Student Notes:

CATIA V5 Training

Foils

Advanced Drafting and Customization

Version 5 Release 19
January 2009

EDU_CAT_EN_DRA_AF_V5R19

Student Notes:

About this course

Objectives of the course

Upon completion of this course you will be able to set and manage all dimension and annotation standards contained in the standard files according to company or projects needs.

Targeted audience

Draftsmen, Drafting Administrators

Prerequisites

CATIA V5 Mechanical Design Fundamentals, knowledge of VB scripting

16 hours

Student Notes:

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Hints & Tips on Interactive Drafting

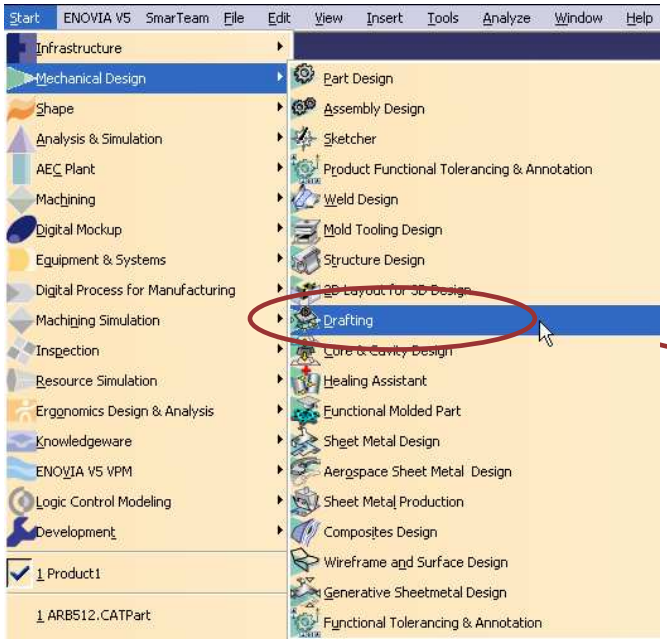
- Accessing the Workbench
- Toolbars Description
- Hints & Tips on Dress Up Commands
- Hints & Tips on Dimension Commands
- Hints & Tips on Text Commands

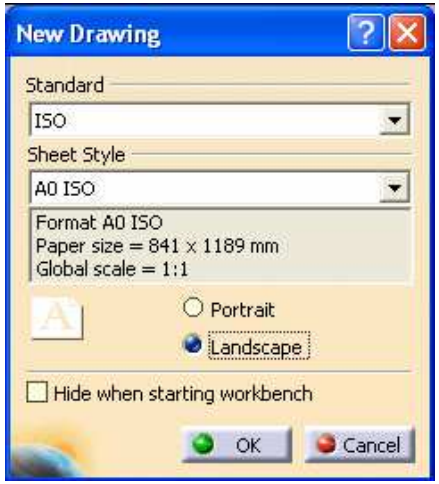
Student Notes:

Accessing the Workbench (1/6)

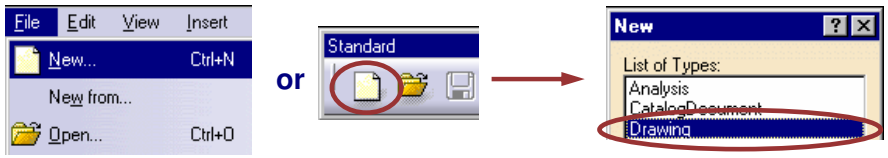
You can access the Drafting Workbench anywhere from :

1 The Start Menu :





2 The File Menu or the New icon in the Standard toolbar :

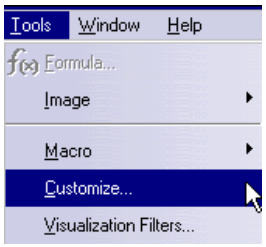


Student Notes:

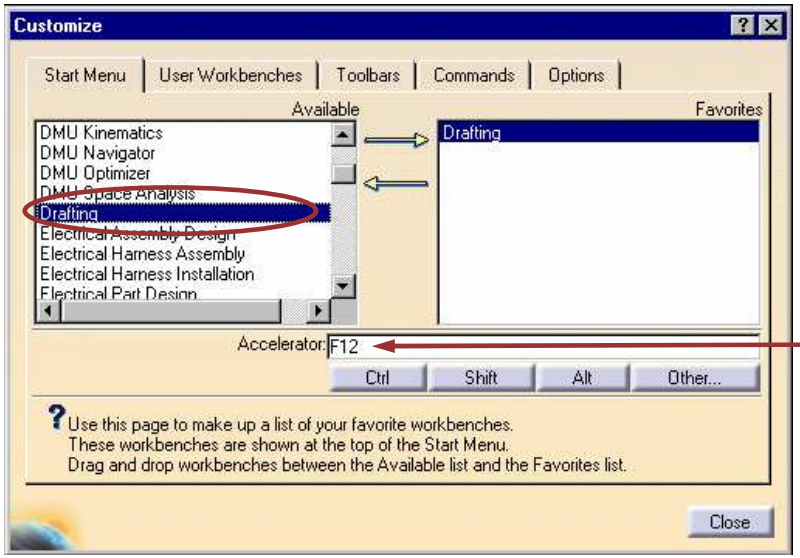
Accessing the Workbench (2/6)

