



CATIA V5 Training

Foils

Student Notes:

Detail Drafting

Version 5 Release 19
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EDU_CAT_EN_DDR_FF_V5R19

About this course

Objectives of the course

Upon completion of this course, you will be able to:

- Create an interactive view and draw a sketch on it
- Insert annotations to dress-up the views
- Use advanced dimensioning tools

You will also be introduced to the 2D-3D links management and to the drafting workbench customization

Targeted audience

Draftsmen

Prerequisites

Students attending this course should be familiar with the basics of 2D views creation in CATIA V5



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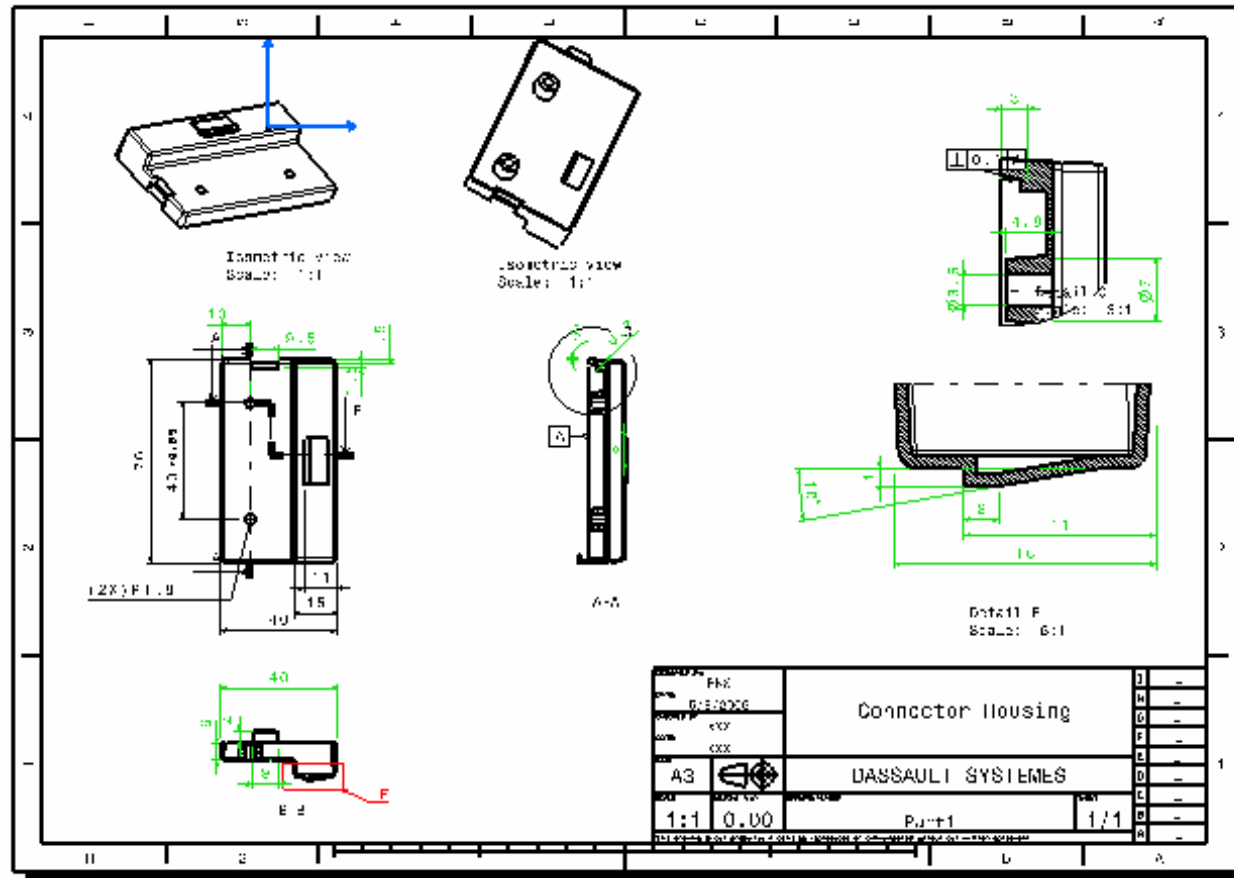
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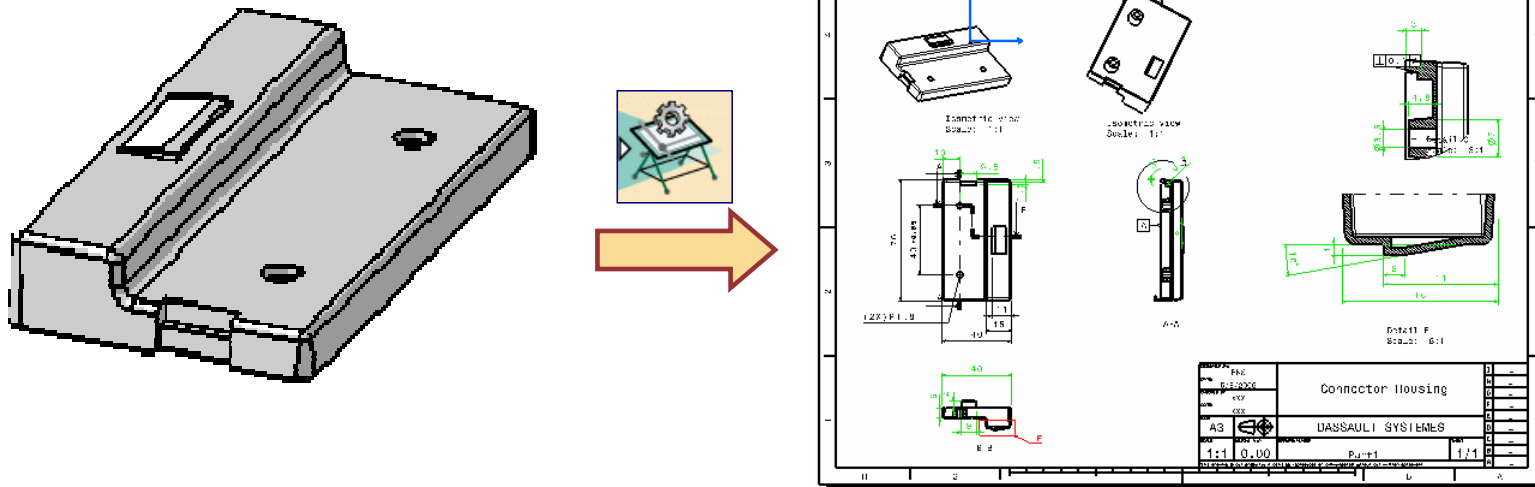
Introduction to Drafting Workbench

You will get familiar with the Generative Drafting Workbench.



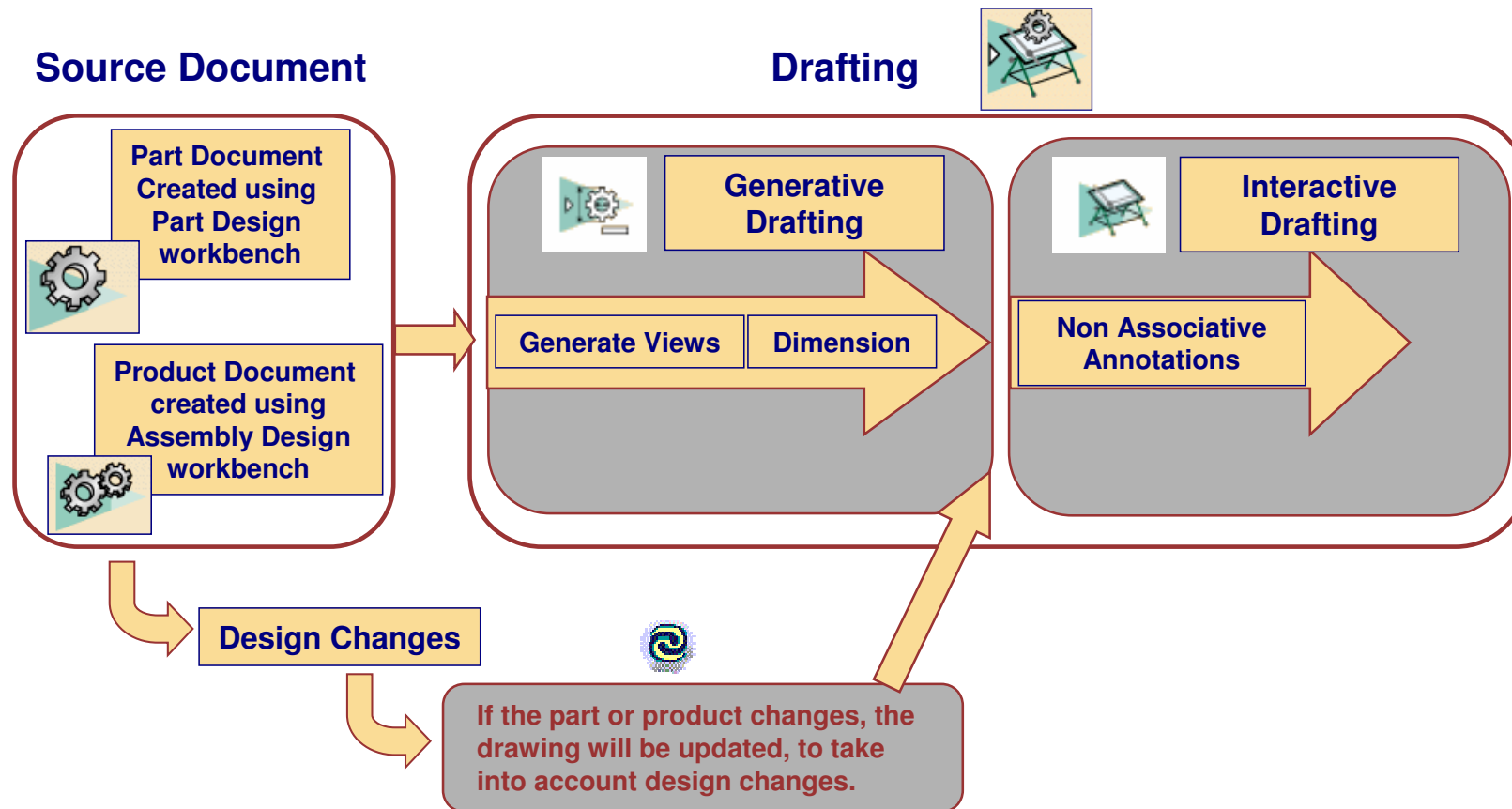
Generative Drafting Presentation

The Drafting workbench allows you to automatically generate engineering drawings from CATIA Part and Assembly documents.



- Engineering drawings generated using this workbench are associative, i. e. any change in the geometry of original Part or Assembly will be reflected in the drawings.
- You can generate associative dimensions on these drawings.
- You can use company standard templates to generate the drawings, and also you can dress-up the drawings with annotations consistent with the company standards.

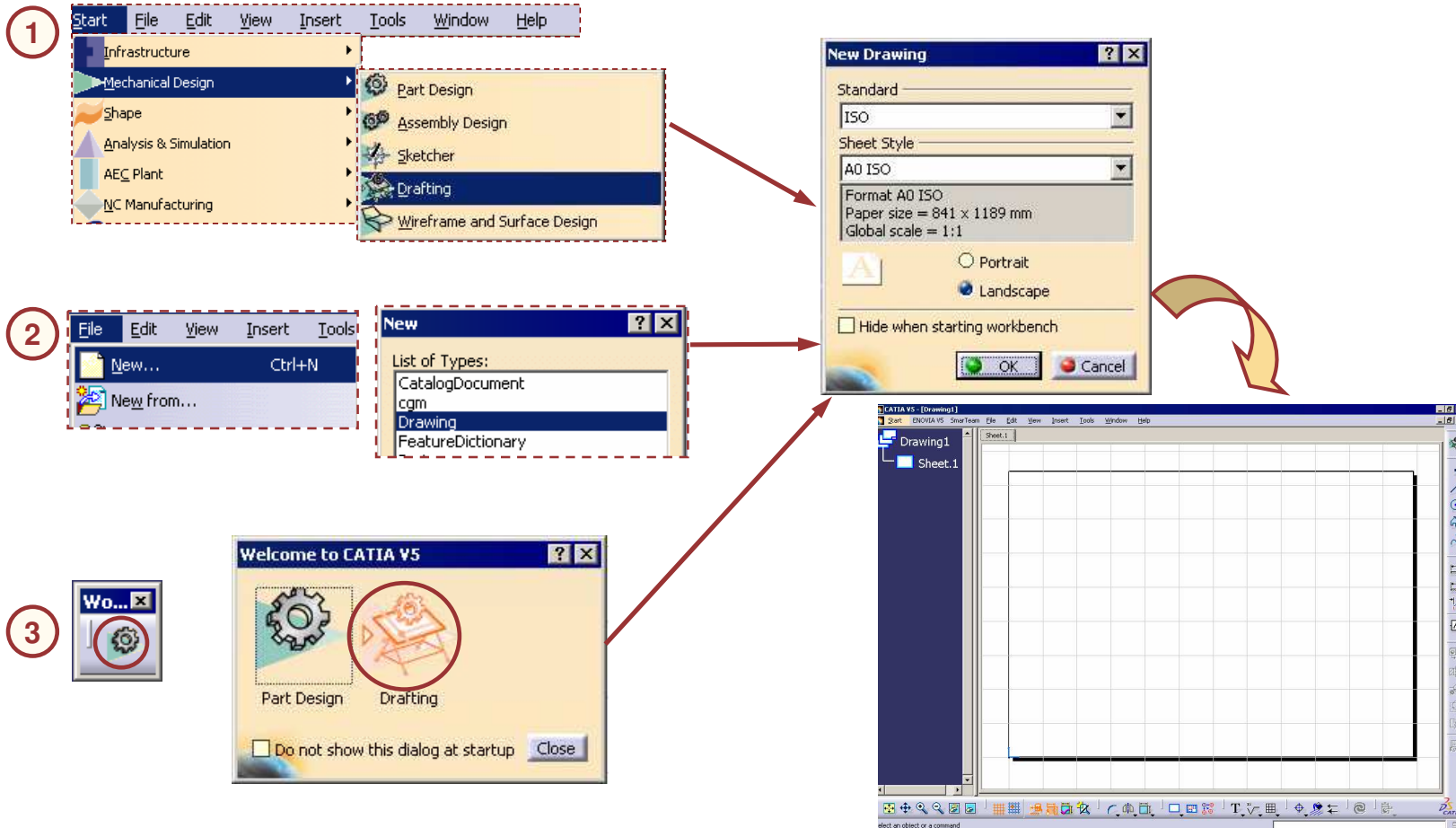
Drafting: General Process



Drafting workbench contains the tools to create views, dimension the views and also the interactive tools to create annotations, company templates.

Accessing the Workbench

You can access the Drafting workbench from 1- Start menu 2- File menu 3- Workbench icon



Drafting Toolbars and Objects

Each toolbar contains objects related to specific tasks. Icons within these toolbars are compressed and can be expanded for accessing additional functionalities.

Dimension properties **Graphic properties** **Text properties**

The image shows the CATIA V5 Drafting environment. A central drawing sheet displays a mechanical part with its front, top, and side views. Surrounding the sheet are several toolbars. Red arrows and numbered circles (1-8) identify specific toolbars:

- 1 - Views:** Located at the top right, containing icons for creating and managing views.
- 2 - Drawing:** Located below the Views toolbar, containing icons for drawing operations.
- 3 - Dimensioning:** Located below the Drawing toolbar, containing icons for dimensioning.
- 4 - Ge... (Geometry):** Located below the Dimensioning toolbar, containing icons for geometry creation and modification.
- 5 - Annotations:** Located at the bottom right, containing icons for text and dimension annotations.
- 6 - Dress-up:** Located at the bottom center, containing icons for modifying the appearance of objects.
- 7 - Geometry Creation:** Located at the bottom left, containing icons for creating new geometry.
- 8 - Geometry Mo... (Geometry Modification):** Located at the bottom left, containing icons for modifying existing geometry.

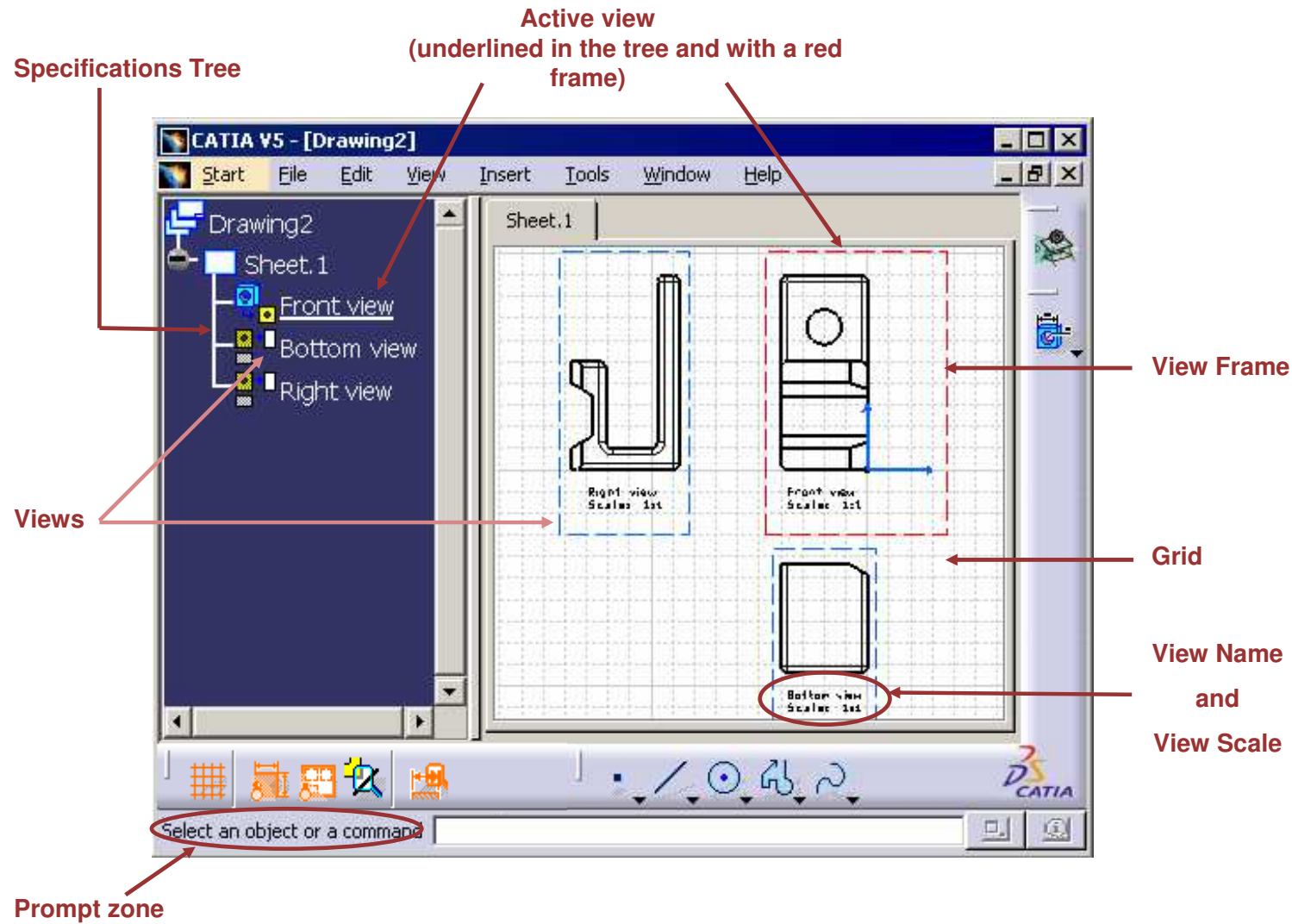
Labels at the top point to the main toolbar area:

- Dimension properties:** Points to the 'no tolerance' dropdown.
- Graphic properties:** Points to the 'None' dropdown.
- Text properties:** Points to the 'Monospaced' dropdown.

On the right, a menu structure is shown:






- Insert
- Tools
- Window
- Help
- Object
 - 1 - Views
 - 2 - Drawing
 - 3 - Dimensioning
 - 4 - Generation
 - 5 - Annotations
 - 6 - Dress-up
 - 7 - Geometry creation
 - 8 - Geometry modification
 - Picture...

Drafting Terminology



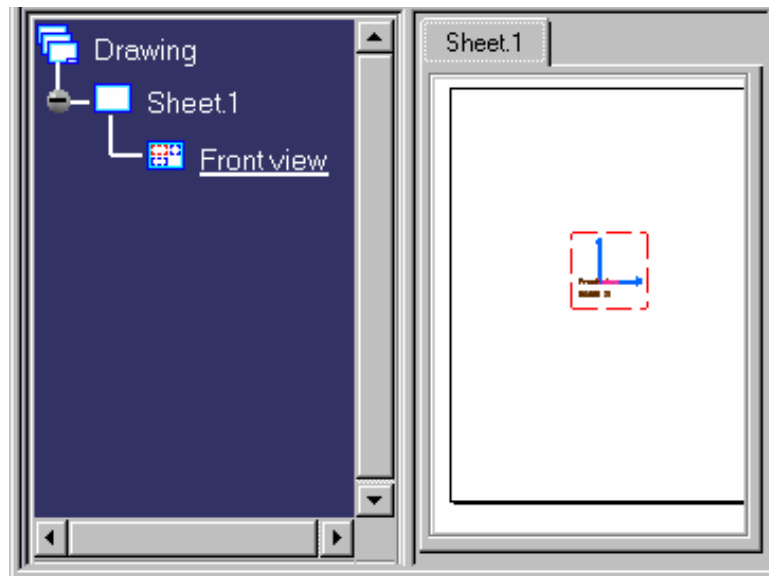
Creating Interactive Views

Drawing views use sketcher tools to create 2D views. Unlike a Generated view, these views have no link to a 3D model.

-  **Starting an Interactive Drawing**
-  **Drawing the Views**
-  **Adding Projection Views**
-  **Methods and Recommendations**
-  **To Sum Up**

Starting an Interactive Drawing

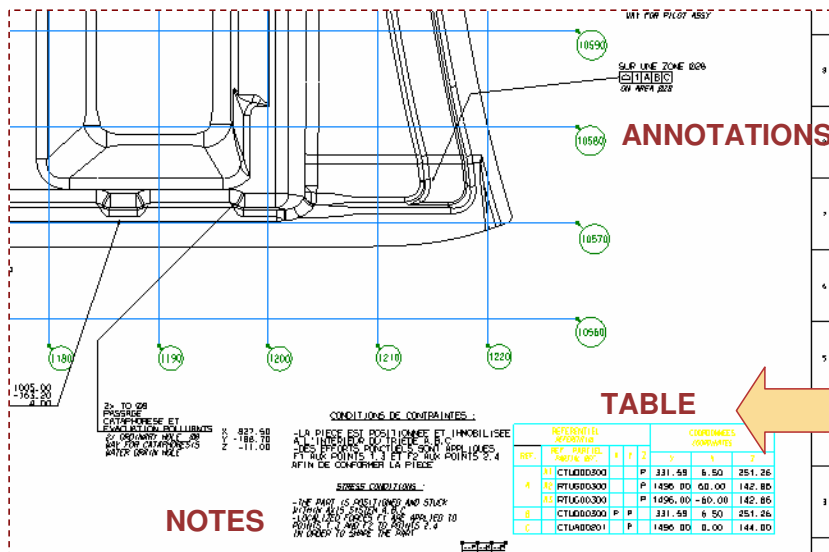
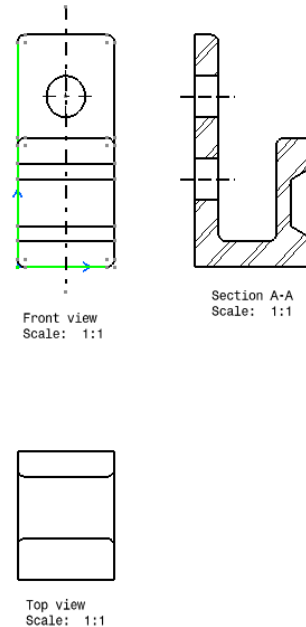
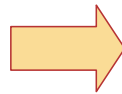
You will learn how to create front view in Interactive Drafting.



Why Interactive Drafting

Interactive Drafting is a drafting system that can be used in a standalone 2D CAD environment. It can be used for traditional drafting purpose i. e. for creating 2D drawings of the components for which you do not have 3D models.

Simple 2D drawings can be created in Interactive drafting without a 3D model. These drawings can be used to express design ideas.



To complete a generative view with interactive elements.

2D interactive functionalities can be used in an advanced production environment for the dress-up and annotation of drawings.

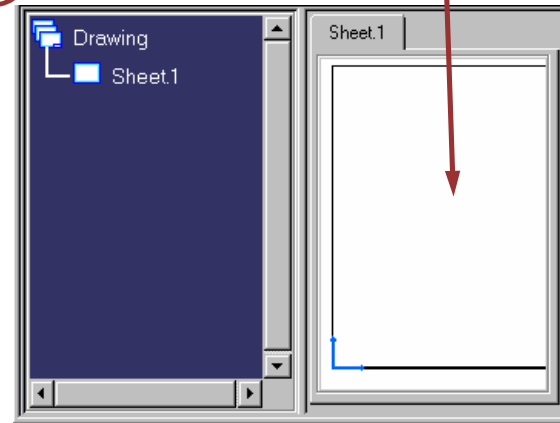
Starting the Front View

When a drawing is created, you need to add a front view before starting to draw lines.

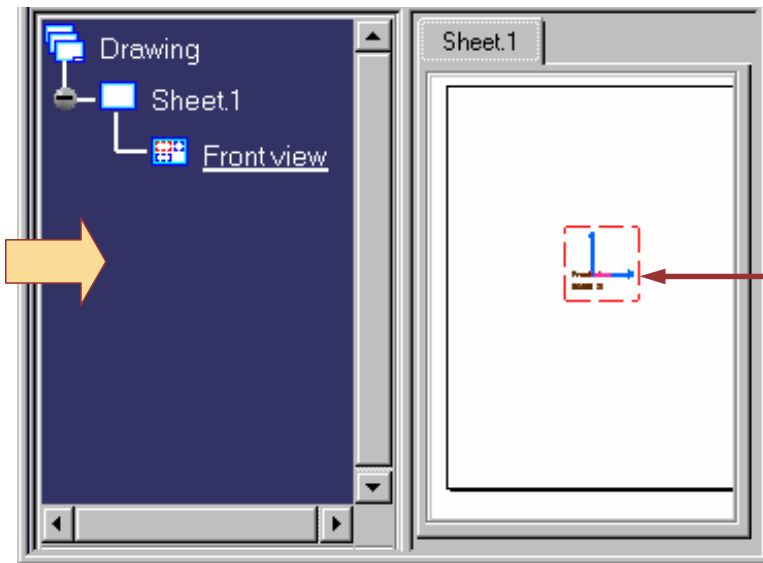
1 Select the New View icon.



