Getting Started

Before getting into the detailed instructions for using **Generative Drafting**, 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. The main tasks covered in this section are the following:

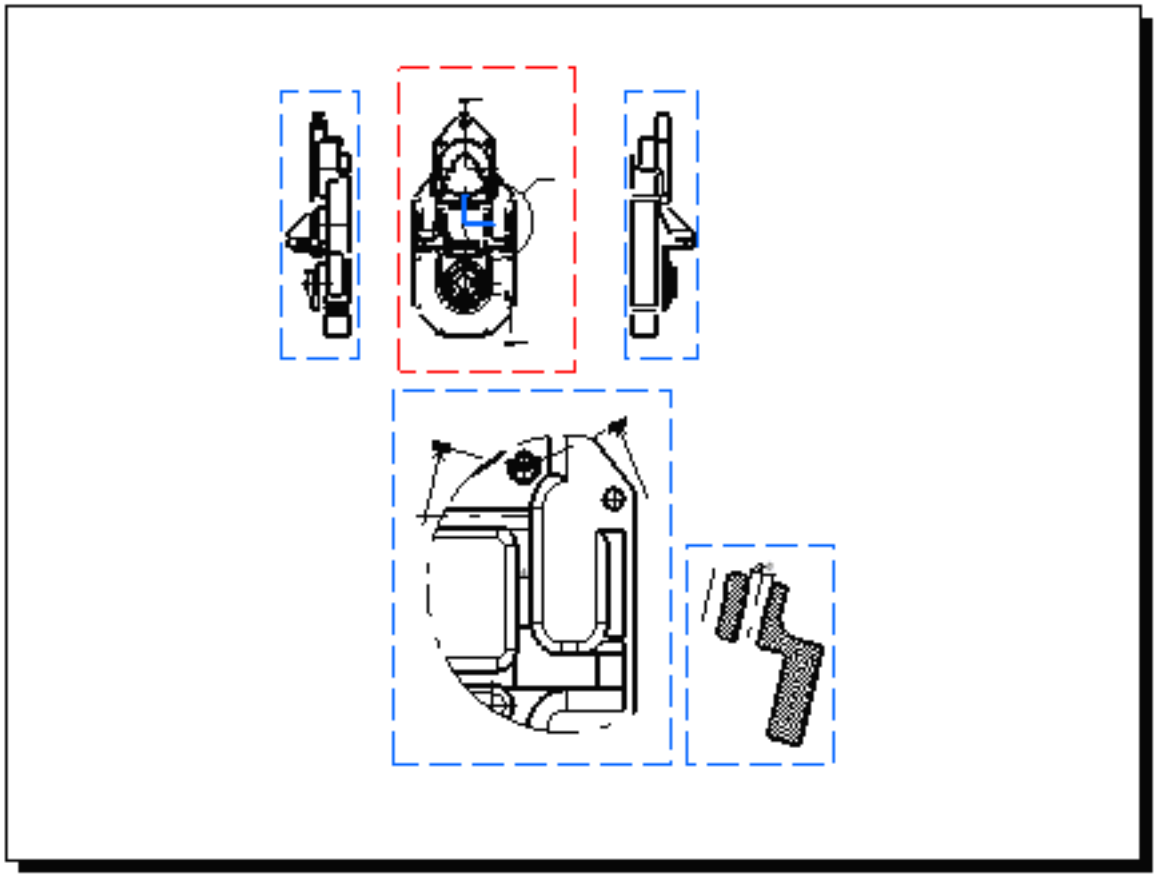
Before You Begin, make sure you customized the following settings:

Grid:
De-activate the Grid icon  from the Tools toolbar (bottom right).

View names and scaling factors:
Go to `Tools->Options` (`Mechanical Design -> Drafting option` at the left of the dialog box, `Layout` tab) and un-check the View name and Scaling factor options.

This step-by-step scenario introduces the basic capabilities of **Generative Drafting**. You just need to follow the instructions as you progress along.

At the end of this getting started, you will be able to print the following sheet:

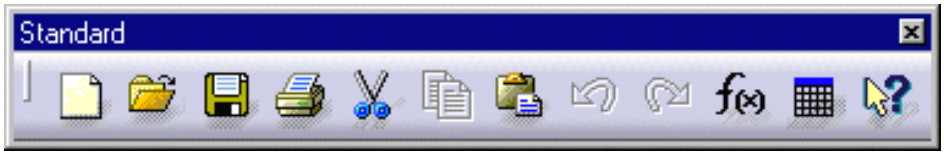


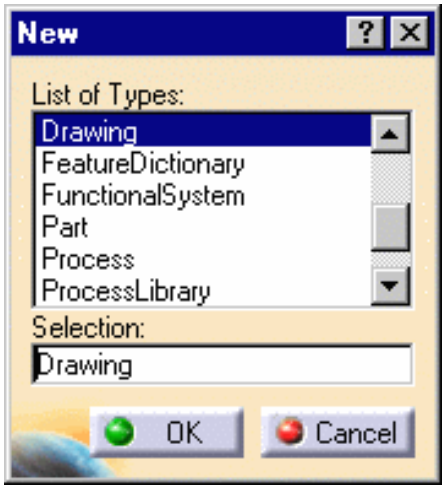
Before discovering this scenario, you should be familiar with the basic commands common to all workbenches. These are described in the *Infrastructure User's Guide*.

Defining the Drawing Sheet

This task shows you how to define the drawing sheet to be used for creating the views described in further tasks.

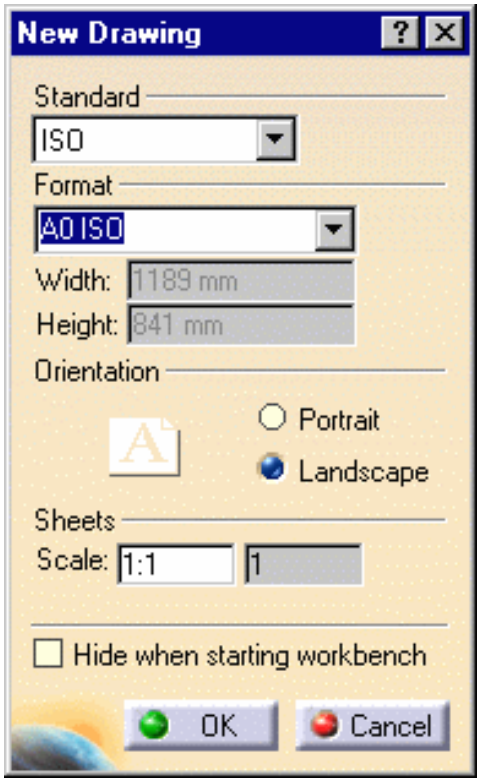
1. Click the New icon from the Standard toolbar or select `File` -> `New...` from the menu bar.





2. Select `Drawing` workbench from the `List of Types` and click `OK`.
3. From the `New Drawing` dialog box, select the `ISO` standard, and the `A0 ISO` format.

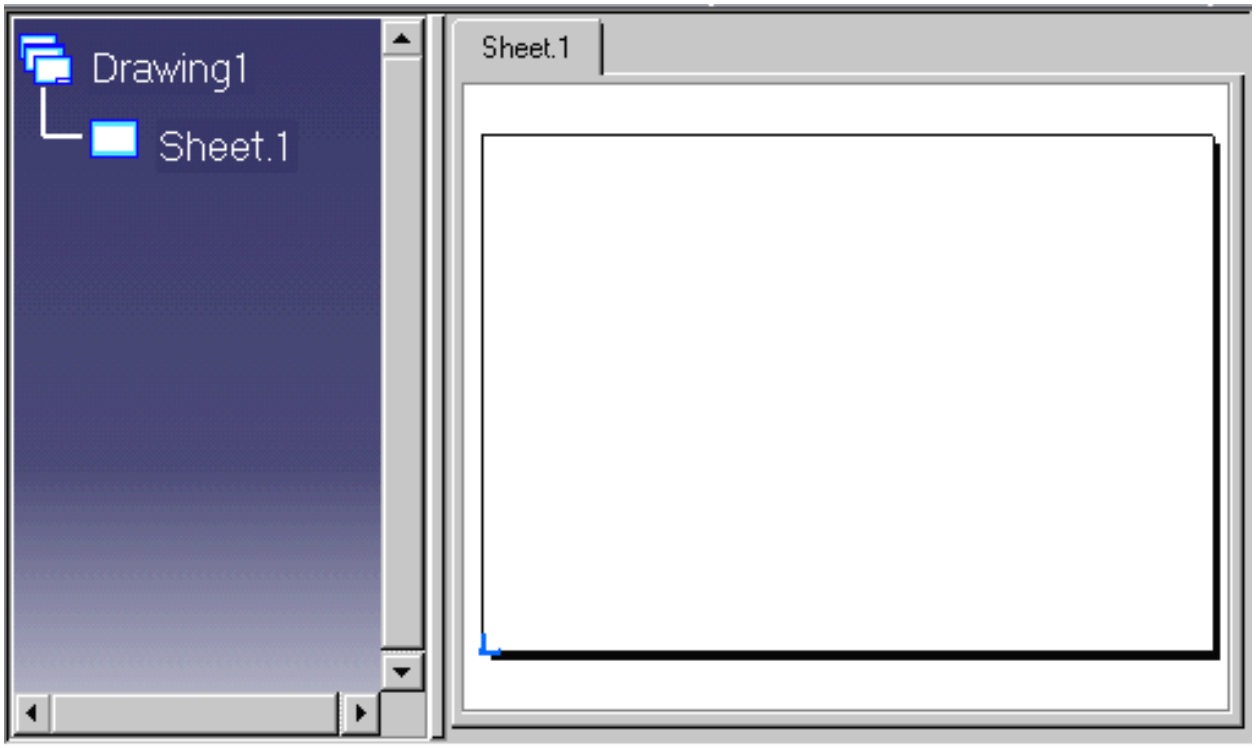
   In this particular case, and all along the guide, we use the `ISO` standard.