You can also create shortcut to access faster the workbench :

1 **Go to Tools Menu and select Customize.**



2 **Select the Drafting Workbench and add it to the Favorites by clicking the right arrow.**



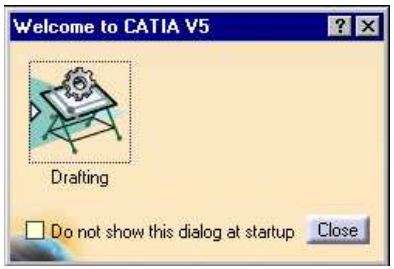
You can add an Accelerator to access directly the workbench by pressing a key.

Student Notes:

Accessing the Workbench (3/6)

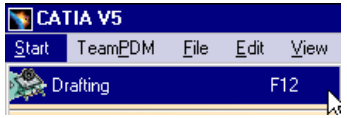
3 **Now you can access the workbench by 3 new ways :**

When you start CATIA



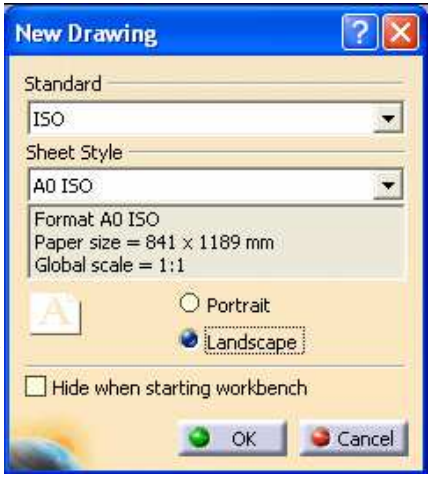
or

Using the Start Menu



or

Pressing F12 Key in this case



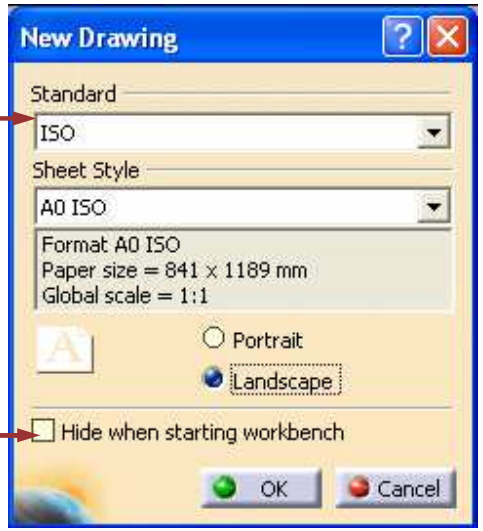
Student Notes:

Accessing the Workbench (4/6)

The New Drawing Panel :

You can choose different Standards : ISO, ANSI, ASME,JIS or your company standard

Use this Option to avoid displaying this panel when accessing the Drafting Workbench.

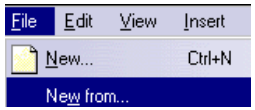


Student Notes:

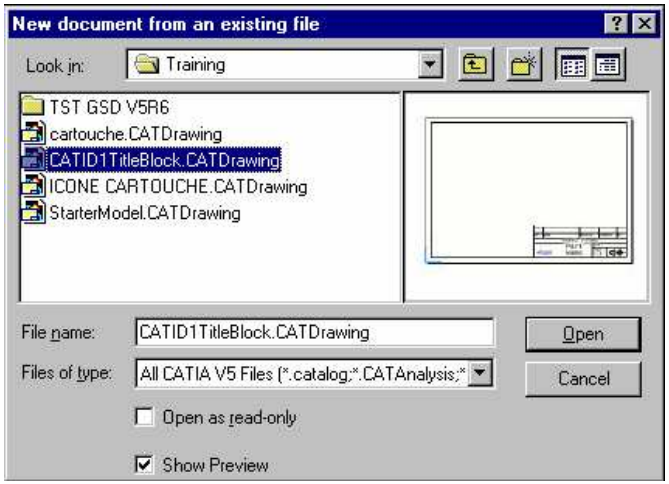
Accessing the Workbench (5/6)

You can use the New From function to access the Drafting Workbench by using an existing drawing with a predefined format :