2 Place the view by clicking on the sheet



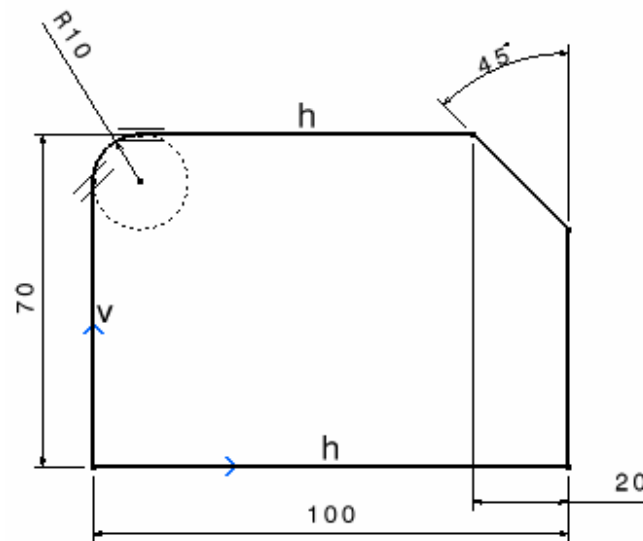
3 The Front view is added in the tree and on the Sheet with a blue HV reference axis and a view frame



View frame : By default, a view frame is displayed and always fits the content of the view and changes with it.

Drawing the Views

You will create basic sketches within the Drafting workbench



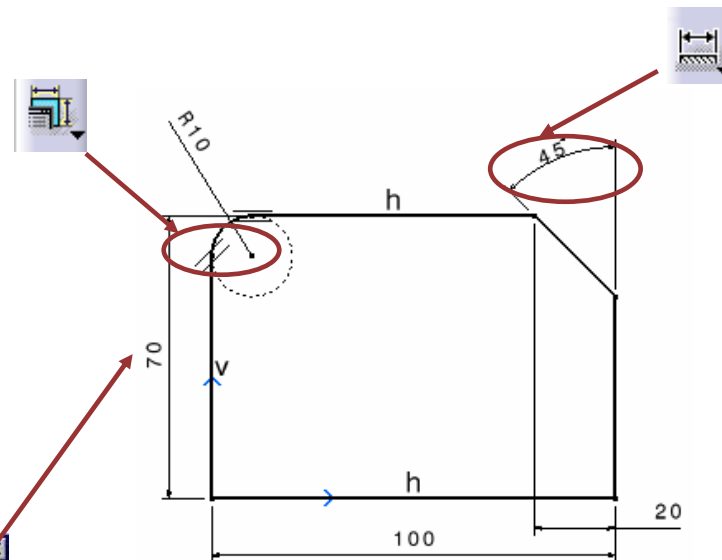
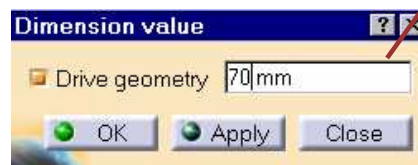
Sketching ...

Sketching in the drafting workbench gives you the ability to create 2D shapes. A sketch is a rough drawing by nature however it is possible to create precise elements.

A typical approach to sketching is made up of three steps:

- Profile Creation
- Creating Sketch Constraints
- Editing Constraints

When first creating a sketch thought should be given to the basic shape of the profile and not precise measurements. Once the profile is created, constraints can be added to help better define the profile. Finally, by editing the applied constraints, the precise profile can be obtained.



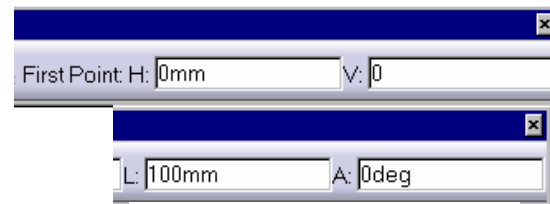
Profile Creation

Sketches drawn with the profile icon help to eliminate the need for the creation of multiple sketched entities such as line, curve and trim. Profiles do not need to be created precisely which speeds up sketching.

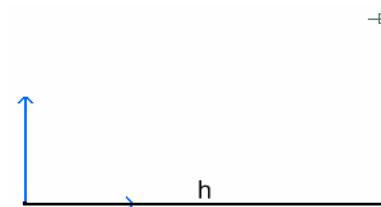
1 Select the Profile icon



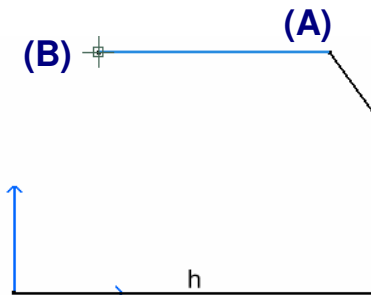
2 Enter values for the First Point, Length and Angle.



3 Enter values L: 50mm & A: 90deg



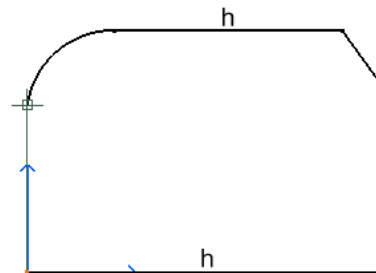
4 Click to create angle (A) and click again to create a horizontal line (B). Blue indicates horizontal.



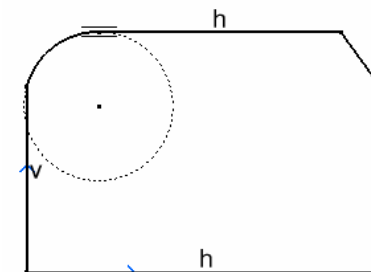
5 Select the Tangent Arc icon



6 Click to create the arc when it is inline with the vertical axis.



Example results :








7 Click on the origin to close the profile.

Student Notes:


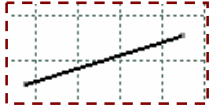

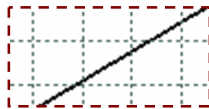

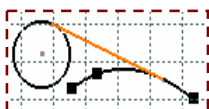

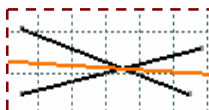


Point Creation

Point types available in the Drafting Workbench:

Type	Icon	Description
Points by Clicking		Create a point just clicking where you want it to be.
Points by Coordinates		Create a point defining its coordinates in the 2D space of the Sketch.
Equidistant Points		Create as many points as you want equidistantly distributed on an existing curve.
Intersection Point		Create the intersection point between two existing curves.
Projection Point		Project an existing point onto an existing curve normally to this curve.

Line Creation


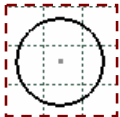

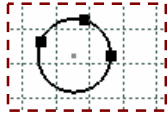

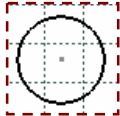

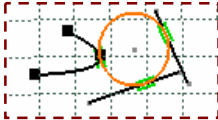
Line types available in the Drafting Workbench:

Type	Icon	Geometry	Description
Line			Create a line by clicking two points to define its extremities.
Infinite Line			Create an infinite line defining its direction by clicking two points.
Bi-tangent Line			Create a line tangent to two existing curves.
Bisecting Line			Create a line bisecting to two existing lines.
Line Normal to Curve			Create a line perpendicular to an element.

Student Notes:




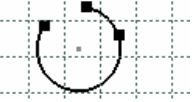



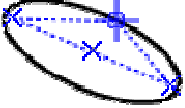
Circle Creation (1/2)

Circle and Ellipse types available in the Drafting Workbench:

Type	Icon	Geometry	Description
Circle			Create a circle defining its center and its radius by clicking.
Three Points Circle			Create a point passing by three points.
Circle by Coordinates			Create a circle giving the coordinates of its center and its radius.
Tri-tangent Circle			Create a circle tangent to three existing curves.


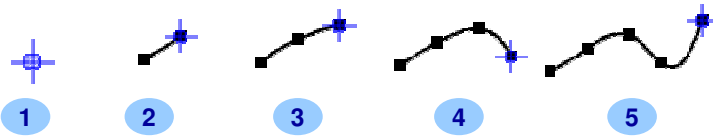

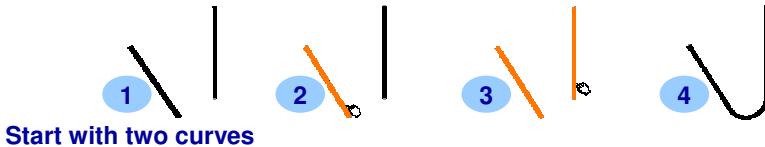

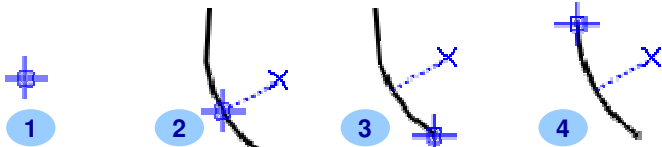
Circle Creation (2/2)

Circle and Ellipse types available in the Drafting Workbench:

Type	Icon	Geometry	Description
Three Points Arc			Create an arc passing by three points, relimited by the first and the last selected points.
Three points arc starting with limits			Create an arc passing by three points, relimited by the first and the second selected points.
Arc			Create an arc defining its center and the two limit points.
Ellipse			Create an ellipse by selecting its center point, and defining its dimensions by clicking two points.


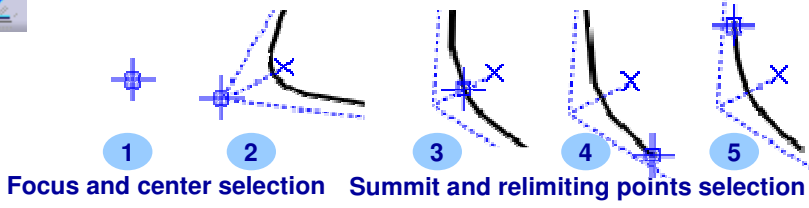
Curve Creation (1/3)

Curve types available in the Drafting Workbench:

Type	Icon	Geometry	Description
Spline			Create a spline curve passing by as many points as you want. End the spline by double-clicking.
Connect		 <p>Start with two curves</p>	Create an arc connecting two curves.
Parabola by Focus		 <p>Focus and summit selection Relimiting points selection</p>	Create a Parabola selecting: <ul style="list-style-type: none"> • The focus point • The summit • The relimiting points

Curve Creation (2/3)

Curve types available in the Drafting Workbench:

Type	Icon	Geometry	Description
Hyperbola by focus		 <p>1 Focus and center selection</p> <p>2 Summit and relimiting points selection</p> <p>3</p> <p>4</p> <p>5</p>	<p>Hyperbola by focus</p> <ul style="list-style-type: none">• Create an Hyperbola selecting:• The focus and center points• The summit• The relimiting points

Student Notes:

Curve Creation (3/3)

- Conic: Conics can be created in several ways. After the ICON is selected, several conic tools appear in the toolbar shown.

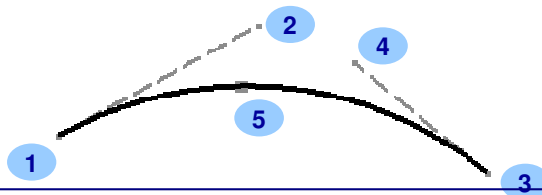


Icon

Geometry

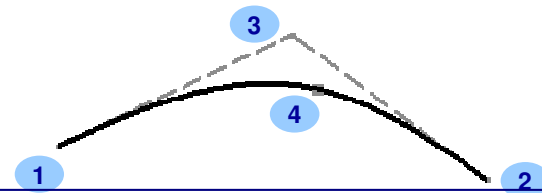
Description

Two point conic
with end tangents



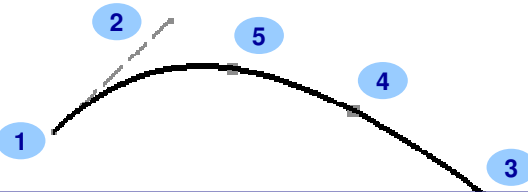
Select an end point, define its tangency, select the other end point and its tangency, select a mid point

Two point conic
with tangent
intersection



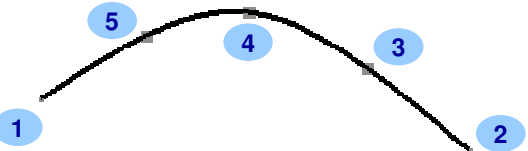
Select two end points, select a point which defines the tangent intersection, select a mid point.

Four point conic
with one tangent



Select the two end points, select two mid points, define one tangency at first or last point.


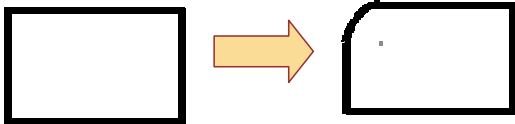

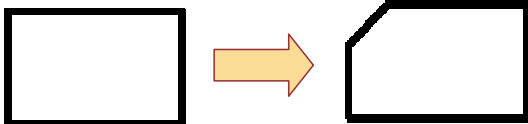

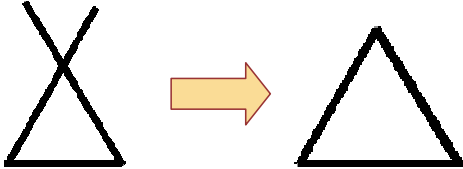


Five point conic



Select two end points, select three mid points.

Relimitation Elements (1/2)


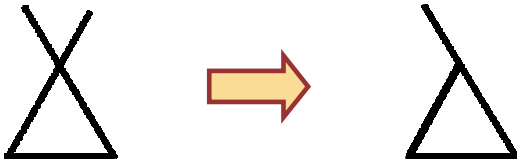

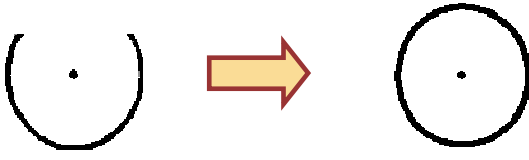

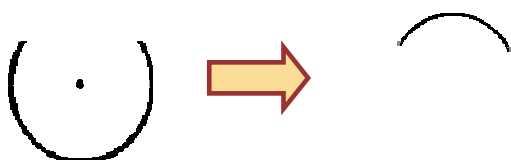
Relimitation types available in the Drafting Workbench:

Type	Icon	Geometry	Description
Corner			Trims two elements to create a round corner. Option available to keep existing edges or to trim them.
Chamfer			Creates a beveled corner at the point where two selected edges meet. Option available to keep existing edges or to trim them.
Trim			Trims or extends elements to form a closed corner. Select elements on the side of the intersection you want to keep.
Break			Divides an element at selected point. Creates two elements from one.

Student Notes:

Relimitation Elements (2/2)

Relimitation types available in the Drafting Workbench:

Type	Icon	Geometry	Description
Quick Trim			Trims the selected element to the intersection of another element.
Close			Connects the endpoints of the element to close it.
Complement			Replaces the existing element with an element that would close the original.

Student Notes:

Transformation Elements (1/2)


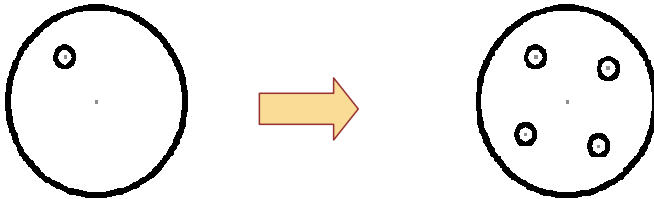

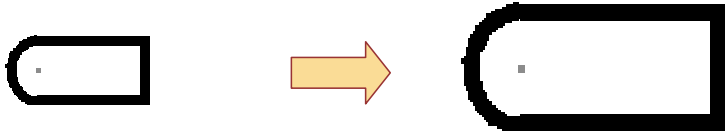

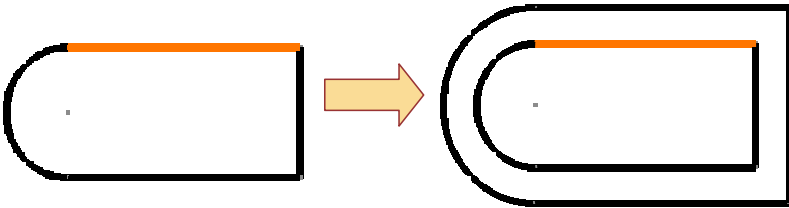
Transformation types available in the Drafting Workbench:

Type	Icon	Geometry	Description
Mirror			Creates mirror image of selected elements about a selected line or axis
Translate			Moves selected elements in the X and/or Y direction. Can also be set to duplicate mode.
Symmetry			Moves selected elements about a selected line or axis

Student Notes:

Transformation Elements (2/2)

Transformation types available in the Drafting Workbench:

Type	Icon	Geometry	Description
Rotate			Rotates selected elements. Can also be set to duplicate mode.
Scale			Enlarges or shrinks selected elements by a ratio of the original.
Offset			Offsets just the selected element. Can also be set to offset all elements connected to the selected element.

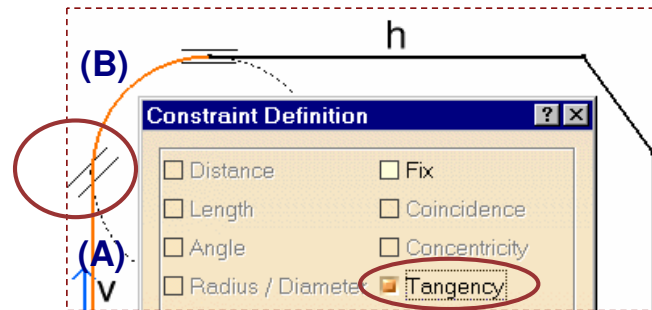
Creating Sketch Constraints

A sketch can be constrained using geometrical constraints, such as tangency, to establish geometrical relationship.

- 1 Select the elements to constrain.
- 2 Select the "Constraint Defined in Dialog Box" icon. Select the Type of geometrical constraint in Constraint Definition dialog box.



Select the vertical line A and arc B



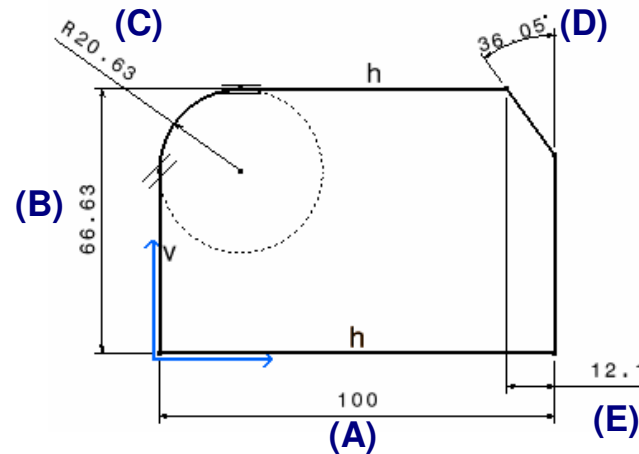
Select Tangency.

Or the sketch can be constrained by creating controlling dimensions which can be edited.

- 3 Select Dimensions icon and the elements to constrain and indicate the location.



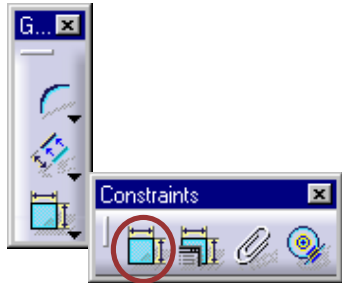
- (A) Length of lower horizontal line
- (B) Distance between horizontal lines
- (C) Radius
- (D) Angle of chamfer relative to vertical
- (E) Distance of chamfer width relative to right side of profile and chamfer point



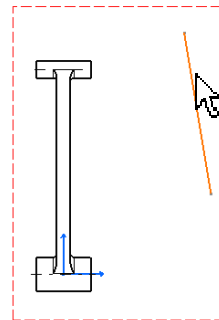
Creating Explicit Geometrical Constraints

Explicit Geometric Constraints can be created between sketched elements or between sketched and generated geometry.

1 Select the Geometric Constraint icon.



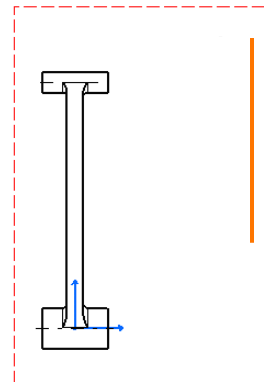
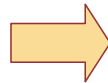
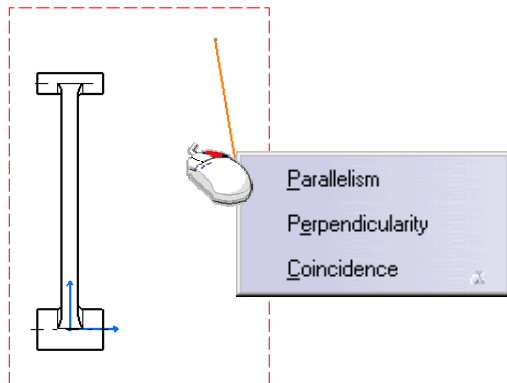
2 Select the elements to Constrain.



(Create)
Horizontal

A predetermined Geometric Constraint will be displayed. If this is the desired constraint, select the drawing background to confirm.

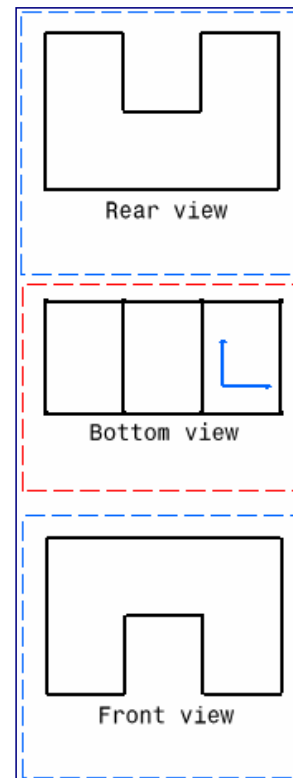
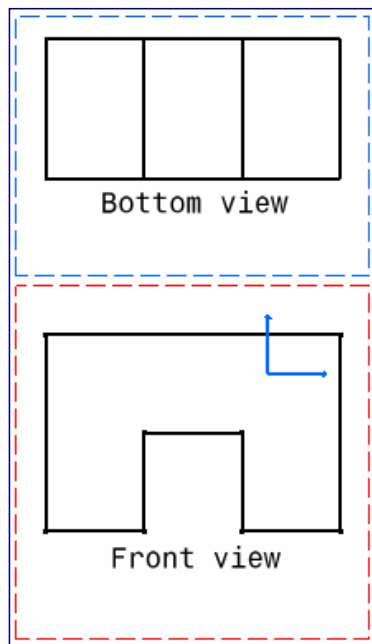
3 Using the contextual menu, select the Geometric Constraint.



It is impossible to create a constraint between sketched and generated geometry through the Constraints Definition dialog box. Use this method to create a constraint between sketched and generated geometry.

Adding Projection Views

You will create additional projection views on a drawing.

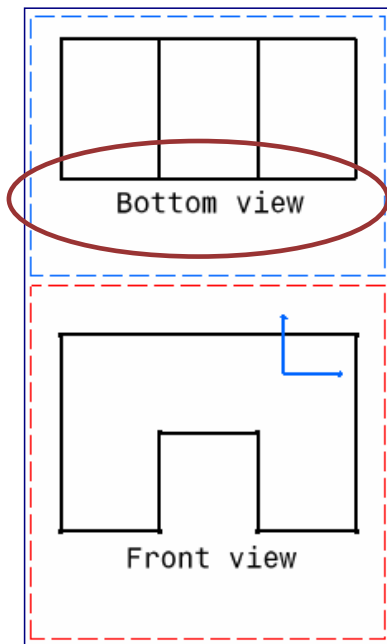


What is the General Method to Add Views ?

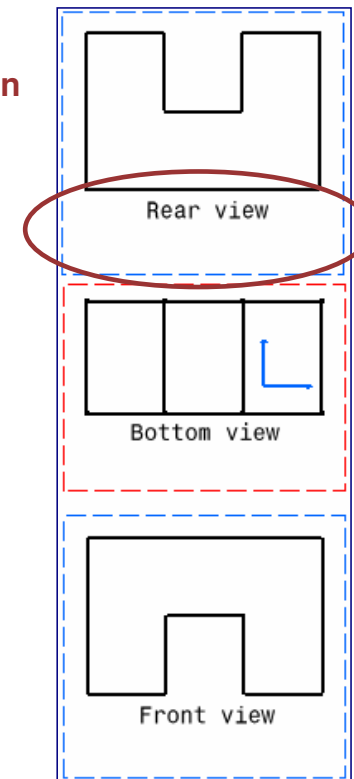
Added Views are usually created as projections of the Front view. The New View icon is used to create projection views one at a time. When an existing view is active, added views are created as projections of that view based on 1st or 3rd Angle Projection.

Any view that is made active becomes the primary view, and added views that are created become a projection of this primary view.

Projection from a Front View is a Bottom View based on 1st Angle Projection



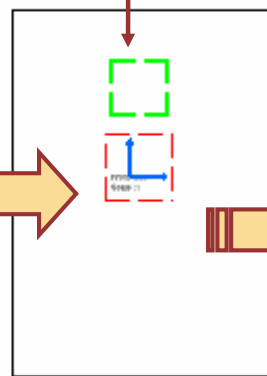
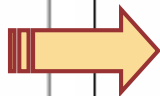
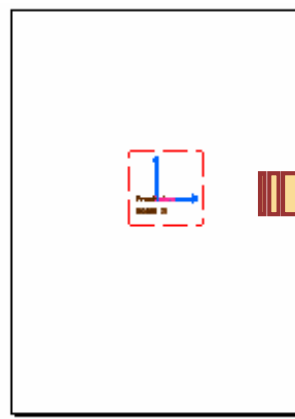
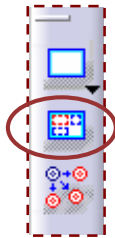
Projection from a Bottom View Results in a Rear View based on 1st Angle Projection



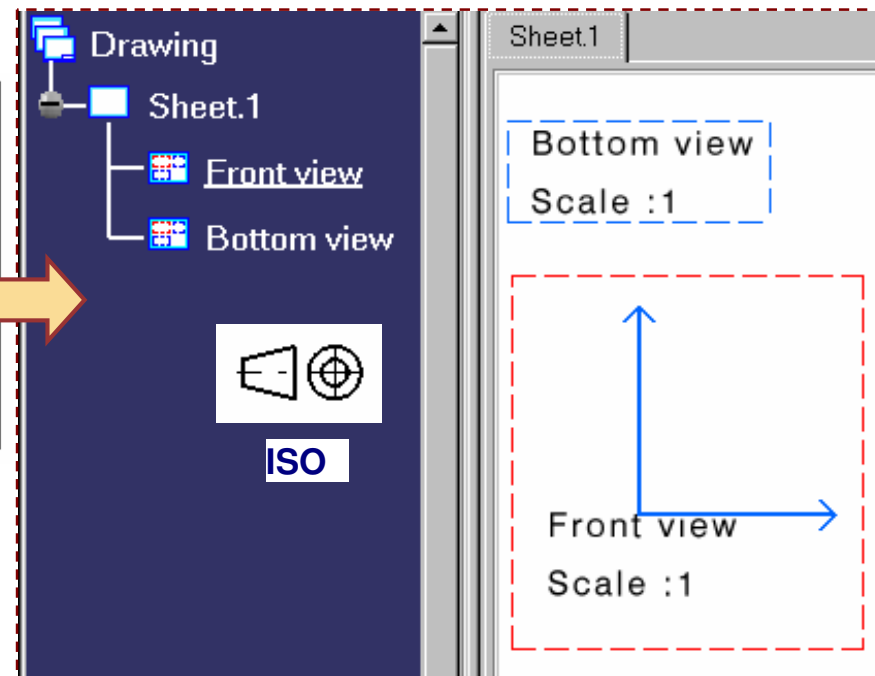
Adding Views

The views added on the drawing are projected from the active view in accordance with the selected standard

- 1 Select the new view icon
- 2 Place the view by positioning the cursor according to the desired View; for example:
 - in ISO, above the Front view will give a Bottom view
 - in ANSI, above the Front view will give a Top view



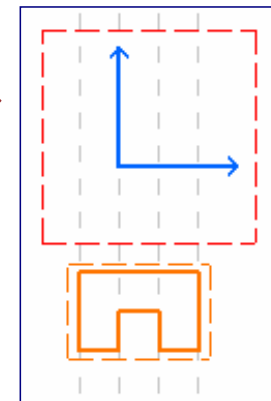
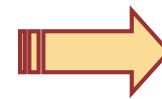
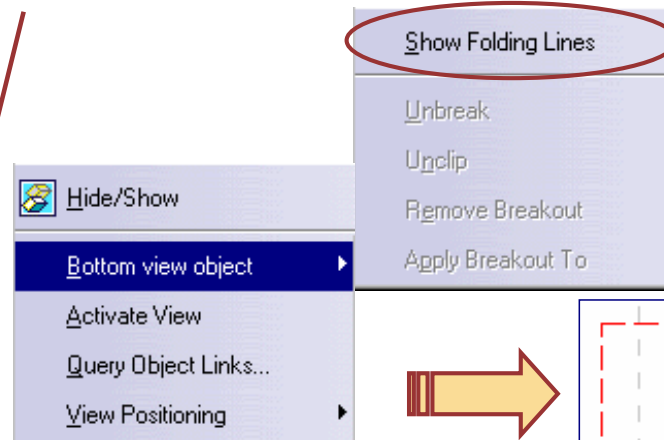
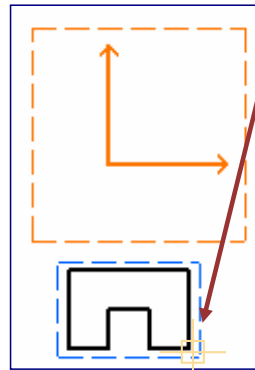
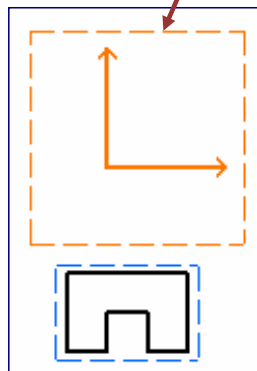
- 3 Select on sheet to create the view



Construction Lines

To easily draw the lines in the views, you can use the construction lines. Construction lines are projections of the lines of one view into the active view and can be used to draw lines.

