

Generative Drafting (ISO)

Version 5 Release 8
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EDU-CAT-E-GDRI-FF-V5R8

Table of Contents (1/2)

1. Introduction to Generative Drafting
 - Generative Drafting Workbench Presentation
3. Starting a Drawing and View Generation
 - Starting a Generated Drawing
 - Defining the Main Views
6. Additional View Generation
 - Section Views and Cuts
 - Secondary Views: Detail, Clipping, Broken, Breakout, Auxiliary, Isometric and Unfolded Views
9. Editing Views Layout and Properties
 - Editing a View and Sheet Properties
 - Adding Sheets to a Drawing
 - Repositioning Views
 - Modifying of Section, Detail and Auxiliary Profiles
 - Modifying of Section, Detail and Auxiliary Graphical Definitions
 - Modifying a Section Hatching Representation
10. Automatic Dimensioning
 - Generative Dimensions
 - Generative Dimensions (step by step)
 - Balloons
13. Finalizing the Drawing and Printing
 - Checking Links to a Solid 3D Part and Updating a Drawing
 - Adding a Title Block
 - Printing the Drawing
16. Setting Drafting Options
 - Drafting Options and Generating Views

Table of Contents (2/2)

- 1. Drafting Visualizations
 - Threaded Holes
 - Using 3D Viewer
- 4. Saving a Drawing Document
 - Saving a Drawing and Checking Links

Introduction To Generative Drafting

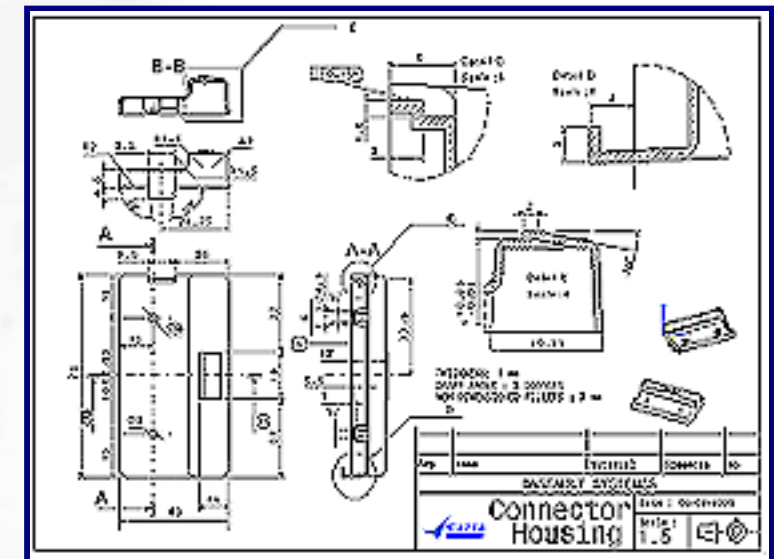
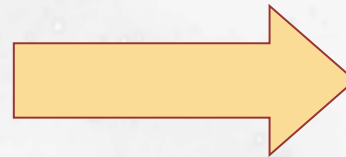
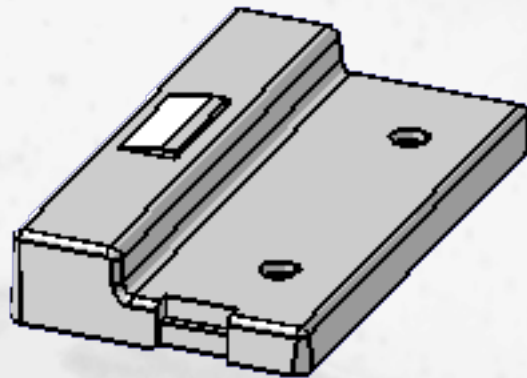
You will become familiar with the Generative Drafting main functionalities.

Generative Drafting Workbench Presentation

Generative Drafting Workbench Presentation

You will learn about the Generative Drafting Workbench by:

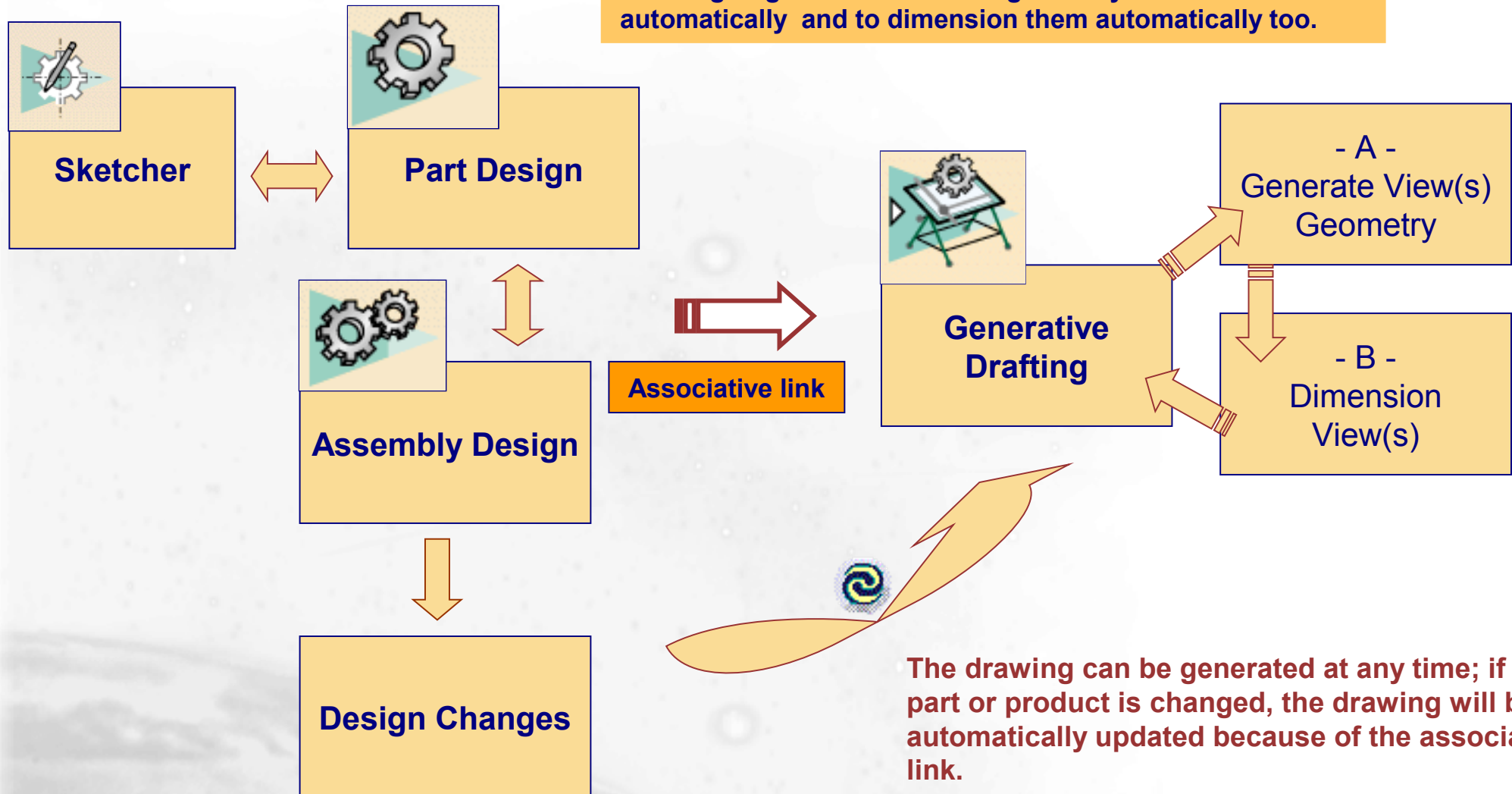
- Understanding the general process
- Accessing the Workbench
- Exploring the User Interface and Terminology



General Process



Generative Drafting workbench and Interactive Drafting workbench are exactly the same except for the tools allowing to generate the views geometry from the 3D automatically and to dimension them automatically too.



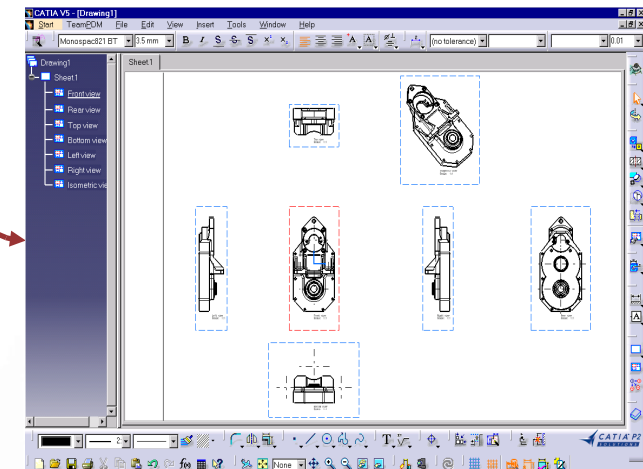
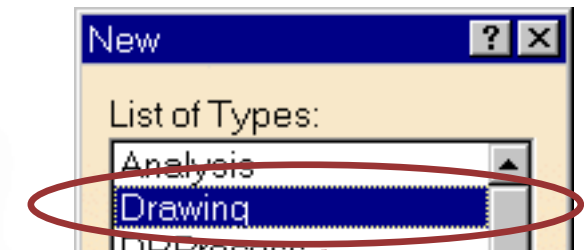
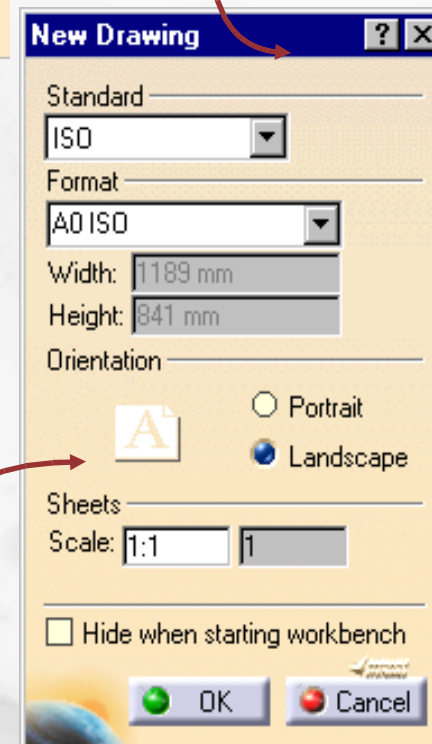
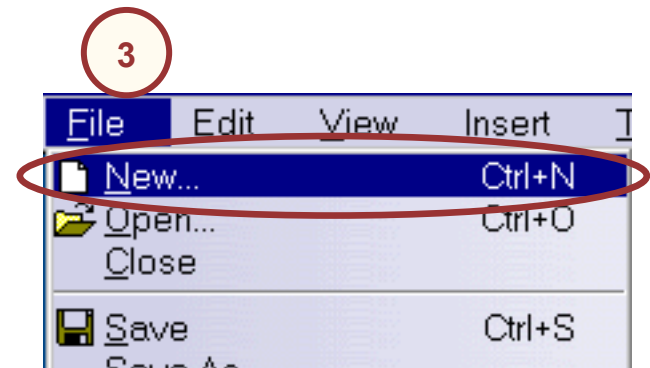
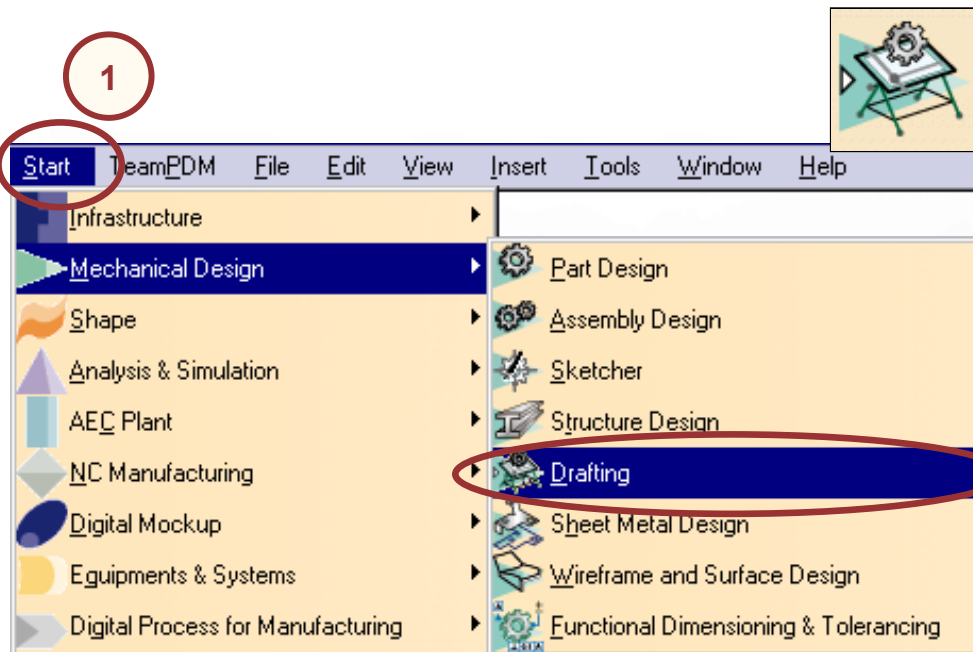
The drawing can be generated at any time; if the part or product is changed, the drawing will be automatically updated because of the associative link.

Accessing the Workbench


From 1- Start menu

or 2- Workbench Icon

or 3- File menu



Drafting Toolbars and Objects

- ▶ Each toolbar contains objects that are related to specific tasks.
- ▶ Objects within these toolbars are compressed and can be expanded for additional capability by selecting the 

Text properties

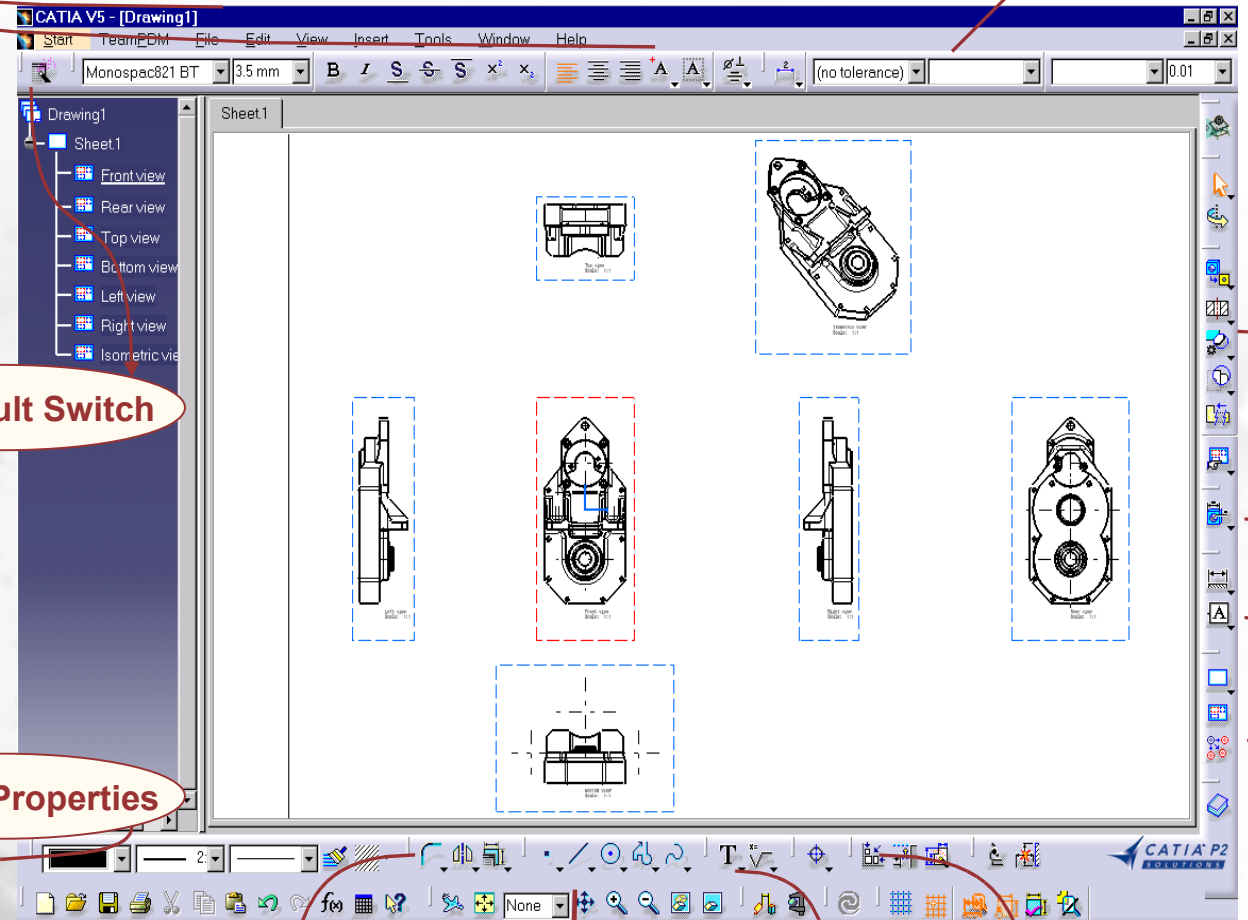
Dimension properties

Set Default Switch

Graphic Properties

Object

- 1 - Views
- 2 - Drawing
- 3 - Dimensioning
- 4 - Generation
- 5 - Annotations
- 6 - Dress Up
- 7 - Geometry creation
- 8 - Geometry modification



The screenshot shows the CATIA V5 Drafting environment. The main window displays a technical drawing of a mechanical part with various views (Front, Rear, Top, Bottom, Left, Right, Isometric). The interface includes a menu bar (Insert, Tools, Window, Help), a toolbar (Monospac821 BT, 3.5 mm, B, I, S, S, x, x, (no tolerance), 0.01), and a left sidebar (Drawing1, Sheet1, Front view, Rear view, Top view, Bottom view, Left view, Right view, Isometric view). The bottom status bar shows CATIA P2 SOLUTION. Red callouts point to specific toolbars and objects: 'Text properties' points to the 'Text' toolbar; 'Dimension properties' points to the 'Dimension' toolbar; 'Set Default Switch' points to the 'Set Default Switch' button; 'Graphic Properties' points to the 'Graphic Properties' button; and numbered callouts 1 through 8 point to the 'Views', 'Drawing', 'Dimensioning', 'Generation', 'Annotations', 'Dress Up', 'Geometry creation', and 'Geometry modification' toolbars respectively.

Views 1

Generation 4

Dimensioning 3

Drawing 2

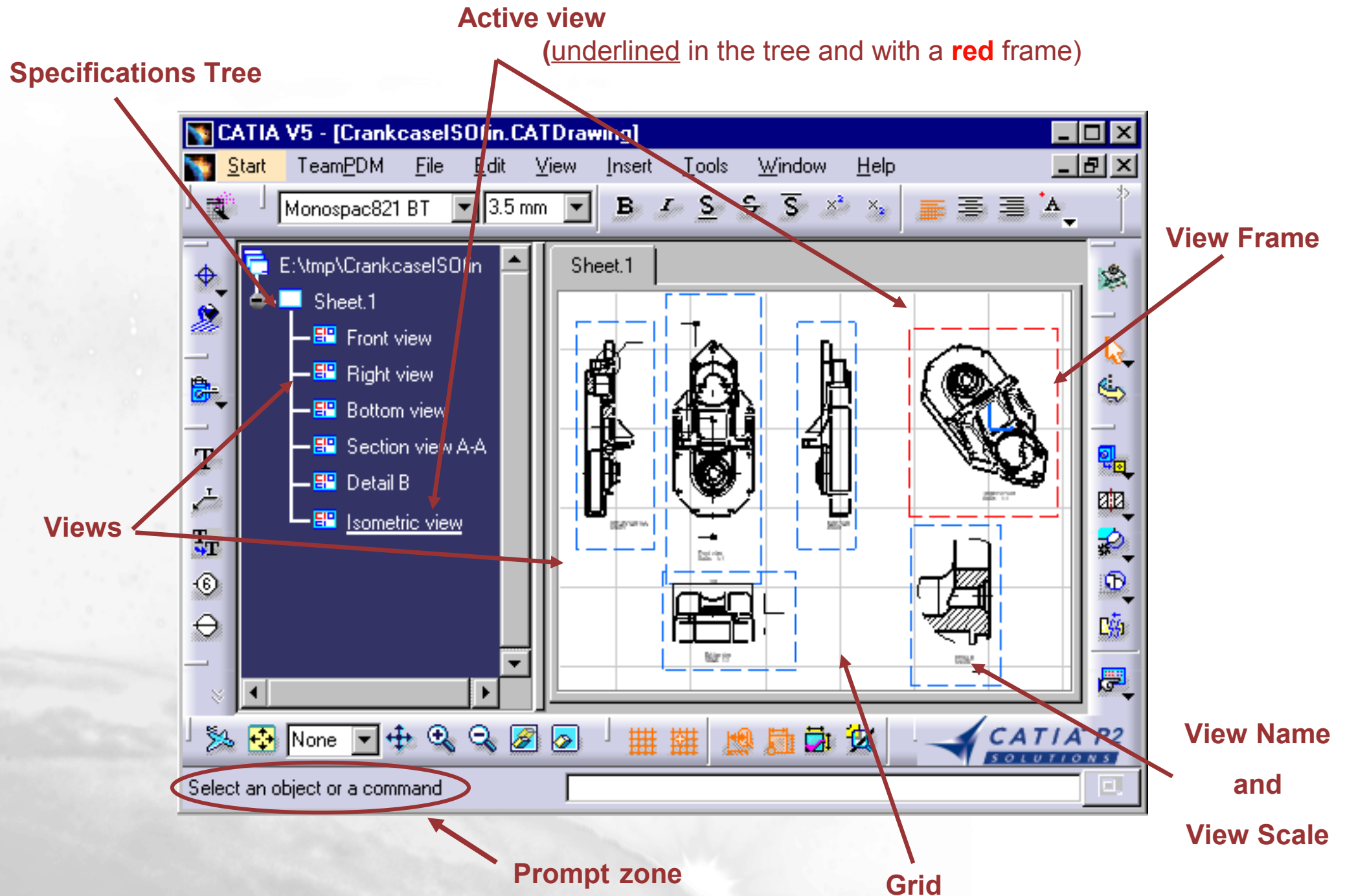
Geometry creation 7

Geometry modification 8

Annotations 5

Dress Up 6

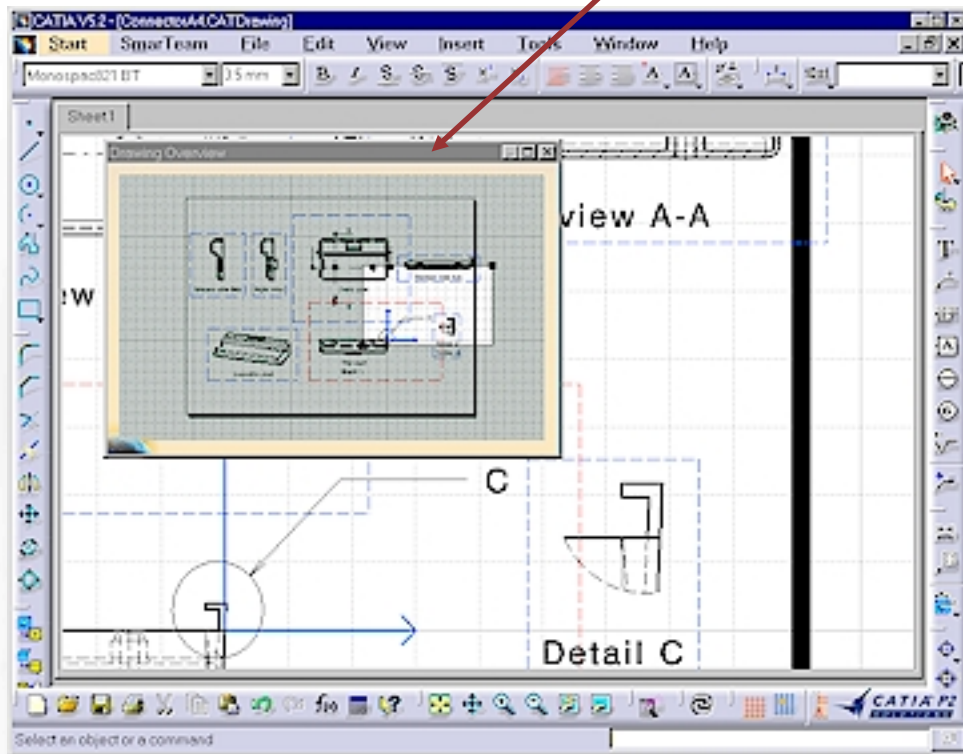
Drafting Terminology



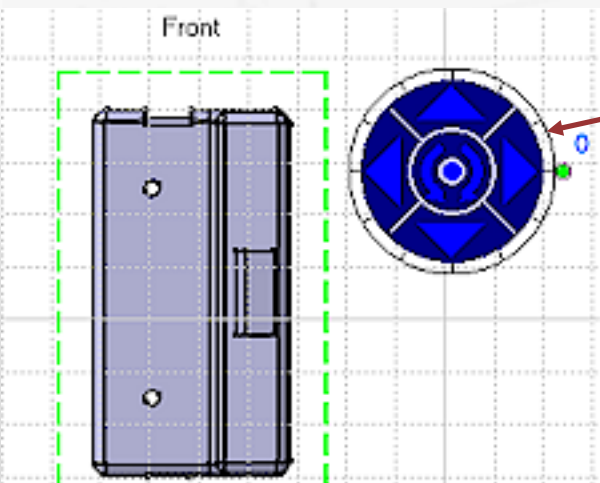
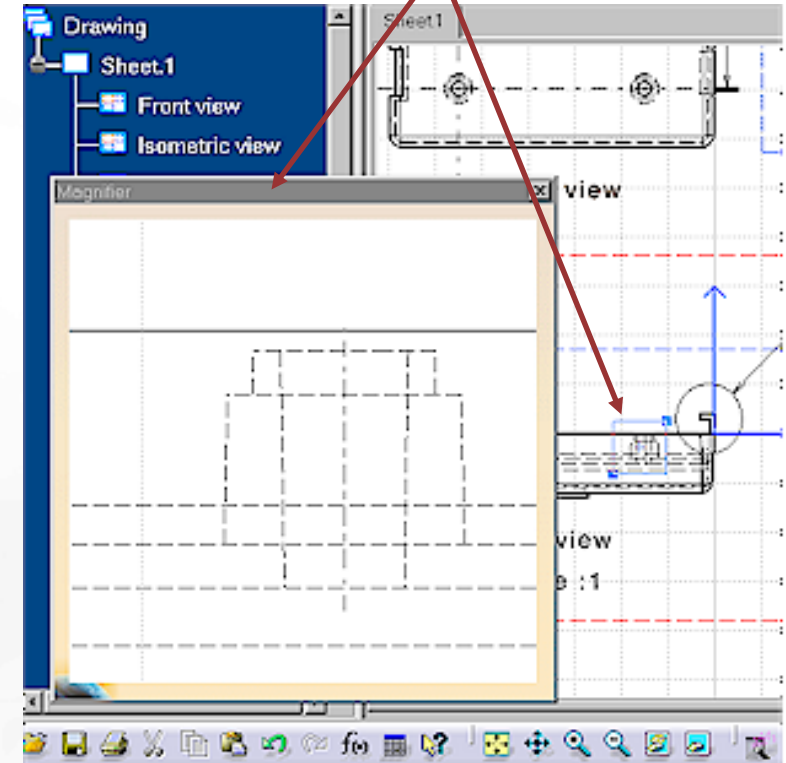
Generative Drafting (P2 Power Tools)

►(View Menu)

- Drawing Overview (on)

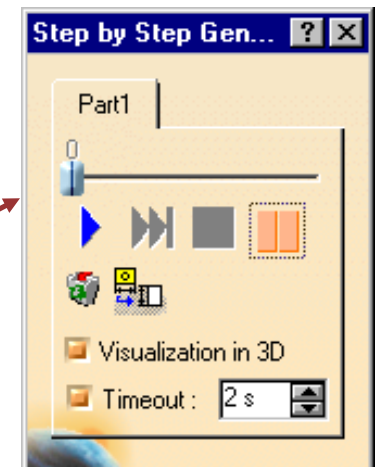


- Magnifier (on)



- View Manipulator Dial
(at Front view creation
with view wizard)

- Step by step Dimension
Generation



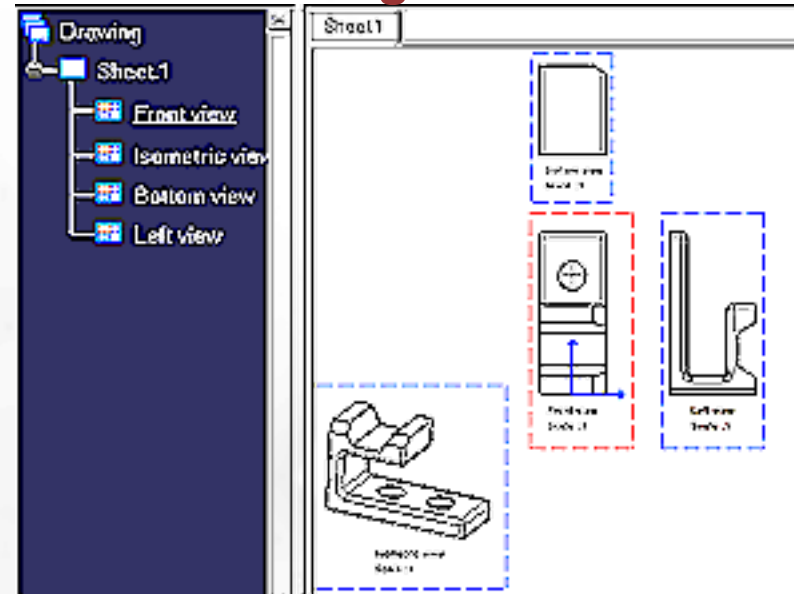
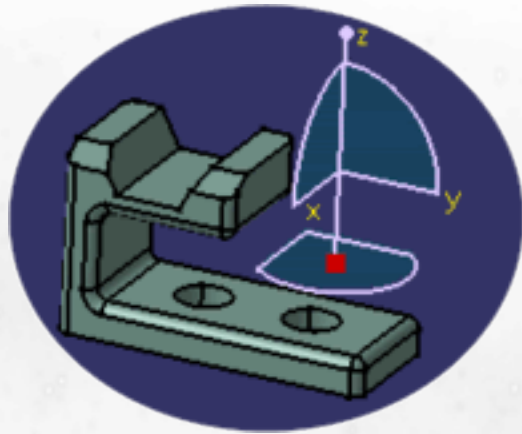
To Sum Up ...

In this introduction to the Generative Drafting Workbench,

- You had a quick tour of the process to create the views of a 3D part and how to access the Generative Drafting workbench
- You have seen the general layout of the user interface, terminology and the basic principles.

Starting a Drawing and View Generation

In this lesson you will learn how to generate a drawing and views for a 3D part.

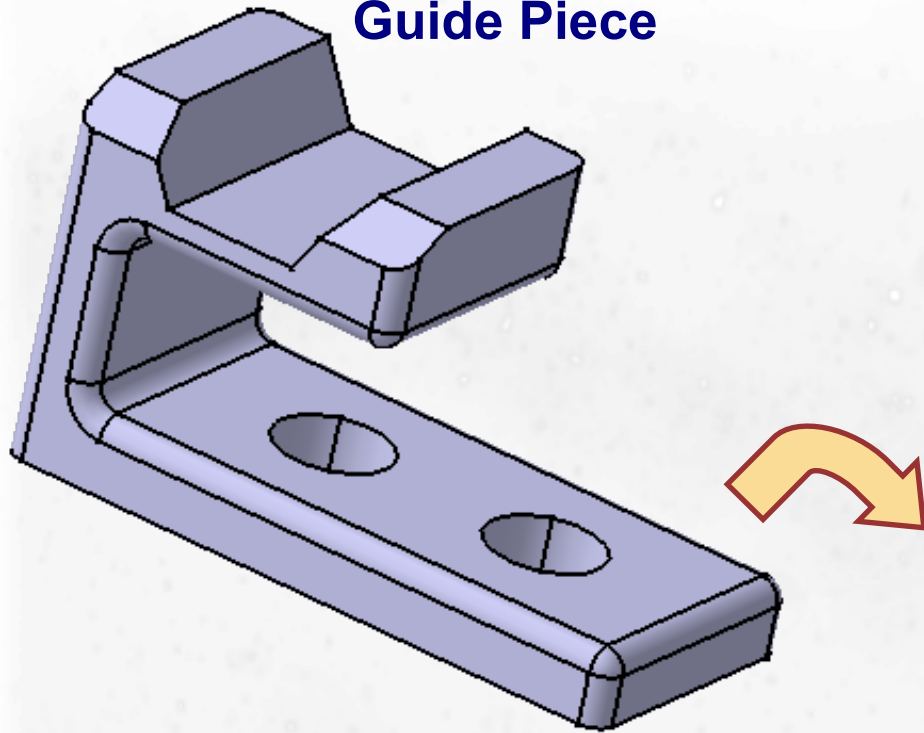


- Starting a Generative Drawing
- Defining the Main Views

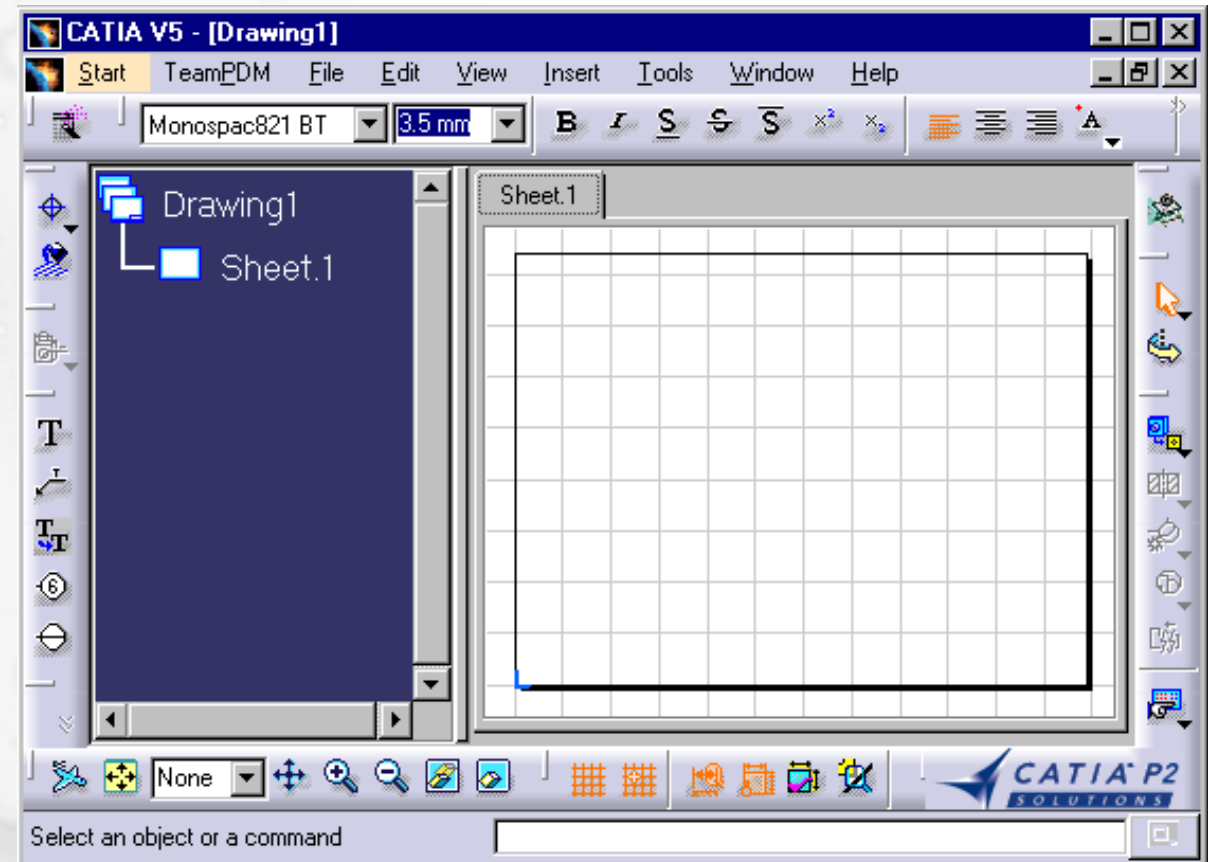
Starting a Generative Drawing

You will learn how to generate a drawing directly from a 3D part.

Guide Piece



Drawing

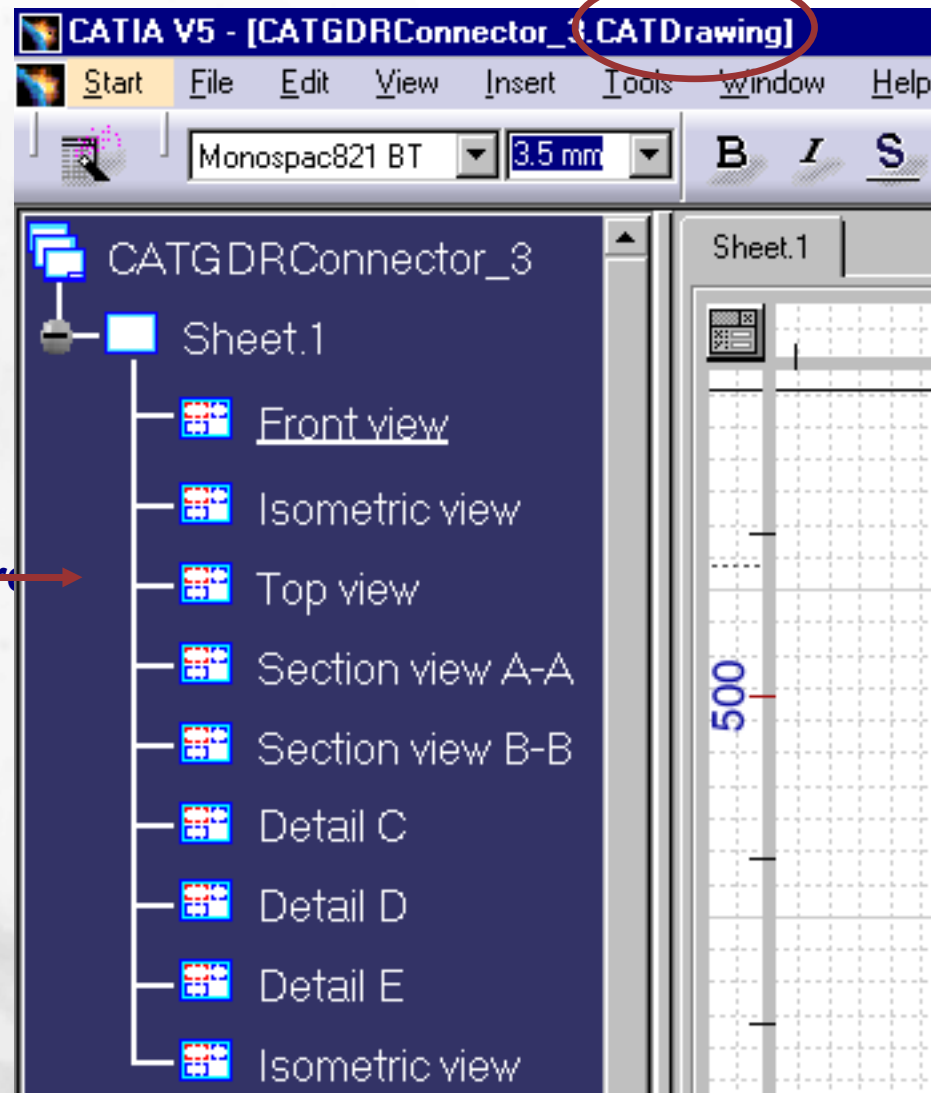


What is a Drawing?

A Drawing Document is a file that is also called a CATDrawing and is identified by its file extension (.CATDrawing).

CATDrawing file extension identification.

A CATDrawing file contains a structure listing of all the sheets and views contained in the document.

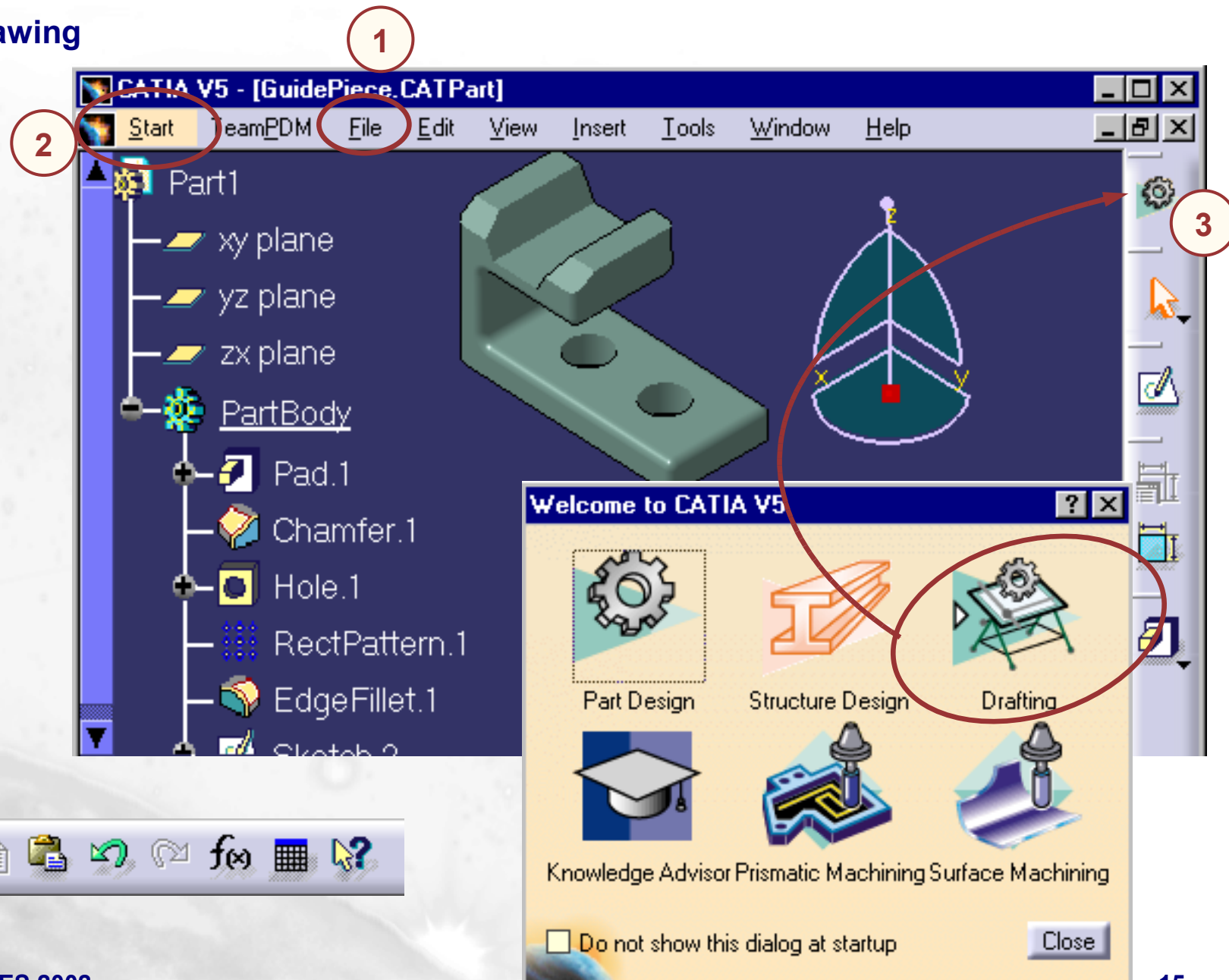


How to Start a Generative Drawing from a CATPart?

Drawing Documents (CATDrawing) can be created in various ways.

Ways to Generate a Drawing

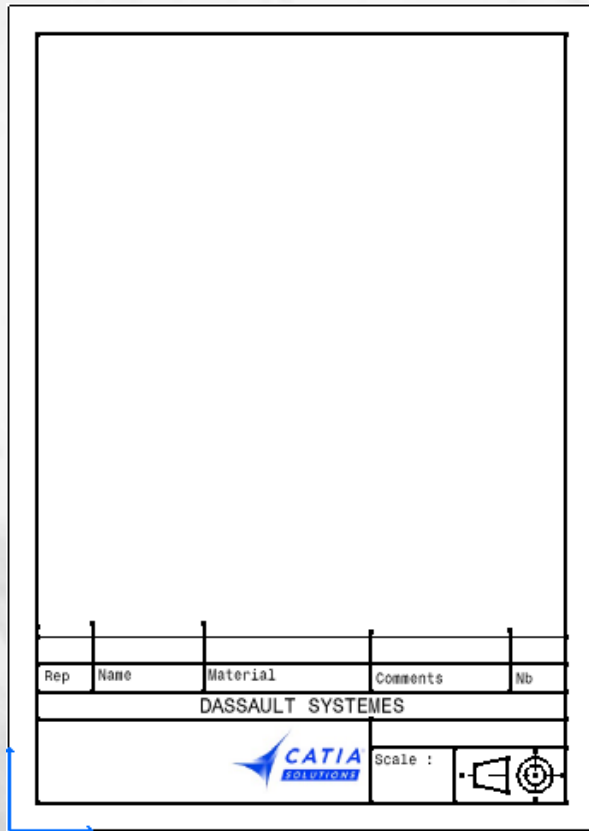
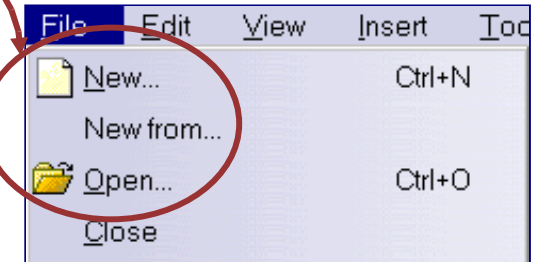
- 1- File menu
- 2- Start menu
- 3- Workbench Icon
- 4- New Icon



Classical Method to Start Generating a Drawing

File Options:

- New... for starting a default drawing document
- New from... for starting a drawing from an existing document
- Open... for opening an existing document

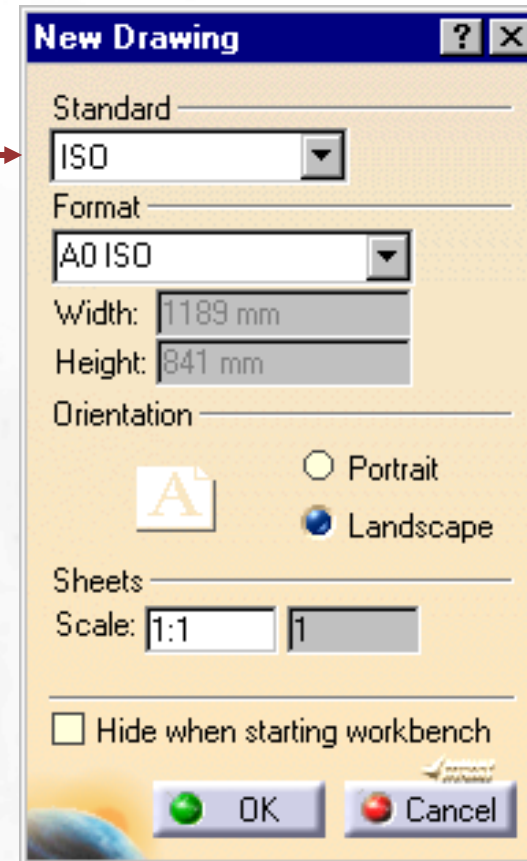


The New from or Open options could be for retrieving company startup documents.

Setting the Drawing Sheet Format and Drafting Standards

The **Modify** option changes the default sheet format and the format standards.

ISO
ANSI
or JIS



The following items maybe set:

- **ISO, ANSI and JIS** standards
- Paper formats (A, B, C, or A0, A1, A2, etc..)
- Orientation (Landscape or Portrait)
- Sheet scale (1 is default and should only be changed with caution)



Format settings can be modified later if necessary: for example the paper orientation or size with menu **File + Page Setup...**

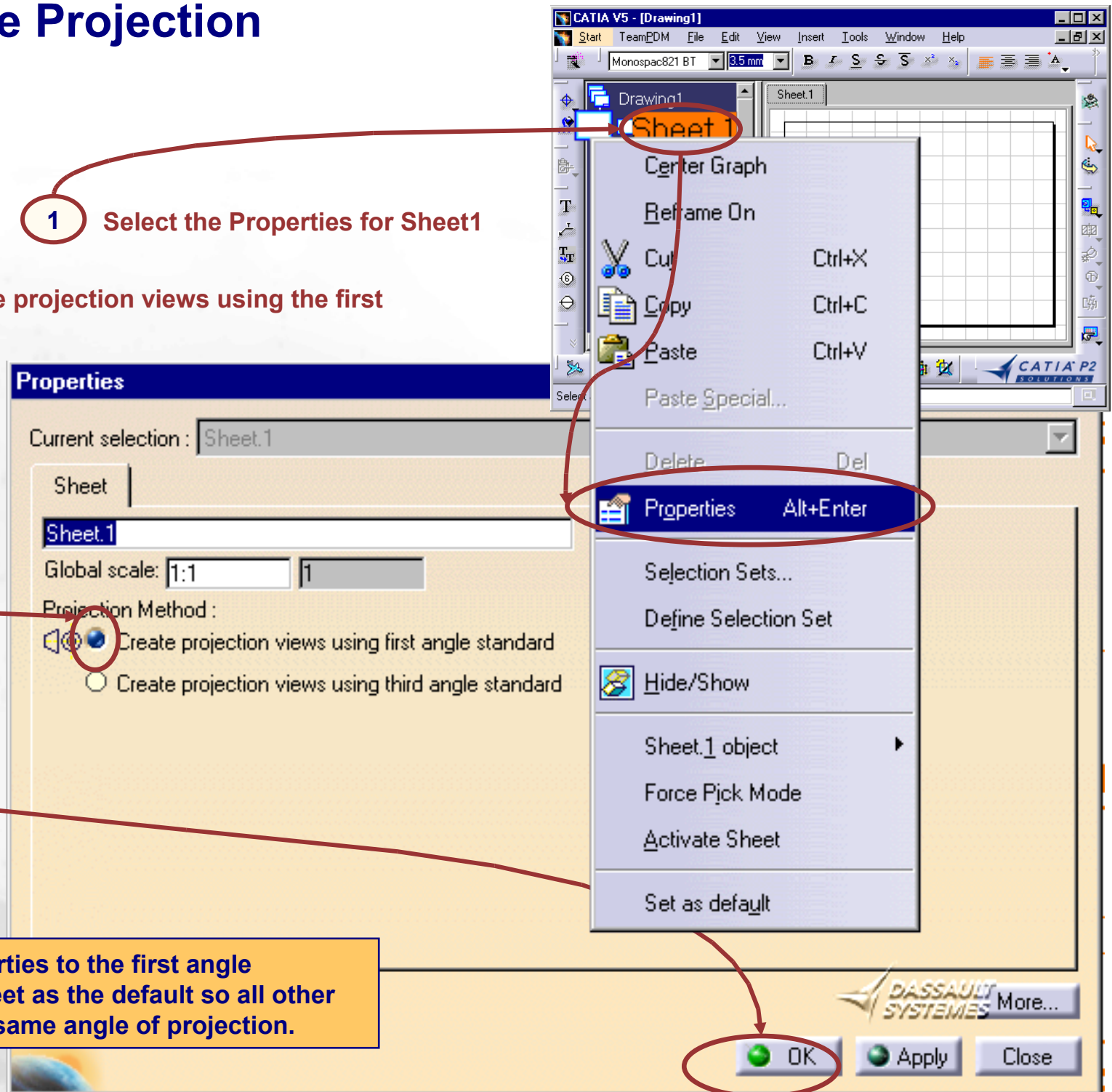
Setting First Angle Projection

1 Select the Properties for Sheet1

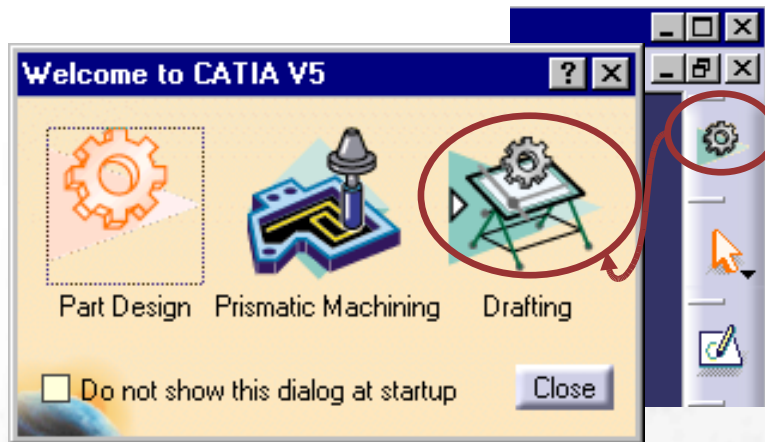
2 Select the option to create projection views using the first angle projection standard

3 Select OK to accept the changes

After changing the Sheet properties to the first angle projection standard, set the sheet as the default so all other sheets will be created with the same angle of projection.

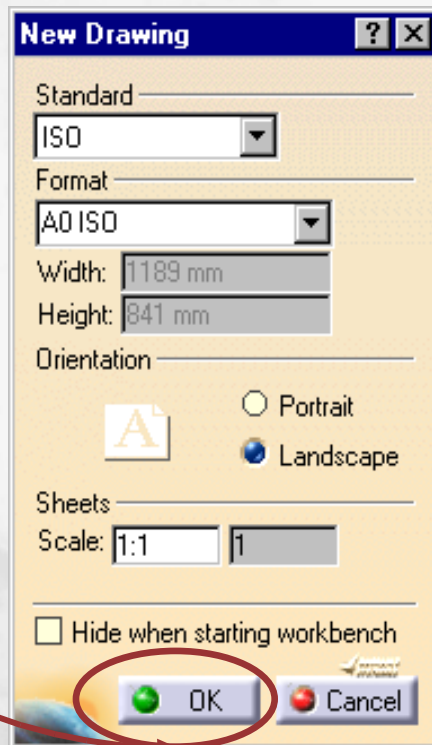


Starting a Drawing with a Blank Sheet

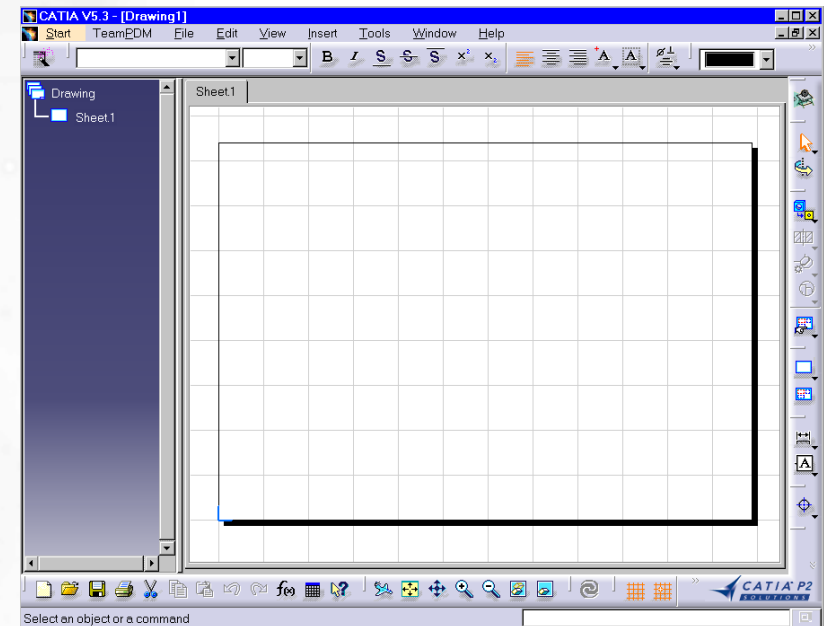


1 Change to the Drafting Workbench from the Part Workbench

2 Select the Sheet option in the Start Wizard

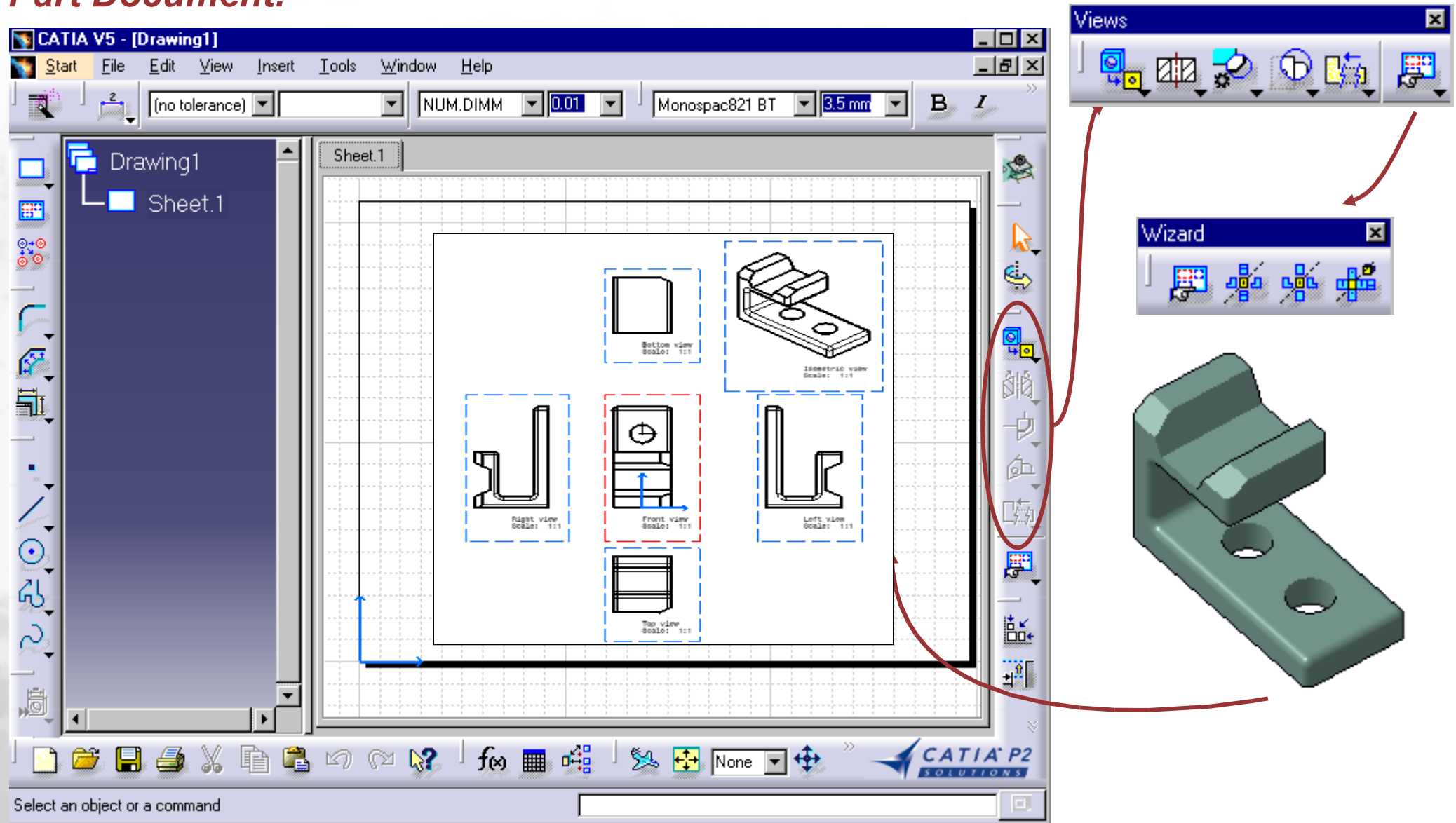


3 Select OK to accept the default drawing standards and Format Size



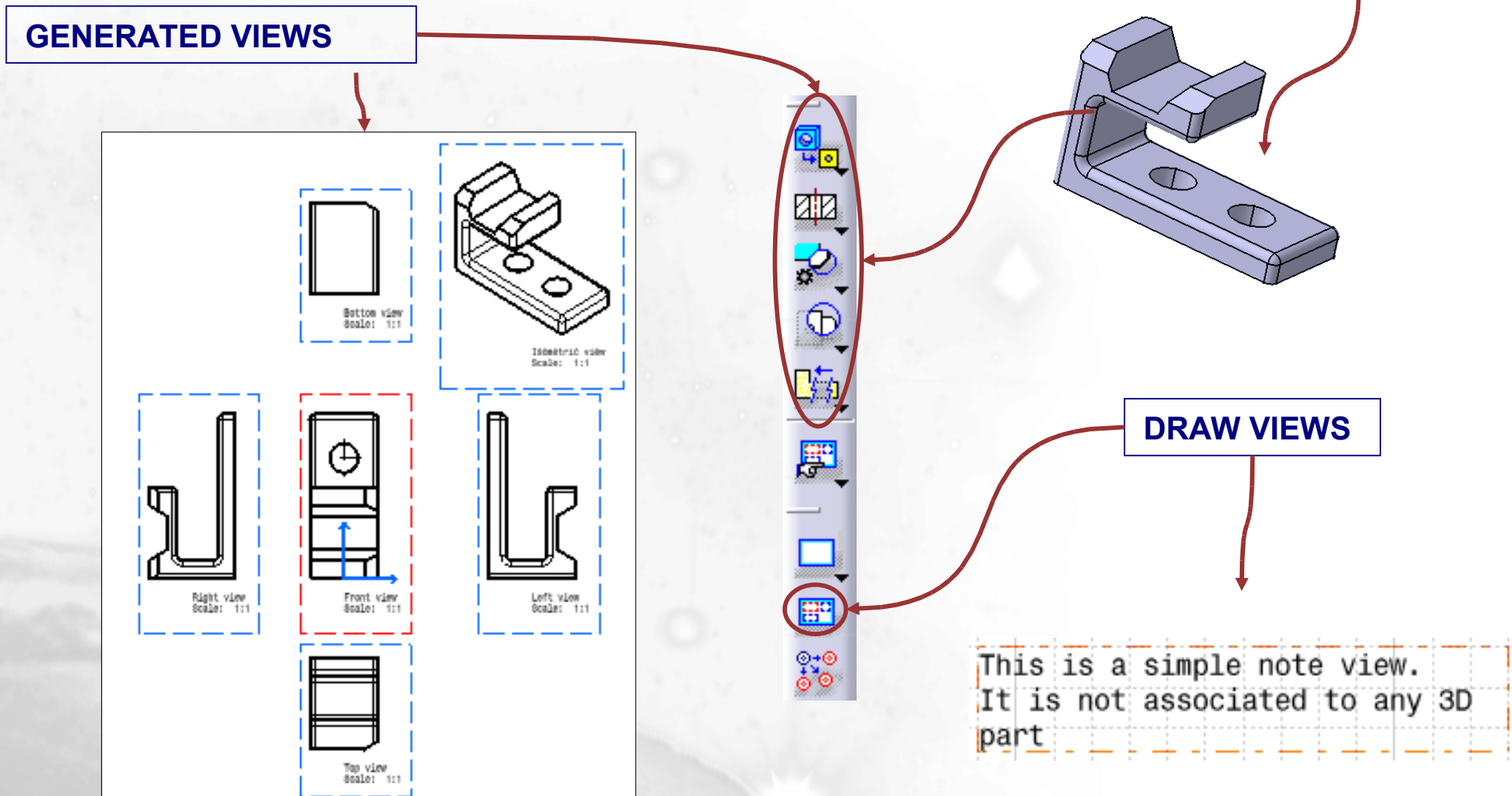
Defining the Main Views

You will learn how to define the main views for a drawing using the View Wizard. The geometry will be generated into these views from the associated Part Document.



What are the different types of Views?

- Views can be associative (linked to 3D parts) or unassociative (unlinked from a 3D part).
- Associative (Linked) Views to a 3D part are called **GENERATED VIEWS**.
- Unassociative (Unlinked) Views to a 3D part are called **DRAW VIEWS**.



Creating Views ...

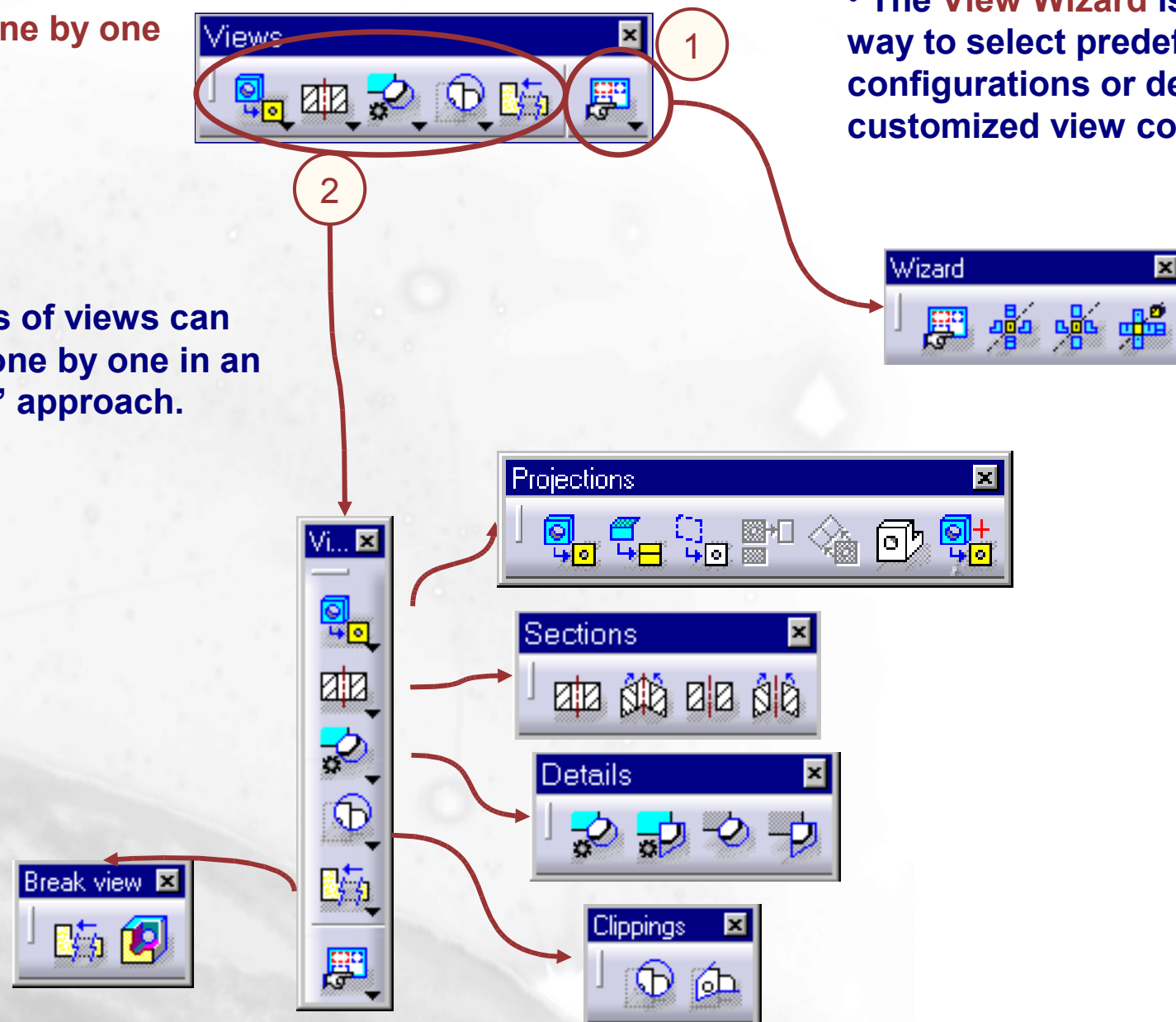
Views can be created in various ways:

1- View Wizard

2- Individually one by one

• Many types of views can be created one by one in an “as needed” approach.

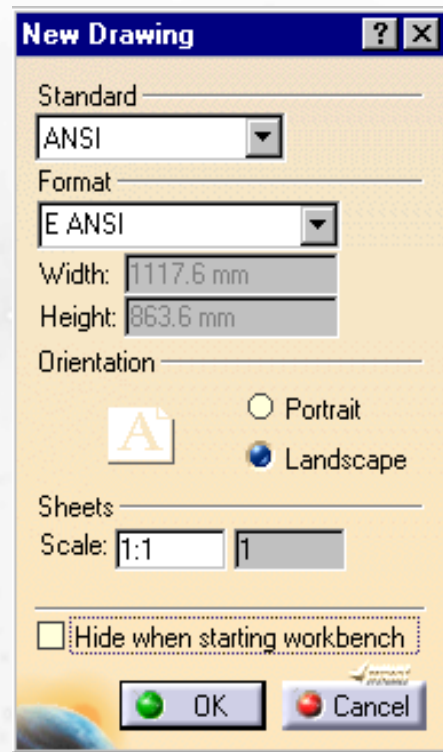
• The **View Wizard** is a quick way to select predefined view configurations or define a customized view configuration.



Creating a Front View (1/2)

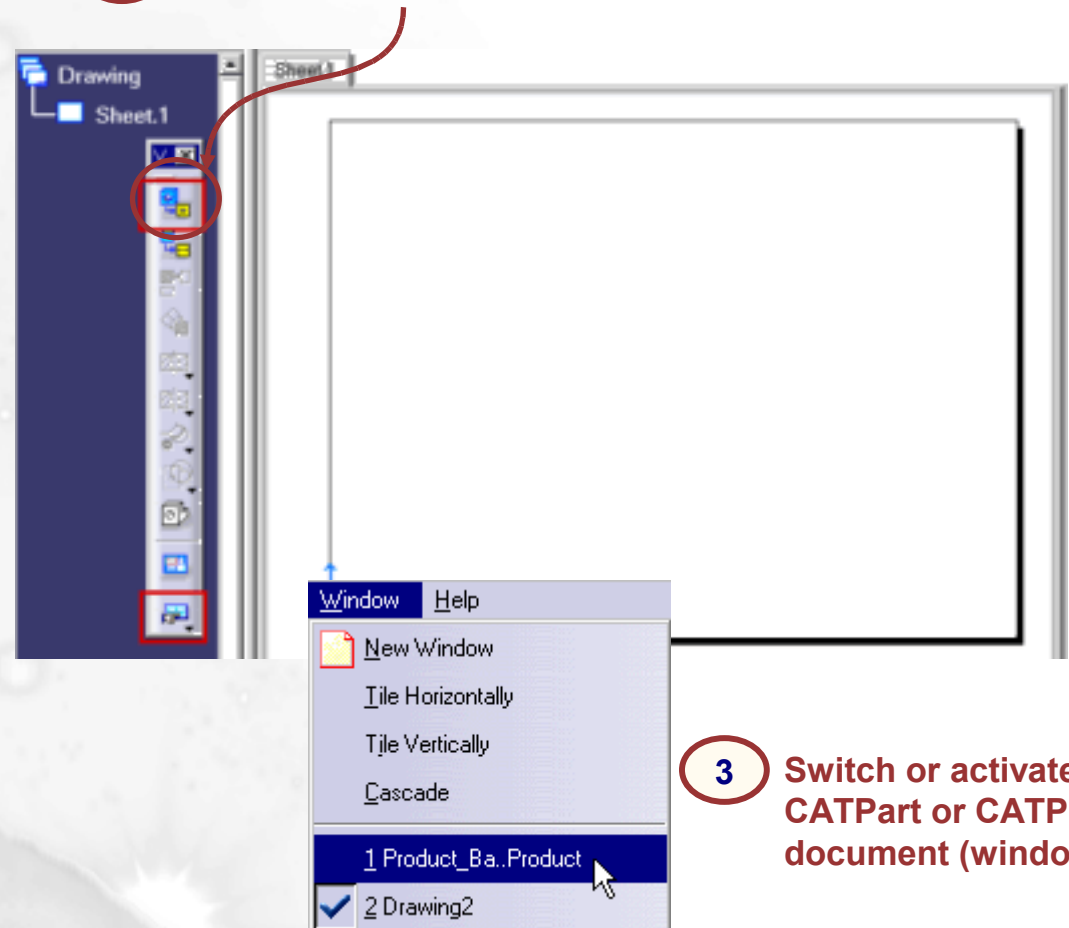
Create a front view either from a part, sub-body of a part, product, or from the sub-part of a product using a reference plane.

1 Start the drawing with a blank sheet



Note: the Front View is used as the defining view when creating projection views.