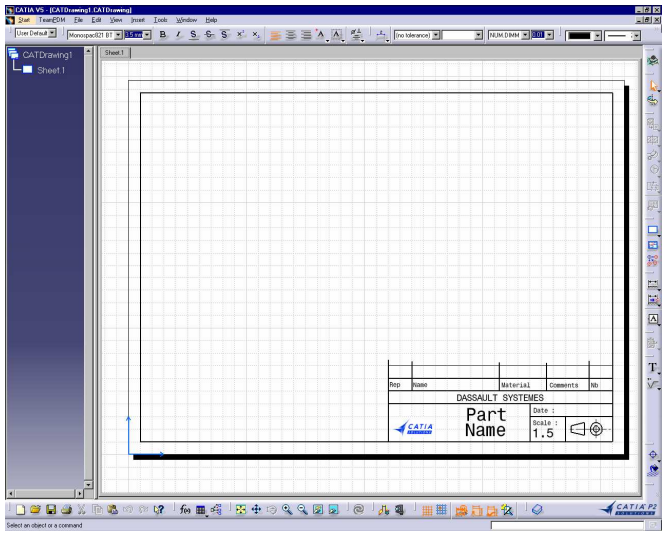
1 **Go to File Menu and select New From.**



2 **Select an existing drawing to use it as reference.**



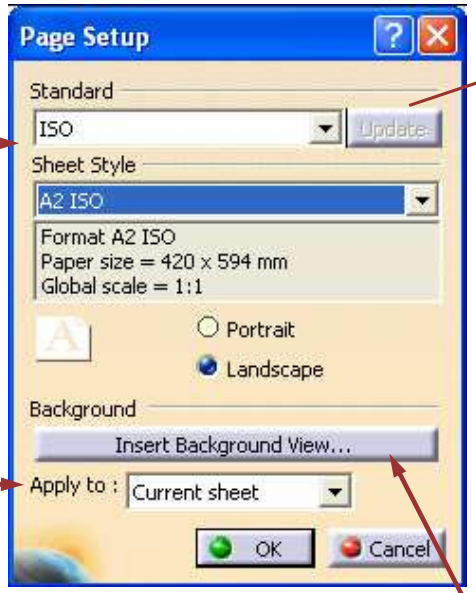
This is a new Drawing and you can save it without keeping any links with the selected file.



Student Notes:

Accessing the Workbench (6/6)

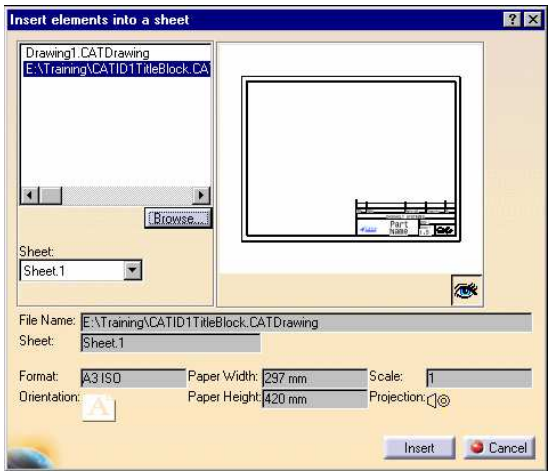
When you have accessed the Drafting Workbench you can still make modifications on the Format, the Orientation or the Background using the File/Page Setup command:



Change the Format

Change the Standard and update the drawing with this new Standard.

You can choose to apply the modifications on the current sheet or on all the sheets of the drawing.

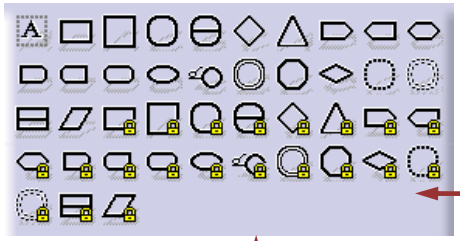


You can insert a background view from a selected sheet of an other drawing document.

Student Notes:

Toolbars Description (1/6)

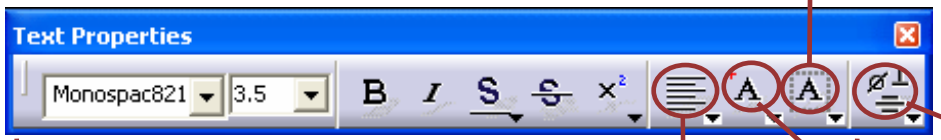
1 **The Text Properties Toolbar :**

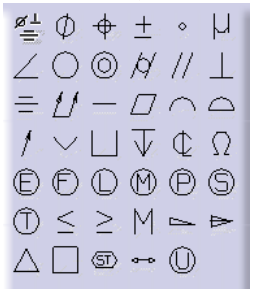


You can select a frame around the text or a dimension.

Be careful, these two specifications are applied at the dimension creation.

You can use different Fonts as Postscript, True type and V4 type.





Default values are defined in the standard styles. After text creation you can change the default values by selecting a font, a size, an alignment, etc.

Text alignment

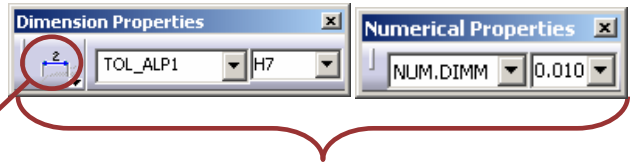
Select the anchor point position of the text.

You can add some symbols in the text. It allows you to add a prefix symbol to a dimension.