- 1 Activate the view in which you want to receive the construction lines by double-clicking on its frame or on its name in the specification tree
- 2 Place cursor on the view frame from which you want to get the construction lines. Then, using the third mouse button, select **Show Folding Lines** from the contextual menu



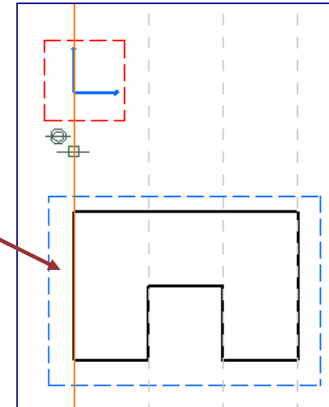
Create Geometry using Construction Lines

When construction geometry is present, Auto Detection recognizes it to anchor newly created geometry.

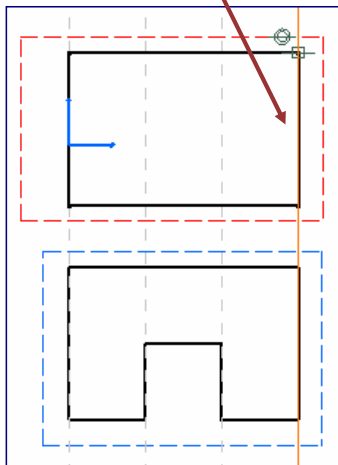
- 1 Select the Rectangle icon.



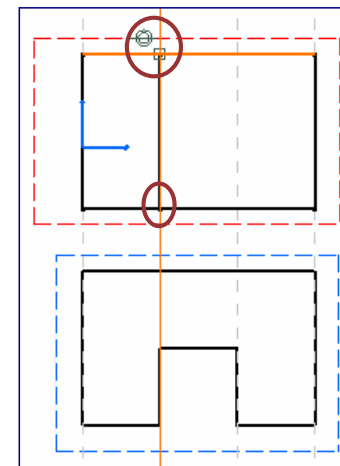
- 2 Select the left construction line using auto detection to begin the rectangle.



- 3 Select the right construction line using auto detection to complete the rectangle.



- 4 Select the line icon. Then use the intersections of the construction line and the rectangle to complete the line.

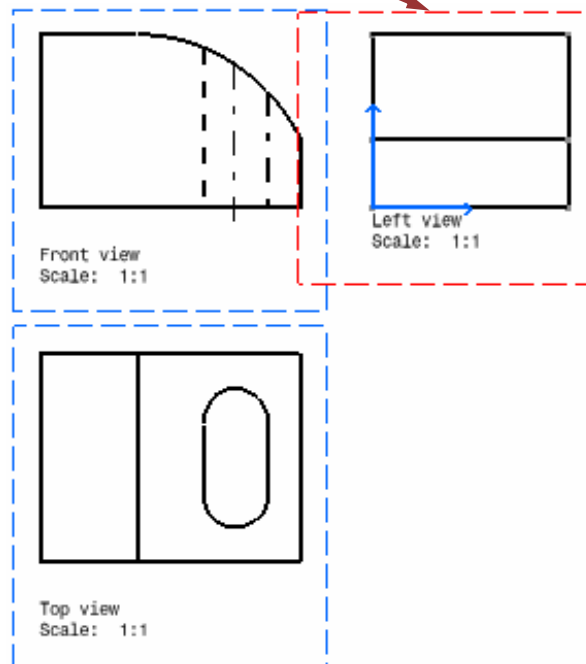


Creating a Multiple View Projection (1/2)

This function allows you to generate geometry in a view by projecting geometry from previously defined views. The projected geometry retains the same attributes it had in the original multi-view.

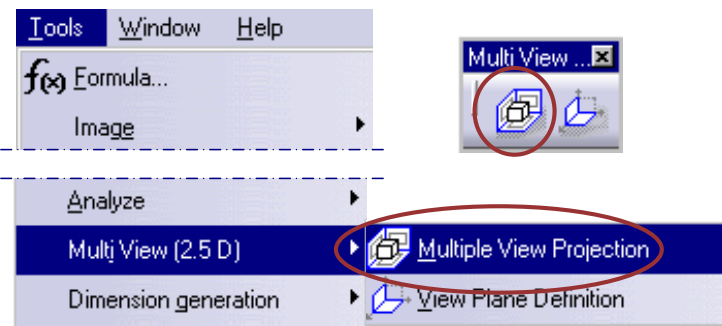
1

Activate the view where you want the projection to appear.



2

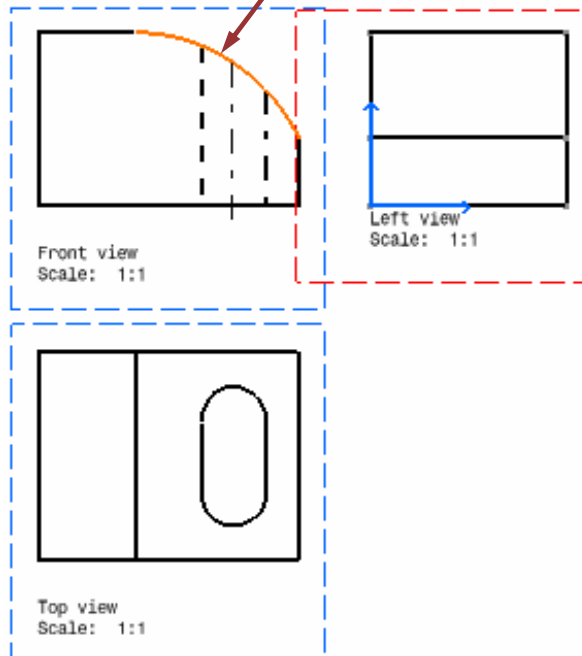
Select the Multiple View Projection icon.



Creating a Multiple View Projection (2/2)

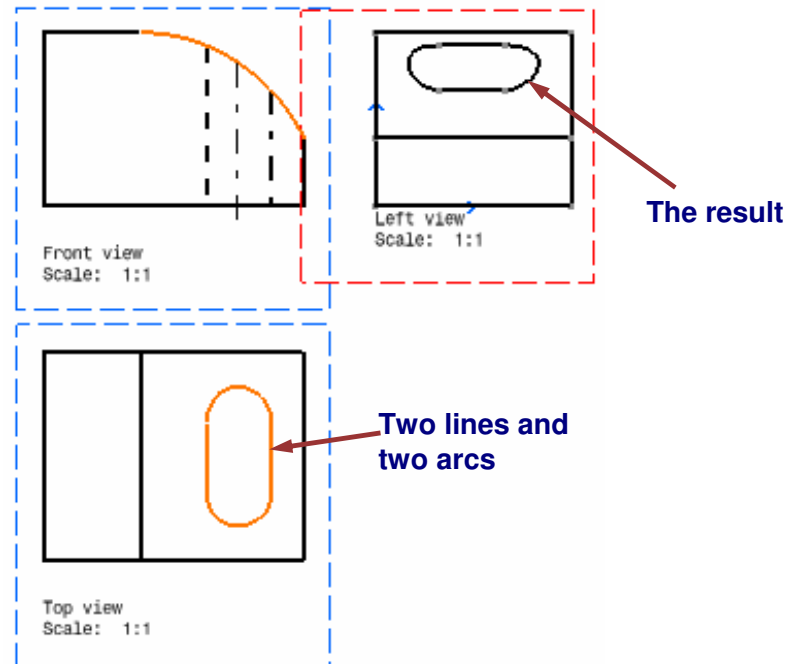
3

Select object defining target plane or surface



4

Select one by one the geometry elements (four) which compose the feature (elongated hole). The result appears automatically

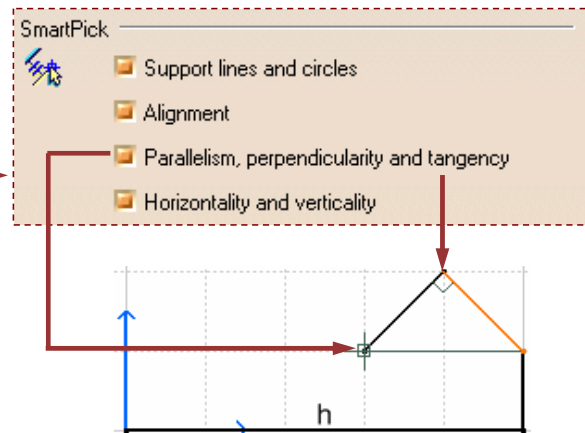
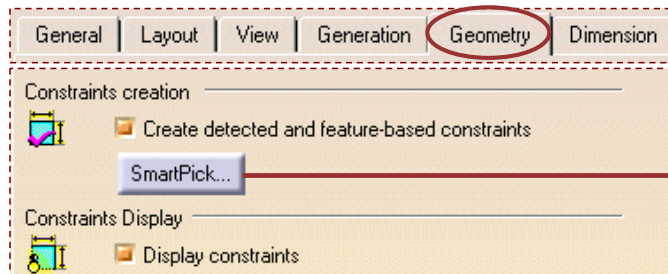
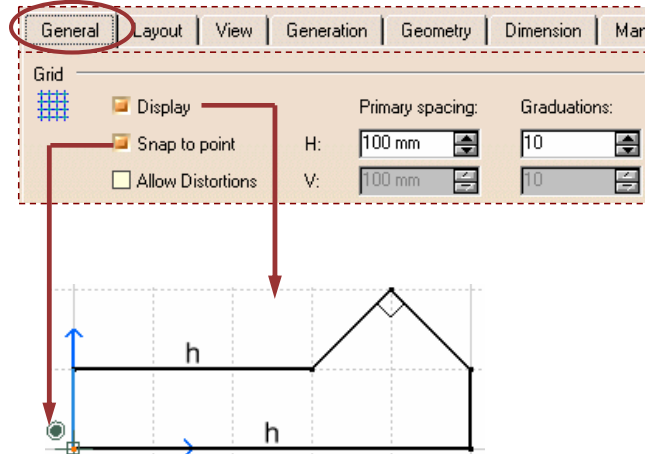
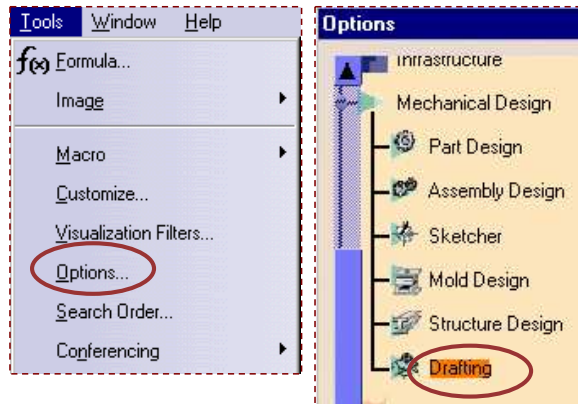


Methods and Recommendations

You will see some suggestions and techniques you can use while creating interactive views. Also, you will learn how to complete the generated drawing using interactive elements.

Sketching Aids (1/2)

There are several sketching aids available in menu Tools >Options > Drafting in the General and Geometry tabs.

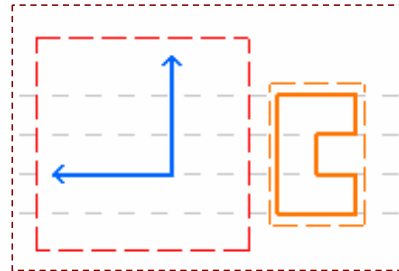


Using these aids will minimize the need to Edit Constraints to make the sketch precise.

Sketching Aids (2/2)

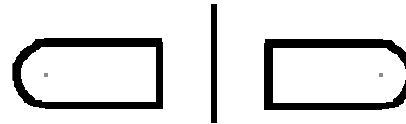
Make use of construction lines and transformation tools to increase your efficiency.

Use construction lines to ensure projection views are in line with the main view.

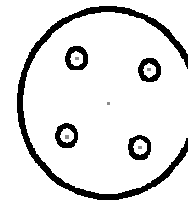


Construction lines.

Take advantage of the transformation tools available to help increase your efficiency while constructing similar or identical geometry.

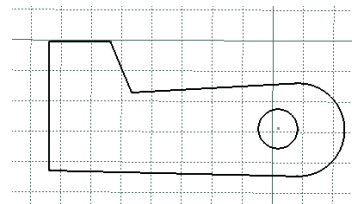


Mirror

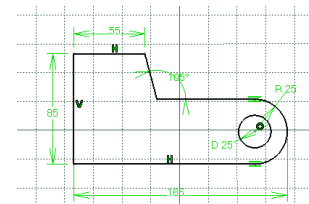


Rotate

Create the approximate profile first and then use constraints to make the sketch more precise.



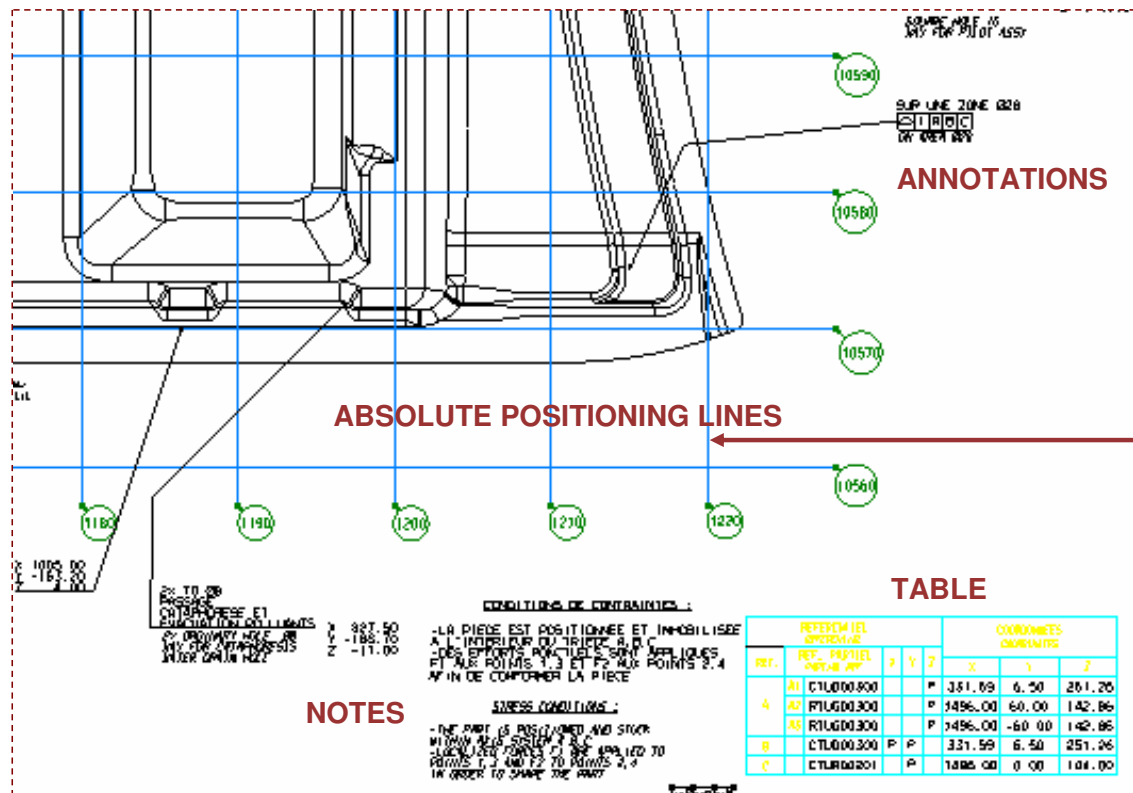
Initial Sketch



Final Sketch

Completing a Generative View

2D interactive functionalities can be used in an advanced production environment for the dress-up and annotation of drawings.









To Sum Up

You have learned:

- How to create interactive views.
- How to use sketcher tools to create geometry.
- Remember, a typical approach to sketching consists of three steps:
 - ◆ Creating Profiles
 - ◆ Creating sketch constraints
 - ◆ Editing constraints

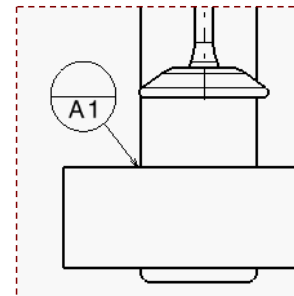
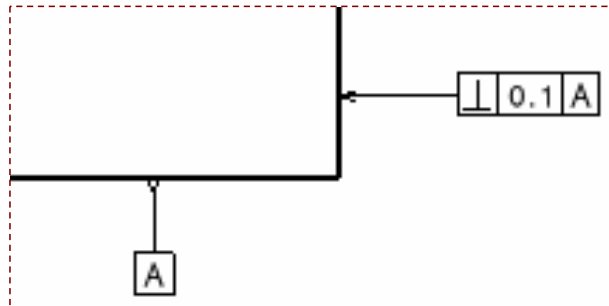
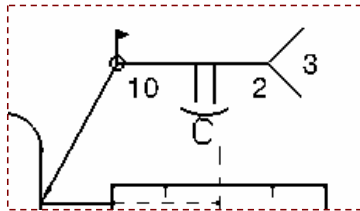
Dressing Up the Layout

To fully detail a drawing it is often necessary to add additional information such as geometric tolerancing, notes, or tables. In this lesson we will look at how to create these.

-  **Adding Annotations and Text**
-  **Creating Tables**
-  **Graphic Dress-up**
-  **Instantiating Elements From a Catalog**
-  **Methods and Recommendations**
-  **To Sum Up**


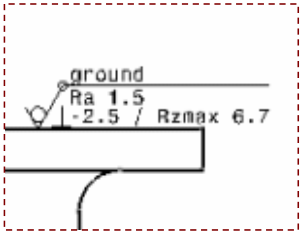

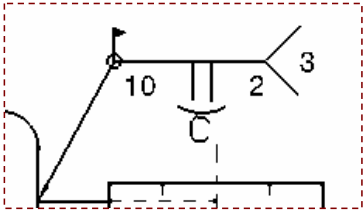

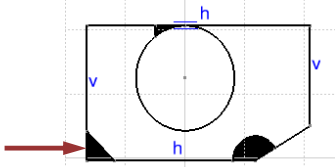
Adding Annotations and Text

You will learn how to add symbols geometrical tolerances and text to the drawing.



Symbols


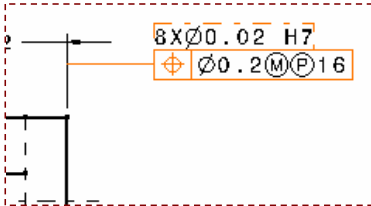

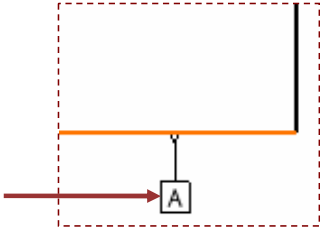

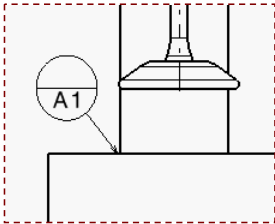
You can add symbols to better define the specifications of the part.

Type	Icon	Geometry	Description
Roughness Symbol			Creates a standard roughness symbol including roughness value, direction of lay, production method etc.
Welding Symbol			Creates a weld symbol including complimentary symbols and weld finish symbols.
Weld			Creates a geometry weld symbol between two elements.

Student Notes:

Geometrical Tolerances

You can add symbols to better define the specifications of the part

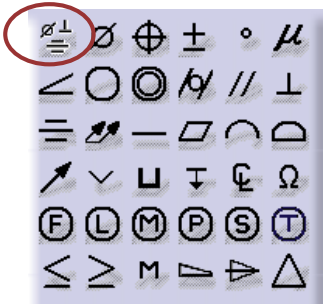
Type	Icon	Geometry	Description
Geometrical Tolerance			Creates standard geometrical tolerances including symbols and references.
Datum Feature			Annotates a Datum Feature.
Datum Target			Annotates a datum target.

Creating Text



Text can be created with or without leaders and can have many properties to customize the font and frame style (shown on the right).

Special symbols can also be inserted into the Text as needed.



Icon	Name	Description
	Font Name	Changes the style of the text.
	Font Size	Changes the size of the text.
	Bold	Changes the weight of the text. Toggles between normal and heavy (bold).
	Italic	Changes the angle of the text. Toggles between normal and slanted (italic).
	Underline	Adds a line under the text.
	Strikethru	Adds a line through the center of the text.
	Overline	Adds a line above the text.
	Superscript	Raises the text above the normal text line.
	Subscript	Lowers the text below the normal text line.
	Left Justify	Aligns multiple lines of text to the left edge of the text frame.
	Center Justify	Centers multiple lines of text within the text frame.
	Right Justify	Aligns multiple lines of text to the right edge of the text frame.
	Anchor point	Changes the position of the point that connects the text to the drawing or to an element. Choices are as follows: <ul style="list-style-type: none">Along the top of the text: left, center or rightAlong the vertical center of text: left, center or rightAlong the bottom of the text: left, center or right
	Frame	Draws a single-line frame around the text. A variety of different shapes is available.

Text without Leader


Text without leaders with a customize symbol and frame style.

- 1 Select the Text icon to create text without a leader



- 2 Select the text anchor (starting) point



- 3 Key the desired text with the appropriate framing or embedded symbols; for example:
 (a) Key "HOLE" then *spacebar*
 (b) Select  from the Symbol icon then press


spacebar

(c) Key "SIZE 3" then press *ENTER* to start new line

(d) Key "SEE NOTE" then press *spacebar*

(e) Select  from the Frame icon



(f) Key "1".

HOLE  SIZE 3
 SEE NOTE 1



- 4 Select anywhere to end the text.

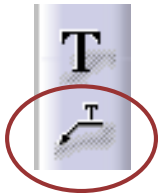
Example results:

HOLE  SIZE 3
 SEE NOTE 1

Text with Leader

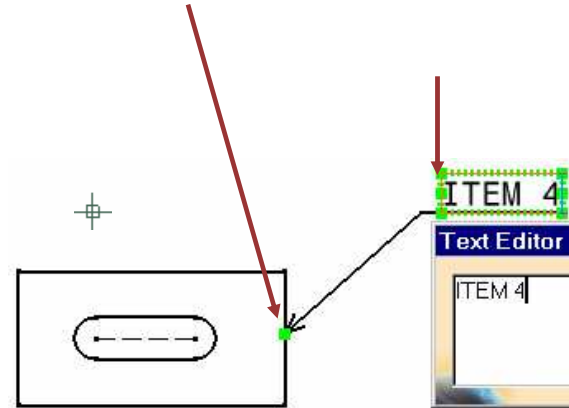
Here is how to create text with leaders:

- 1 Select the Text with Leader icon



- 3 Select the text anchor point

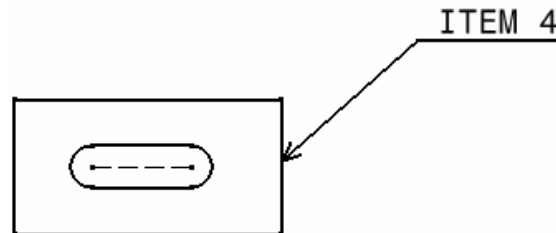
- 2 Select an anchor point for the arrow head



- 4 Key desired text (ITEM 4)

- 5 Click anywhere to end the leader text

Example results:



Non-Associative Balloons

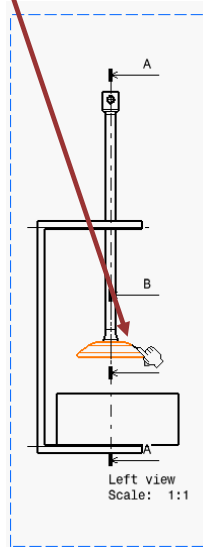
1

Select the Balloon icon



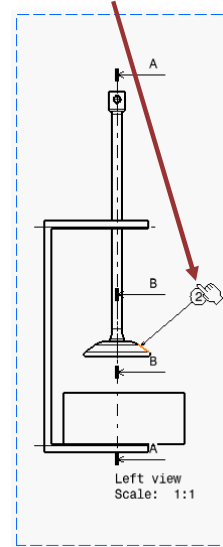
2

Select an anchor point for the arrow head



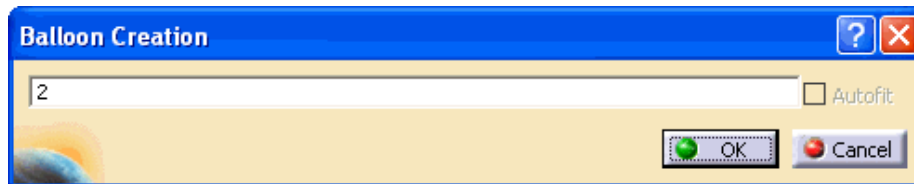
3

Select location for the balloon

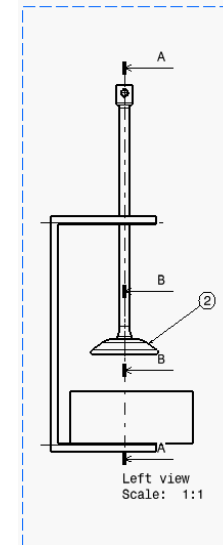


4

Key desired text (2)



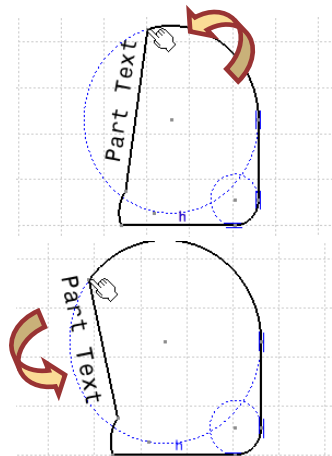
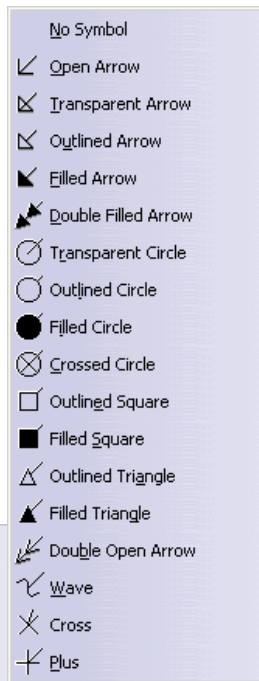
Example results:



Editing Text

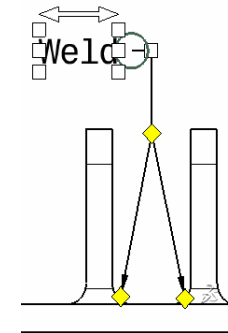
Text can be modified to change leader style the content, style or the orientation of the text.