You can add an unlimited number of customized standards using Standard files that you will create and/or, if needed, modify. Once created, this standard will appear in the New Drawing dialog box. For more details on standards, see the Standards Administration section. Care that any user-defined standard is based on one of the four international standards (ANSI, ISO, ASME or JIS) as far as basic parameters are concerned.

4. In the **Generative Drafting** workbench, a grid is set by default. Throughout this documentation, we decided not to display the grid. To do this, de-activate the Grid icon  from the Tools toolbar (bottom right).

   The drawing sheet appears as shown here:



From now on, you will work on the created sheet unless you define a new sheet.

Opening a Part

This task will show you how to open the part to be used in the **Generative Drafting** workbench to create views. You may use either a 3D part or an assembly.

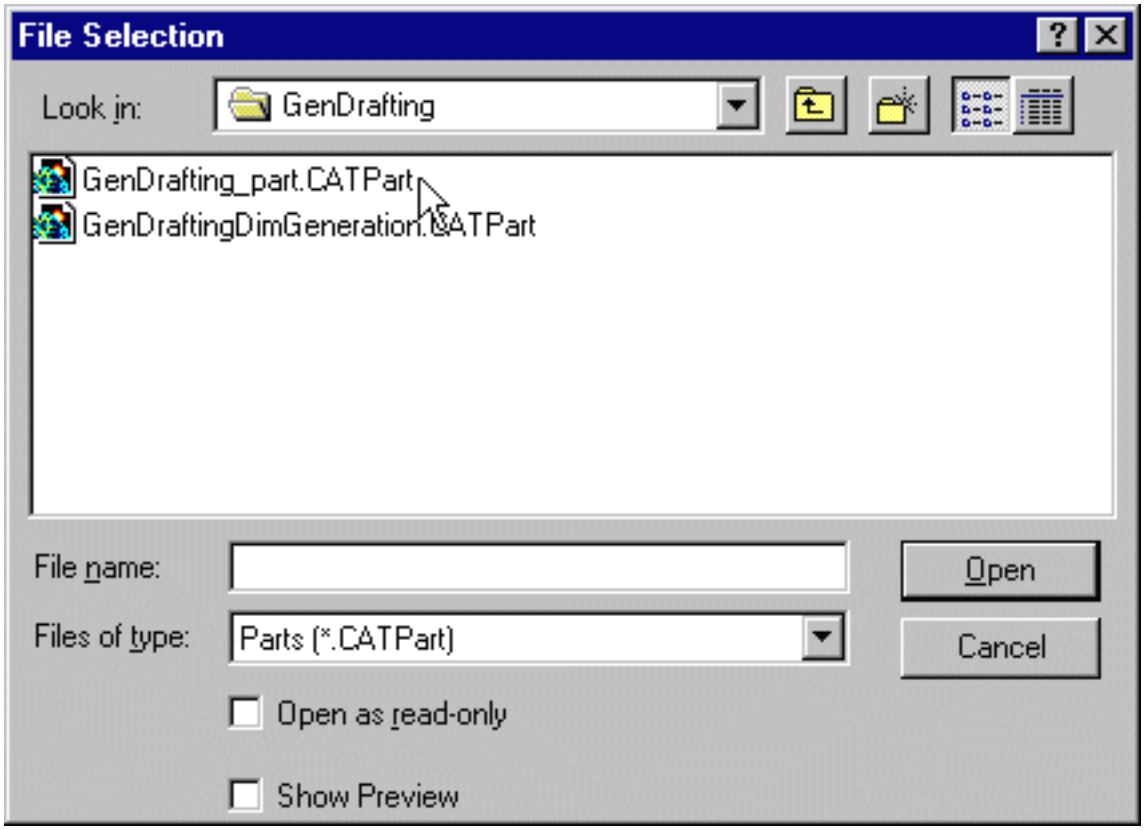
Before You Begin and all along the current Getting Started chapter, make sure you customize the following settings:

Grid:
De-activate the Grid icon from the Tools toolbar (bottom right).

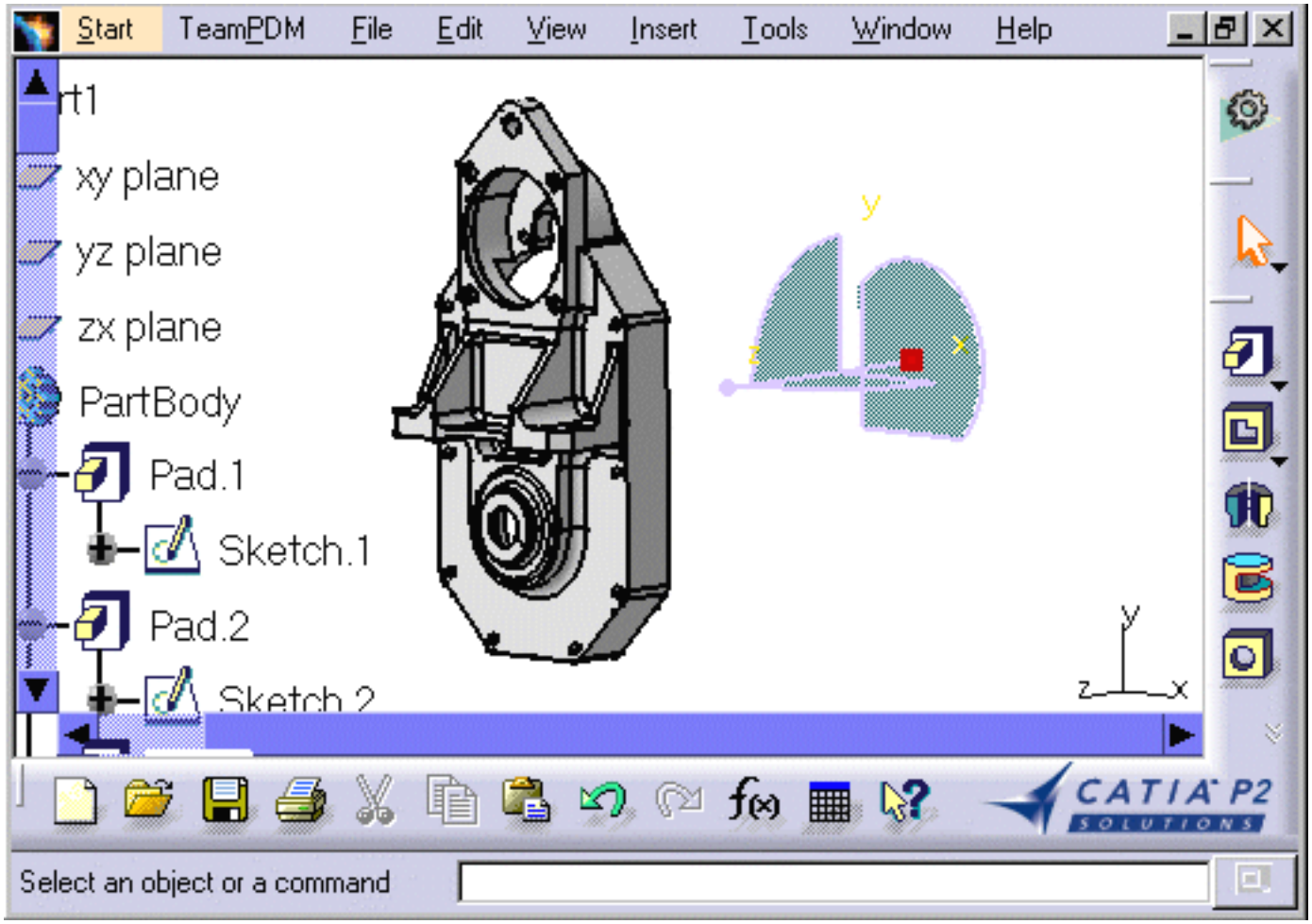
1. Click the Open icon from the Standard toolbar or select `File->Open...` from the menu bar.