Student Notes:

Toolbars Description (2/6)

2 **The Dimension Properties and Numerical Properties Toolbars:**



You can modify the dimensional representation of the dimension.

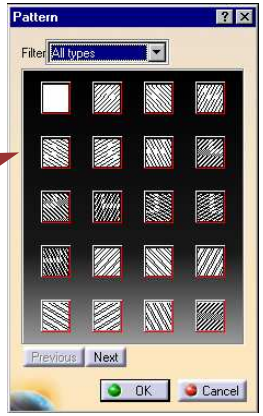
Default values are the one defined in the drafting standards. After dimension creation you can choose different values.

Student Notes:

Toolbars Description (3/6)

3 The Graphic Properties Toolbar :

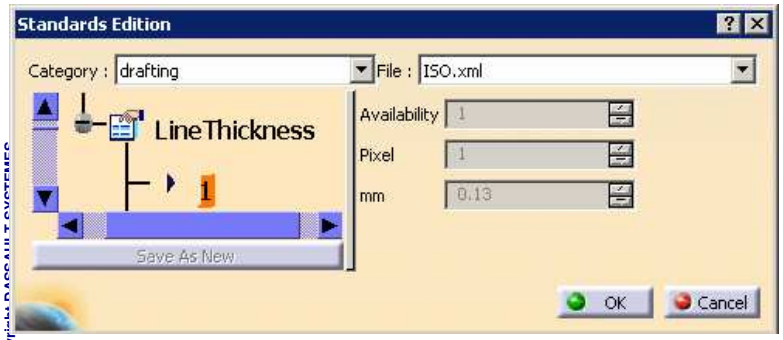
You can copy the format of one object on selected objects.



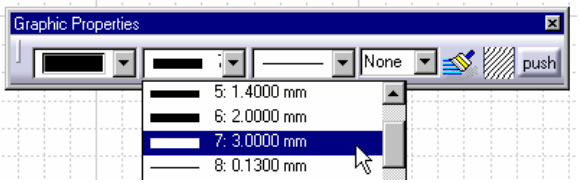
You can select here the line color, the line thickness, the line type and point type. When you choose a specification, it will be applied on all the graphics you will create; it becomes a specification by default.

You can also select a graphic then choose a specification to apply on it. In this case, this specification won't be applied on the next created graphics.

You can select here a pattern for the Area Fill function.



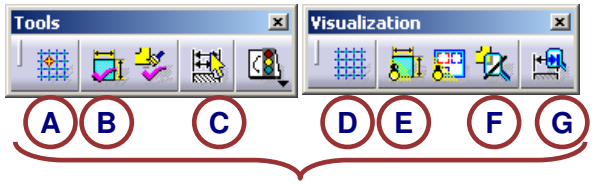
In Standards, you can add new thickness values



Toolbars Description (4/6)

4 **The Tools and Visualization Toolbars:**

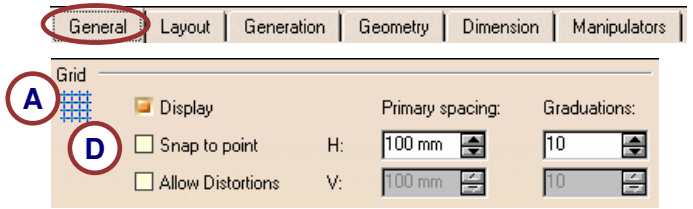
These Toolbars are specific because they contain some standard functions and displays specific information in accordance with the command you select. Most of time, you have to drag & drop the Toolbar on the screen to display it entirely.

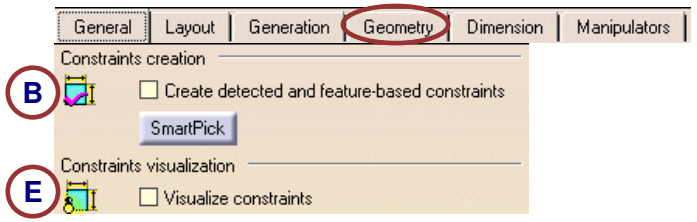


Here is the standard Tools toolbar. It's a kind of shortcut of the menu Tools – Options – Drafting.

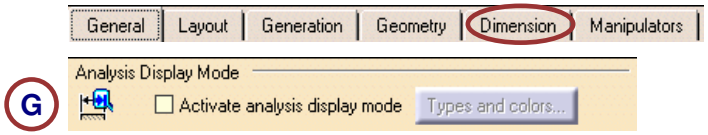
C **Dimension system selection mode.**

F **You can differentiate 2D elements (Interactive workbench) from generated elements (Generative workbench) within the same view.**





If you want the constraints to be created, you must have selected the Constraints creation Option before to create geometry.



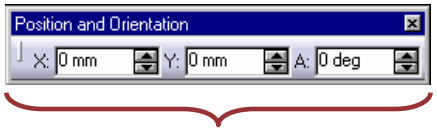
Student Notes:

Student Notes:

Toolbars Description (5/6)

5 **The Position and Orientation Toolbar:**

This Toolbar is not located on a side of the workbench in the default set-up. You have to go in the View/Toolbars command to select it.