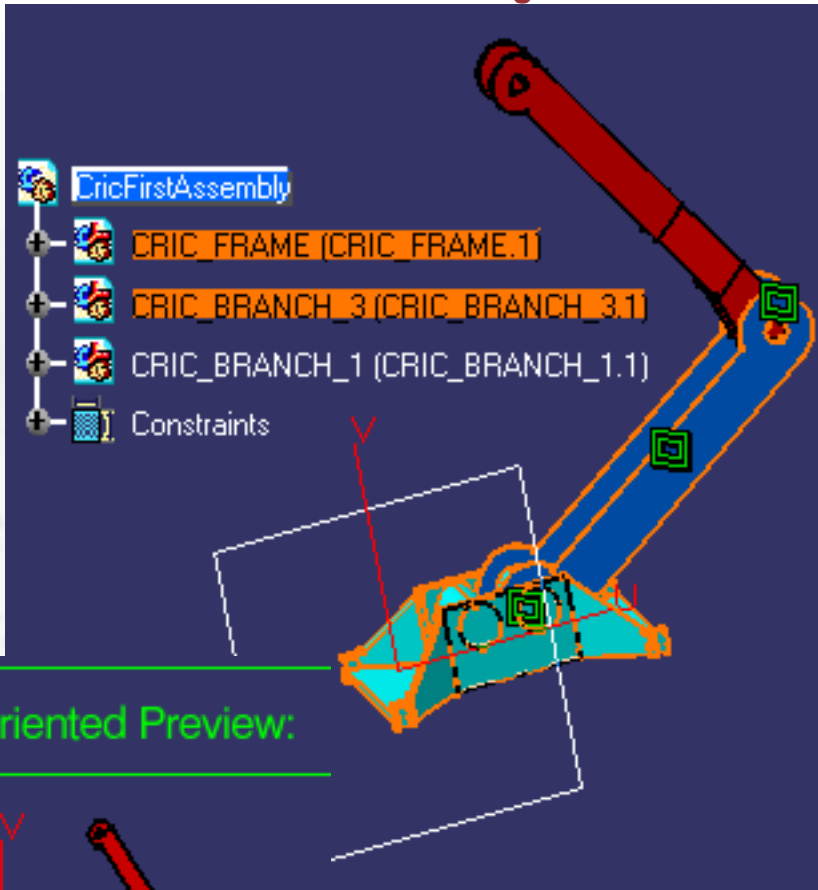
2 Select the Front View icon



3 Switch or activate the CATPart or CATProduct document (window)

Creating a Front View (2/2)

- 4 (Optional) Select the bodies or parts to display in the tree (use Ctrl key). If none is selected, all bodies or parts will be displayed. Or select a local view axis to set the view origin.

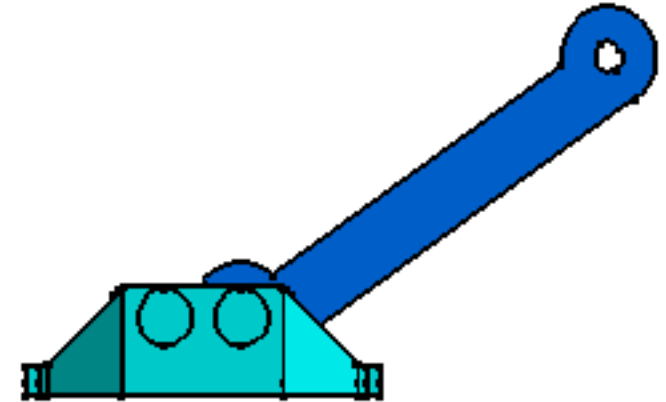


Oriented Preview:

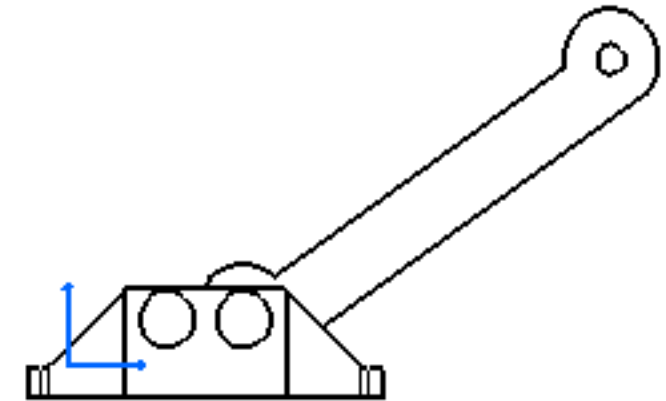
Note: When selecting a view plane, an orientation preview will be displayed when pre-selected (highlighted under the cursor)

- 5 Select your front view plane. A preview will be shown on the drawing sheet.

Preview of the front view



- 6 Select anywhere on the drawing sheet to generate the view

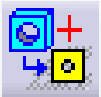


Front view
Scale: 1:1

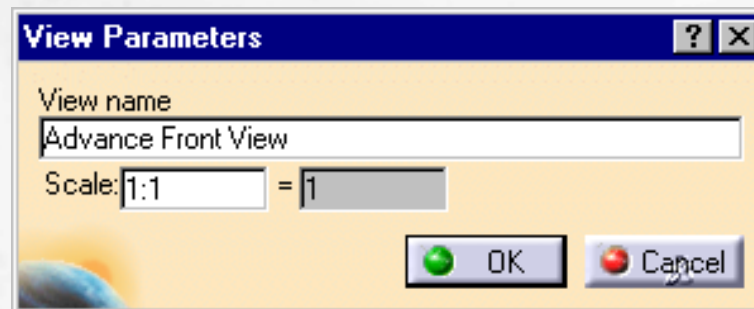
Creating an Advanced Front View

- The advance front view icon allows the creation of a Front View as shown previously, while defining several choices at view creation, such as view name and view scale.

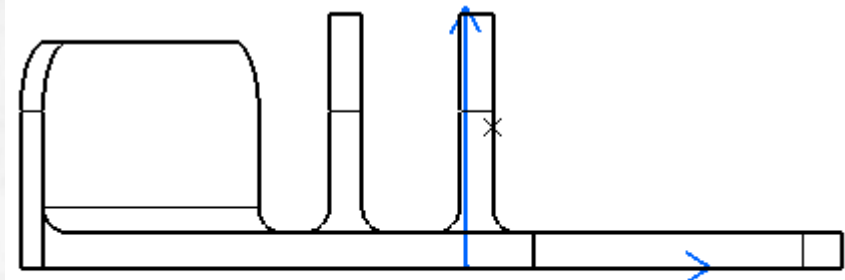
1 Select the Advance Front View icon



2 Key in the desired view name and scale. Select the OK button.



3 Complete the Front View as previously shown



Advance Front View
Scale: 1:1

25

Rotating the Front View Background with the View Manipulators (1/2)

Before accepting the view, it can be reoriented with the view manipulator

clicking up arrow

clicking right arrow

clicking center left arrow

dragging green handle

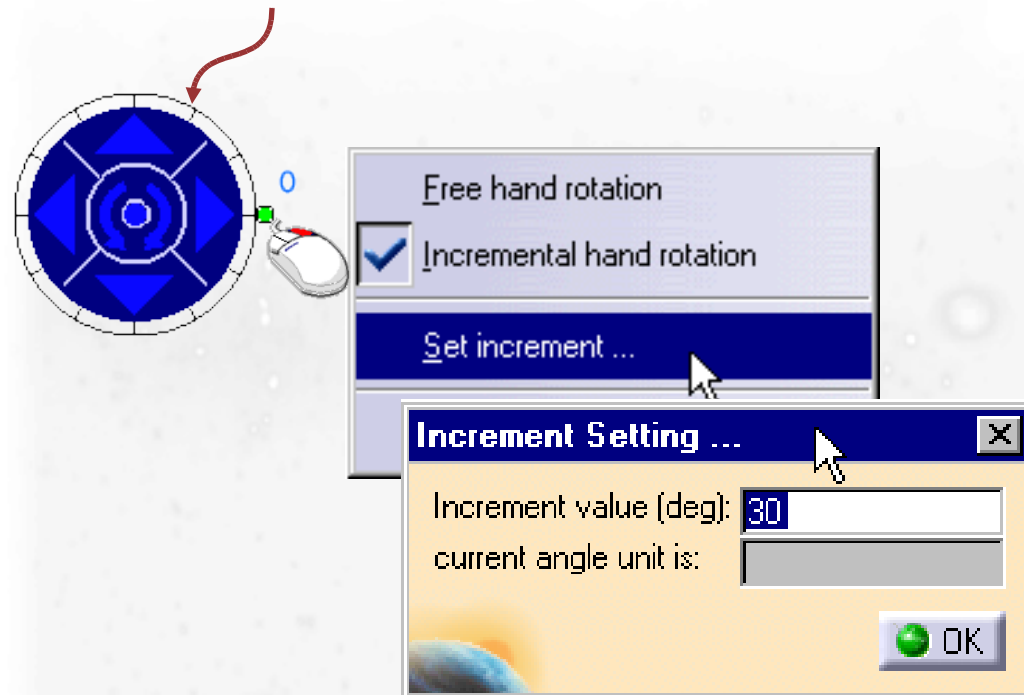
Front view
Scale :1

- **Up, down, right and left** arrows flip background plane view **90 degrees**.
- **Center left and right** arrows rotate view **30 degrees** in the same plane. The 30 degrees increment can be changed with the contextual menu on the dial.
- The **green handle(dot)** rotates view freely on the same plane.
- When finished, click on dial center or anywhere on sheet to generate the Front View.

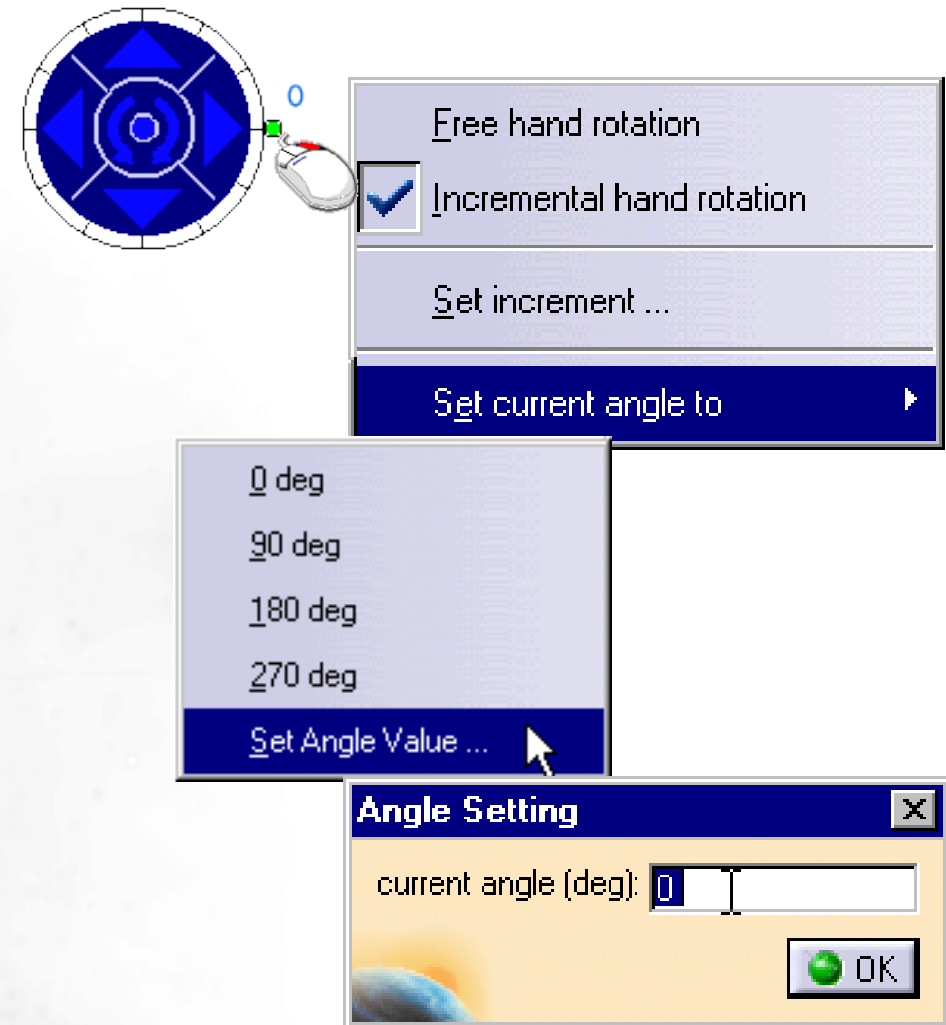
Rotating the Front View Background with the View Manipulators (2/2)

View Manipulator contextual menu settings

• Set the increment of Rotation



• Set the angle of Rotation

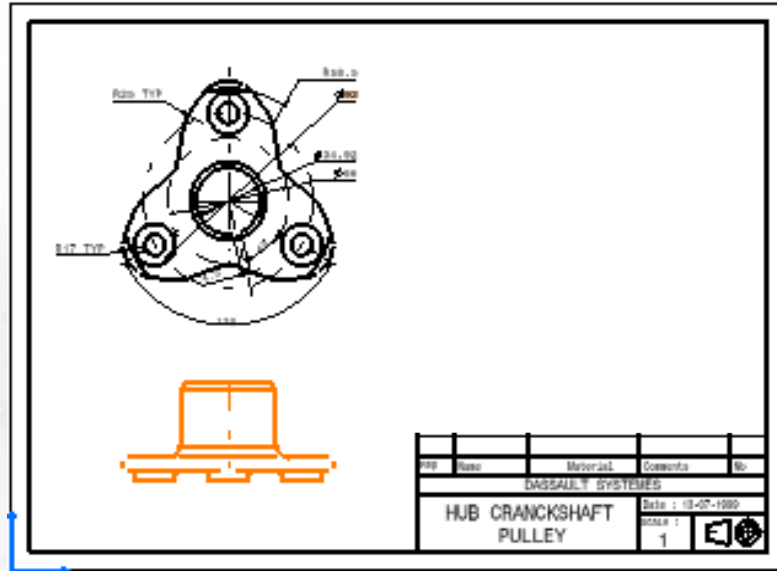


• Set free Rotation

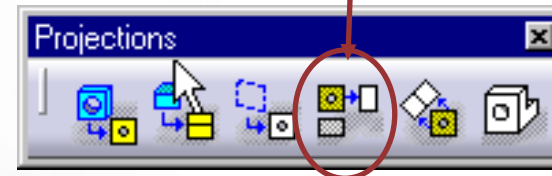


Adding a Projection View on a Drawing

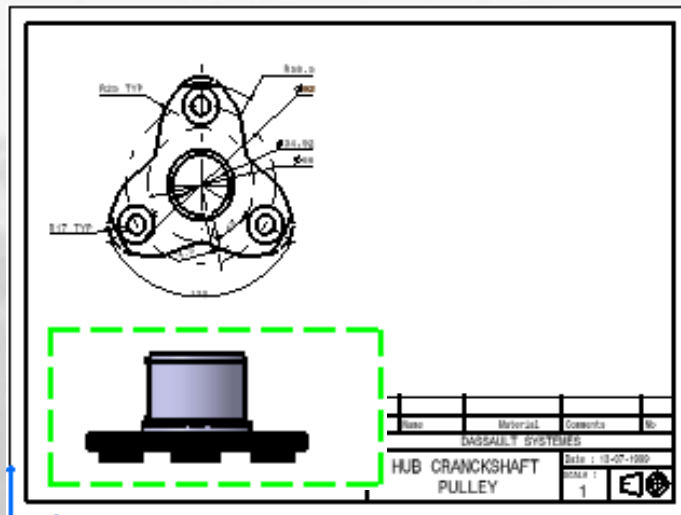
1 To add a view on this drawing



2 Activate the view (front) to project from (blue axes on and underlined in tree). Select the View Projection icon



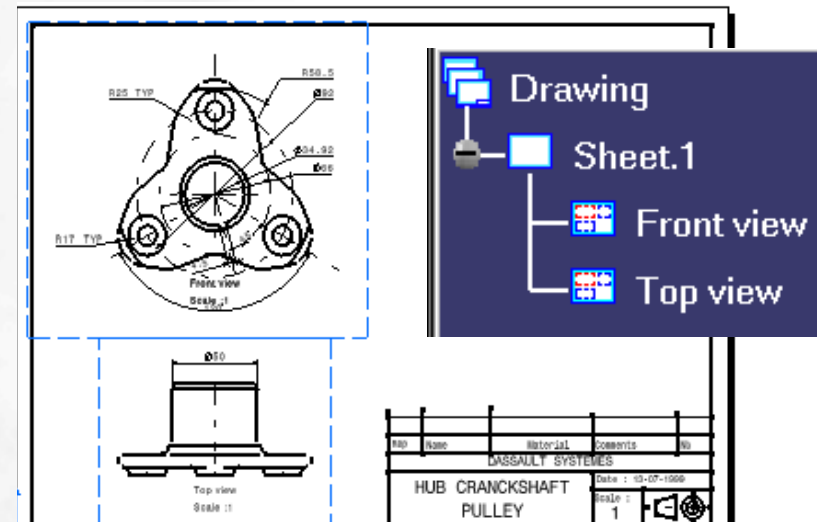
3 Move cursor where View is to be projected



A preview of the projection view appears.



The Top view (ISO Standard) is generated with the default projection standard set to first angle projection. With the ANSI Standard, a Bottom view is generated.



The View Wizard

The **VIEW WIZARD** provides the ability to quickly create a variety of standard view configurations or build a specific view configuration.



All Views

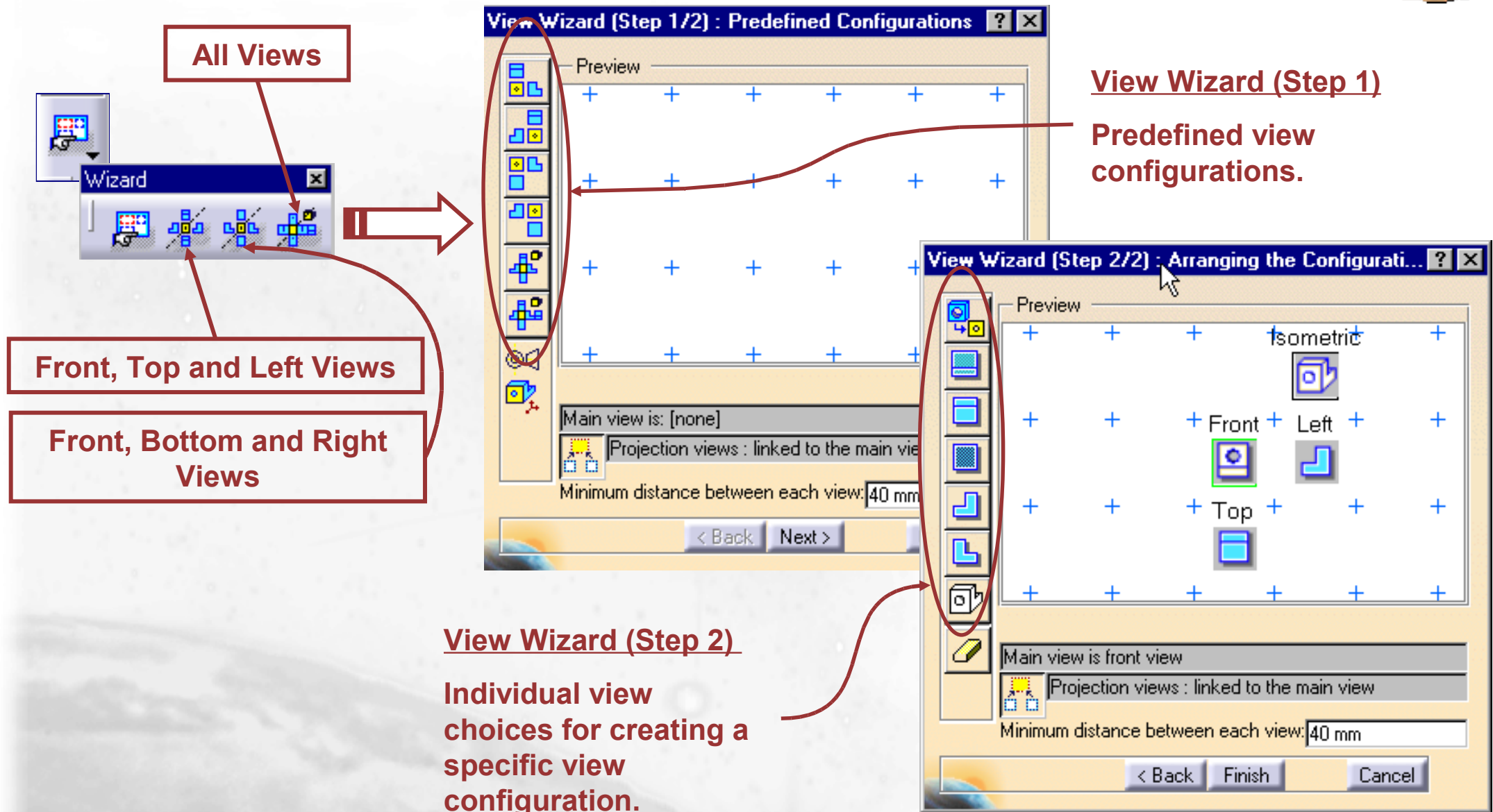
Front, Top and Left Views

Front, Bottom and Right Views

View Wizard (Step 1)
Predefined view configurations.

View Wizard (Step 2)
Individual view choices for creating a specific view configuration.

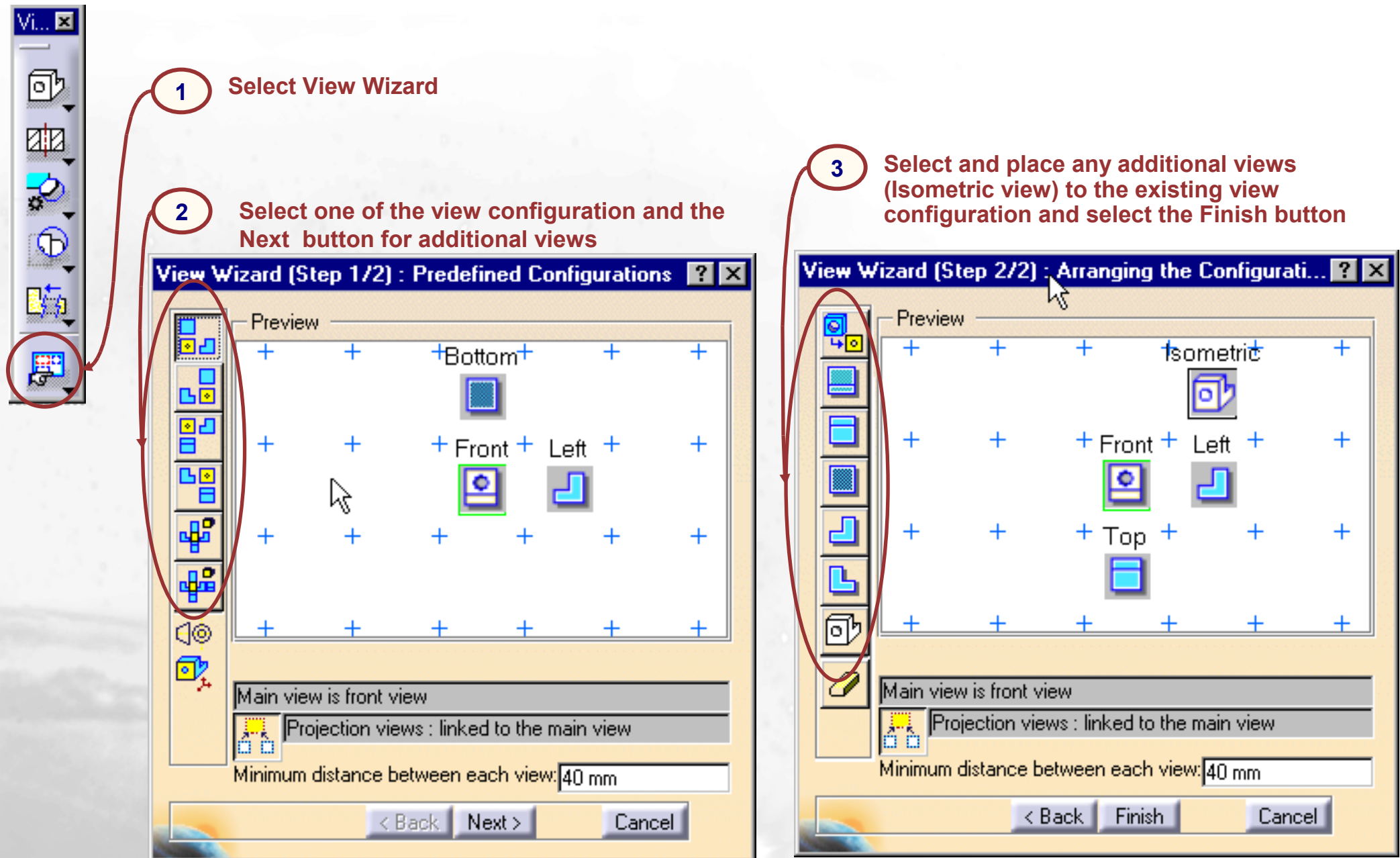
Add, delete and rearrange the views as needed.



The image displays two screenshots of the View Wizard software interface. The first screenshot, titled 'View Wizard (Step 1/2) : Predefined Configurations', shows a grid of predefined view configurations. A red oval highlights the left sidebar containing various view icons. Red arrows point from text boxes on the left to specific icons in this sidebar: 'All Views' points to the top icon, 'Front, Top and Left Views' points to the second icon, and 'Front, Bottom and Right Views' points to the third icon. The main area of the dialog shows a preview grid with blue plus signs. The second screenshot, titled 'View Wizard (Step 2/2) : Arranging the Configurati...', shows a more detailed configuration screen. A red oval highlights the left sidebar, which now includes options like 'Isometric', 'Front', 'Left', 'Top', and 'Bottom'. Red arrows point from text boxes on the left to specific icons in this sidebar: 'Front, Top and Left Views' points to the 'Front' icon, and 'Front, Bottom and Right Views' points to the 'Bottom' icon. The main area of the dialog shows a preview grid with blue plus signs and the text 'Main view is front view' and 'Projection views : linked to the main view'. At the bottom of the dialog, there are buttons for '< Back', 'Finish', and 'Cancel'.

Generating Main Views with the View Wizard Quick Method (1/3)

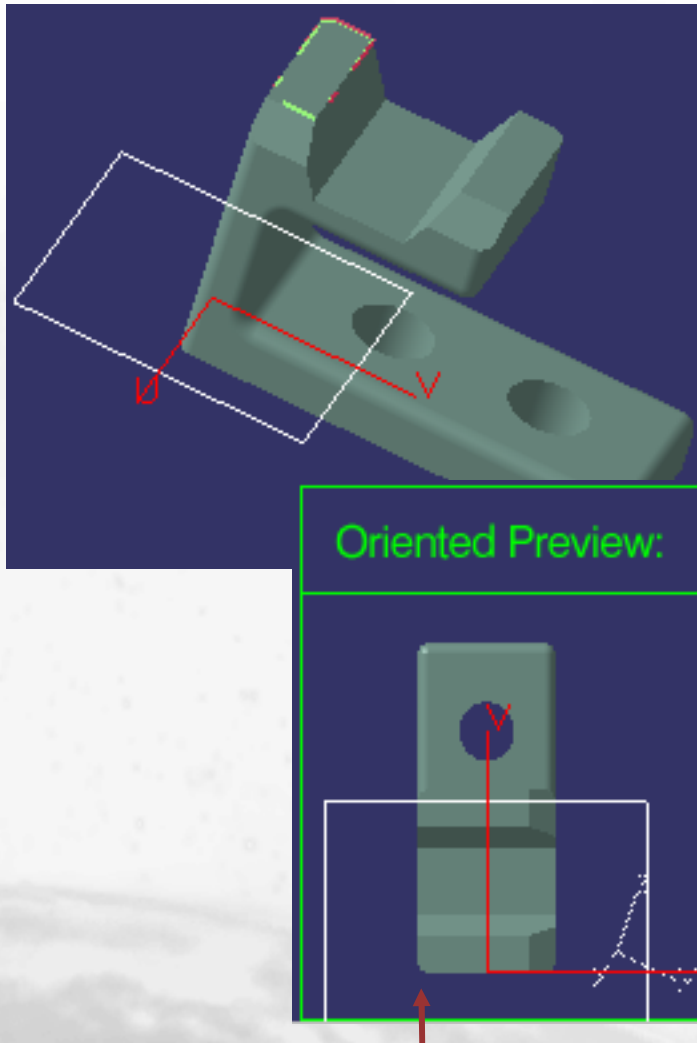
The process to use the view wizard to quickly build views on a blank drawing sheet.



Generating Main Views with the View Wizard Quick Method (2/3)

4

Select the face on the 3D part for the Front view background plane.



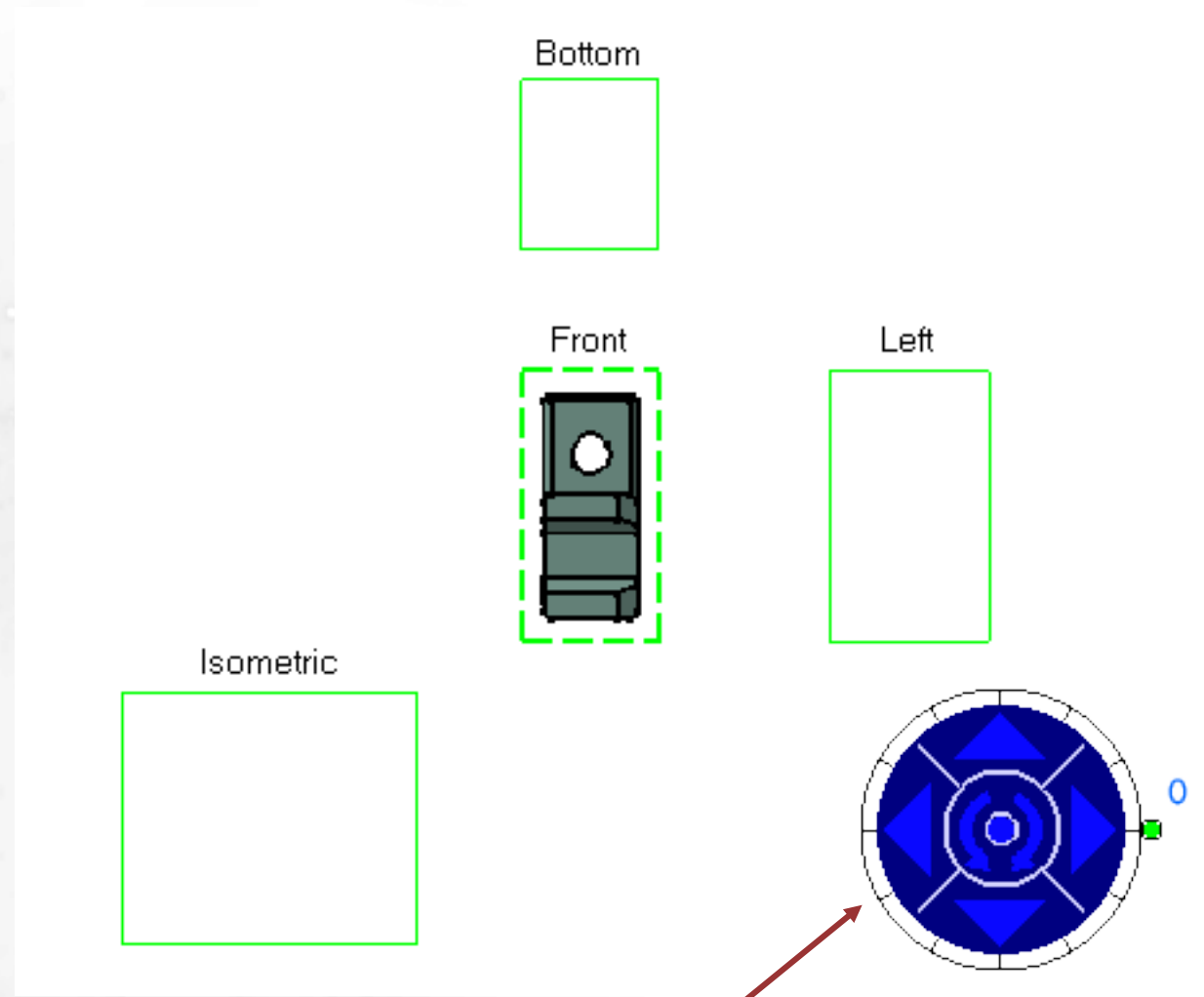
A preview of the Front view appears in Part Design Workbench when pre-selected (highlighted) by the cursor

5

A preview of your view configuration appears on the drawing sheet.



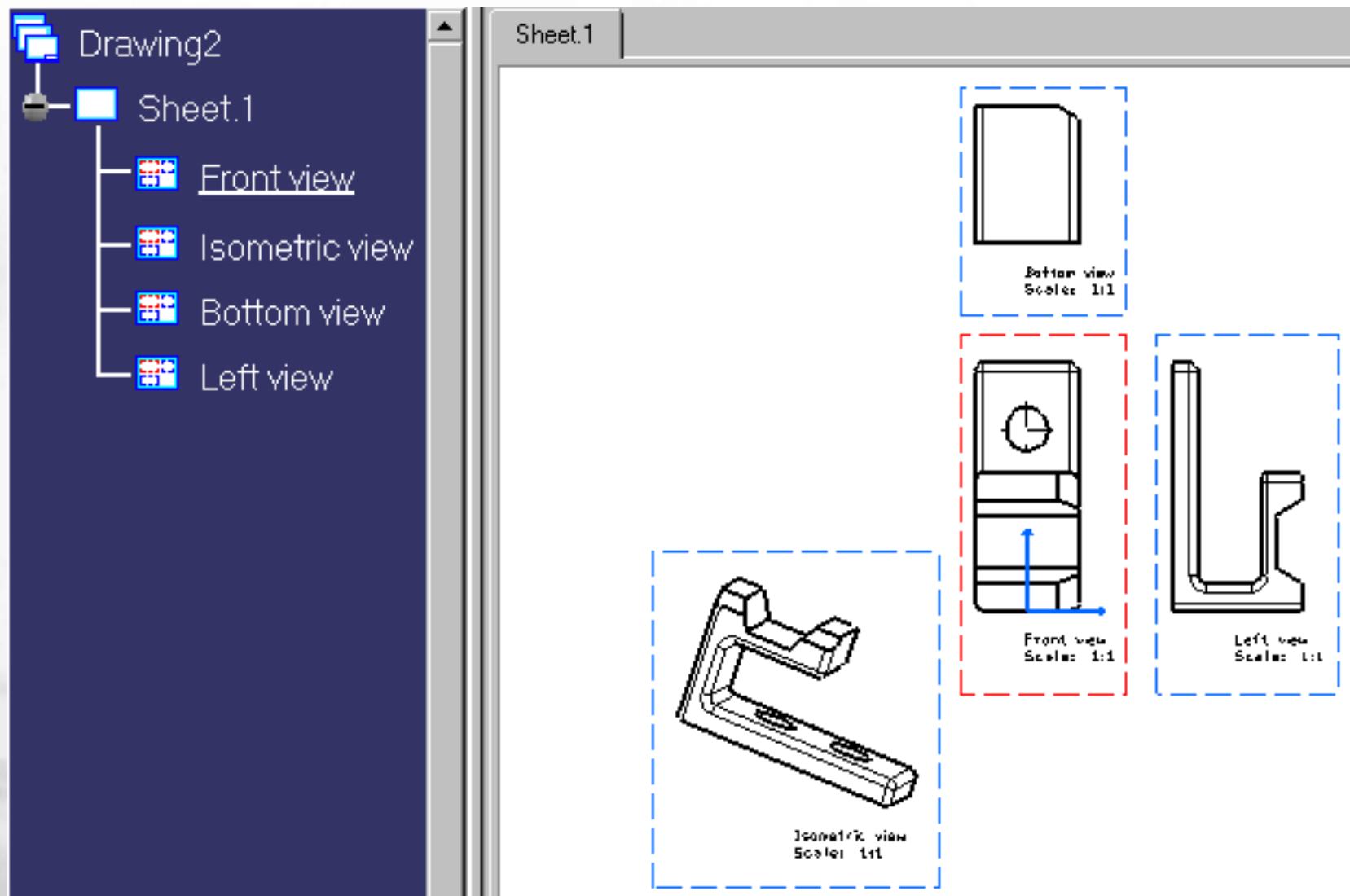
To preview a view, move the cursor to the green preview box of the desired view



If needed, use the View Manipulator to reorient the Front view.

Generating Main Views with the View Wizard Quick Method (3/3)

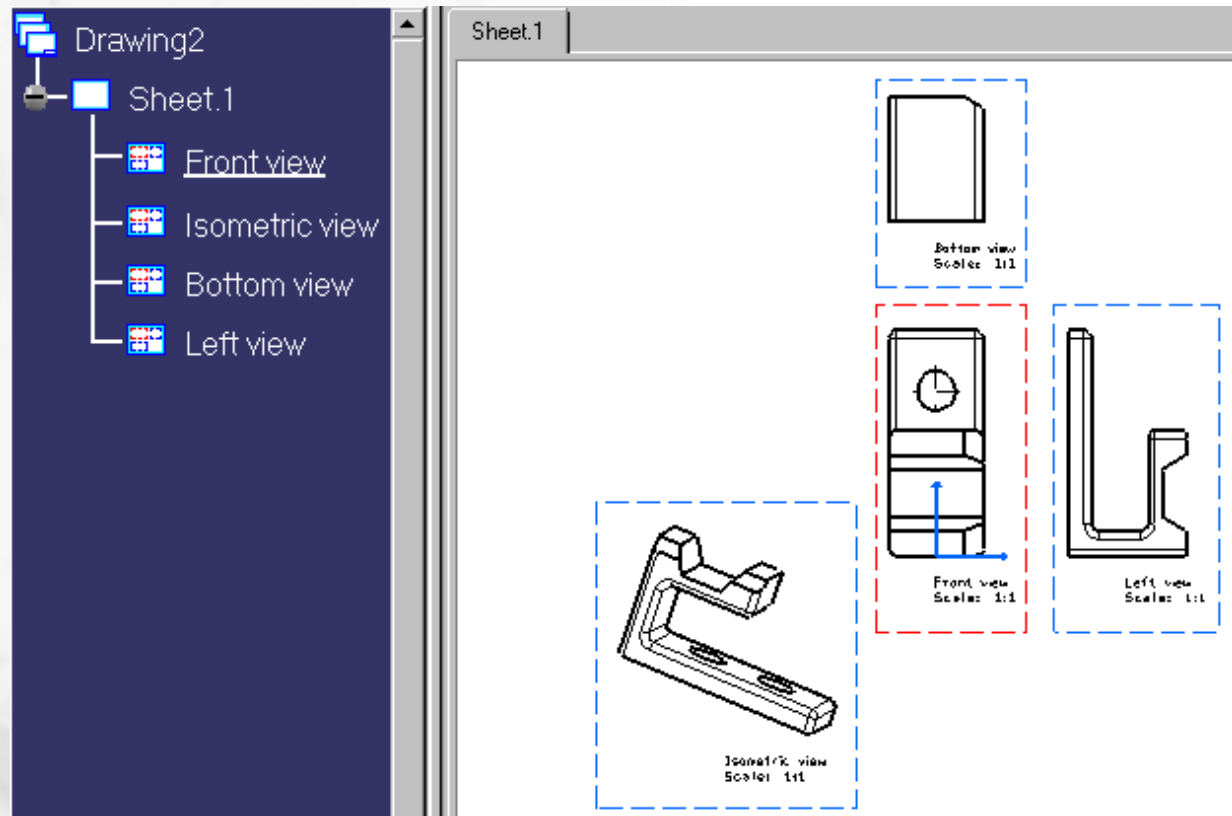
- 6 Select anywhere on the drawing to generate and modify the individual view location as needed



To Sum Up ...

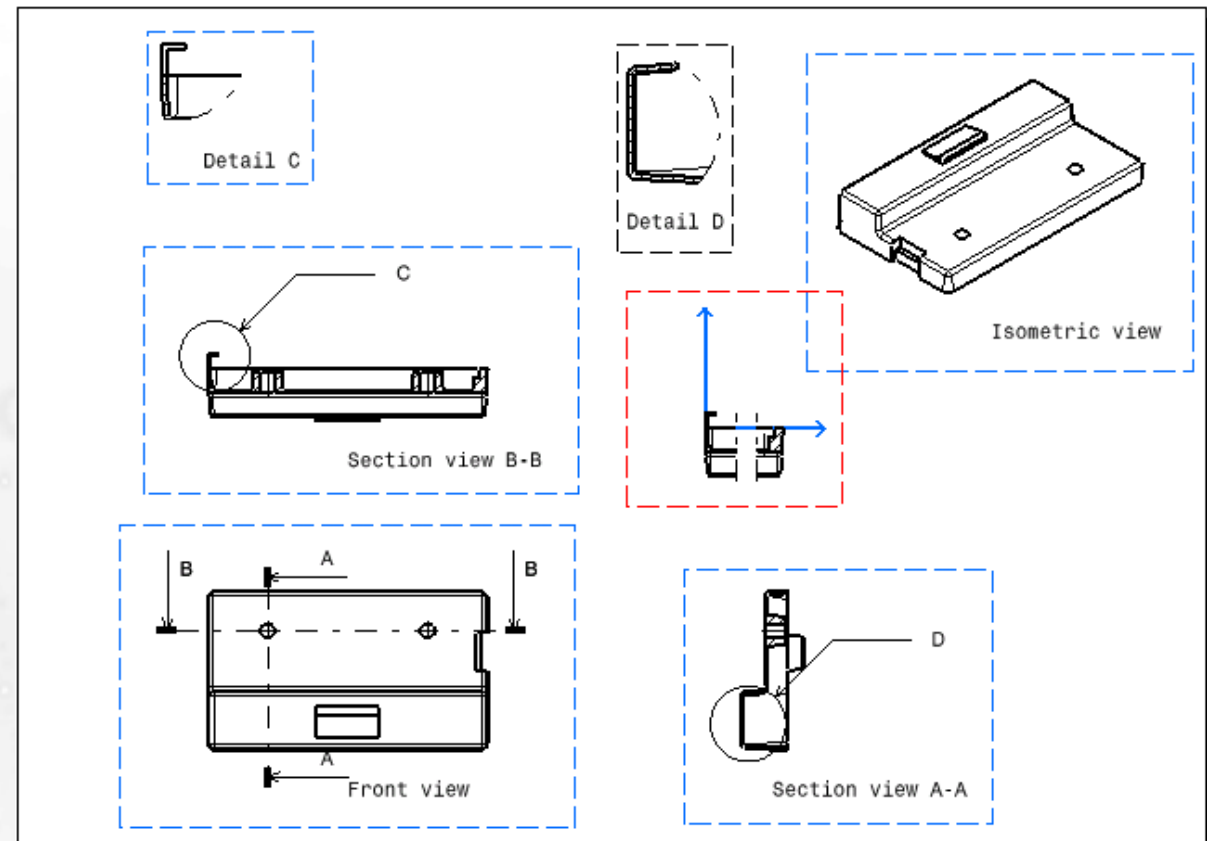
In this lesson you have seen...

- How to generate the geometry of the main views either automatically or manually while the part to be drawn is in the Part Design workbench
- Ways to define views on a blank drawing



Additional View Generation

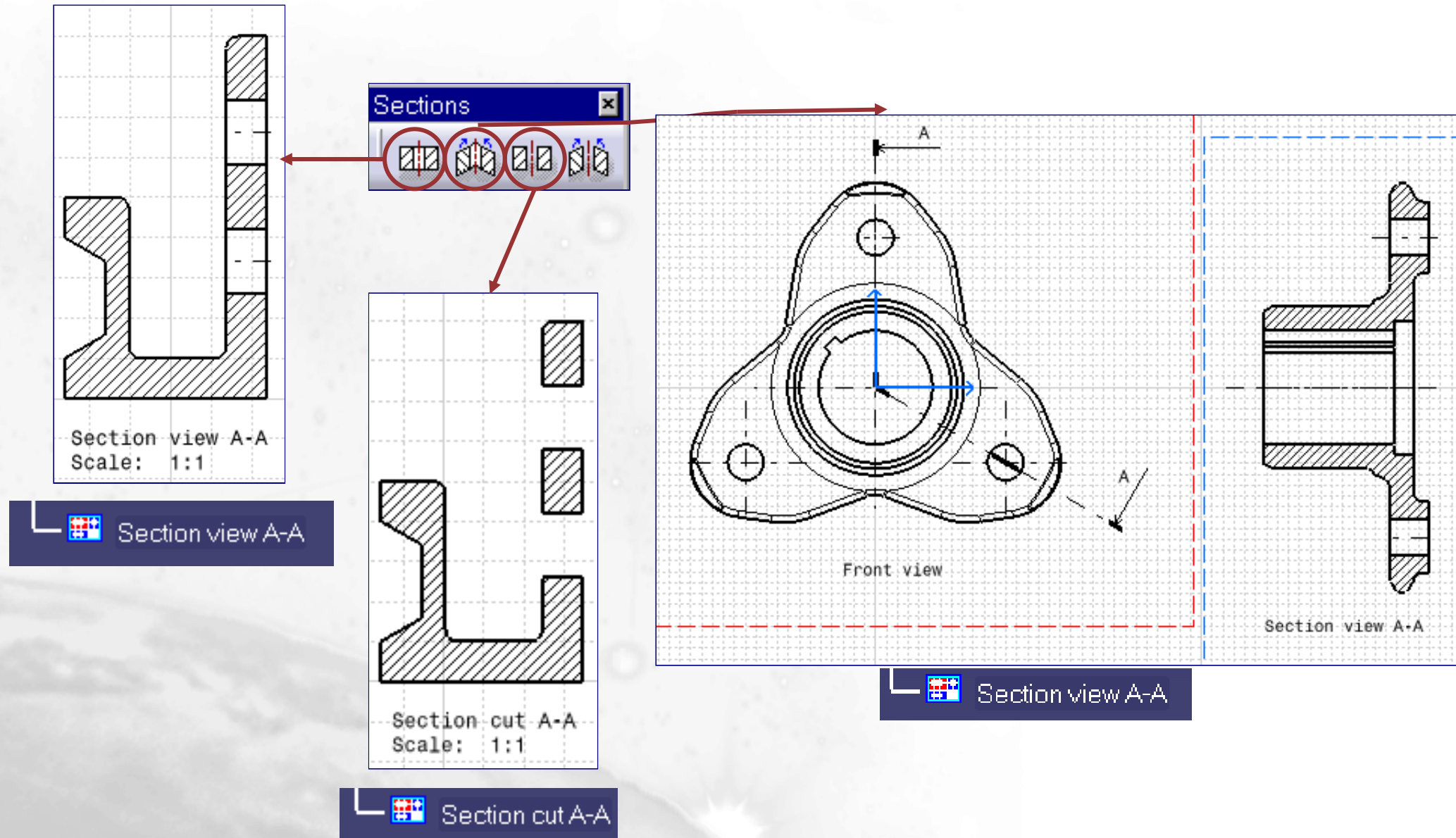
In this lesson you will generate section, detail, auxiliary, isometric and broken views on a drawing.



- Section Views and Cuts
- Secondary Views: Detail, Clipping, Broken, Breakout, Auxiliary, Isometric and Unfolded Views

Adding Section Views and Cuts

You will learn how to add section views and section cuts to the drawing

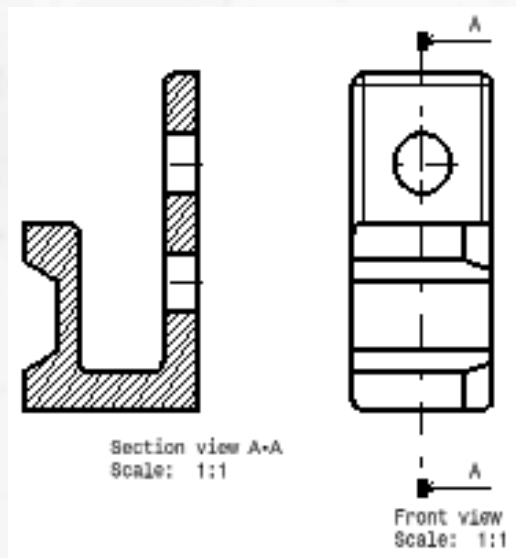


What are the different types of Sections Views ?

There are 3 main types of sectional views; "full section", "offset" or "aligned" depending on how the cutting plane is defined.

Offset Sections

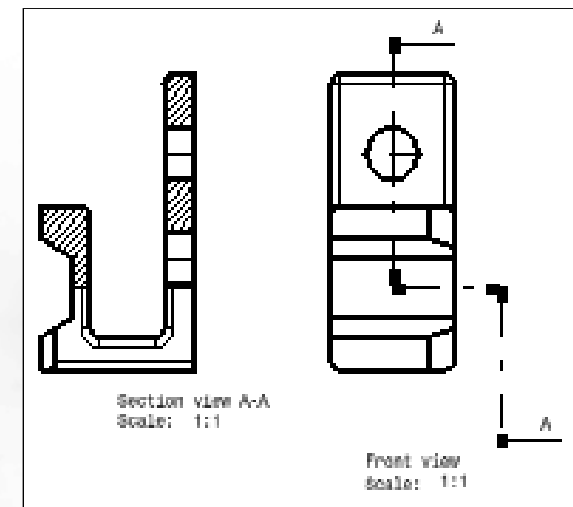
Simple Offset Section



Simple offset section - a full section with a simple single cutting plane **fully** passing through the part

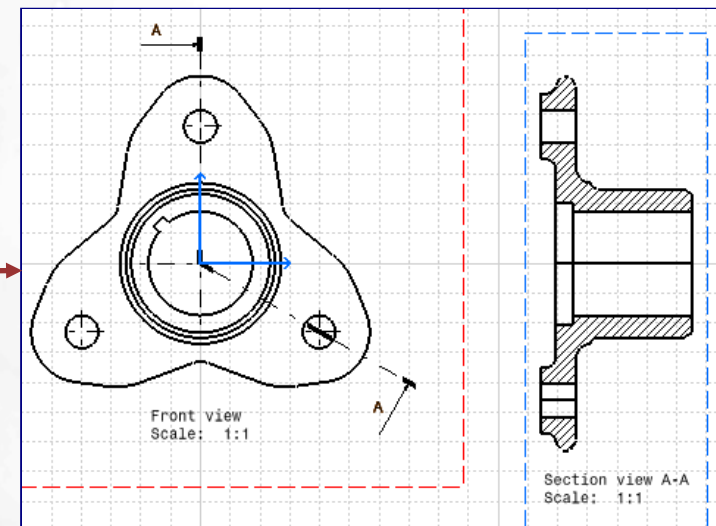
Complex offset section - a full section in which the cutting plane is **offset** to pass through important features

Complex Offset Section



Aligned Sections

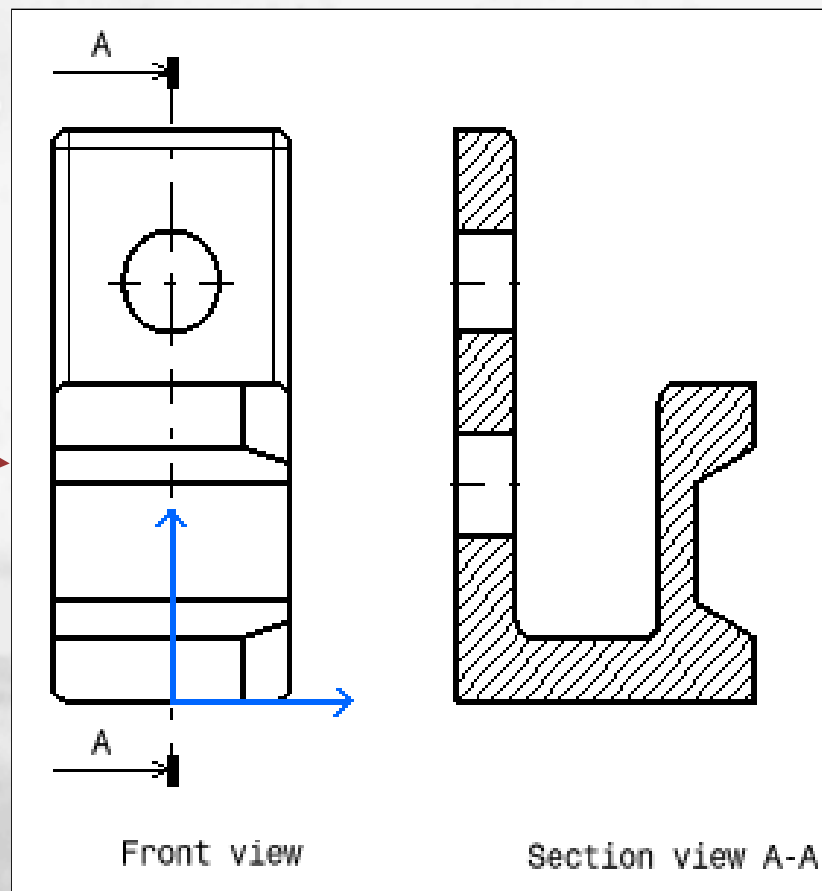
Aligned sections, also called "revolved" sections, has a cutting plane revolving around an axis



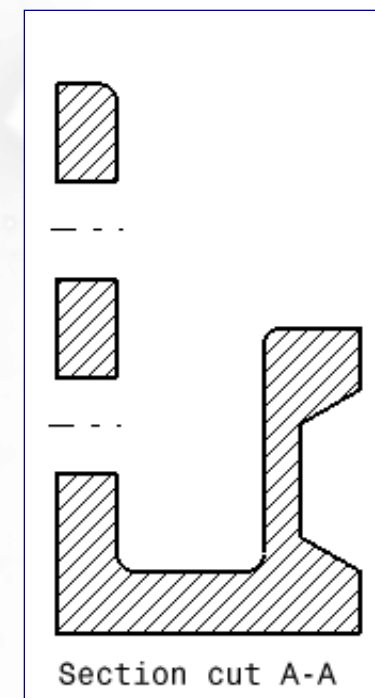
Section Views and Section Cuts ...

Reminder of the differences of a **section view** (Offset or Aligned) and **section cut** (Offset or Aligned)

Section View: A view of the cutting plane and any geometry that extends beyond the cutting plane in the direction of the sight (arrows).

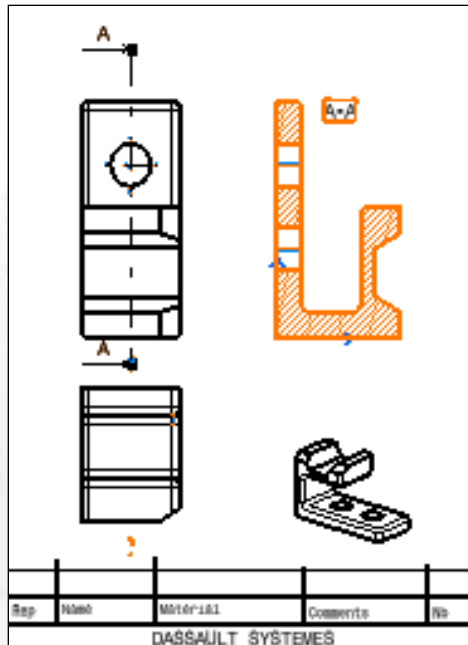


Section Cut: A view of only the material that the cutting edge touches when passing through the part.

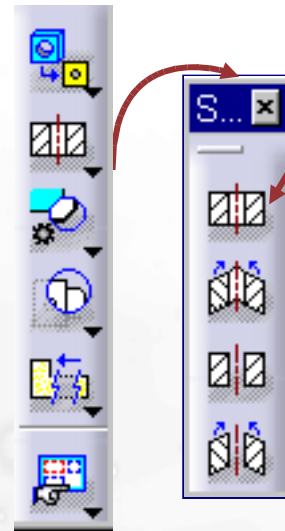


Adding a Simple Section View on a Drawing

1 To add a section view on this drawing



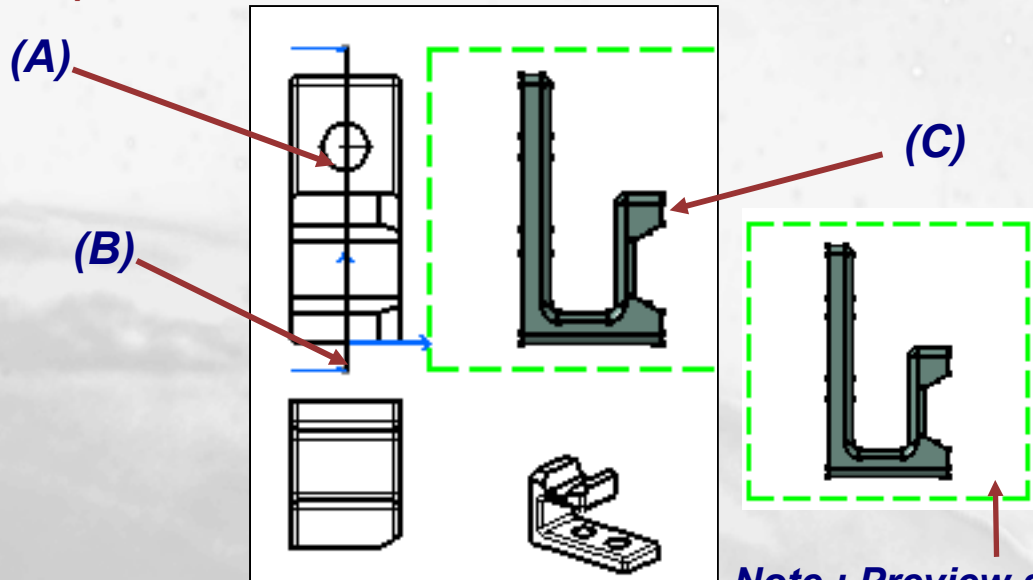
2 Activate the front view and select the desired Section View icon



Front view is active when the blue axis is visible and the view name is underlined in the tree. If the Frame option is set on then the frame color will be red around the active view.

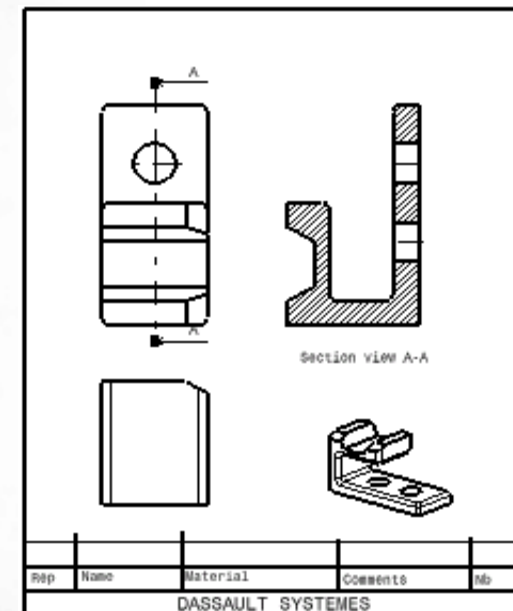
Use the section cut icon and the same process to make a section cut view

3 Select the circle A, double click at B, then place the view C



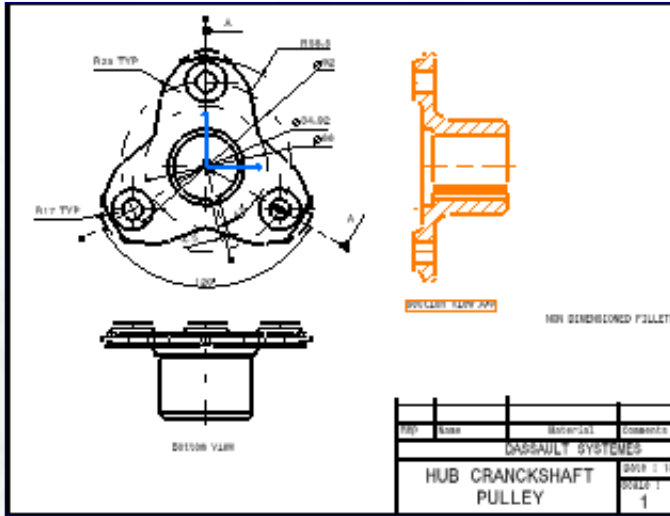
Note : Preview appears

4 The section is added

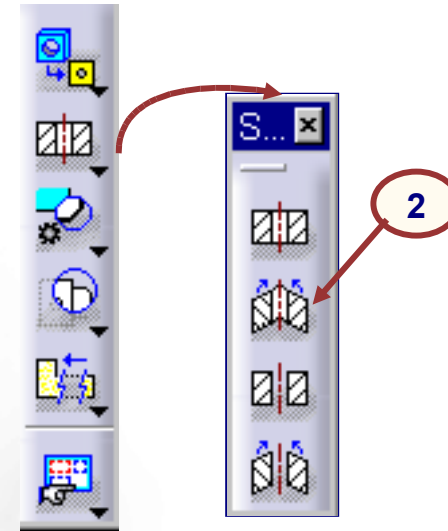


Adding an Aligned Section View on a Drawing

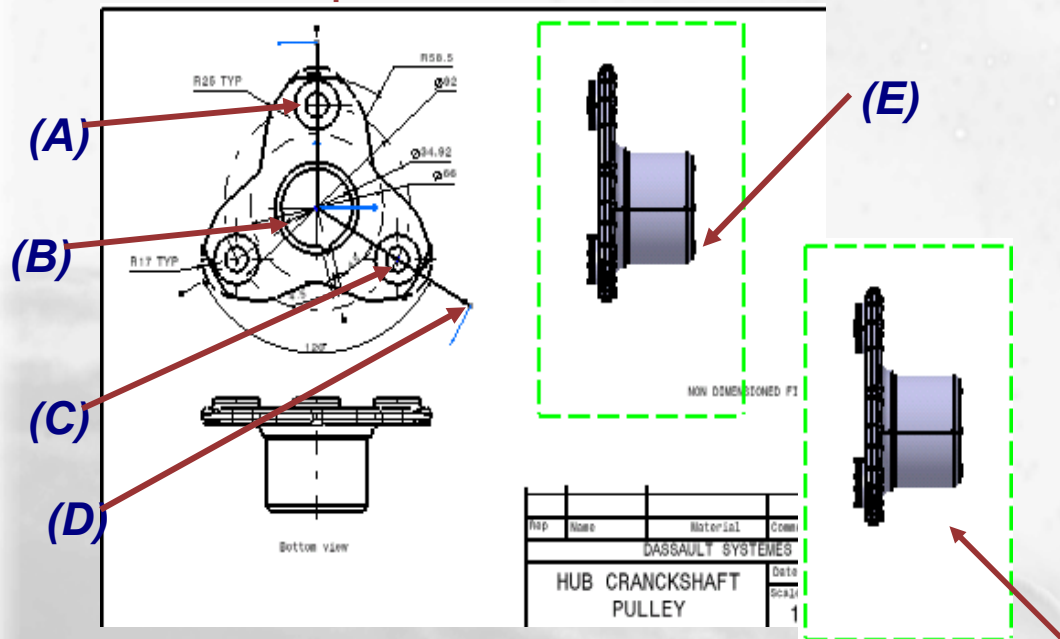
- 1 To add an aligned section view on this drawing



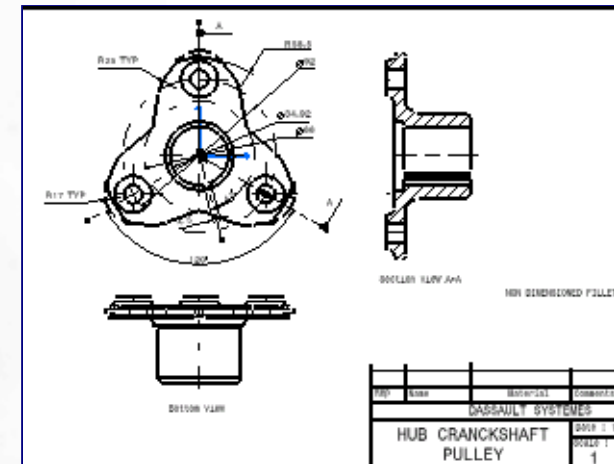
- 2 Activate the front view, select the desired Aligned Section icon



- 3 Select the circle A, select the circle B, select the circle C, double click at D and place the view at E

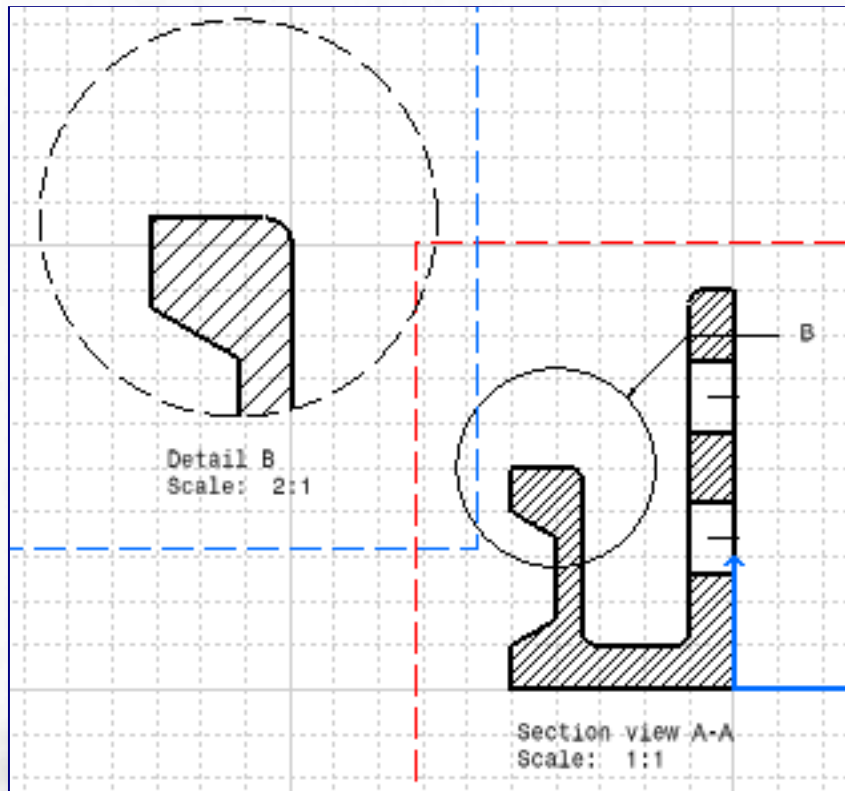


- 4 The aligned section geometry is created

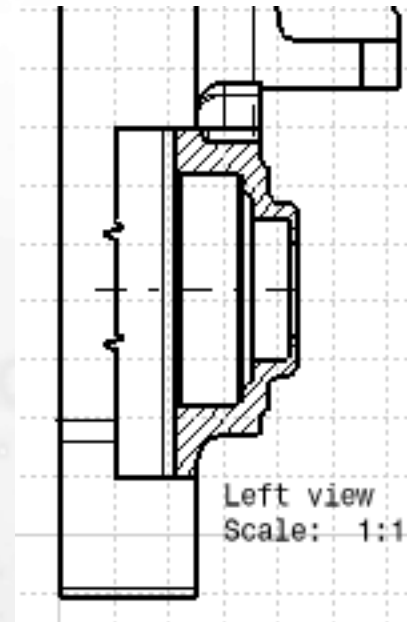


Adding Secondary Views

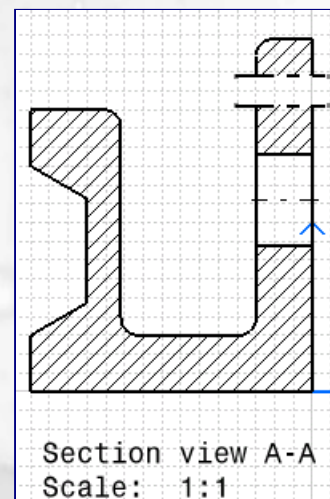
You will learn how to add secondary views



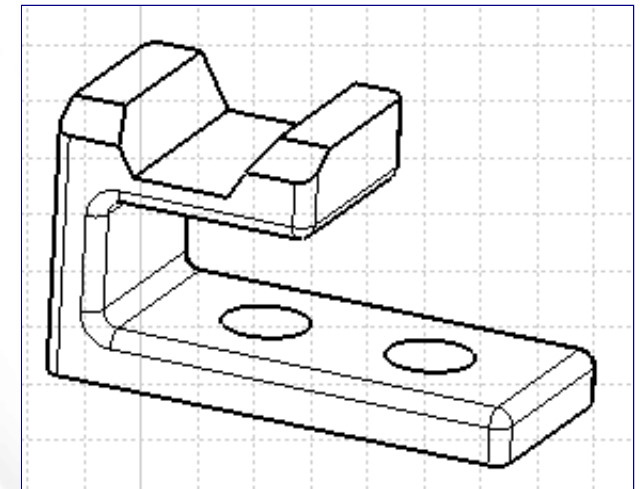
Detail Views



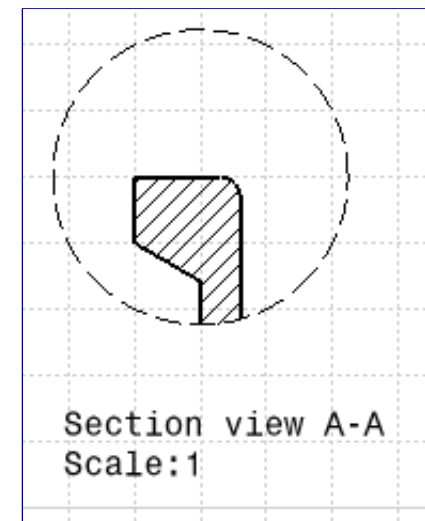
Breakout Views



Broken Views



Isometric Views



Clipped Views

What are secondary Views ? (1/2)

Secondary Views are added to improve the clarity of the description of a part through better visualization and/or to aid in dimensioning. There are seven types of secondary views.

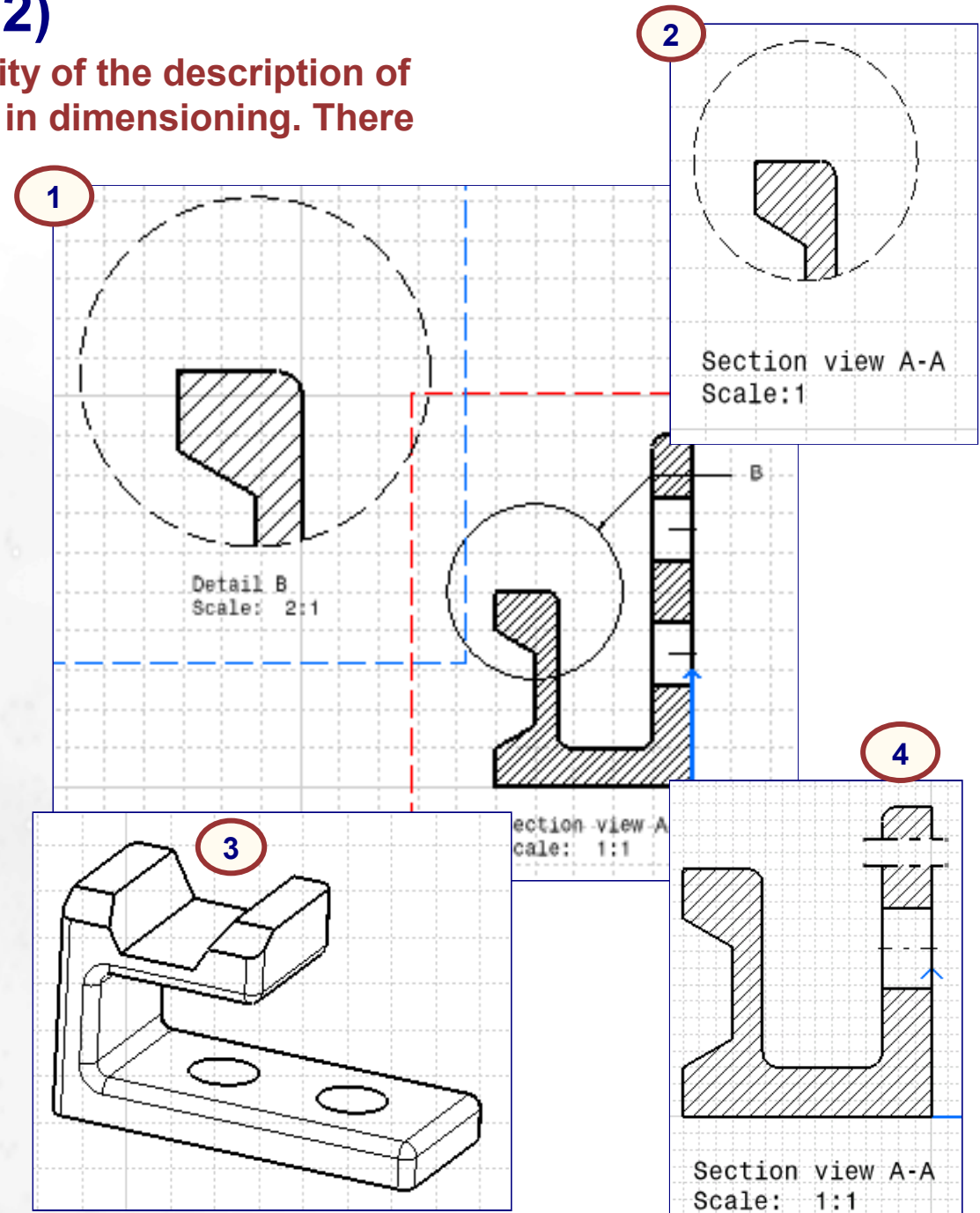
Secondary Views:

1) **Detail Views:** A detail view is defined by a "callout" on an existing view around the area to be enlarge for the new view. The callout can be a circle or an free-hand sketched profile.