Change arrow style



Change text orientation

Add breakpoints and create multiple leaders



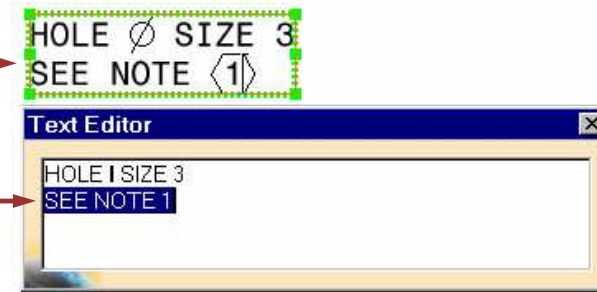
Change Text font or justification



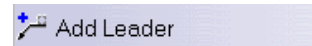
Text Manipulation

Text can be manipulated in the following ways: Edited, Repositioned and/or Leaders added or removed

- 1 Double click text to open the Text Editor.
- 2 Swipe the cursor across the text (second line) within the Text Editor window and edit or delete.
- 3 Select any part of a note and drag to reposition it

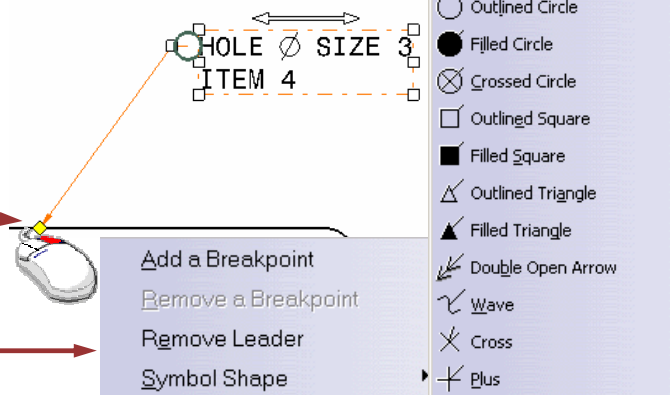


- 4 To add a leader; Right click the note and select:
Indicate or select an element to anchor the arrow head.



This process can be repeated to create multiple leaders.

- 5 Select the anchor point of a leader arrow and drag it to reposition it
- 6 Right click on the anchor point of the leader arrow, select the option from the contextual menu

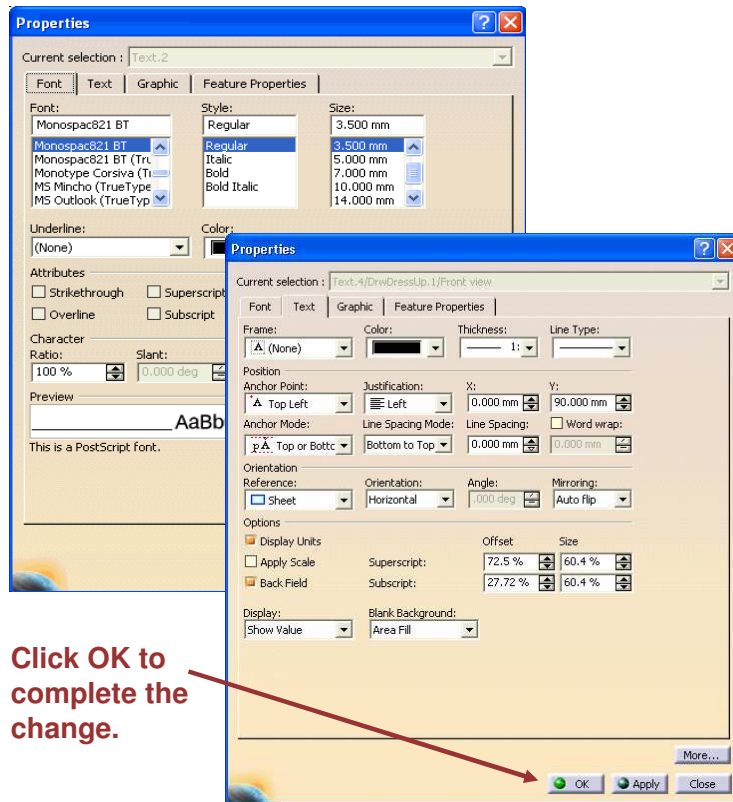


Text Properties

Text font and justification can be modified through the properties dialog box or the Text Properties toolbar.

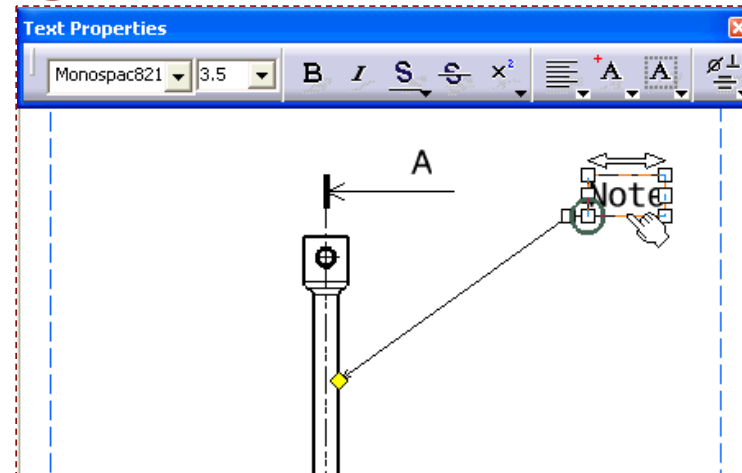
1a Right click on the note and select Properties from the contextual menu.

2a Make necessary changes.



3 Click OK to complete the change.

1b Highlight the note.



2b Use the Text Properties toolbar to make changes.



Text with Breakpoints and Multiple Leaders

Here is how to create text with multiple leaders and breakpoints.

Add Breakpoint:

- 1 Select the Text with Leader
- 2 Using the contextual menu, Select the anchor point (yellow symbol) and Add a Breakpoint item.
- 3 Select the breakpoint and drag to desired location

Add Leader:

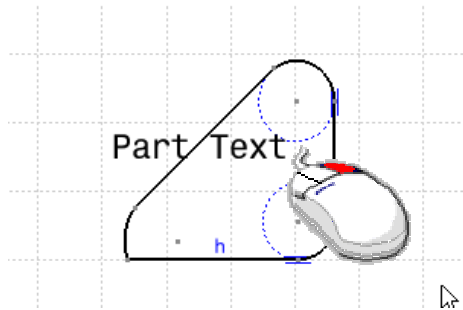
- 1 Using the contextual menu, select a breakpoint and the Add an Extremity item.
- 2 Select an element or location to end the leader

Example result:

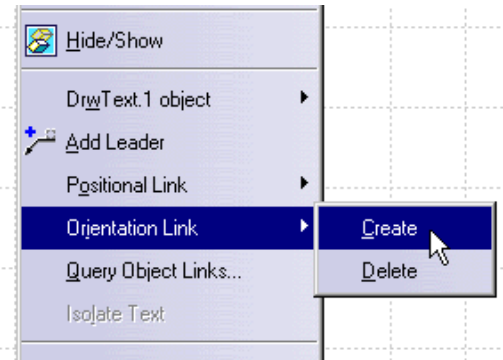
Associated Text

Define a link to 2D geometry to determine the orientation of text

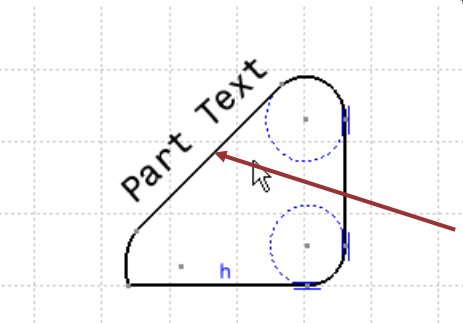
1 Right click on the text



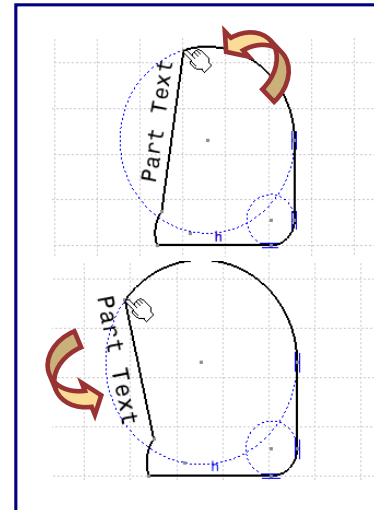
2 Select option Orientation Link and Create



3 Select orientation element



As the element rotates, the text will flip to avoid being backwards or upside down unless the text property Auto Flip is disabled



Creating Tables

You will add a Table to a drawing and modify it

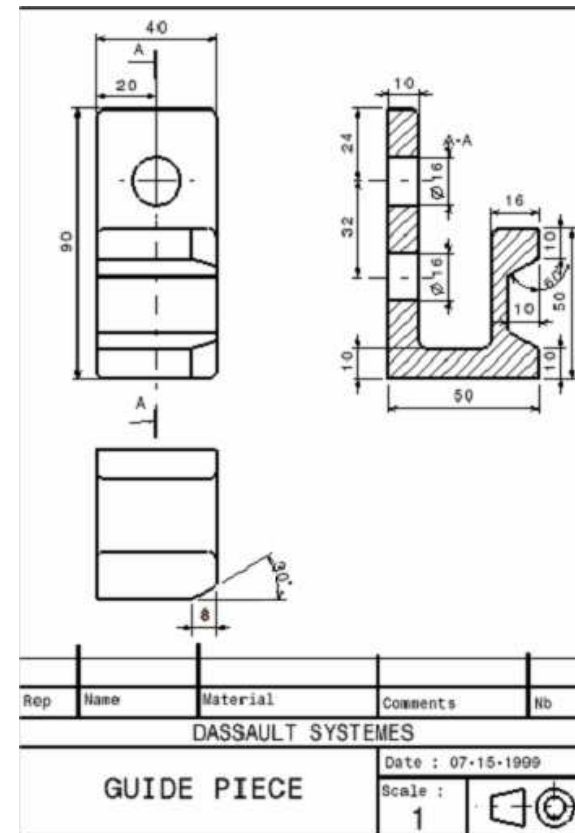
	X	Y
A	0.25	0.25
B	0.50	0.35
C	0.75	0.45
D	1.00	0.55
E	1.25	0.65
F	1.50	0.75
G	1.75	0.85
H	1.80	0.90

	X	Y
A	0.25	0.25
B	0.50	0.35
C	0.75	0.45

	X	Y
D	1.00	0.55
E	1.25	0.65
F	1.50	0.75

	X	Y
G	1.75	0.85
H	1.80	0.90

Bill of Material		
Quantity	Part Number	Buyer
1	PN100-10	Mwho
2	FN1200	Dname
1	PN300-20A	
2	PN200-F	Mfirst
2	FN60-30-21-5	Iamgood
1	PN-DF-120	Ataboy
4	FN1600-N	Bsmart
4	FN1600-W	Bsmart

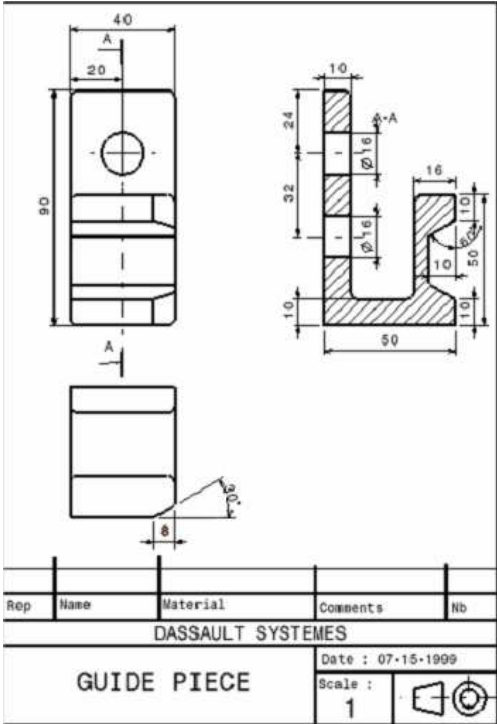


Student Notes:

Tables

Tables are used to capture drawing information in tabular form. Typically tables are used to handle title block information or bill of materials.

Tables can be created each time or they can be created in another program saved as a CSV file and imported into one or many drawings.



Bill of Material		
Quantity	Part Number	Buyer
1	PN100-10	Mwho
2	FN1200	Dname
1	PN300-20A	
2	PN200-F	Mfirst
2	FN60-30-21-5	Iamgood
1	PN-DF-120	Ataboy
4	FN1600-N	Bsmart
4	FN1600-W	Bsmart

Creating a Table

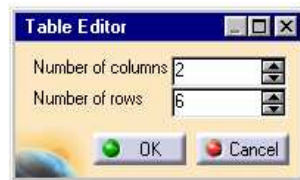
To create a Table of data for Bill of Material list, Revision Notes, or etc...

1 Select the key Table icon



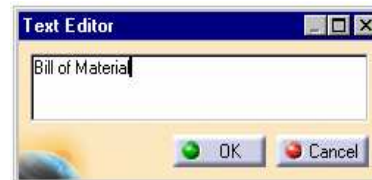
2 Select the Table location

3 Key the number of columns and rows for the Table



4 Double click the Table to select it.

5 Double click a Table cell to edit it.



Bill of Material	



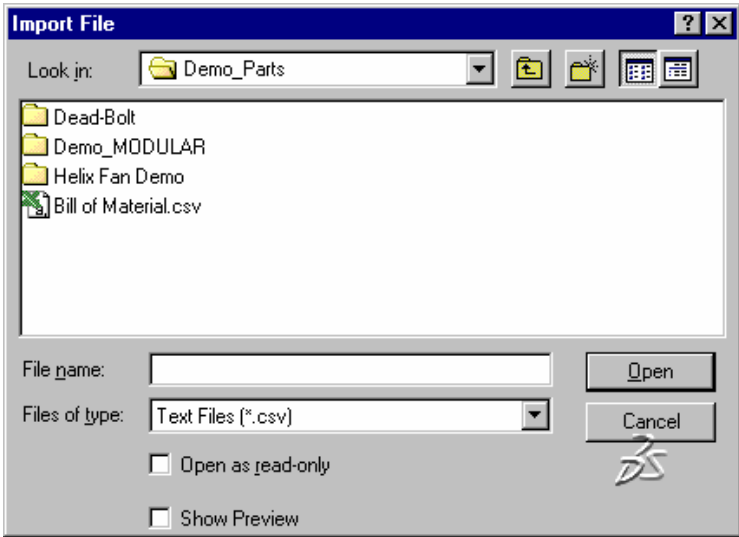
Continue step 5 to fill in all necessary cells.

Creating a Table from a csv file

To create a Table of data from a csv file for Bill of Material list, Revision Notes, or etc...

1 Select the key Table icon 

2 Select the csv file



3 Select a location for the Table

Bill of Material		
Quantity	Part Number	Buyer
1	PN100-10	Mwho
2	FN1200	Dname
1	PN300-20A	
2	PN200-F	Mfirst
2	FN60-30-21-5	Iamgood
1	PN-DF-120	Ataboy
4	FN1600-N	Bsmart
4	FN1600-W	Bsmart

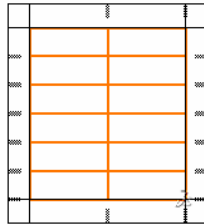


A CSV file is a text file that is comma delimited.
To create a CSV file: save the file created in another program as a .csv file.

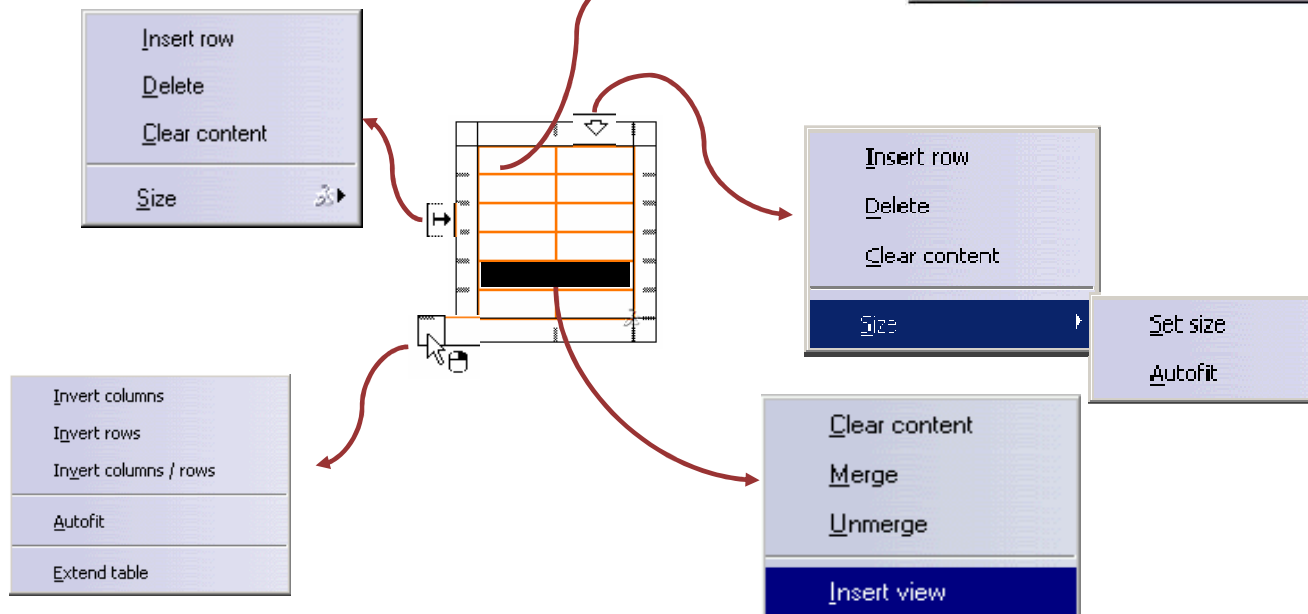
Editing a Table

To edit a Table element to add or delete rows and columns, resize the rows or columns, and invert the rows or columns.

- 1 Double click the Table to select it



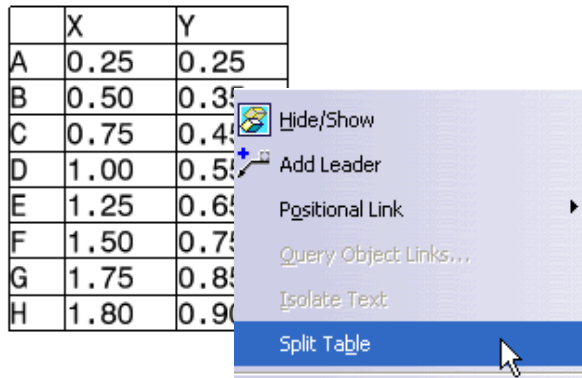
- 2 Double click a Table cell to edit , Use the contextual menu to select the item to modify or select a row/column marker to resize.



Splitting Tables

A table can be split if it becomes larger than a given number of rows

- 1 Right click on the table and select Split Table from the contextual menu



Example results:

	X	Y
A	0.25	0.25
B	0.50	0.35
C	0.75	0.45

	X	Y
D	1.00	0.55
E	1.25	0.65
F	1.50	0.75

	X	Y
G	1.75	0.85
H	1.80	0.90

- 2 Enter in the Split Criteria

Tick box if the first row of the original table is to be copied to the split tables



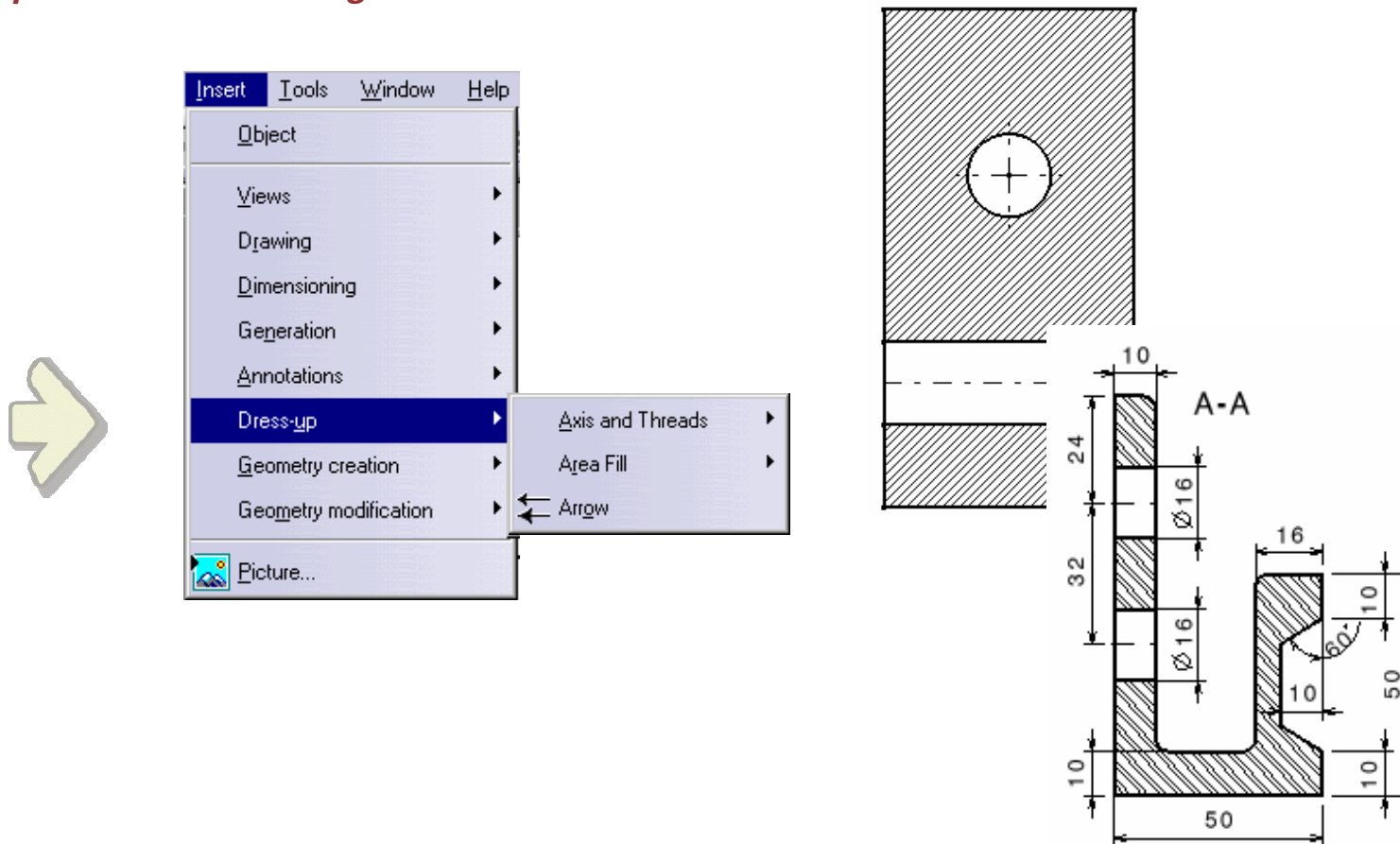
Enter number of rows the table can have before it splits or the maximum height before the split

Enter distance between the split tables

- 3 Select the OK button to complete

Graphic Dress-up

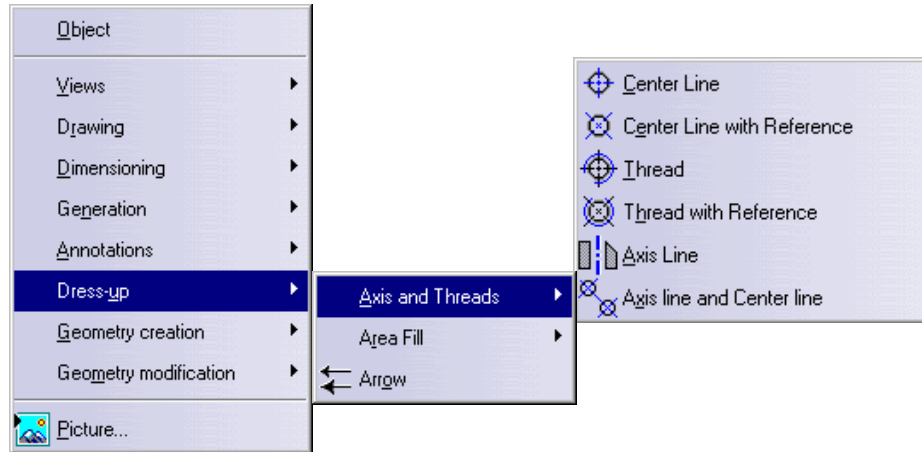
You will learn how to apply dress-up such as centerlines threaded representation and patterns to a drawing.



Student Notes:

Center Lines

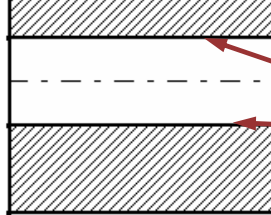
Centerlines for various applications are available in the Insert > Dress-up > Axis and Threads menu. Centerlines can be resized after the creation.



1 Select the Center Line icon 

2 Select the circle 

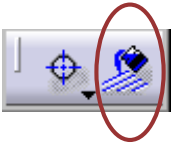
1 Select the Axis Line icon 

2 Select two reference lines to fit the axis line between. 

Adding a Pattern on a Section

To draw a section cut, you need to add a sectioning symbol to represent the material. You can choose between several types of symbols in a Pattern Table.