The `File Selection` dialog box appears.

2. Select the part to be opened. In this example, the user selects GenDrafting_part. CATPart document.



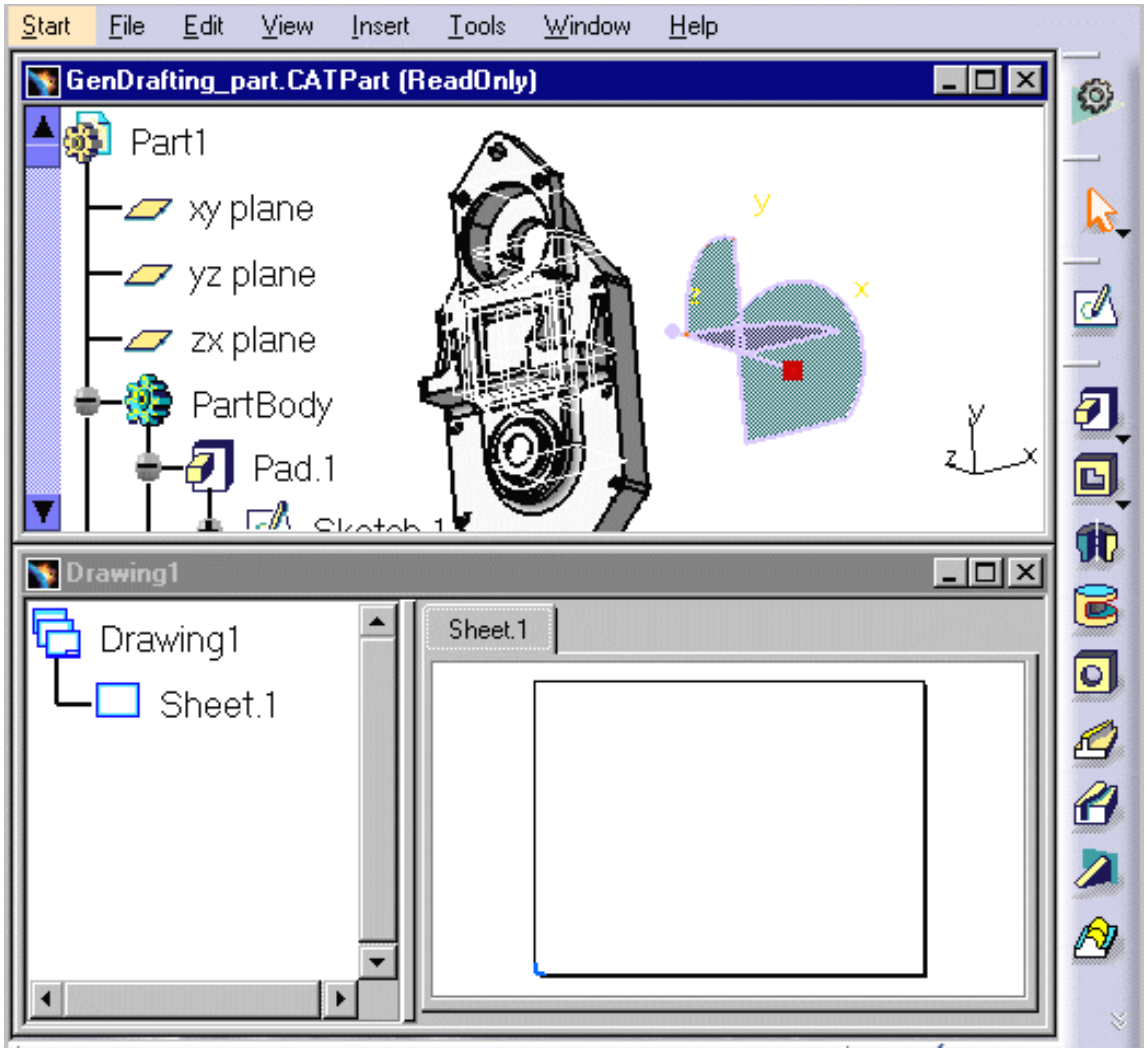
The part is opened and will remain displayed in the window whatever the views you will create from this part.


Start
TeamPDM
File
Edit
View
Insert
Tools
Window
Help
xy plane
yz plane
zx plane
PartBody
Pad.1
Sketch.1
Pad.2
Sketch.2
CATIA P2
SOLUTIONS
Select an object or a command

Creating a Front View

This task will show you how to create a front view on the sheet previously defined and from the 3D part you previously opened.

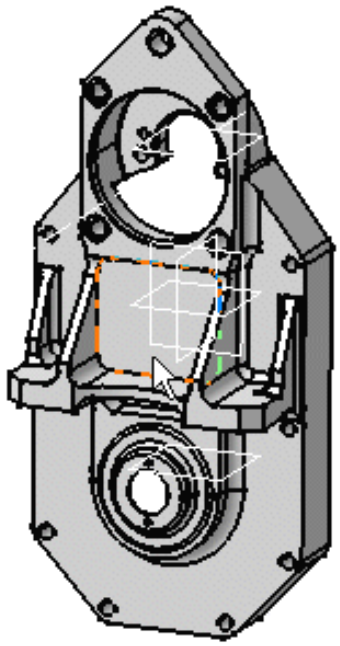
At this step, we strongly advise that you tile screen horizontally . For this, go to `Window -> Tile Horizontally` options from the menu bar.



If you do not want to have the specification tree displayed, press the PF3 key.

1. Click the Drawing window, and click the Front View icon  from the Views toolbar (Projections sub-toolbar).

2. Select the desired planar surface of the 3D part you opened, from the 3D Part viewer.

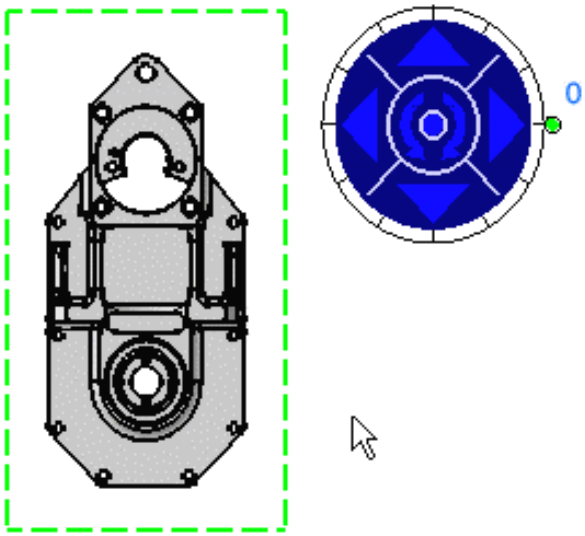


Blue arrows and a green frame including a preview of the view to be created appear on the sheet.

These frame and arrows allow defining the location and orientation of the view to be created.

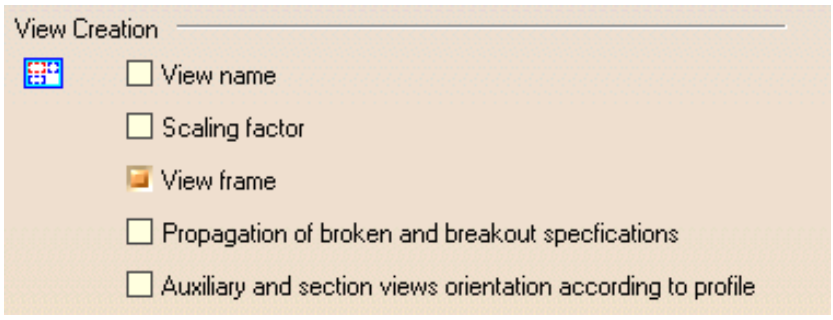
3. Click on the drawing sheet or at the center of the blue manipulator to generate the view.

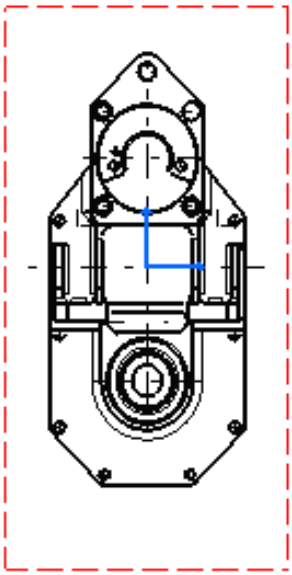
As long as you see the green frame, you can define the frame position using the blue manipulators: top, bottom, left, right or rotated according to a given snapping, or else according to an edited rotation angle.



In the **Generative Drafting** workbench, the view name, scaling factor and view frame are set by default. Throughout this documentation, we decided not to display view names and scaling factors. For this:

Go to Tools->Options->Mechanical Design->Drafting option (Layout tab) and un-check the View name and Scaling factor options.