2) **Clipped Views:** A clipping view is defined by a "callout" on an existing view. The callout can be a circle or an free-hand sketched profile and the clipping will remove all the existing view's geometry that is not in the callout.

3) **Isometric Views:** An isometric view is the projection of the 3D part and its relation to the current rotation of the XYZ plane.

4) **Broken Views:** A broken view is defined by adding break lines to determine an area of the view that will be removed. Views can be broken horizontally or vertically.



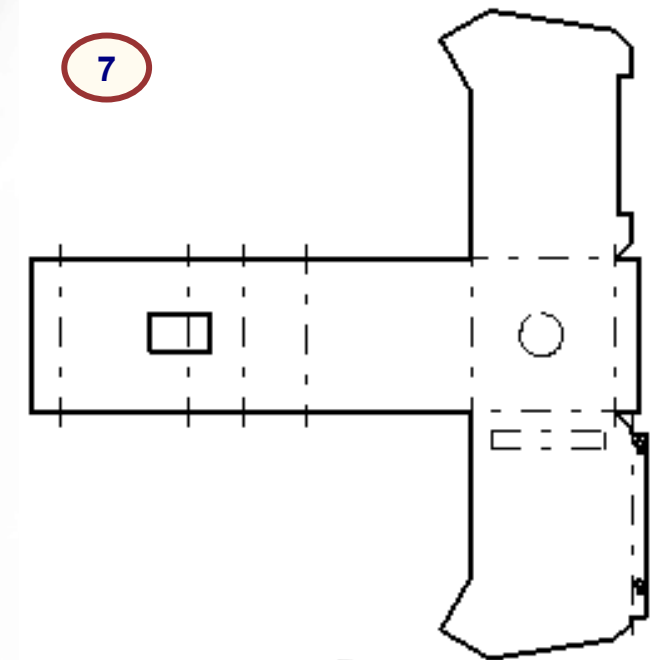
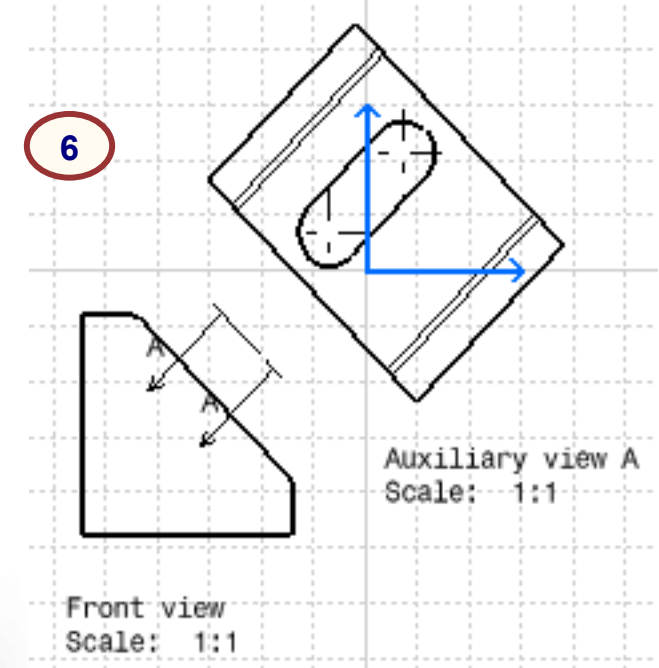
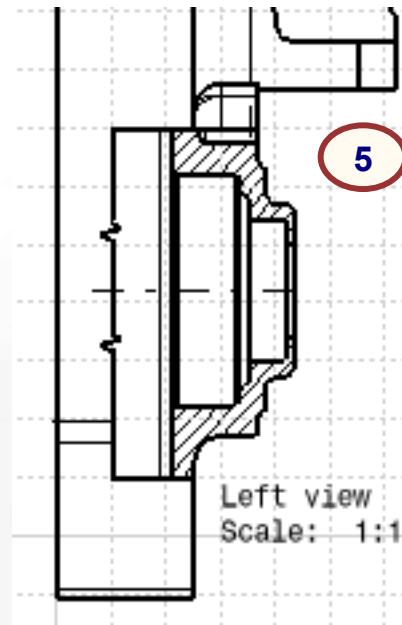
What are secondary Views ? (2/2)

Secondary Views:

5) **Breakout Views:** A breakout view allows the creation of a local cut (by a plane) in order to see the inside of a part without cutting it totally.

6) **Auxiliary Views:** An auxiliary view is a view created in a given direction which is not a direction that can be obtained with a standard view

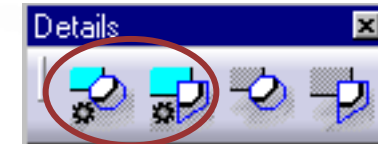
7) **Unfolded Views:** An Unfolded view is a view that can only be obtained from a Sheet Metal part. This kind of view unfolds the sheet metal part in accordance with the rules which have been applied to the bends



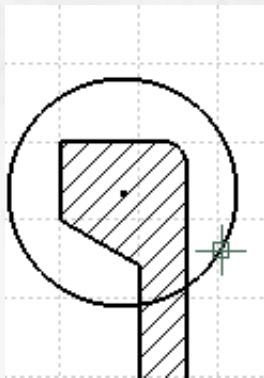
Detail Views Types

Detail views can be full detail views that are calculated from the 3D part or "quick" detail views that are calculated from the defining view. Both types of detail views can be defined by a callout using a circular perimeter or a profile perimeter.

- 1 **Full Detail:** The geometry for the view is calculated from the 3D part with a boolean operation of the callout's perimeter.



Circular Callout



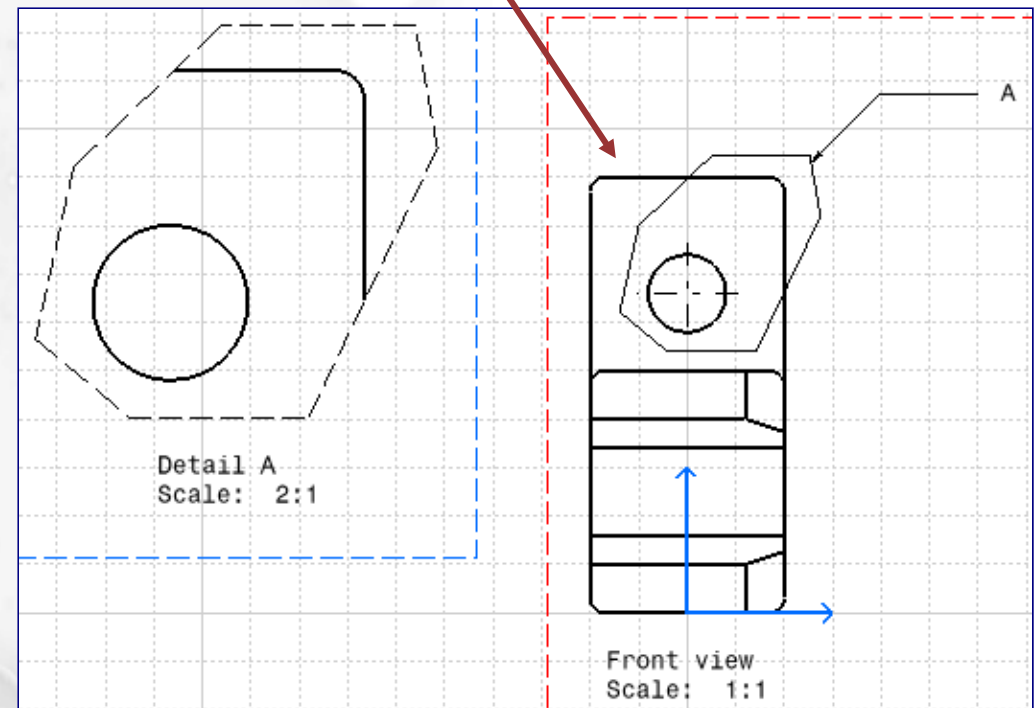
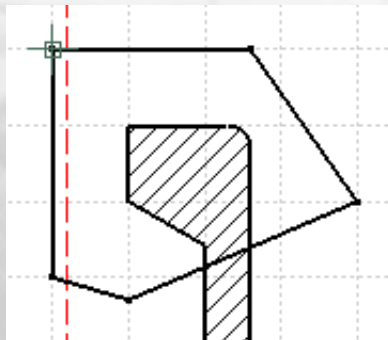
- 2 **Quick Detail:** The geometry for the view is calculated from the defining view.



Profile Callout

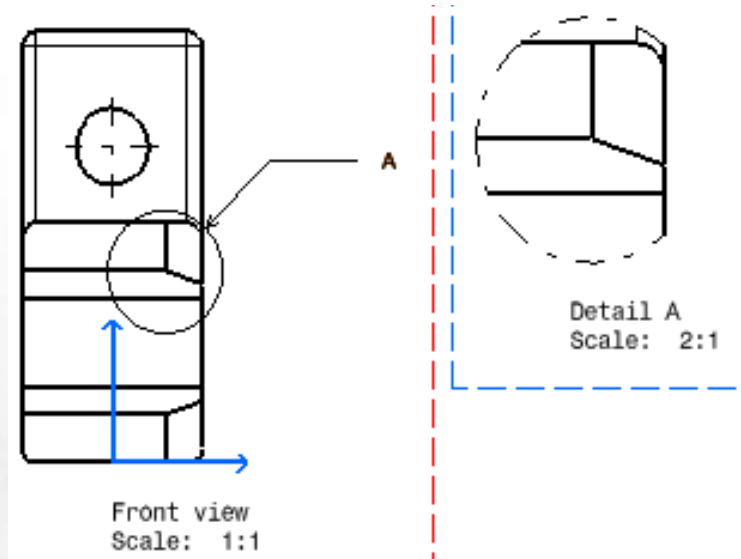


Profile Callout

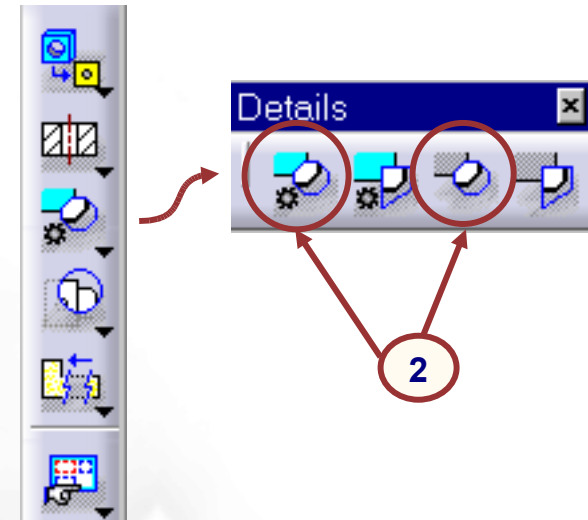


Adding a Detail View

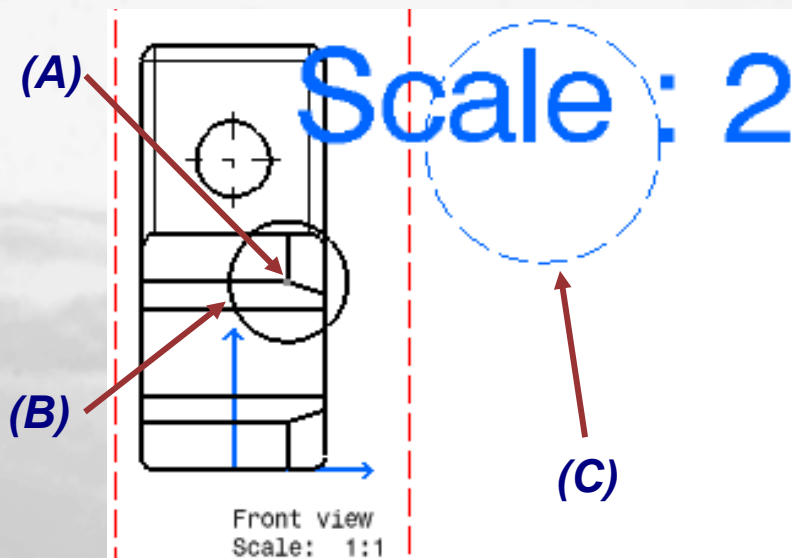
1 To create this detail view



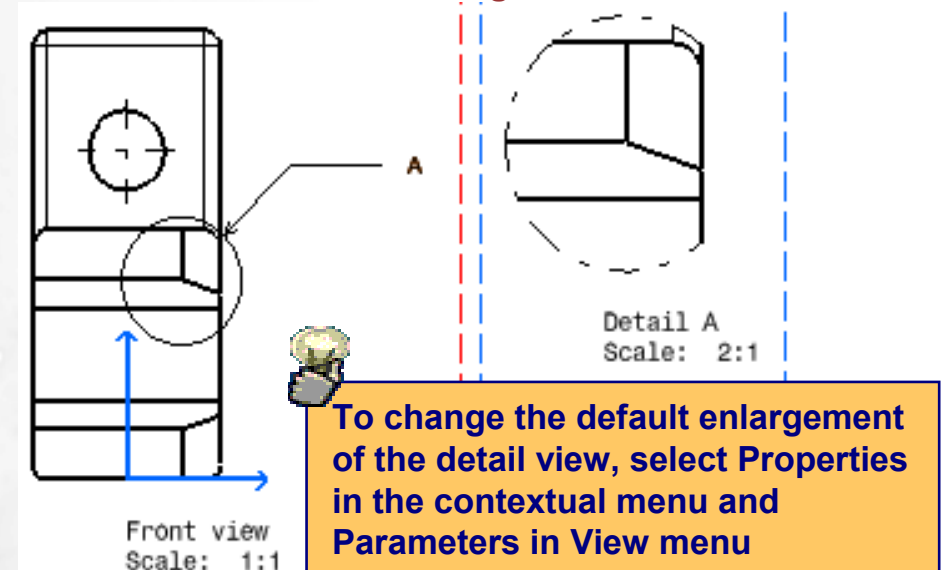
2 Activate the front view, then select the Detail View icon or the Quick Detail View icon.



3 Define the center of the circle by clicking (A), click B to define the circle radius then move mouse to place the detail view at C with a click

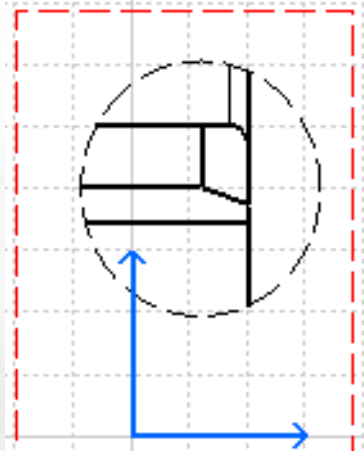


4 The view is generated; the default enlargement is two times the scale of the defining view

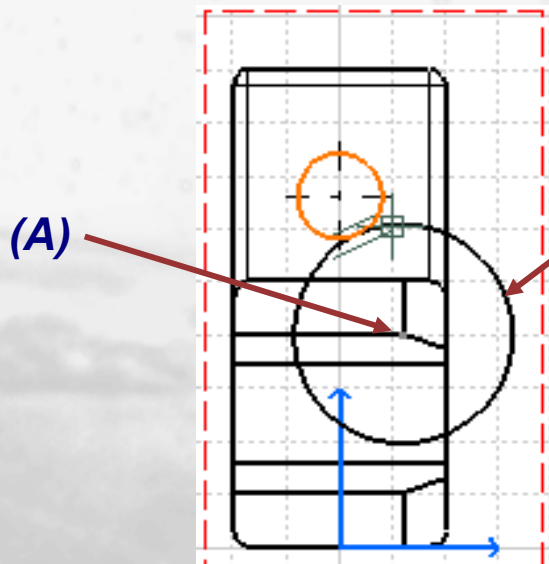


Creating a Clipping View

1 To create this clipping view



3 Define the center of the circle by clicking (A), click B to define the circle radius

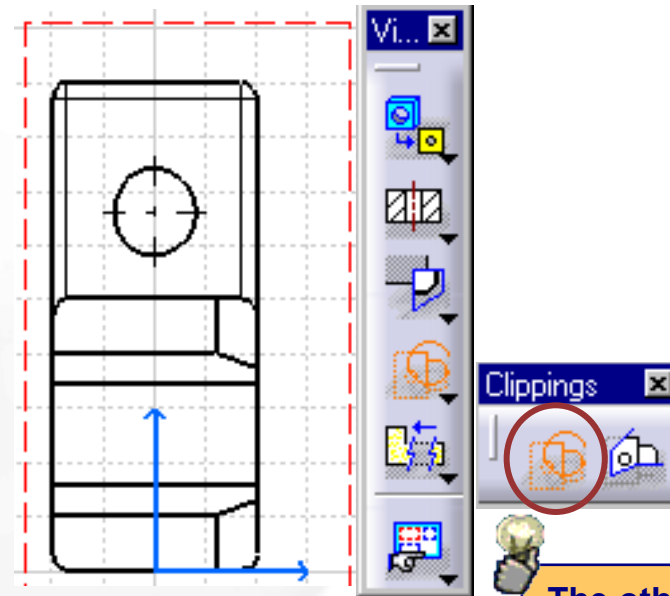


(B)



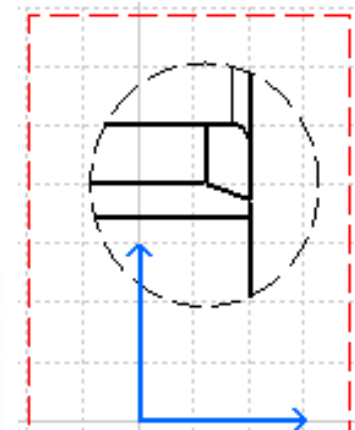
The annotations which are not cutting by the clipping circle can be seen in the No Show mode.

2 Activate the front view, then select the Clipping View icon



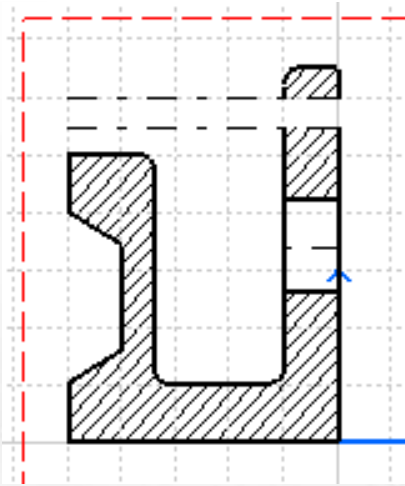
The other Clipping icon creates a clipping view by a profile.

4 The result is displayed

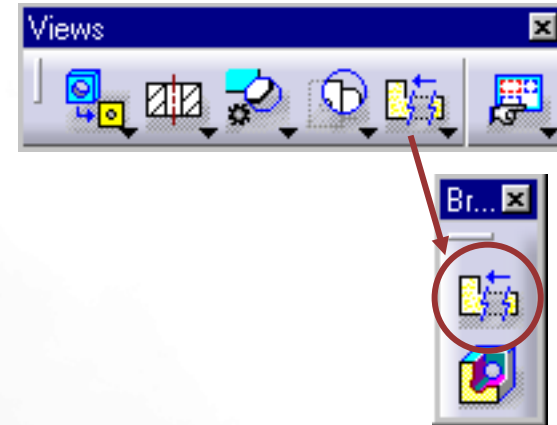


Breaking a View

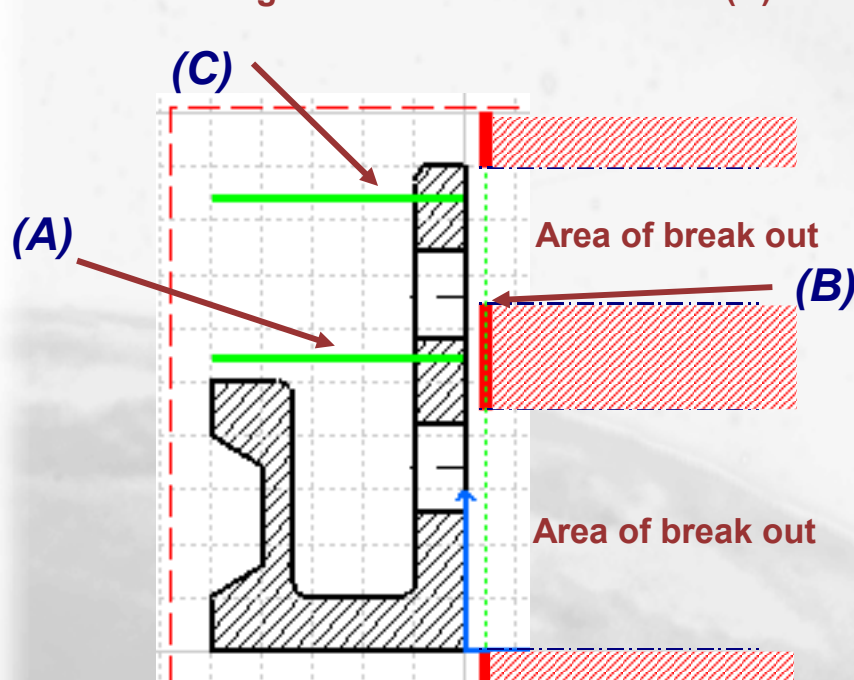
1 To create this Broken view



2 Activate the Section A-A view and select the “breaking a view” icon



3 Define the break out area by clicking (A) the location for the first break limit line, click (B) to delimit the height of the red area and click (C) to locate the second break limit line.



The second break limit can fall anywhere designated by the green dashed line. The solid red line represents the red zone for the areas that cannot be selected for creating the second break limit.

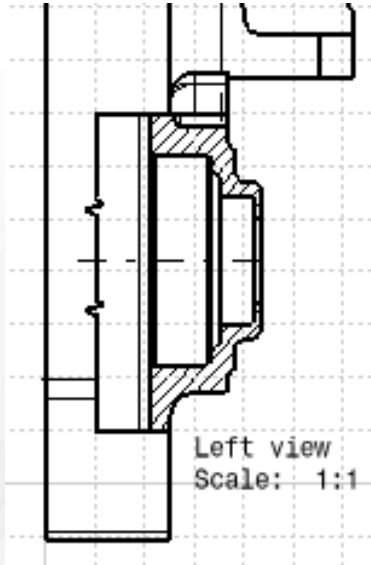
4 Select anywhere on the sheet to modify the section view into a break through section view.

The broken view can be restored with contextual mouse button and select UNBREAK

Note: A view can contain multiple break definitions given the definition is in the same direction and the two breaks do not overlap.

Performing a Breakout View

1 To create the Breakout View below

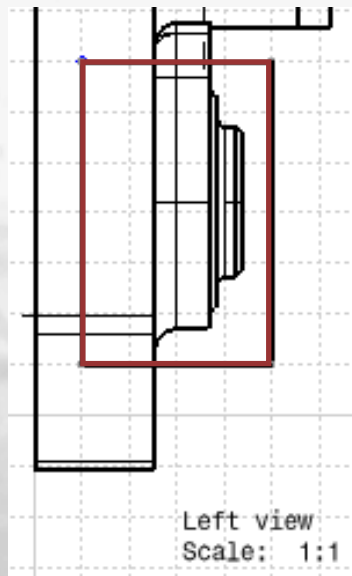


2 Activate the Left view and select the “breakout view” icon

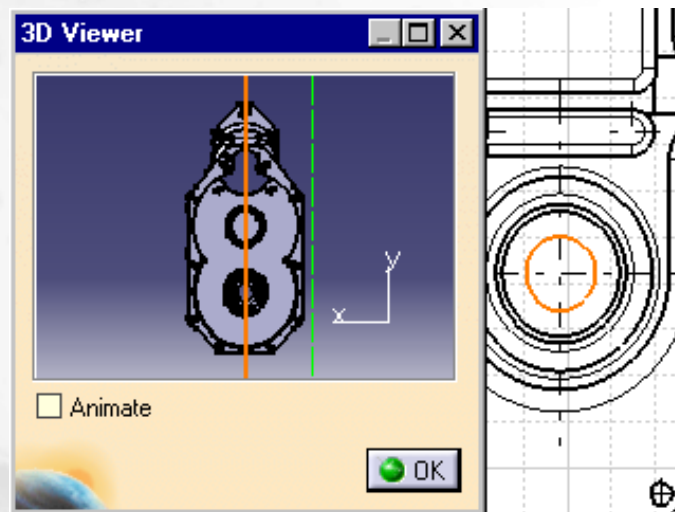


Can create only one simple breakout per view. Cannot generate another view from a breakout view. Once created the breakout view profile cannot be modified.

3 Create points which allow you to build the breakout profile. If necessary, double-click on the first point to close the profile



4 The 3D Viewer window appears. Drag & Drop the green continuous line or work in parallel with the drawing to get the desired cut plane



If Animate is checked you can visualize the 3D part in accordance with the position of the cursor on the drawing

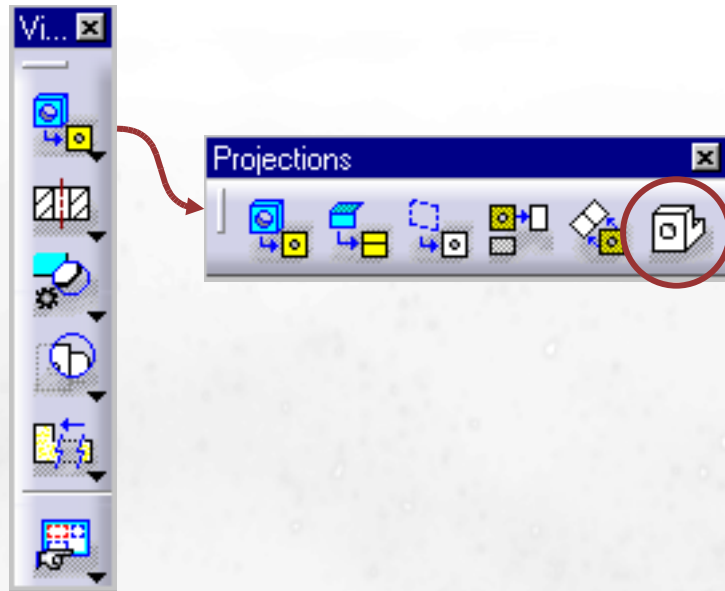
5

Select OK in the 3D Viewer window. The breakout is created.

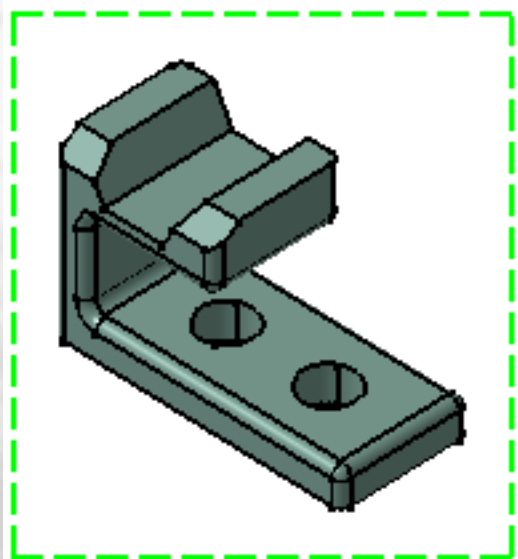
Note: A view can contain multiple breakout definitions.

Adding an Isometric View

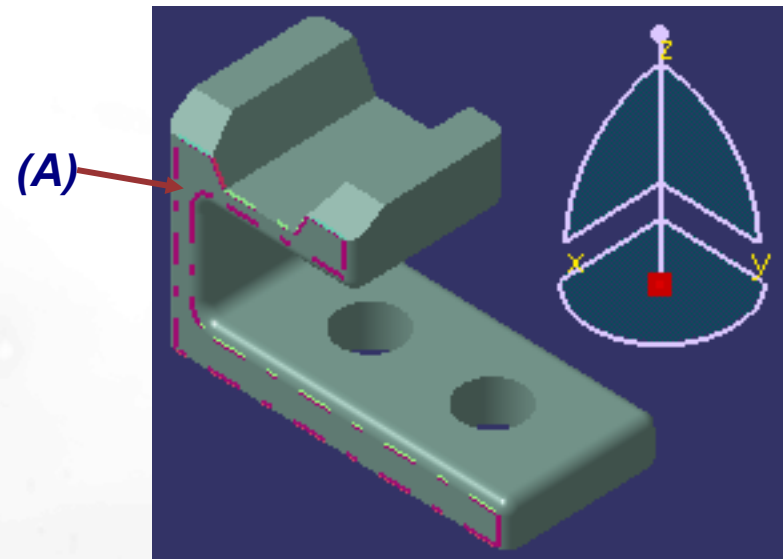
- 1 Select the Isometric icon



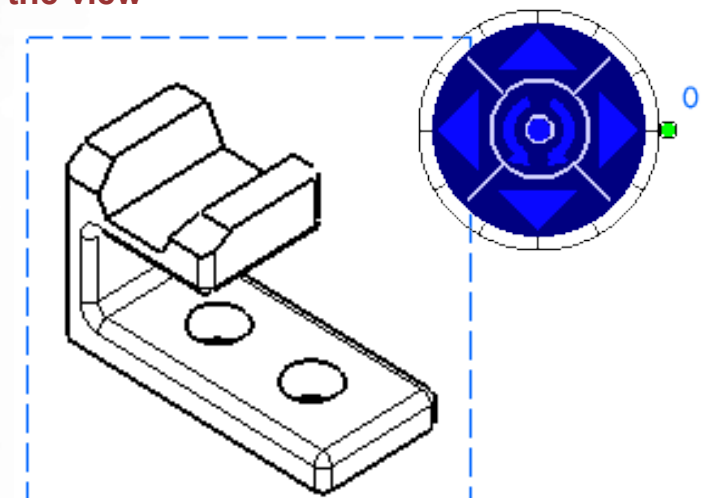
- 3 A preview of the Isometric View displays



- 2 Select a face on the part in the Part or Product document



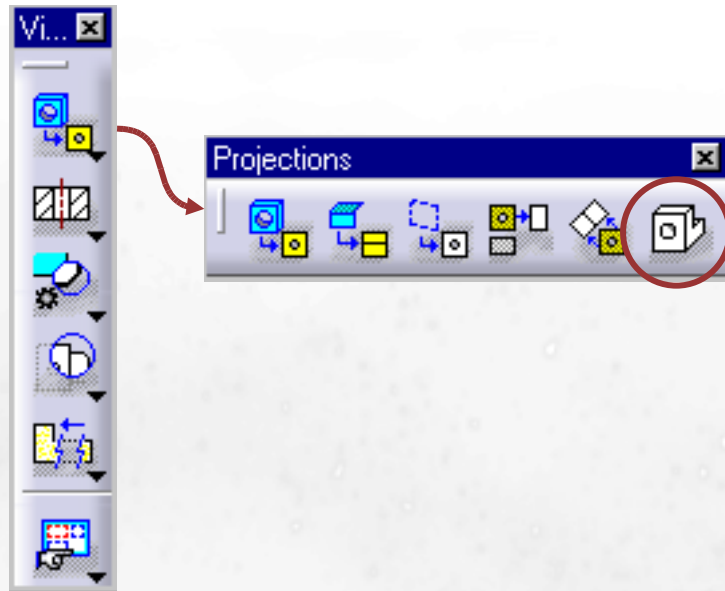
- 4 Select anywhere on the drawing to generate the view



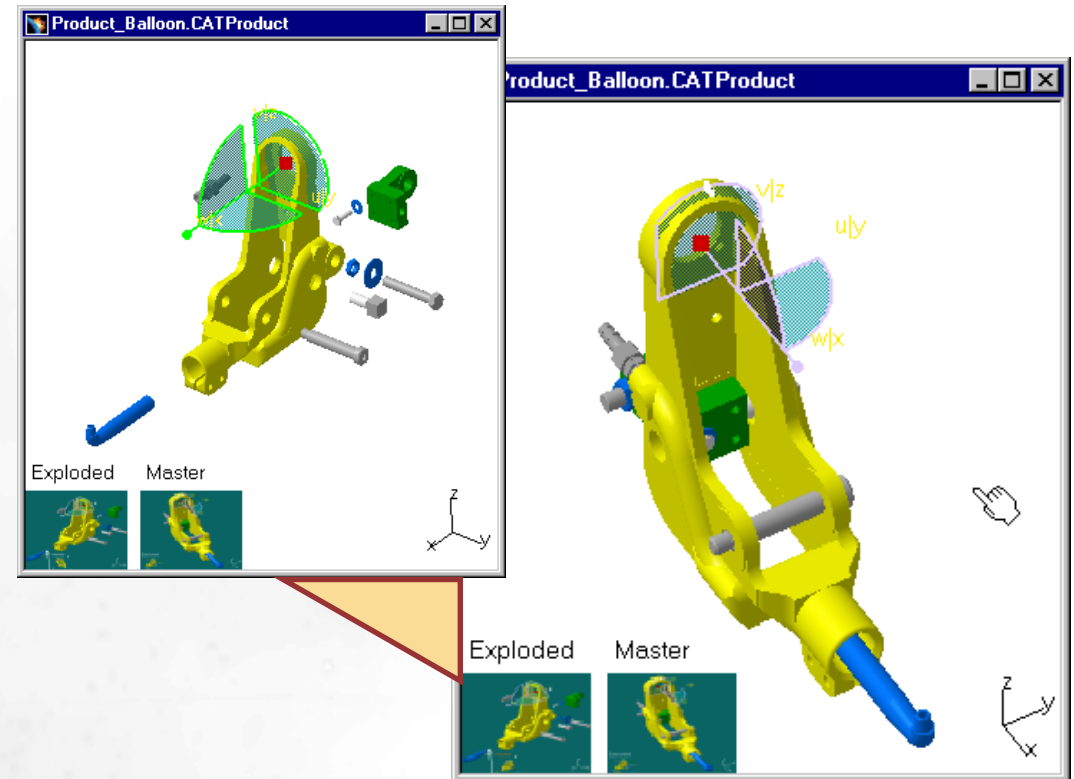
If needed, use the View Manipulator to reorient the Isometric view.

Adding an Exploded Isometric View using a Scene

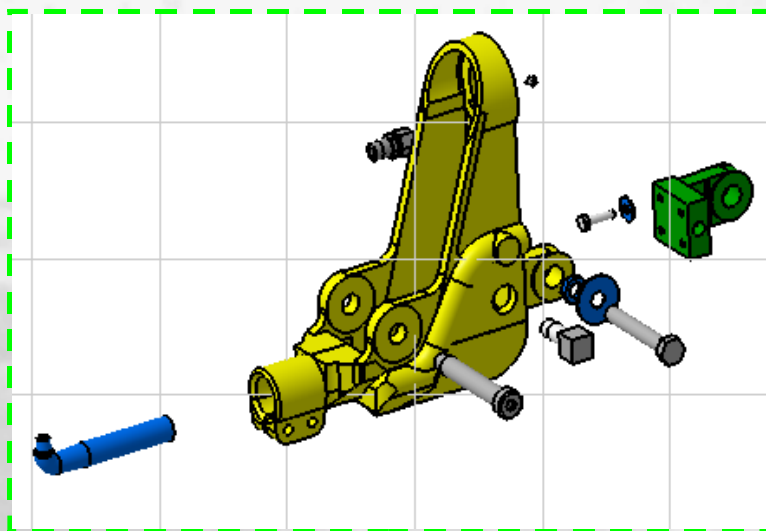
1 Select the Isometric icon



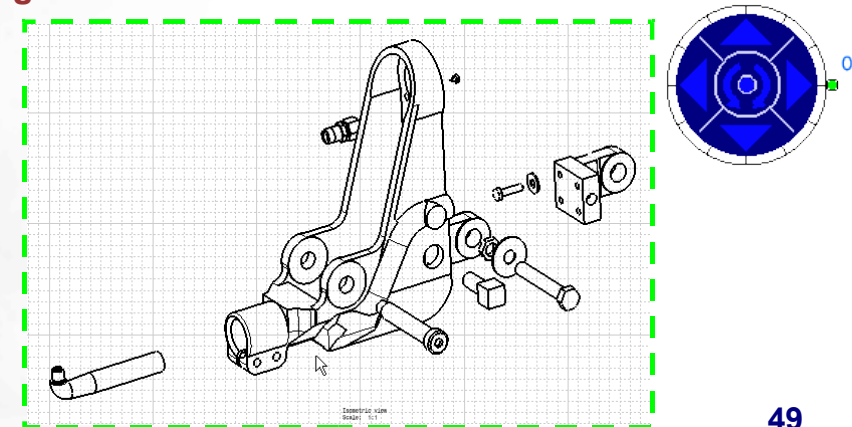
2 Select a Scene in the Part or Product document and a view orientation



3 A preview of the Isometric View displays



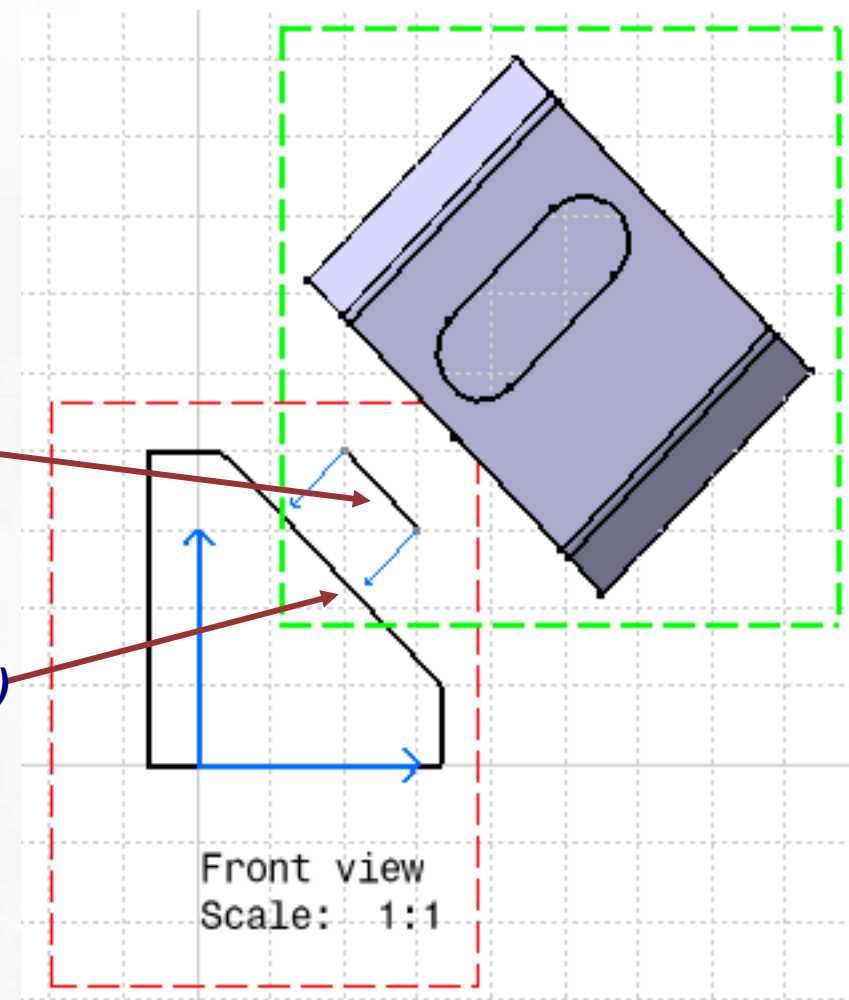
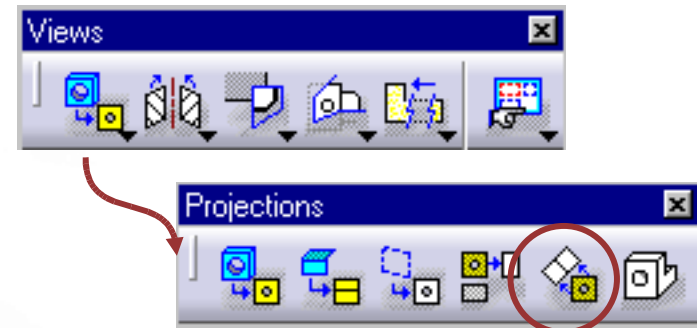
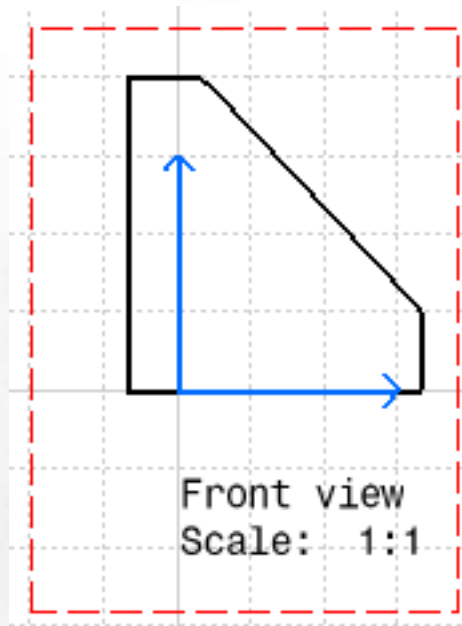
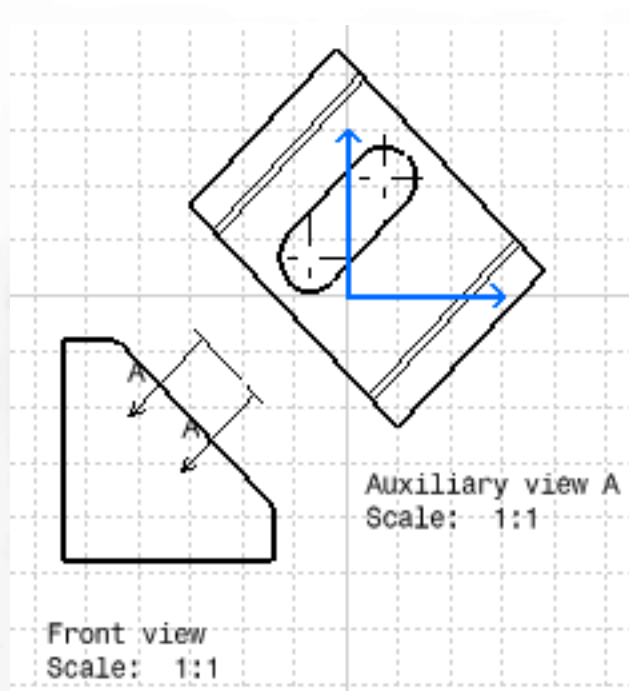
4 Select anywhere on the drawing to generate the view



Adding an Auxiliary View

1 To create the auxiliary view below

2 Active the front view and select the “auxiliary view” icon



3 Sketch the representation of the plane (A) or select an edge (B) on the drawing and drag the mouse to see the preview of the auxiliary view.

4 Select anywhere on the drawing to create the auxiliary view.

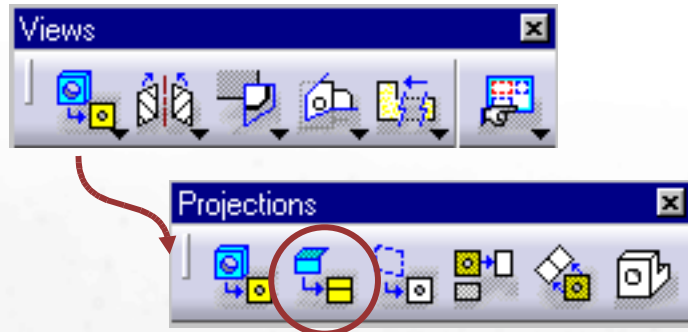


To double-click on an arrow allows to invert the profile with the “Invert Profile Direction” icon ().

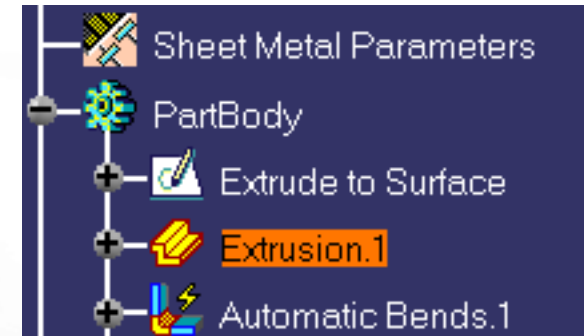
Select “End Profile Edition” icon () to return on the drawing.

Creating Unfolded Views with dashed Bend Lines

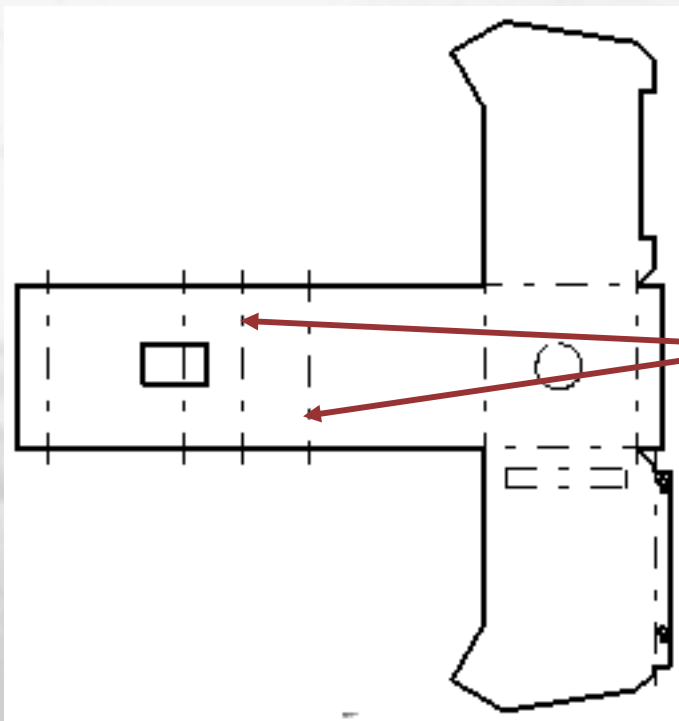
1 Select the Unfolded view icon



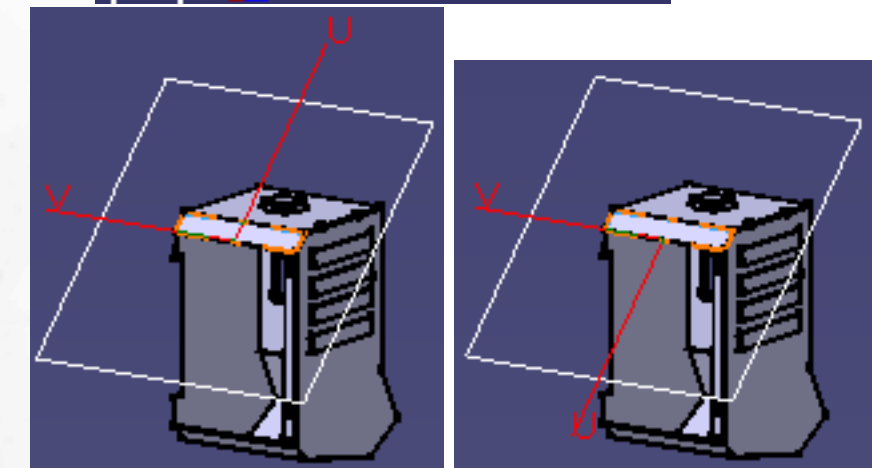
2 Select the first wall of the Sheet Metal part as a 3D reference and choose the reference axis system.



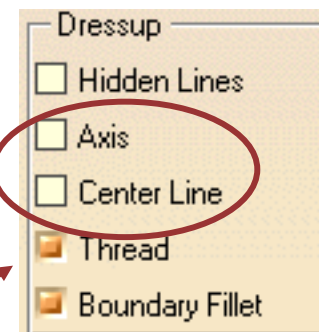
3 Select anywhere to generate the unfolded view on the drawing



Dashed bend lines



To not generate the bend lines, uncheck the Center Line and Axis options in the dressup Properties panel

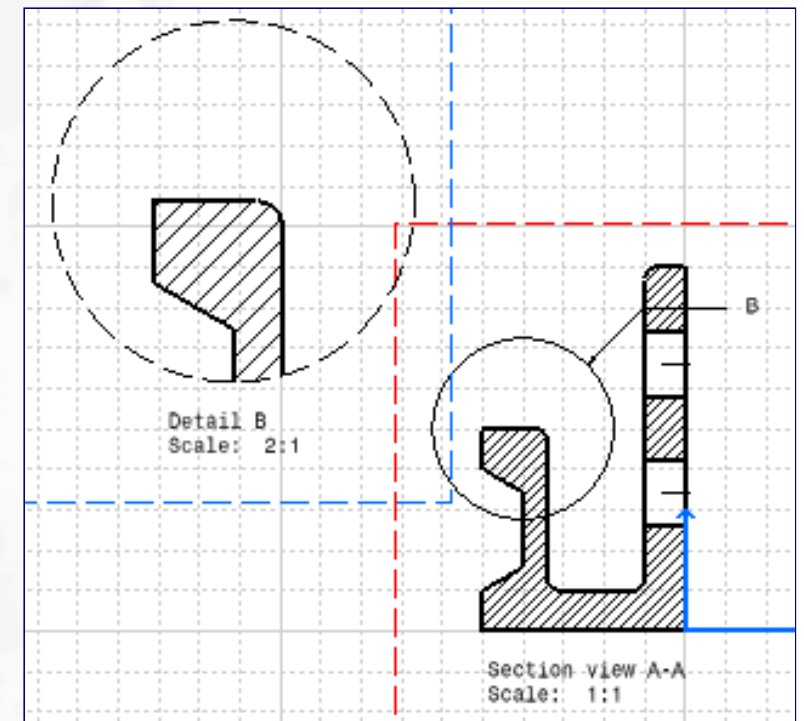


To Sum Up...

In this lesson you have seen...

■ How to add section views and section cuts to a drawing

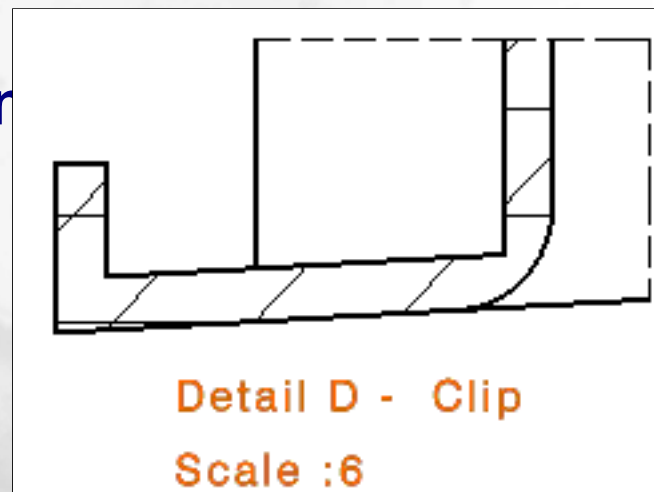
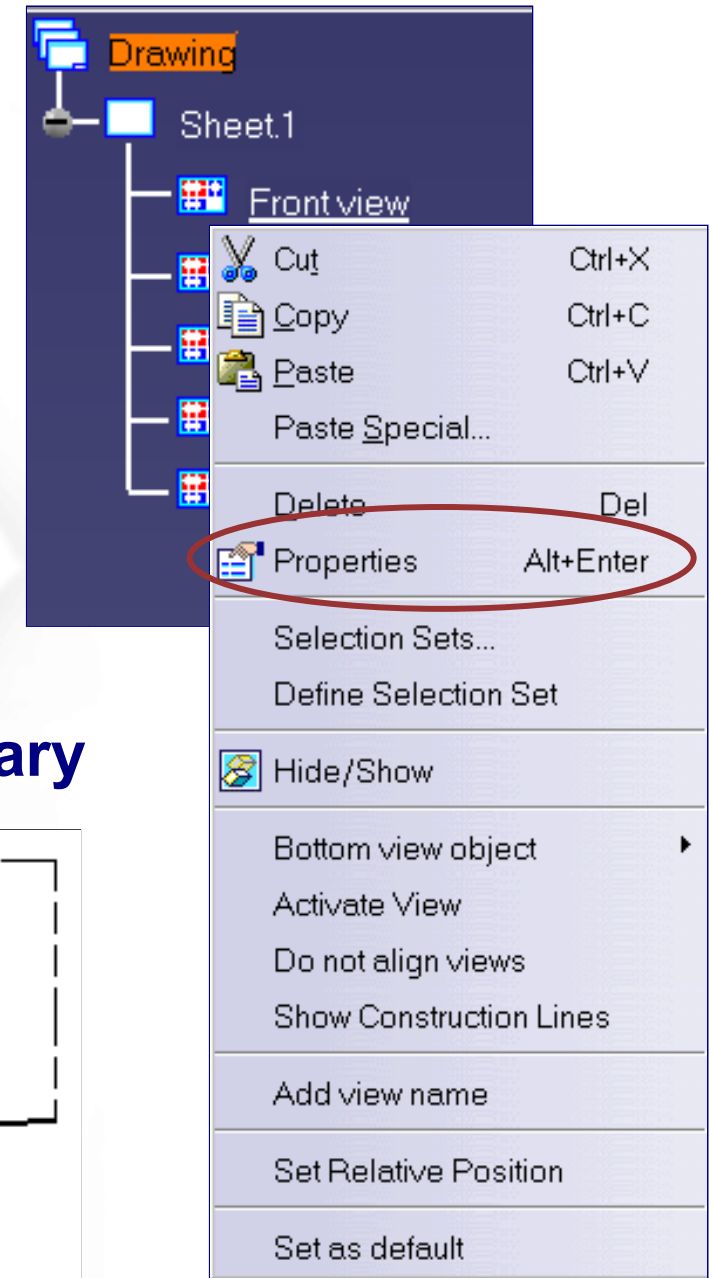
■ How to add secondary views



Editing View's Layout and Properties

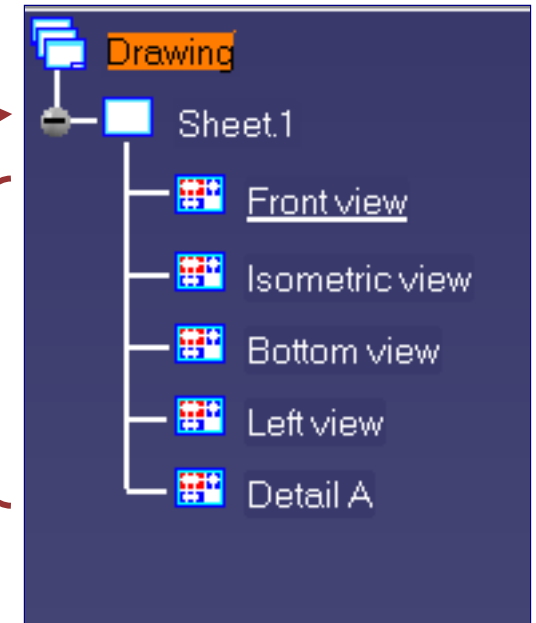
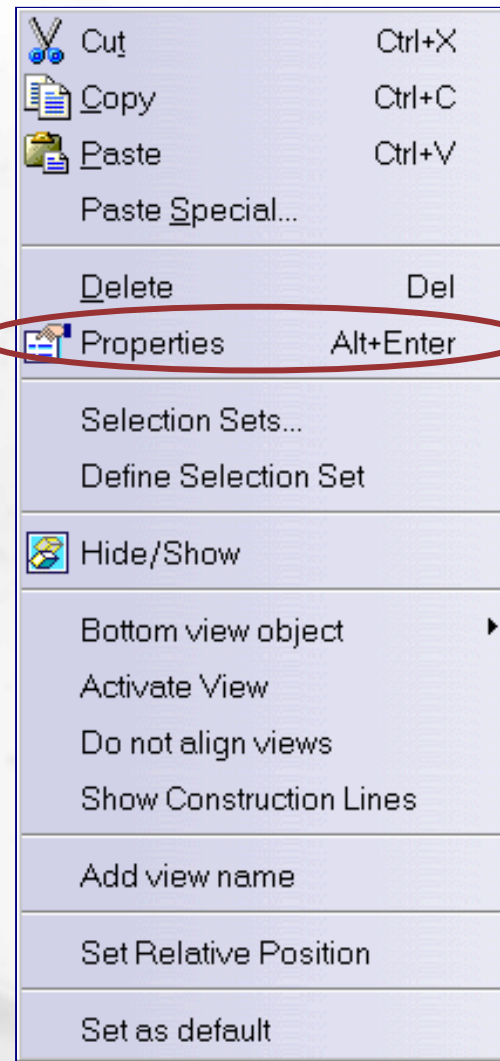
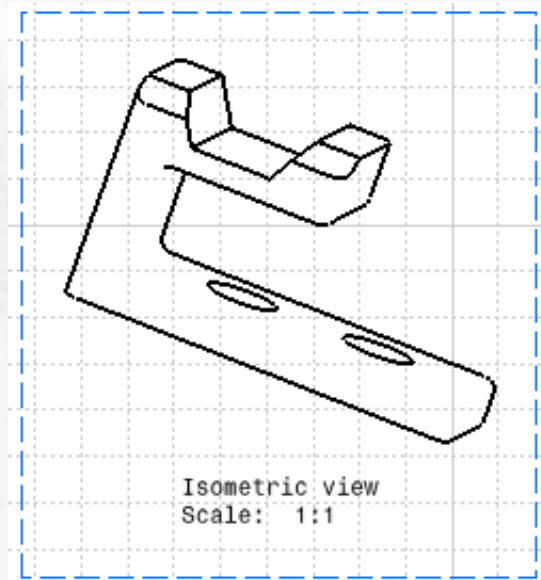
In this lesson you will change the layout of views on a drawing and change the properties of the views.

- ▢ **Editing a View and Sheet Properties**
- ▢ **Adding Sheets to a Drawing**
- ▢ **Repositioning Views**
- ▢ **Modifying Views**
- ▢ **Modifying of Section, Detail and Auxiliary Profiles**
- ▢ **Modifying of Section, Detail and Auxiliary Graphical Definition**
- ▢ **Modifying a Section Hatching Representation**



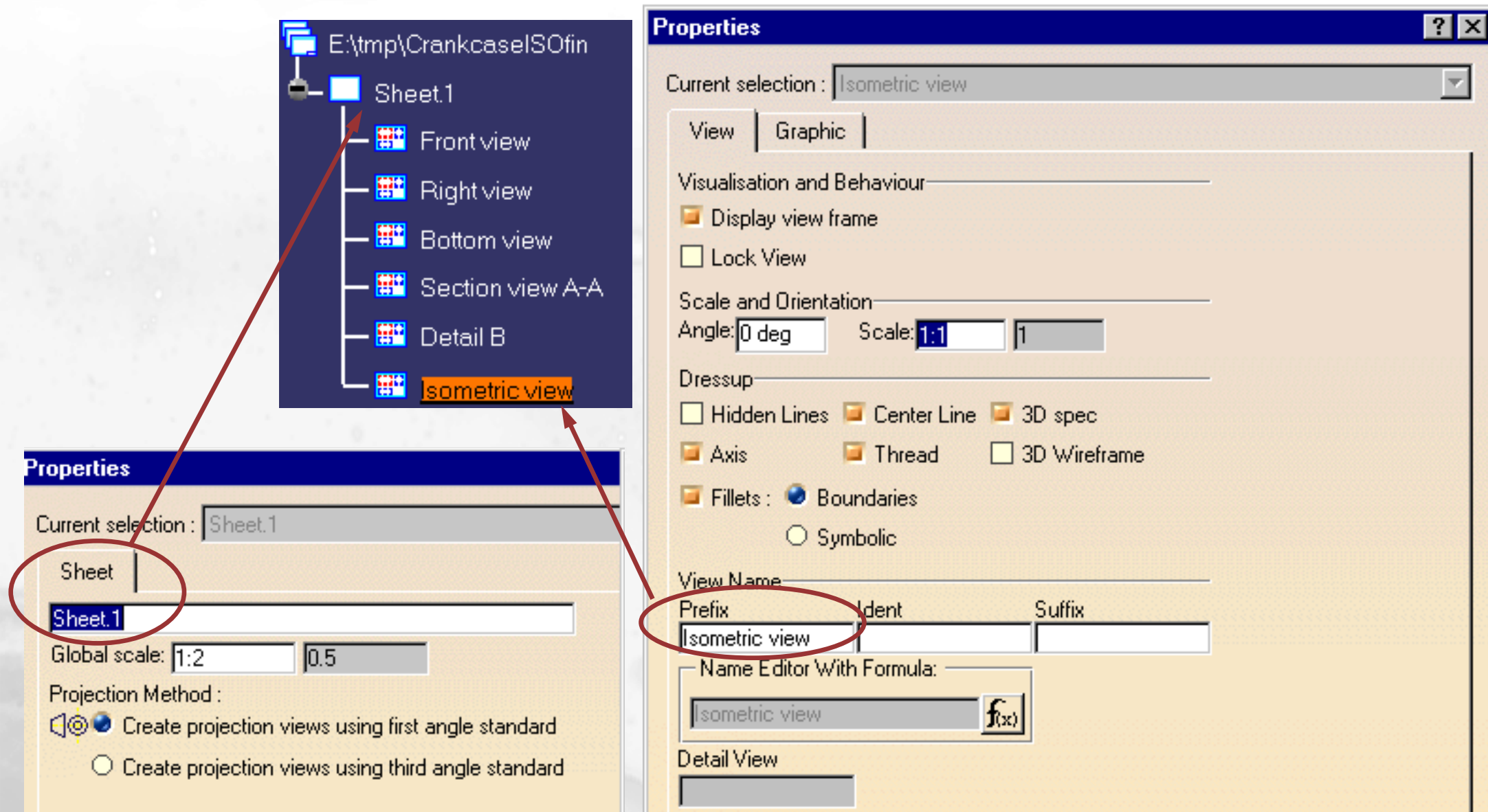
Editing view and sheet properties

You will learn how to edit the view and sheet properties.



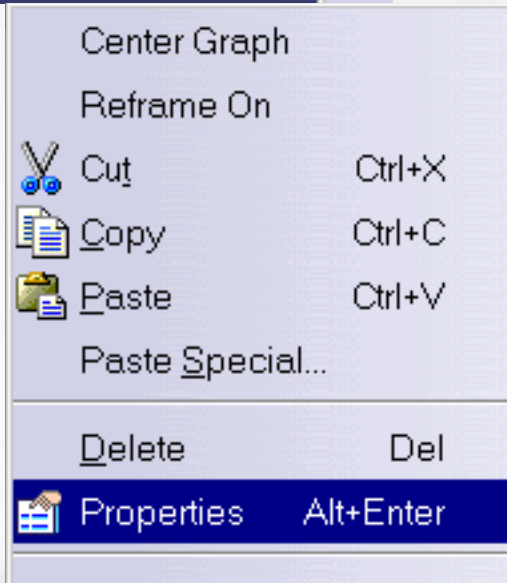
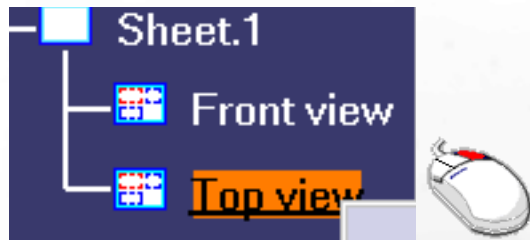
What are view and sheet properties?

View and Sheet properties control all the variables related to specific views and sheets of the drawing.



View Properties

- 1 Select the view to modify in the tree or View frame and Properties with the contextual menu

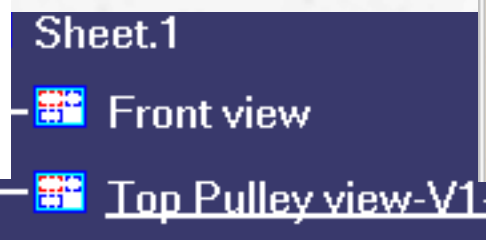
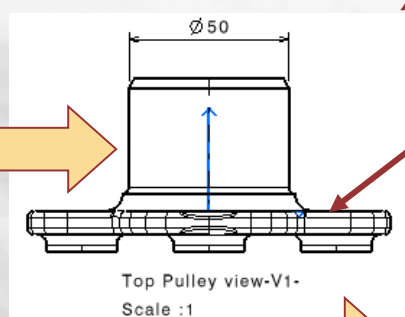
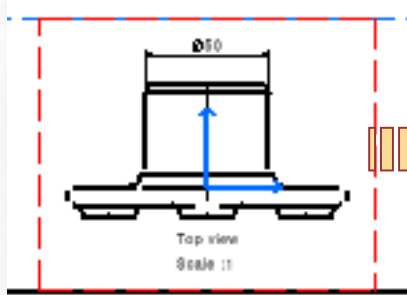
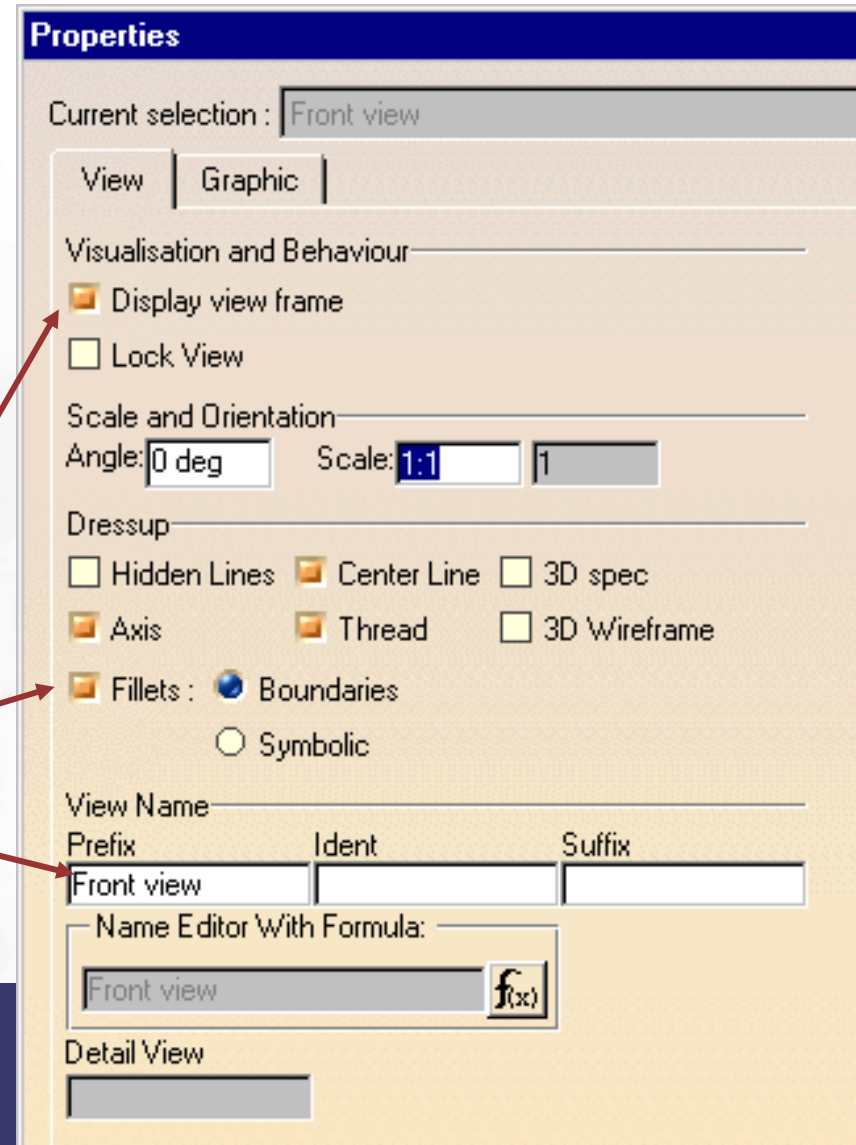


Multi-selection of views is allowed on a single sheet or on different sheets

2

- 2 Select the View or Graphic tab and change the necessary options, here :

- (A) View name
- (B) Dressup to have fillets on
- (C) Visualization to remove the frame

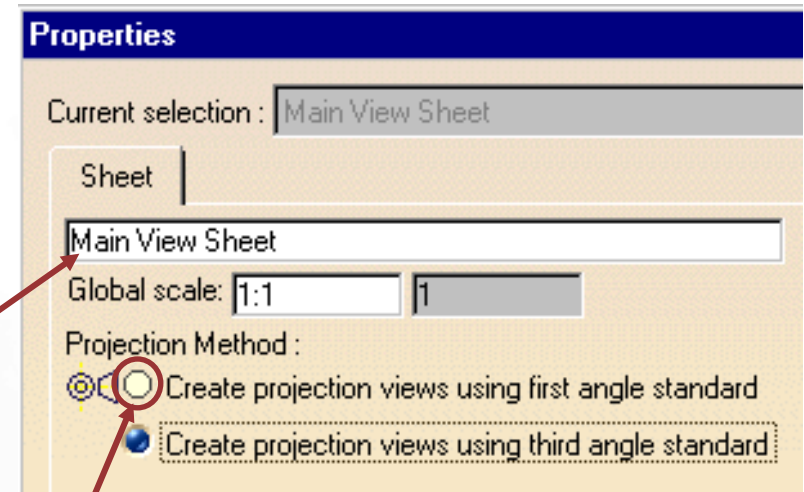
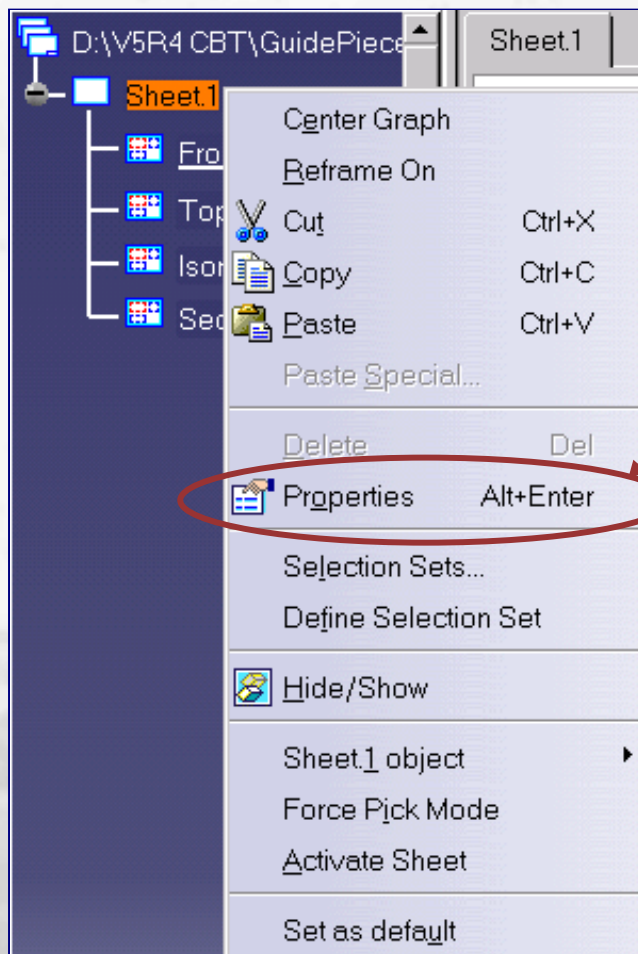


Sheet Properties

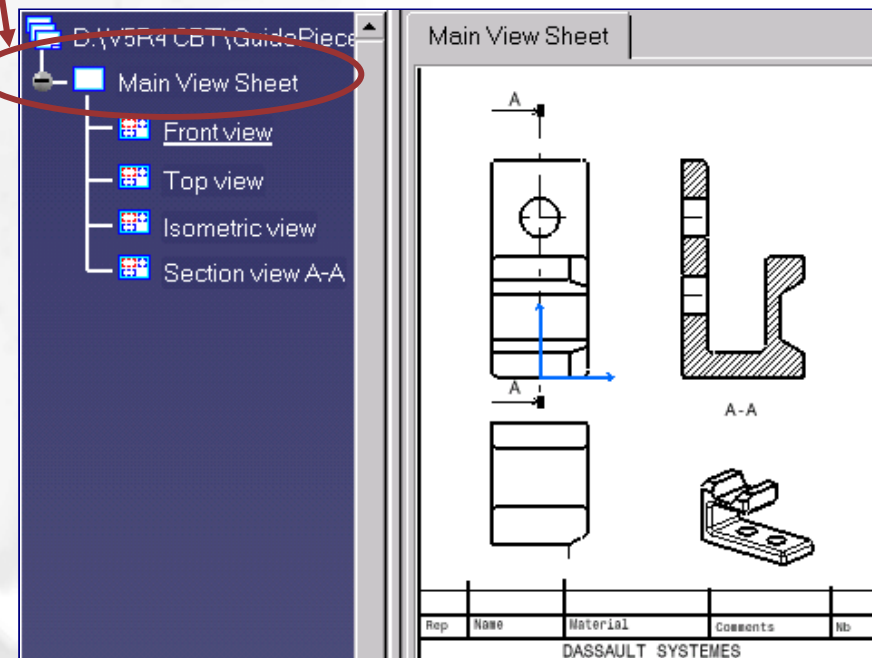
- 1 Select the Sheet (Sheet1) in the tree and Properties in the contextual menu to change its name.