- 1 Select the Area Fill icon

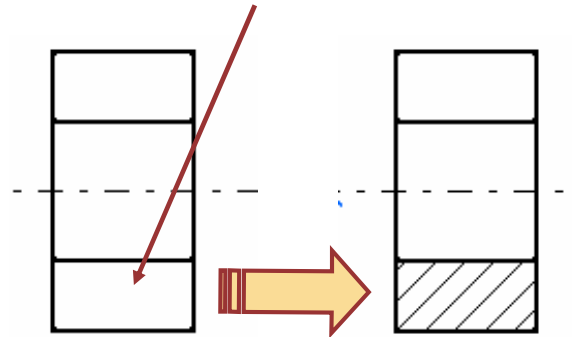


- 2 Select automatic Area detection.



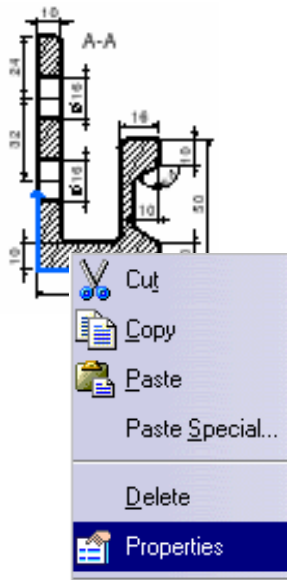
The "With Profile Selection" option allows you to manually define the area to be filled

- 3 Select in the zone to fill



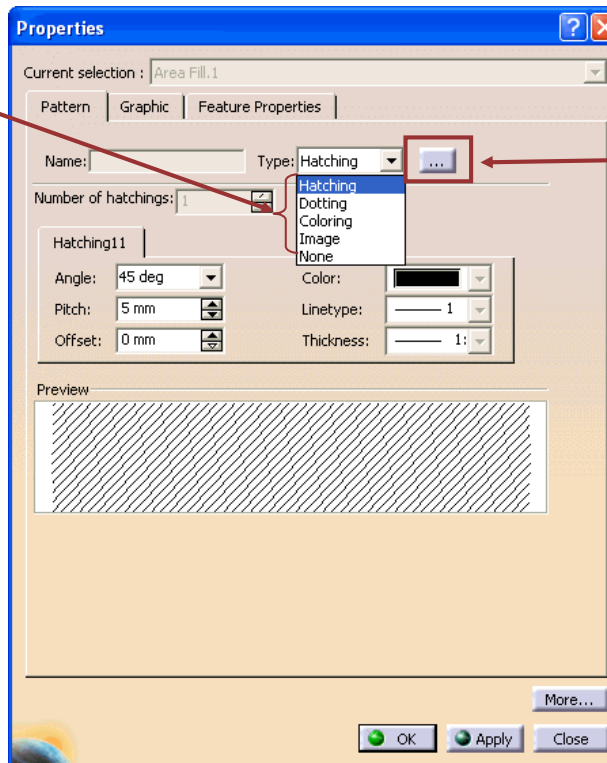
Changing Hatching Pattern (1/2)

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




- 2 Modify the current patterns attributes or select a new type of pattern from the type pull-down

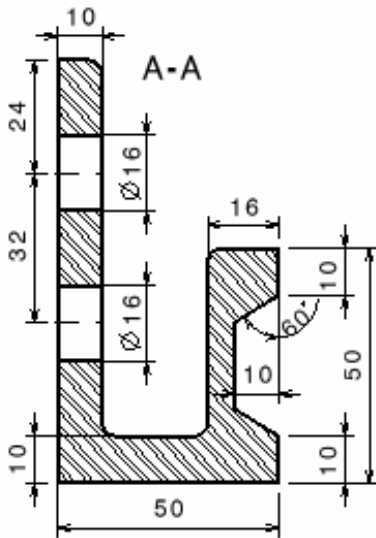
Note the different types of Patterns that can be selected from the pull-down



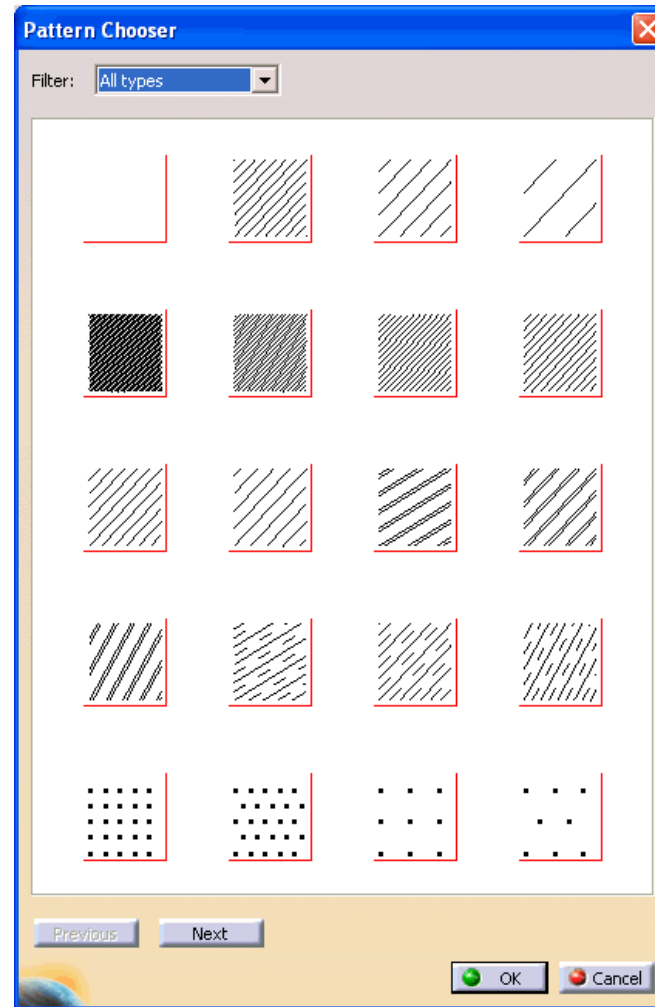
- 3 Click the button to browse to access the Pattern Chooser dialog box. (Discussed on the next Foil)

Changing Hatching Pattern (2/2)

- 3 Select the  to open the pattern Chooser.
- 4 Choose the Pattern type and Select OK to return to the Properties window.
- 5 Change the attributes of the new pattern type if necessary and select the OK button to update the pattern

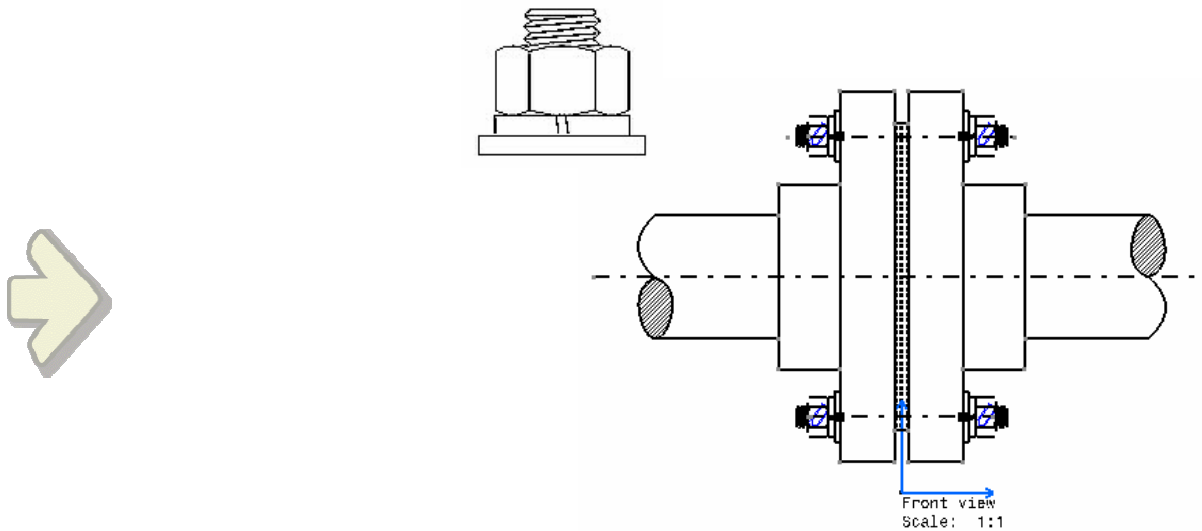


The pattern is changed on the section.



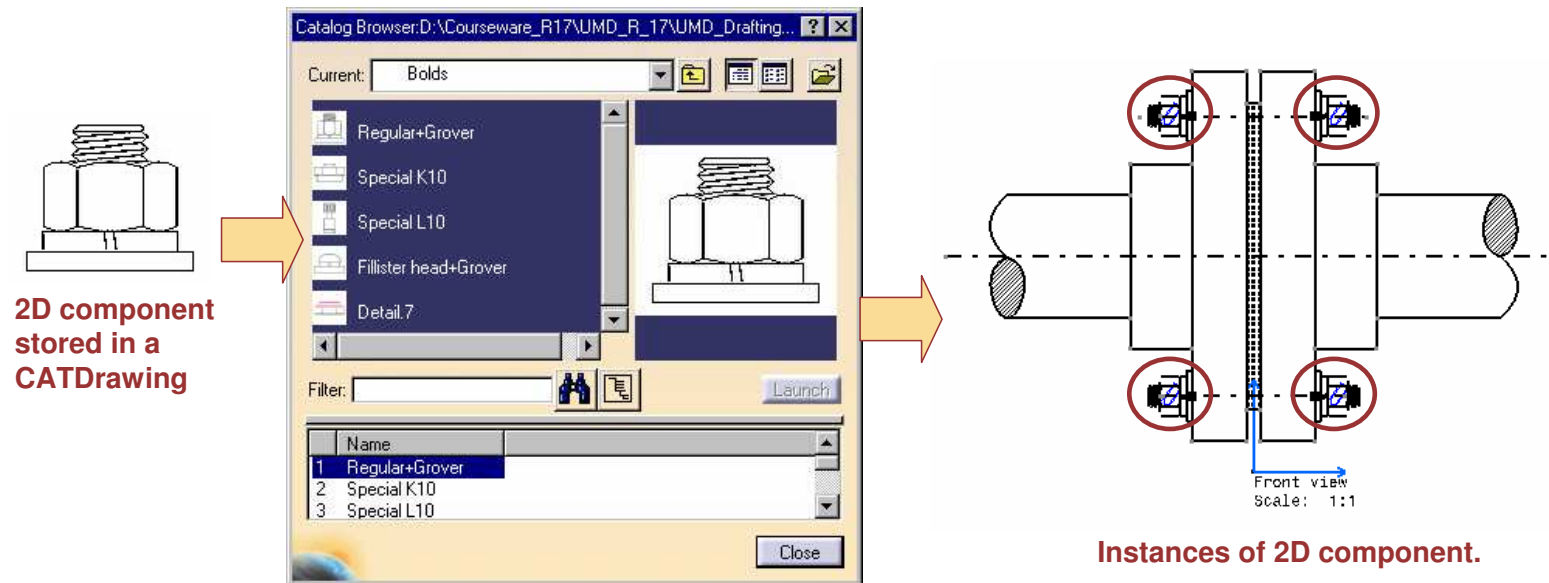
Instantiating Elements From a Catalog

You will learn how to instantiate a 2D elements from a catalog and how to synchronize links between CATDrawing and the catalog file.



Why Instantiate a Component From a Catalog

A 2D component is a re-usable set of geometry and annotations. This component is stored in a CATDrawing referred by the catalog. The 2D component can be instantiated several times, each instance providing a component with a specific orientation, position and scale.

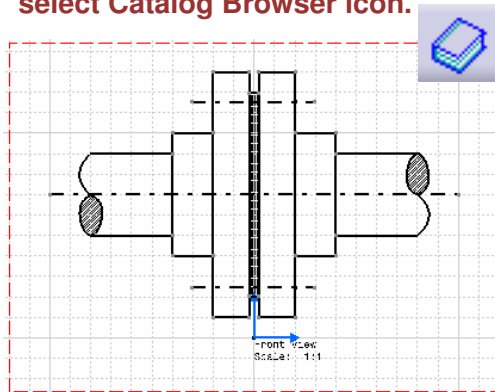


Link is maintained between the Instances and the original 2D component. You can update the instances to reflect the changes made in original 2D component.

How to Instantiate a 2D Component From a Catalog (1/2)

You can multi-instantiate a 2D component from catalog, reorient the component and explode them at the time of instantiating.

- 1 Activate the view in which you want to instantiate the 2D component and select Catalog Browser icon.



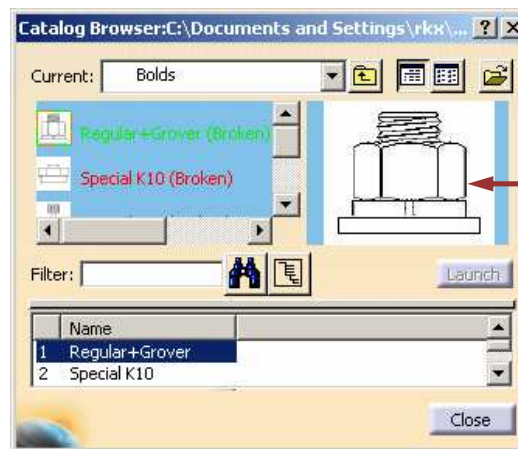
- 2 Open the source catalog file, using Browse another catalog tool.

Catalog Browser dialog box gives you list of the chapters in the catalog file.



- 3 Double click on the chapter in which required 2D component is placed.

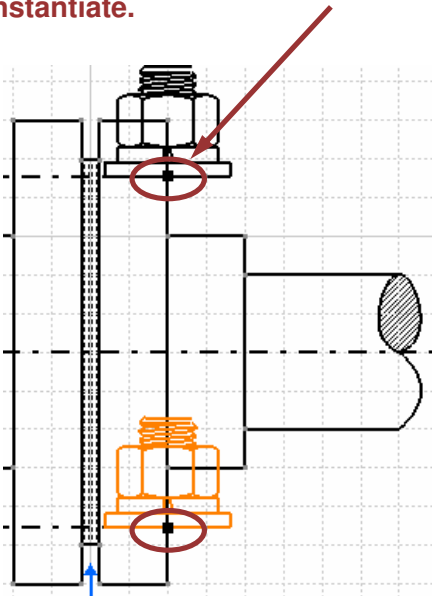
- 4 Select the 2D component you want to instantiate.



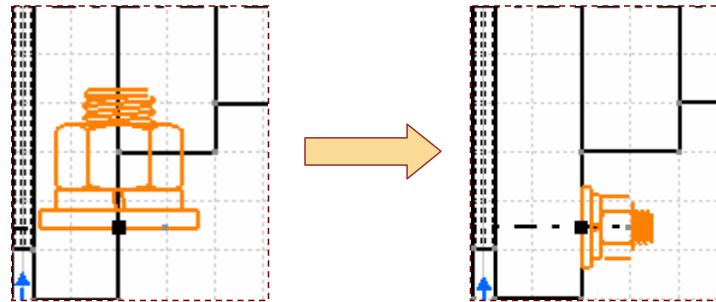
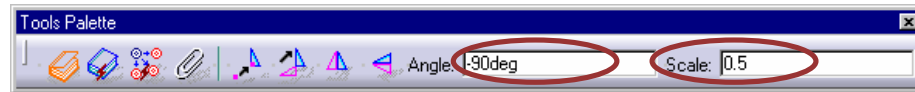
Preview of the component will be displayed.

How to Instantiate a 2D Component From a Catalog (2/2)

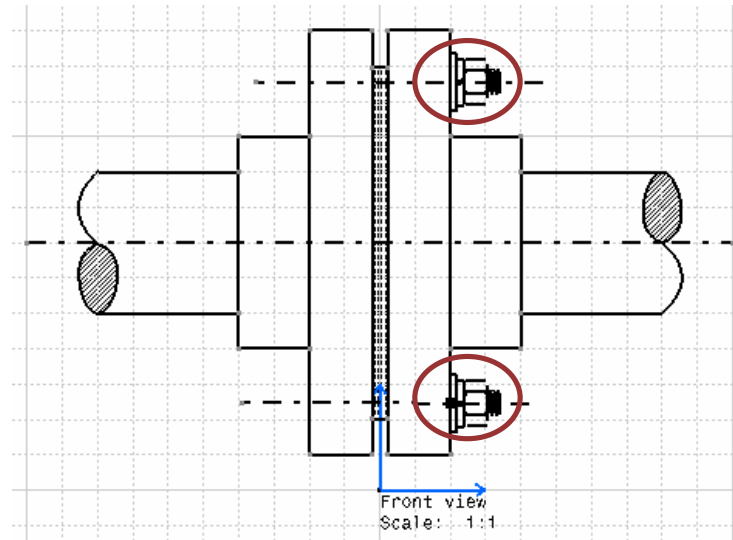
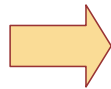
- 5 Drag the component from Catalog Browser into the drawing window, and select the points where you want to instantiate.



- 6 Set the Angle and Scale values



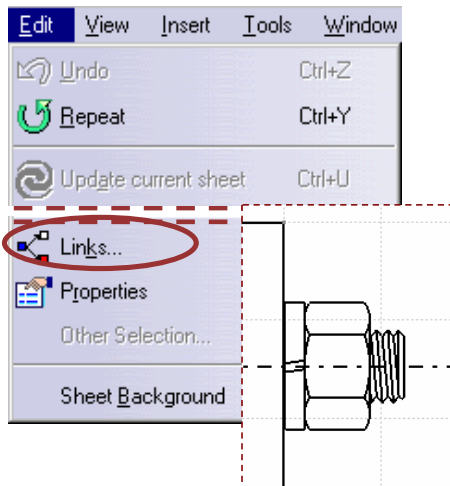
- 7 Click on the empty space in the sheet to exit the command.



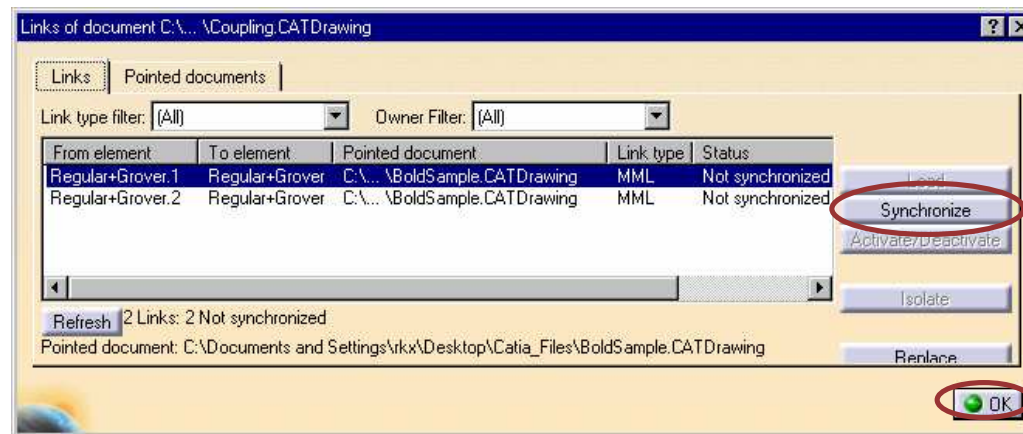
How to Synchronize the Links

When a original 2D component is modified, you need to synchronize the link between the 2D component and the instance to reflect the changes made in original 2D component.

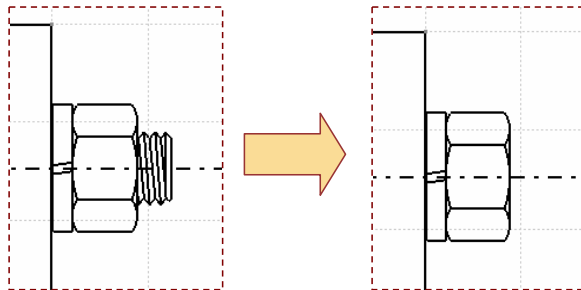
1 Select Edit > Links.



2 In the Links dialog box select the MML link between the 2D Component and the instance, select Synchronize.



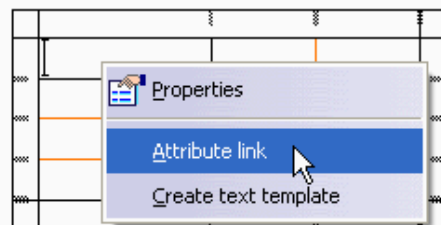
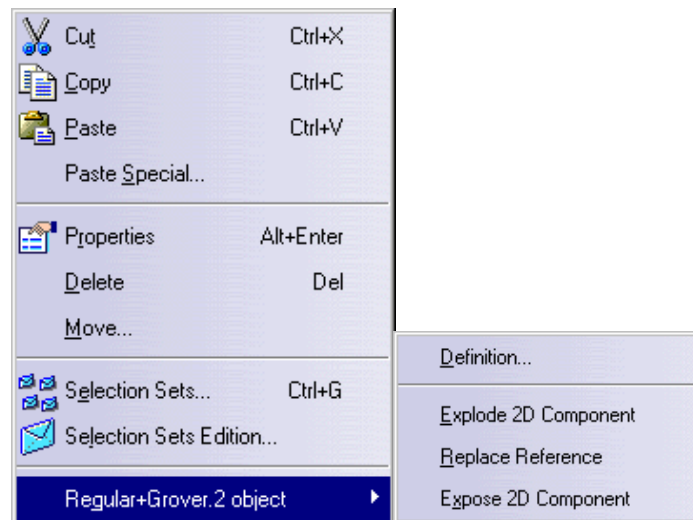
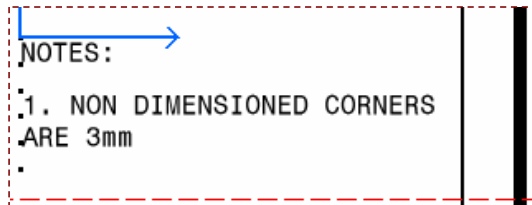
3 Select OK inside the Links dialog box.



The instance is modified as per the changes in the original 2D component.

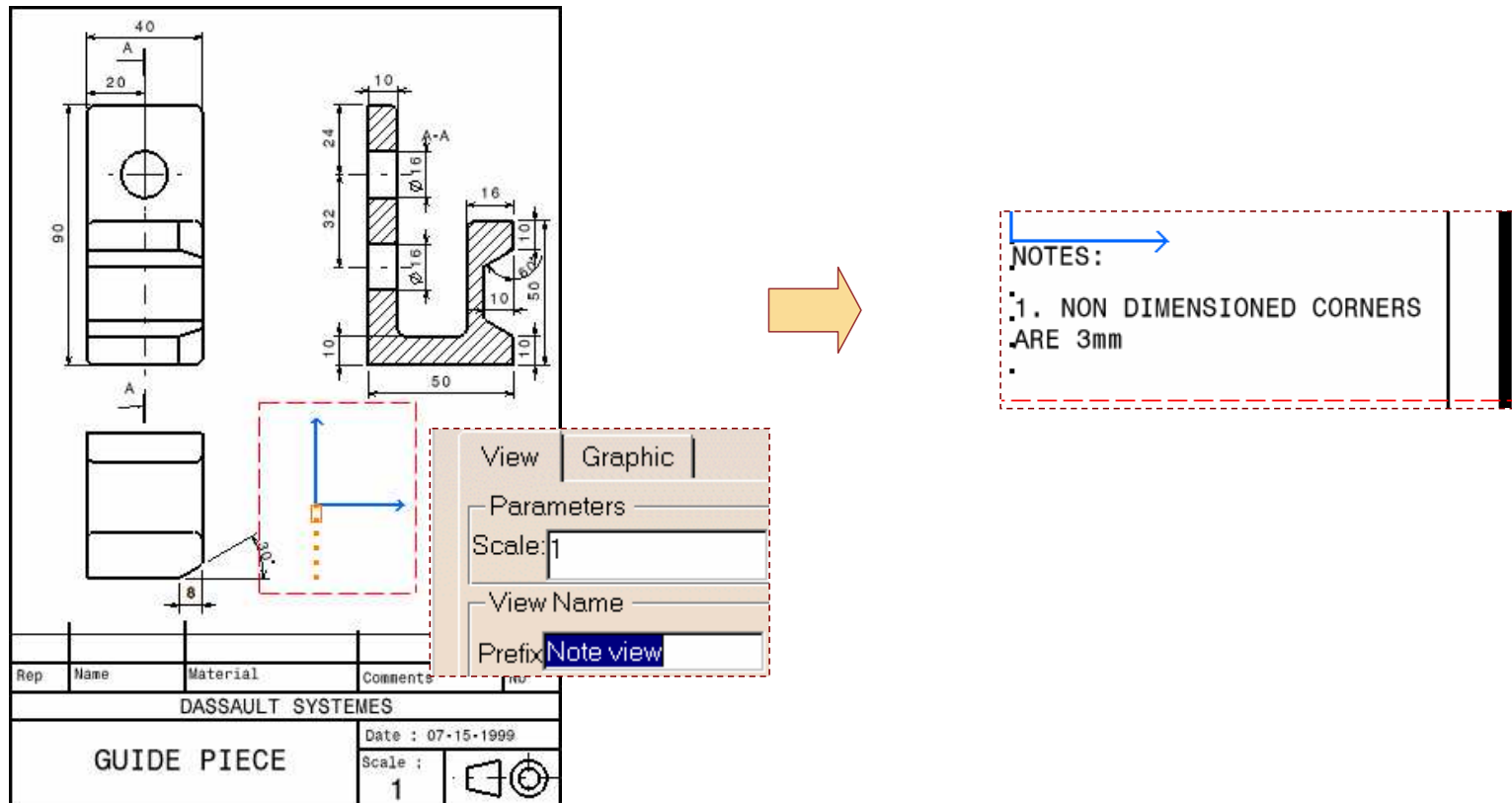
Methods and Recommendations

You will learn about notes view, attribute links and links of a 2D component instance.



Using a Notes View for Text and Table

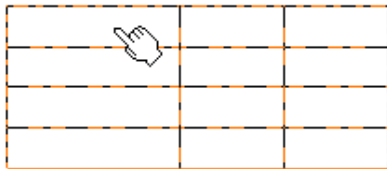
A note view is used to add the notes in the drawing. A Note View is a stand alone view that is not projected from an existing View. Create a front view and rename it to “Notes View” and then add notes to this view.