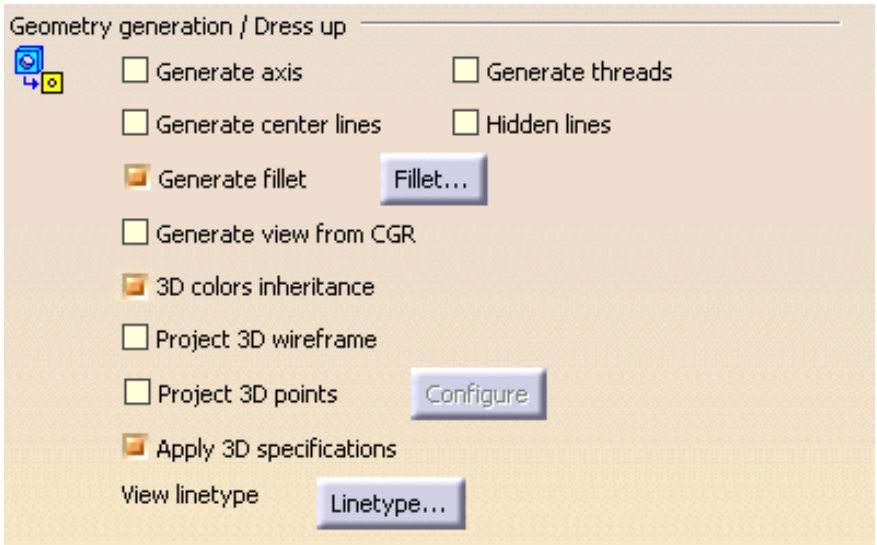




The front view is created.

From now on, you will work on the created sheet unless you define a new sheet.

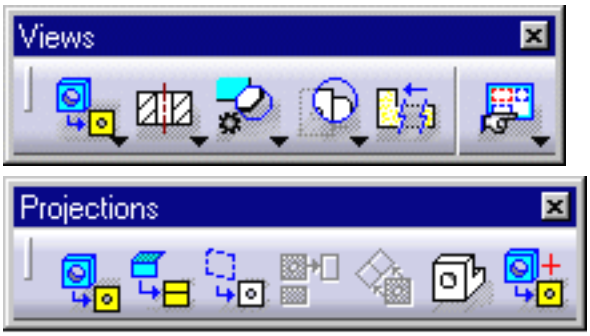
By default, a number of elements can be visualized on generated views. If needed, go to Tools->Options -> Mechanical Design -> Drafting option (Generation tab) to change this by un-checking the required options.



Creating a Projection View

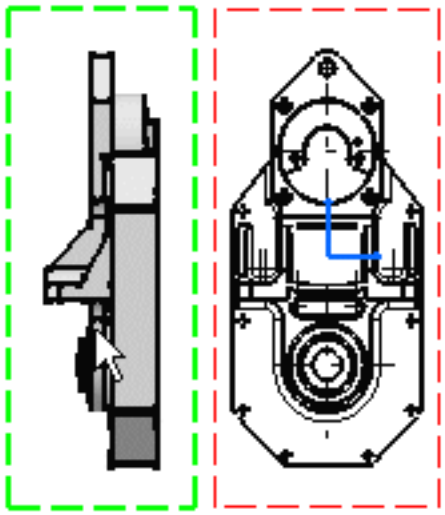
This task will show you how to create projection views on the sheet.

1. Click the Drawing window and click the Projection View icon from the Views toolbar (Projections sub-toolbar).

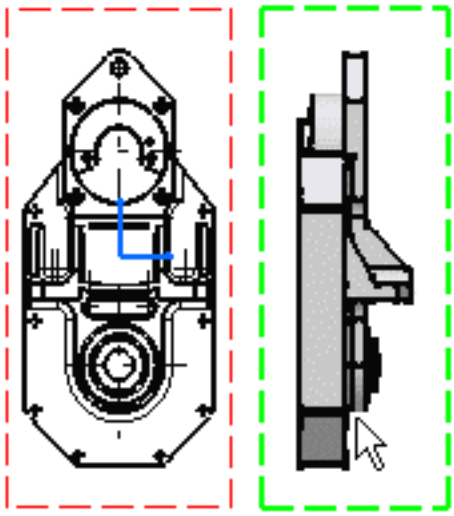


A preview of the view to be created appears. By default, the projection view is aligned to the front view.

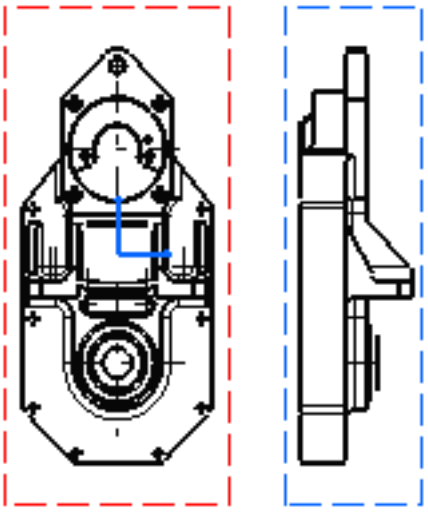
As you move the cursor, a preview of the view to be created appears, as long as you keep the cursor positioned at any possible projection view location (at the left, right, top or bottom of the red frame).



2. Define the projection view position, for example the right view position, using the cursor.

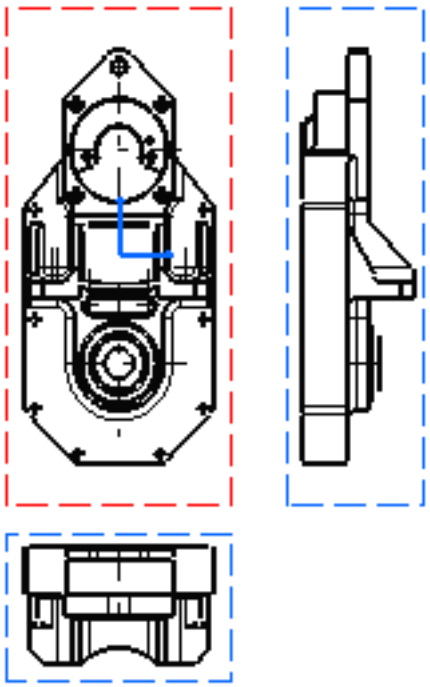


Note that the left view shown here was created and therefore positioned according to ISO standards and the First Angle Projection method. For more information, please refer to Creating Views via the Wizard.



3. Click to generate the view.
4. Click the Drawing window, and click the Projection View  using the cursor.
5. Define the top view position.
6. Click at the left, right, top or bottom of the red frame to generate the view.

   The views result as shown here.

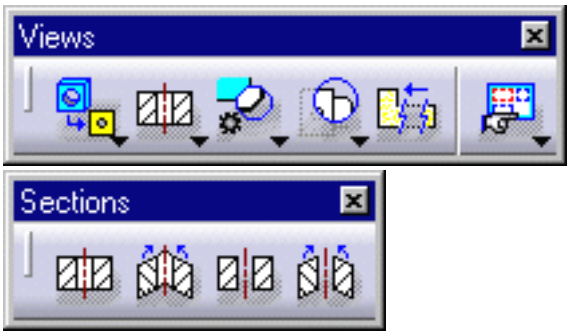


Creating a Section View

This task will show you how to create a section view using the front view previously generated.

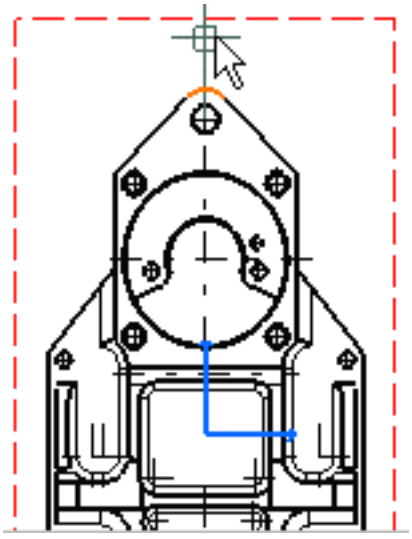
This section view will make drawings more readable by replacing the hidden elements of parts including holes with filled areas.

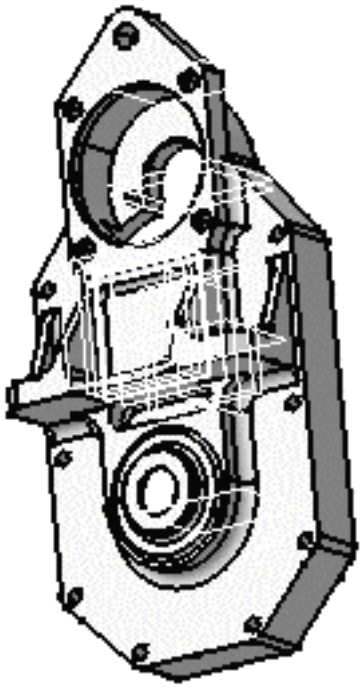
1. Click the Drawing window, and click the Offset Section View icon from the Views toolbar (Sections subtoolbar).