- 2 Modify the Sheet name, scale, or projection method (ANSI or ISO).

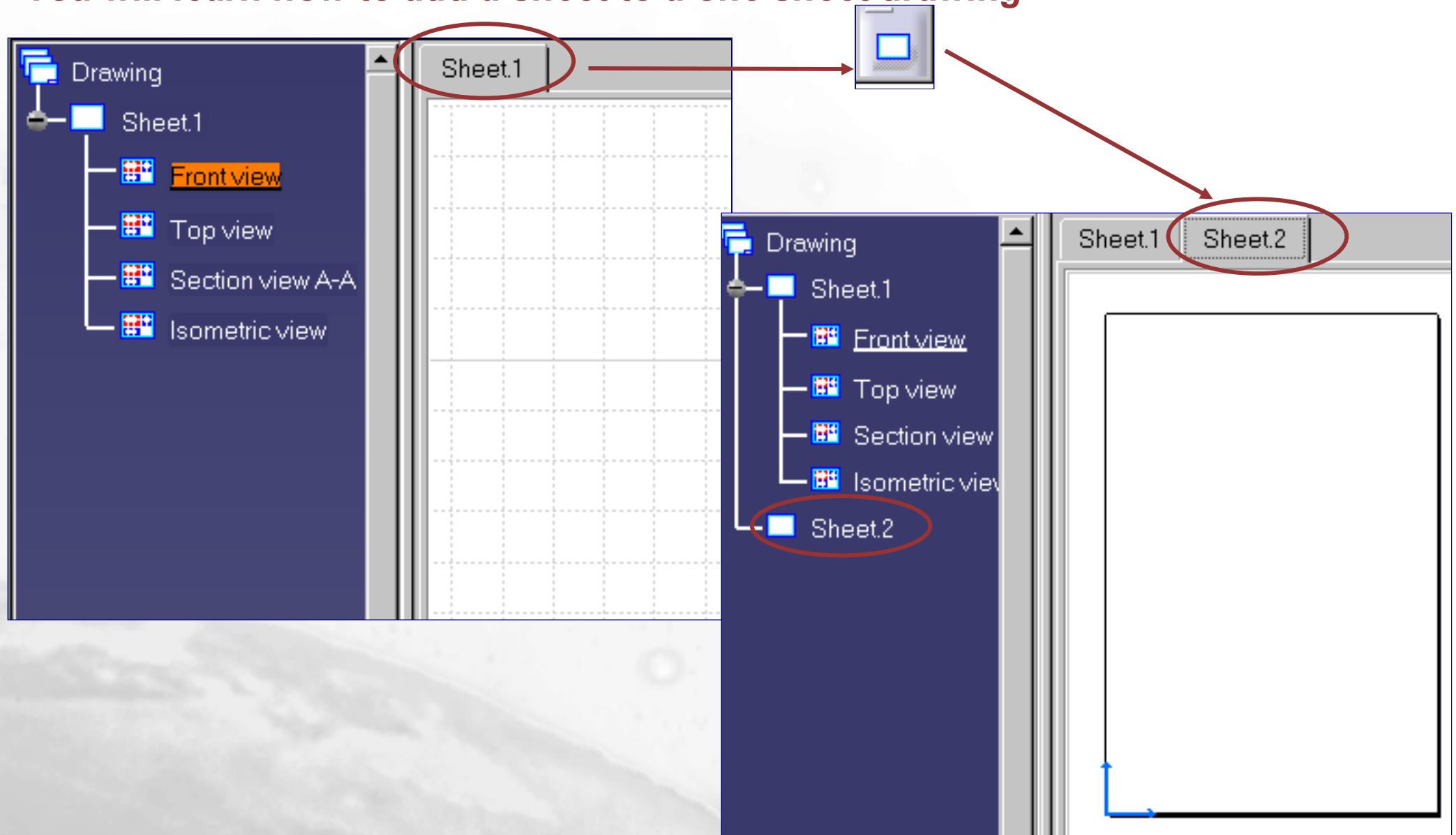


Use this option to generate the views on the sheet using the 1st angle projection method (ISO)



Adding sheets to a drawing

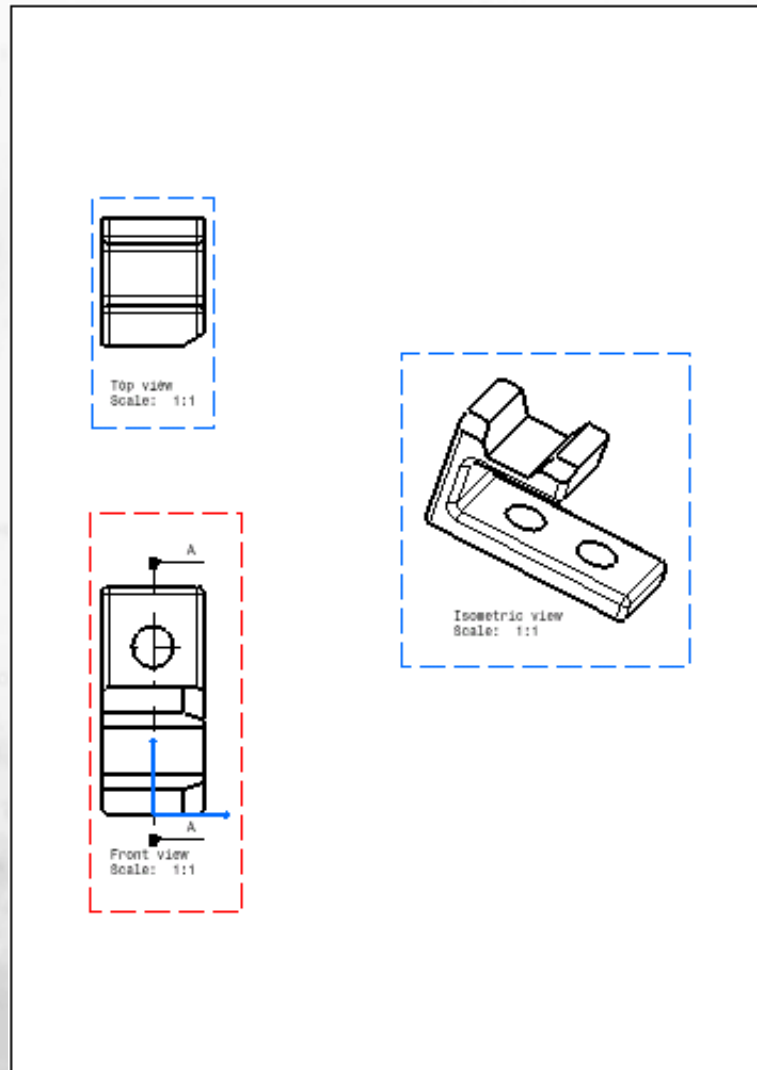
You will learn how to add a sheet to a one sheet drawing



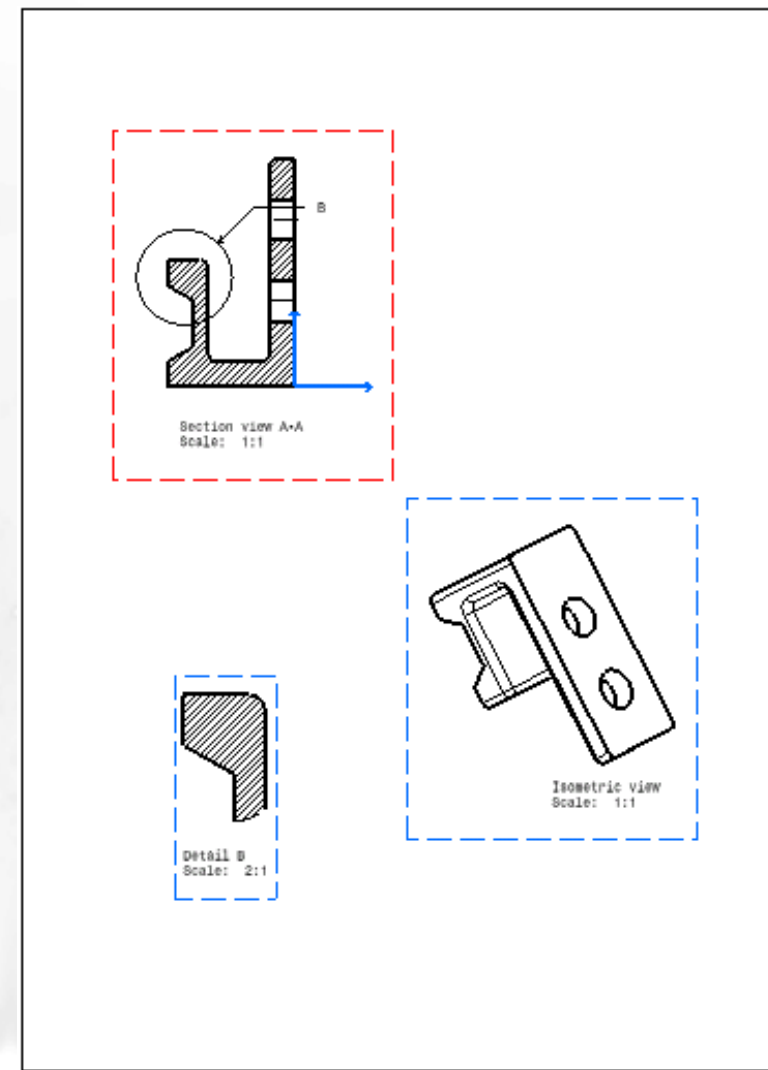
Why add sheets to a drawing?

Sheets are added to a drawing to improve clarity and manage views or annotations that are cluttering a single sheet drawing.

Sheet 1 of 2



Sheet 2 of 2



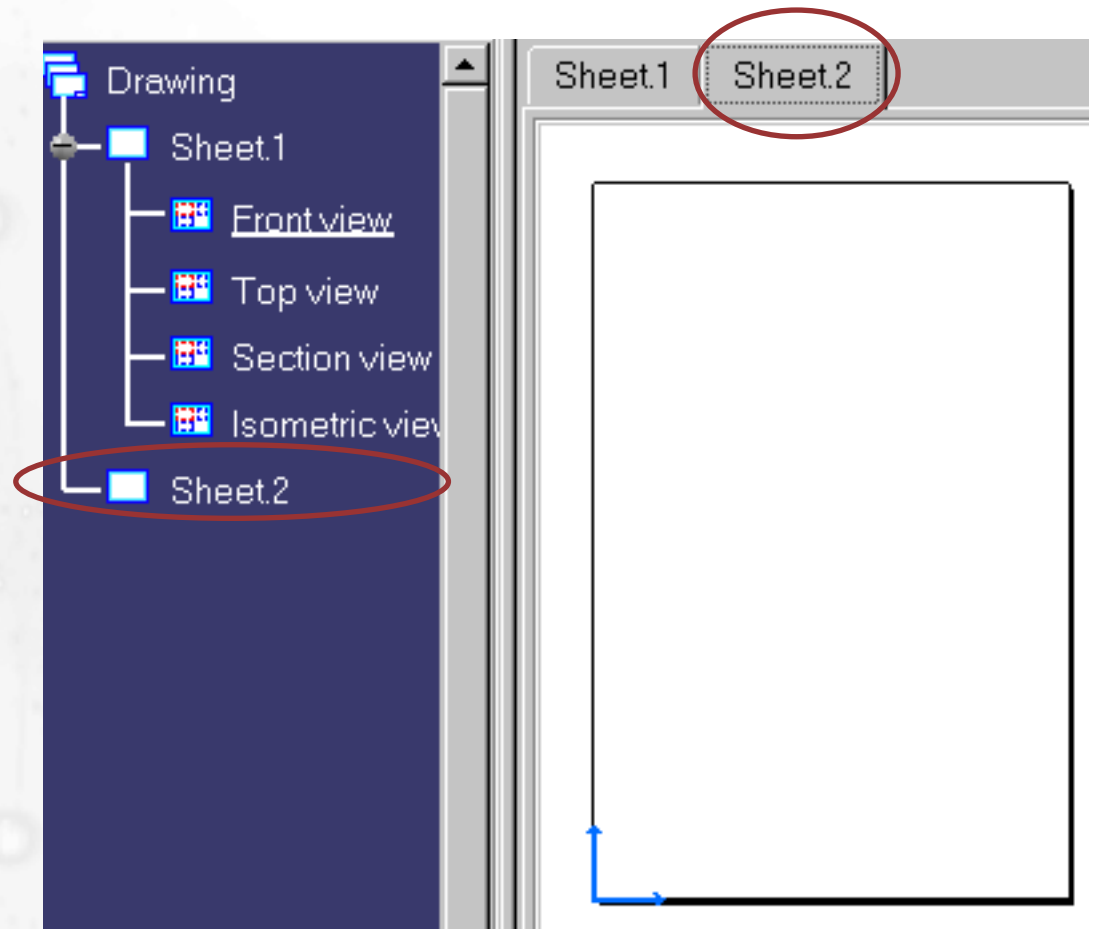
Adding a sheet to a drawing

1

Select the New Sheet icon
It creates an empty sheet with the next sheet number (Sheet 2)

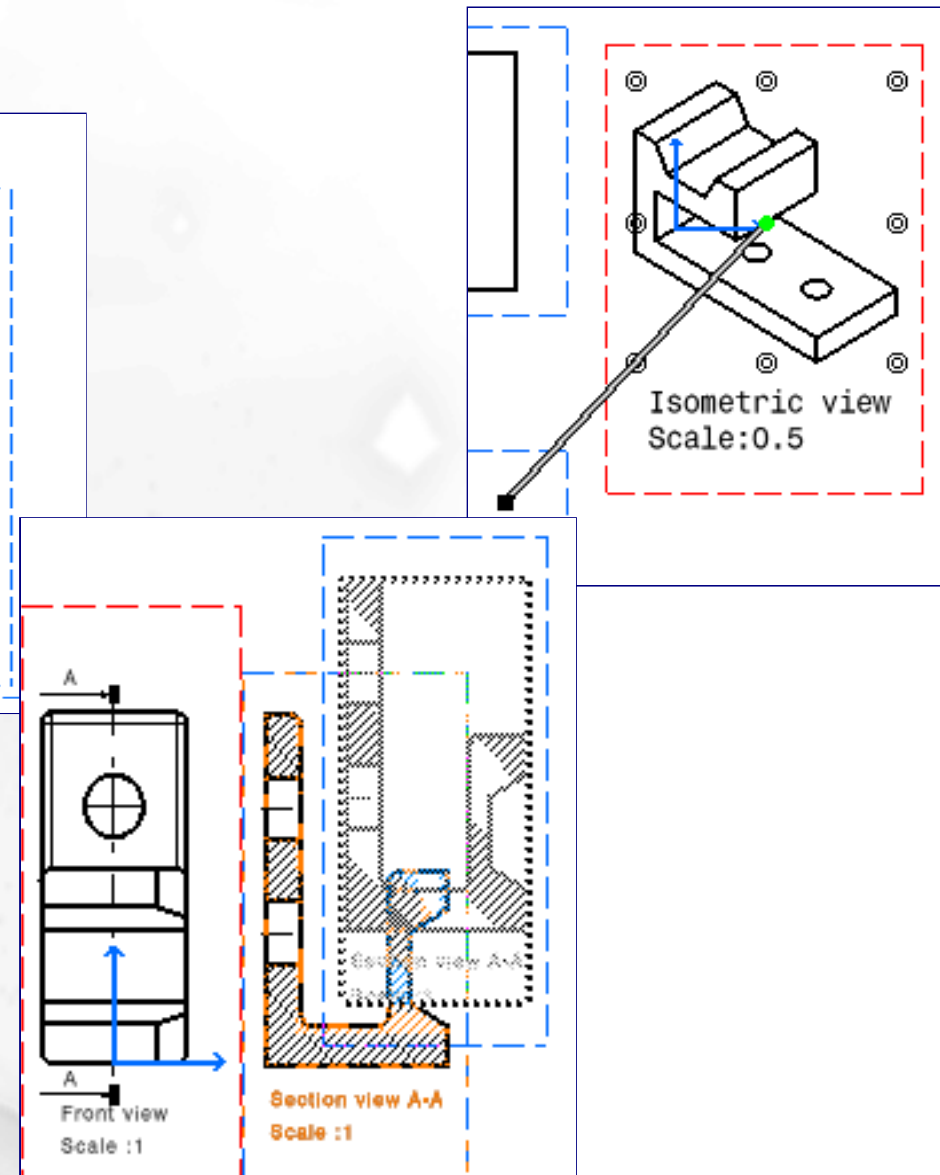
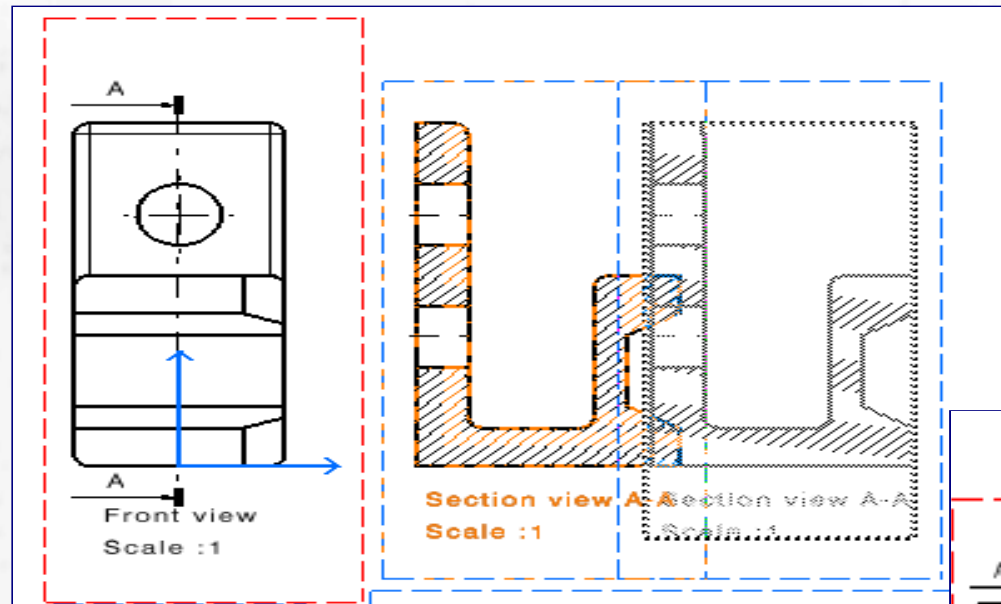


The new sheet is assigned the same standard, format and orientation as the first created sheet



Repositioning Views

You will learn how to reposition views on a single sheet and how to move views from one sheet to another sheet.

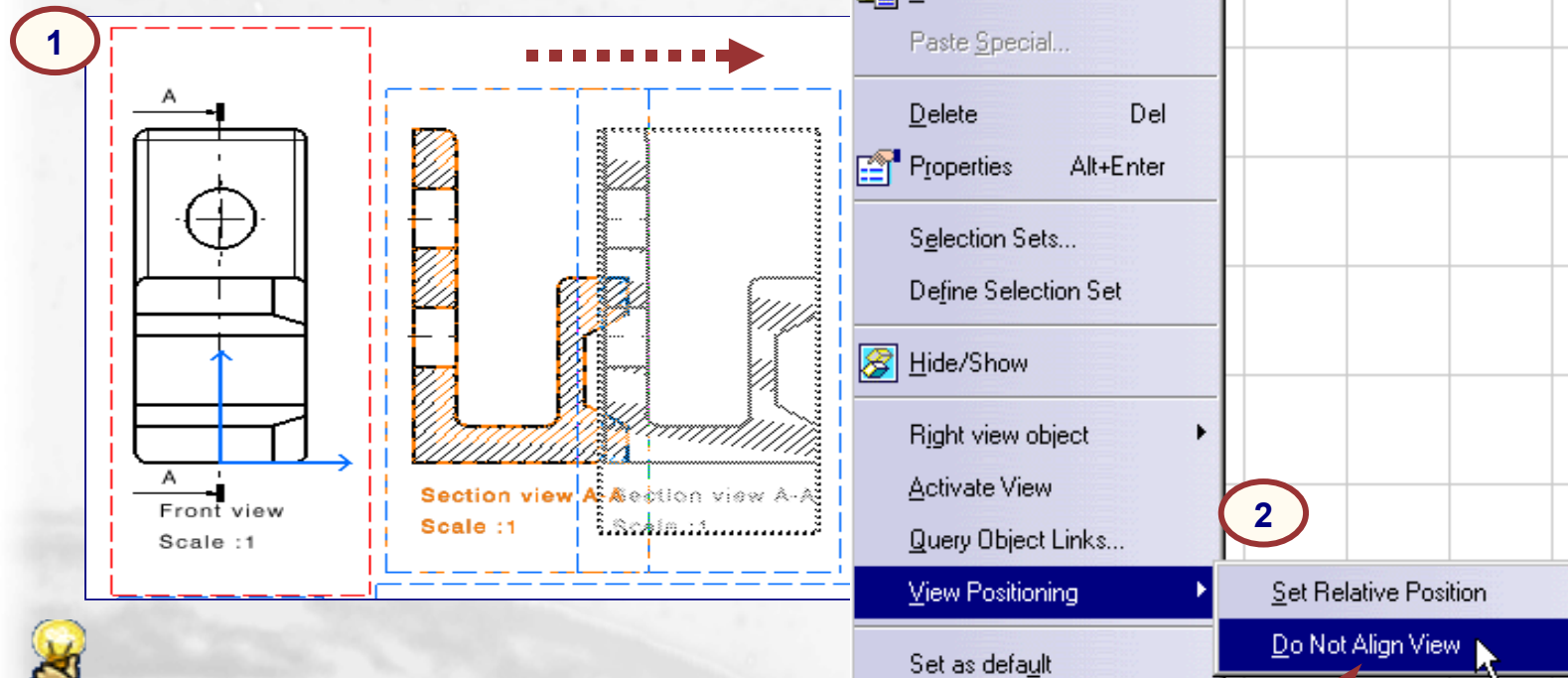


Repositioning views on a single sheet

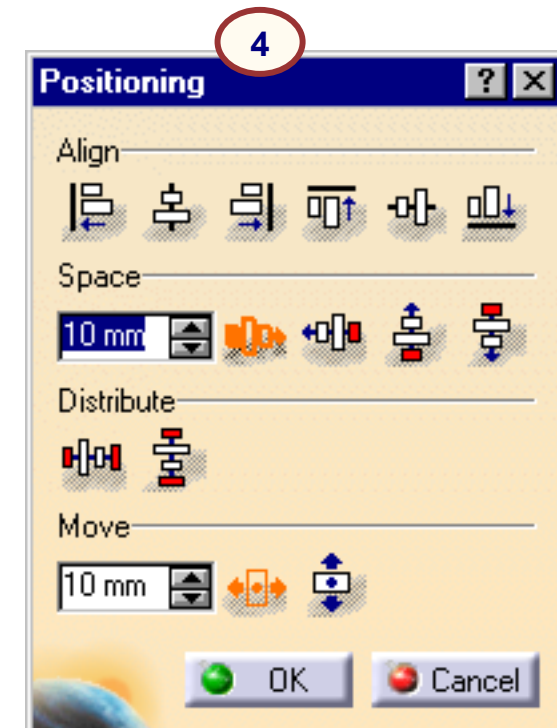
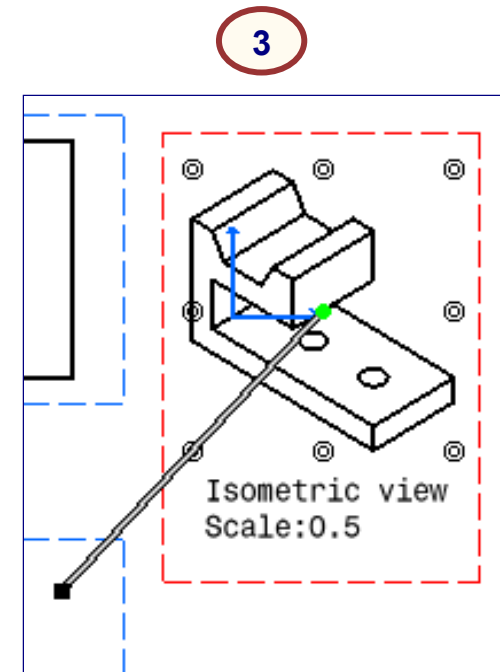
Views can be repositioned in four different ways.

Repositioning Views Options:

- 1 Maintaining alignment with “Parent” Front view
- 2 Without maintaining alignment with “Parent” Front view
- 3 Relative positioning
- 4 Text and View positioning tool



The “Parent” Front view relationship is established when views are added from the front view or created with the Wizard. These “children” views will maintain the alignment link with the “parent” front view unless the alignment is broken.

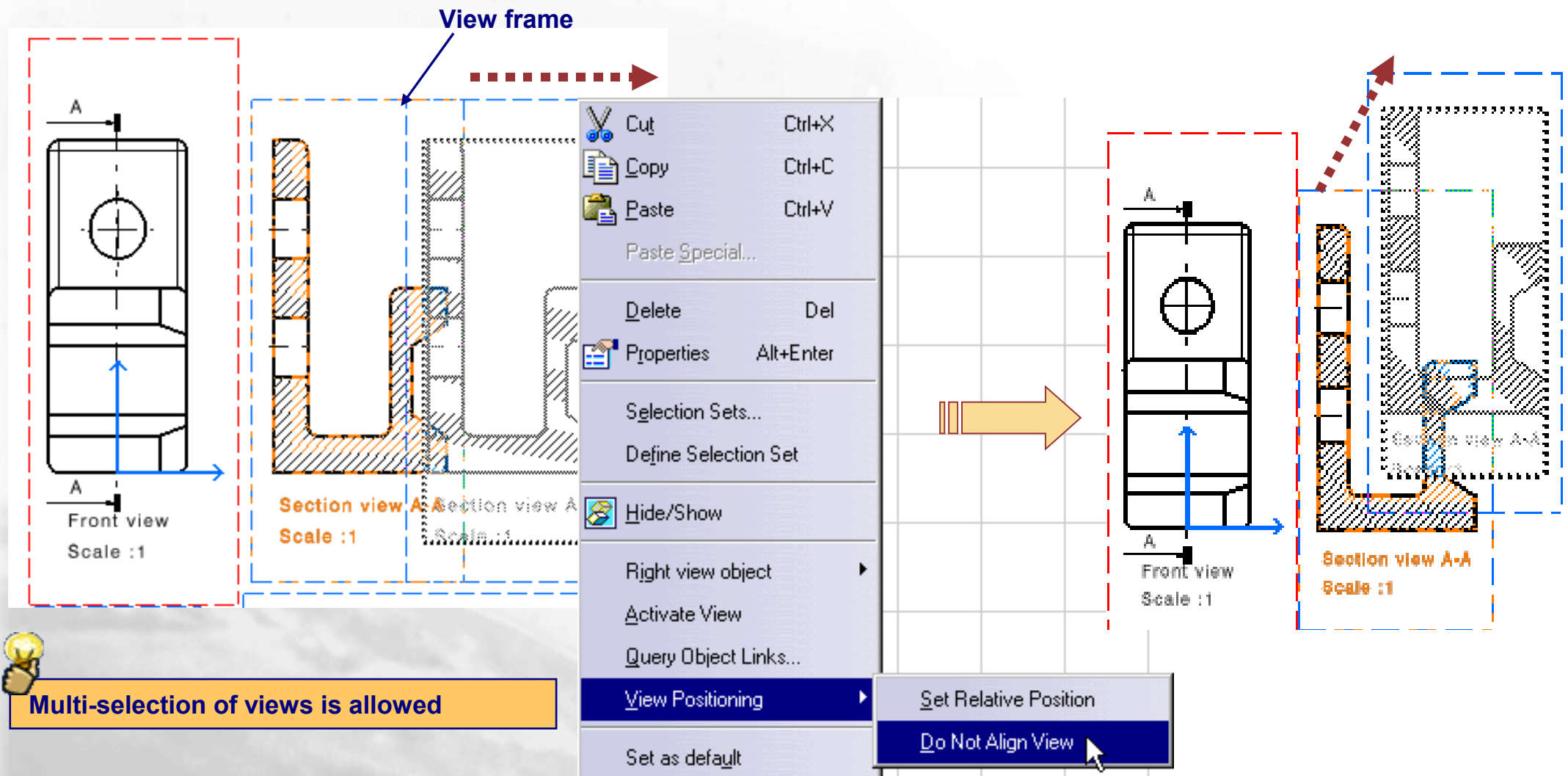


Repositioning views on sheet

A view can be repositioned (moved) to another location

1 Select the frame of the view to move and drag it to the correct position

2 Select the “Do not align views” in the contextual menu first if the desired position is to be not aligned with the Front view.



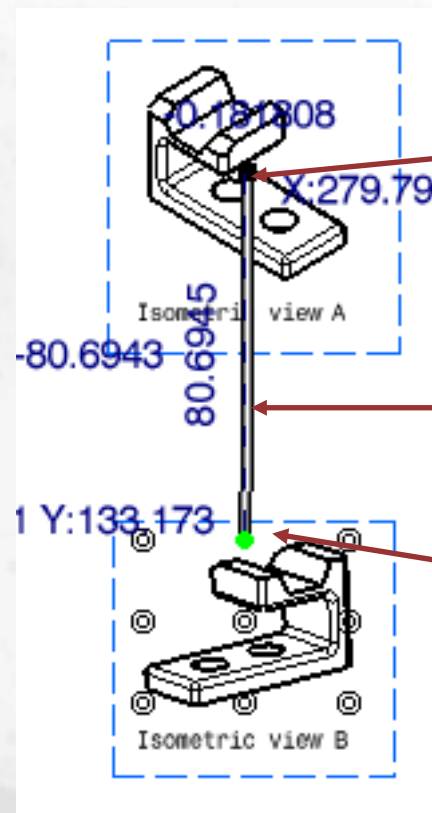
Relative Positioning

A view can be positioned at an exact position on a Sheet and relatively to another view already on the sheet.

Here view B is positioned relatively to view A

View A

View B



Geometric center of view

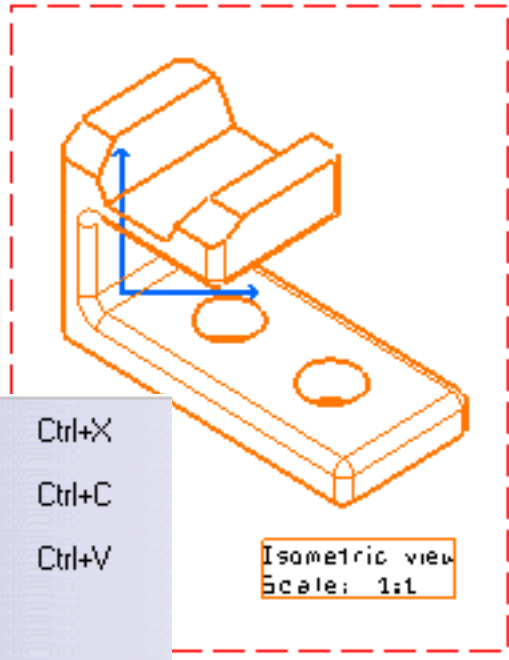
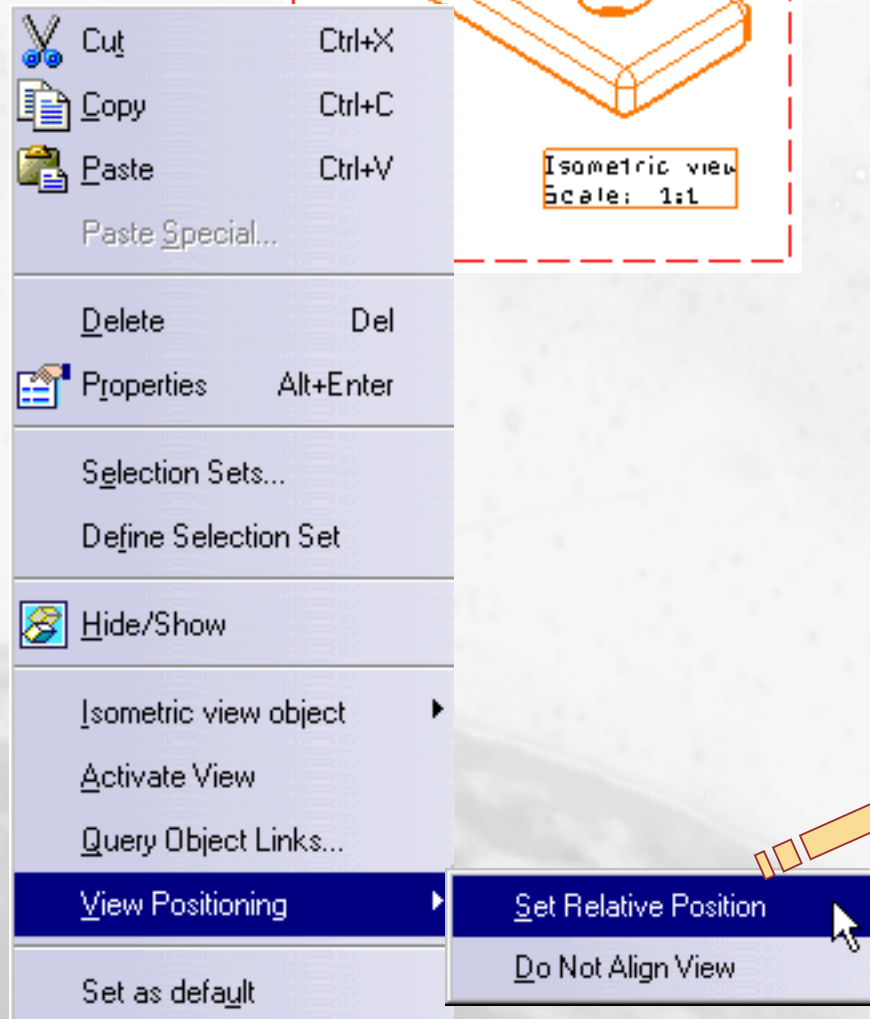
Positioning stick showing distance

Middle top anchor point



This is often used for isometric views, and particularly for exploded views of assemblies

Repositioning a view with Relative Positioning (1/2)



- 1 Select « View Positioning » then « Set Relative Position » in the contextual menu

A positioning stick appears with anchor points around the view and in its center

Positioning Stick

Anchor points

The stick has 3 handles:

- the green end
- the stick
- the black end

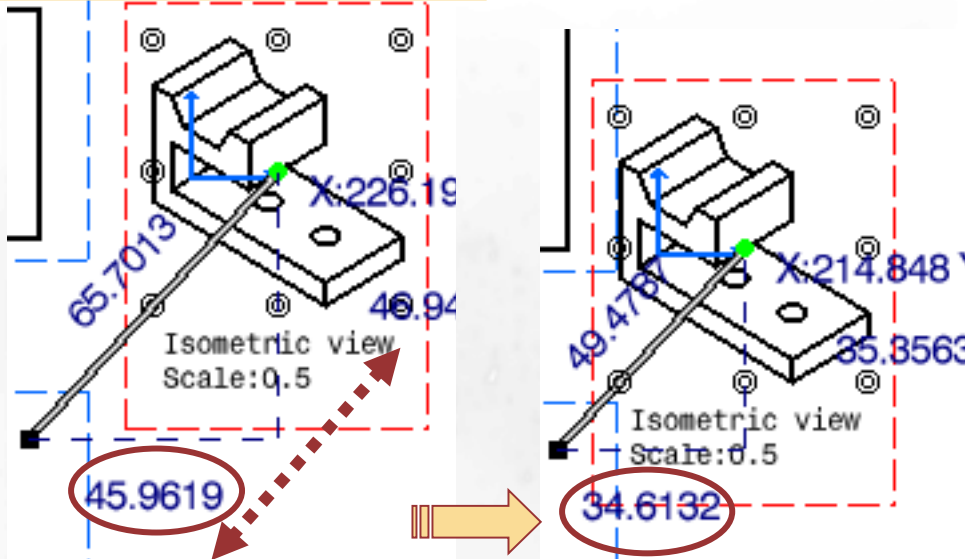
By default, the green handle is on the center anchor point and the stick is oriented SW (south-west).

Note: Remember the contextual menu can be accessed from the tree structure or the view frame. Therefore, the view frame does not need to be on to use the relative positioning option.

Repositioning a view with Relative Positioning (2/2)

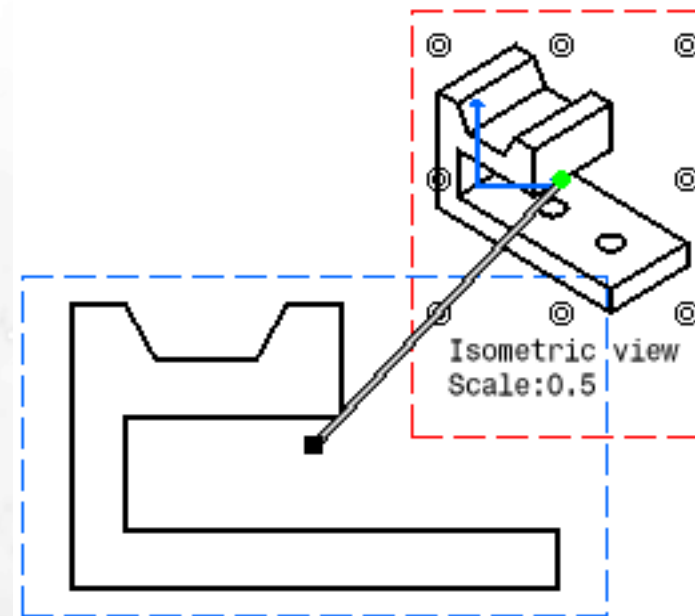
- 2 There are 4 ways to move a view with the positioning stick
You need to combine these to position the view

(A) Selecting the Stick



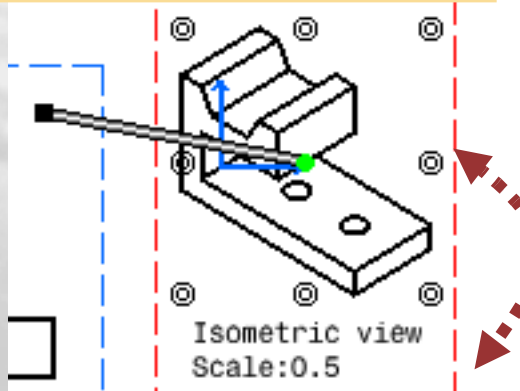
Stick length and distances values appear. Drag along the stick to make it shorter or longer: the view moves accordingly.

(B) Selecting the black end and another view frame



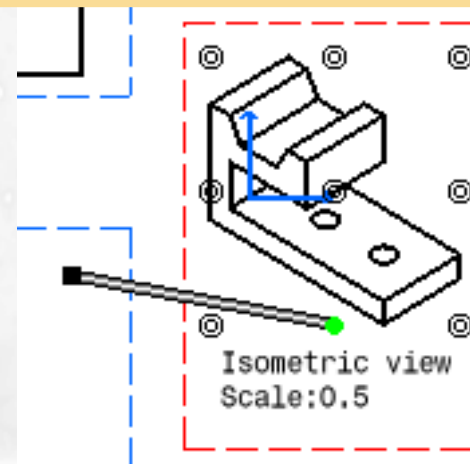
The black end snaps on the center of the selected view frame and the view follows

(C) Selecting the green handle



Drag to rotate the view around the black end

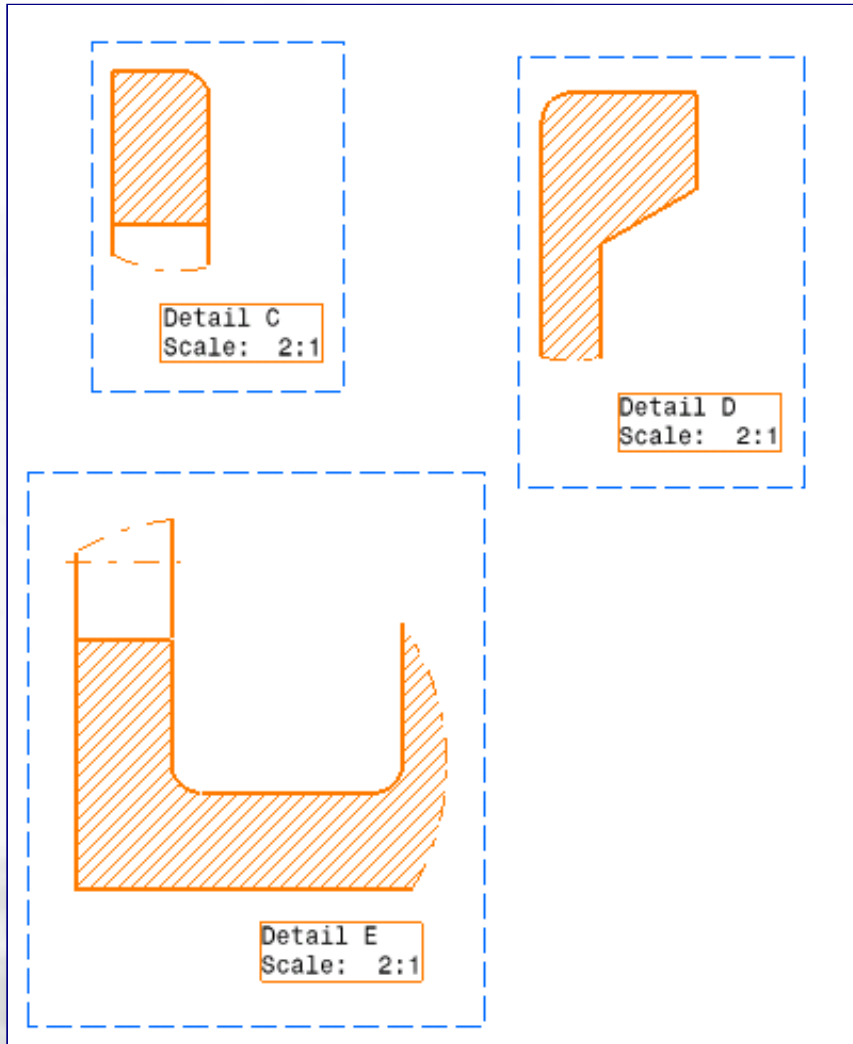
(D) Selecting another anchor point



The green end moves to the selected anchor point and the view is moved (ex. here bottom middle anchor was selected; view moved up)

Moving views on a sheet with the Positioning Tool

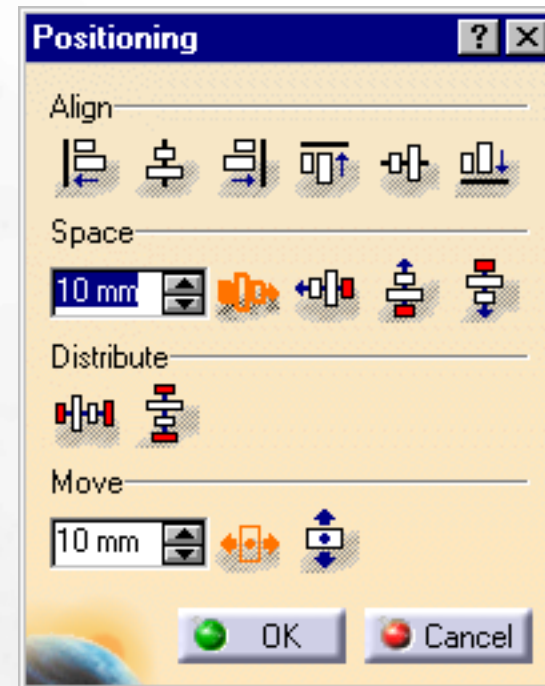
1 Select the view or views to be aligned



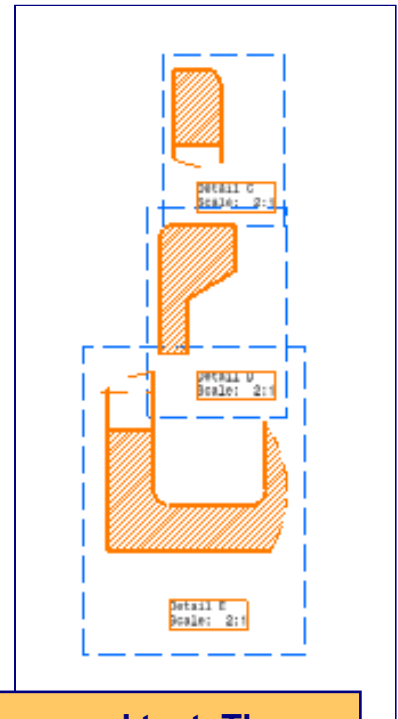
2 Select the positioning tool icon



3 Select the desired positioning option (Vertical Distribute)



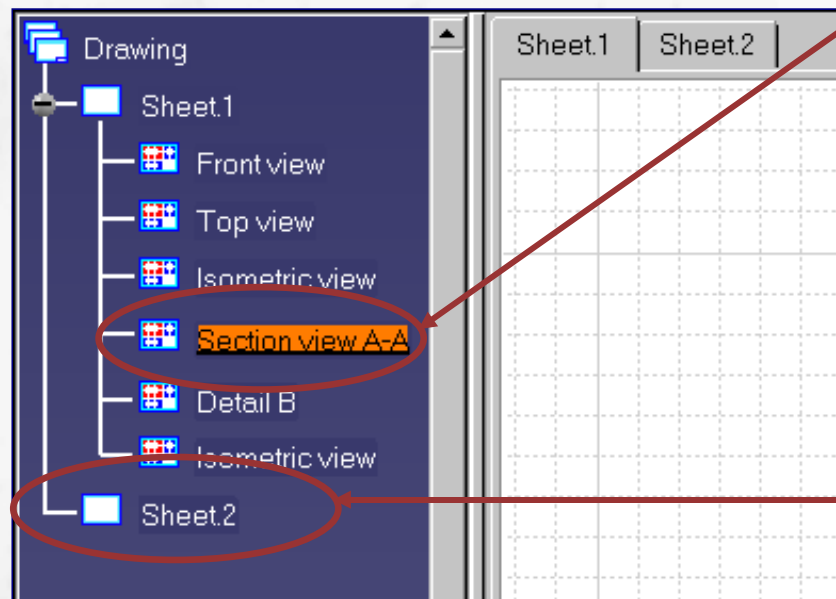
4 The views will move accordingly (aligned vertically at an evenly distributed distance)



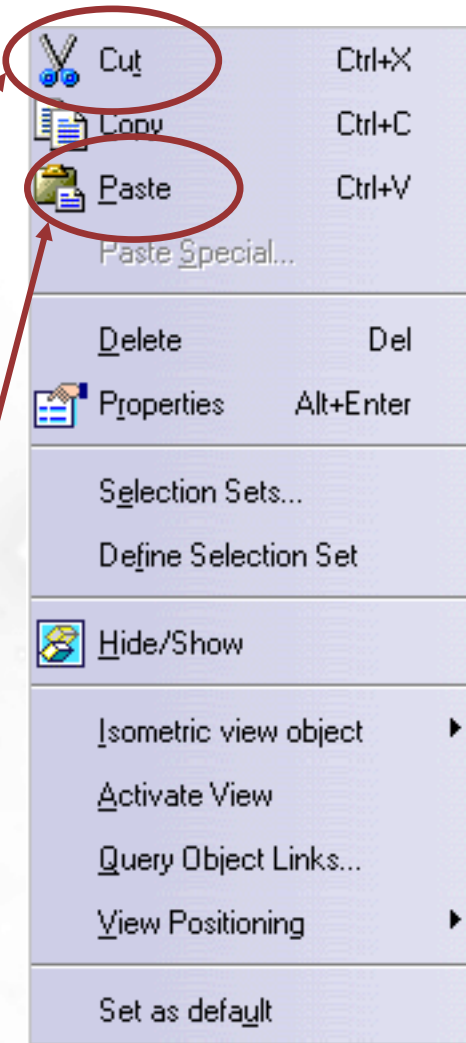
The positioning tool can be position both views and text. They can be aligned from an element, Spaced vertically or horizontally, distributed or even moved a specific distance

Moving views from one sheet to another sheet

1 Cut the view on the sheet to be moved



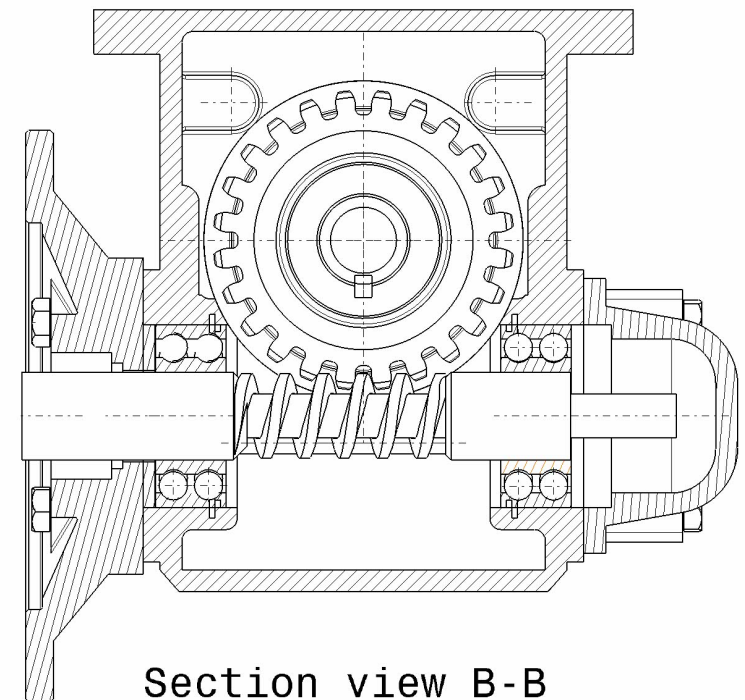
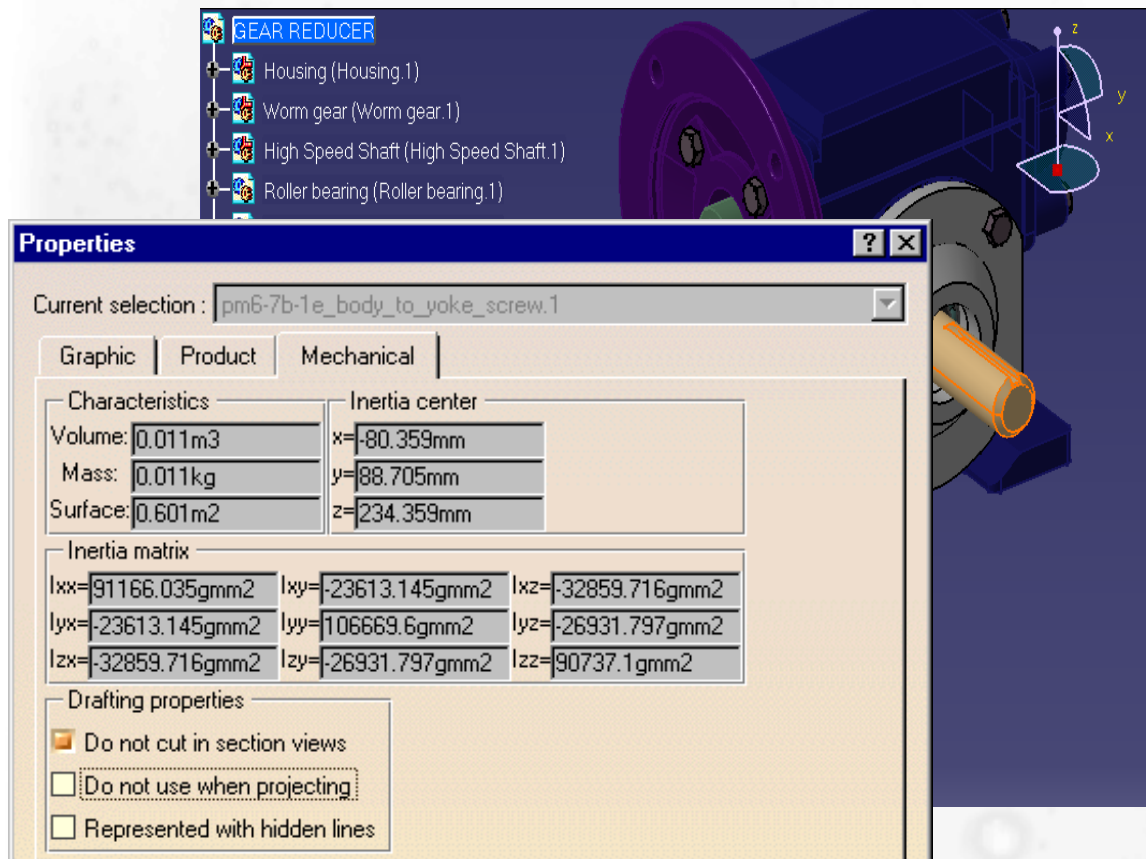
Contextual Menu



2 Paste the cut view on the desired sheet in the specification tree

Modifying Views

You will learn how to modify views by deleting views, isolating views, duplicating geometry in a view, and product instance filtering in a view.



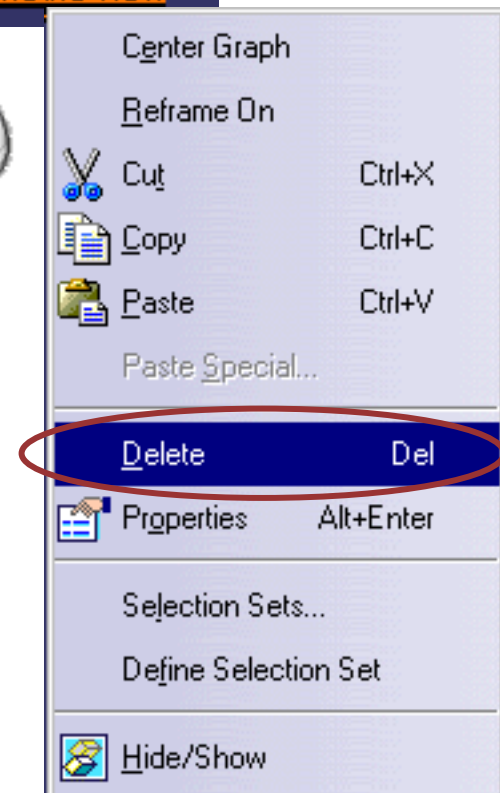
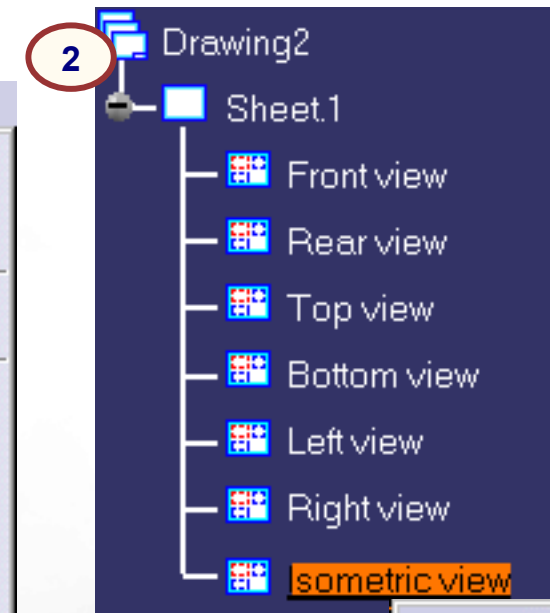
Section view B-B
Scale: 1:3

Deleting Views

Views can be selected from the specification tree or from the geometry on the drawing.

They can be deleted with the following options:

- 1) **Edit + Delete** to delete the selected views
- 2) **Contextual Menu Delete** option
- 3) Using the **Delete** key on the keyboard to delete the selected views

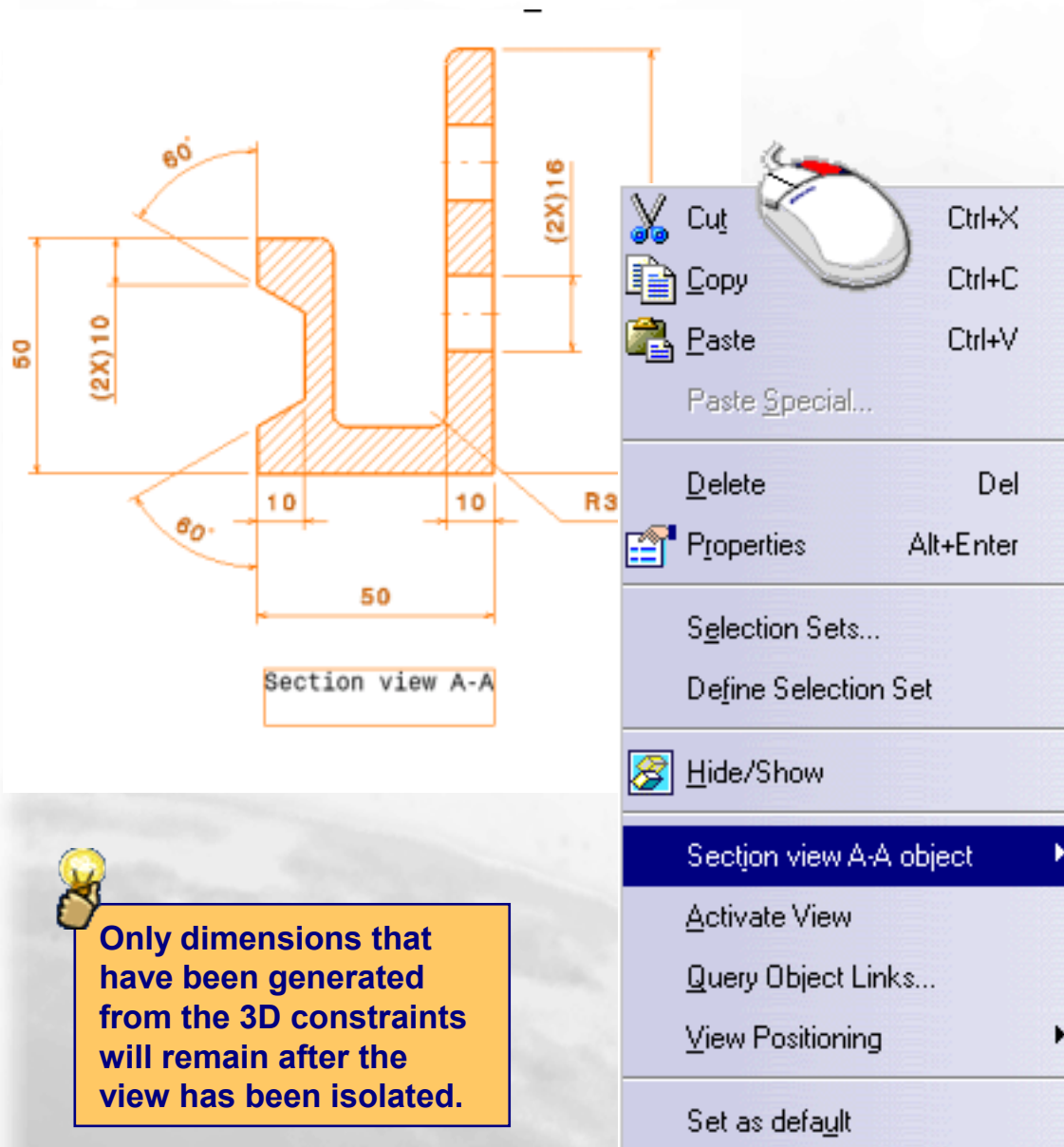


More than one view can be deleted by Multi-selecting from the tree or the geometry.

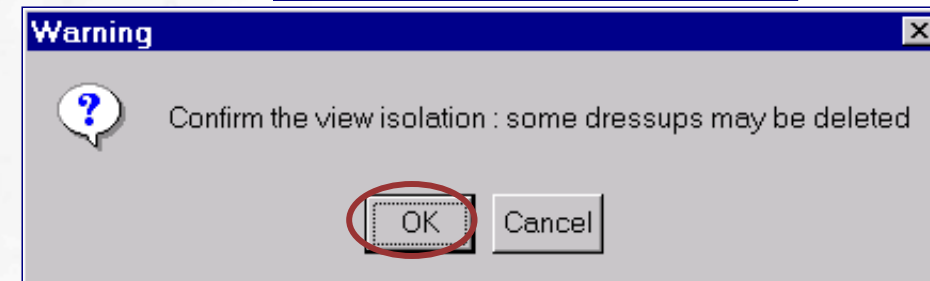
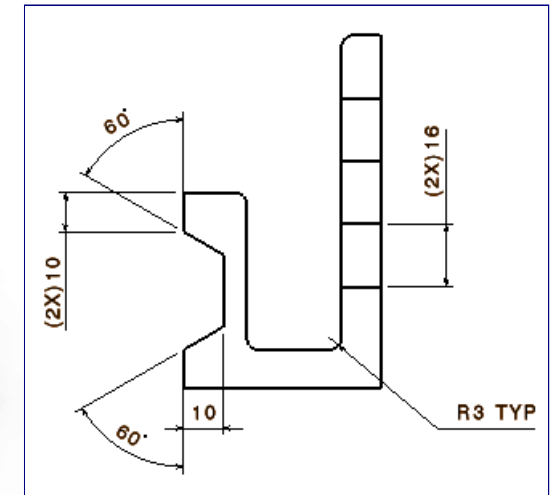
Isolation of a Generative View

Generated Views can be isolated from the 3D geometry and therefore no longer be associative to the parent geometry.

- 1 Select **Isolate** option with the Contextual Menu



- 2 Select **OK** to acknowledge the warning "some dressup elements may be deleted" and to accept the Isolation.



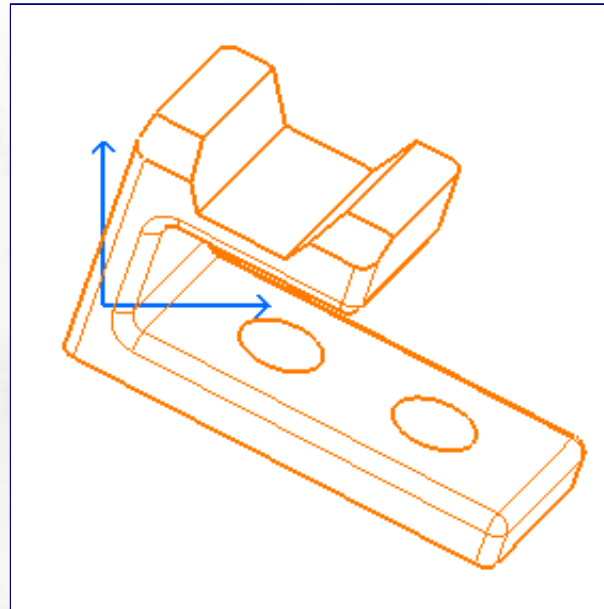
Once a view is Isolated the link to the 3D data can never be reestablished.

Duplicating elements in a Generative View

Duplicate interactive elements can be created over the generative elements

1 Activate the View containing the elements to duplicate

2 Within the active view, select the elements to be duplicated



3 Select the contextual menu Duplicate Geometry



Note: The selected geometry is duplicated on the view at the same position. It is highlighted after creation



The Generative elements that lie underneath the interactive elements will be the only elements effected by the 3D geometry changes.