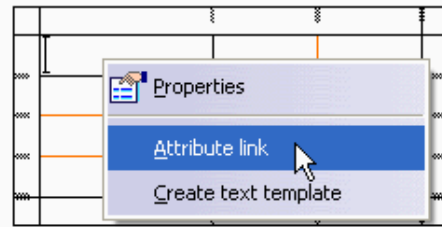
Adding an Attribute Link

To create an attribute link in a note or table cell use the same technique

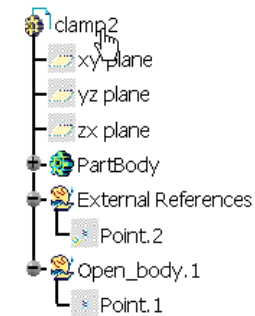
- 1 Double click on the text you want to add the link to



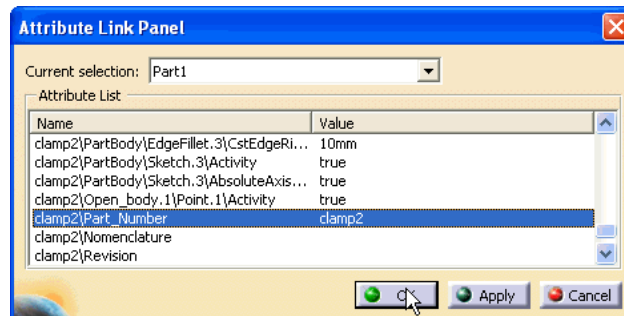
- 2 Right click and select Attribute Link from the contextual menu



- 3 Select the object you want the text to link to



- 4 Select the attribute from the list and click the OK button to complete



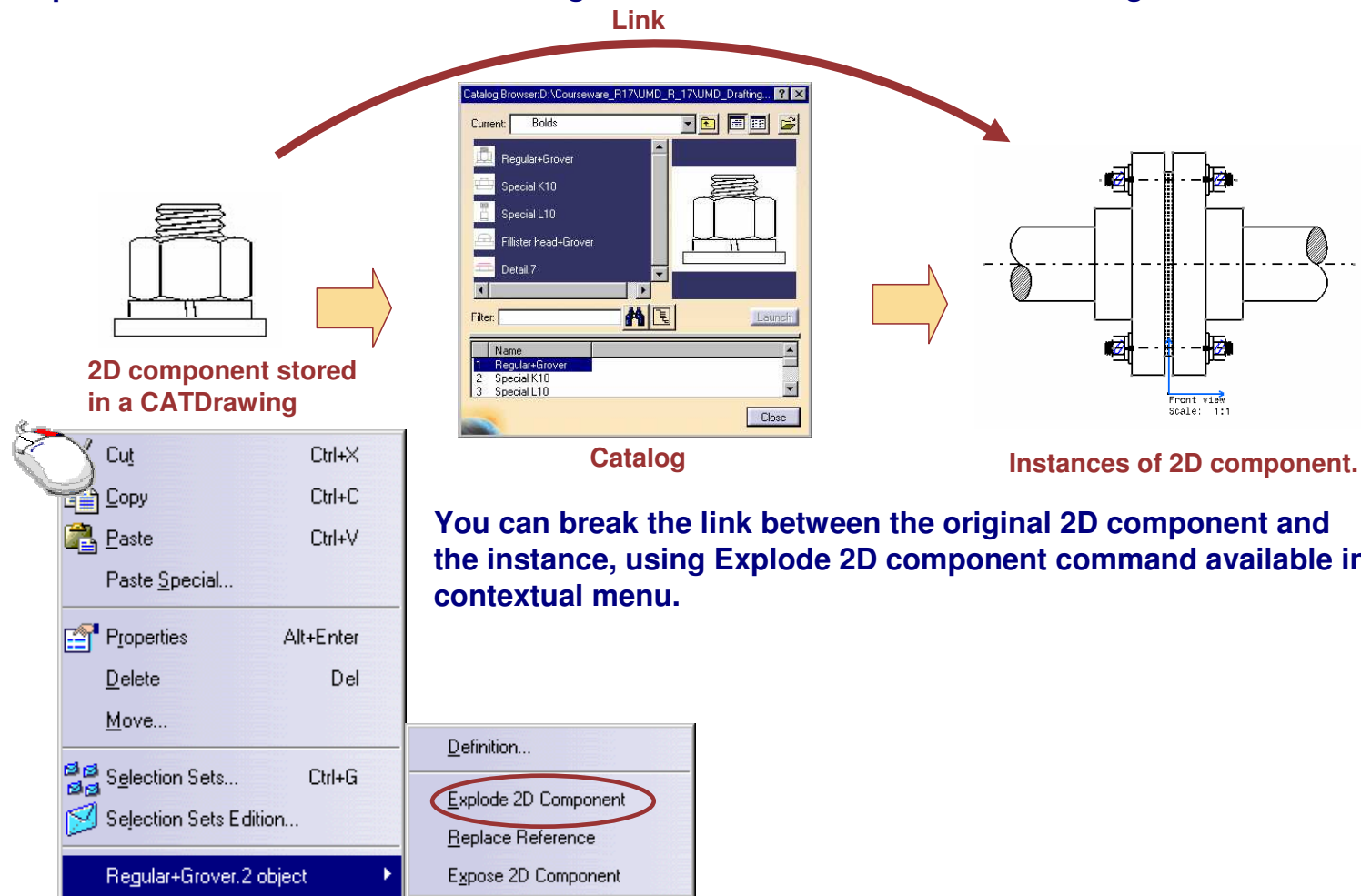
- 5 Example results:

clamp2		

An attribute link will update when changes are made in the parameter it is linked to, provided the Automatic Update mode is turned on in the Options > Mechanical Design > Part Design > General Tab

Links to the Catalog

When a 2D component is instantiated from a catalog a link is created between the 2D component instance and the CATDrawing file which is used to create the catalog.



To Sum Up

You have learned how to:

- Add symbols to a drawing
- Select the appropriate symbol from the Symbol toolbar
- Add Tolerance
- Use the Dimension Properties toolbar for quick access to tolerance
- Create GD&T symbols
- Use the Tolerancing Toolbar
- Create text
- Use the Annotation toolbar
- Create a table

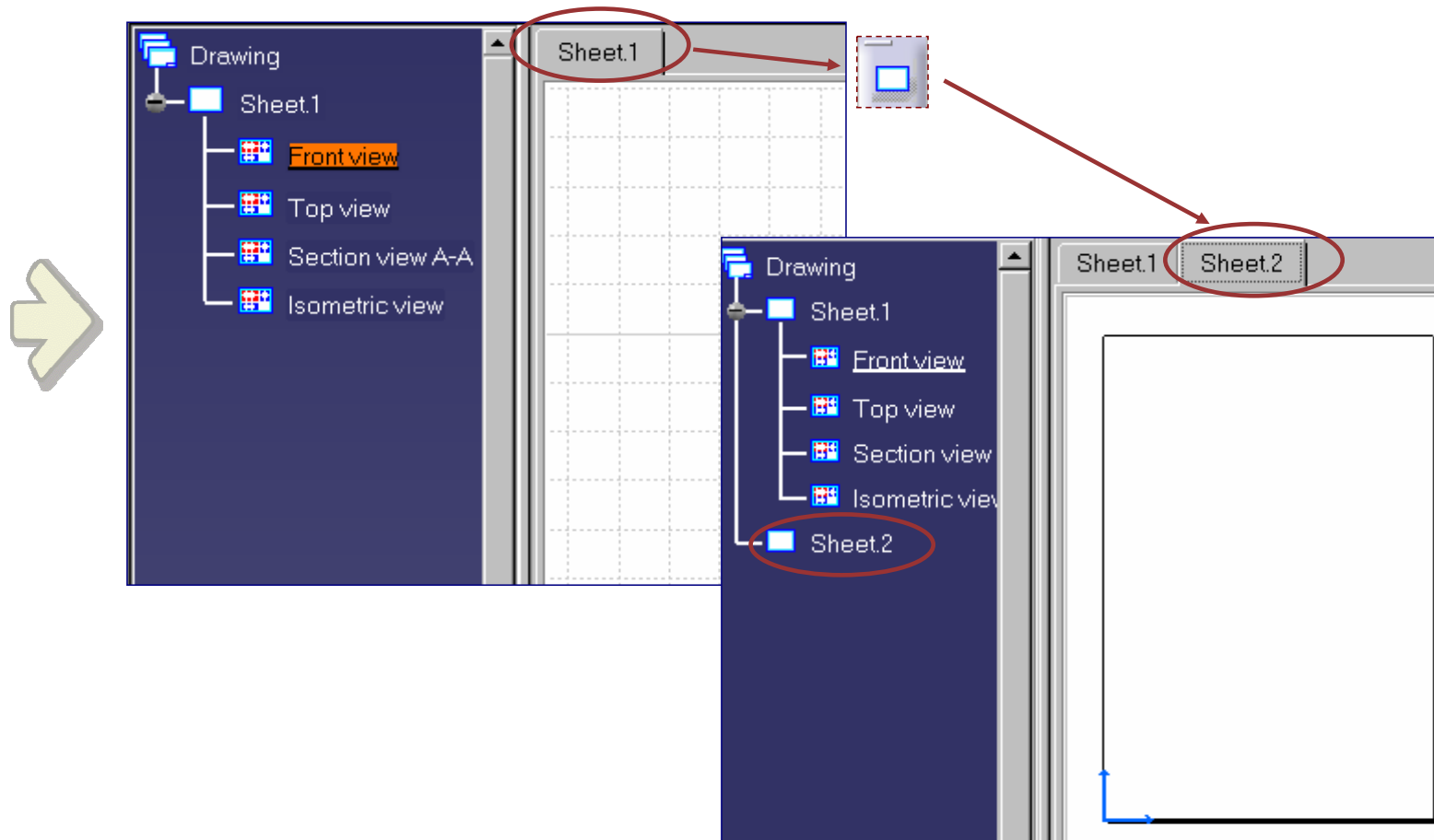
View and Sheet Management

It is sometimes necessary to modify sheets and views properties to improve layout to better suit the needs of a particular drawing.

-  **Sheets Management**
-  **Views Management**
-  **To Sum Up**

Sheets Management

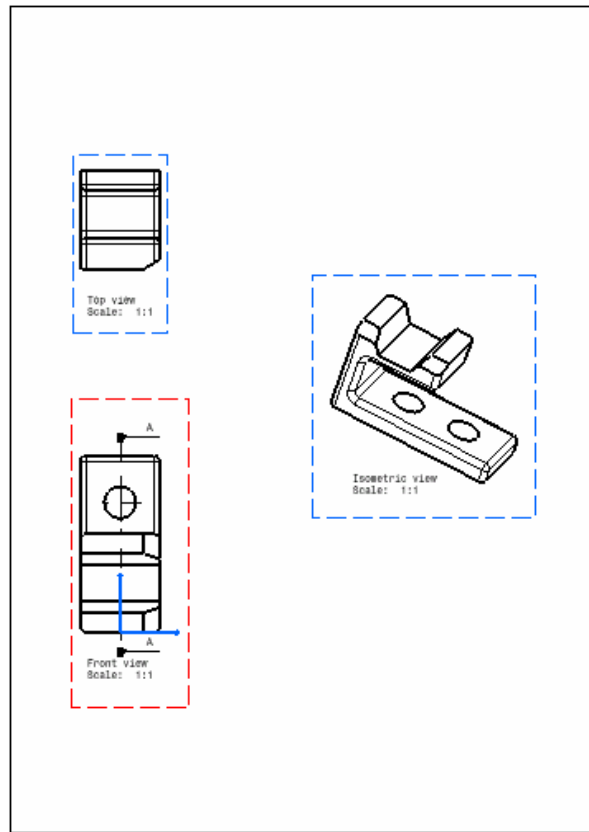
You will learn how to add a new sheet to a drawing, and how to change the properties of a sheet.



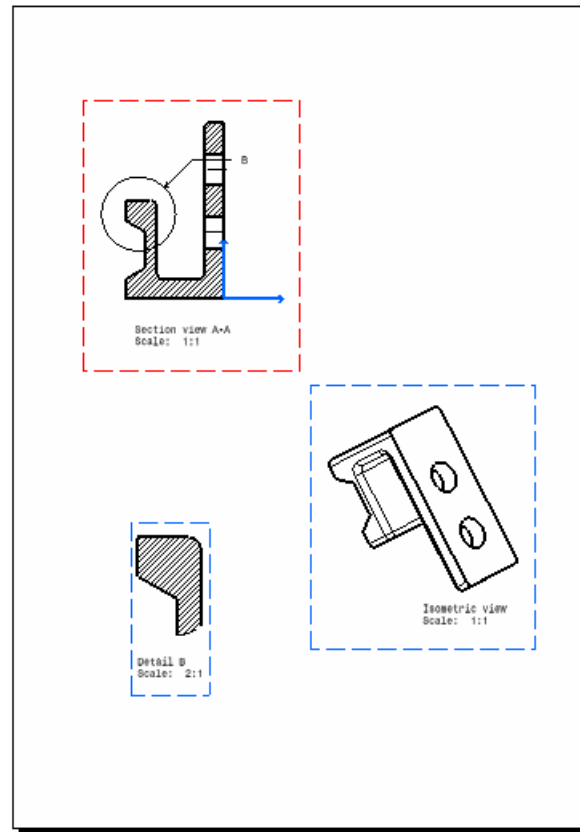
Why Add Sheets to a Drawing?

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

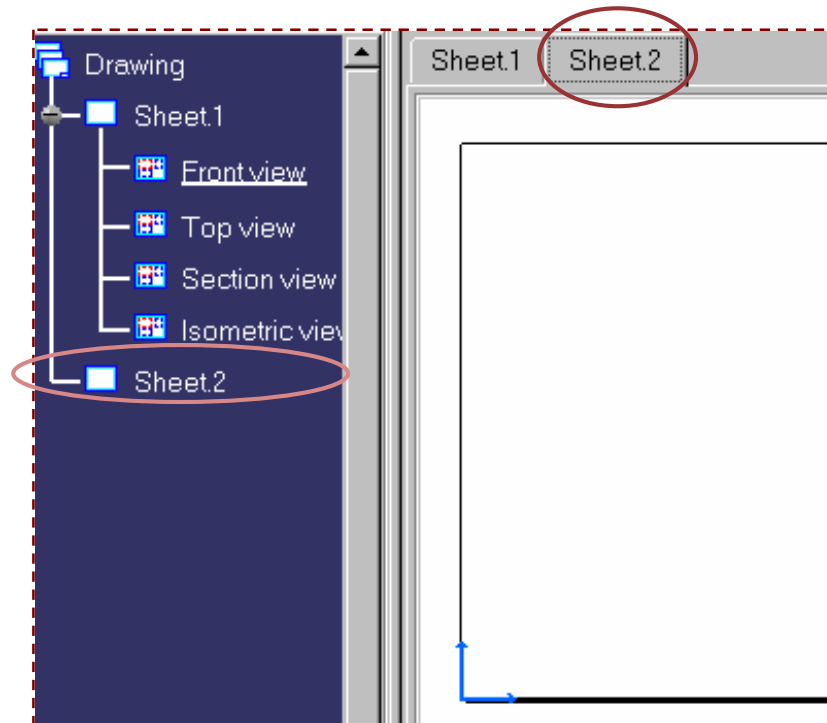


Sheet 2 of 2



Adding a Sheet to a Drawing

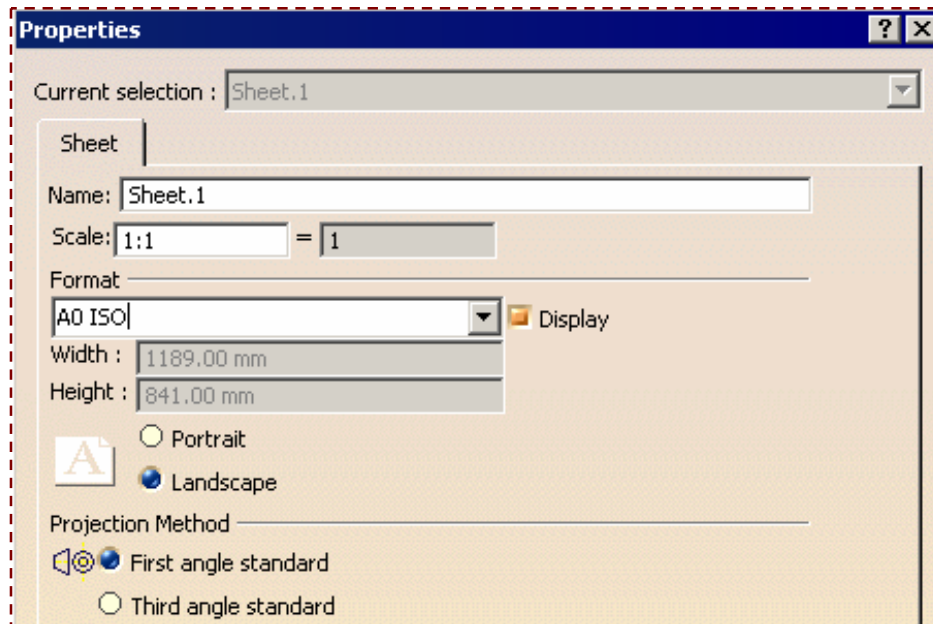
- 1 Select the New Sheet icon
It creates an empty sheet with the next sheet number (Sheet 2)



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

Why Change Sheet Properties?

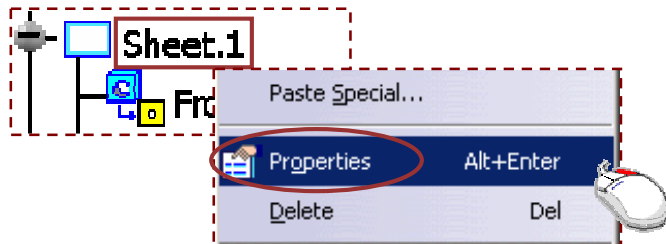
There are times when one or more sheets in a drawing may require properties that are different from the rest of the drawing sheets. By accessing Sheet Properties you can customize individual sheets.



Sheet Properties

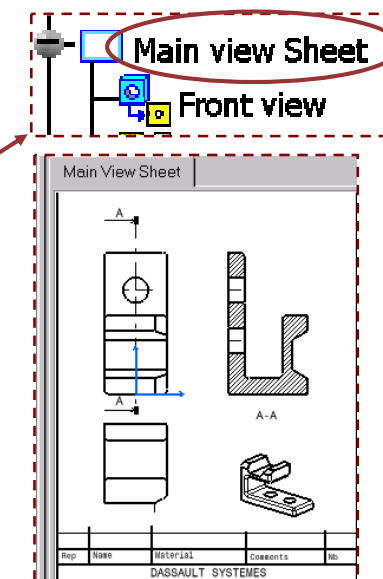
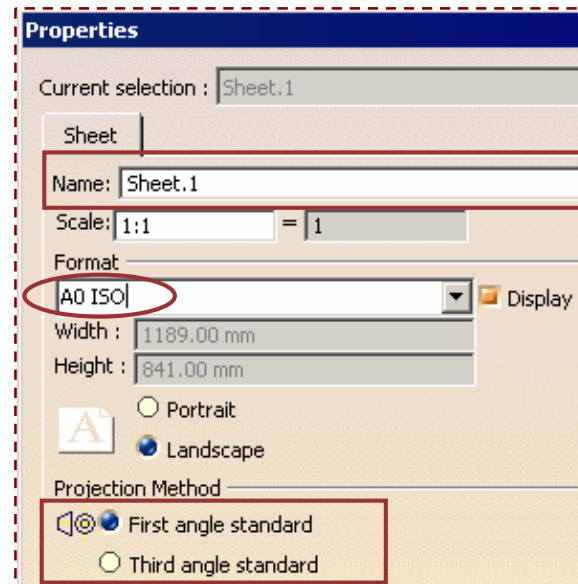
1

Select the Sheet in the tree and Properties in the contextual menu to change its properties.



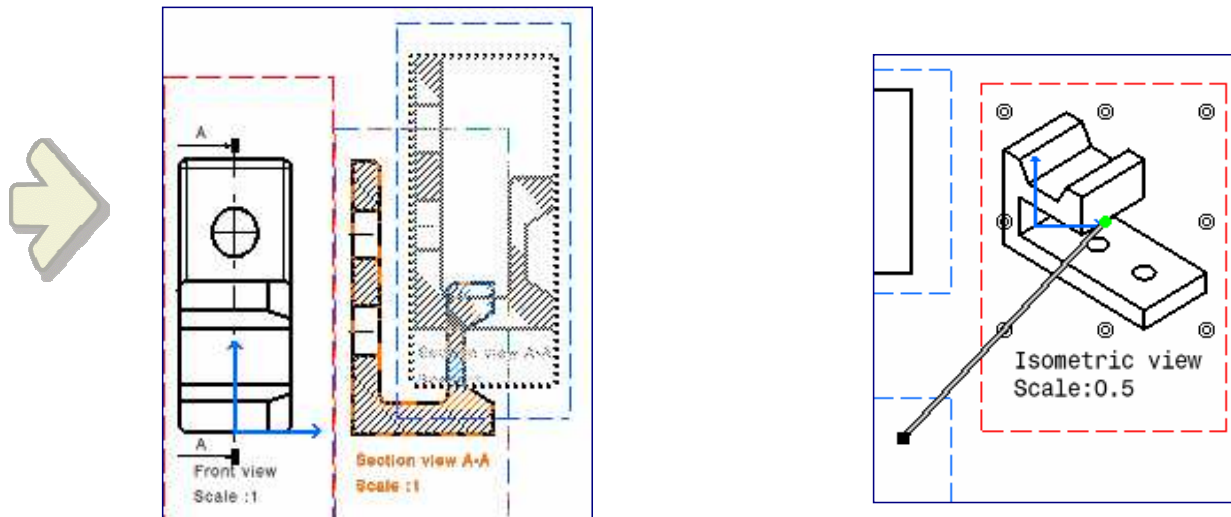
2

Modify the sheet name, the scale, the format (ANSI or ISO) or the projection method (first angle or third angle).



Views Management

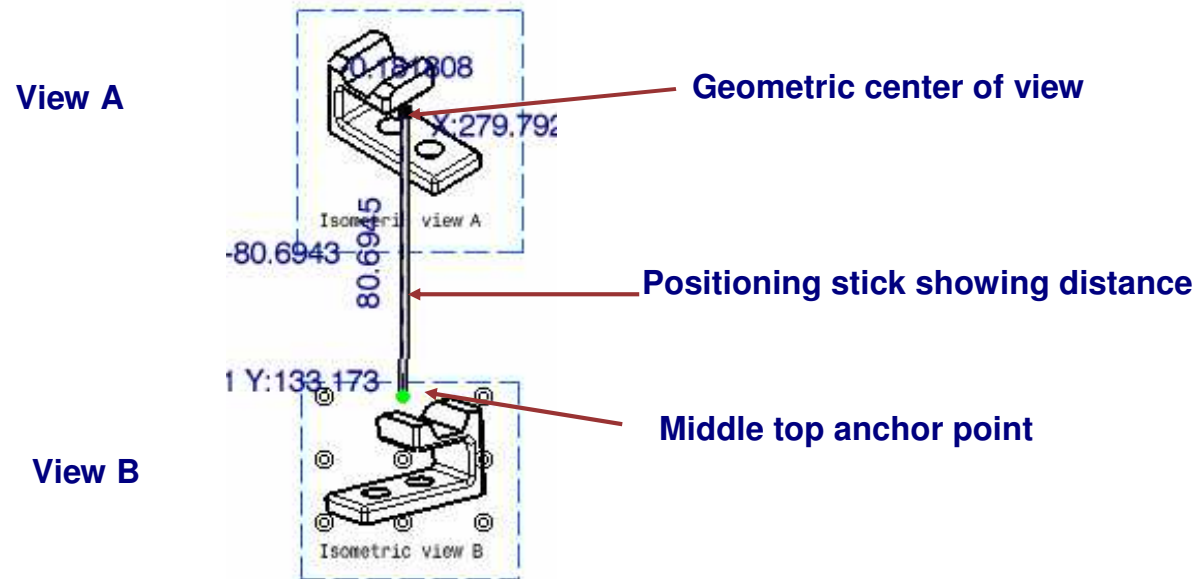
You will learn how to reposition views on a single sheet and how to move views from one sheet to another sheet.



Relative Positioning

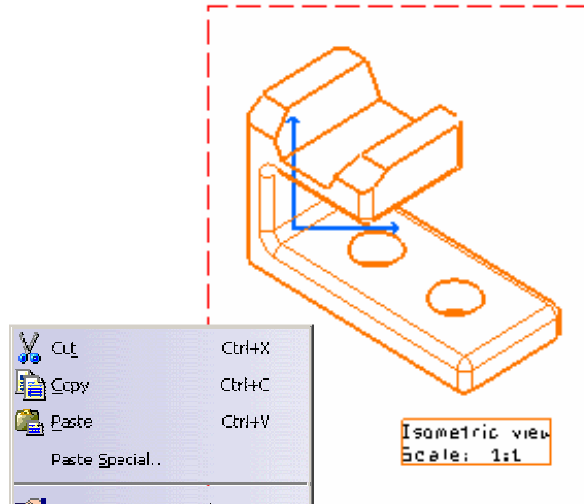
A view can be positioned at an exact position on a Sheet and relatively to another view already on the sheet.

Here view B is positioned relatively to view A



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

Repositioning a View with Relative Positioning (1/3)



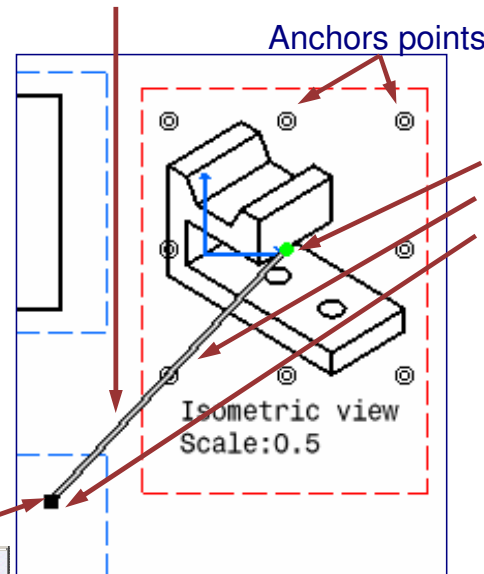
- 1 Select « View Positioning » then « Set Relative Position » in the contextual menu

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



Set Relative Position
Position Independently of Reference View
Superpose
Align Views Using Elements

Positioning Stick



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).

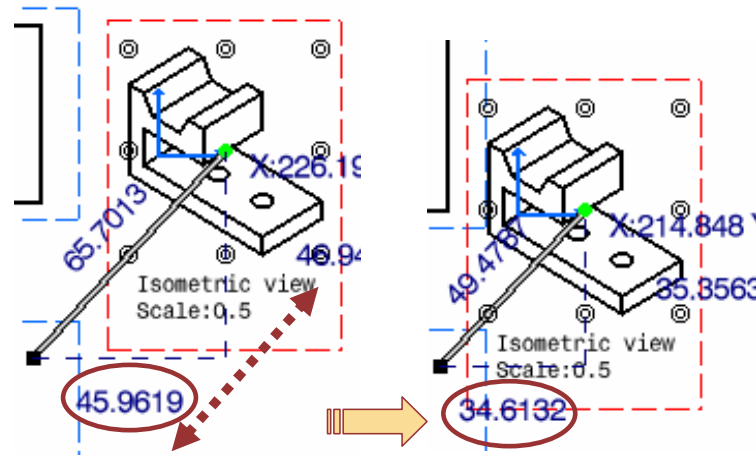
Remember the contextual menu can be accessed from the tree structure or the view frame. Therefore, the view frame does not need to be on to use the relative positioning option.