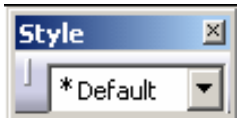
You can apply a translation and a rotation to a view, a text or a datum.

For a view the reference point is the origin point and for a text or a datum, the reference point is the anchor point.

Student Notes:

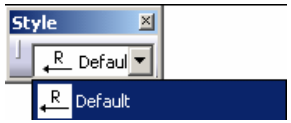
Toolbars Description (6/6)

6 The Style Toolbar :

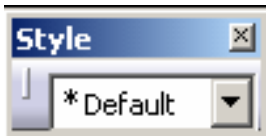


The styles available in the toolbar depend on what your administrator specified in the standards.

1 Create a circle and add a radius dimension. The Style toolbar displays the styles available for radius dimensions. In our example, only one style is available, therefore it will be used by default.



2 In the Graphic Properties toolbar, select another color, red, for example=>In the Style toolbar, an asterisk appears in front of the selected style: this asterisk indicates that the style of the element you are creating has been overloaded compared to the style which is defined in the standards.



3

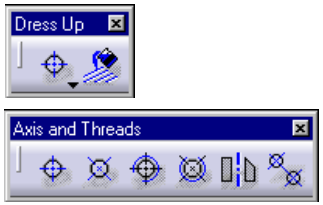
- Then you can either revert to the standard-defined values (i.e. reset the toolbar properties to their original values) by re-selecting this style from the Styles toolbar, and then clicking to validate and end the dimension creation. The asterisk will disappear.
- Or you can apply the modified style by clicking to validate and end the dimension creation. For the purpose of this scenario, do this.

The dimension is created with the selected style, as defined in the standard and overloaded by the properties you changed.

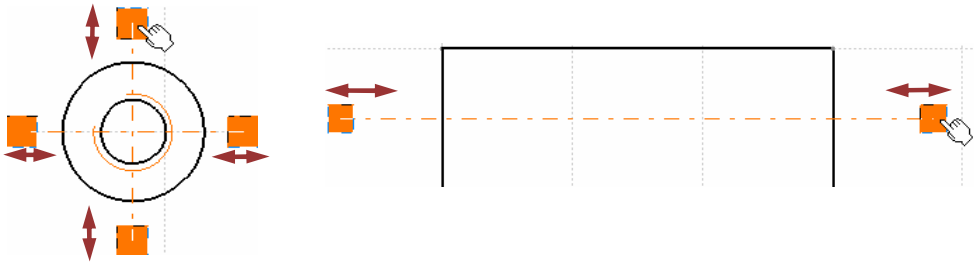
Student Notes:

Hints & Tips on Dress Up Commands

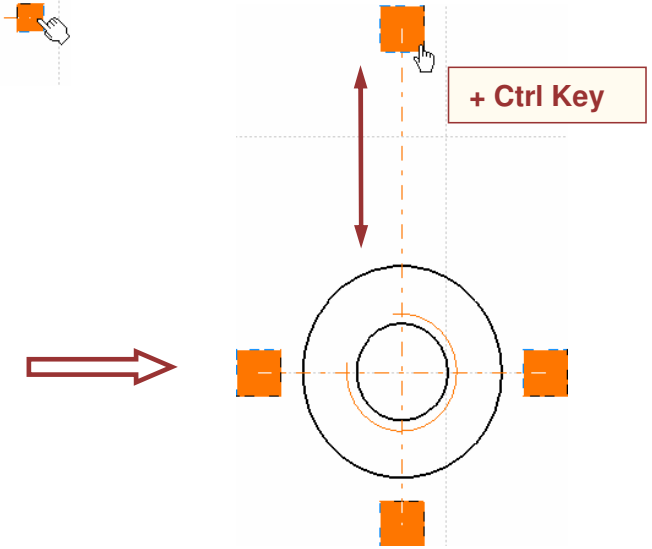
How to move Axis or Centre Line.



If you select a manipulator you will move the Centre Line and the Axis Line along all the directions with the same length. The modification will be symmetric.



If you want to move just one line along only one direction and not symmetrically, you can do it by selecting the concerning manipulator and holding on the Ctrl Key while you are moving the manipulator.