Product Part Filtering in Views (1/3)

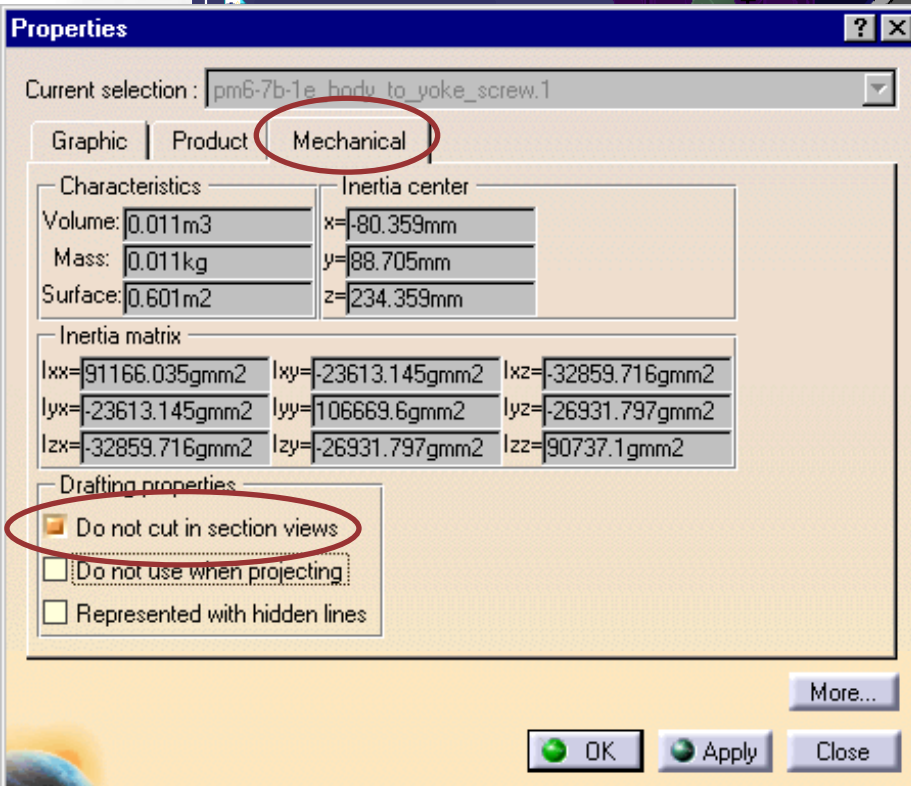
Parts within a product can be set to not be sectioned when generating views

1 Within the Assembly Design Workbench, select the part to define

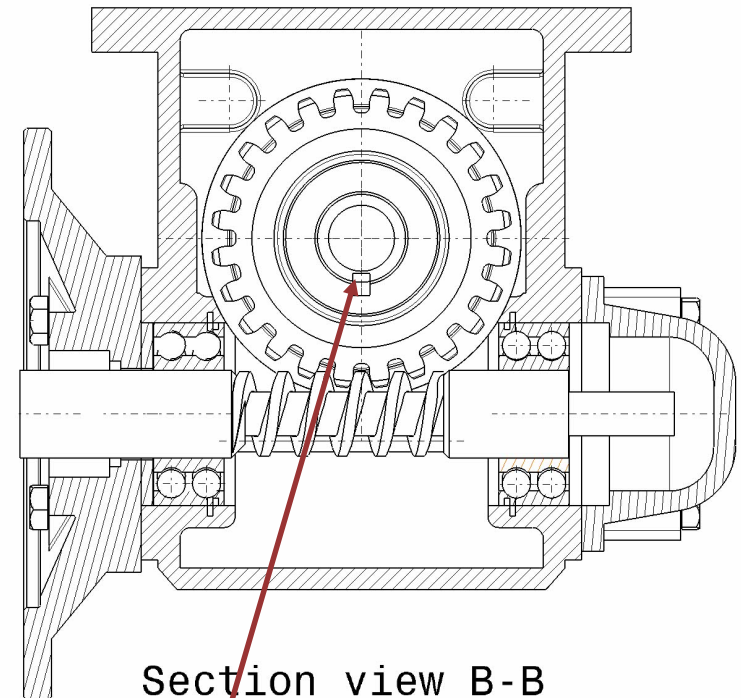
2 From the contextual menu, select Properties



3 Select the Mechanical tab and activate the “Do Not cut in section views” option



4 Create the section or breakout view or update an existing view



Section view B-B
Scale: 1:3

Part **not** cut in
context of assembly

Product Part Filtering in Views (2/3)

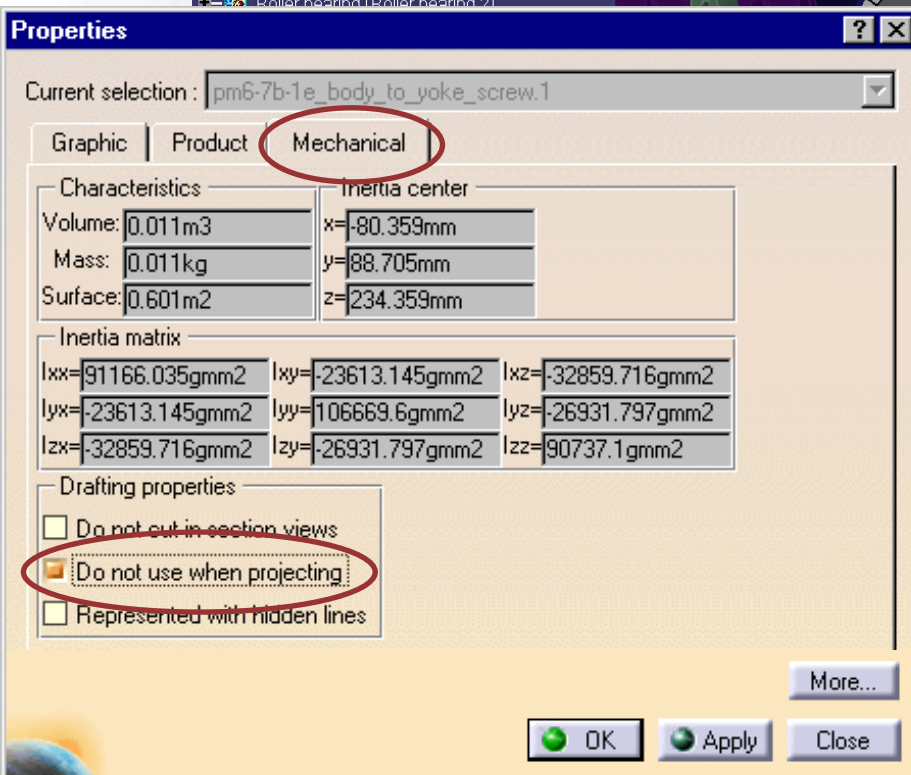
Parts within a product can be set to not be used when projecting views

1 Within the Assembly Design Workbench, select the part to define

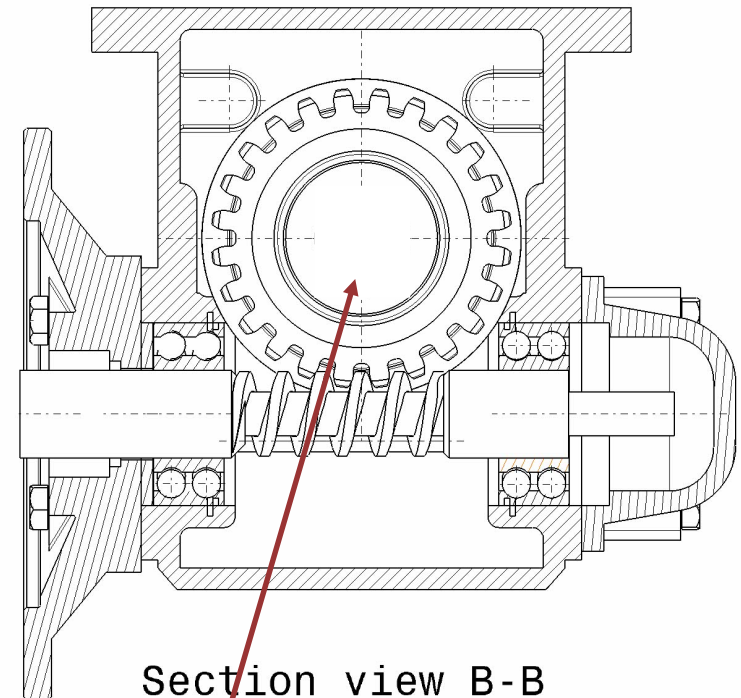
2 From the contextual menu, select Properties



3 Select the Mechanical tab and activate the “Do Not use when projecting” option



4 Create the section or breakout view or update an existing view



Section view B-B
Scale: 1:3

Part **not** used in
context of assembly

Product Part Filtering in Views (3/3)

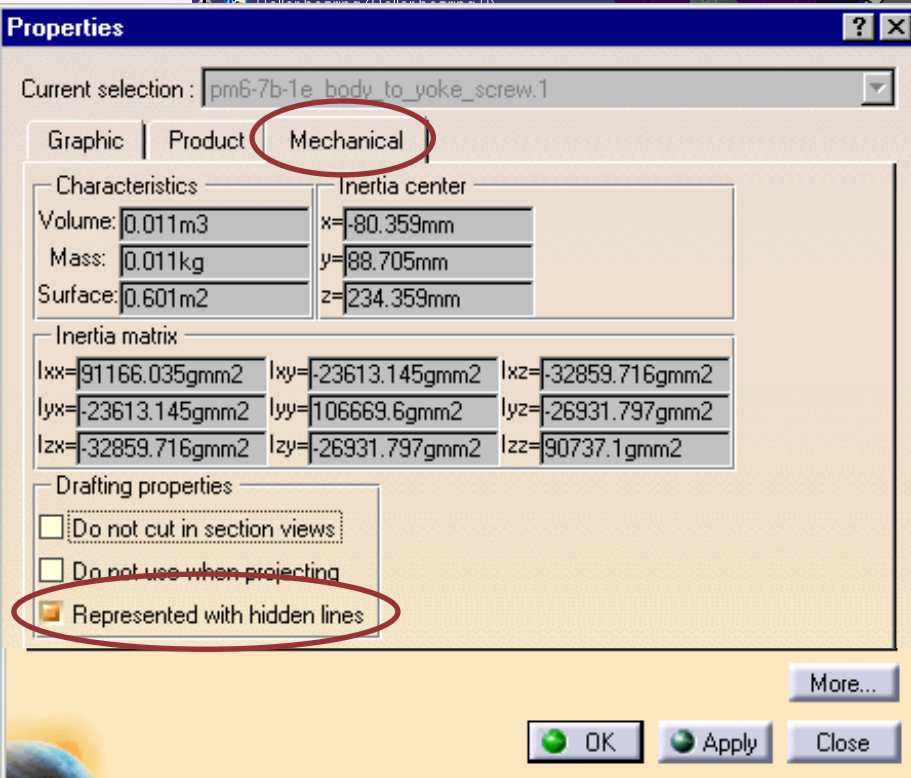
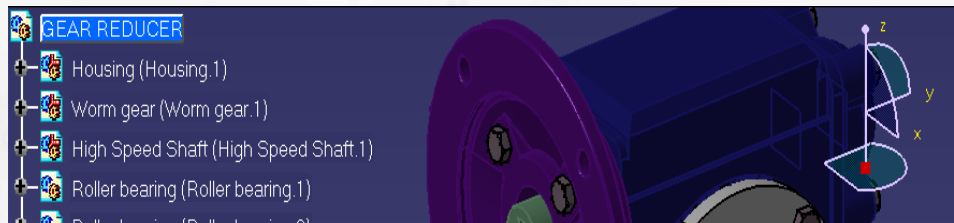
Parts within a product can be set to be represented with hidden lines in views

1 Within the Assembly Design Workbench, select the part to define

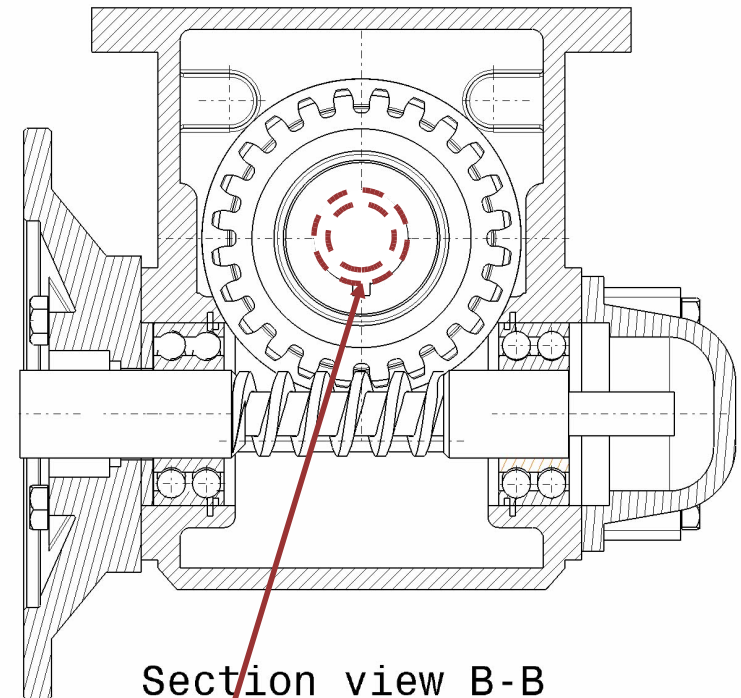
2 From the contextual menu, select Properties



3 Select the Mechanical tab and activate the "Represented with hidden lines" option



4 Create the section or breakout view or update an existing view



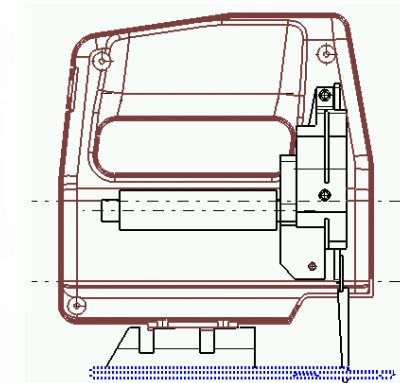
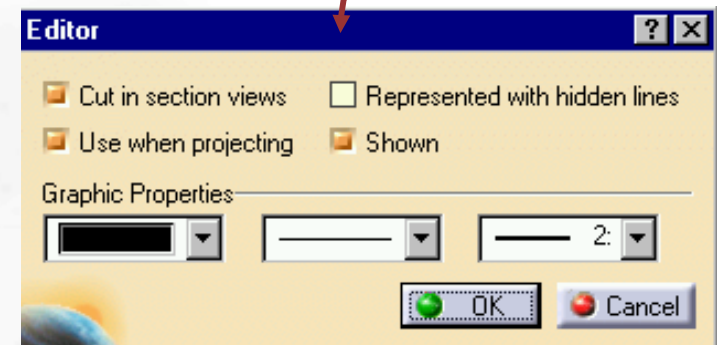
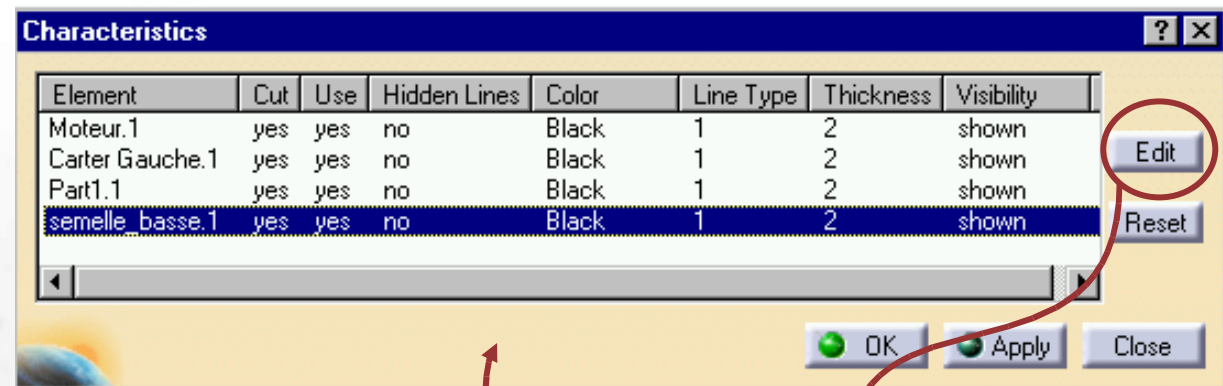
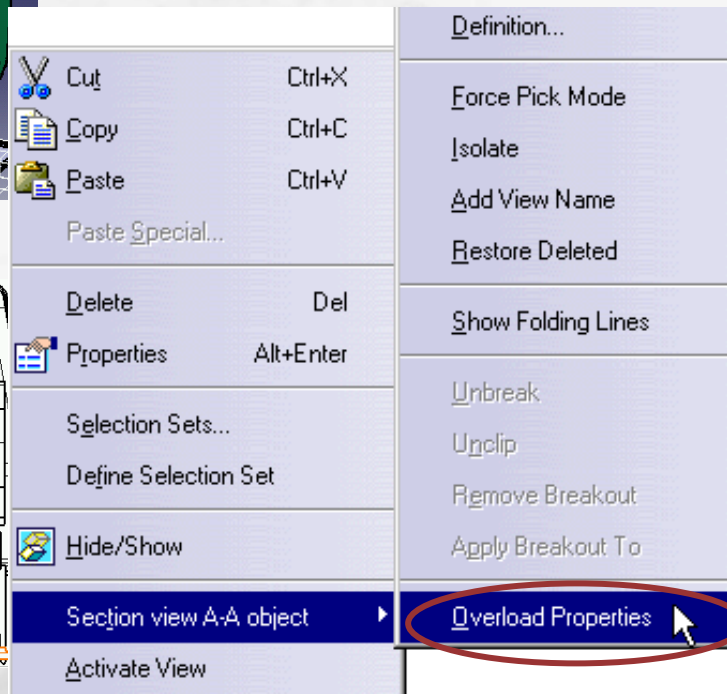
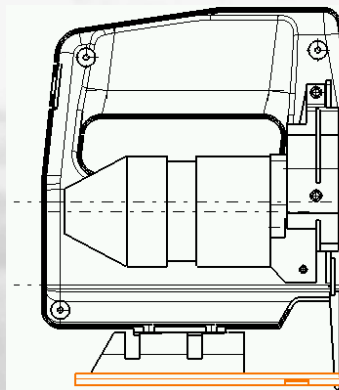
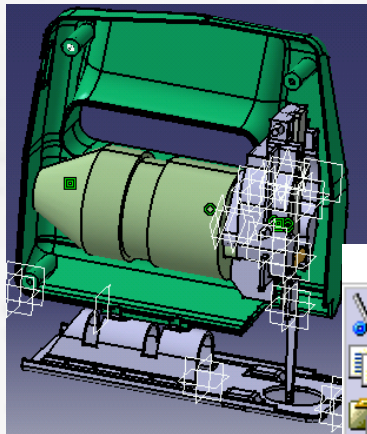
Section view B-B
Scale: 1:3

Part represented with hidden lines in context of assembly

Product Filtering Management for each View

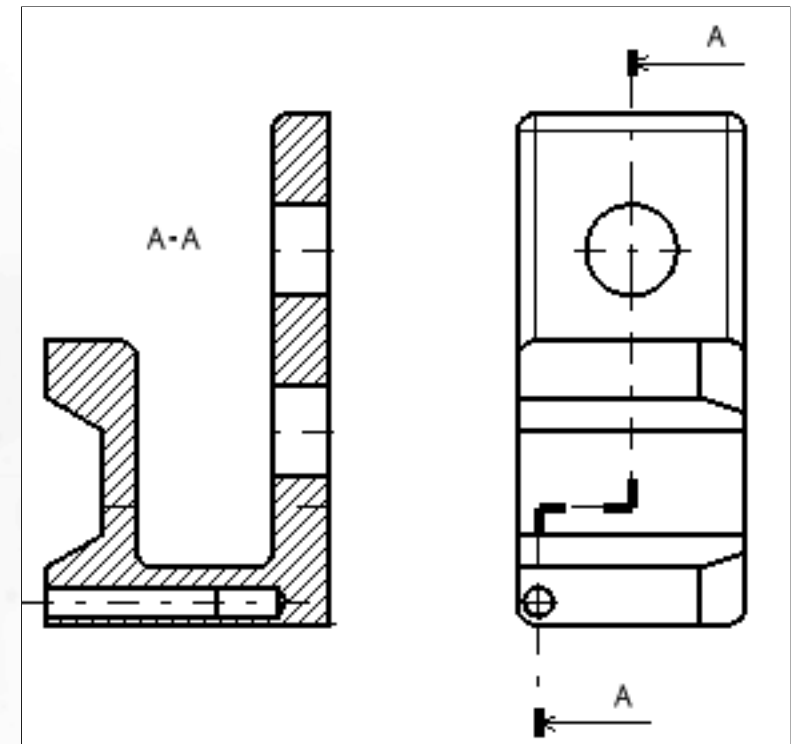
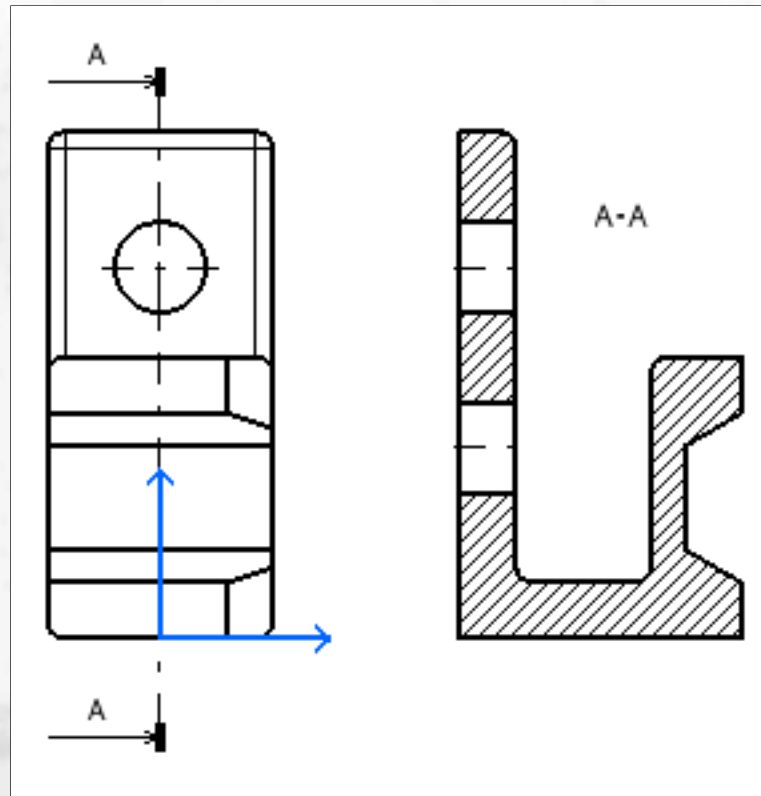
Product Instances filtering for each view

- ◆ Show/No show
- ◆ Use/Unuse
- ◆ Cut/Uncut
- ◆ Color



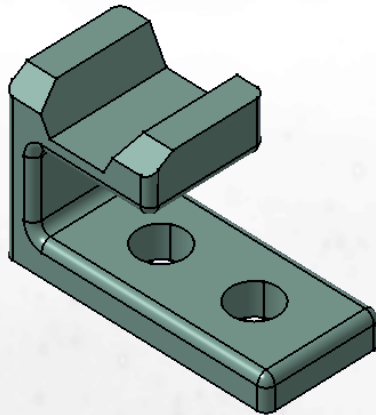
Modifying of Section, Detail, Auxiliary Views

You will learn how to modify the geometrical properties of the definition profile of Section, Detail and Auxiliary views..

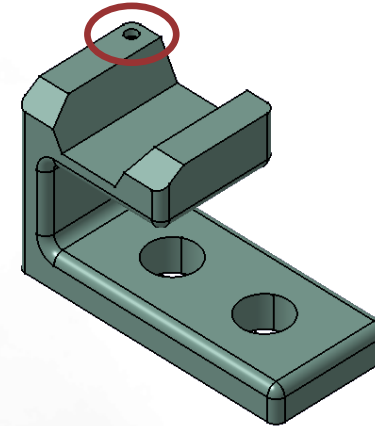


Why change the Profile?

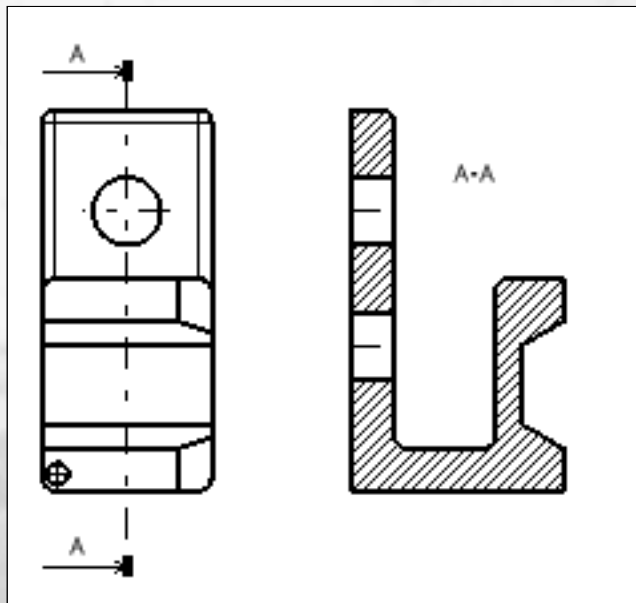
The design of your parts or products will evolve with time. CATIA allows you to modify the arrangement of views, and to modify the section, detail and auxiliary views to clarify your drawings.



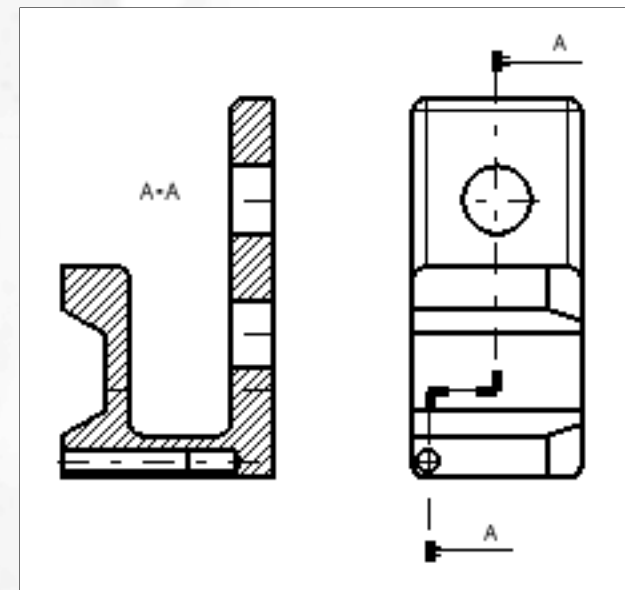
Original part



New version of the part



Original associated drawing



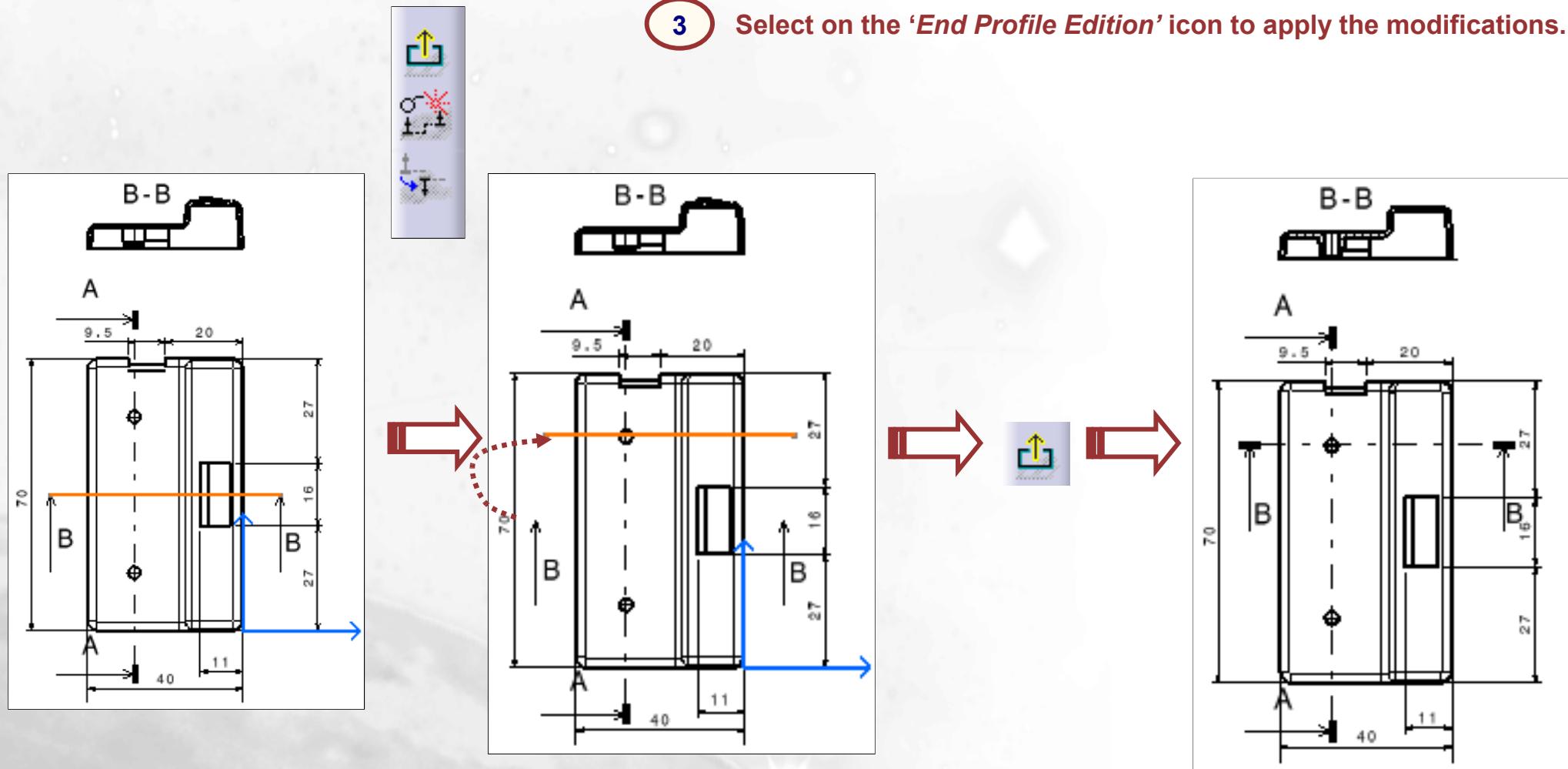
Updated drawing, section view modified

Modifying Section View Profile (1/3)

1 Double click on the Section view callout to open the 'Edit/Replace' toolbar which allows you to perform several kinds of modifications.

2a Move the section profile : select the callout. Drag and drop at a new location.

3 Select on the 'End Profile Edition' icon to apply the modifications.

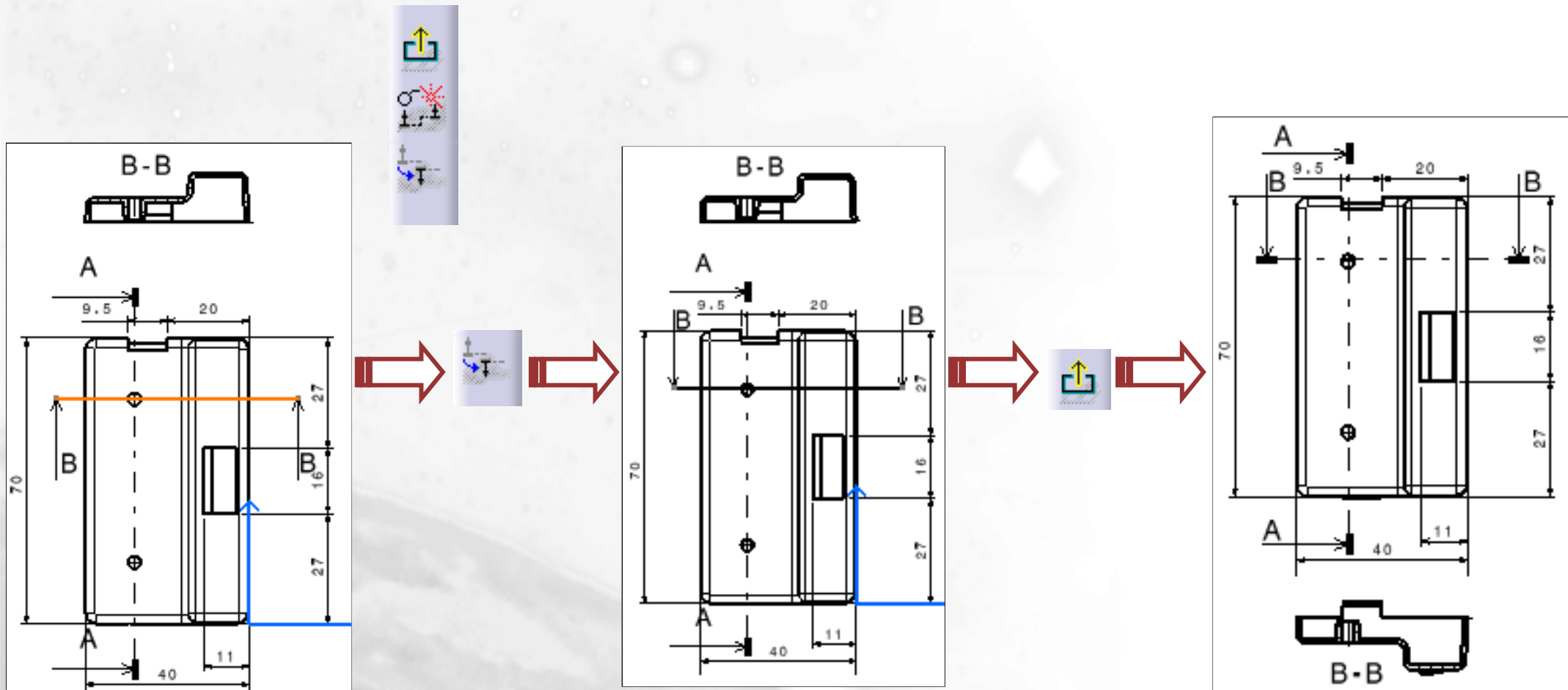


Modifying Section View Profile (2/3)

1 Double click on the Section view callout to open the 'Edit/Replace' toolbar which allows you to perform several kinds of modifications.

2b Inverse the view direction : select the '*InvertProfile direction*' icon.

3 Select on the '*End Profile Edition*' icon to apply the modifications.

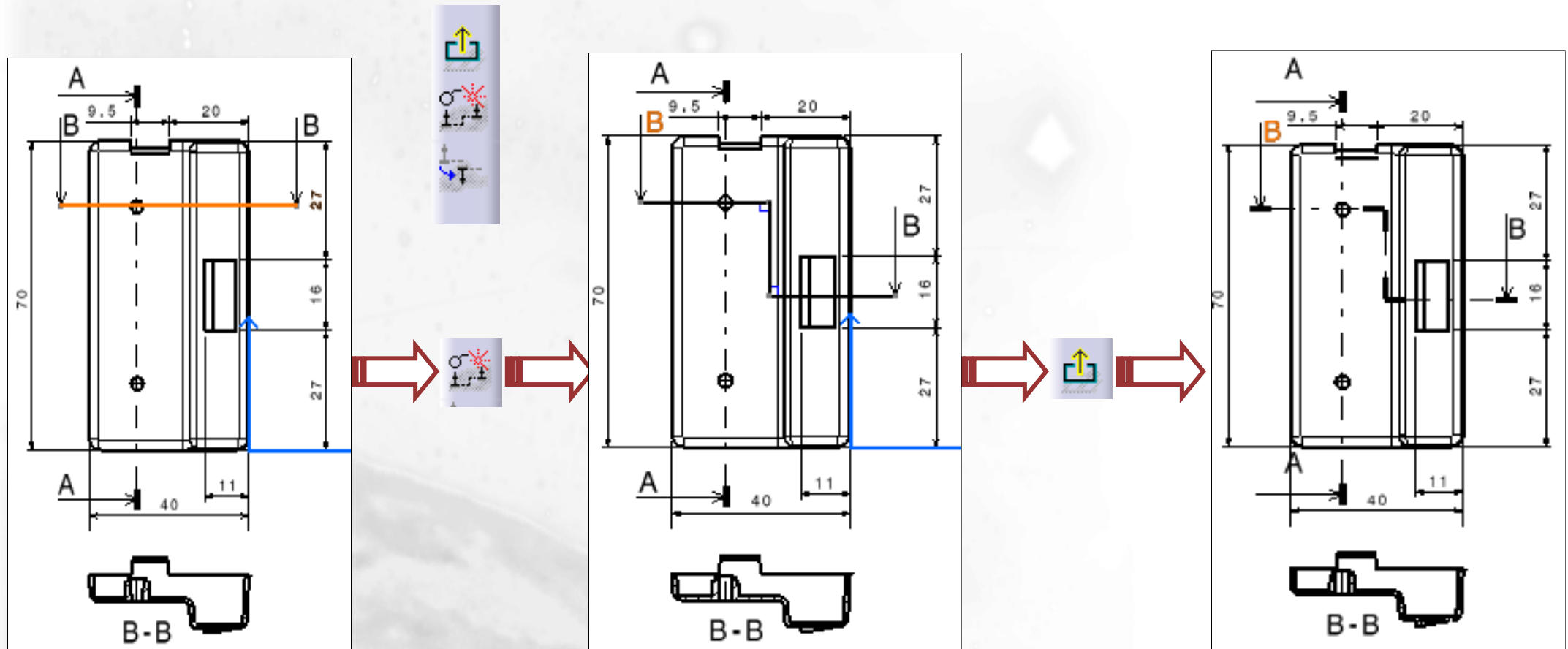


Modifying Section View Profile (3/3)


1 Double click on the Section view callout to open the 'Edit/Replace' toolbar which allows you to perform several kinds of modifications.

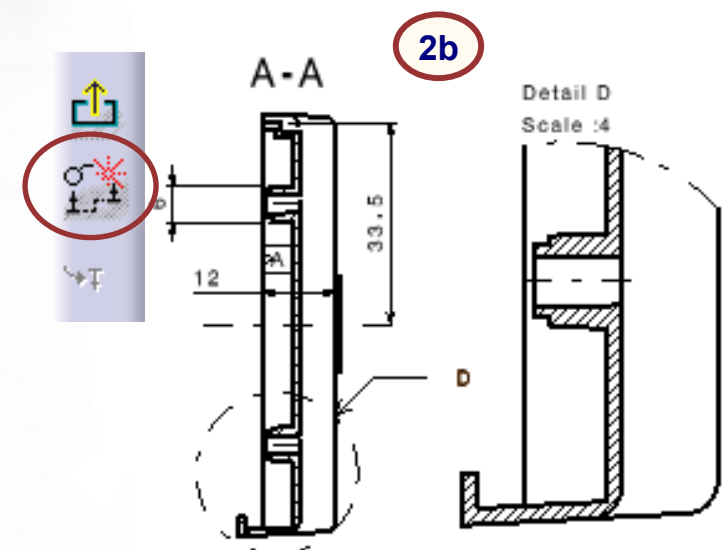
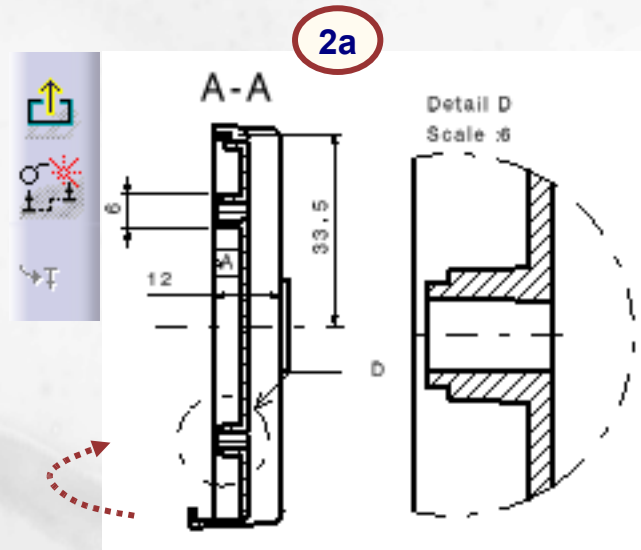
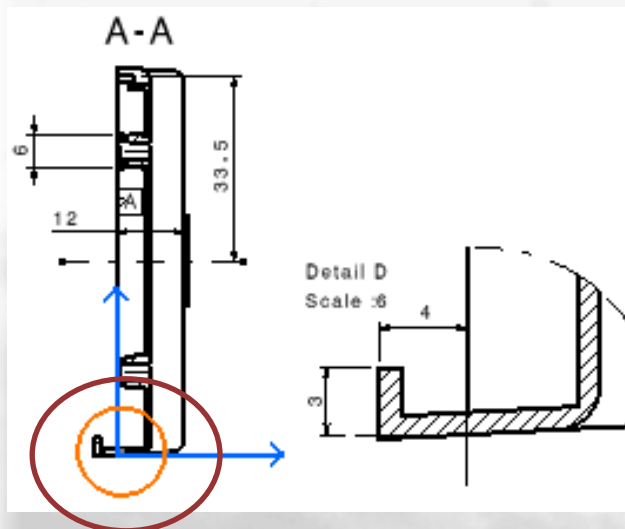
2c Replace the profile : select the 'Replace Profile' icon. Create your new profile to replace the old one.

3 Select on the 'End Profile Edition' icon to apply the modifications.



Modify Detail View Profile

- 1 Double click on the Detail view callout to open the 'Edit/Replace' toolbar which allows you to perform several kinds of modifications.
- 2
 - (a) Move the Detail profile : select the callout. Drag and drop it at the desired location.
 - (b) Replace the Detail view : select the 'Replace Profile' icon. Create your new detail callout profile.
- 3 Select on the 'End Profile Edition' icon  to apply the modifications.



Modifying Auxiliary View Profile

1

Double click on the Auxiliary view callout to open the '*Edit/Replace*' toolbar which allows you to perform several kinds of modifications.

2

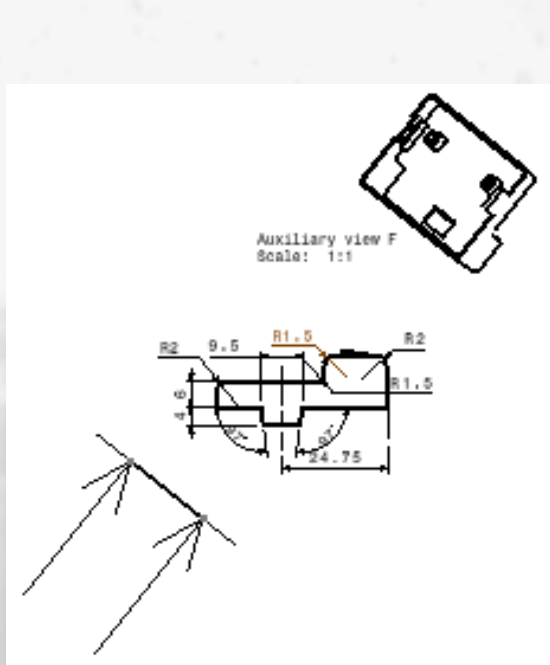
- (a) Move the Auxiliary view profile : select the callout. Drag it to a new location.
- (b) Inverse the view direction : select the '*Invert Profile direction*' icon.

3

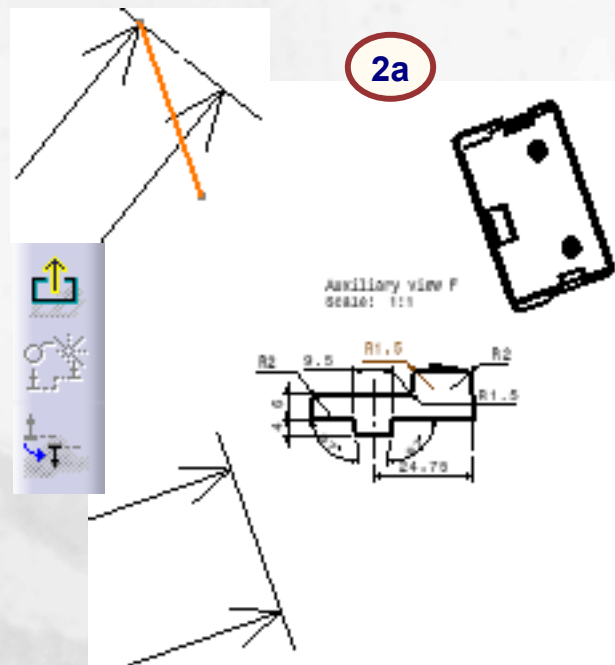
Select on the '*End Profile Edition*' icon



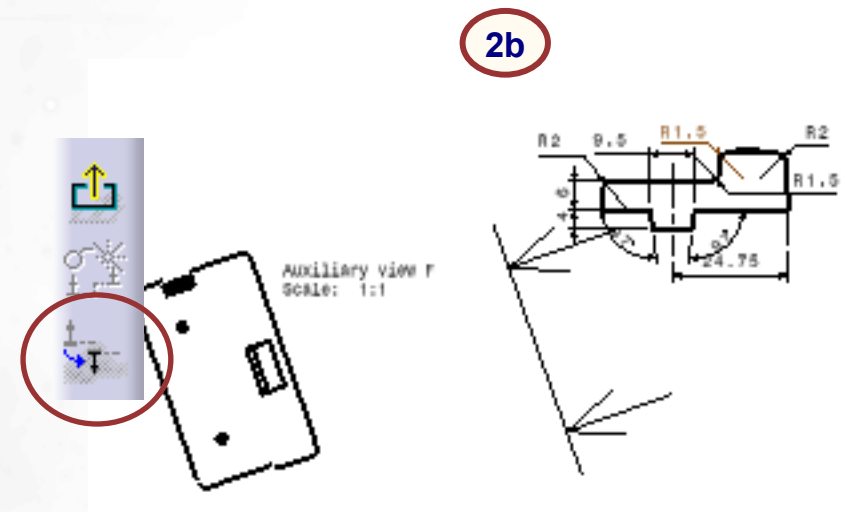
to apply the modifications.



2a

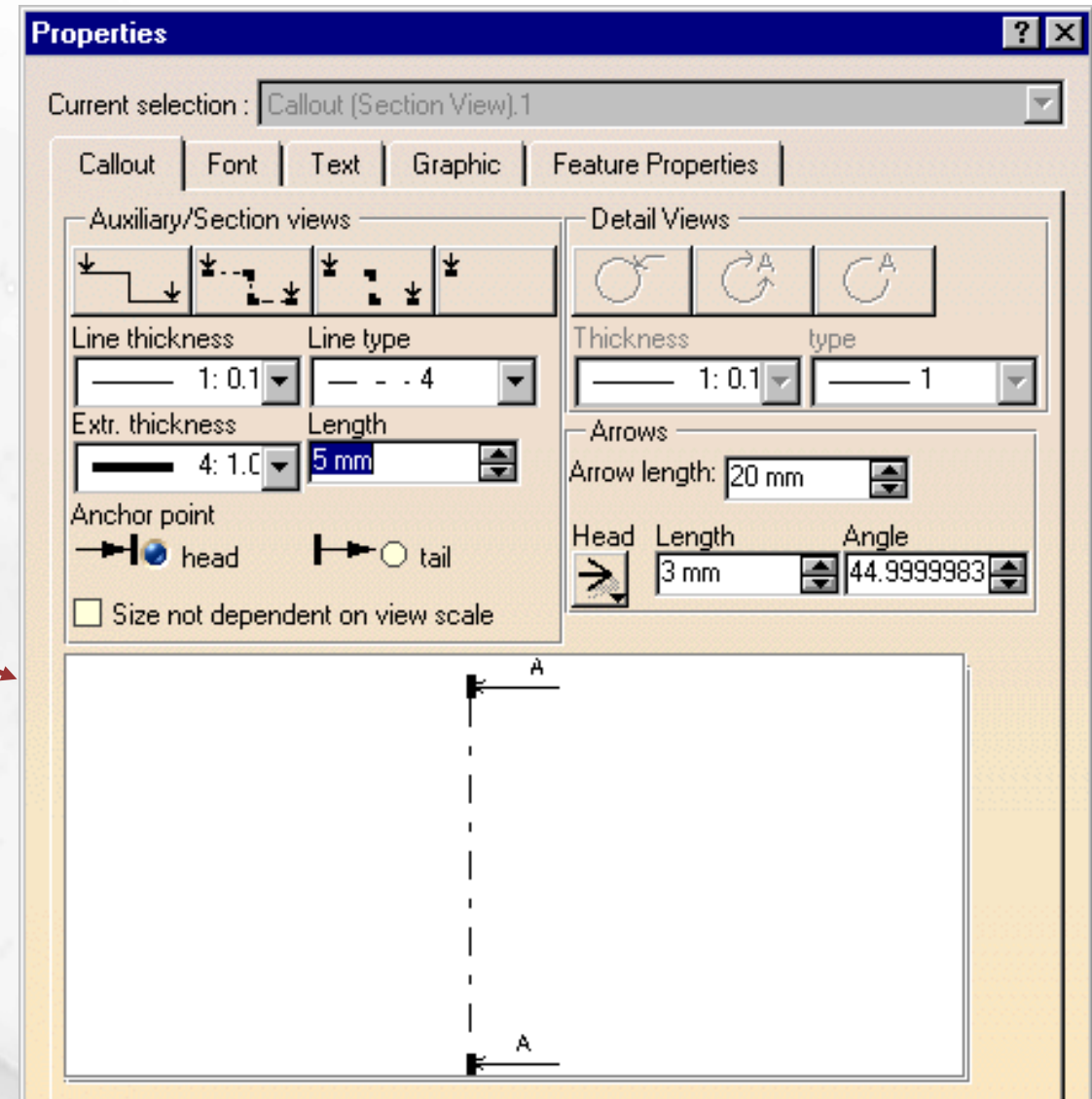
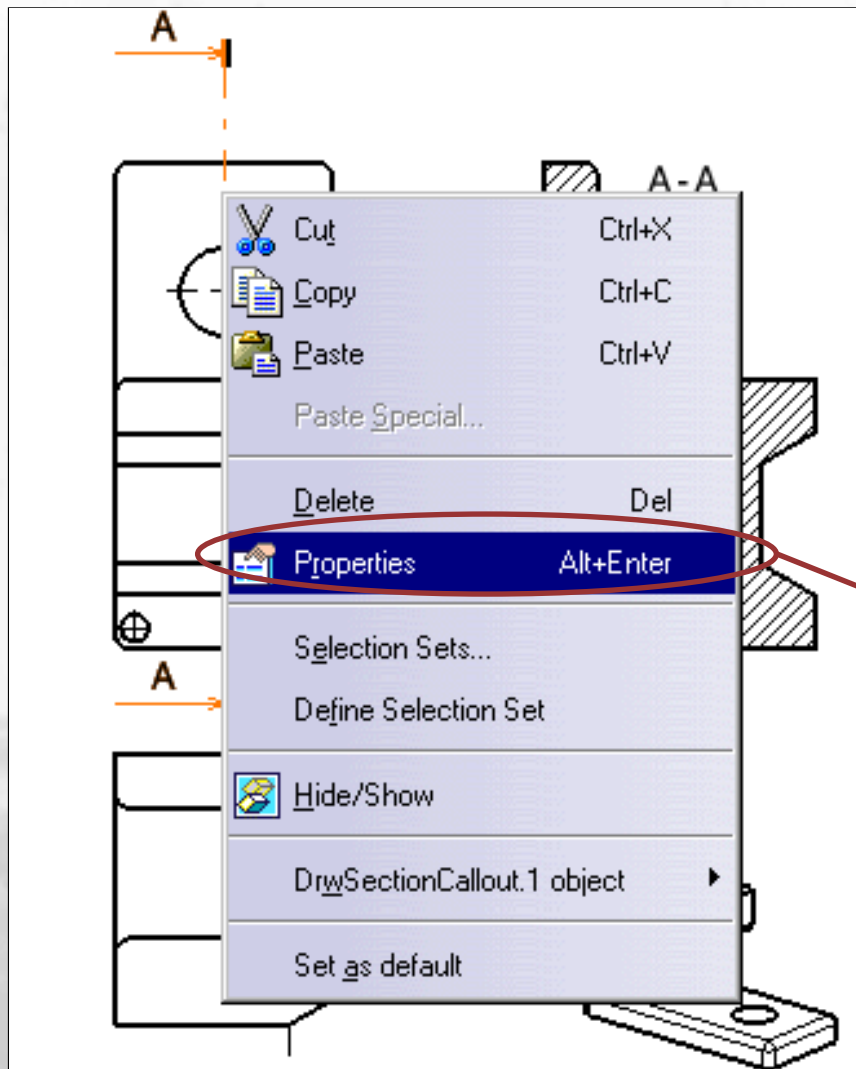


2b



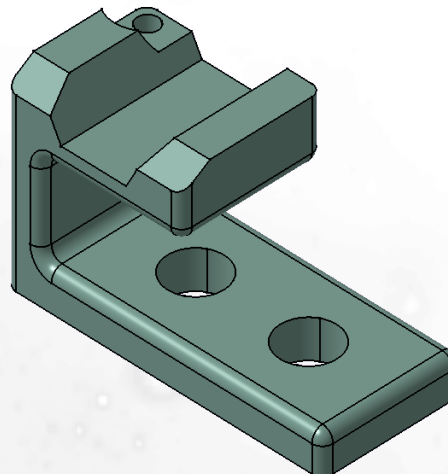
Modifying a Section, Detail, Auxiliary graphical definition.

You will learn how to modify the graphical attributes of the callout for Section, Detail and Auxiliary views.

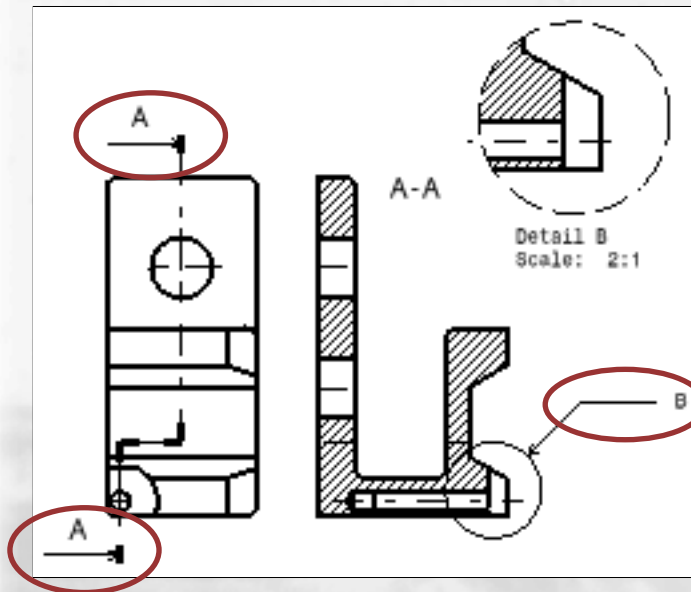


Why Change the Graphical Definition ?

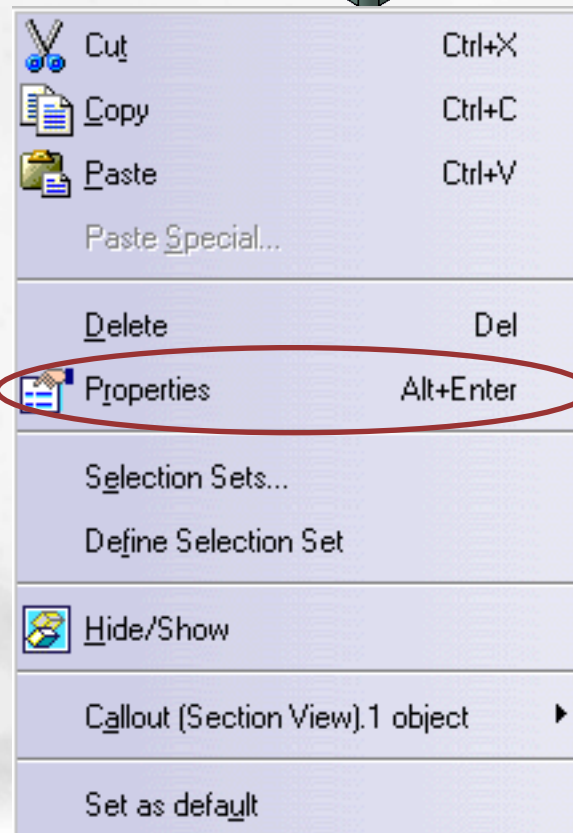
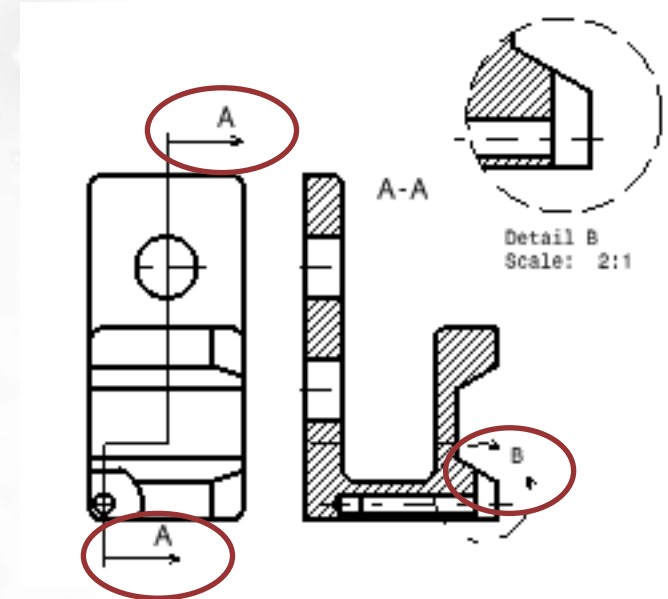
According to your or your customer's needs, CATIA allows you to modify the graphical attributes of Section, Detail or Auxiliary views. This allows you to clarify your drawings or to adapt them to different standards.



Original associated drawing



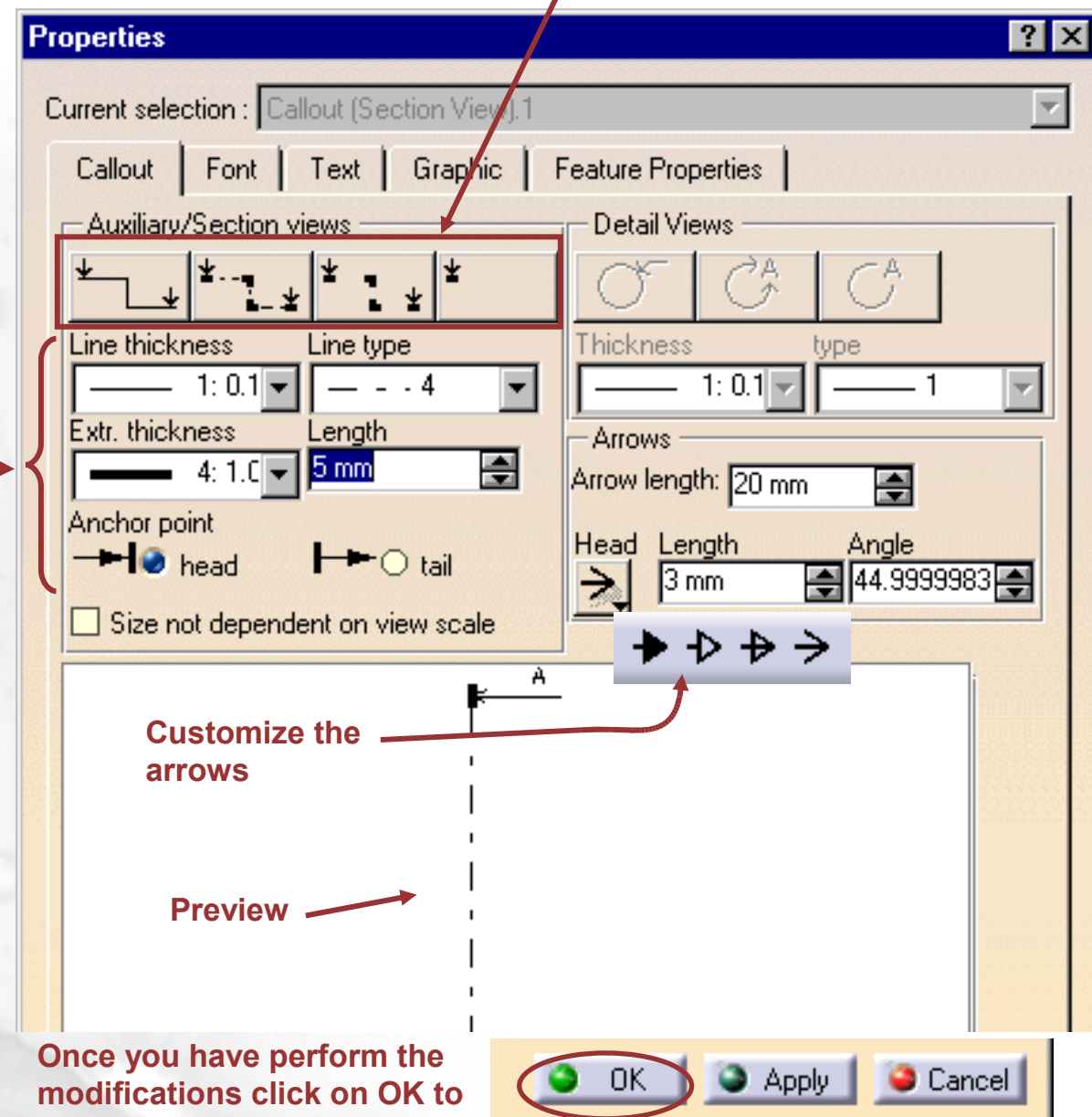
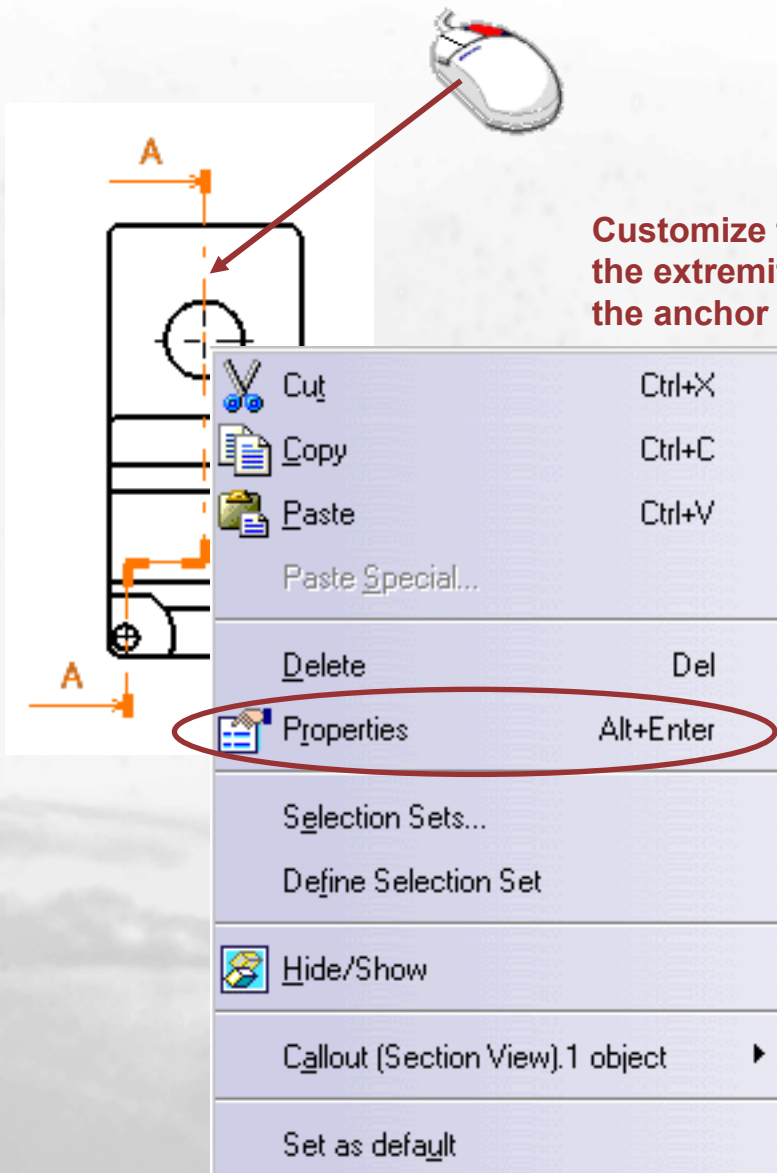
Modified associated drawing



Modifying Section View Graphical Definition

- 1 Select (or multi-select) the Section view callout to modify Properties with the contextual menu.

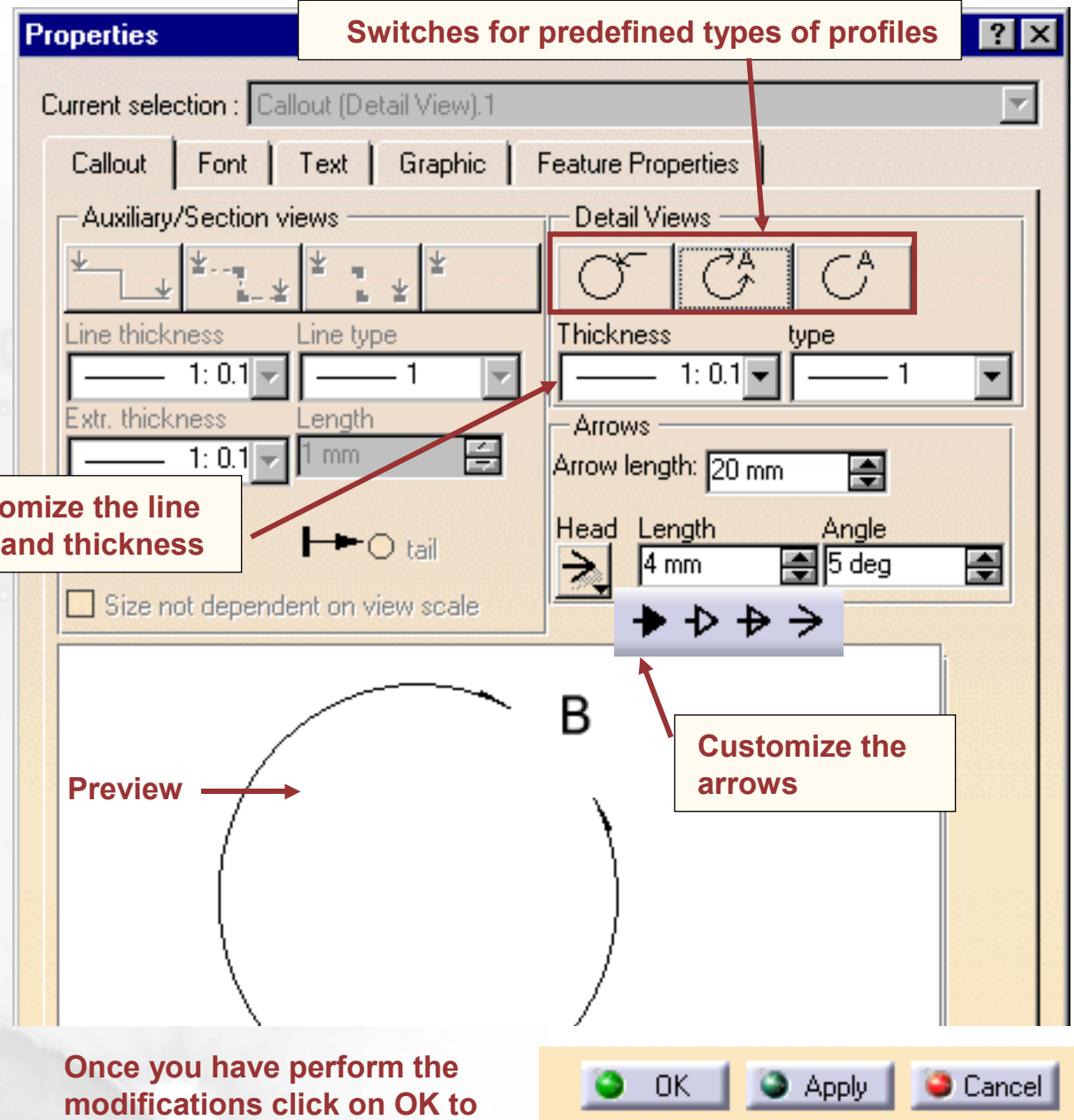
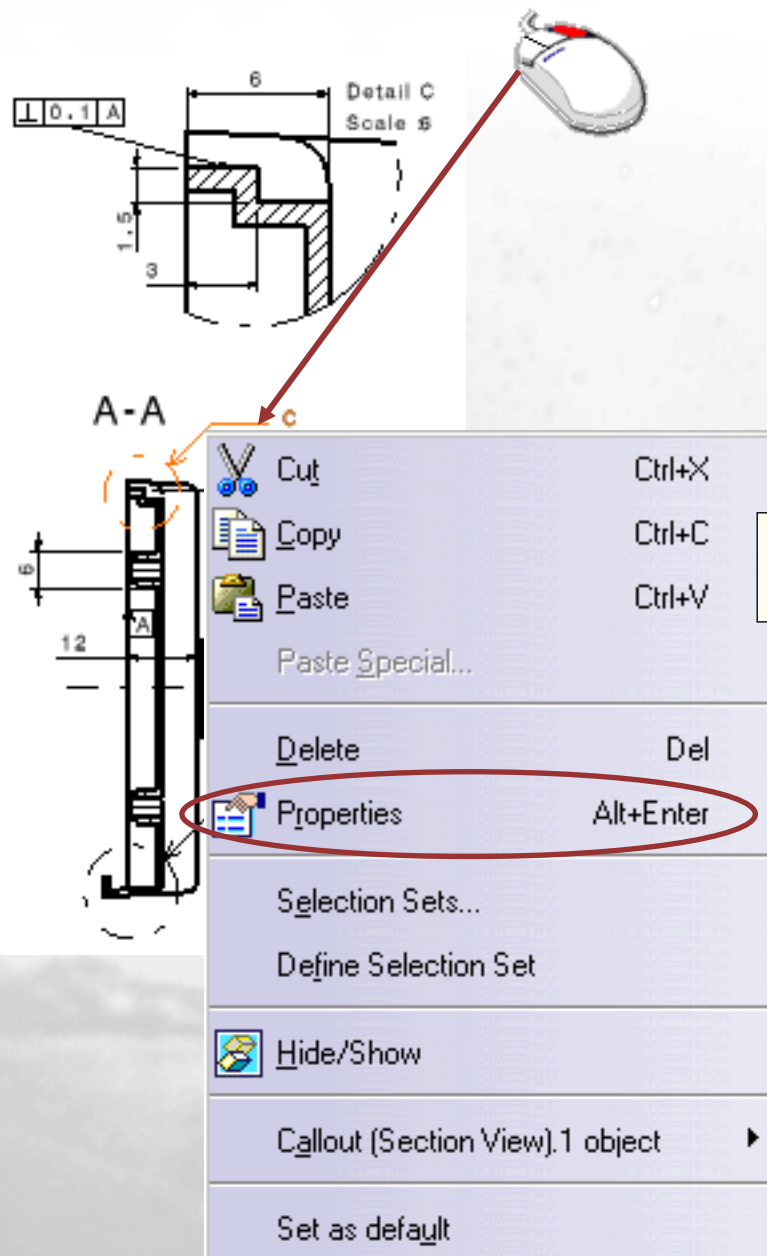
- 2 Select the Callout tab. Use the different commands to customize the drawing.



Modifying Detail View Graphical Definition

- 1 Select (or multi-select) the Detail view callout to modify and Properties with the contextual menu.

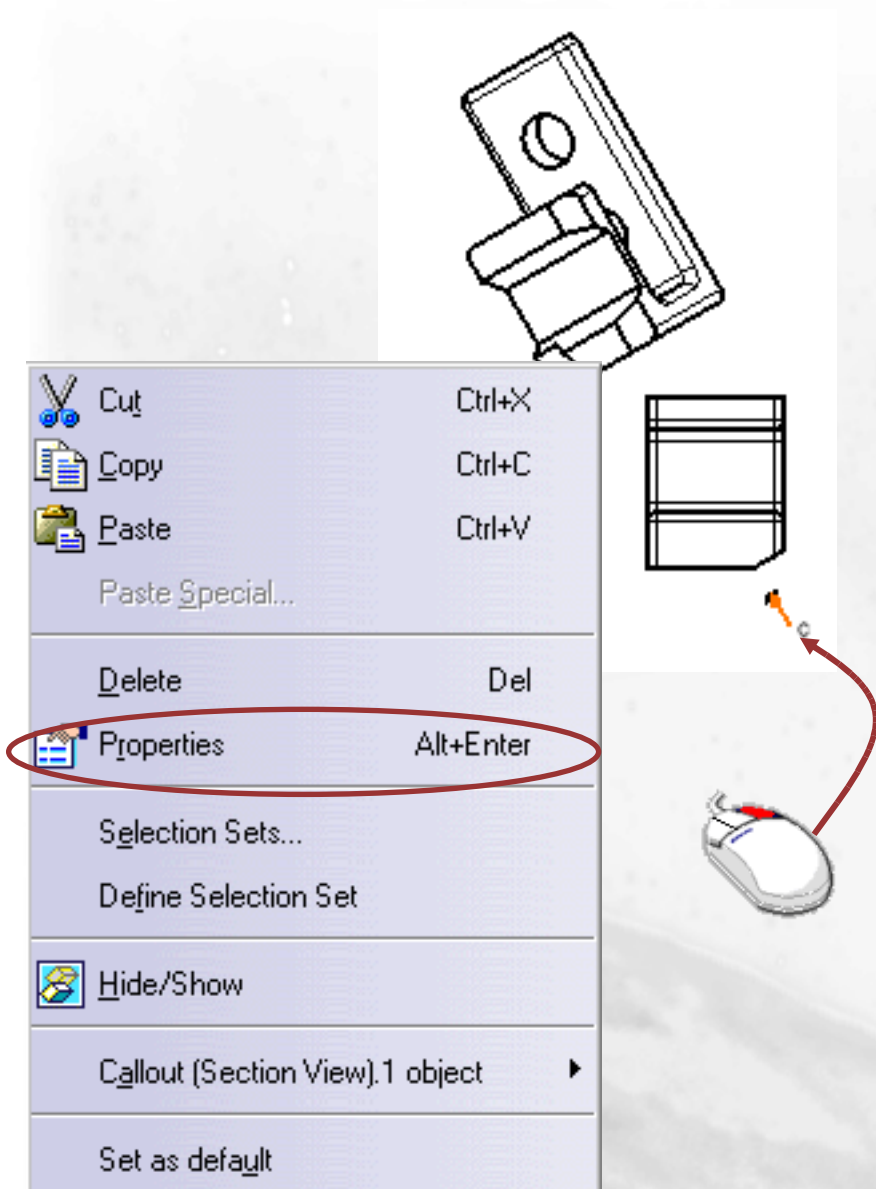
- 2 Select the Callout tab. Use the different commands to customize the drawing.



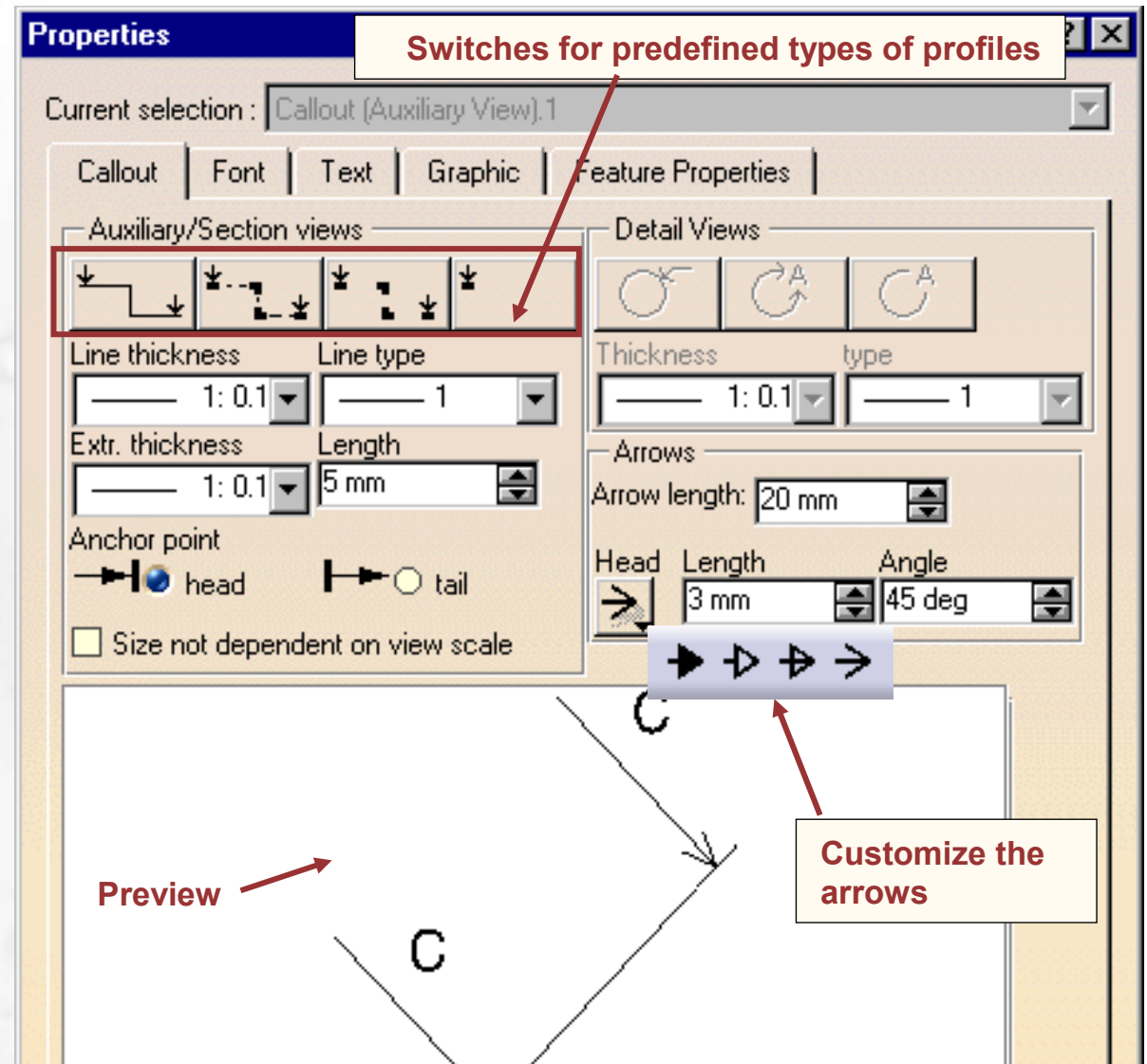
Once you have performed the modifications click on OK to validate.

Modifying Auxiliary View Graphical Definition

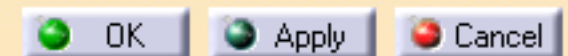
- 1 Select (or multi-select) the Auxiliary view callout to modify and Properties with the contextual menu.



- 2 Select the Callout tab. Use the different commands to customize the drawing.

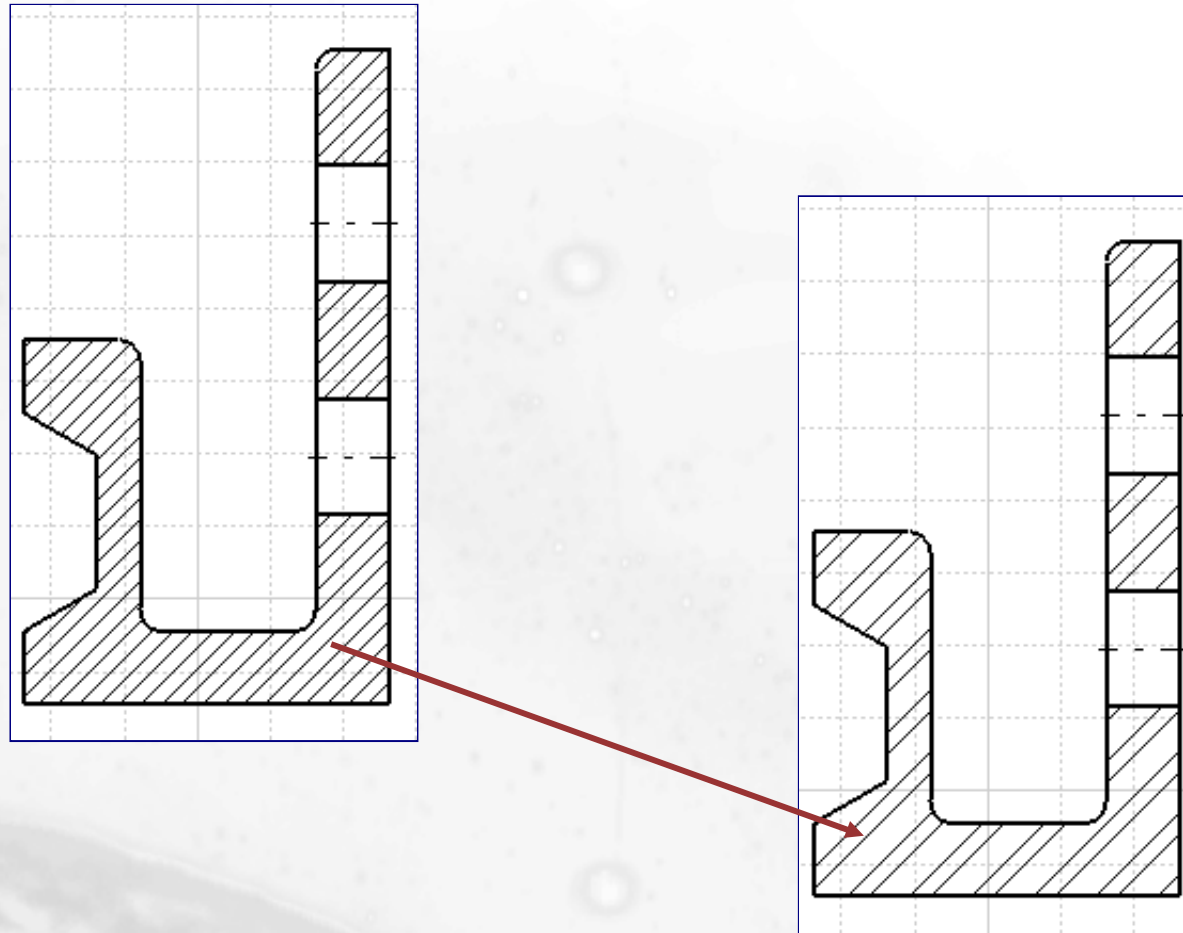


Once you have perform the modifications click on OK to validate.