Repositioning a View with Relative Positioning (2/3)

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

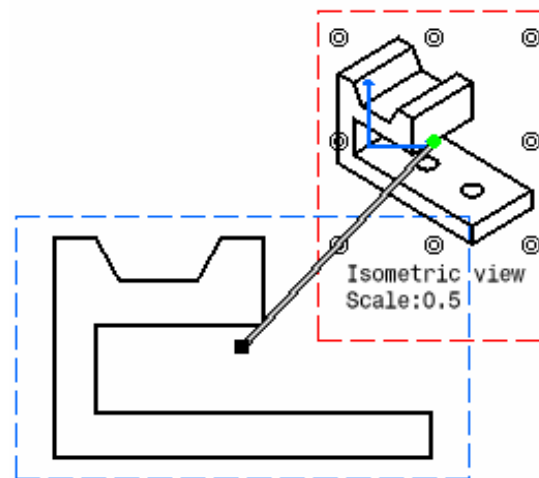
(A) Selecting the Stick

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

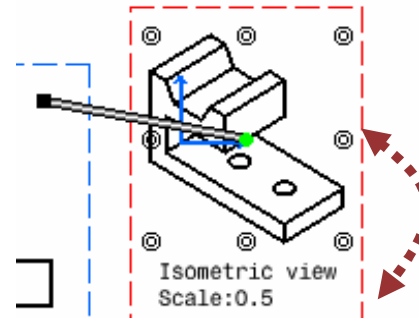
The black end snaps on the center of the selected view frame and the view follows



Repositioning a View with Relative Positioning (3/3)

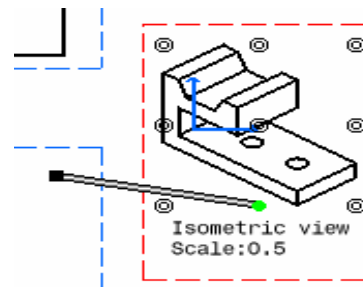
(C) Selecting the green handle

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)

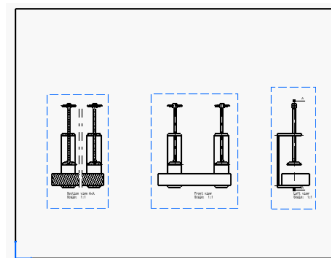
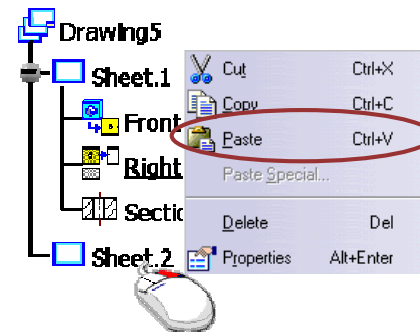
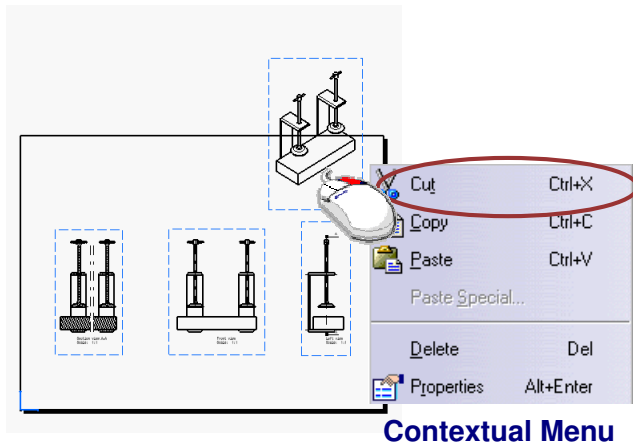


Moving Views from one Sheet to Another Sheet

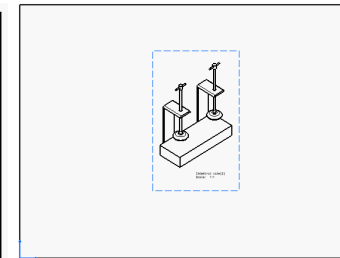
A view can be moved from one sheet in a drawing to another. This can be useful when a sheet becomes too cluttered.

1 Cut the view on the sheet to be moved.

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



Sheet 1

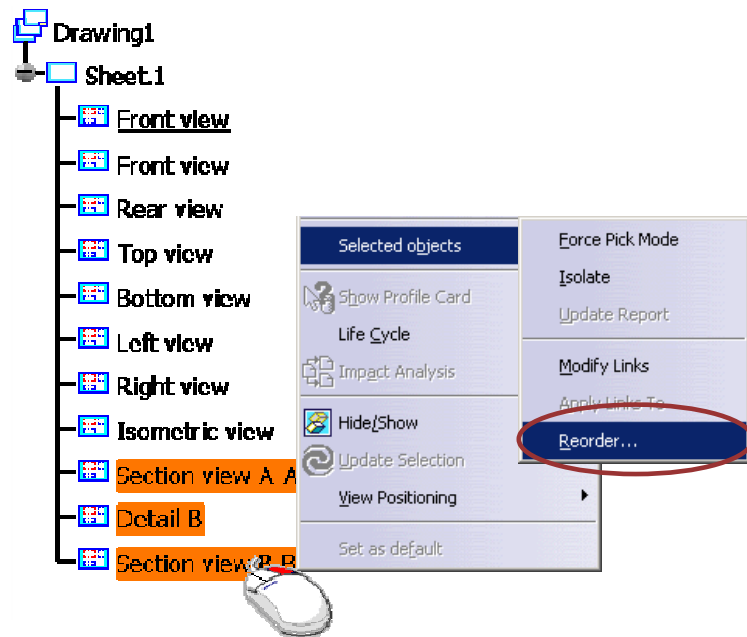


Sheet 2

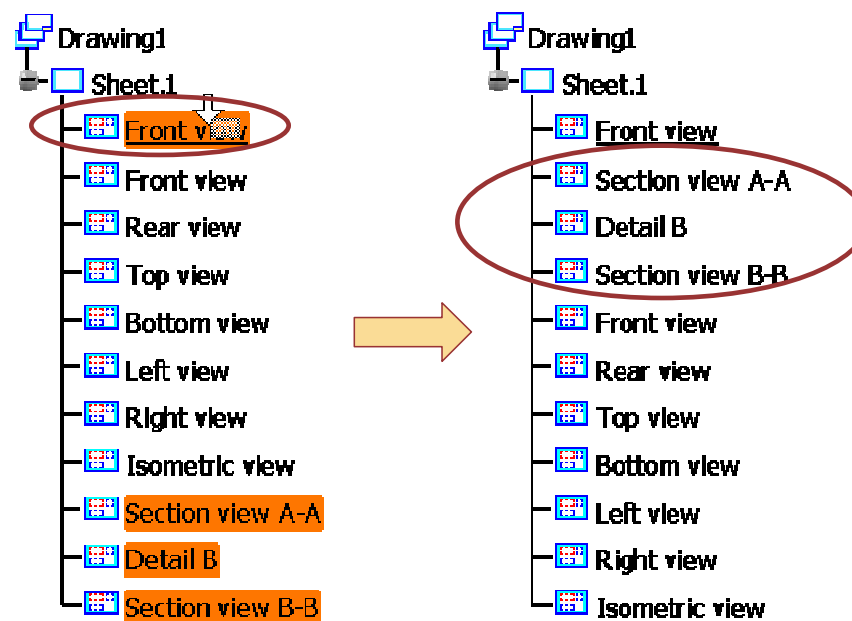
Reordering Views in the Specification Tree

You can organize your specification tree by using reorder command. The children views can be arranged after the parent view.

1 Select the views you want to reorder and launch reorder command from contextual menu.



2 Select the view after which you want to place the reordered views.



Section view A-A, Detail D
and Section view B-B
reordered after the front view

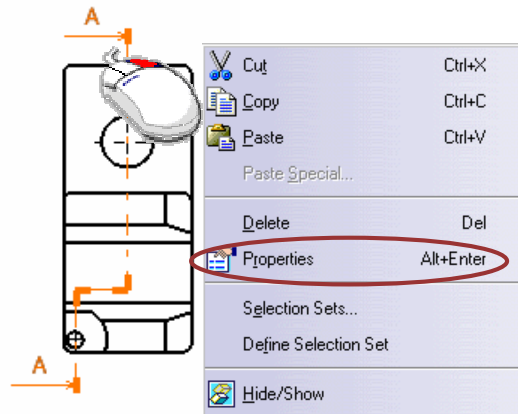


In the same way, you can reorder the sheets also.

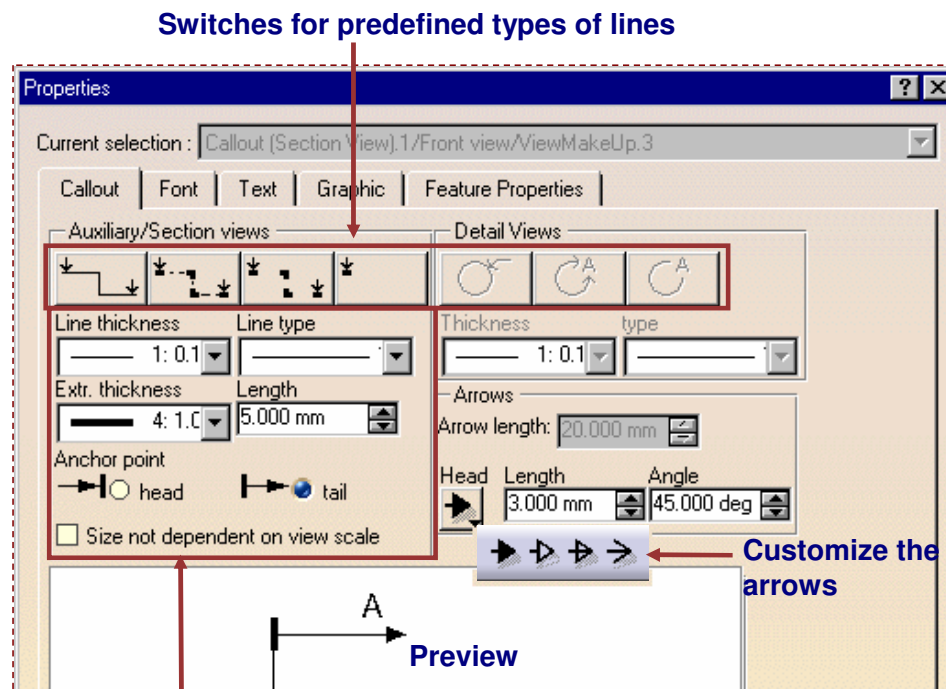
Modifying Graphical Definition for Views

According to your or your customer's needs, CATIA allows you to modify the graphical attributes of Section, Detail or Auxiliary views.

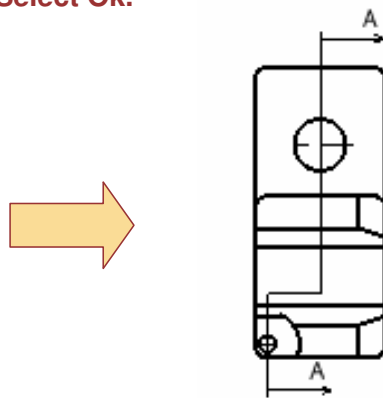
1 Select the view callout and select Properties with the contextual menu.



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



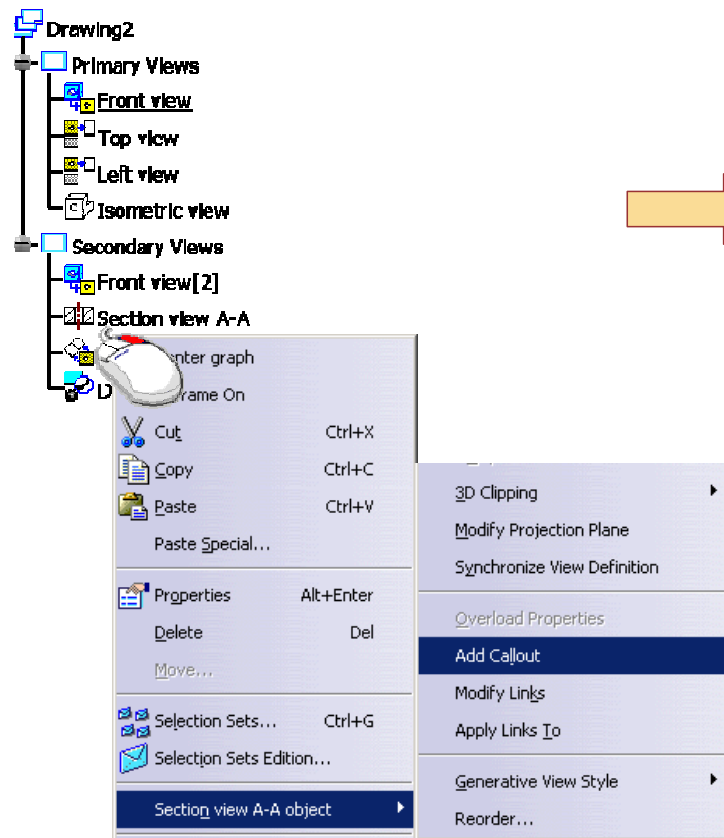
3 Select Ok.



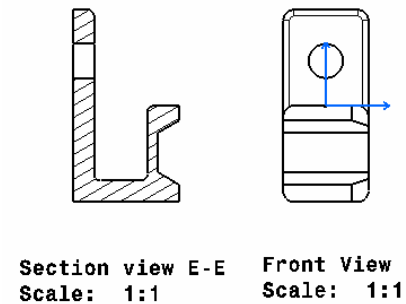
How to Restore the Graphical Definition?

You can restore the deleted graphical definitions of section views, auxiliary views and detail views.

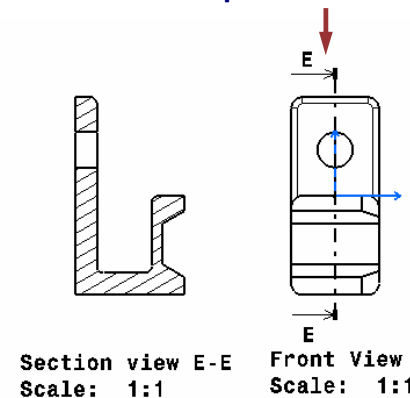
- 1 Right click on the section view and select Section view XX object > Add Callout



- 2 Select the parent view



Graphical representation of the section view restored in its parent view.







To Sum Up

In this lesson you have learned:

- How to add additional sheets.
- How to modify sheet properties.
- How to edit the properties of a sheet.
- How to reposition the views.
- How to move a view from one sheet to another.
- How to modify graphical definition for views.

Customizing the Drafting Workbench

In this lesson we will look at options available to you to customize the Drafting workbench.

-  **Drawing Customization Options**
-  **Setting Drafting options**
-  **Standard Management**
-  **To Sum Up**

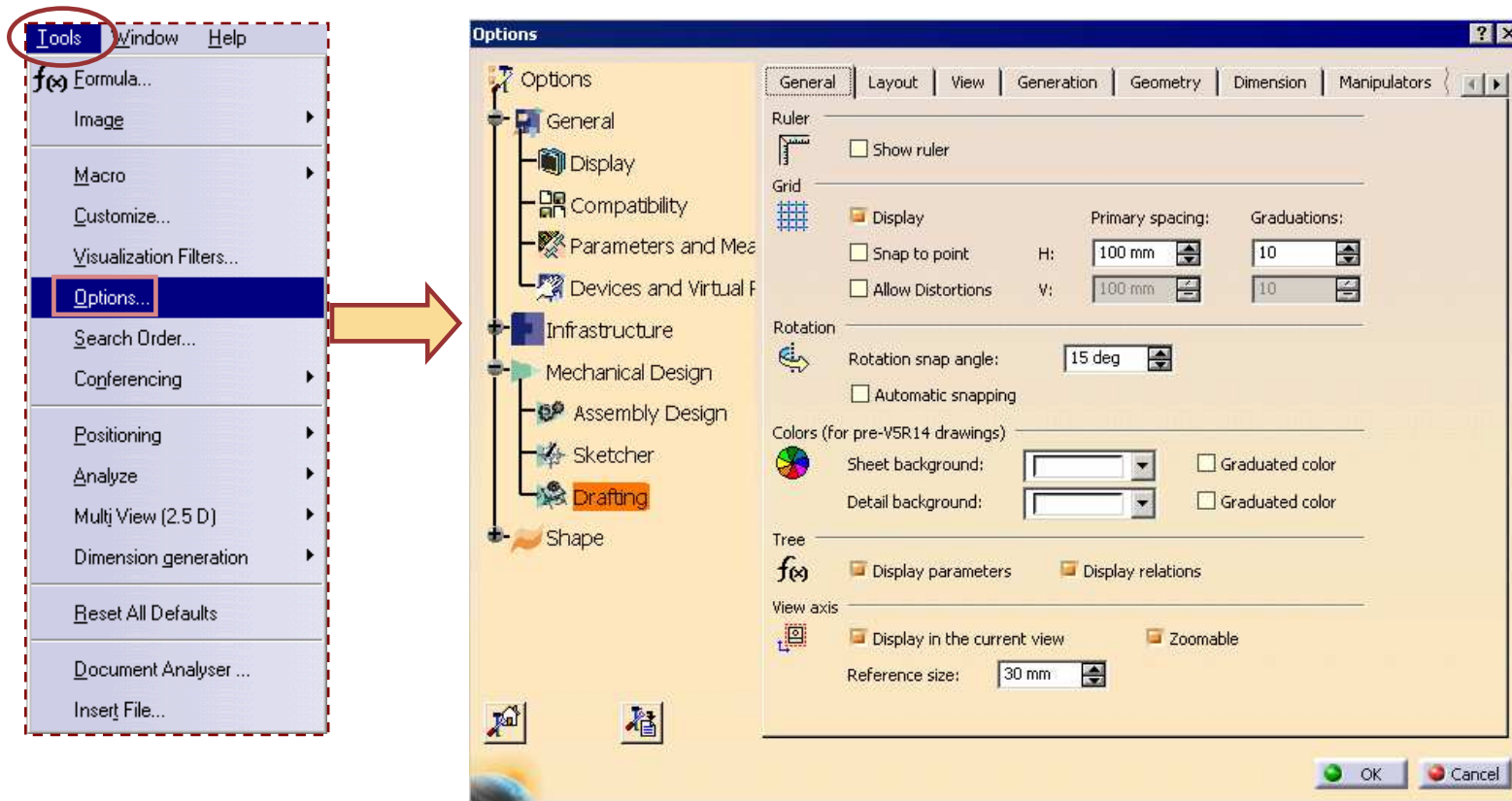
Drawing Customization Options

The options shown in this course are just some of the numerous options available to customize the Drafting workbench.

The options that will not be discussed in this course are typically controlled by an administrator and not a user.

Setting Drafting Options

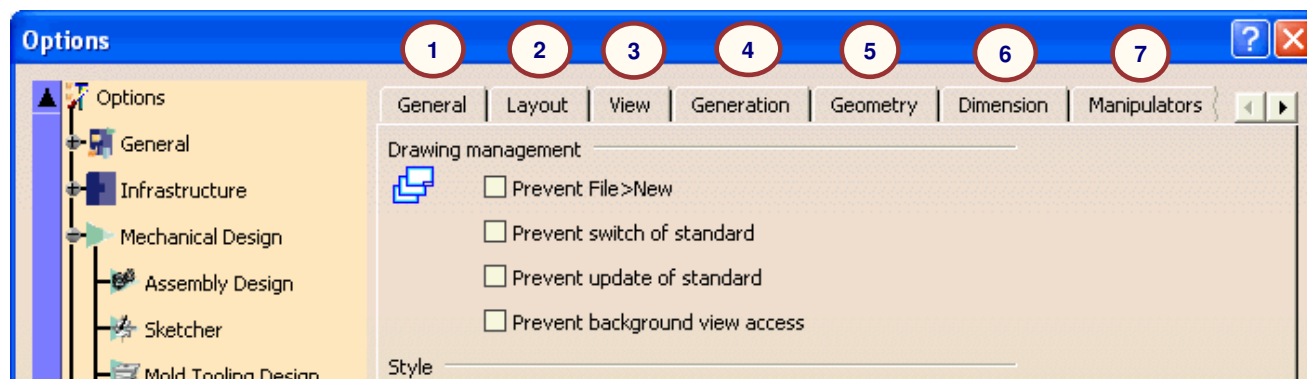
You will learn how to set the session's default drafting options



What are Drafting Options? (1/2)

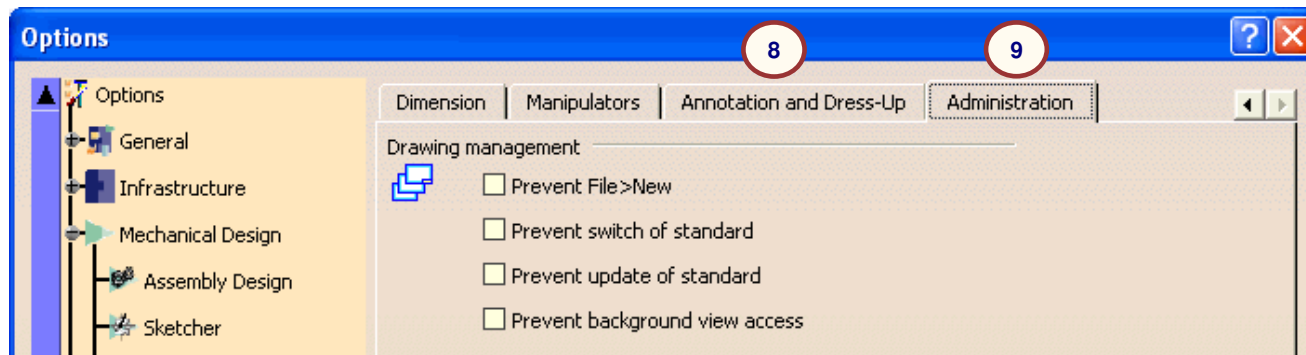
There are primarily **nine** Drafting Option tabs that allow the user to customize the drafting interface.

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. **View** – Determines the display of geometry and view generation
4. **Generation** - Determines dimension and geometry generation
5. **Geometry** - Aids to create geometry such as display of center points, auto-detection for orientation, and constraint creation and display
6. **Dimension** - Position, Line-up dimensions and Analysis display mode
7. **Manipulators** – Turns on / Off the manipulators for dimension creation or modification



What are Drafting Options? (2/2)

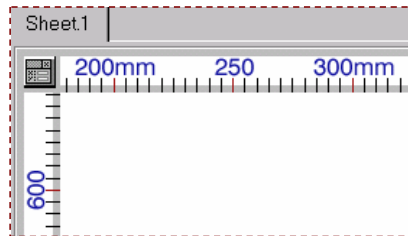
8. **Annotation and Dress-Up** – Turns on/off the controls for annotations
9. **Administration** – Controls for administrator



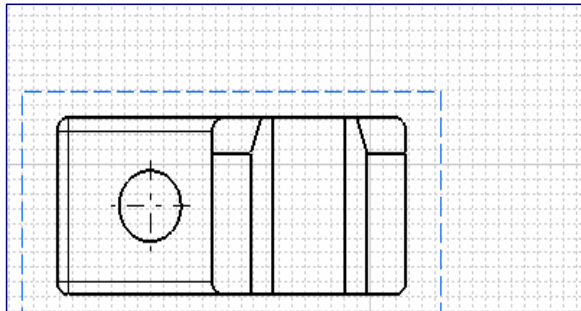
General Options (1/2)

Set the following General options:

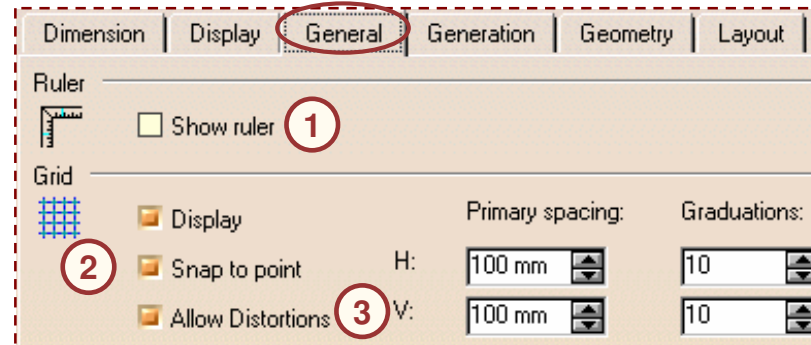
- 1 **Show Ruler:** In the OFF position the ruler along the top and left side of the screen will not be displayed.



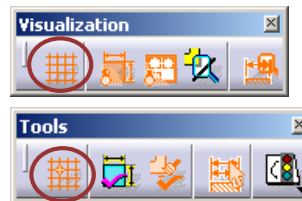
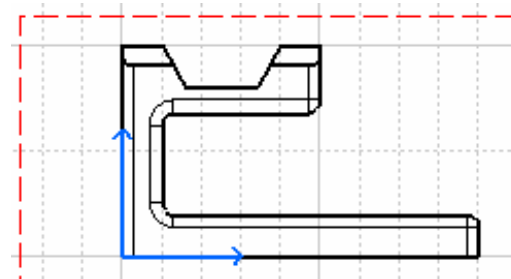
- 2 **Grid:** With the Display turned ON and Snap to point turned ON. Adjust the Primary spacing and graduations to aid in dimension placement.



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



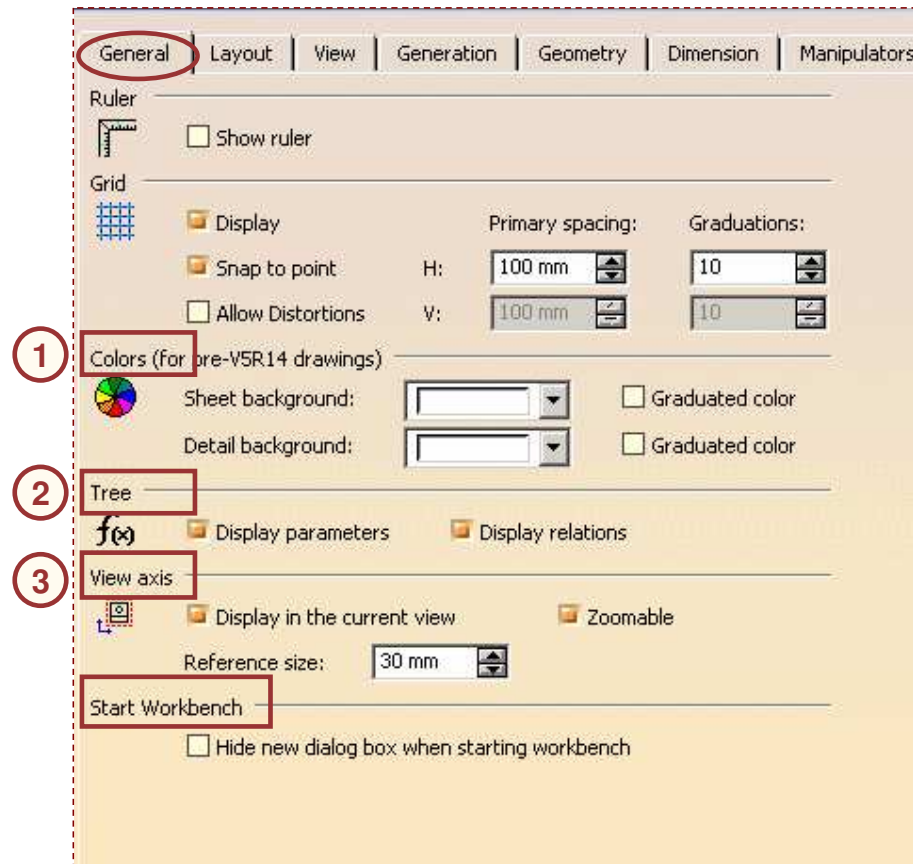
- 3 **Allow distortions:** Allow you to change the scale of H and V on the grid.