2. Select the holes and points required for sketching the callout on the view.

Selecting a circular, a linear edge or an axis line (for example, a hole) amounts to making the callout associative by default to the 3D feature. If you select a circle, the callout will go through the circle center. If you select an edge, the callout will be parallel to the selected edge.

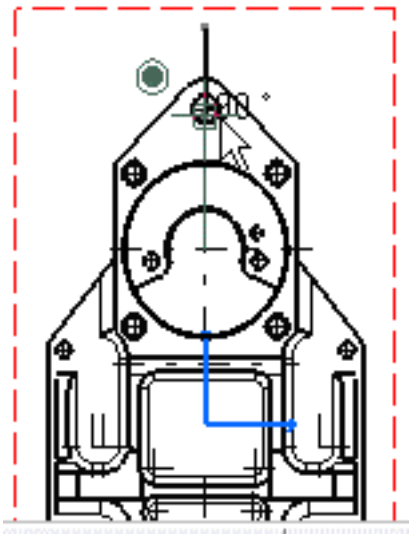


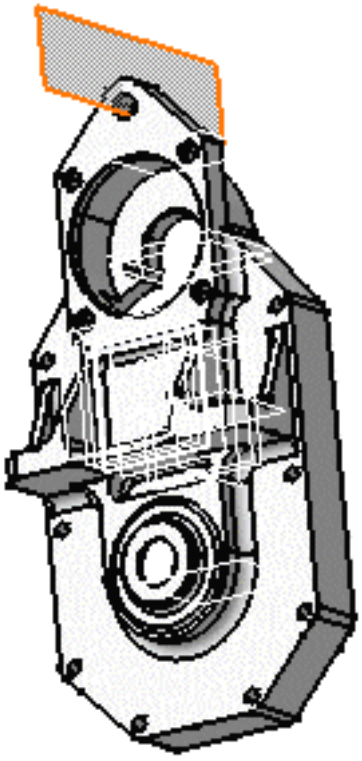


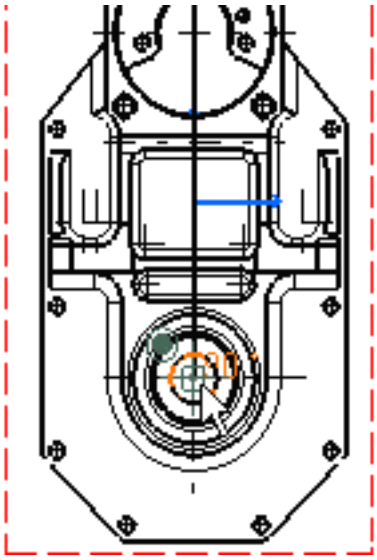
If you are not satisfied with the profile you create, you can, at any time, use Undo  or Redo  icons.

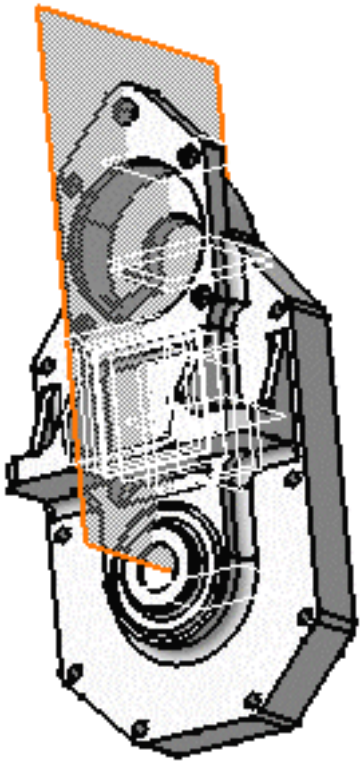
Note that SmartPick assists you when creating the profile.

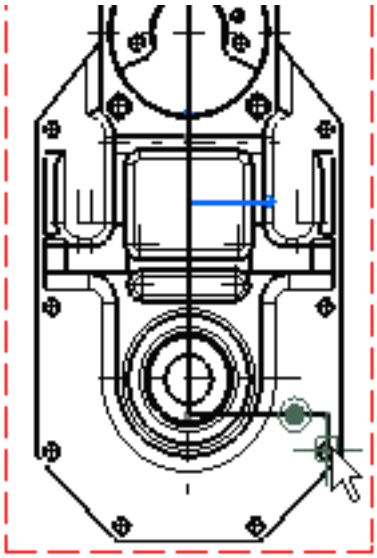
The section plane appears at the second point you select and moves dynamically on the 3D part as you create the callout on the drawing. This section plane will automatically disappear as you will double-click to end the callout creation.