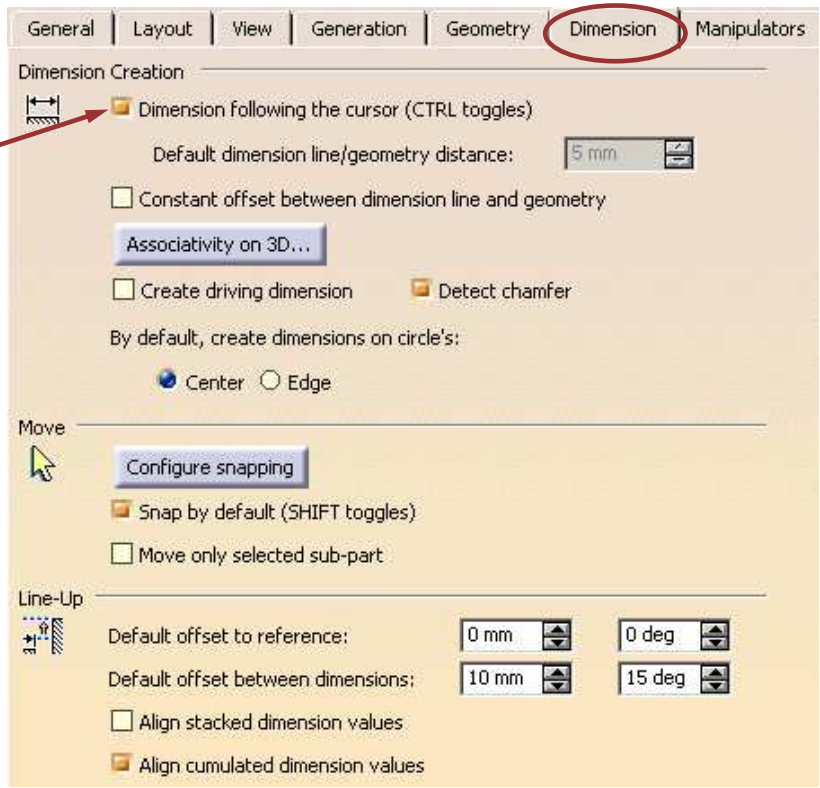
Student Notes:

Hints & Tips on Dimension Commands (1/13)

The way that you can set and manipulate dimensions depends on the options that you have checked in the Tools Options Drafting panel.

1 The Dimension following the cursor Option:

By selecting this option, when you create a dimension, the dimension line is dynamically positioned following the cursor. If you want to deactivate temporarily this mode, you can do it by pressing the Ctrl Key.

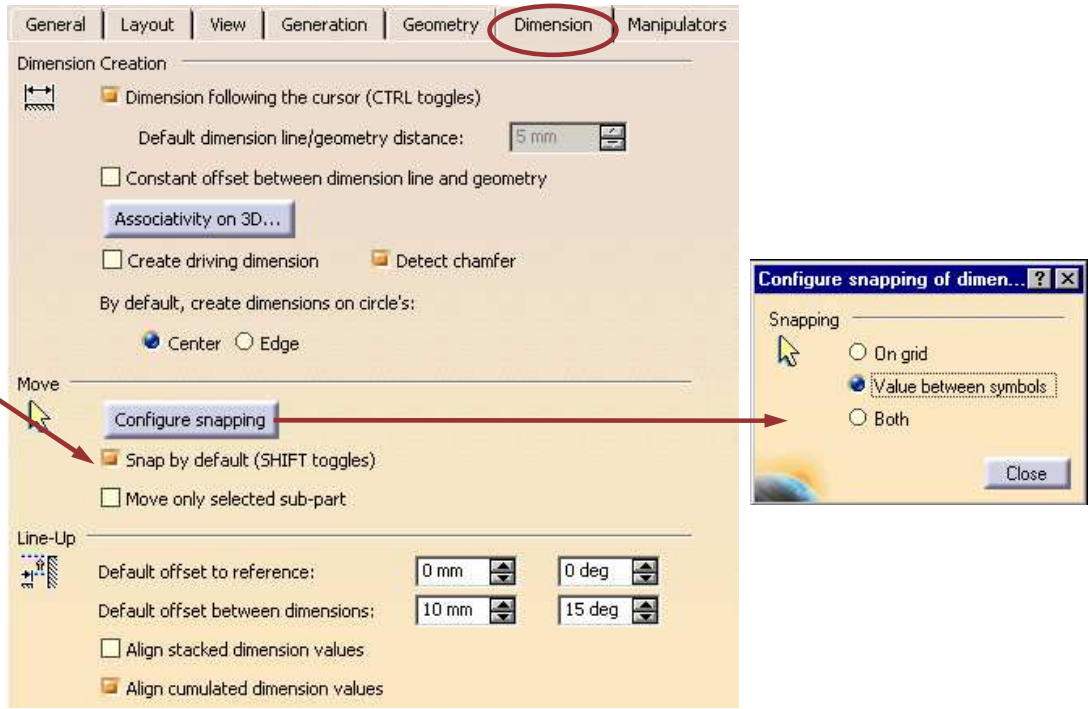


Student Notes:

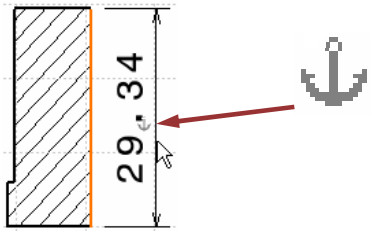
Hints & Tips on Dimension Commands (2/13)

2 The Snapping Option :

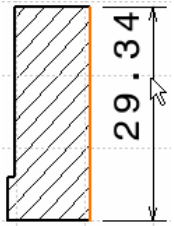
By selecting this option, when you create a dimension, the dimension is snapped on the grid or/and the dimension value to be located at its default position between symbols. If you want to avoid this, you can do it by holding on the Shift Key.