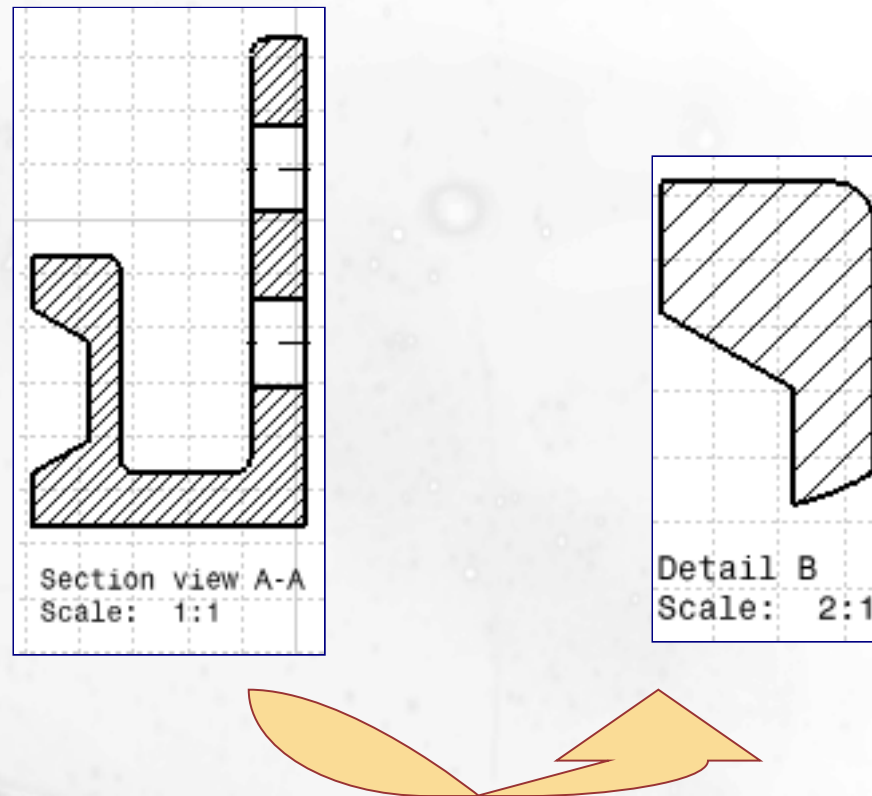
Modifying a Section Hatching Representation

You will learn how to Modifying a section Hatching Representation



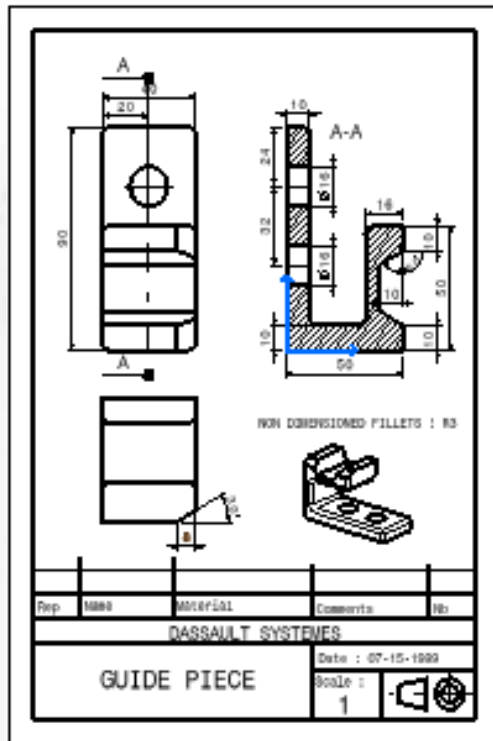
Why Change Hatching Pattern?

Hatching Patterns are changed to modify the default material that was assigned to the 3D part or to accommodate the size of the part or the size of the view.

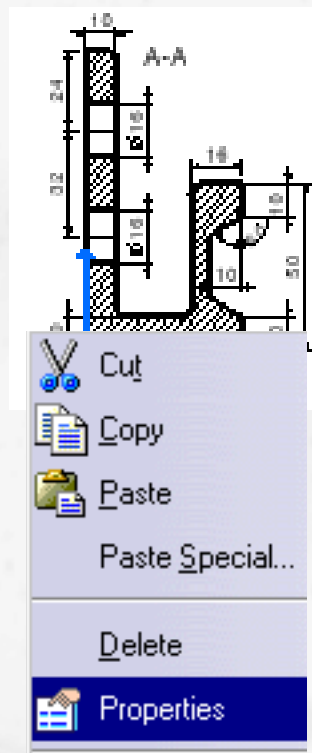


Changing Hatching Pattern (1/2)

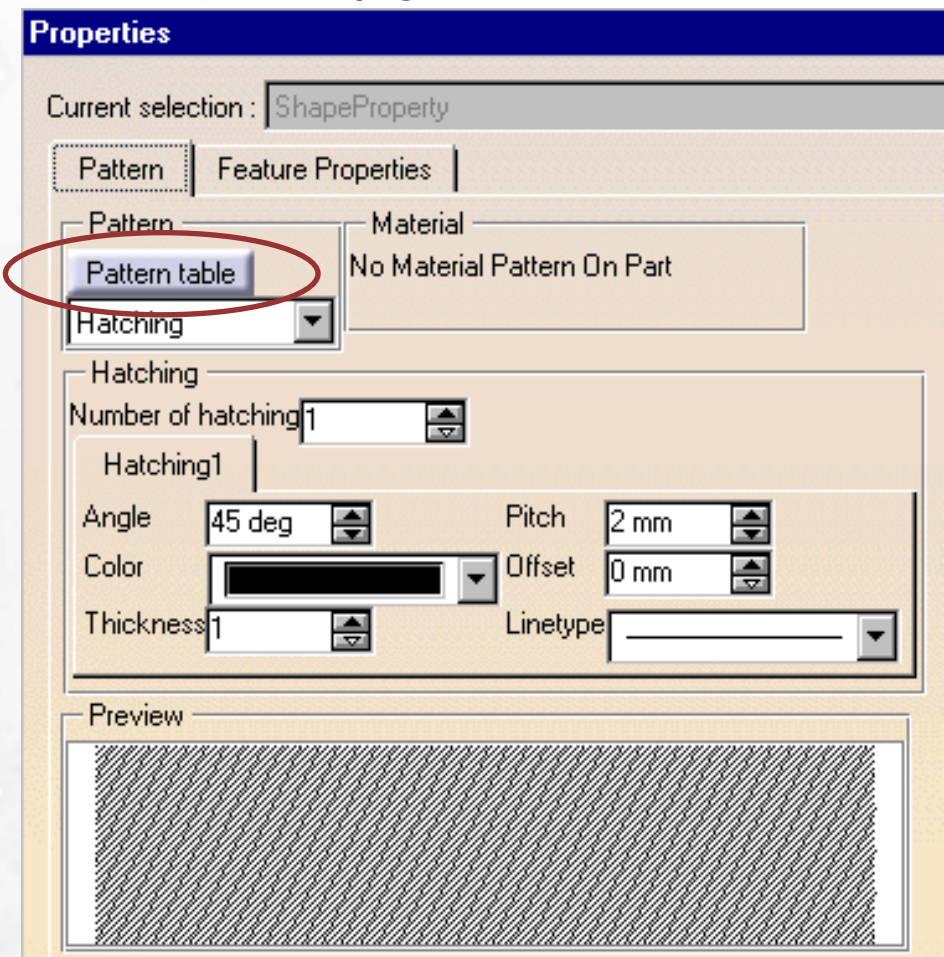
1 To change the steel cross hatching to an aluminum pattern.



2 Select the hatching pattern to modify. Select the properties using the right mouse button.



3 Activate the Pattern table (see next page)

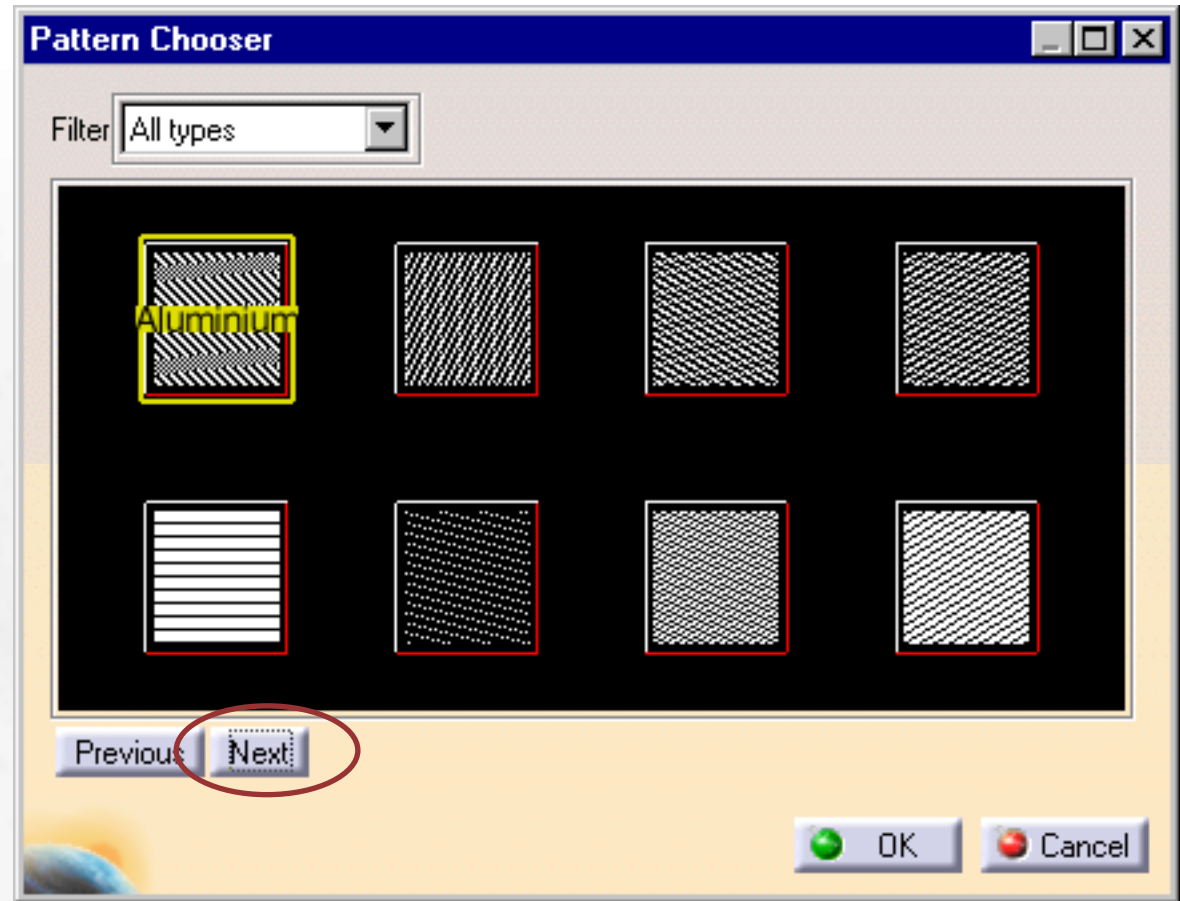
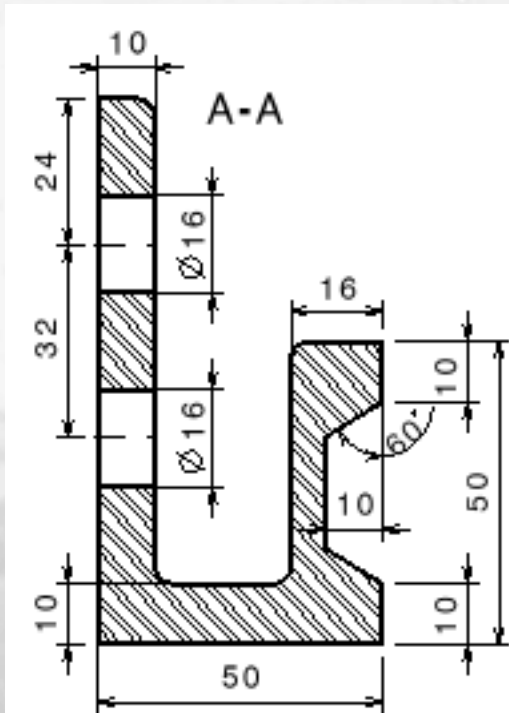


Changing Hatching Pattern (2/2)

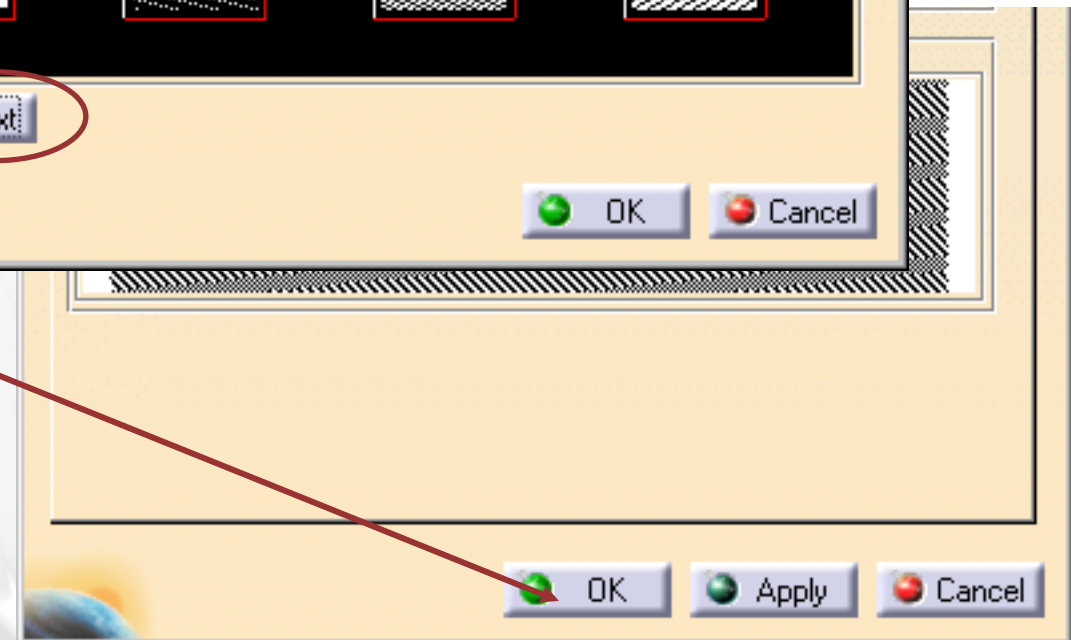
4 Using the Next button, display the Aluminum pattern. Select it and select OK.

5 Select OK in the Properties window.

The pattern is changed on the section.



2



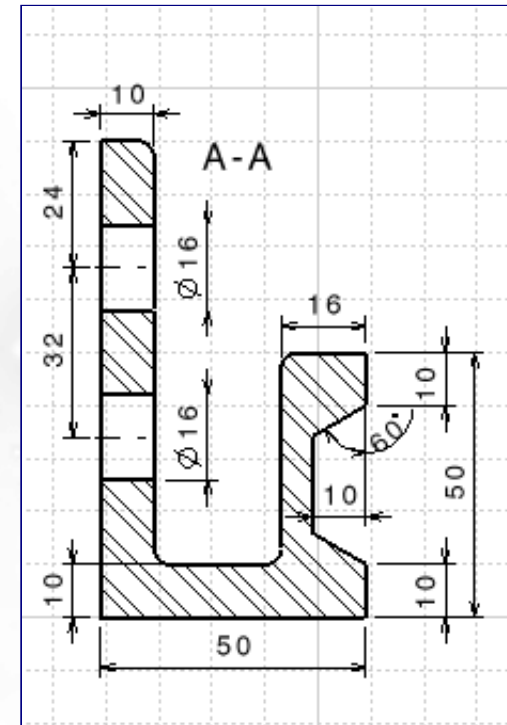
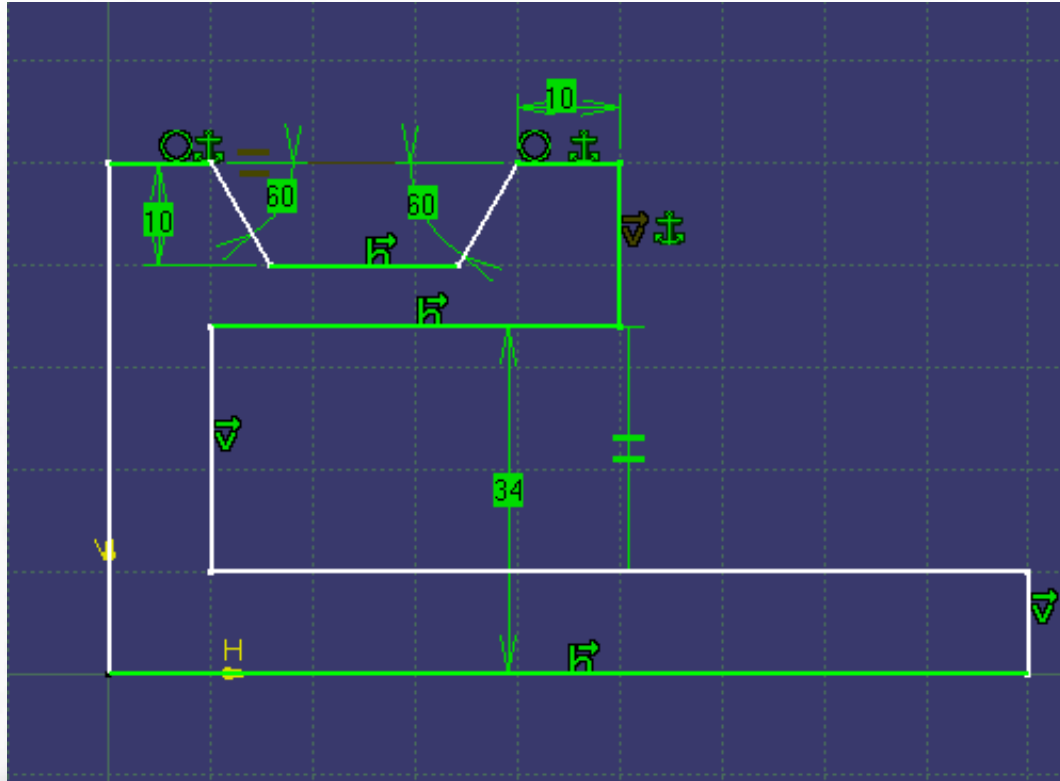
To Sum Up...

In this lesson you have seen...

- How to Edit properties for a view, sheet and drawing
- How to add sheets to a drawing
- How to manage views
- How to duplicate generative geometry
- How to modify Section, Detail and Auxiliary view profiles
- How to modify Section, Detail and Auxiliary view's graphical definition
- How to modify section hatching

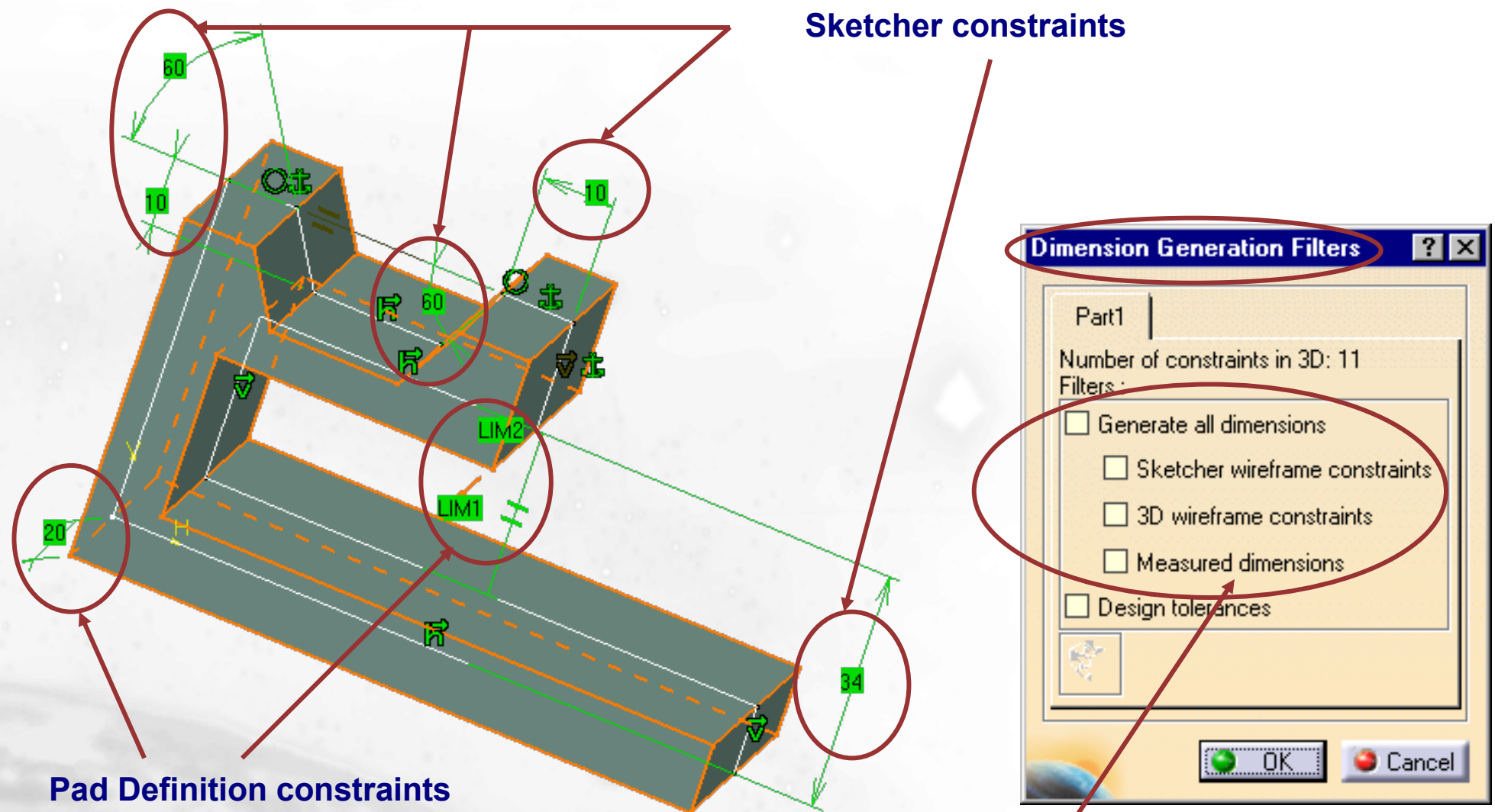
Automatic Dimension Generation

You will create the automatic dimensions and balloons for a generative drawing



What are Generated Dimensions ?

Generated dimension are dimensions that are created from existing 3D Part constraints



The dimensions that are generated from the 3D part are determined from the filter settings in the drafting options.

Generated Dimensions ...

Dimensions can be generated with two different methods

Ways to Generate Dimensions

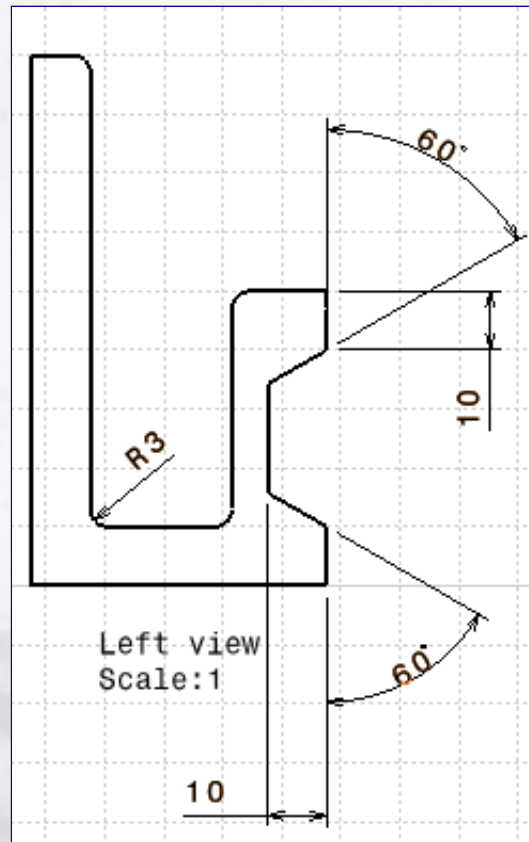
1- In One Step

2- Step by step

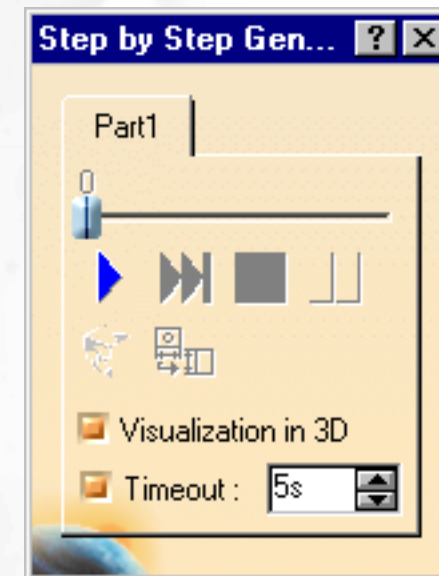
For each one you need to set up options



1



2



One Step Dimensioning Options

Dimension Generation options are set on the Generation Tab within the drafting options.

The image shows two dialog boxes from a CAD software. The main dialog is 'Options' with the 'Generation' tab selected. It contains several checkboxes: 'Generate dimensions when updating the sheet' (unchecked), 'Filters before generation' (checked), 'Automatic positioning after generation' (unchecked), 'Allow automatic transfert between views' (checked), and 'Analysis after generation' (checked). A spinner box shows '2 s' for the delay between generations. A second dialog, 'Dimension Generation Filters', is open, showing a list of filters: 'Generate all dimensions' (checked), 'Sketcher wireframe constraints' (unchecked), '3D wireframe constraints' (unchecked), 'Measured dimensions' (unchecked), and 'Design tolerances' (unchecked). Red circles and arrows point from text boxes to these specific options.

Options

Infrastructure
Mechanical Design
Part Design
Assembly Design
Sketcher
Mold Design
Structure Design

Dimension | Display | General | **Generation**

Dimension Generation

- ☐ Generate dimensions when updating the sheet
- ☒ Filters before generation
- ☐ Automatic positioning after generation
- ☒ Allow automatic transfert between views
- ☒ Analysis after generation

Delay between generations for step-by-step mode: 2 s

Dimension Generation Filters

Part1

Number of constraints in 3D: 11

Filters :

- ☒ Generate all dimensions
- ☐ Sketcher wireframe constraints
- ☐ 3D wireframe constraints
- ☐ Measured dimensions
- ☐ Design tolerances

OK Cancel

The allow automatic transfer between views only applies when using the step by step method.

The analysis after generation provides valuable information on the constraints found and the constraints generated.

Dimension Generation filter option allows you to choose what type of 3D constraints will be included in the generation of dimensions. It is recommended to have this setting on to assure desired dimensions are generated.

The Filter before generation option allows the selection of desired views for the dimensions to be generated into.

Step by Step Dimensioning Options

The step by step options allow you to control the generation and modify the dimensions one by one as they are visualized.



Up to End option stops the step by step method and generates the remaining dimensions in one step.

Next dimension generation

Trash

Visualization of dimensions to be generated in 3D

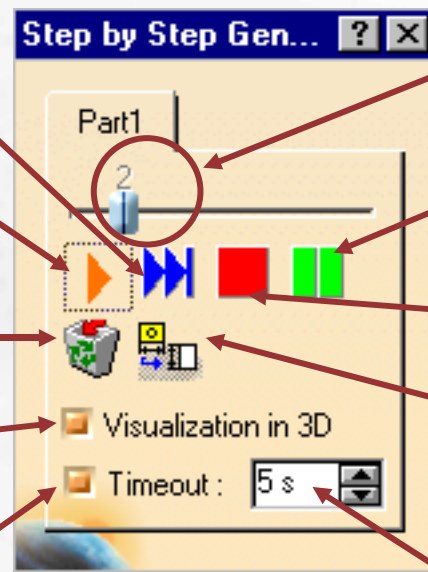
Number of dimensions generated

Pause

Abort the dimension generation process

Transfer dimension to another view

Seconds between dimension generation



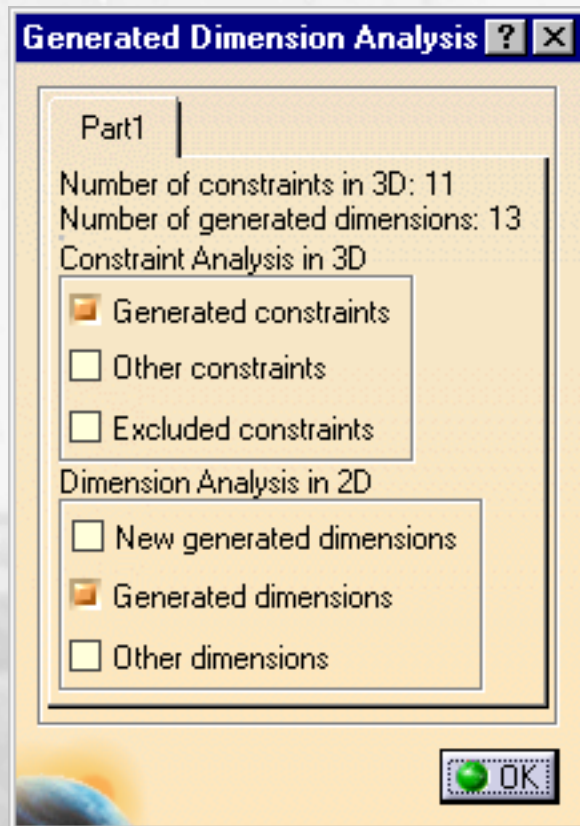
If the TIMEOUT option is not selected the Next arrow must be selected for each dimension to be generated.

Dimensioning Generation in One Step

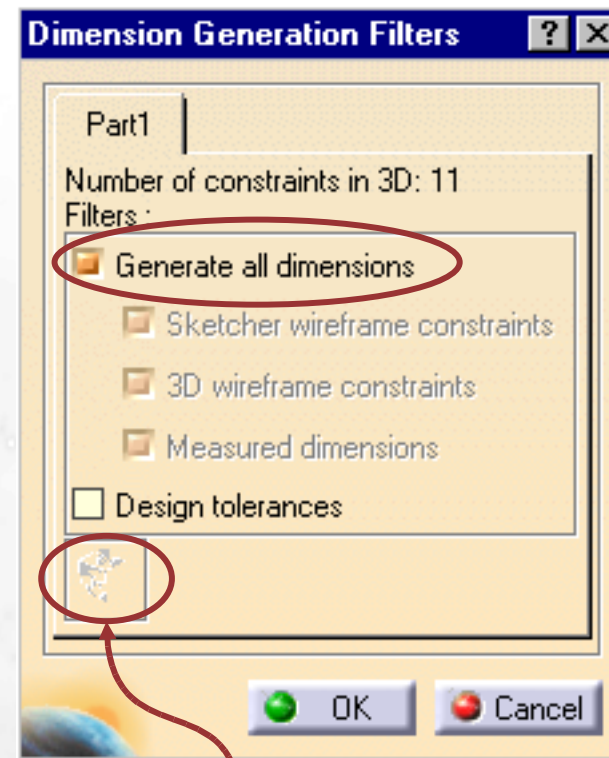
- 1 Select the one step dimension generation icon



- 3 Select options from the analysis panel

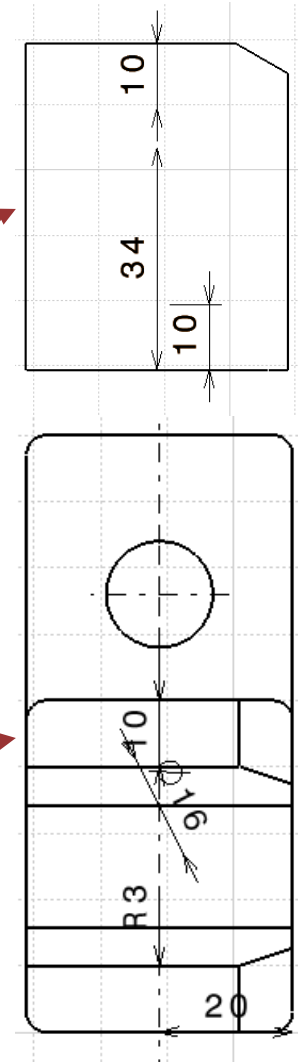


- 2 Select the views that are to receive the dimensions and set the filter option to generate all dimensions and OK



Constraints can be recovered by selecting the Excluded constraints icon and the constraint to be recover from 3D geometry.

4



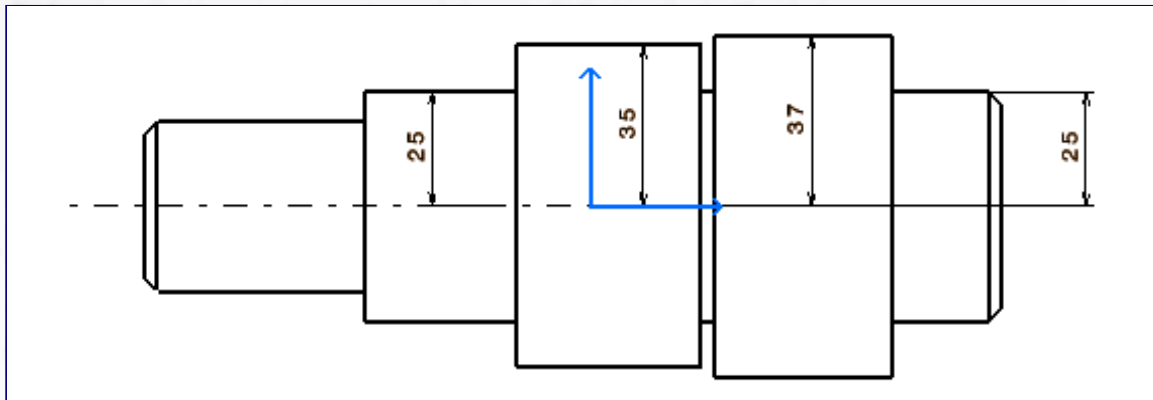
Dimensions are generated in the selected views. If none of the views are selected the dimensions will generate in the most appropriate view such that they will not have to be dimensioned anywhere else.

Automatic Positioning of Generated Dimensions

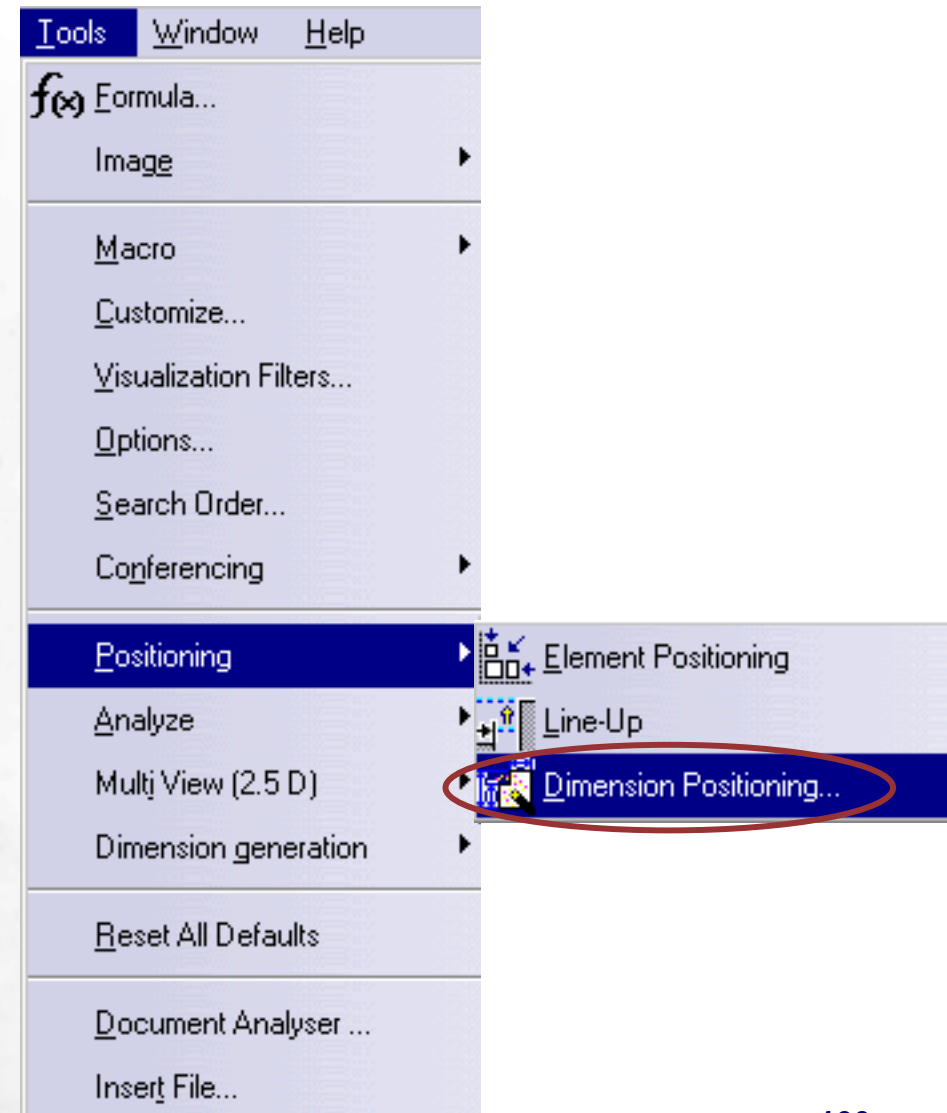


Often when Dimensions are generated in one step they require repositioning. The automatic positioning option does this operation in one step.

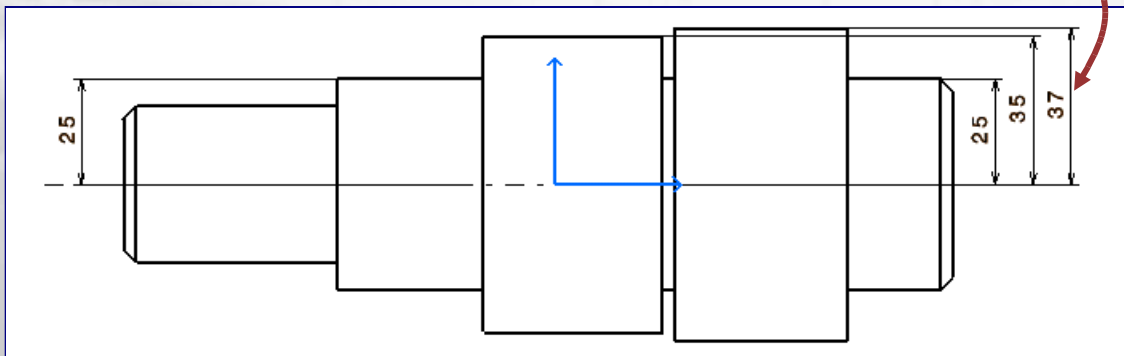
- 1 Select the view or views that require the dimensions be repositioned



- 2 Select the Dimension Positioning Icon or Tools + Positioning



- 3 Dimensions are rearrange automatically

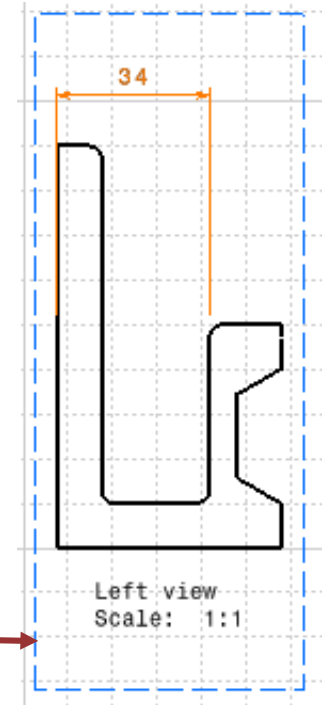
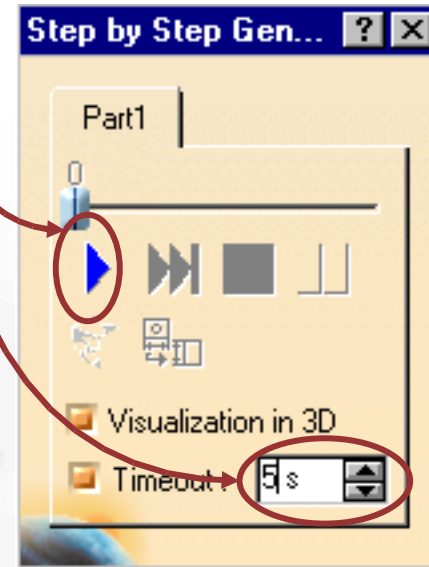


Dimensioning Generation Step by Step

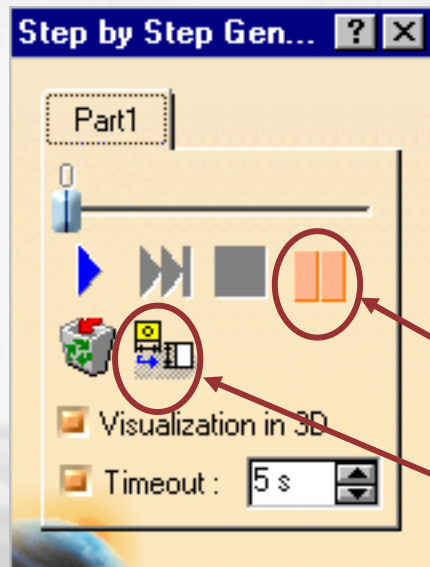
- 1 Select the step by step dimension generation icon and set the filter option to generate all Dimension.



- 2 Select the NEXT icon to start the generation process with a 5 second delay between each dimension generated.



- 3 Select the pause and the transfer option to relocate a dimension to another view

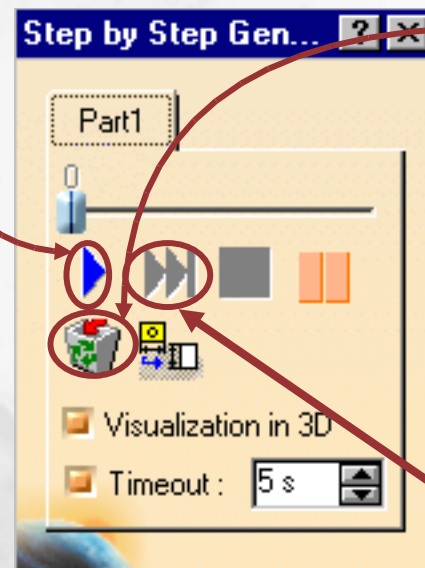


Pause

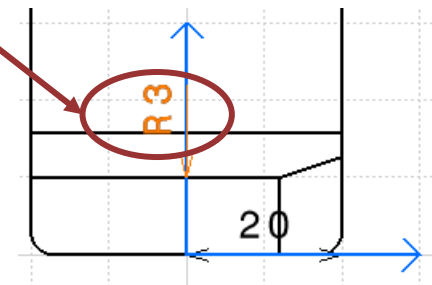
Transfer

- 4 Select the frame of the view that the dimension is to be transferred to

- 5 Continue the generation process by selecting NEXT icon again.



- 6 Select the trash icon to discard any unwanted dimensions.



- 7 Finish the dimension generation by selecting the "up to end" Icon and the desired analysis options.



The Pause can also be used to relocate dimensions during the generation process.

Dimension Interference Analysis...

Dimension Interference can be analyzed if the dimensions have been Generated from 3D or Manually created.

The analyze panel provides the following:

(A) Interfering Dimension list:

The interfering dimensions can be viewed as a whole list or a filtered to display an optimized list

(B) Total number of elements in the current list:

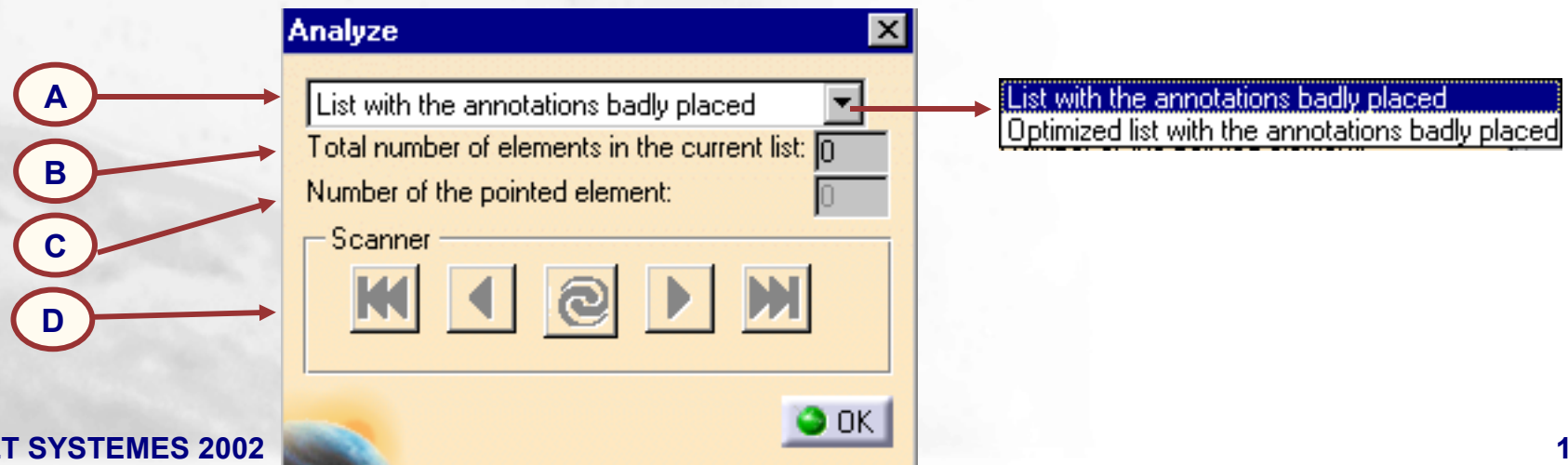
As Dimensions are relocated the update switch will correct the total number of dimensions that are poorly placed

(C) Number of the pointed element:

Each dimension that has an interference is assigned a number and this field displays the number of the elements currently pointed at as an interfering element

(D) Scanner (Start, Before, Update, Next, End)

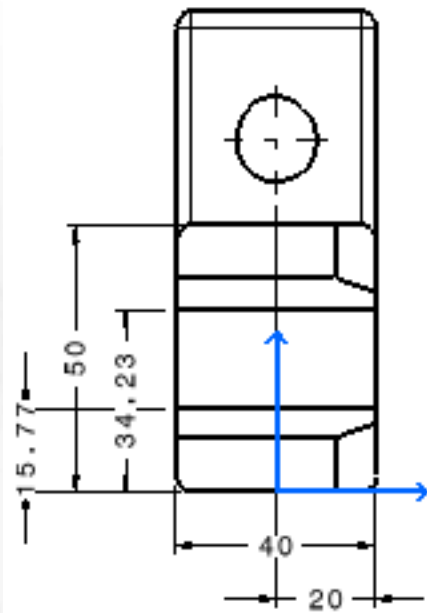
To navigate among the list of interfering elements



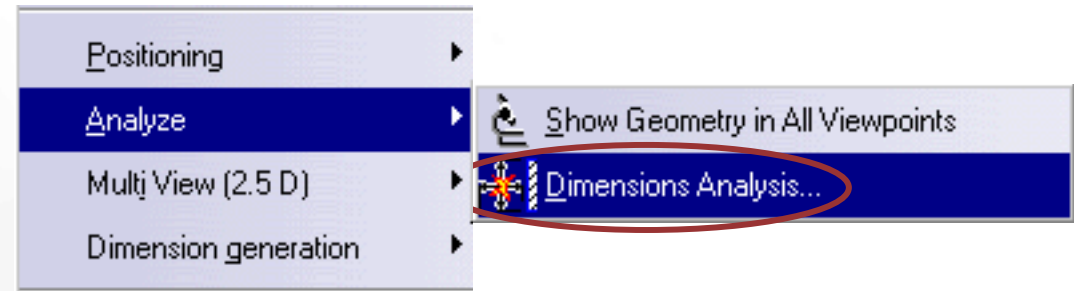
Analyzing the Interference of Dimensions (1/2)



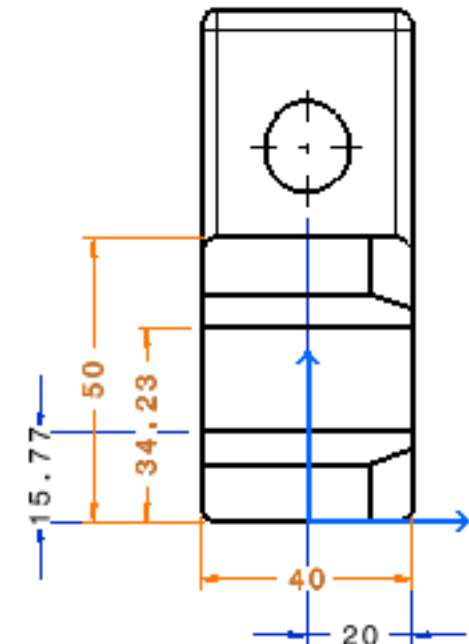
- 1 Make the view active that is to be analyzed



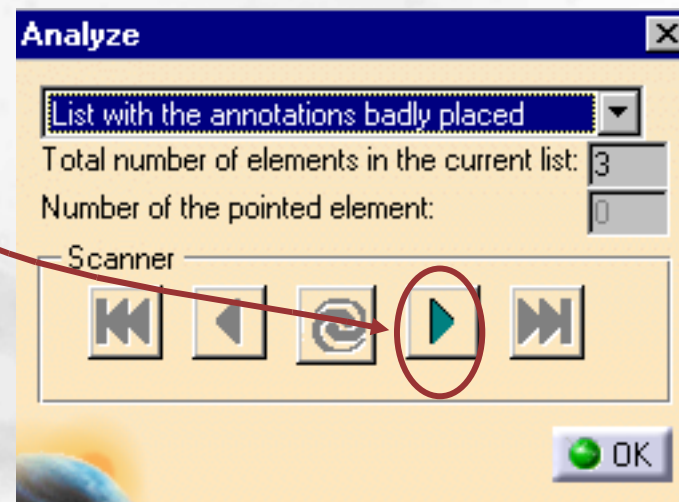
- 2 Select Dimension Analysis Icon or the Tools + Analyze



- 3 The interfering dimensions are automatically displayed in the red orange color.

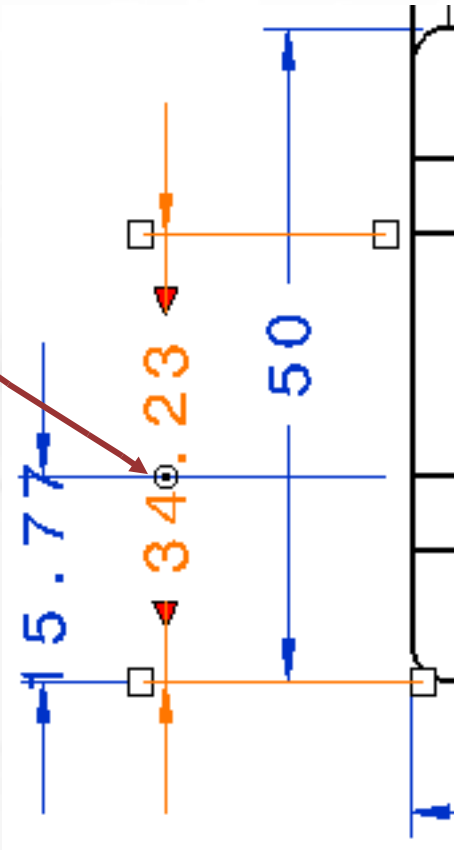


- 4 Select the Next arrow to begin scanning the Interfering dimensions

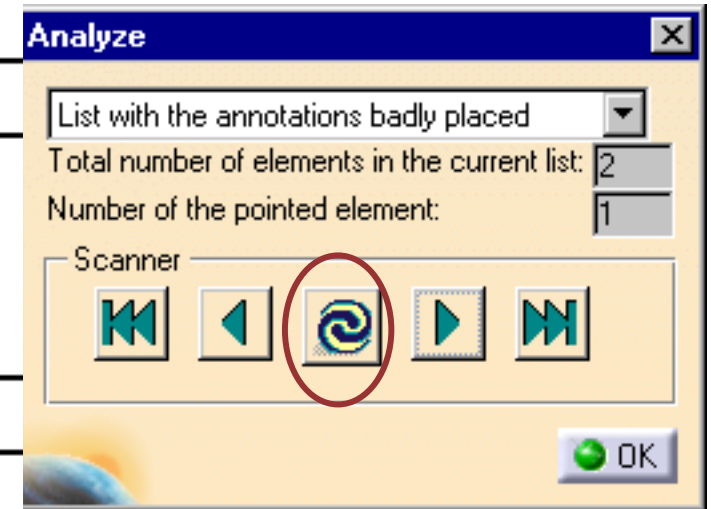
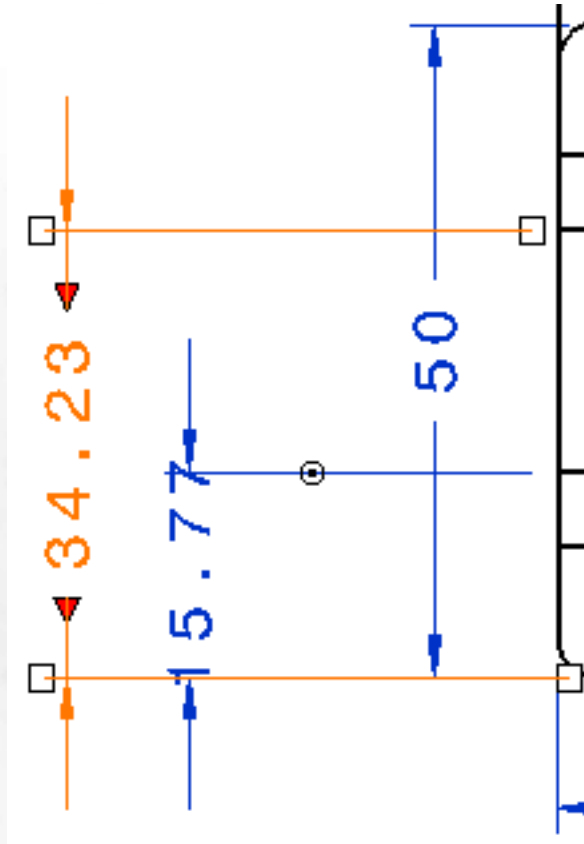


Analyzing the Interference of Dimensions (2/2)

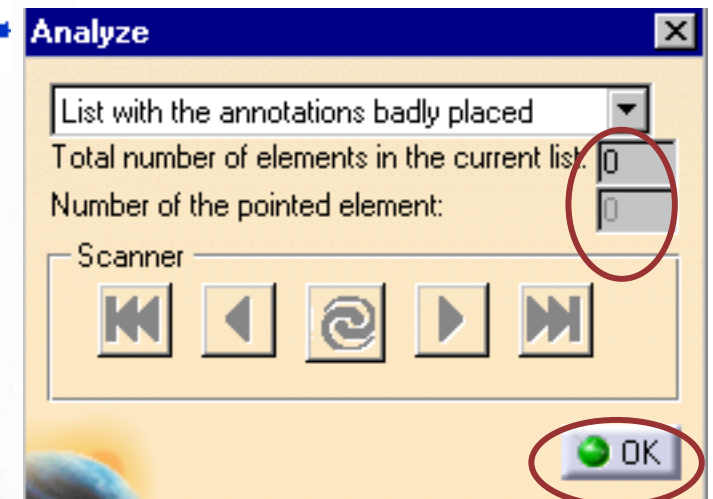
- 5 Small circles allow visualizing the interference location



- 6 Move the Dimension such that it no longer interferes with any element and select the update icon



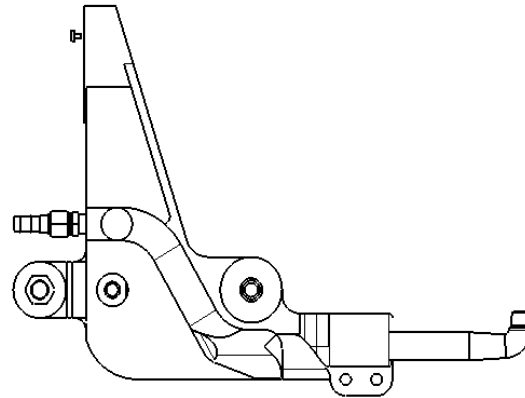
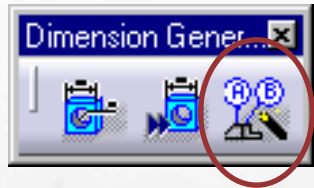
- 7 Continue the process until the dimension interference is reduced to zero and OK to close the panel



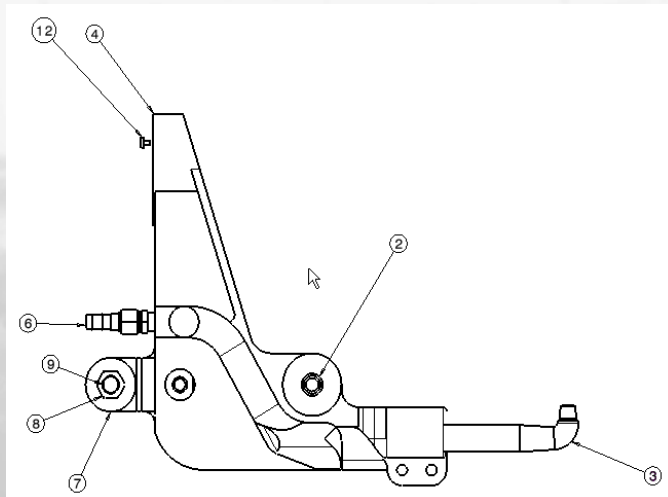
Balloons

Creates balloon annotations from the CATProduct information on an assembly

- 1 Activate the desired view and Select the Balloon Icon

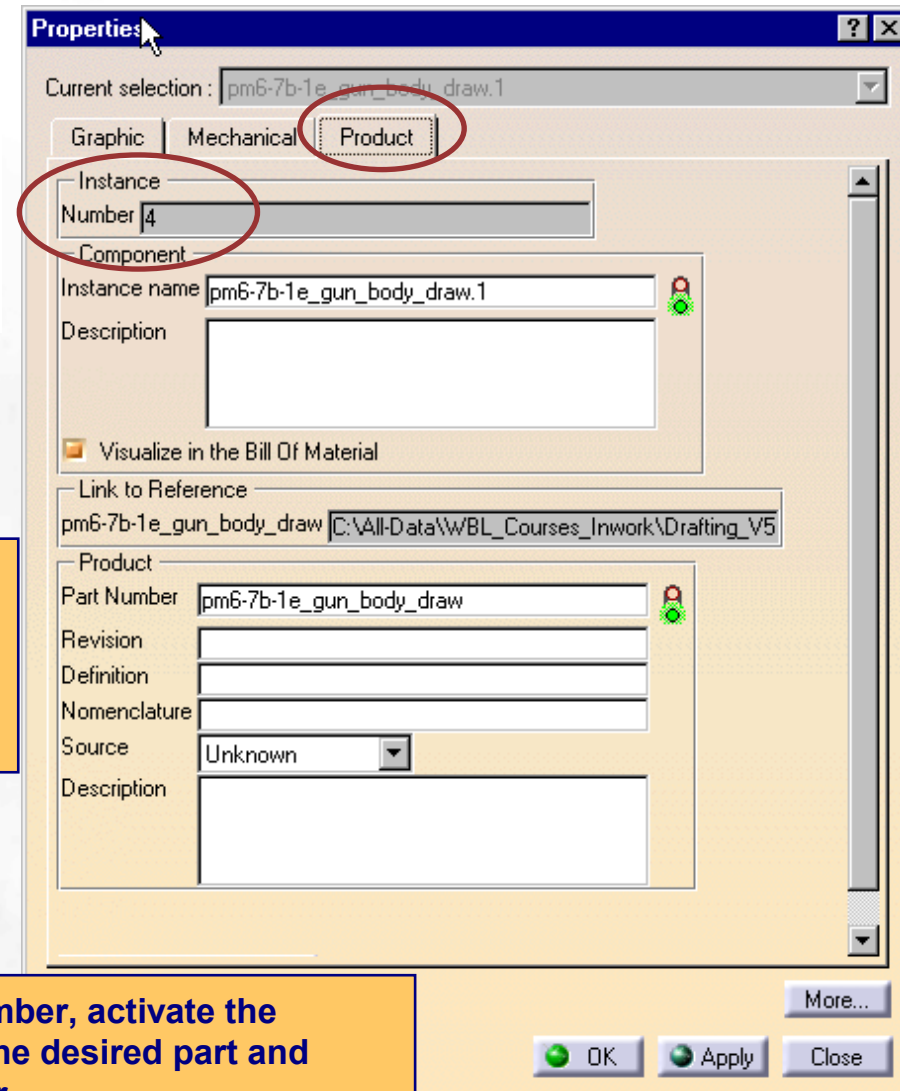


Example results:



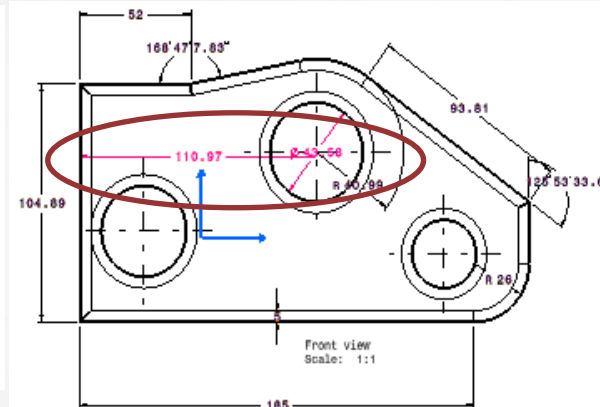
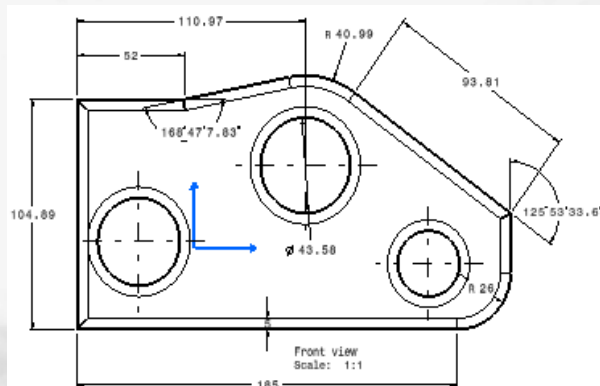
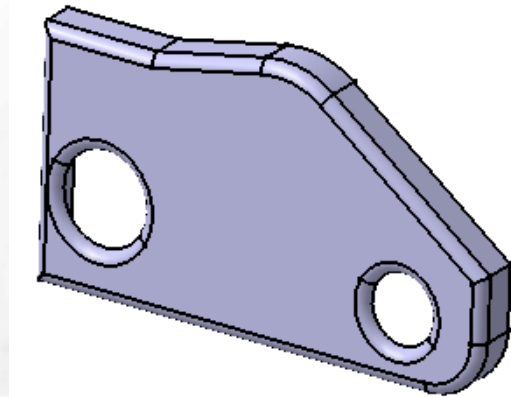
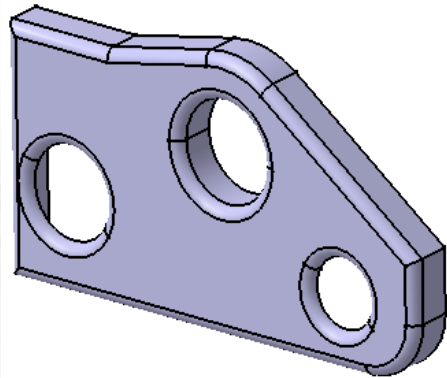
Arrange the balloons as needed after creation. You can also change the Font properties of the balloons as needed.


To change the instance number, activate the CATProduct properties of the desired part and modify the instance number.

A screenshot of the 'Properties' dialog box in a CAD application. The 'Product' tab is selected and circled in red. The 'Instance Number' field is also circled in red and contains the value '4'. Other fields include 'Instance name' (pm6-7b-1e_gun_body_draw.1), 'Description', 'Link to Reference' (C:\All-Data\WBL_Courses_Inwork\Drafting_V5), 'Part Number' (pm6-7b-1e_gun_body_draw), 'Revision', 'Definition', 'Nomenclature', 'Source' (Unknown), and 'Description'. At the bottom are 'OK', 'Apply', and 'Close' buttons, and a 'More...' button on the right.

Dimension Associativity

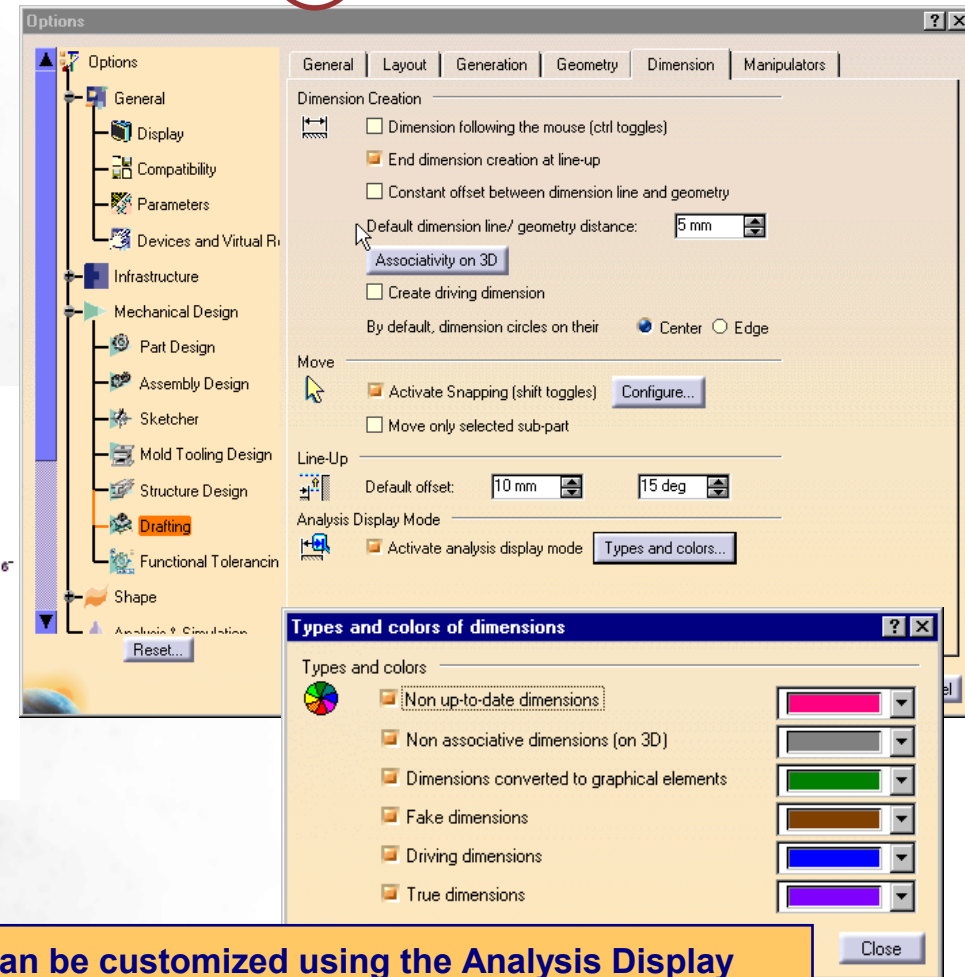
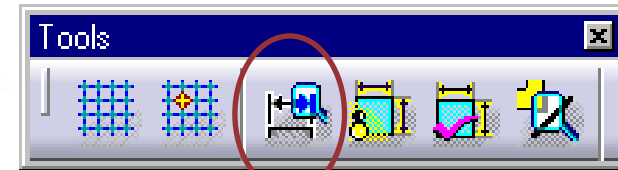
If one parent element of the dimension is deleted or deactivated, as soon as you update the drawing (either 3D Generative or 2D Interactive drawing), the orphan dimension becomes **purple** on the condition you activated the Analysis Display Mode



Update () the drawing in accordance with the 3D, any non associative dimension will disappear.



Colors can be customized using the Analysis Display Mode option from the Tools toolbar or via Tools+ Options (Drafting option to the left of the dialog box, Dimension tab).





To Sum Up...

In this lesson you have seen...