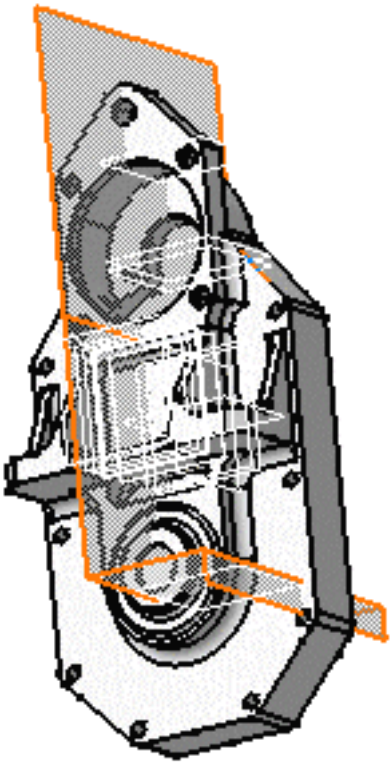


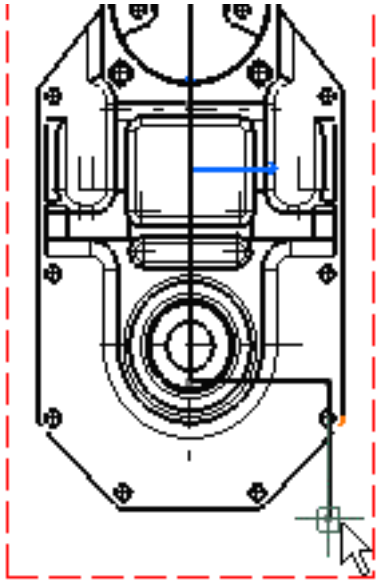


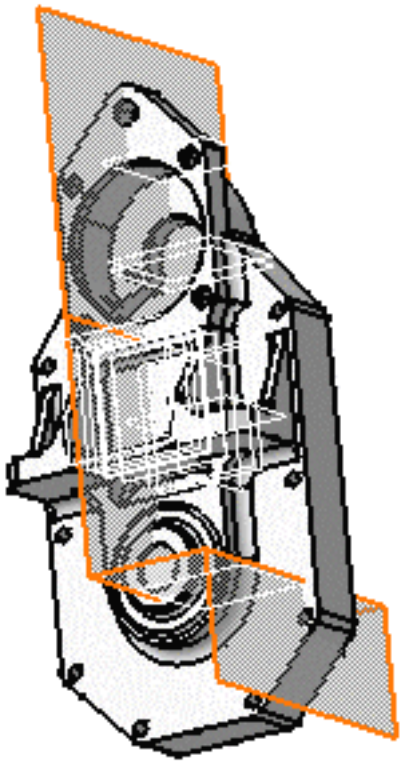






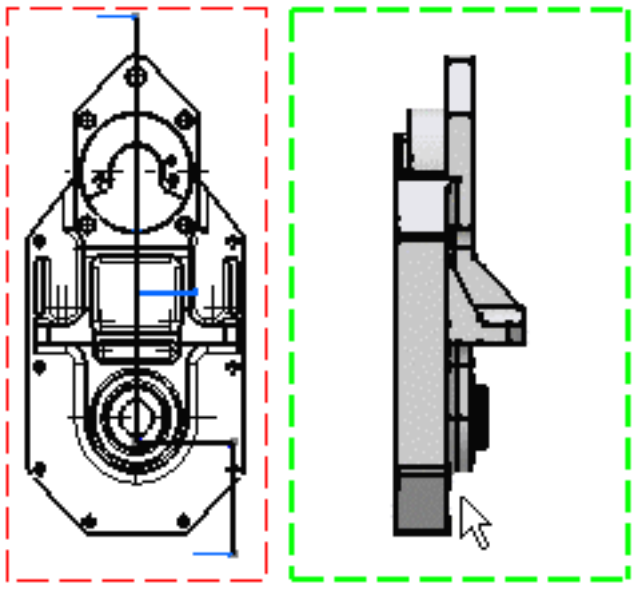


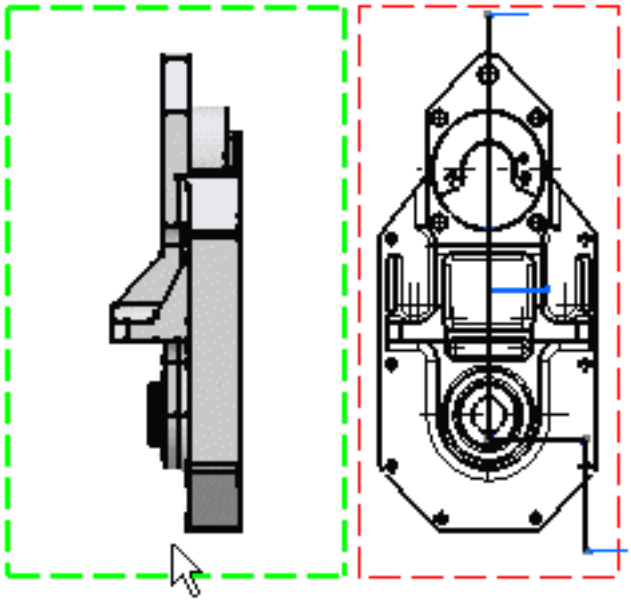




3. Double-click to end the cutting profile creation.

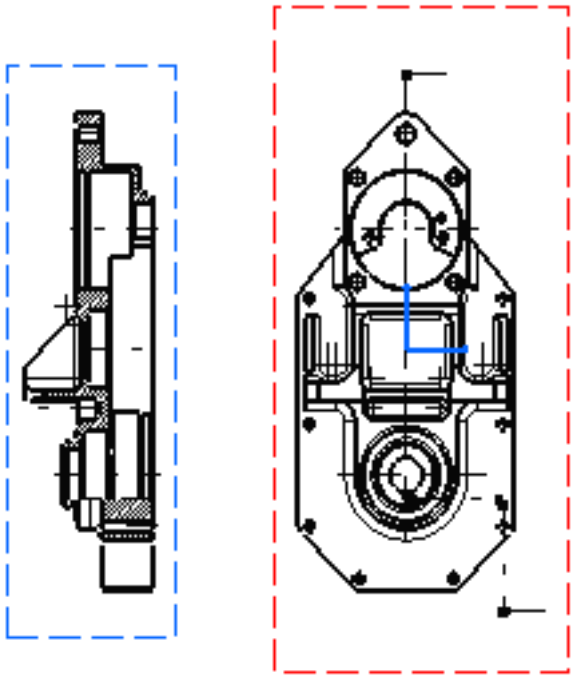
Positioning the view amounts to defining the section view direction. The callout blue arrows direction is modified according to the cursor position. In other words, this preview behaves as if it were either a left or a right projection view you need to position.





4. Click to generate the view.

Using the cursor, you can then position the section view so that it is or not aligned to the front view.



You can modify the hatching pattern by pressing the right mouse button on the section view pattern and selecting the `Properties` option from the contextual menu. You will then display a `Properties` dialog box in which you will either select a new hatching pattern or modify the graphical attributes of the existing hatching pattern.

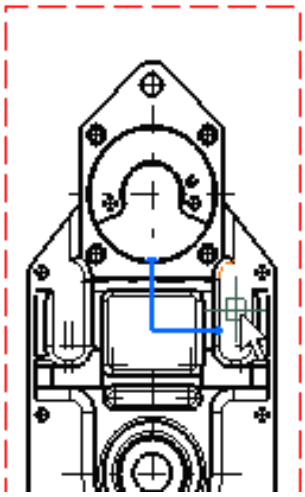
Creating a Detail View

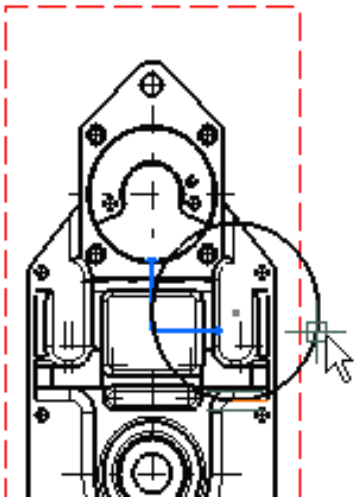
This task will show you how to create a detail view from the front view you previously generated.

1. Click the Drawing window, and click the Detail View icon  from the Views toolbar (Details subtoolbar).

2. Click the callout center.

3. Drag to select the callout radius.





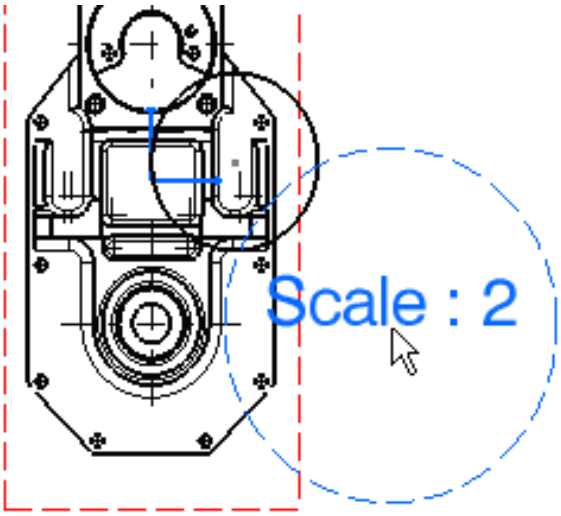
4. Click a point on the callout.

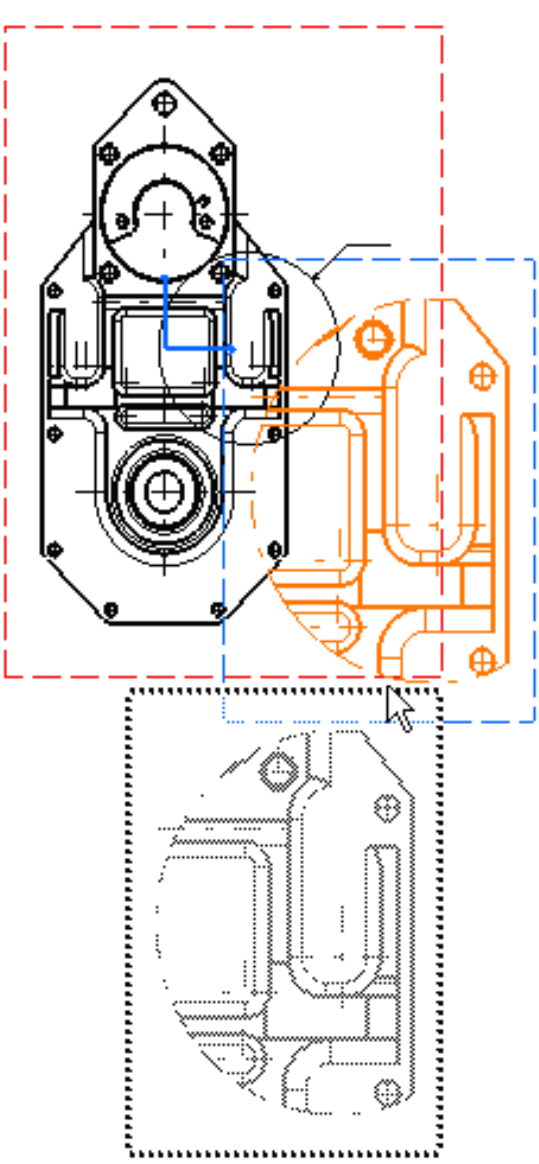
   A blue circle appears at the position of the cursor.

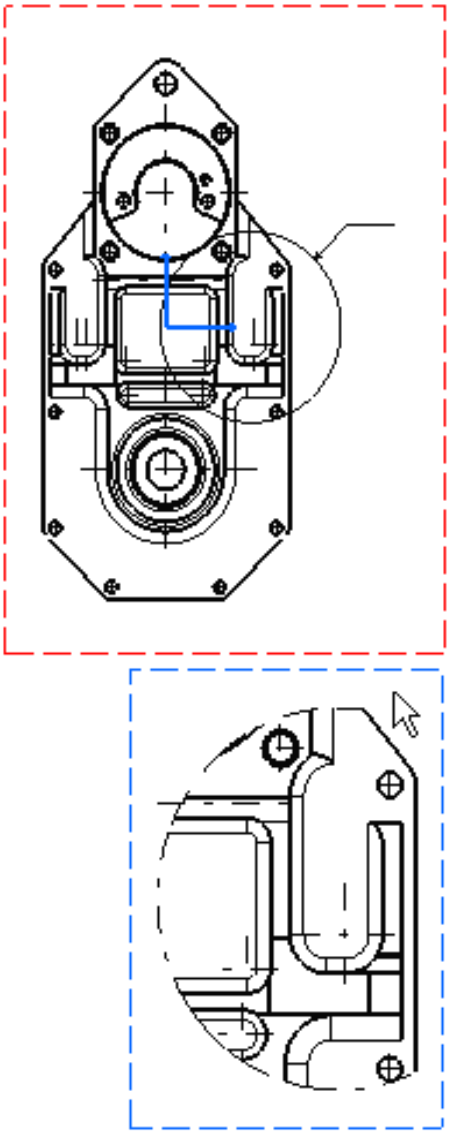
5. Move the previewed detail view to the desired location.

