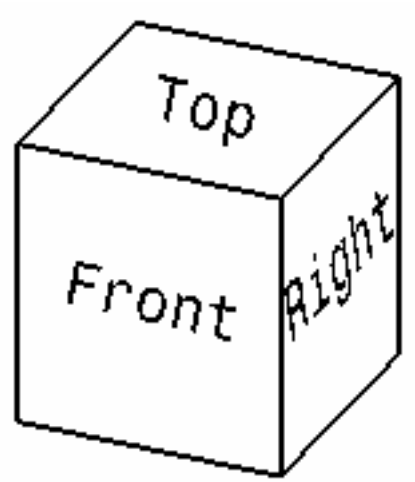
V

view box

In the 3D space, the projection views of a layout can be seen as a box (or a cube) containing the part to be designed. This cube is made of basic views: Front, Left, Right, Top, Bottom and Rear.

A view box gathers all the data needed to fully define the layout of a view set in the 2D window, as well as the position of each view in the 3D space.





view content All 2D elements aggregated under a given view.

view filter View filters modify the visualization of some elements that are represented in the background of layout views. They allow users to overload global graphic properties, thus making some background elements invisible, unpickable or low-intensified. See also Display filter and Mask filter.

view plane definition Corresponds to the specific position of design views in the 3D space.

view set Corresponds to the set of views which have been created from a primary view, that is the primary view, its related basic views, as well as any isometric view, section view, section cut and/or auxiliary view created from any view belonging to the view set.

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