- How to create the automatic dimensions for a generative drawing
- How to create the automatic dimensions for a generative drawing using step by step
- How to create the automatic balloon annotations for a generative drawing

Finalizing the Drawing and Printing

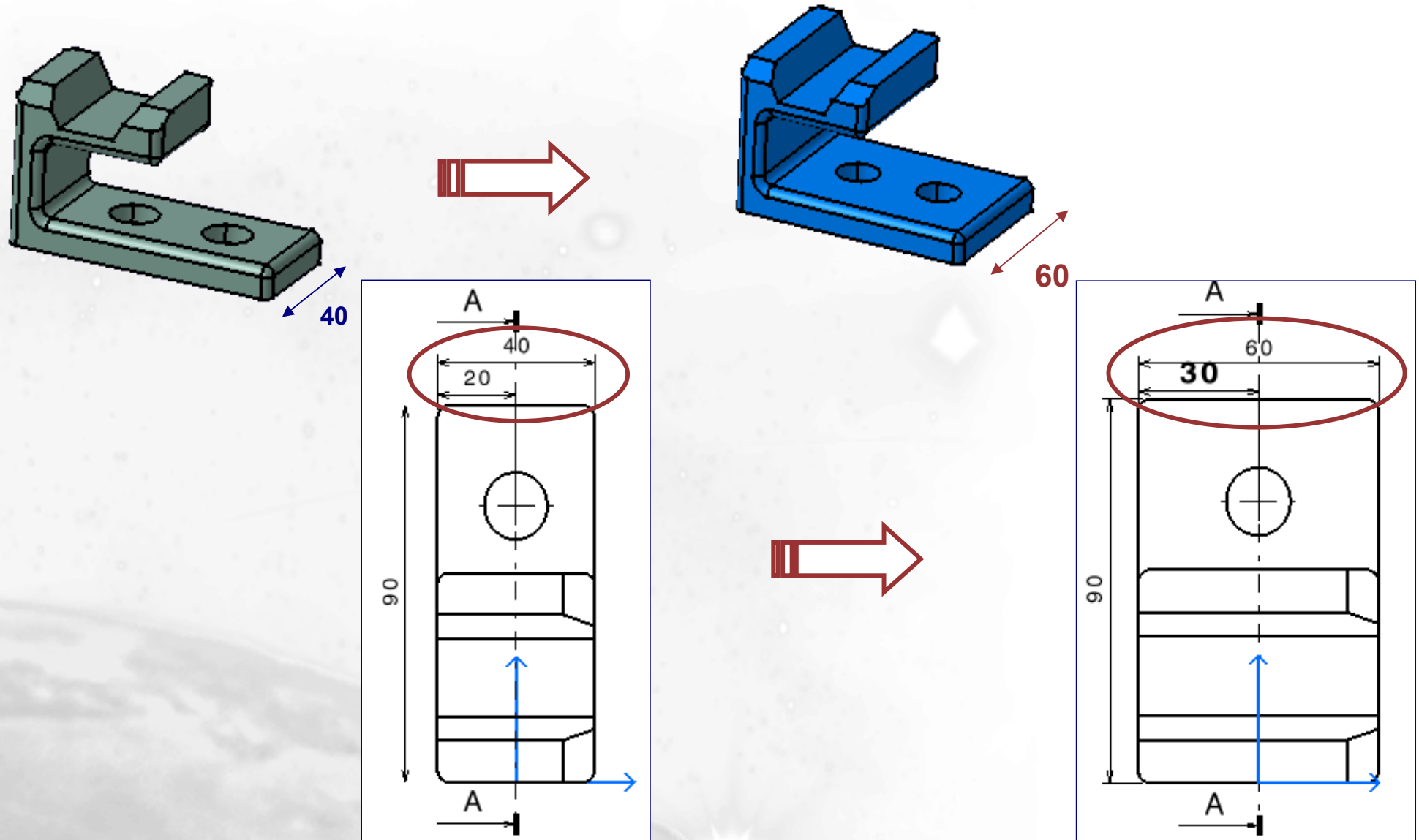
In this lesson you will see how to update a drawing upon notification that the part geometry has changed in 3D. You will also see how to add a Title Block and print your drawing.

| | | | | |
|---|------|-------------------|---|----|
| | | | | |
| Rep | Name | Material | Comments | Nb |
| DASSAULT SYSTEMES | | | | |
|  Connector Housing | | Date : 06-07-1999 | | |
| | | Scale : 1.5 |  | |

- ▣ Checking Links to Solid 3D Part and Updating a Drawing
- ▣ Adding a Title Block
- ▣ Adding a BOM (Bill of Material)
- ▣ Printing the drawing

Checking for Changes

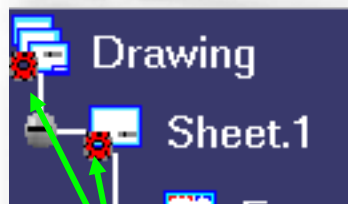
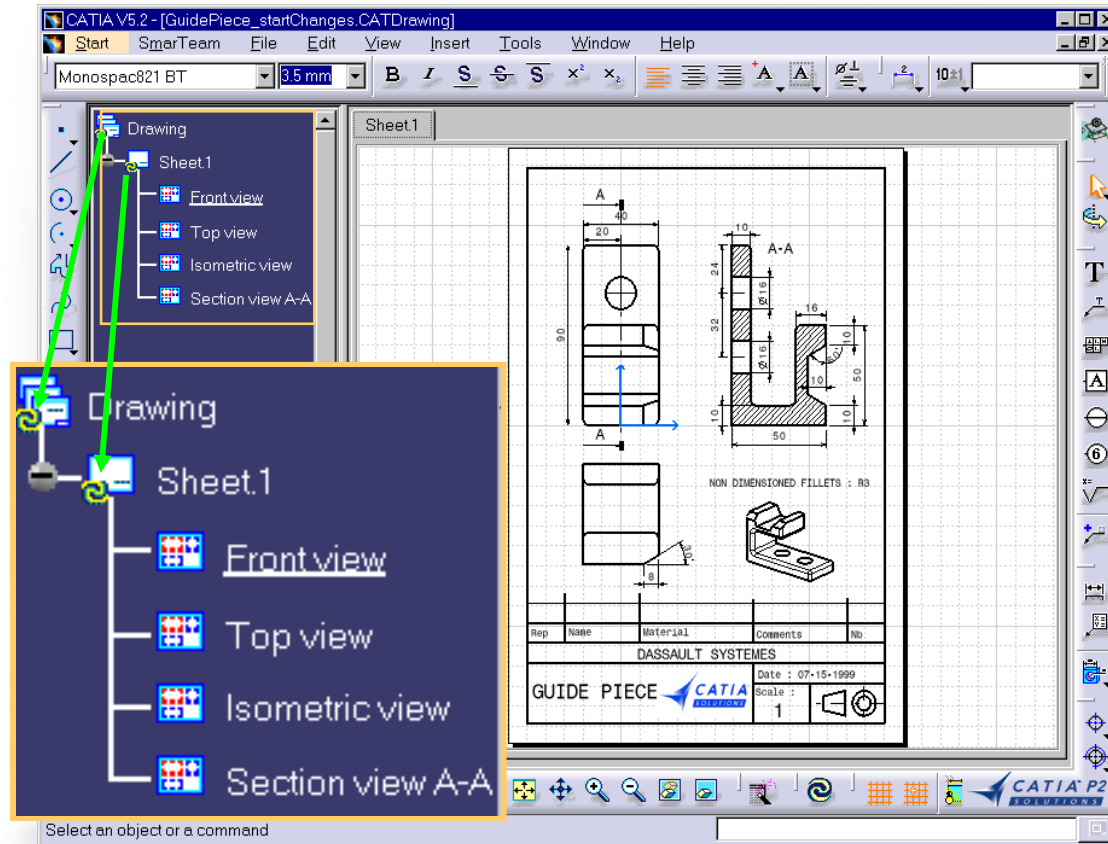
You will learn how to check the drawing for 3D changes in the Part



Matching Drawing with Modified 3D Part

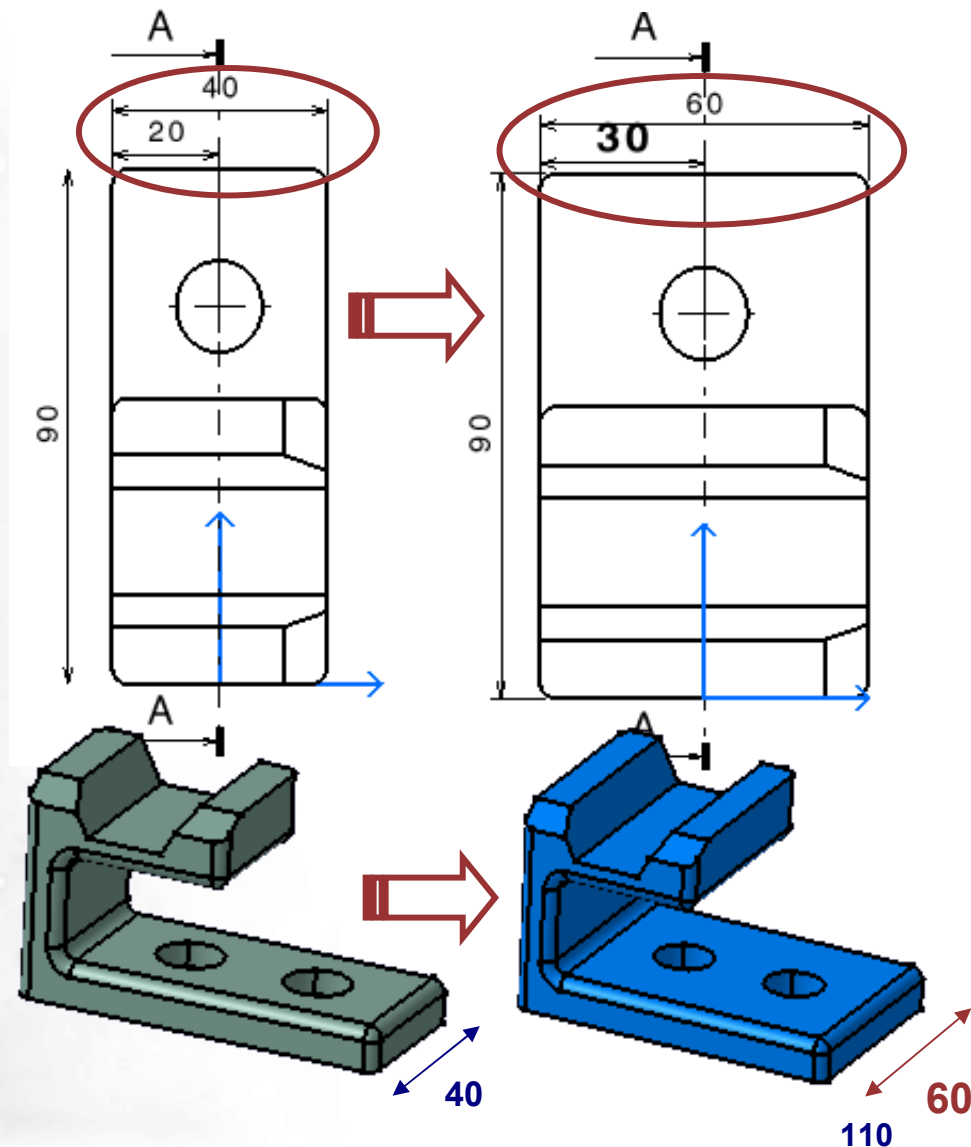


Watch out for the Update icon: if it is highlighted, it means that the drawing needs to be updated to reflect the changes that were made on the 3D part it represents



1

Select the update icon to re-generate the view and modify any dimensions



The red circle in the tree indicates that the 3D part is not loaded
If the part is not in the Part Design workbench, you can use
Edit + Links to check if both representations match.

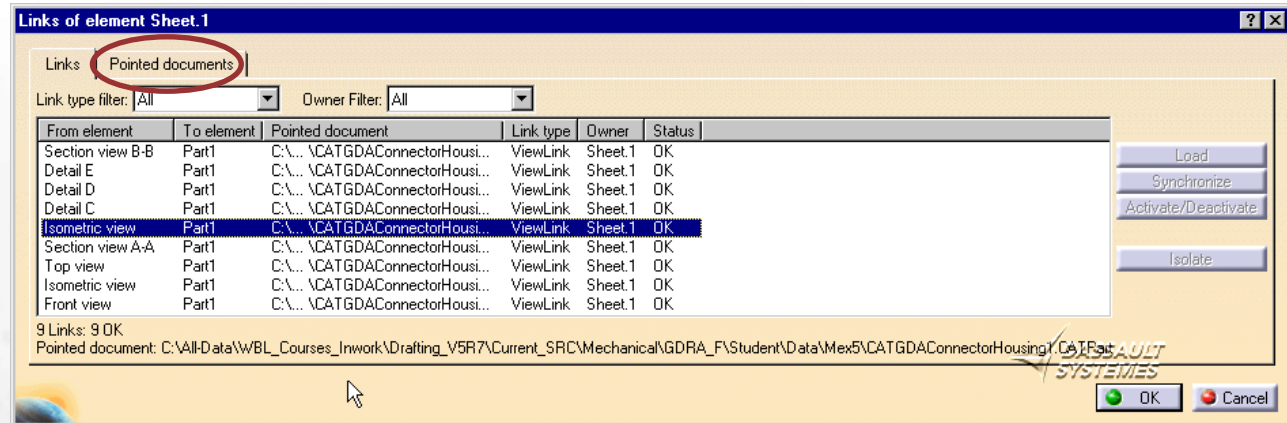
Checking Links to 3D Parts

To check if the drawing is up to date, you may have to load the represented part(s).

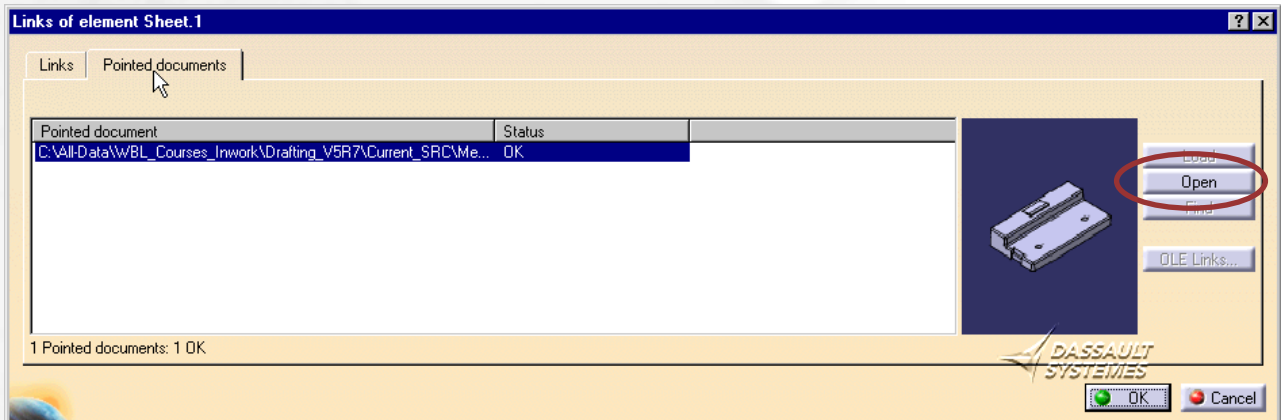


1 Select the **Edit + Links** menu

2 Select the desired view.



3 Select the “Pointed Documents” tab when you do not know the name of the 3D part.

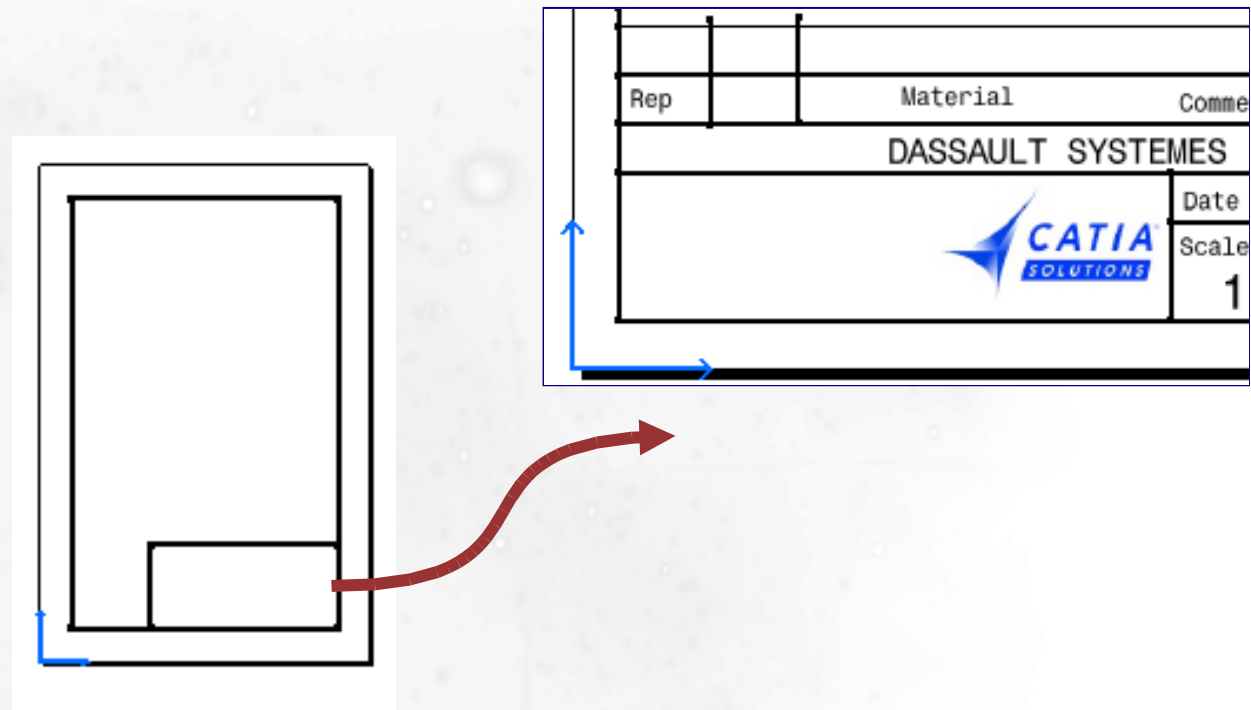


4 Select the “Open” button to load the part automatically

If the modified 3D part has another name (new version number for example), you can use the “Find...” button to have your drawing pointing to it. You will still have to update it afterwards.

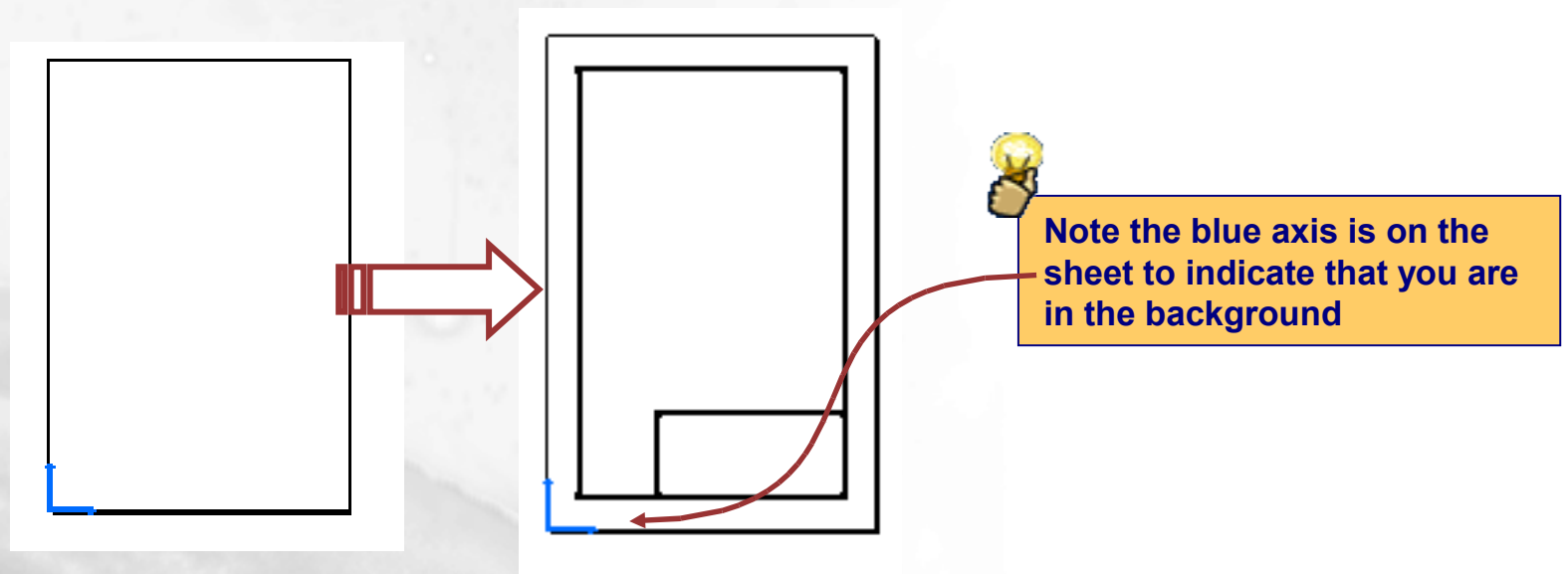
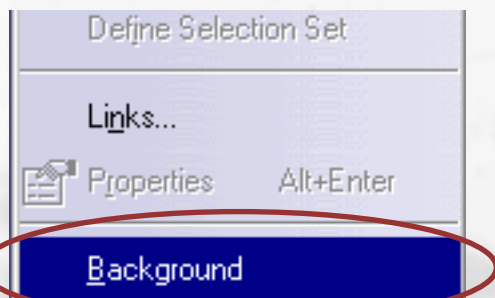
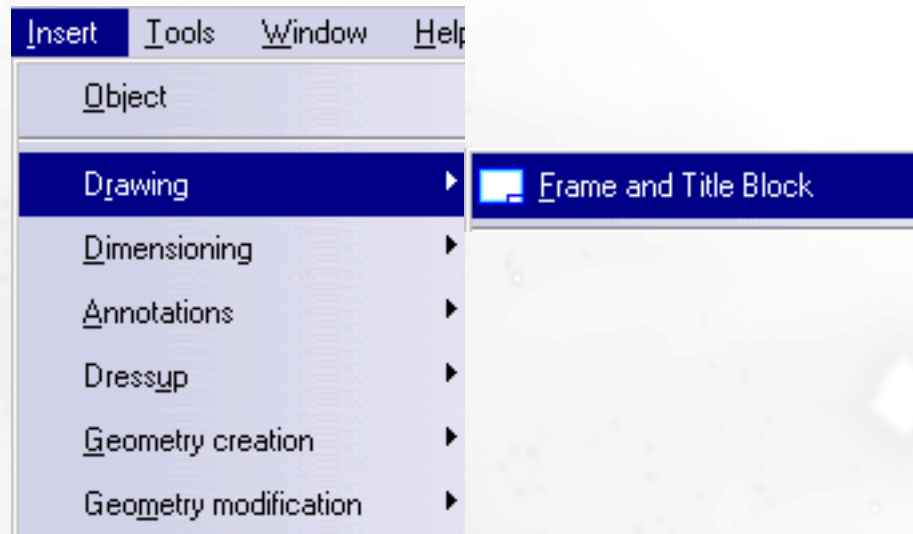
Adding a Title Block

You will learn how to add a title block to the drawing



Adding a Title Block (1/2)

- 1 Change to the sheet "Background" with **Edit + Background** menu
- 2 Select **Insert + Drawing + Frame and Title Block** menu, to set the 2 main frames



Adding a Title Block (2/2)

- 4 (a) Use the geometric tools (lines, circles, etc.) to draw the title block
- (b) Use the Text icon and the Text Properties to fill in all necessary information
- (c) Use a VB script to complete the Title block geometry and text.

- 5 When finished, do not forget to go back to (select) **“Working View”** from the Edit menu

4a

Geometry Creation

4b

T

3c

Macro

Macro Name:

C:\Program Files\Dassault Systemes\B07\intel_a\VBScript

C:\Program Files\Dassault Systemes\B07\intel_a\VBScript

Run

Cancel

Edit

Create

Select

Delete

Macro in: External File

Description

DASSAULT SYSTEMES

Text Properties

Monospac821 BT

3.5 mm

B I S

Geometry Creation

Tools

| Rep | Material | Comments | Nb |
|-------------------|----------|-------------------|----|
| DASSAULT SYSTEMES | | | |
| CATIA SOLUTIONS | | Date : 07-15-1999 | |
| Scale : 1 | | | |

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6

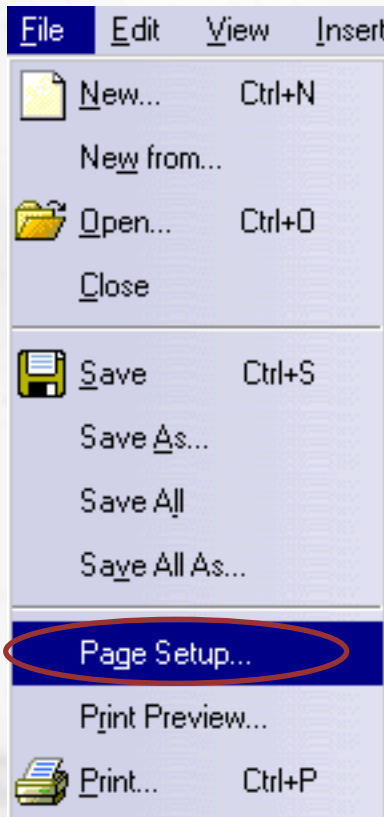
x=

✓

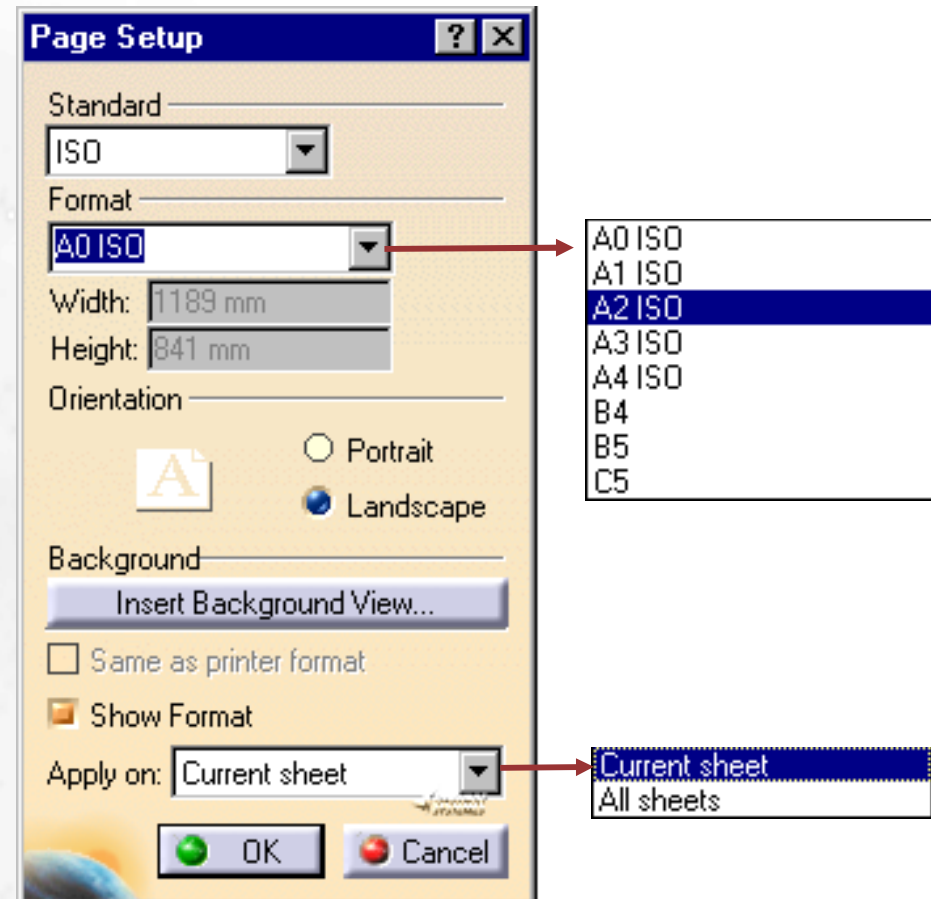
Changing Drawing size

Drawing size can be changed anytime in the drafting process.

1 Select File + Page Setup menu



2 Select the new size format, the desired orientation, the sheets effected and OK.



While changing drawing size an option to insert a Background view from another document is available.

Adding a BOM (Bill of Material)

You will learn how to add a BOM (Bill of Material) to the drawing

Bill of Material: Mecanique

| Quantity | Part Number | Type | Nomenclature | Revision |
|----------|-----------------|----------|--------------|----------|
| 1 | Roue dentee | part | | |
| 1 | Bague | part | | |
| 1 | Boitier | part | | |
| 1 | Coulisse | part | | |
| 1 | Coussinet-cyl | part | | |
| 1 | Ensemble Moteur | assembly | | |
| 1 | bagueD7 | part | | |

Bill of Material: Ensemble Moteur

| Quantity | Part Number | Type | Nomenclature | Revision |
|----------|---------------|------|--------------|----------|
| 1 | Moteur_carter | part | | |
| 1 | Axe | part | | |
| 1 | clavette | part | | |

Recapitulation of: Assemblage

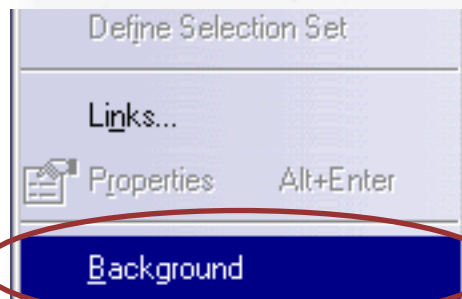
Different parts: 13

Total parts: 13

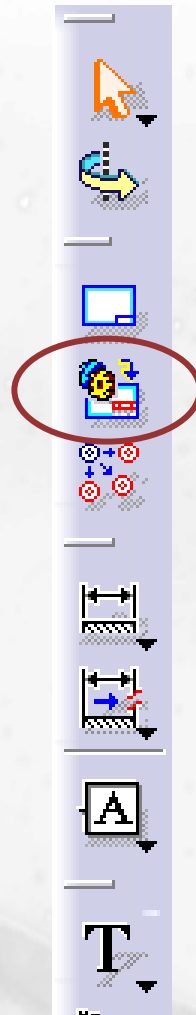
| Quantity | Part Number |
|----------|---------------|
| 1 | Semelle haute |
| 1 | semelle_basse |
| 1 | Roue dentee |
| 1 | Bague |
| 1 | Boitier |
| 1 | Coulisse |
| 1 | Coussinet-cyl |
| 1 | Moteur_carter |
| 1 | Axe |
| 1 | clavette |
| 1 | bagueD7 |
| 1 | Carter Gauche |
| 1 | lame |

Adding a BOM (Bill of Material) (1/2)

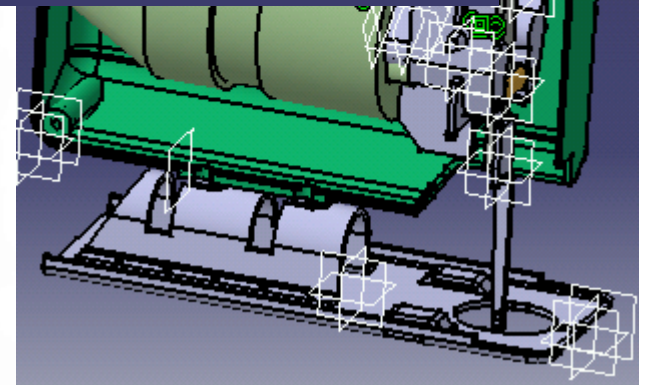
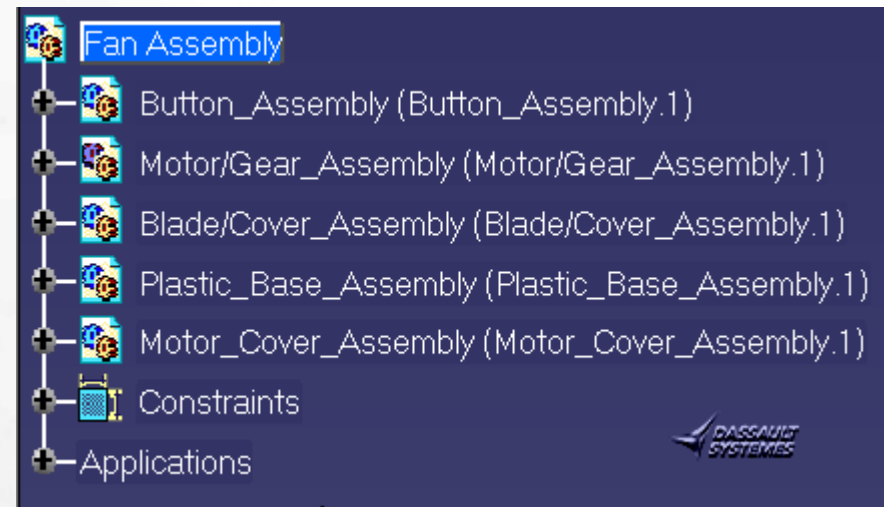
- 1 Change to the sheet "**Background**" with **Edit + Background** menu



- 2 Select **Insert Bill of Material** icon



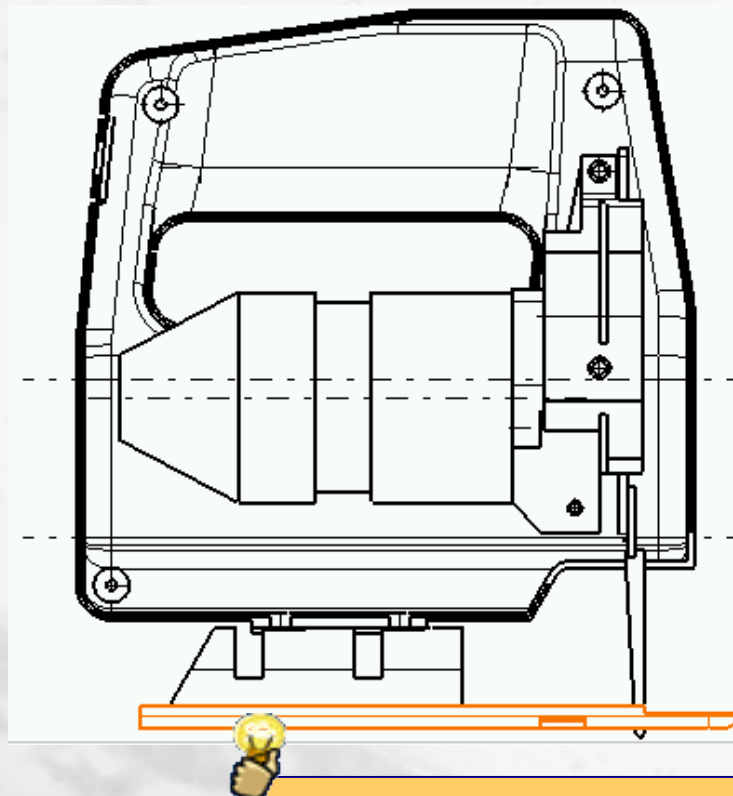
- 3 From the Assembly, Select the Product structure to which generate the BOM



Adding a BOM (Bill of Material) (2/2)

4

Select a location on the drawing to position the BOM



Bill of Material: Mecanique

| Quantity | Part Number | Type | Nomenclature | Revision |
|----------|-----------------|----------|--------------|----------|
| 1 | Roue dentee | part | | |
| 1 | Bague | part | | |
| 1 | Boitier | part | | |
| 1 | Coulissee | part | | |
| 1 | Coussinet-cyl | part | | |
| 1 | Ensemble Moteur | assembly | | |
| 1 | bagueD7 | part | | |

Bill of Material: Ensemble Moteur

| Quantity | Part Number | Type | Nomenclature | Revision |
|----------|---------------|------|--------------|----------|
| 1 | Moteur carter | part | | |
| 1 | Axe | part | | |
| 1 | clavette | part | | |

Recapitulation of: Assemblage

Different parts: 13

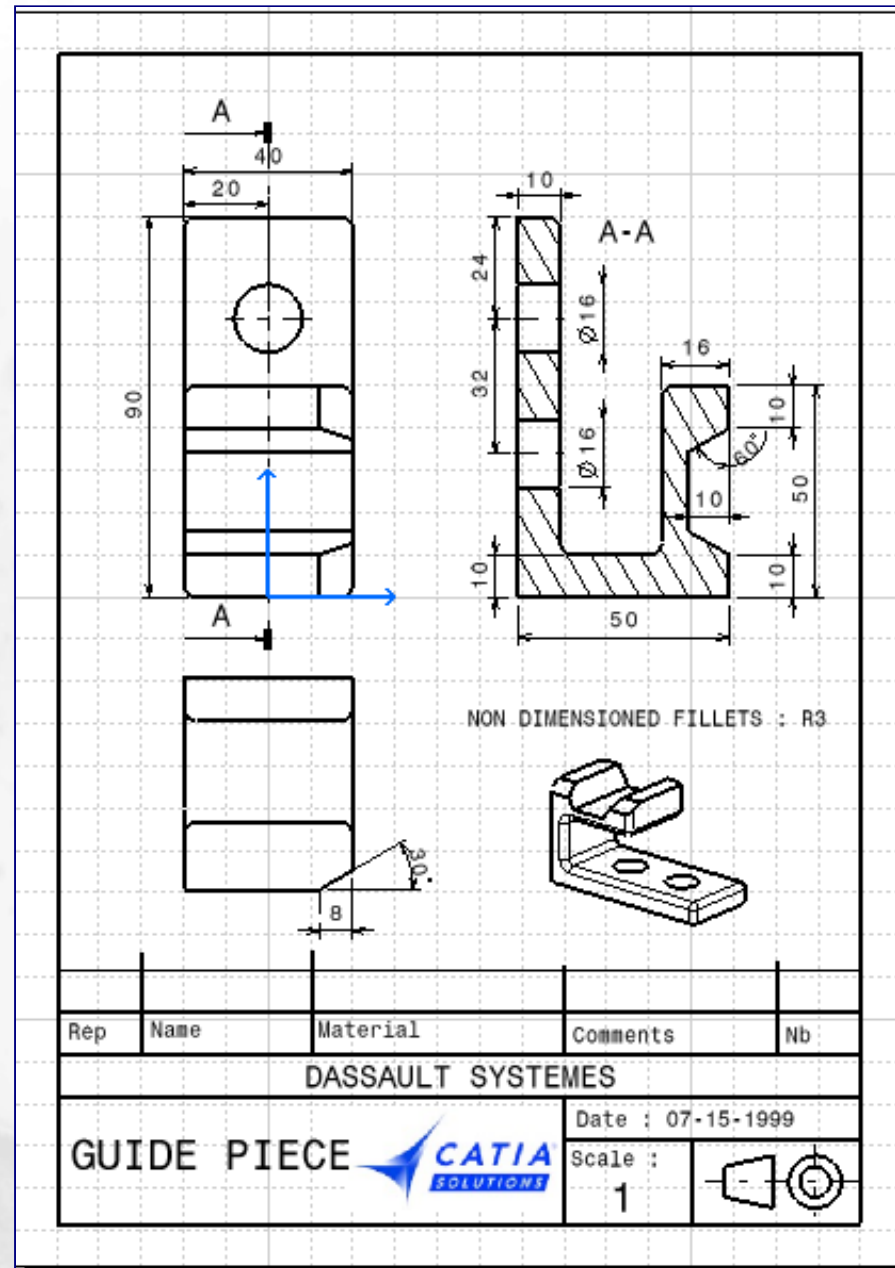
Total parts: 13

| Quantity | Part Number |
|----------|---------------|
| 1 | Semelle haute |
| 1 | semelle_basse |
| 1 | Roue dentee |
| 1 | Bague |
| 1 | Boitier |
| 1 | Coulissee |
| 1 | Coussinet-cyl |
| 1 | Moteur carter |
| 1 | Axe |
| 1 | clavette |
| 1 | bagueD7 |
| 1 | Carter Gauche |
| 1 | lame |

Note: a BOM can be updated to reflect when a part is added or deleted to the product structure

Printing the Drawing

You will learn how to Print the drawing

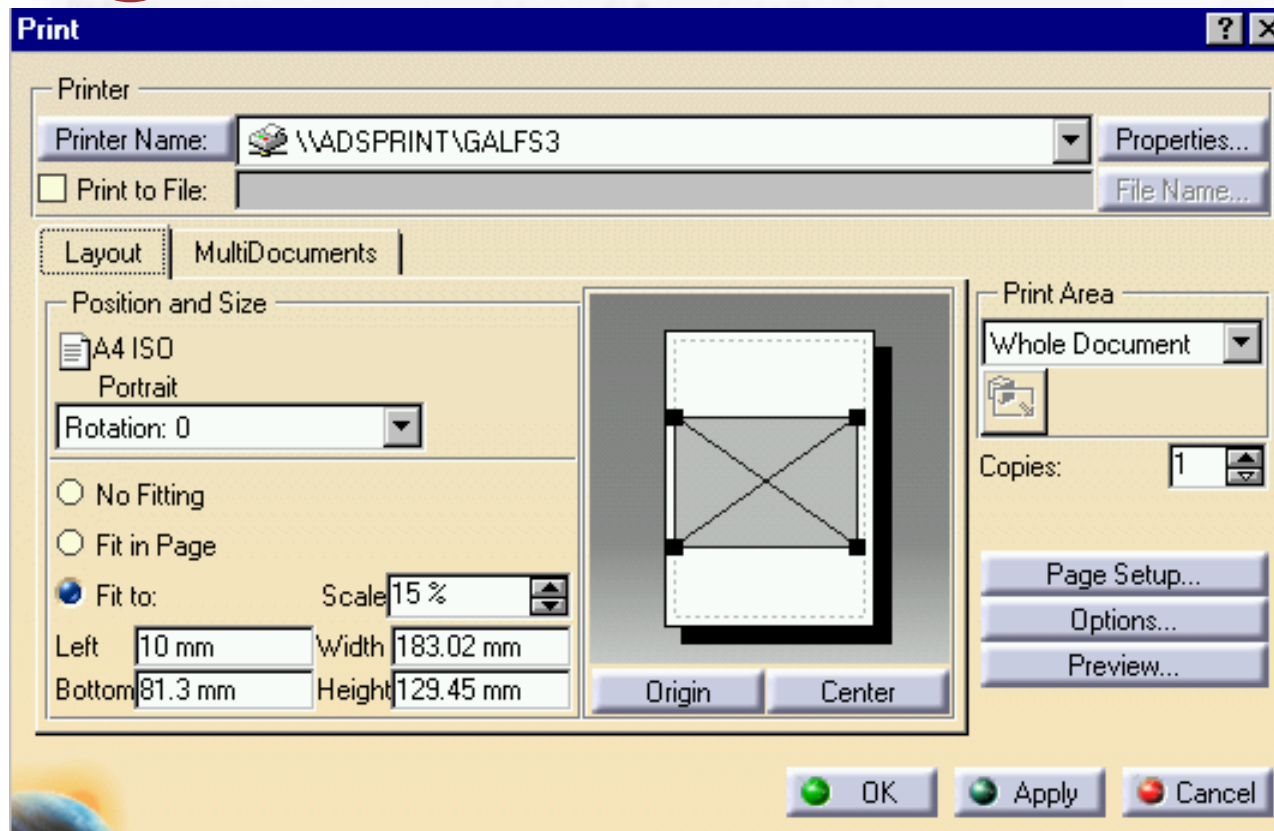


Printing a Drawing

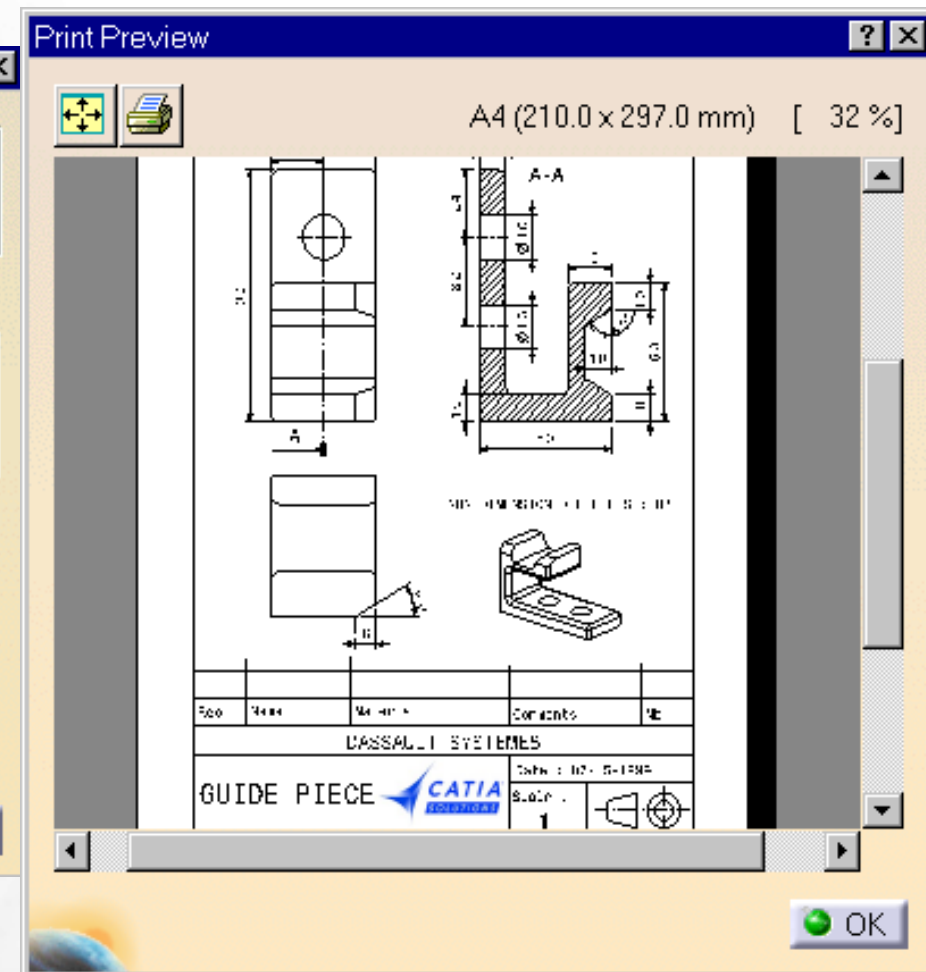
From the File menu,

2. use Print to direct the drawing to the selected printer or plotter
3. use Preview button... to check what will be printed

1

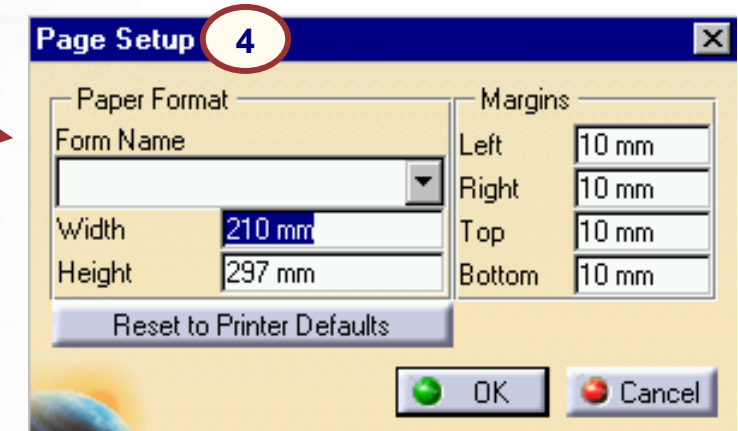
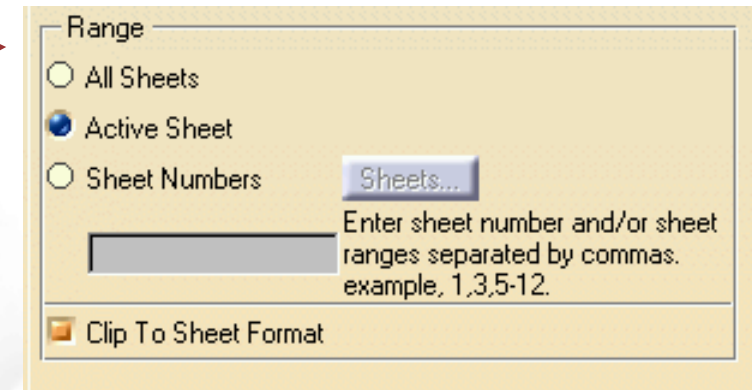
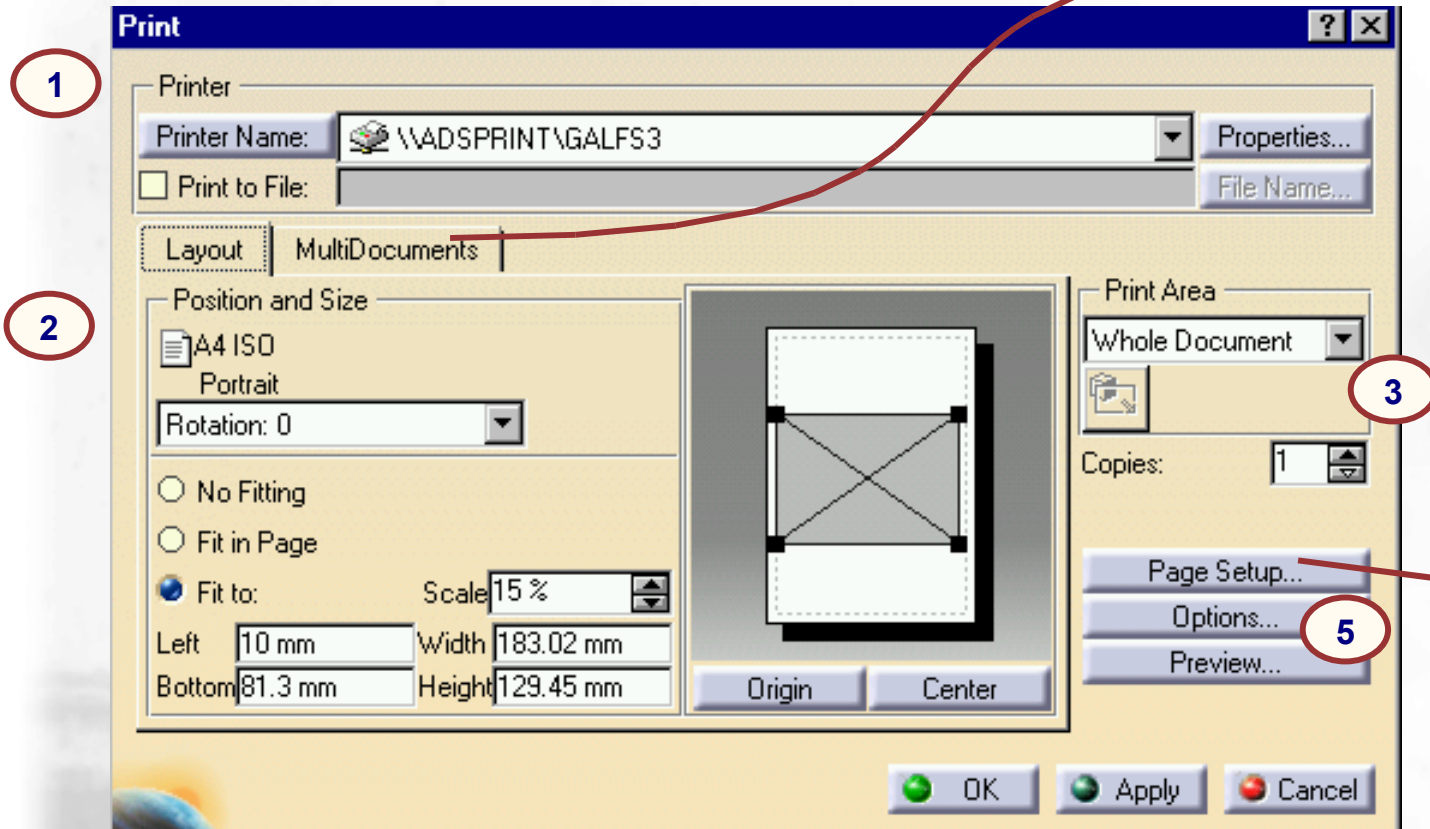


2



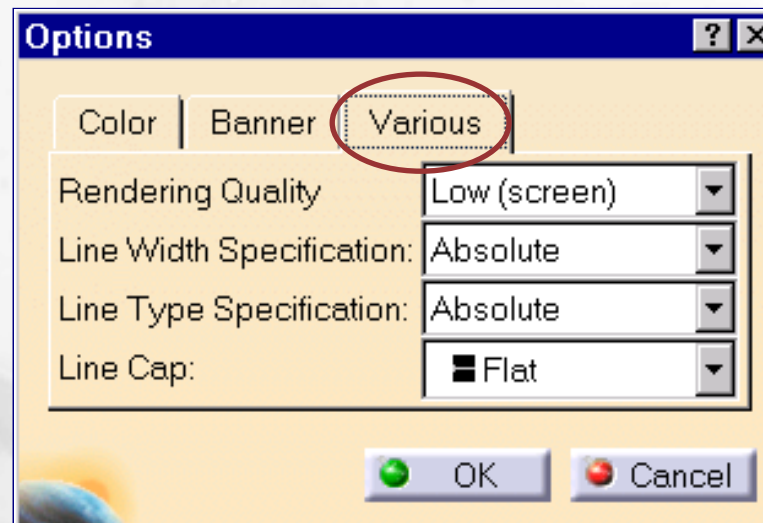
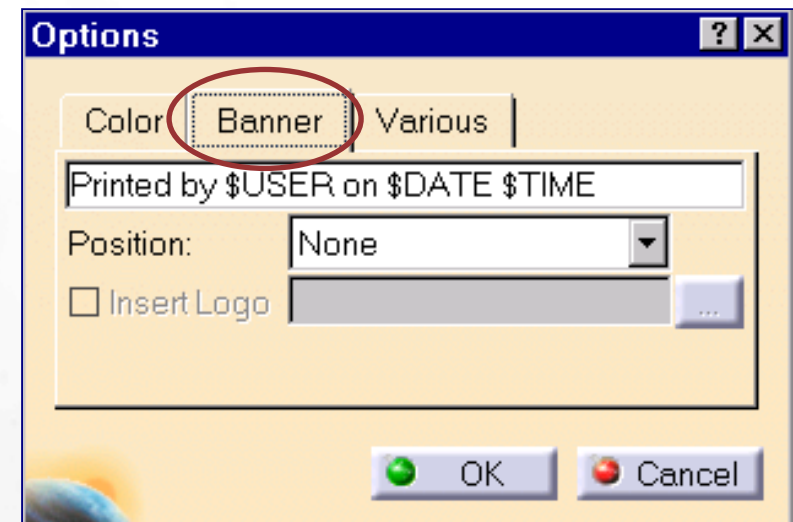
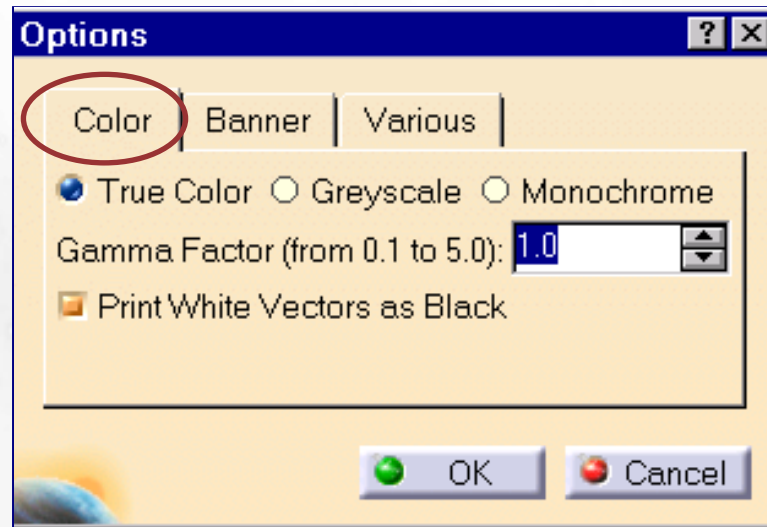
Print User Interface

- 1) **Printer:** Select the printer or key in a file name to print to.
- 2) **Position and Size:** Define the position and size of the geometry on the page
- 3) **Print Area:** Define the area to print
- 4) **Page Setup:** Define the area to print
- 5) **Print Options:** Define the print options



Printing Options

Three options tabs are provided for customizing the printed output



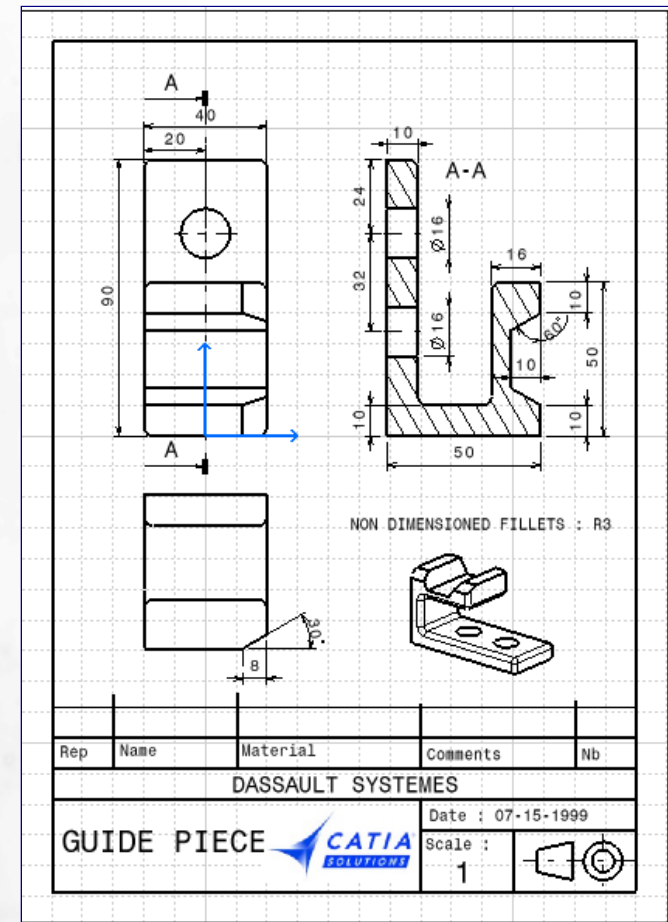
To Sum Up ...

In this lesson you have seen...

■ How to Check Links to Solid 3D Part and Updating a Drawing

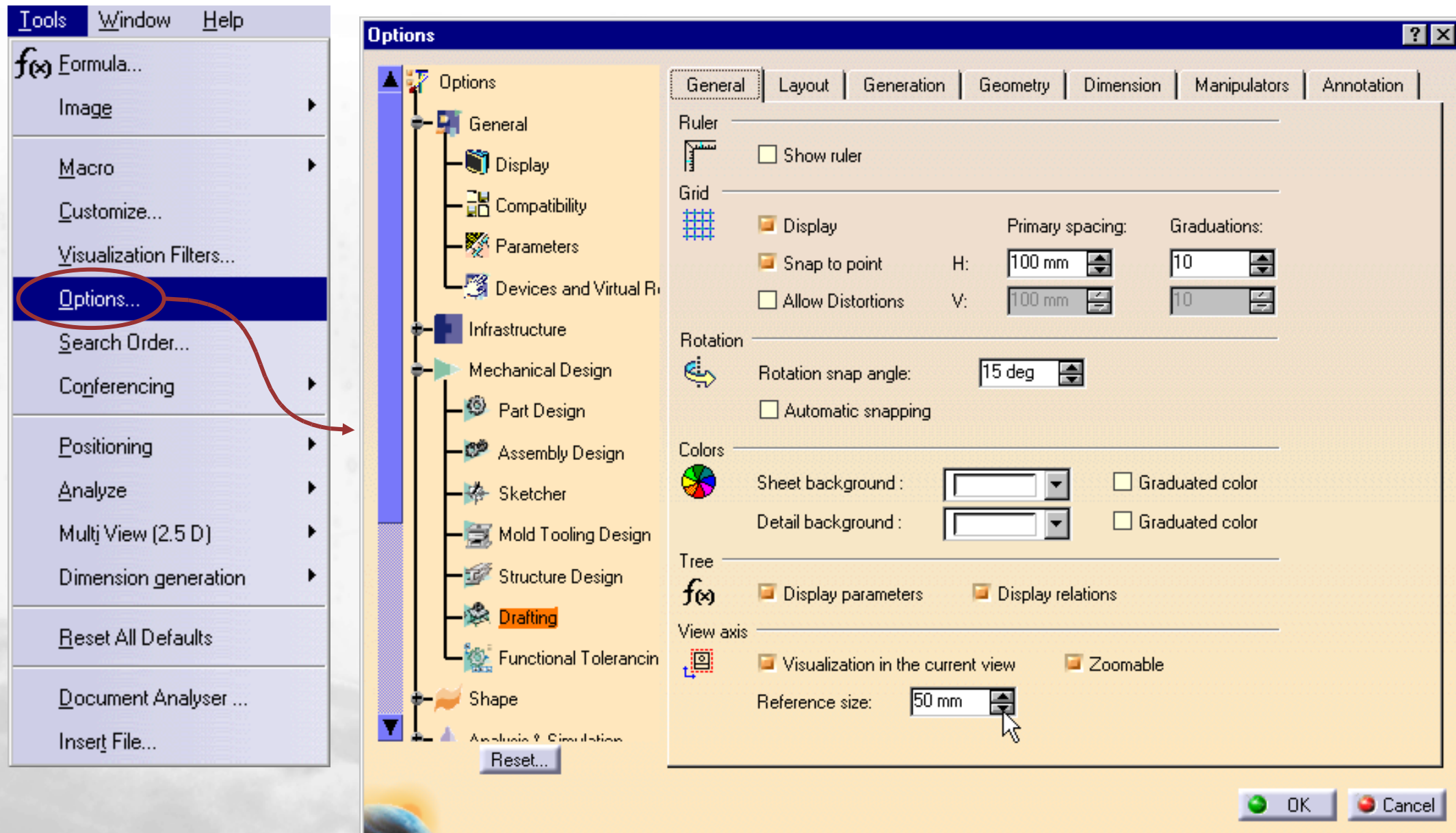
■ How to Add a Title Block

■ How to Print a drawing



Setting Drafting Options

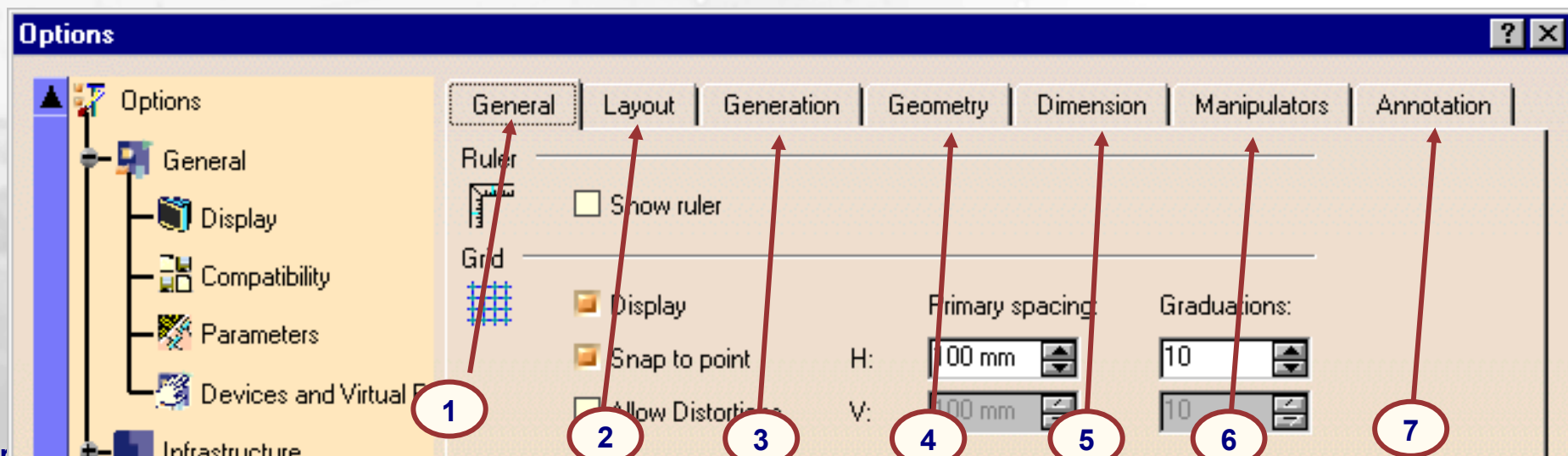
You will learn how to set the session's default drafting options



What are Drafting Options?

There are primarily **seven** Drafting Option tabs that allow the user to customize the drafting interface.

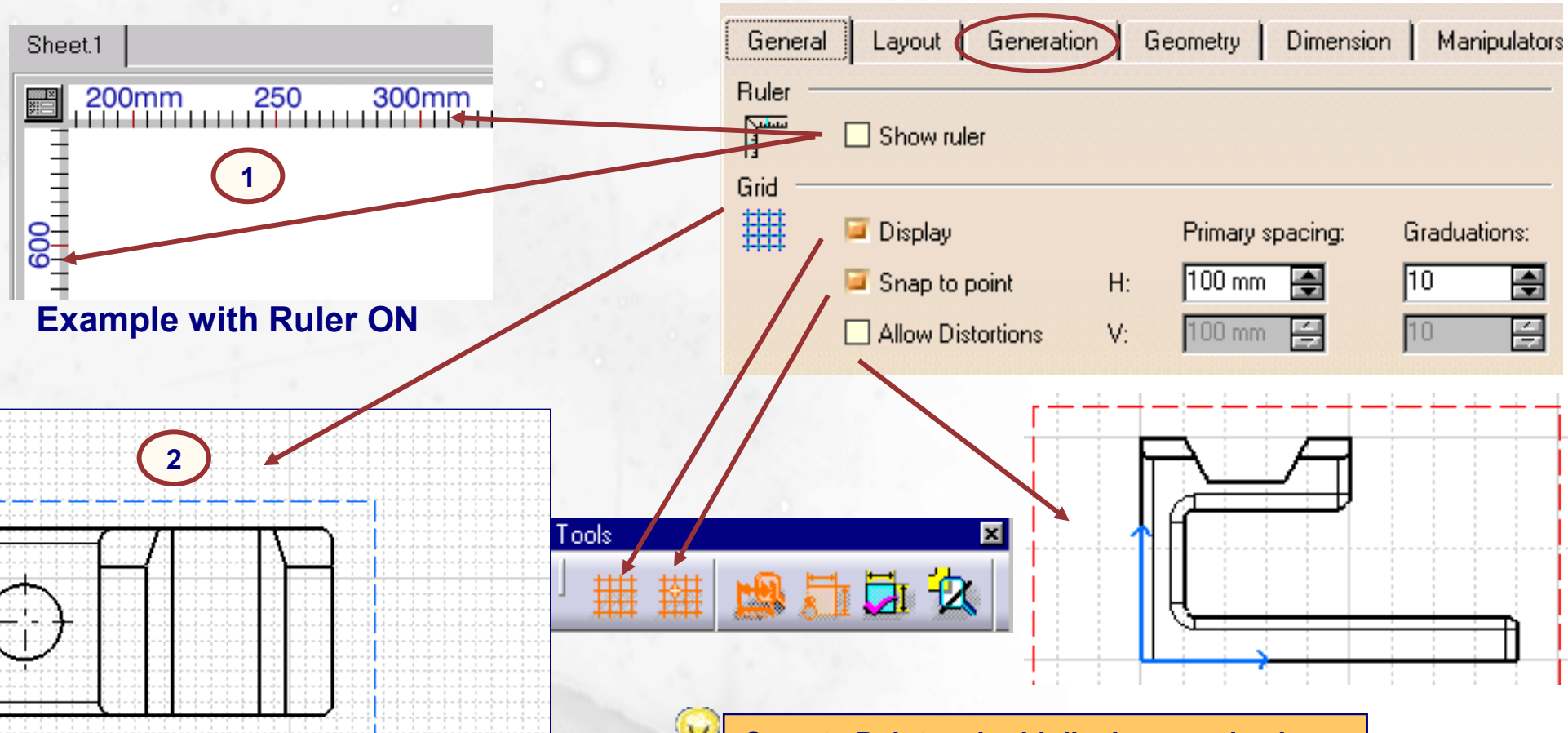
- (1) **General** - Determine display of ruler, grid, background colors and Tree Display
- (2) **Layout** - Determine display of View name, scale, frame and determines new sheet parameters settings
- (3) **Generation** - Determines dimension and geometry generation
- (4) **Geometry** - Aides to create geometry such as display of center points, auto-detection for orientation, and constraint creation and display
- (5) **Dimension** - Position, Line-up dimensions and Analysis display mode
- (6) **Manipulators** – Turns on/Off the manipulators for dimension creation or modification
- (7) **Annotation** – Turns on/off the controls for annotations



General Options (1/2)

Set the following General options:

- 1) **Show Ruler:** In the OFF position the ruler along the top and left side of the screen will not be displayed
- 2) **Grid:** With the Display turned ON and Snap to point turned ON. Adjust the Primary spacing and graduations to aid in dimension placement.
- 3) **Allow distortions:** Allow you to change the scale of H and V on the grid

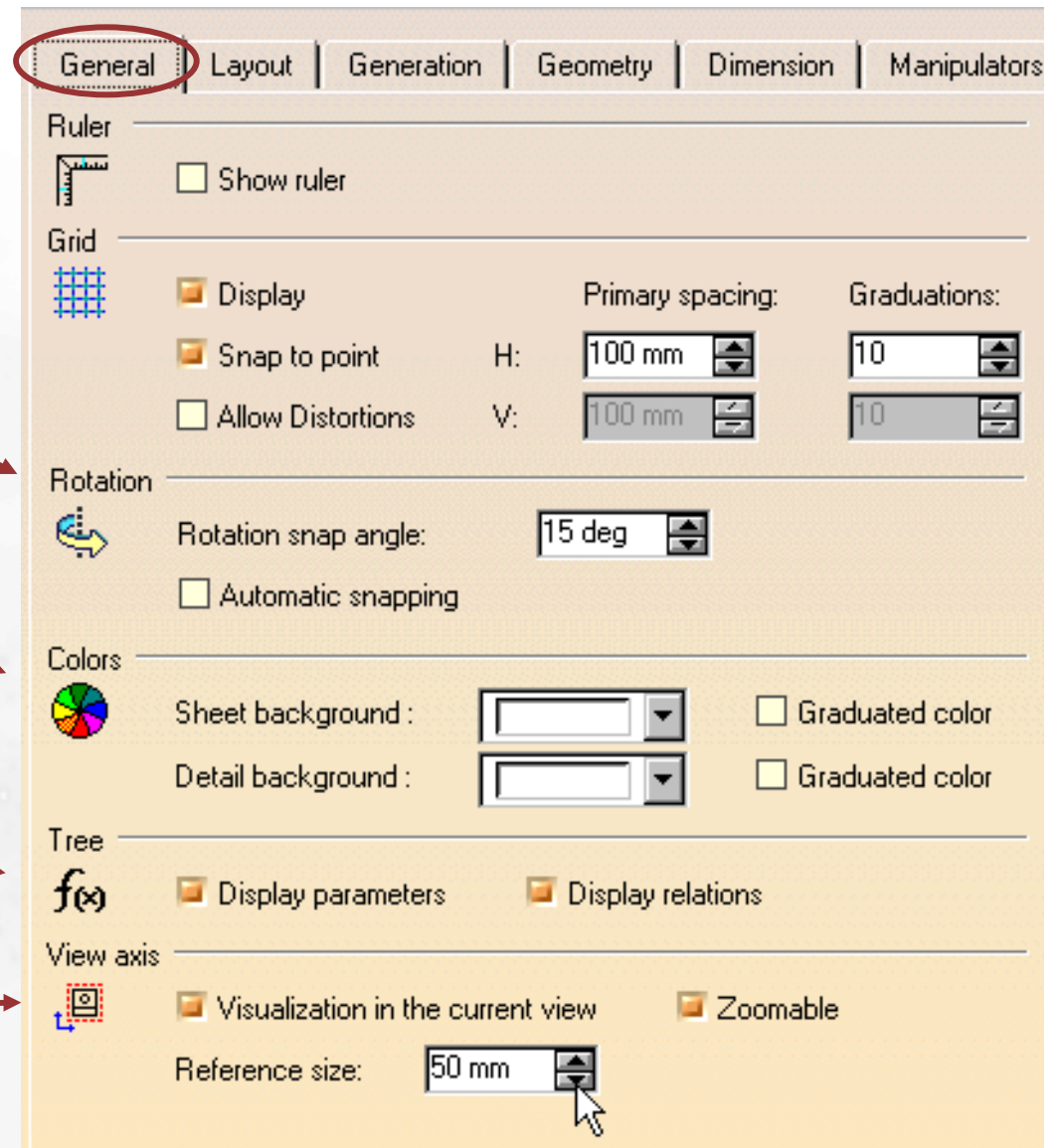


Snap to Point and grid display can also be controlled from the standard toolbars.