6. Click inside the blue circle to position the detail view at the desired location.

7. If needed, drag the detail view to a new position.

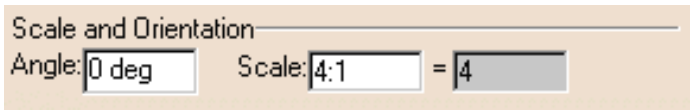






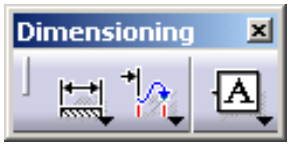
As shown above, the scale by default is twice that of the active view. You can modify this scale. For this:

8. Right-click the detail view and select the `Properties` option from the contextual menu, `View` tab, enter `4` as `Scale` and click `OK`.



In this particular case, we will add a dimension to the detail view. For this:

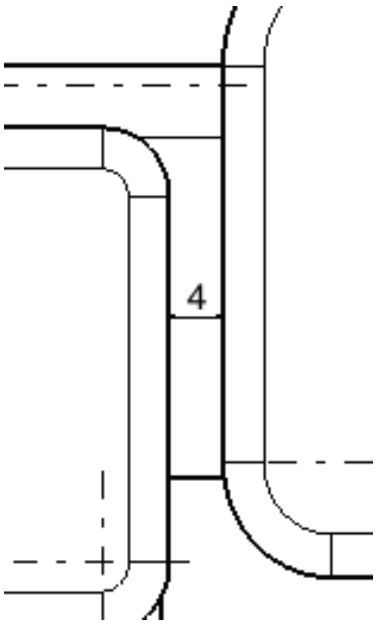
9. Click the Dimensions icon from the Dimensioning toolbar.

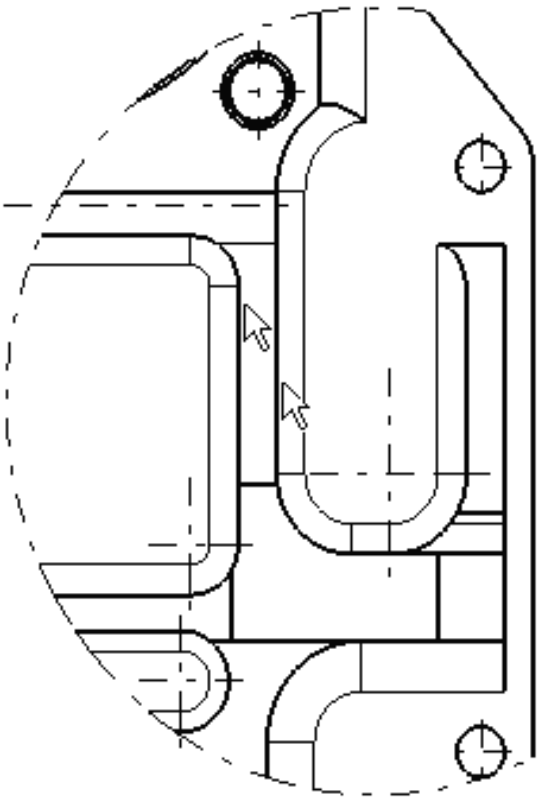


10. Click both required elements in the view as shown here.

11. Enter `7mm` as graphical value font size, in the `Text Properties` toolbar.

The dimension appears:

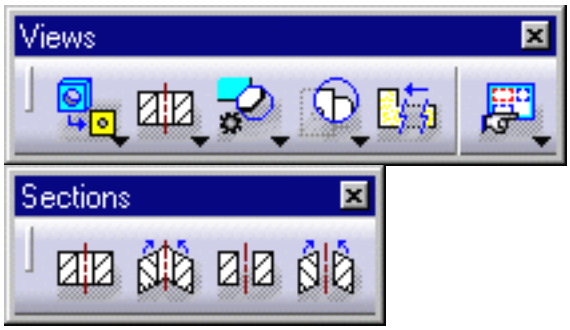




Creating a Section Cut

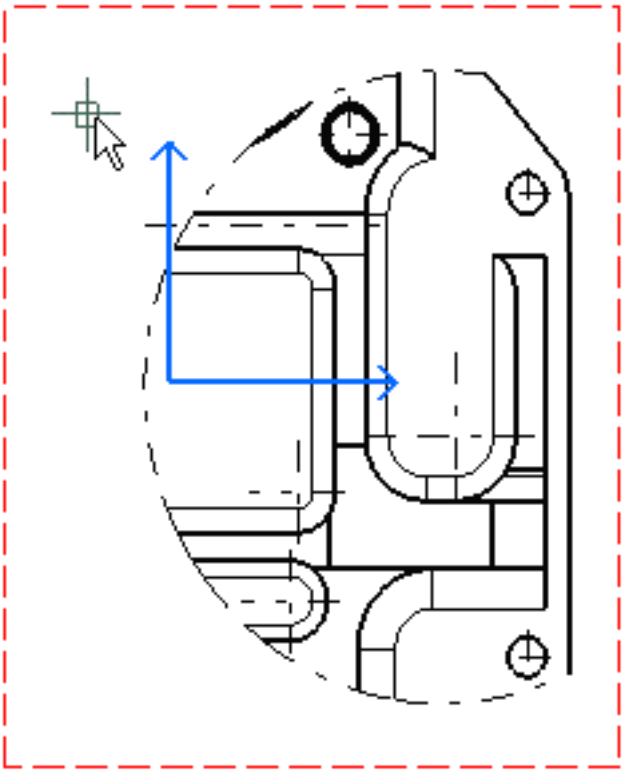
This task will show you how to create a section cut from the detail view you just created.
Be careful: the scale of the section cut will depend on the scale of the view this section cut is generated from. In this case, the section cut is generated from a detail with a scale 4: The section cut scale will also be 4.

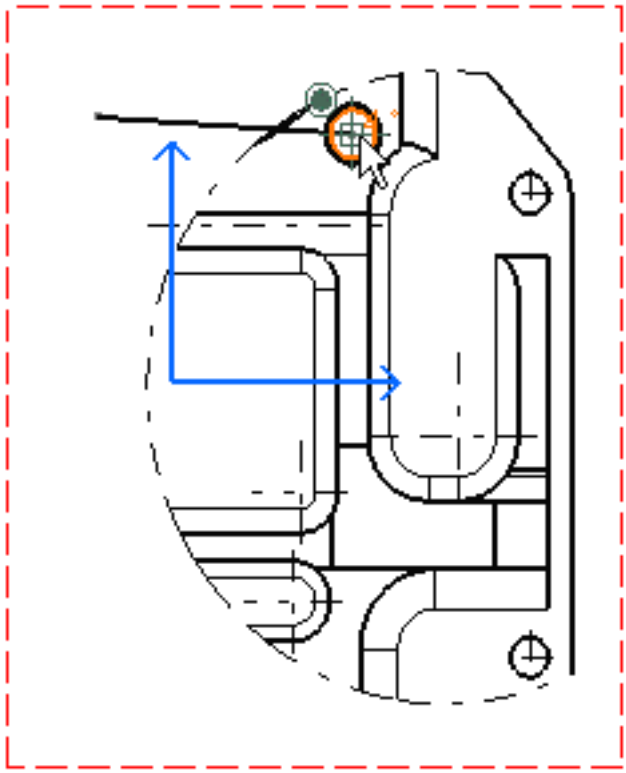
1. Right-click the detail view and select the `Activate View` option from the contextual menu.
2. Select the Drawing window, and click the Aligned Section Cut icon from the Views toolbar (Sections subtoolbar).



3. Select the holes and points required for sketching the cutting profile.

Selecting a circular, a linear edge or an axis line (for example, a hole) amounts to making the cutting profile associative by default to the 3D feature.





If you are not satisfied with the profile you create, you can, at any time, use Undo or Redo icons.