General Options (2/2)

Set the following Display options:

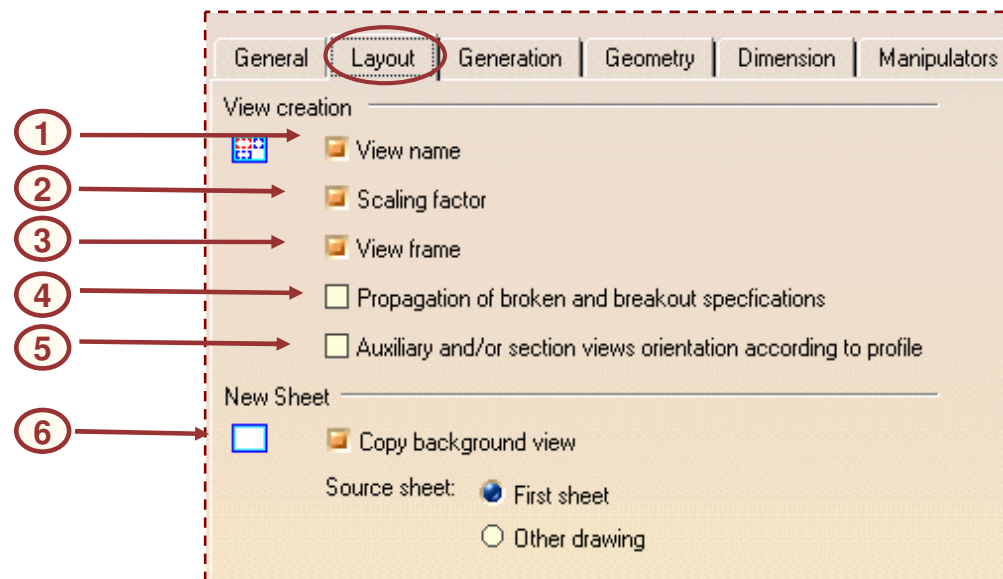
- 1 **Colors:** Change colors of the sheet and detail background.
- 2 **Tree :**Display parameters and relations in the specification tree.
- 3 **View Axis:** Provides a blue axis in the view that is current.
- 4 **Start Workbench:** 'New' dialog box when starting the drafting workbench.



Layout Options

Set the following view Layout options:

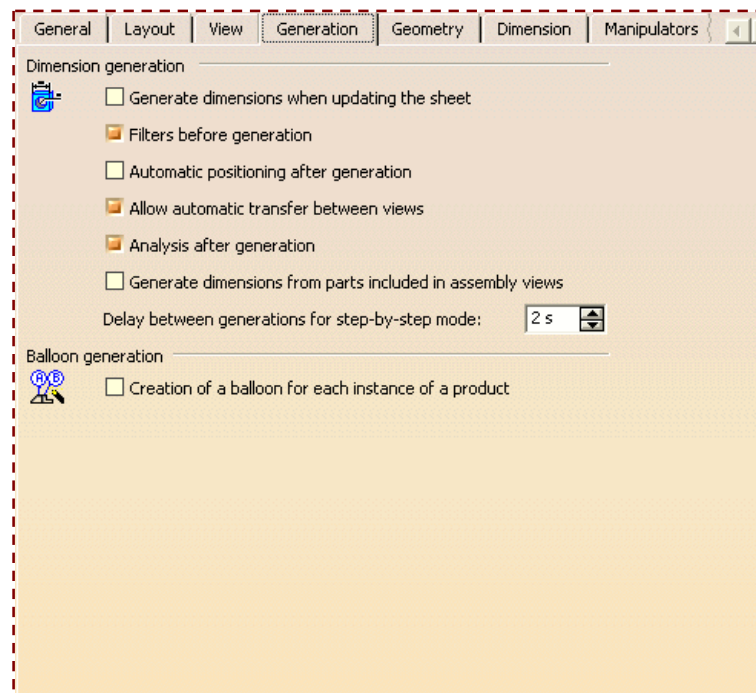
- 1) View name: Check that it is OFF since primary and projected view names are not normally necessary.
- 2) Scaling factor: Check that it is OFF since primary and projected view scale will be declared on the drawing as a global scale for the drawing.
- 3) View frame: Turn on to easily understand which 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:

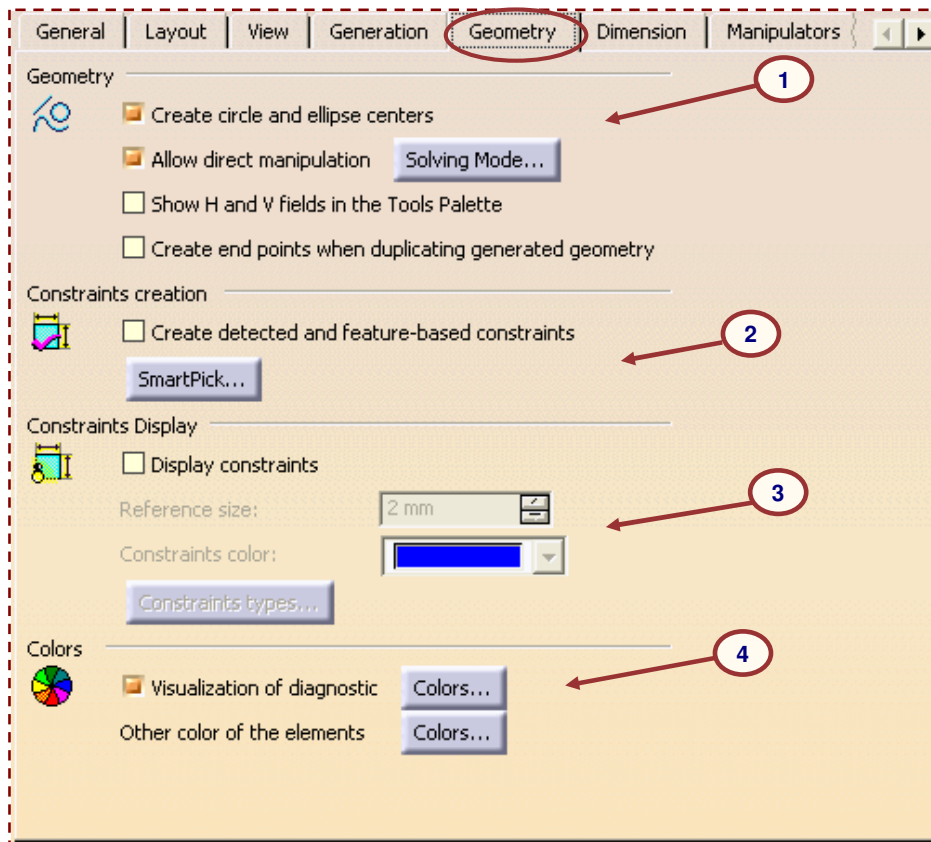
Dimension Generation: Allows dimensions to be automatically positioned after generation, automatic transfer between views and analysis of dimensions that have been generated



Time delay between dimension generation 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:



1) **Geometry**: Interactive geometry creates circle and ellipse centers and end points included with drag elements

2) **Smart Pick**: Provides an aid for creating geometry relative to existing features and geometry

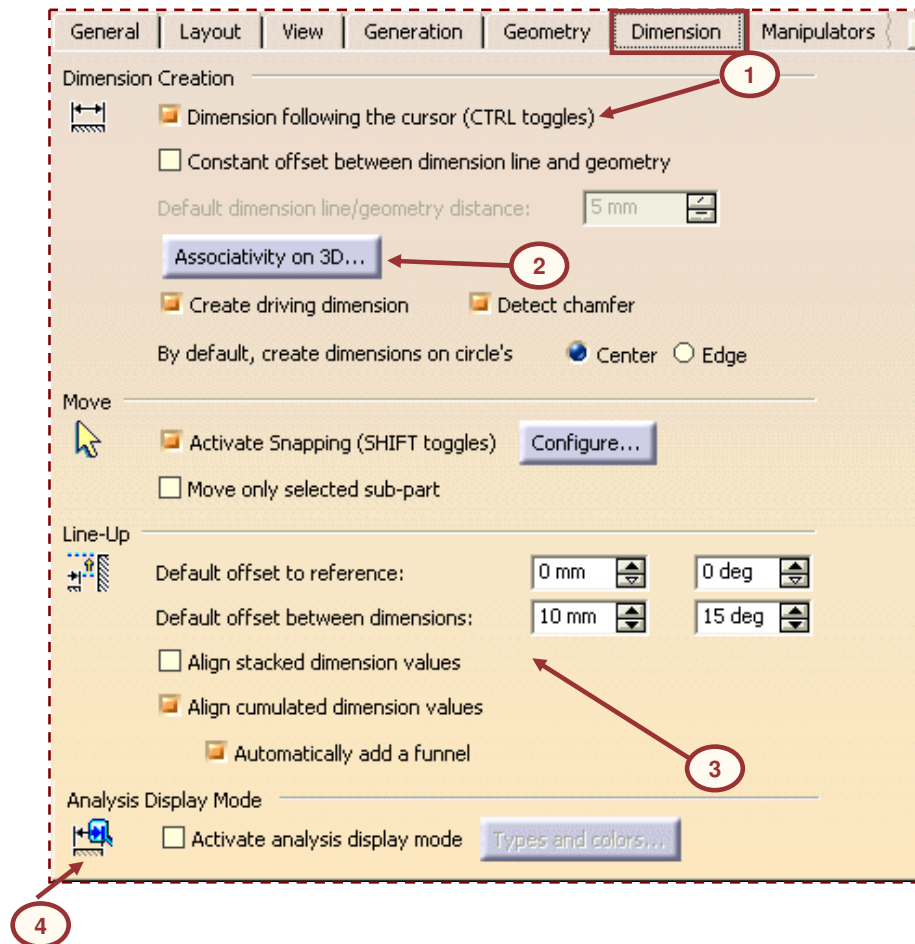
3) **Constraint Display**: Allows what constraints will be visualized and the constraints color and size

4) **Colors**: Allows you to visualize and choose colors for geometry elements

Dimension Options

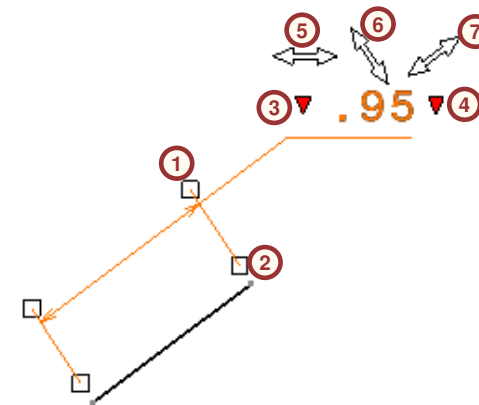
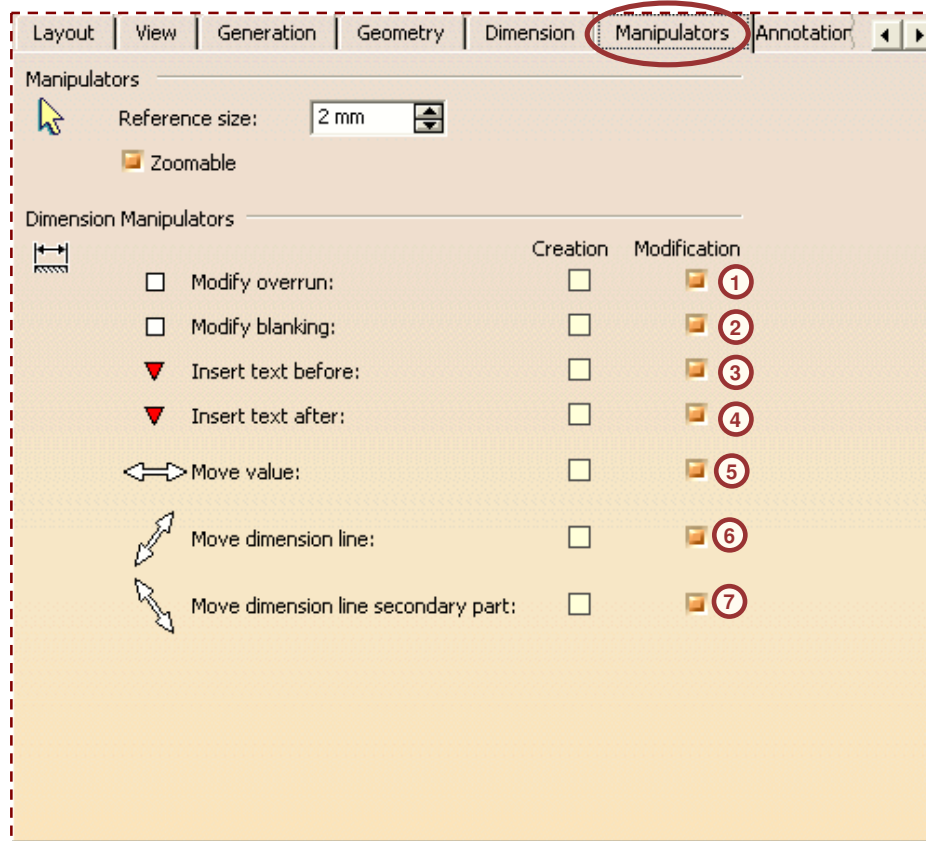
Before creating any dimensions, turn on the following Dimension options:

1. **Manual positioning at creation:** allows full freedom for dimension positioning.
2. **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.
3. **Line-Up default:** a default spacing between dimensions when a Line-Up and a reference dimension are selected
4. **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



Manipulators Options

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



Annotation Options

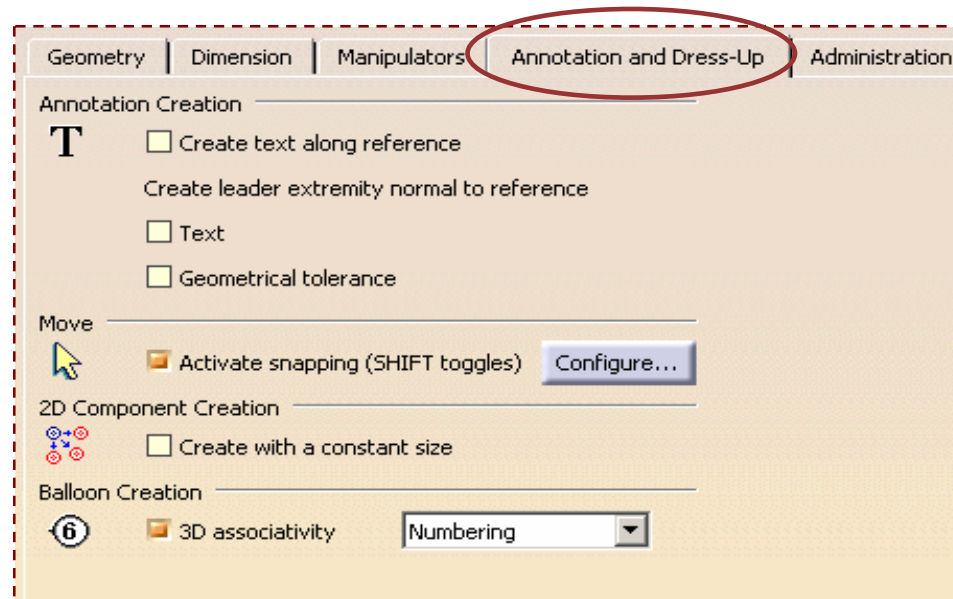
Options to enable SNAPPING to control the precise location or properties of text, leader text, or GDT during creation or modification

Annotation Options for Text, Leader Text, and GD&T

1. Settings to choose the leader default behavior
2. Stay Horizontal/Vertical with leader creation
3. Free orientation

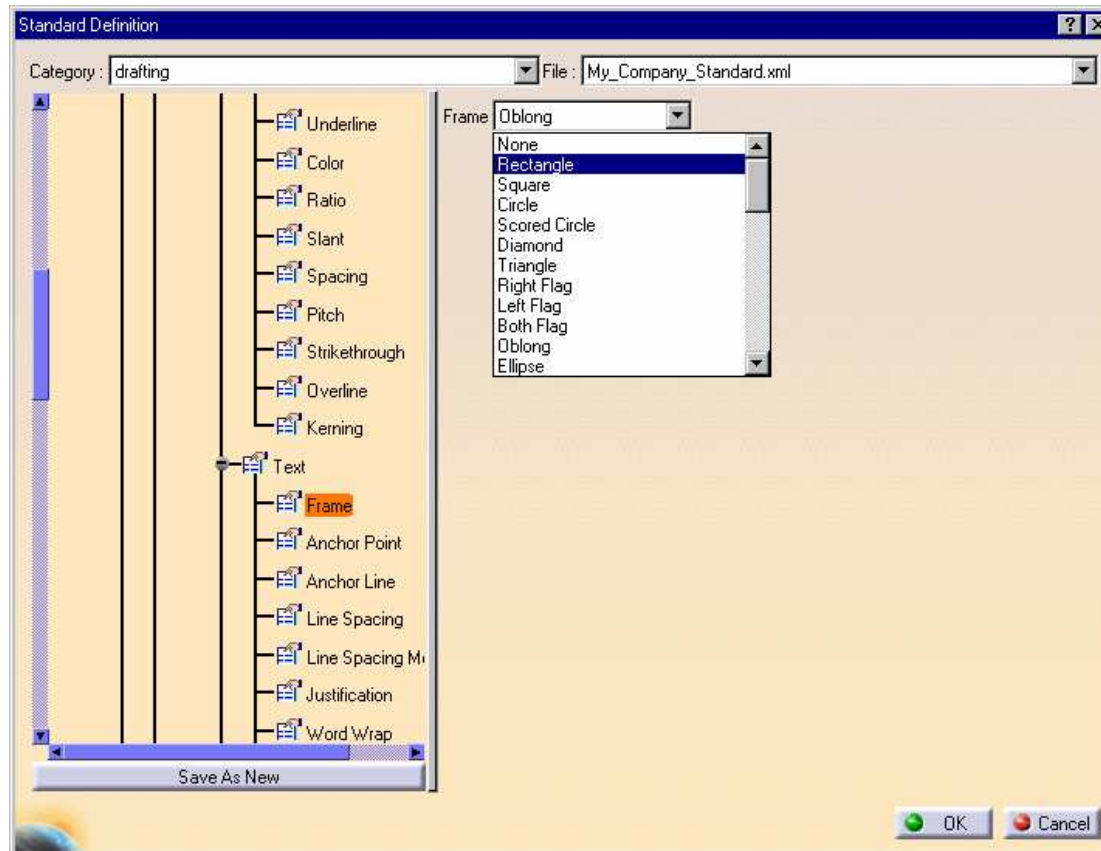
Ability to Swap the text or GDT orientation during creation

1. With the Ctrl key, swap from vertical to horizontal
2. With the Shift key, free or lock perpendicular the leader



Standard Management.

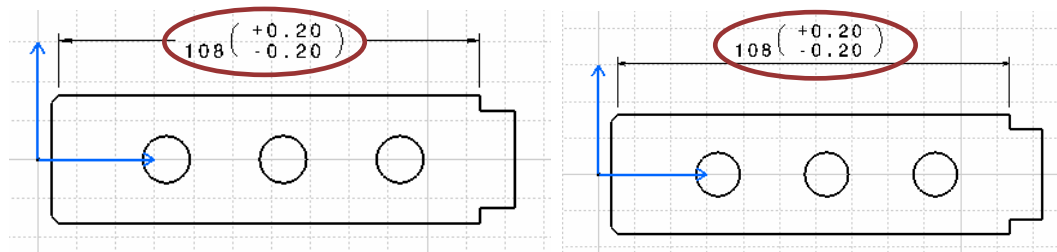
You will learn about customizing the drafting standards.



Why Standards?

Drafting standards are used to have uniformity in the drawings created. All Draftsmen referring to the same standards, ensures the uniformity in the graphic conventions drawings. Administrator can set the standards complying to the specific organization.

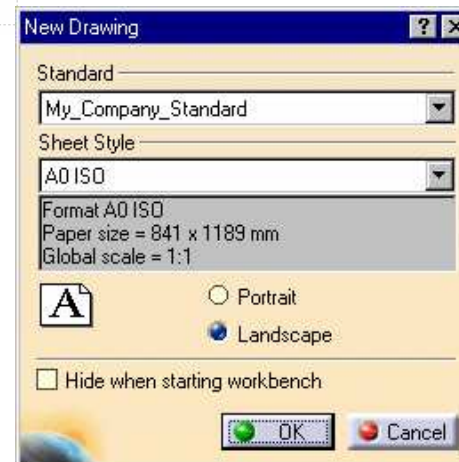
These standards govern the different parameters of the drawing elements such as Angle of Projection of the views, dimension style, line types and many more.



Dimension in ANSI standard.

Dimension in ISO standard.

Administrator can create company specific standard files and draftsmen will use these standards while creating new drawings.

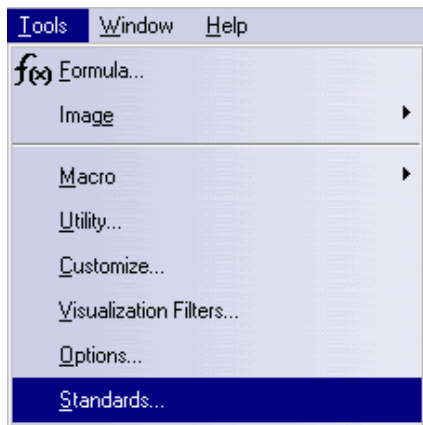


To maintain integrity of the standards only administrator can change the standards. Following slides will tell how Administrator can change the standards.

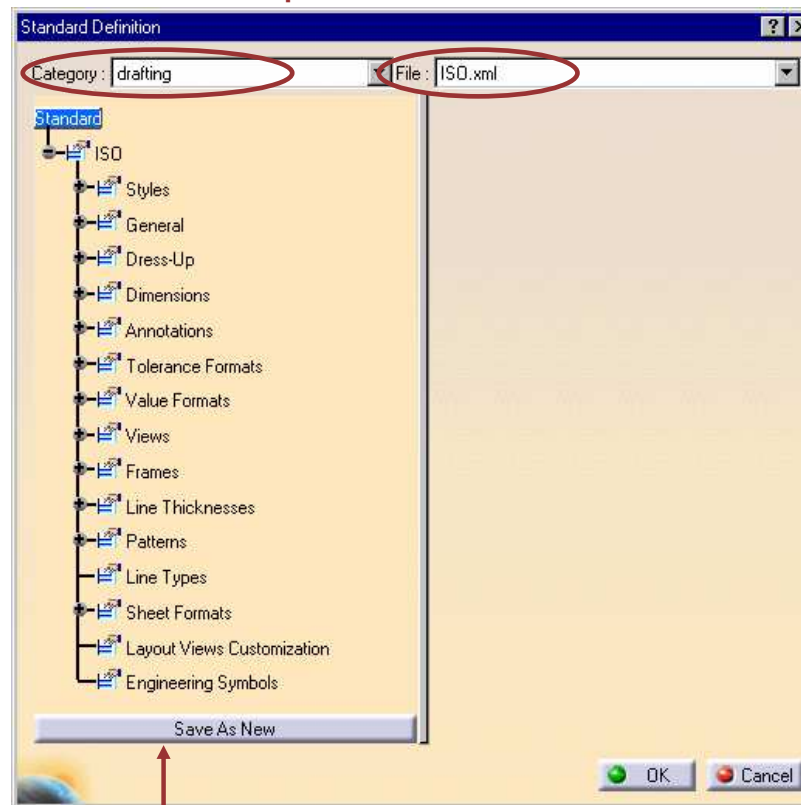
Accessing the Standards

Standards are stored in a xml file. Administrator can edit these files using an interactive editor.

- 1 Select Tools > Standards menu.



- 2 Select drafting Category and select the respective standard file.



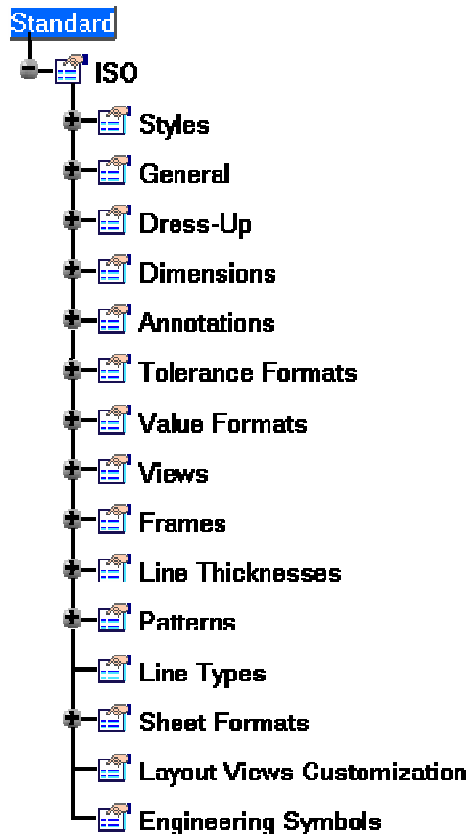
This editor provides an easy-to-use graphic interface to customize the parameters included in the standard files.



Save the changes made in the standards in a new xml file. To modify the standards you need to start the CATIA in Administration mode.

Standards Content

Standard file contains several main sections, each dealing with a specific aspect of drafting customization. Following are few important sections.



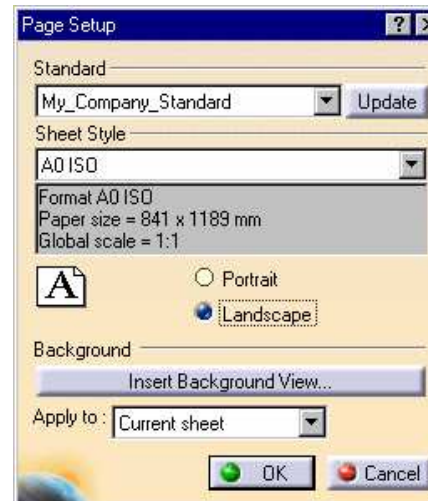
- **Styles:** Administrators can set the default values that will be applied to all properties of the elements such as sheets, geometry, annotations, dimensions, dress-up and dress-up symbols etc.
- **General:** Administrator can control and restrict the values that are available for some element properties e.g. Text fonts used by draftsmen can be controlled using this property.
- **Dress-Up:** Administrator can control the appearance of Dress-Up parameters such as threads and markup arrows.
- **Dimensions:** Using this node administrator can control the appearance of Dimensions e. g. appearance dimension value.
- **Tolerance Format:** The tolerance format parameters drive the representation of a dimension tolerance, and include parameters such as type of tolerance, font size for tolerance etc.
- **View Generation:** This node lets administrator customize settings that should be applied when generating views in a Generative Drafting context.
- **Frames:** Frames around text and annotations can be customized using this node.
- **Line Types:** Line types applied to the elements in a drawing, such as lines, curves, dimension lines etc. can be customized using this node.

Taking Into Account Standard Modification

When draftsmen create a CATDrawing document they specify the standard that will be associated with this document. The values of the parameters in the specified standard file are then copied into the CATDrawing document.

When Administrator modifies a standard file, updated standards will be applied to the new drawings created afterwards.

To take in account these modifications in previously created files, you have to open them and reapply new standard using **File > Page Setup** command.



To Sum Up

You have learned:

- How to customize the CATIA Drafting workbench
- Options to customize the drafting workbench.
- How standards are used to bring in uniformity in the drawings
- How to modify the standards