General Options (2/2)

Set the following General options:

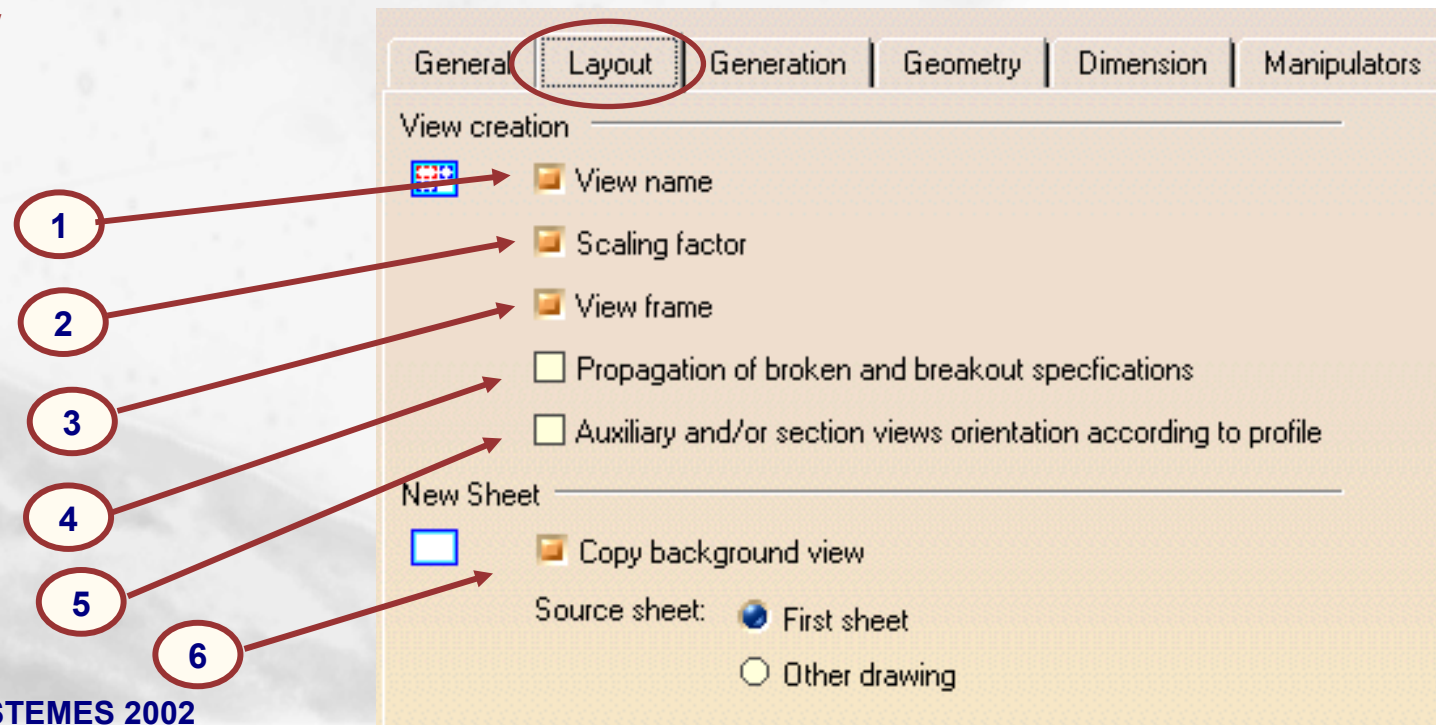
- 1) **Rotation:** Allows to set the View manipulator default rotation angle
- 2) **Colors:** Allows easy identification of a Detail sheet by altering the background color
- 3) **Tree:** Allows parameters and relations to be displayed in the graph tree
- 4) **View Axis:** Provides a blue axis in the view that is current



Layout Options

Set the following view Layout options:

- 1) View name: Check that it is OFF since primary and projected view names are not normally necessary
- 2) Scaling factor: Check that it is OFF since primary and projected view scale will be declared on the drawing as a global scale for the drawing
- 3) View frame: Turn on to easily understand which view is active and to quickly access view properties
- 4) Propagation of broken and breakout specifications: Allows the propagation a Broken or Break-out specification during the creation of a projection or auxiliary view
- 5) Auxiliary and/or section views orientation according to profile: Allow the view axis to be orientation according to profile
- 6) New Sheet : Allows the selection to determine where the sheet properties will be copied from and an option to copy the background view



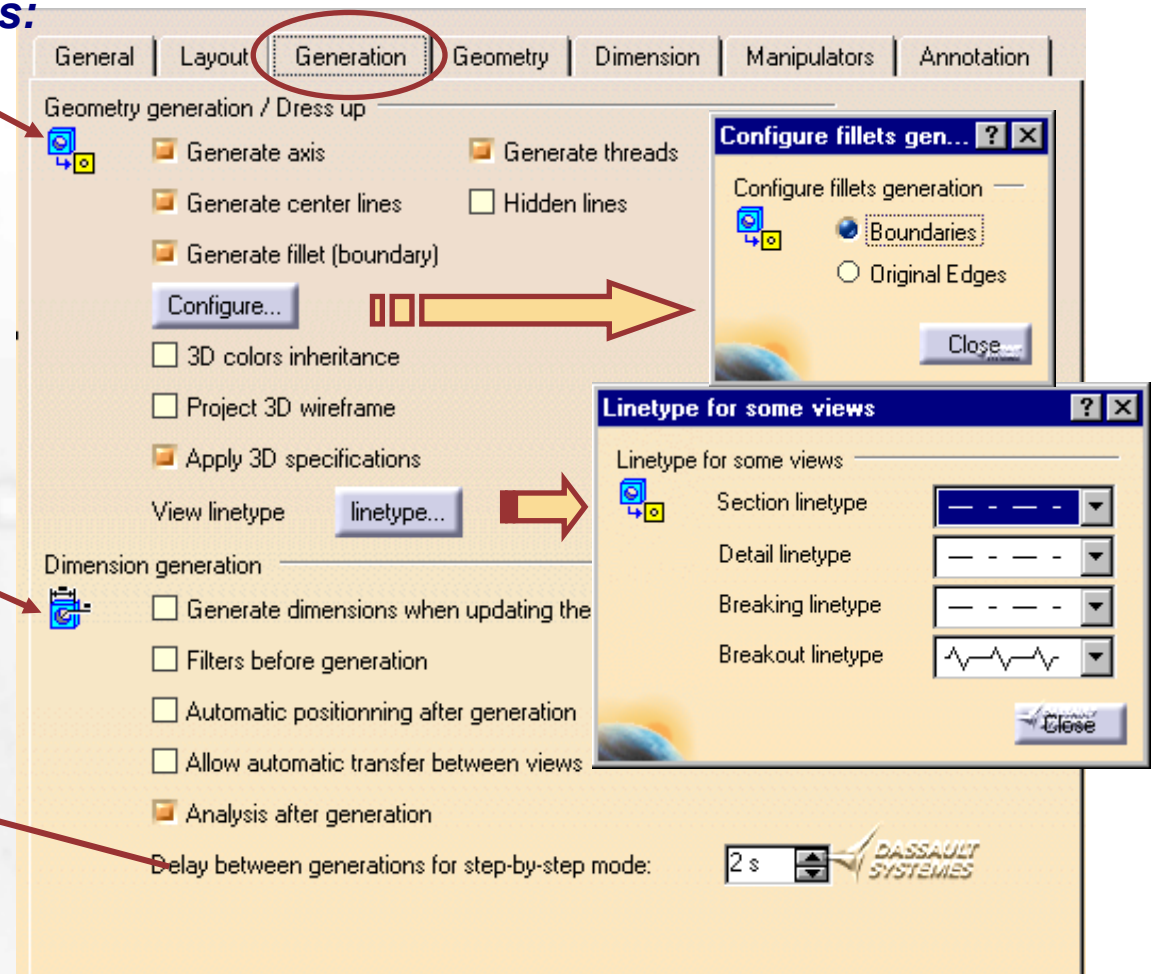
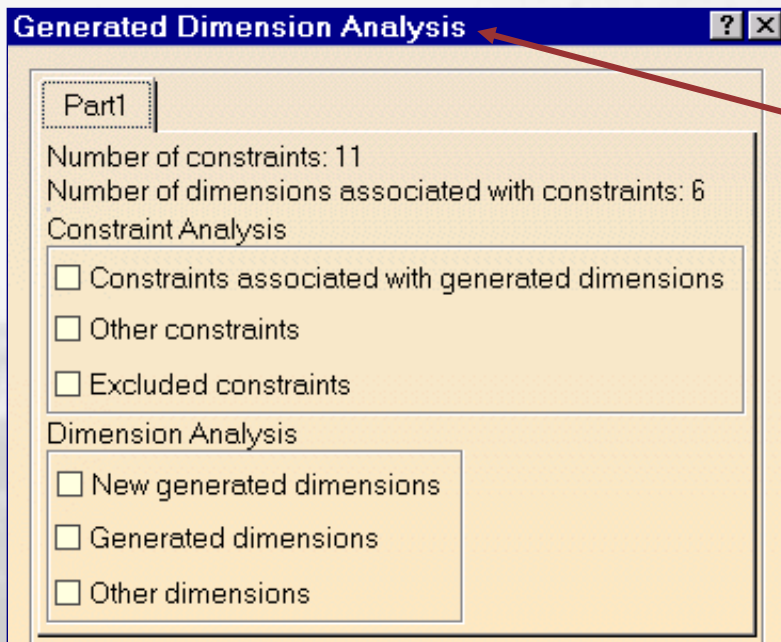
Generation Options

Set the following view Generation options:

- 1) **Geometry Generation:** Allows the automatic creation of axis, centerlines, thread representation, fillet boundaries, hidden lines, 3D colors inheritance, project 3D wireframe and set the view linetype



Threads are generated in top views, side views and section views.



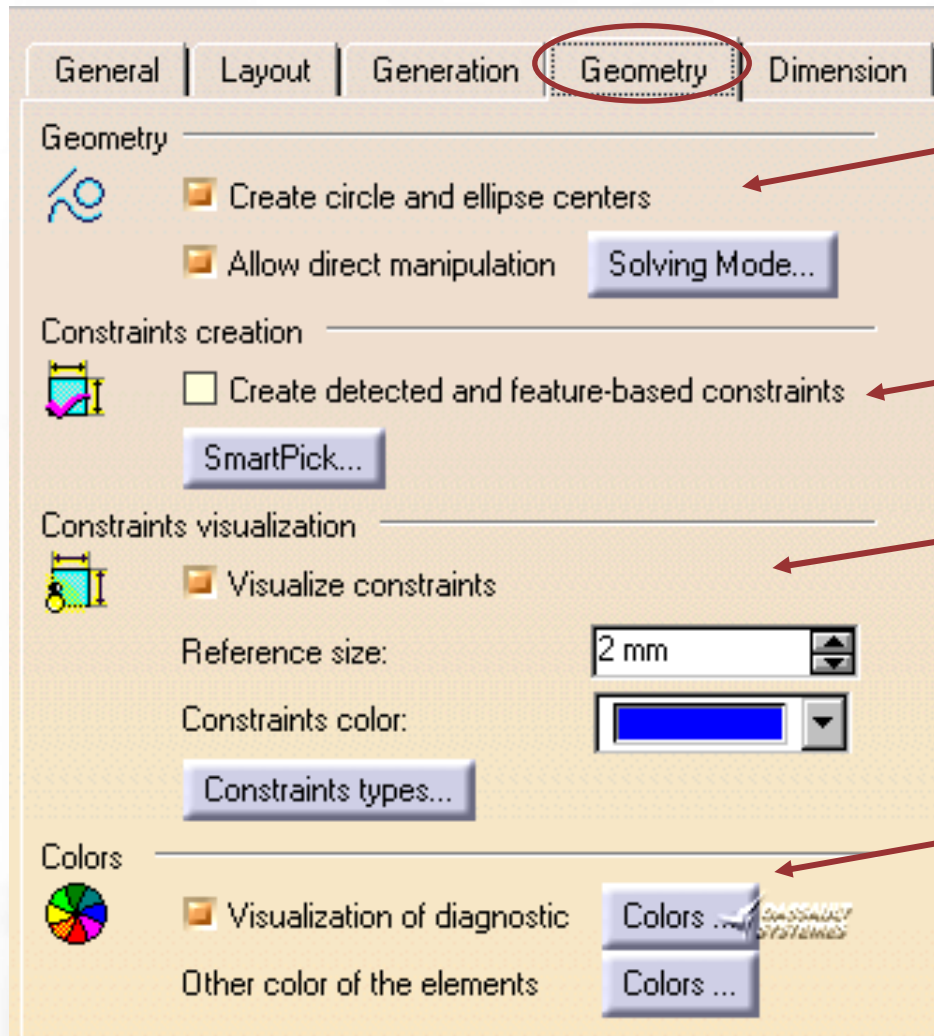
- 2) **Dimension Generation:** Allows dimensions to be automatically positioned after generation, automatic transfer between views and analysis of dimensions that have been generated



Time delay between dimension when using the step by step method can also be set prior to starting the generation process.

Geometry Options

Set the following view Geometry options:

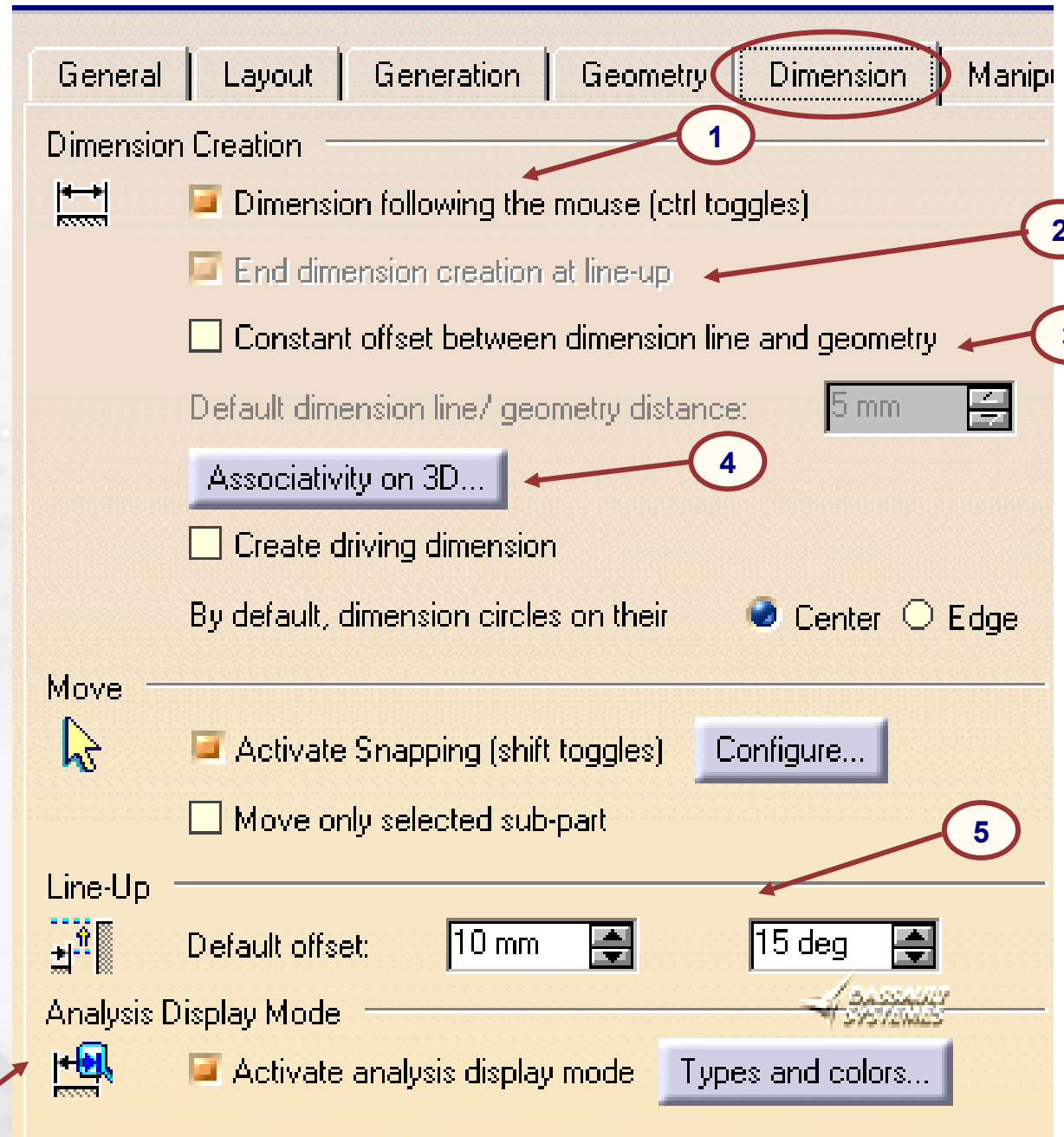


- **Geometry:** Interactive geometry creates circle and ellipse centers and end points included with drag elements
- **Constraint Creation :** Allows for creation of feature based constraints
- **Constraint Visualization:** Allows what constraints will be visualized and the constraints color and size
- **Colors:** Allows you to visualize and choose colors for geometry elements

Dimension Options

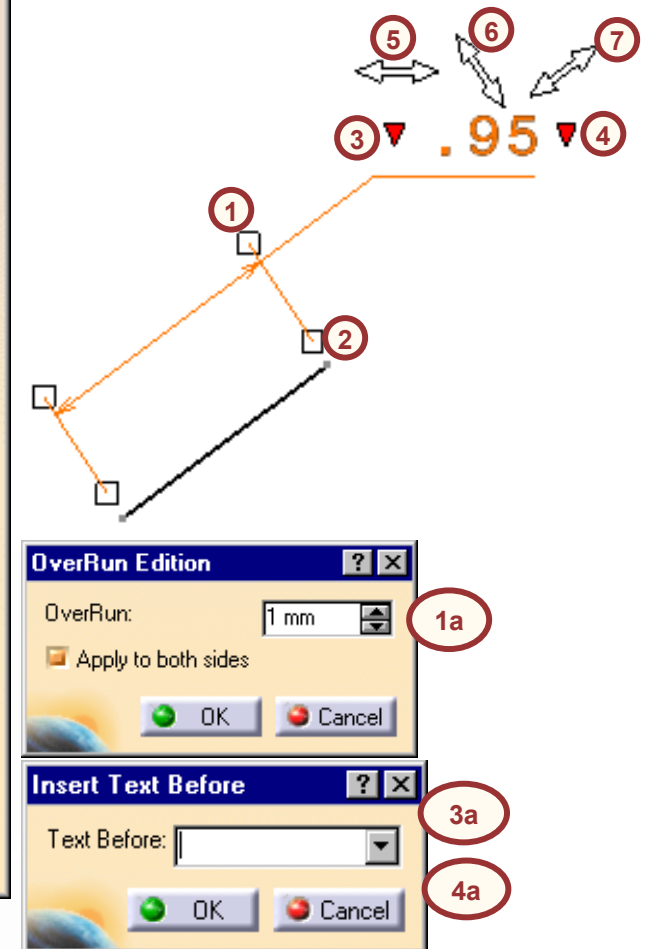
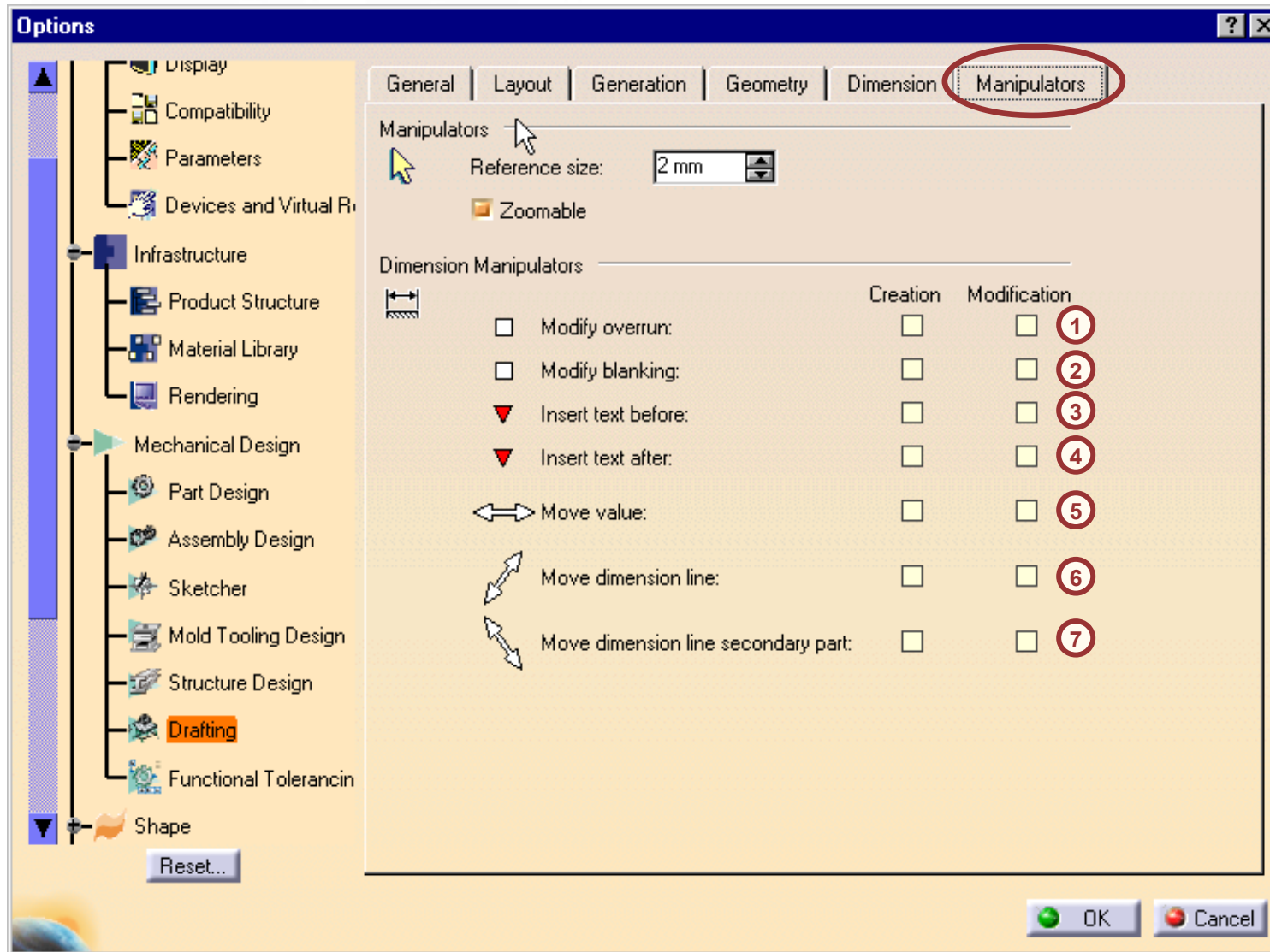
Before creating any dimensions, turn on the following Dimension options:

- **Manual positioning at creation:** allows full freedom for dimension positioning.
- **End dimension creation at line-up:** use if line-up is the last step in the normal dimension creation process.
- **Create associativity dimension line/geometry:** the distance between the created dimension and the geometry remains the same as set by the value.
- **Associativity on 3D :** a link can be applied between a dimension and the 3D part. As a result, when you update the drawing, the dimension is automatically re-computed. If you do not check this option, when you perform the update, you need to re-create the dimension afterwards.
- **Line-Up default:** a default spacing between dimensions when a Line-Up and a reference dimension are selected
- **Activate analysis display mode:** Displays dimension status of Non up-to-date dimensions, Non associative dimensions, converted dimensions, Fake dimensions, Driving dimensions and True dimensions



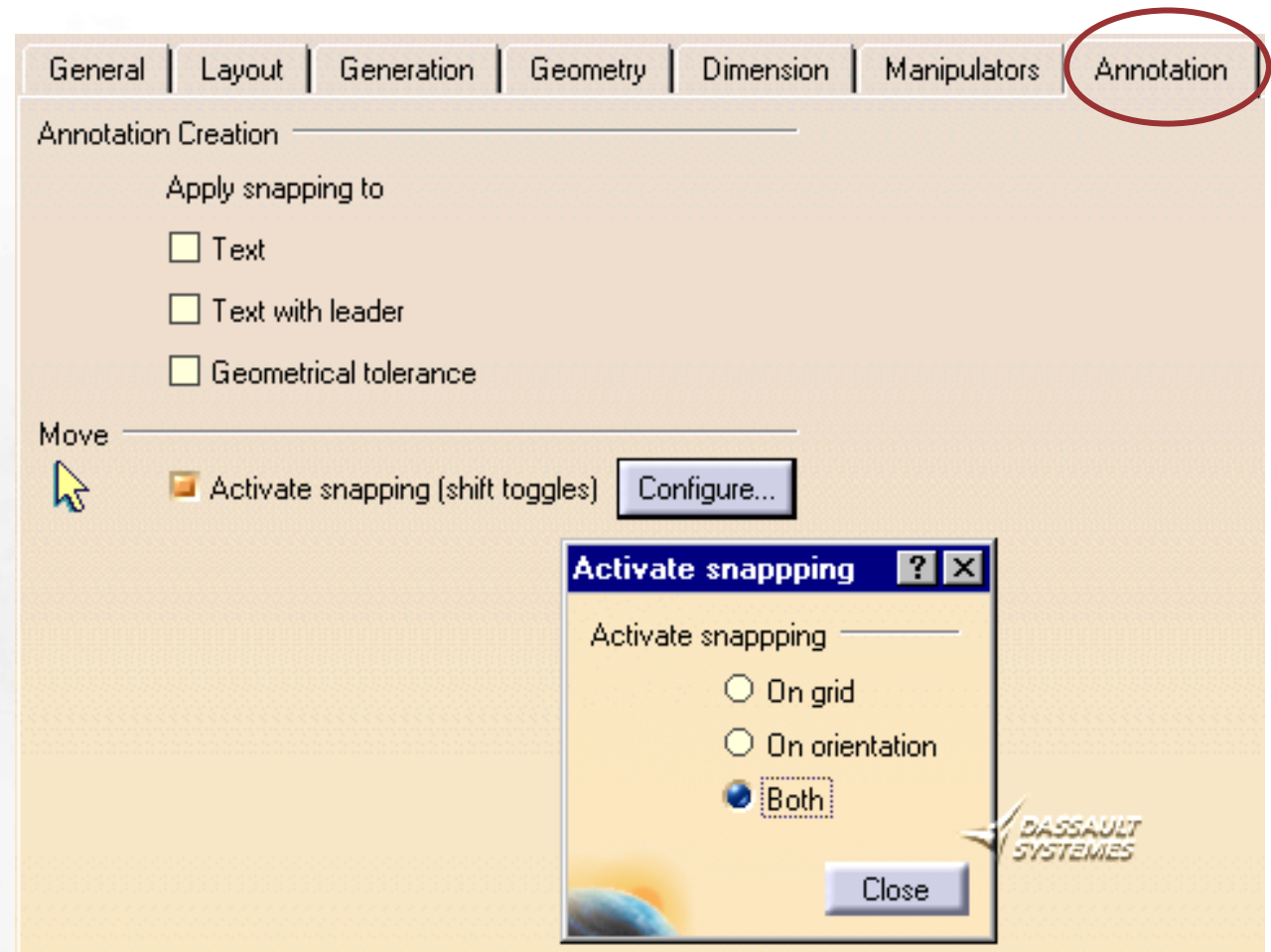
Dimension Manipulators

Option to enable dimension manipulators to control the precise location or properties during creation or modification of dimensions



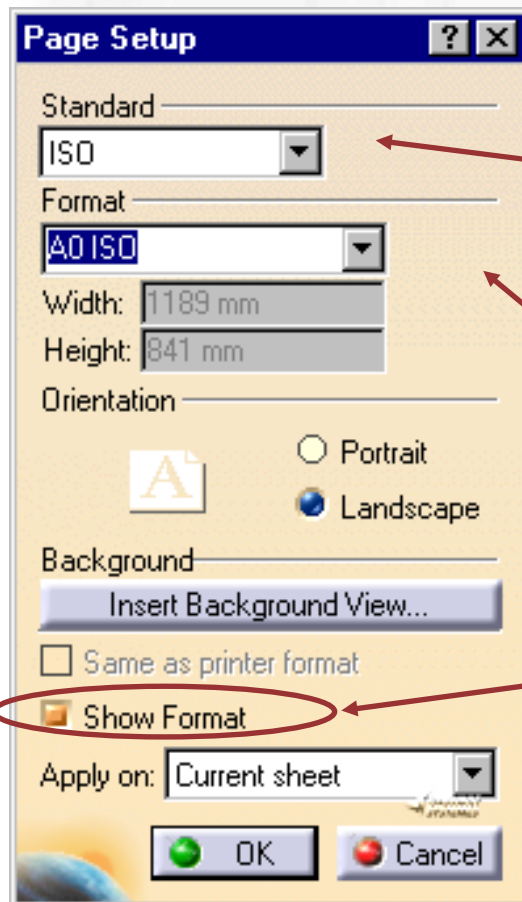
Annotation Options

- Annotation Option for Text, Leader Text, and GDT
 - Settings to choose the leader default behavior
 - Stay Horizontal/Vertical with leader creation
 - Free orientation
- Ability to Swap the text or GDT orientation during creation
 - With the « ctrl » key, swap from vertical to horizontal
 - With the « shift » key, free or lock perpendicular the leader



Drafting Page (Drawing) Settings

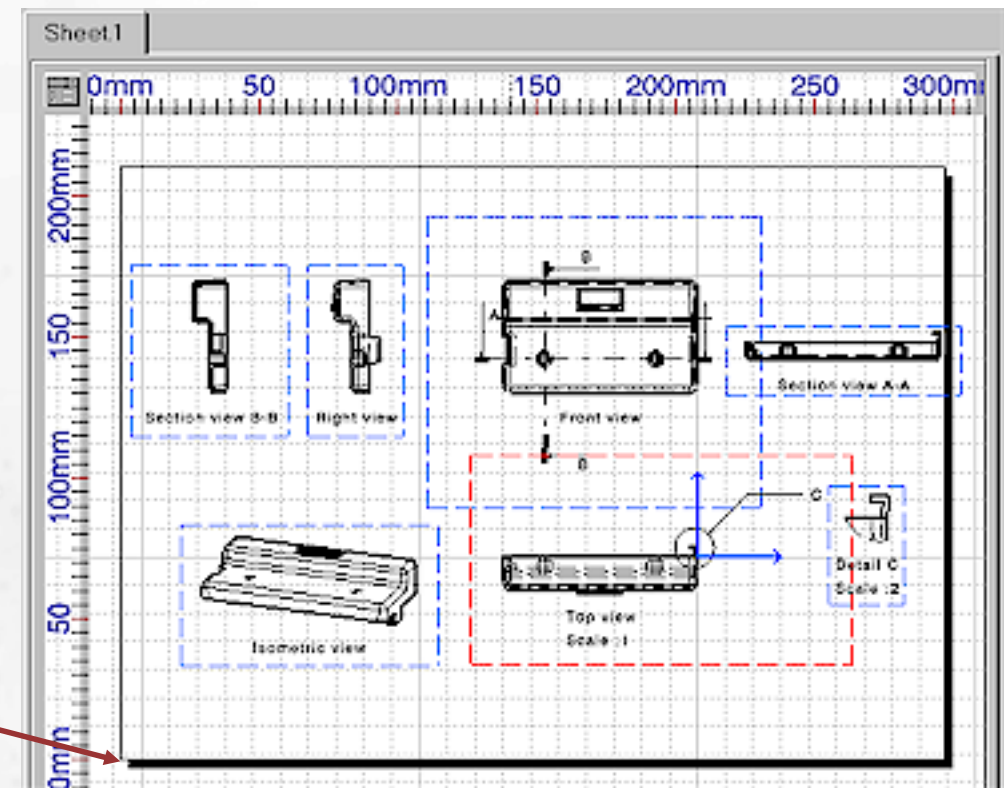
Menu File + Page Setup Properties:



• Drawing Standard

• Drawing Size and Orientation

• Format: on/off

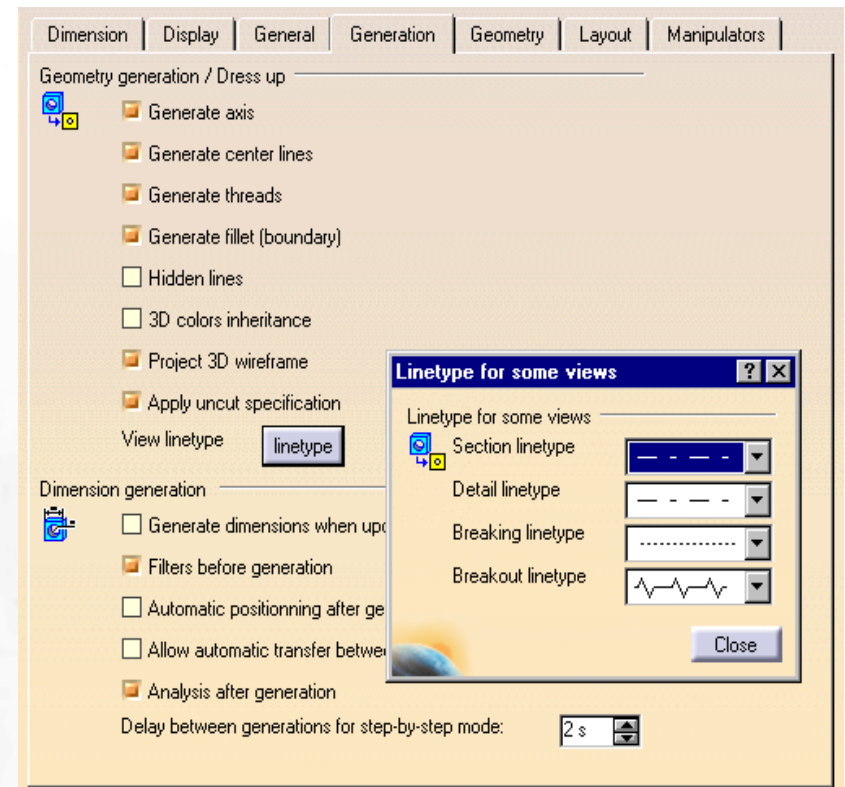


To Sum Up...

In this lesson you have seen...

 **How to set drafting options using the Dimension, Display, General, Generation, Geometry, and Layout tabs**

 **How to set the drafting page standards**



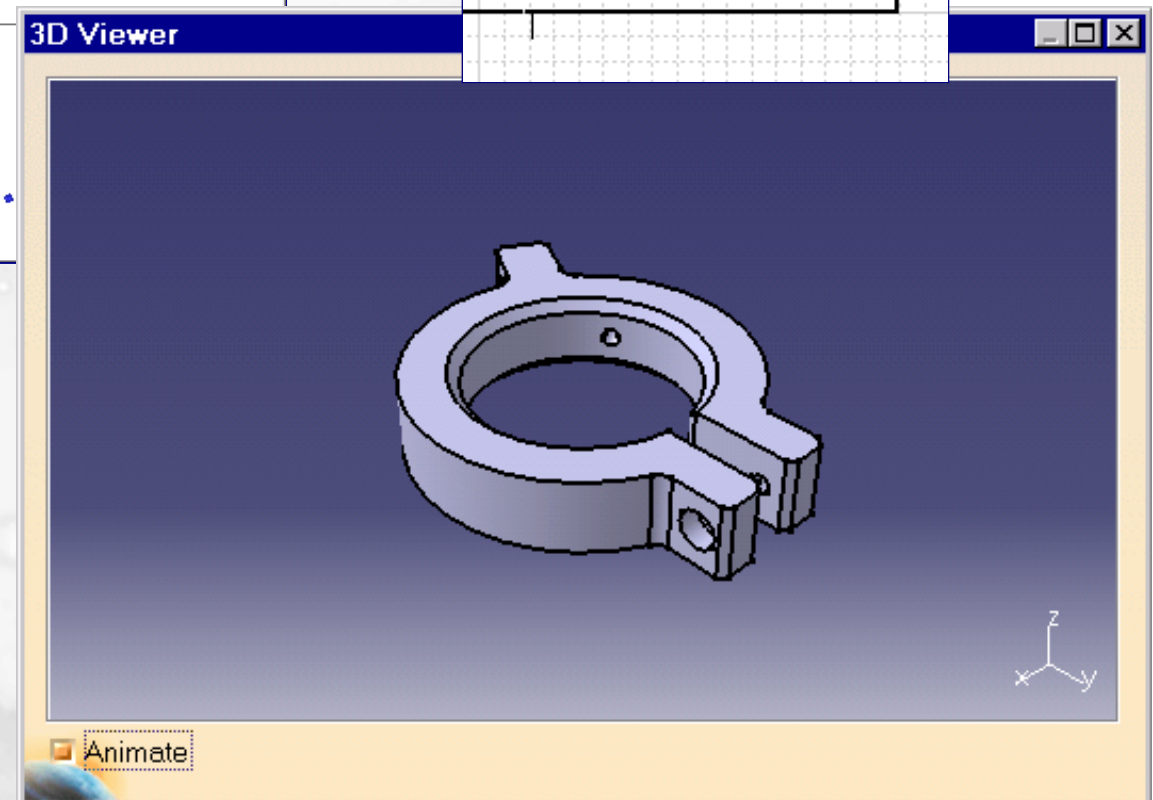
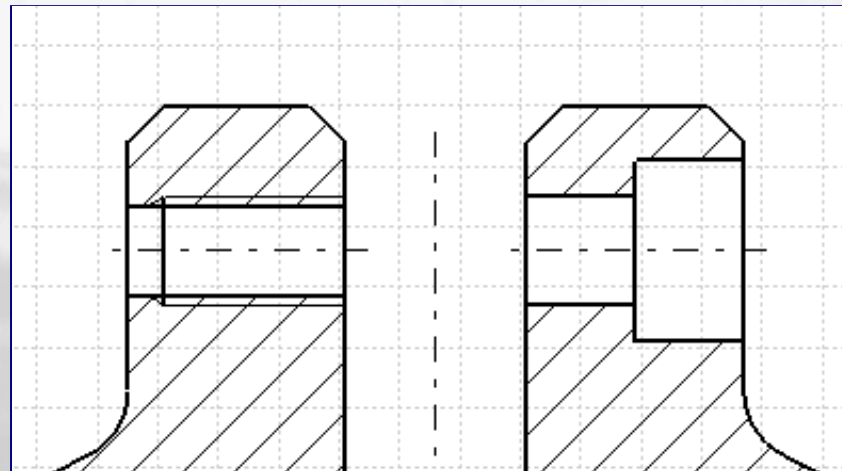
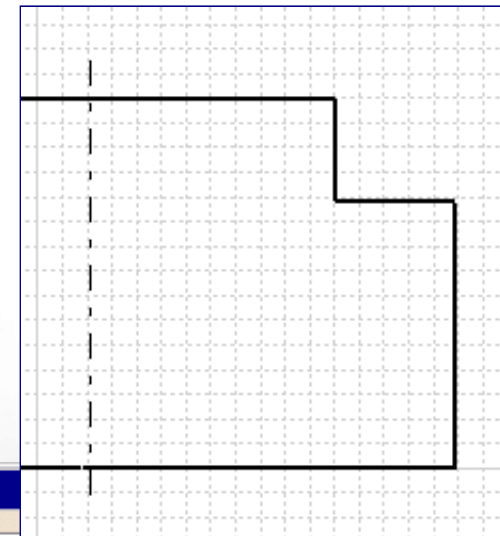
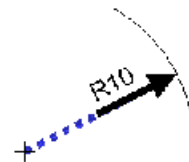
Drafting Visualizations

You will learn how to take advantage of visualization aids and manage drafting visualization standards

Standard section

General Dimension Parameters

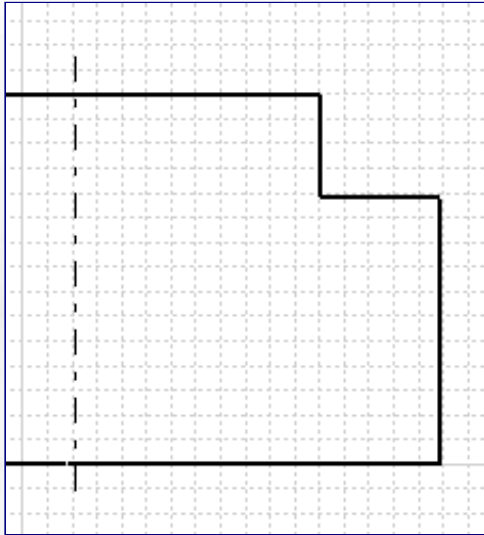
| DESCRIPTION | PARAMETER NAME | VALUES | DESCRIPTION |
|---|--------------------------|---|-------------|
| International standard | ParentStandard | [ISO/ANSI/JIS] | |
| Extension of dimension line on radius dimensions (value inside circle) | RadiusIntReachCenter | [YES/NO] YES = Till center NO = Till value | |
| Extension of dimension line on radius dimensions (value outside circle) | DIMLRadiusExtReachCenter | [YES/NO] YES = Till center NO = constant over-run | |
| | DIMLRadiusExtLength | (mm) | |



What are the Different Types of Visualization Aids or Standards?

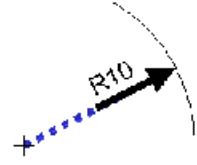

▶ Setting Company Standards

▶ Extraction of Bends in dashed lines for unfolded views



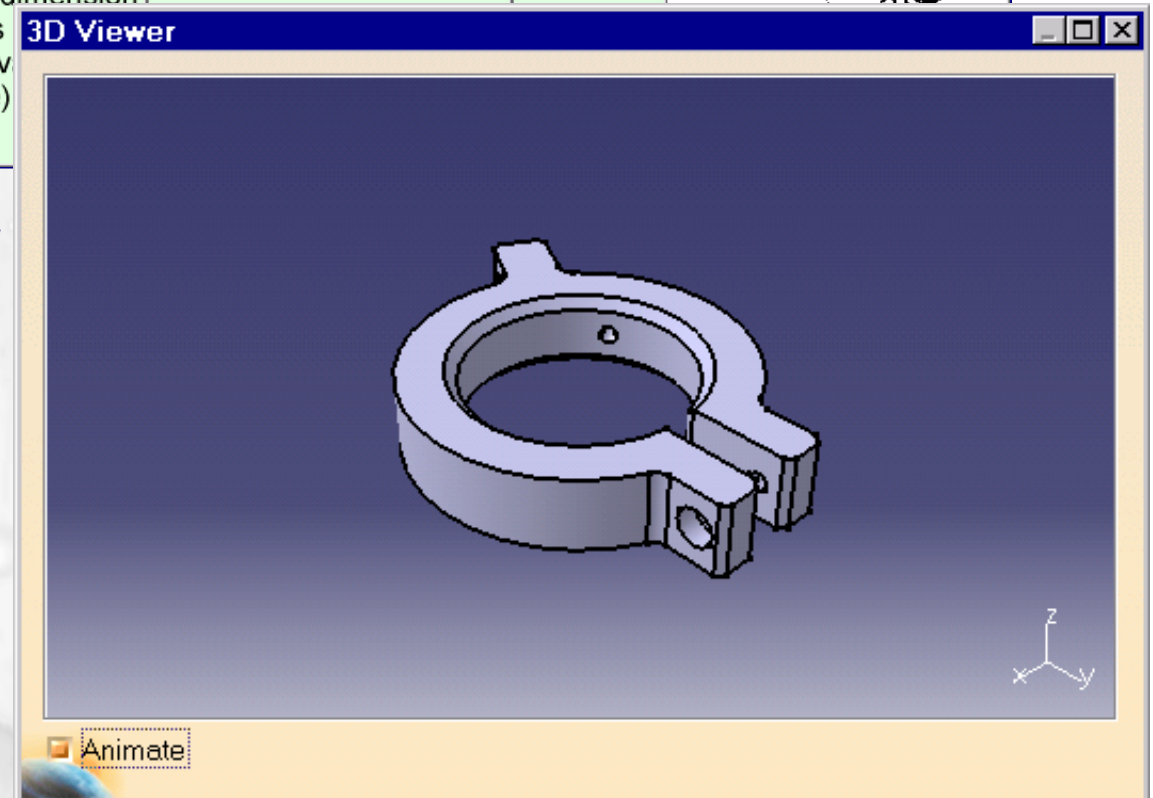
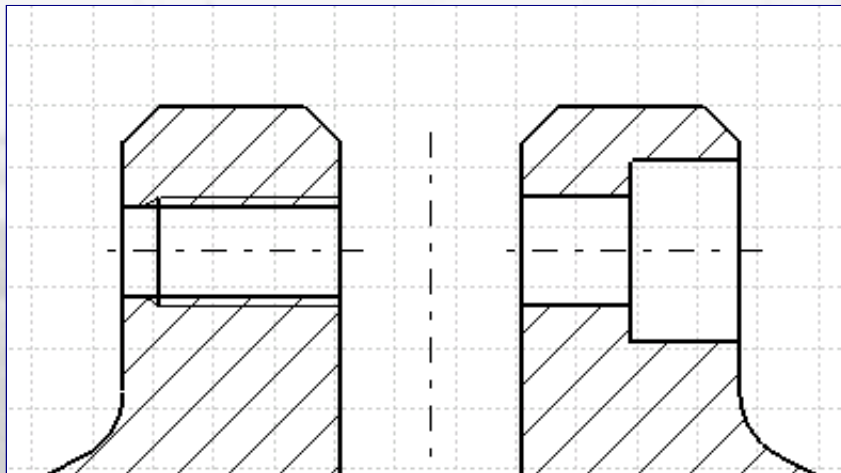
Standard section

General Dimension Parameters

| DESCRIPTION | PARAMETER NAME | VALUES | DESCRIPTION |
|---|----------------------|--|---|
| International standard | ParentStandard | [ISO/ANSI/JIS] | |
| Extension of dimension line on radius dimensions (value inside circle) | RadiusIntReachCenter | [YES/NO] YES = Till center NO = Till value |  |
| Extension of dimension line on radius dimensions (value outside circle) | | [YES/NO] |  |

▶ 3D Viewer

▶ Thread Generation:


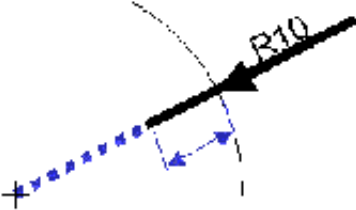


Managing Company Standards

The Standards File controls the representation of text and dimensions. Users or administrators can use this file to create or modify a preferred company style.

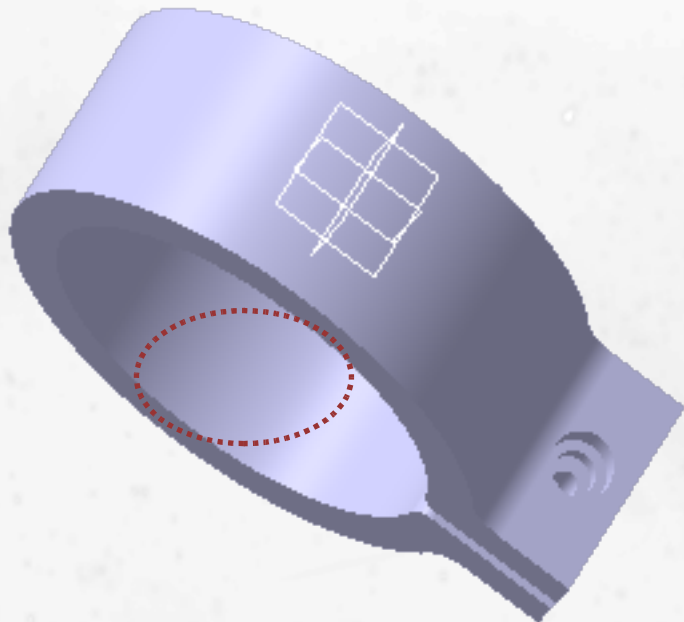
Standard section

General Dimension Parameters

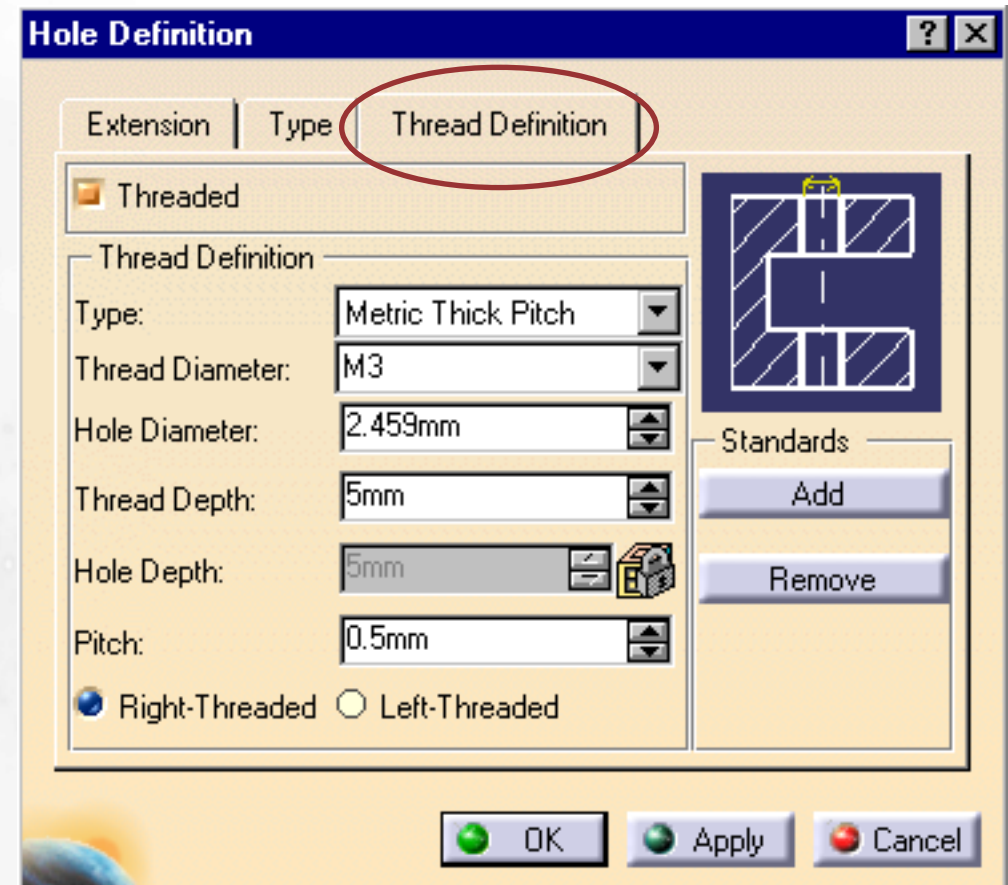
| DESCRIPTION | PARAMETER NAME | VALUES | DESCRIPTION |
|---|--------------------------|---|---|
| International standard | ParentStandard | [ISO/ANSI/JIS] | |
| Extension of dimension line on radius dimensions (value inside circle) | RadiusIntReachCenter | [YES/NO] YES = Till center NO = Till value |  |
| Extension of dimension line on radius dimensions (value outside circle) | DIMLRadiusExtReachCenter | [YES/NO] YES = Till center NO = constant over-run |  |
| | DIMLRadiusExtLength | (mm) | |

Creating Threaded Holes Representation (1/2)

1 In Part Design Workbench, double-click on the threaded hole

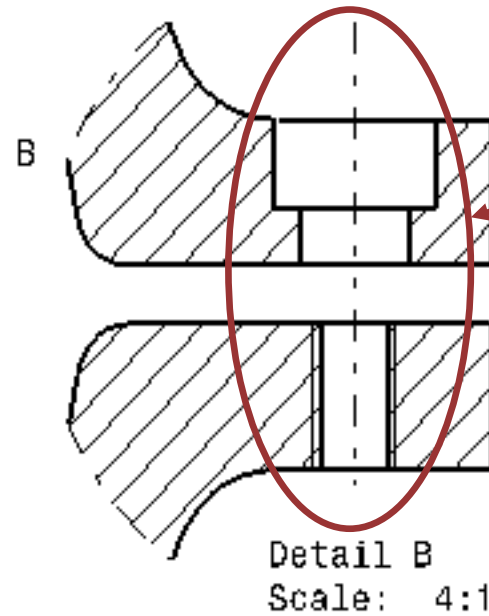
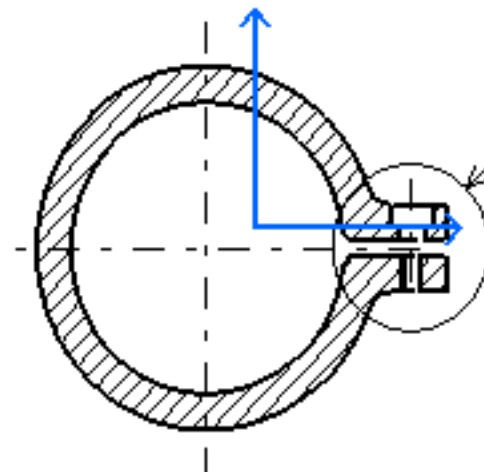
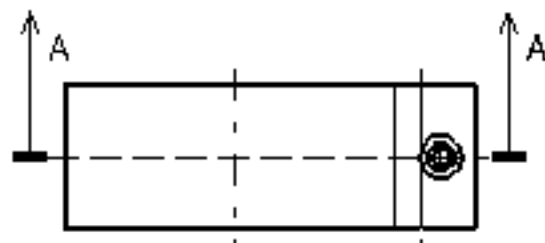


2 Set the thread definition by selecting Thread Definition menu



Creating Threaded Holes Representation (2/2)

- 3 Generate the views or update the existing views of the part.

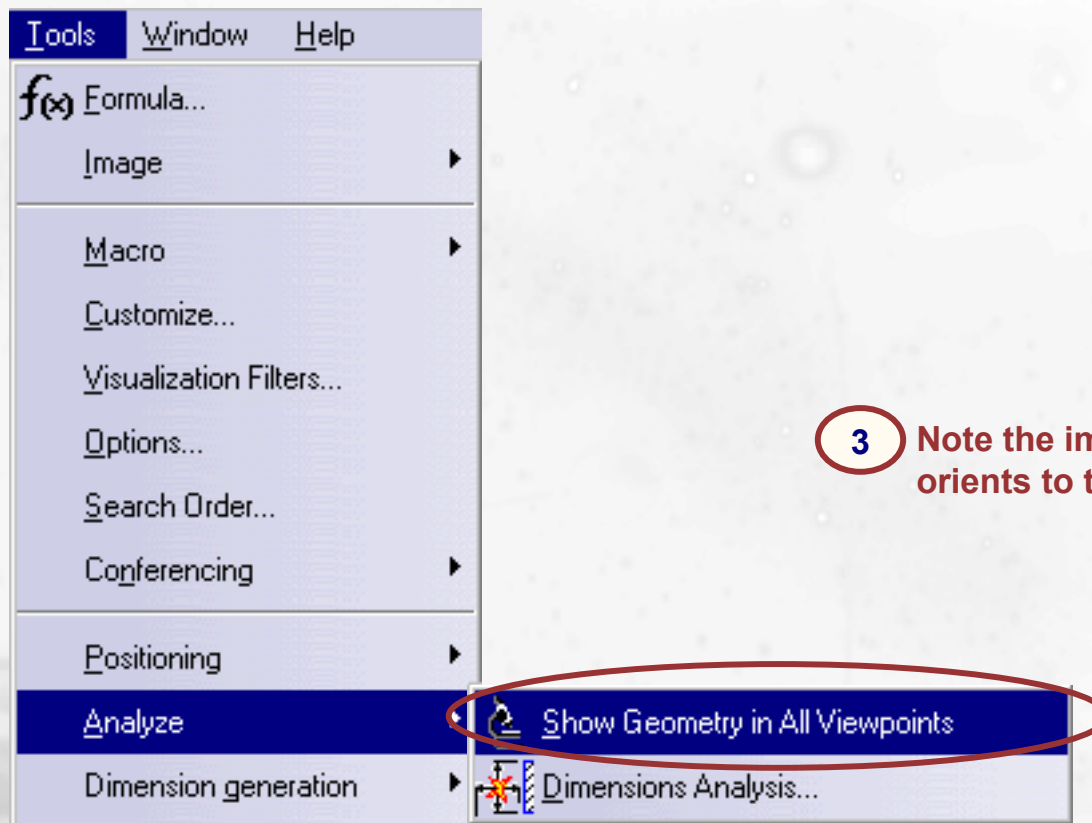


Threaded Hole
Representation

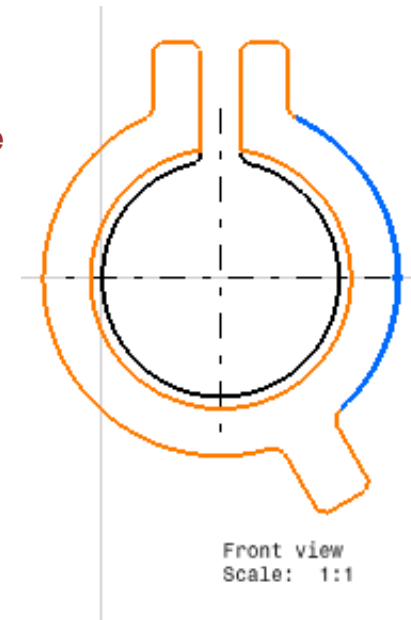
Using the 3D Viewer

The 3D Viewer enables visualization of the 3D element's surface or edge in the views that the element corresponds with

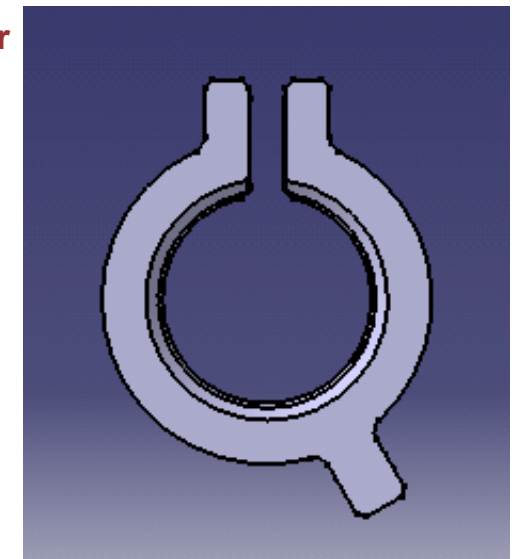
1 Select Tools + Analyze and select Show Geometry in all Viewpoints



2 Move the cursor over a surface in one of the views



3 Note the image in the 3D viewer orients to this surface



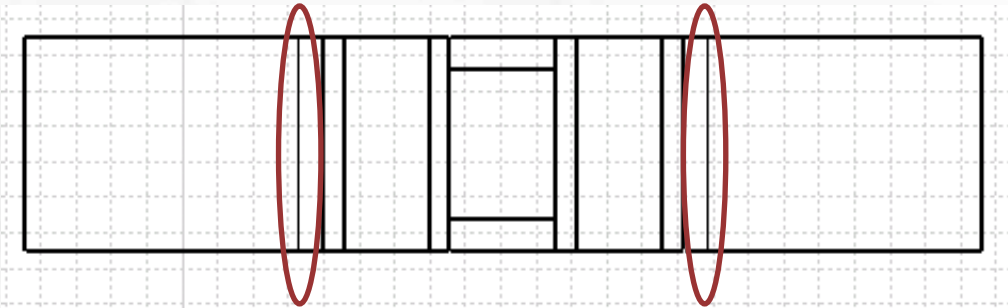
The 3D viewer is only for previewing surfaces and is not available in select mode

Update Persistency of Generated Geometry With Graphical Representation

1 Hide unwanted edges

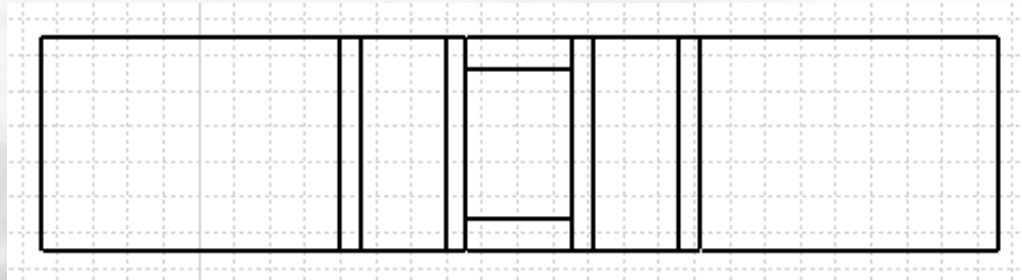
2

Modify the part in any way such that the views require an update



3

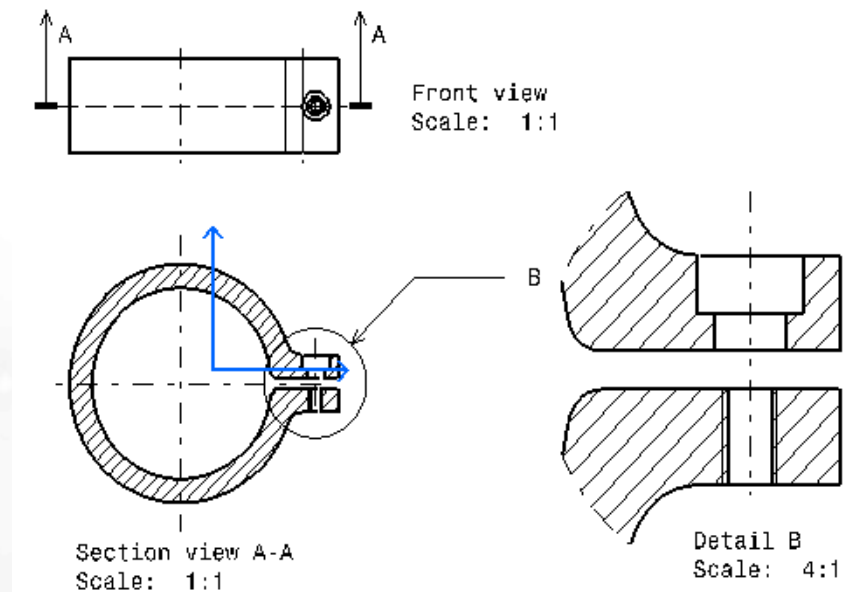
When the views are updated the hide state of the fillet edges is maintained



To Sum Up...

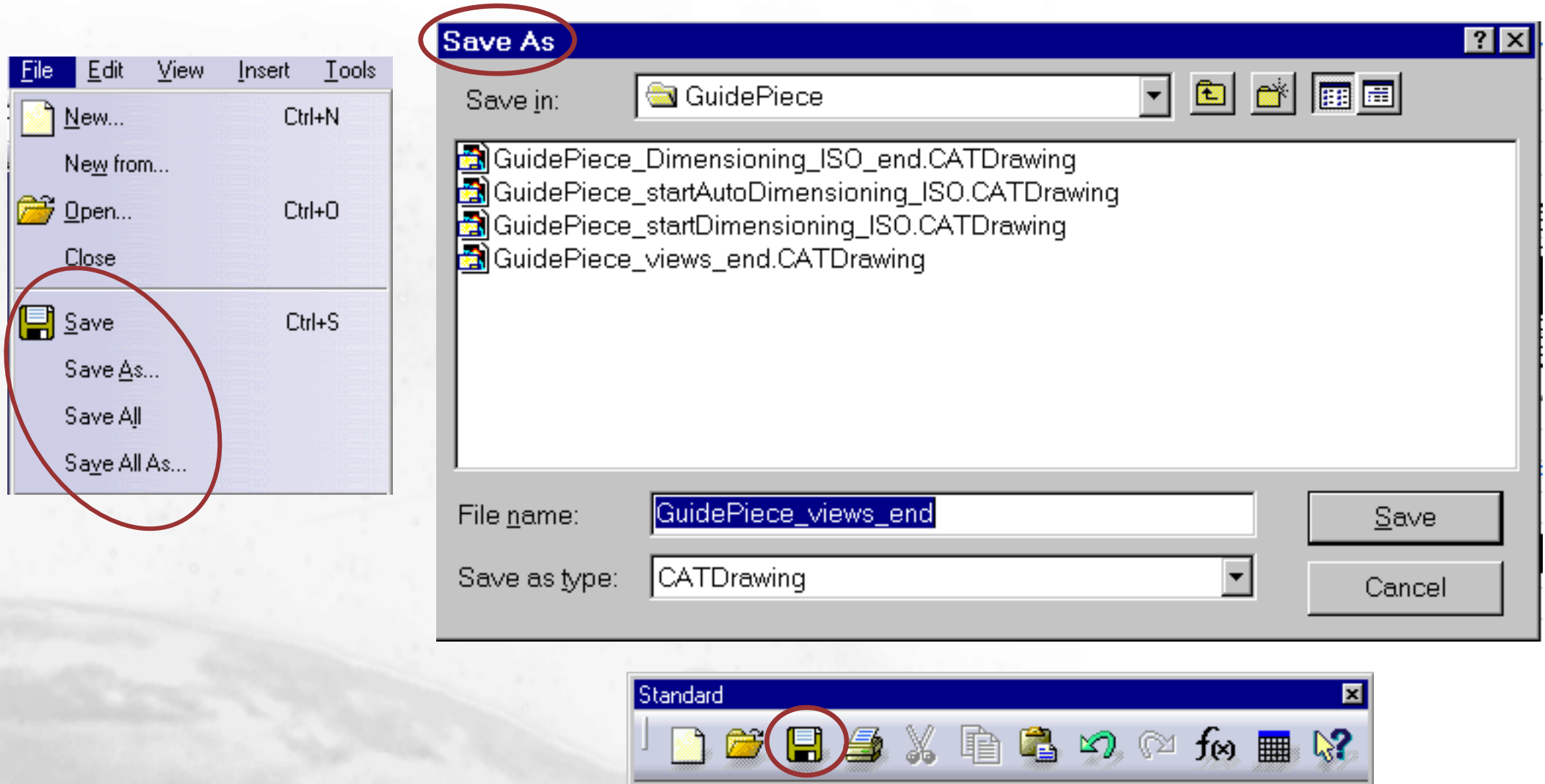
In this lesson you have seen...

- How to set the Threaded hole representation
- How to use the 3D viewer
- How to update Persistency of generated geometry with Graphical Representation



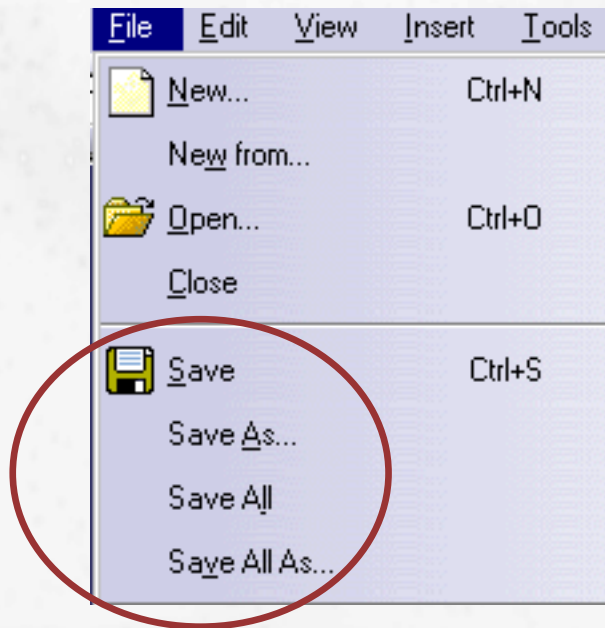
Saving a Drawing Document

You will learn how to save an Drawing Document



Saving Drawing Documents...

There are various ways to save a Drawing Document



Save will save the active drawing document

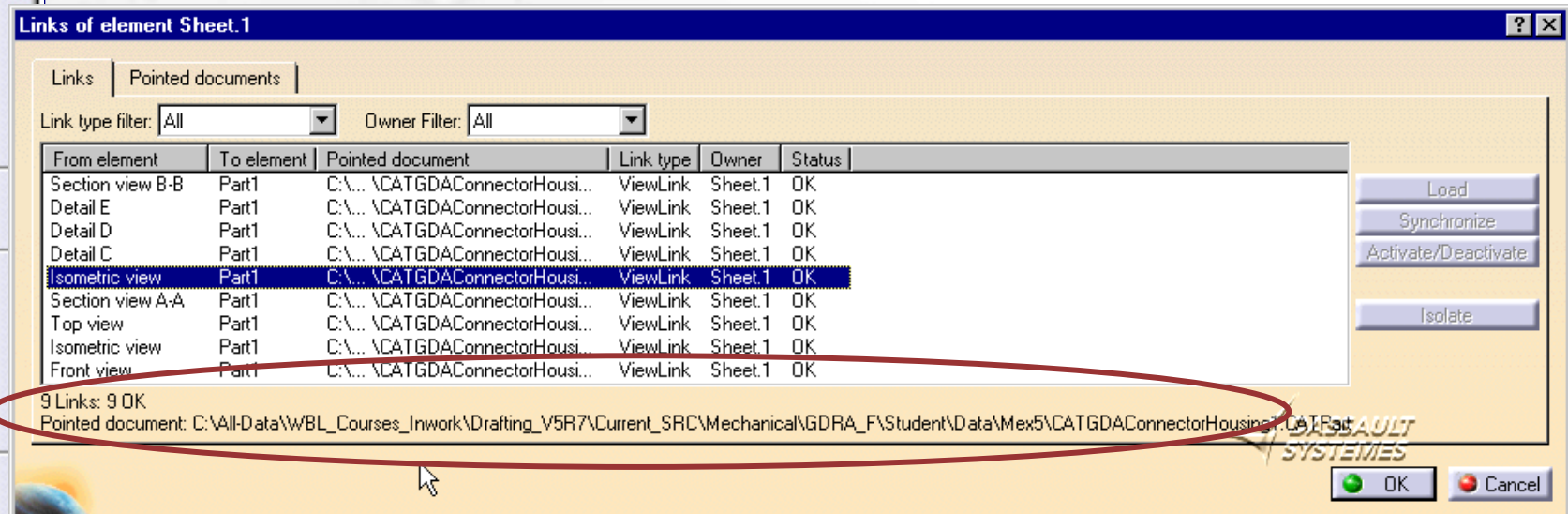
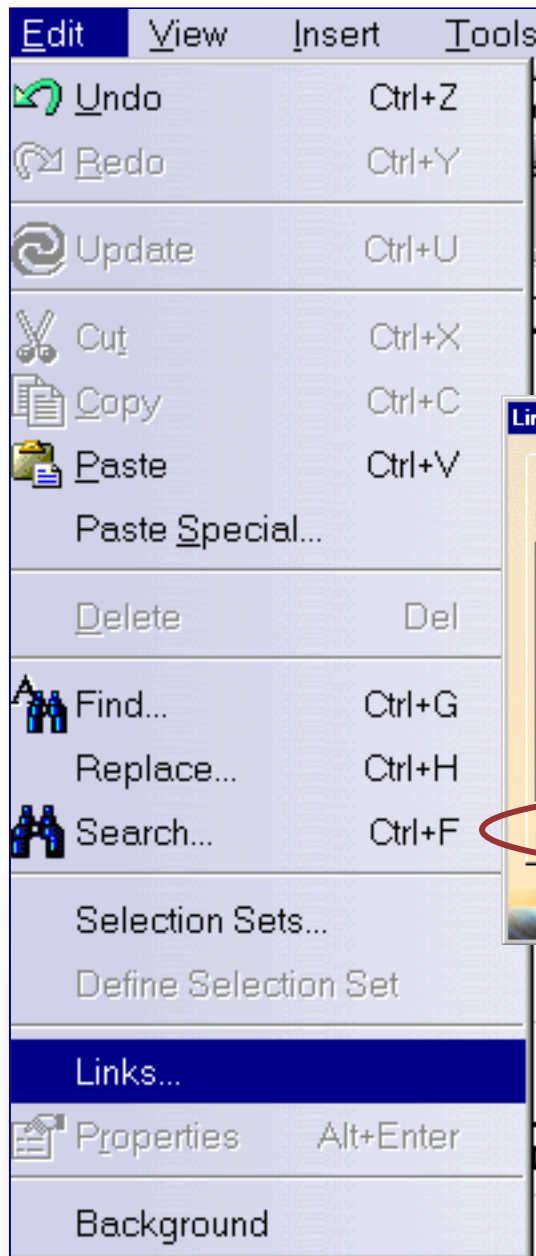
Save As... is similar to Save, but allows you to specify the name and folder for the active document

Save All will propose saving all modified, open documents and children of these document, but you can control which documents actually get saved

Save All As identifies the state of ALL open documents (new, modified, open but not modified), allows you to select which documents to save and allows you to specify the name and folder for these documents

Checking the Links to the 3D Part

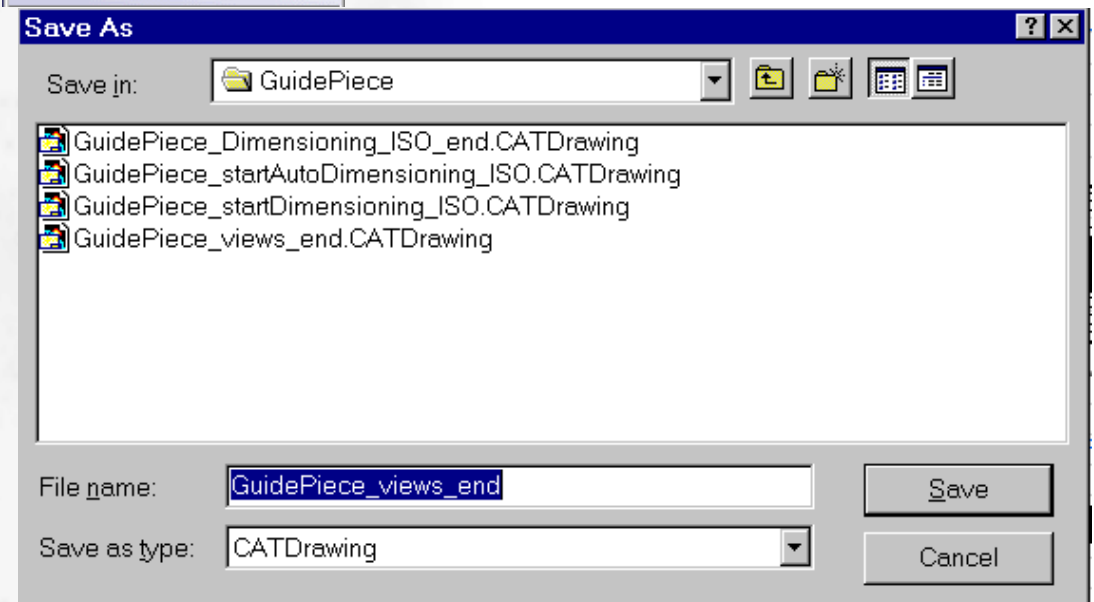
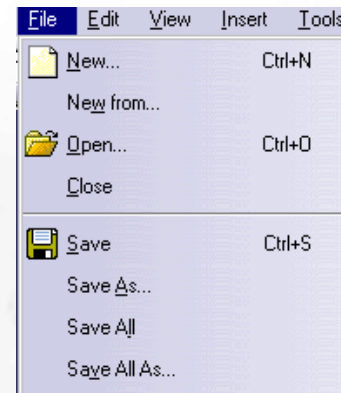
- When a generated drawing is saved, the link between each view and the original 3D part is saved as well as “Stored source”.
- To check the information stored , use the **Edit + Links ...** menu right after saving - this might help prevent any surprises next time the document is opened:



To Sum Up...

In this lesson you have seen...

- How to save a drawing
- How to check the links of a saved drawing



To Sum Up...

In this course you have seen...

- Introduction to the Generative Drafting Workbench
- How to start a generative drawing and view generation
- How to create any additional section, auxiliary, isometric or exploded views
- How to edit a view's layout and properties
- How to finalize a drawing and print
- How to set the drafting options
- How to set the drafting visualizations
- How to save a drawing document and check the links