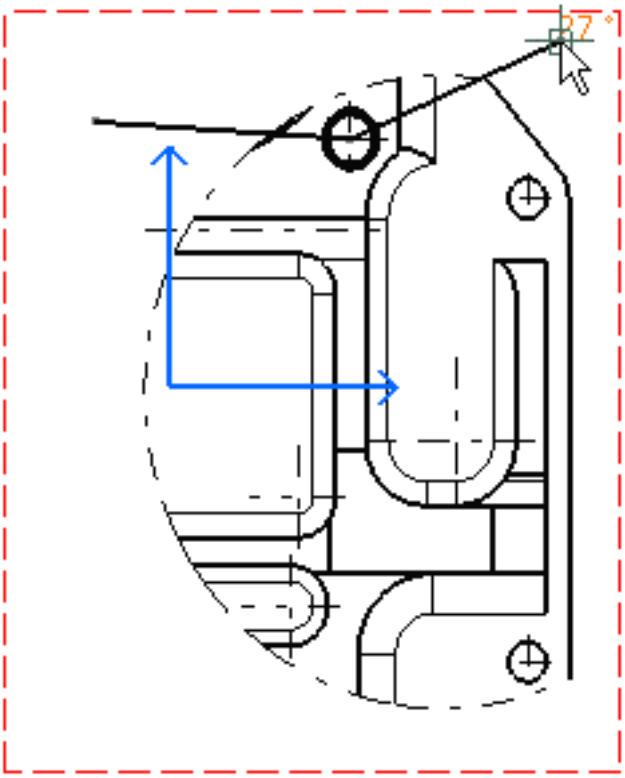
Note that SmartPick assists you when creating the profile.

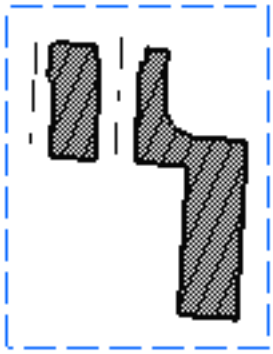
4. Double-click to end the cutting profile creation.

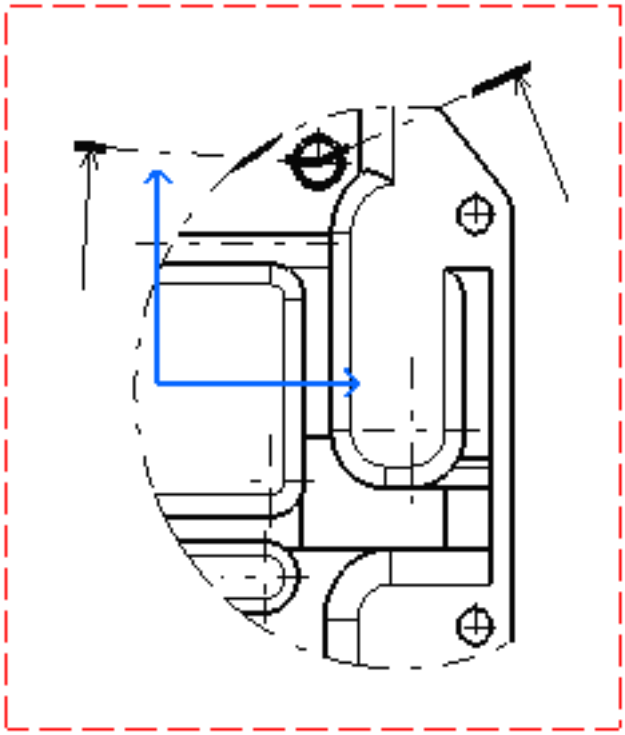
A preview with the view to be created appears. Positioning the section cut either to the right or to the left amounts to defining the section cut direction (as if it were a projection view).

5. Click to generate the section cut.

Once you have clicked, you can modify position of the section cut relatively to the detail view on the condition you right-click the section view and select the Do not align views option from the contextual menu.





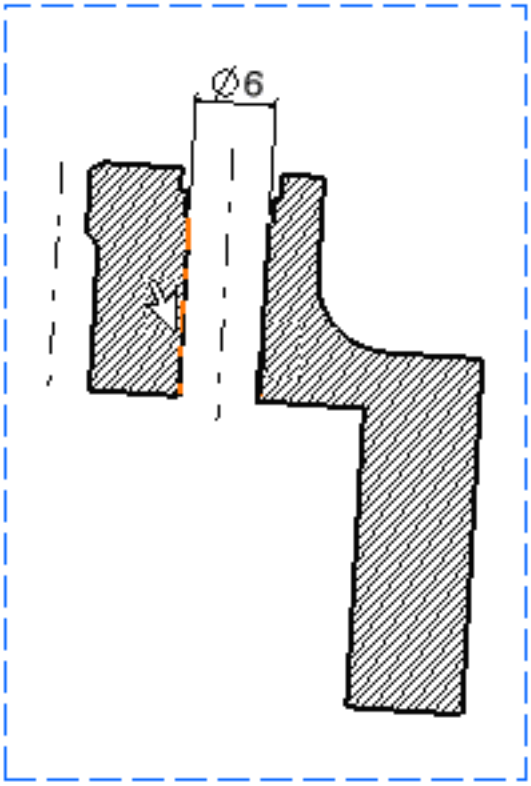


You can select an existing edge within the view and define automatically the direction of the cutting profile. You can also select a reference plane in 3D or a 3D wireframe plane. For more information, please refer to Creating an Offset Section Cut/Section View.

In this particular case, we will add a dimension to the detail view. For this:

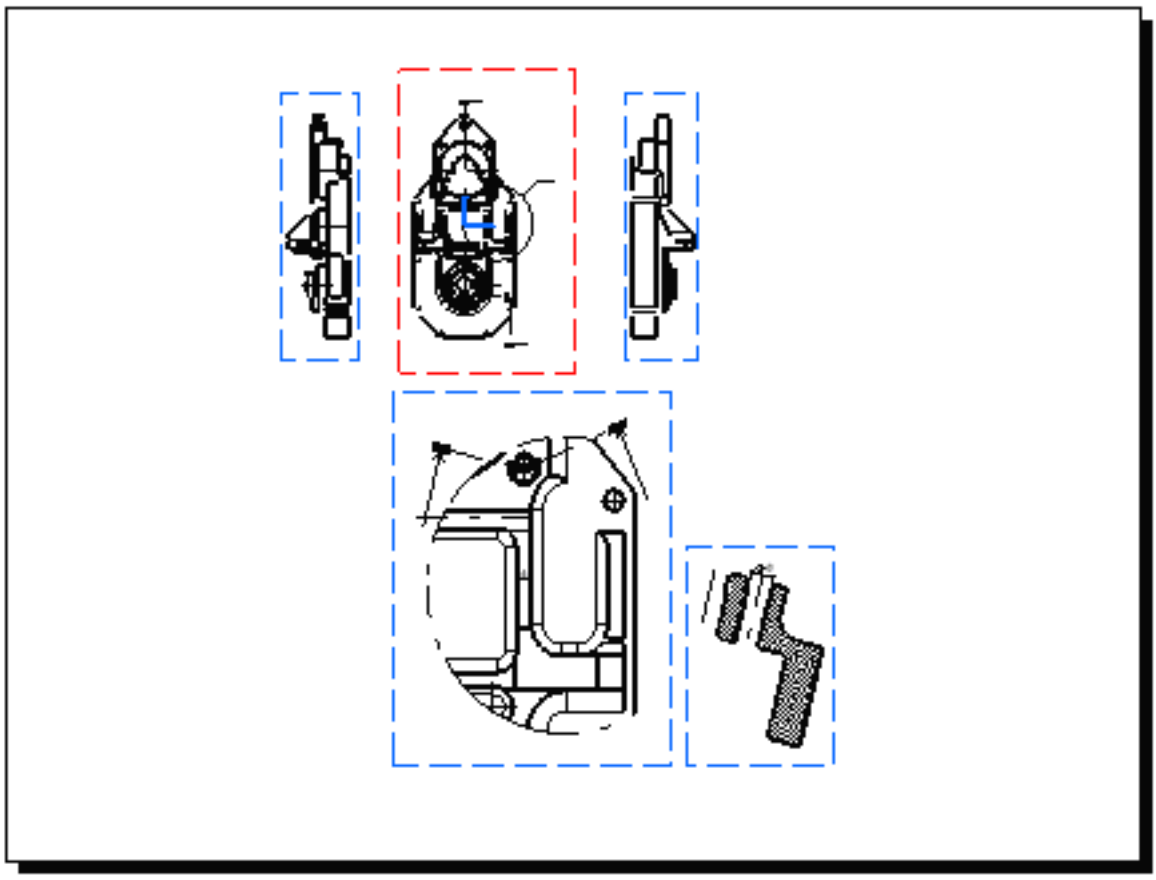
6. Add a diameter dimension to the section cut by clicking one edge only.

In this case, we applied a higher font size to the dimension value.



You can modify the hatching pattern by pressing the right mouse button on the section cut pattern and selecting the `Properties` option from the contextual menu. You will then display a `Properties` dialog box in which you will either select a new hatching pattern or modify the graphical attributes of the existing hatching pattern.

Now the resulting sheet is as shown here. Note that in this case, we re-positioned the views.



You can now print this sheet. For this, select `File -> Print` from the menu bar. Make sure the print format of the current sheet is the same as the print format of the printer.