With the *Value between symbols* option, the dimension value will remain in the middle of the two extension lines only if the mouse cursor stays between the two extension lines.



+ Shift Key



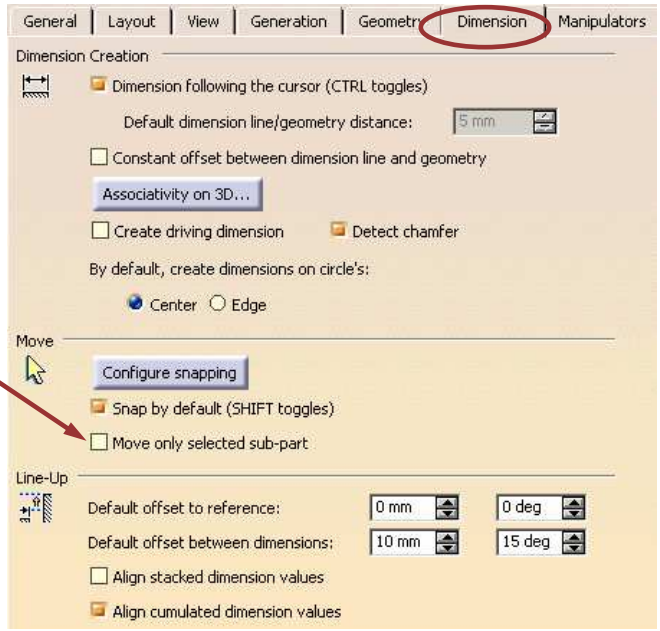
You can position the dimension value where you want.

Student Notes:

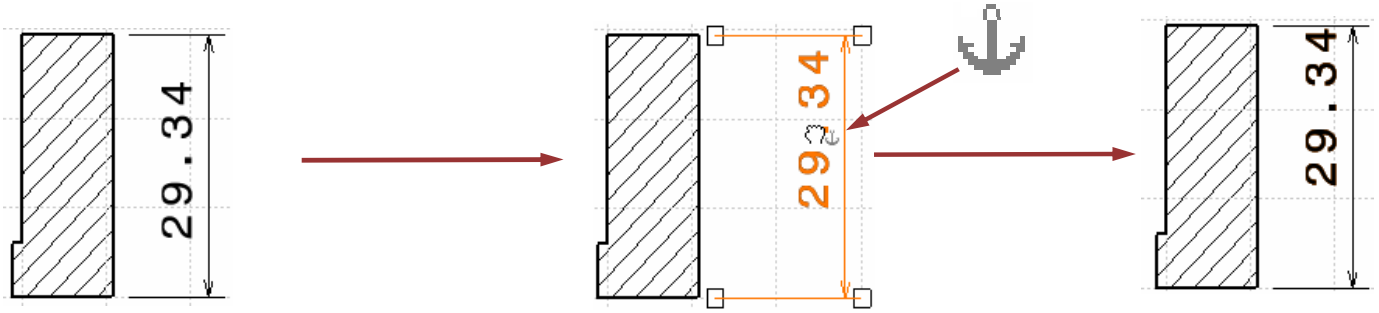
Hints & Tips on Dimension Commands (3/13)

3 **The Move only selected sub-part Option :**

The *Move only selected sub-part* option allows you to move only a dimension sub-part like a text, a line, the dimension value, etc.



For example, you can move the dimension value even if you have selected the *Snap by default* option.

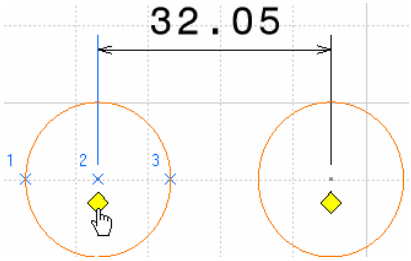


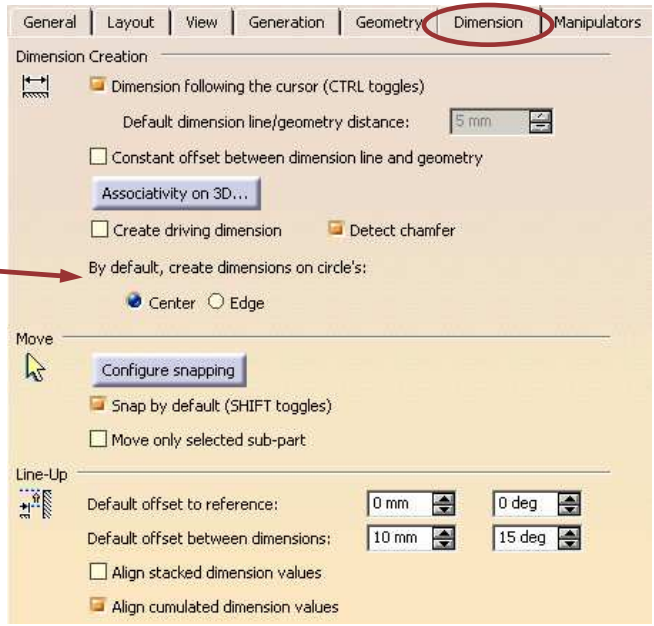
Student Notes:

Hints & Tips on Dimension Commands (4/13)

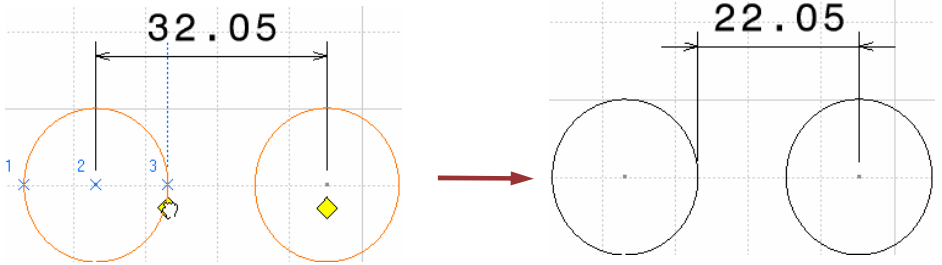
4 The Dimension Circle Option :

By selecting this option, the dimension you will create between a circle and another element will be either on the circle center or on the circle edge. If you want to change this option during the creation, you can do it by using the yellow manipulator which is displayed.





Select the manipulator with MB1 and drag it to choose one of the 3 proposed locations. If you have chosen the *Dimension following the cursor* option use CTRL KEY.

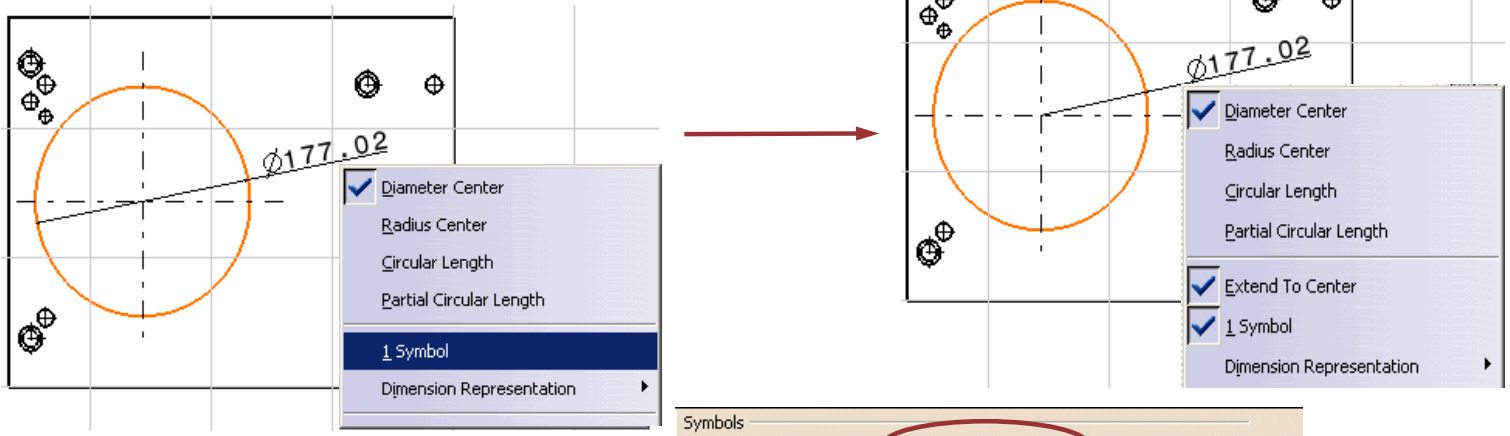


Student Notes:

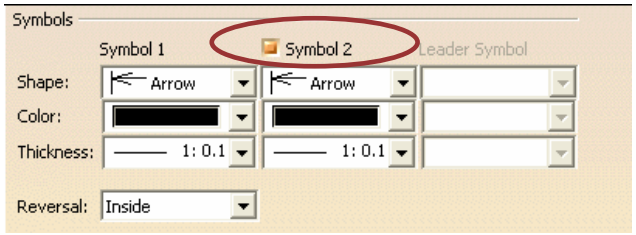
Hints & Tips on Dimension Commands (5/13)

5 The Diameter Dimension Options :

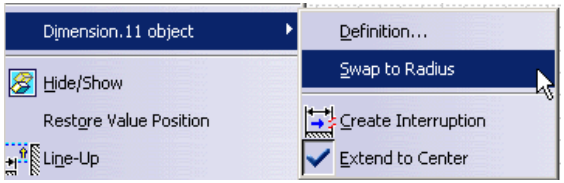
During the dimension creation step, you can switch between one-symbol or two-symbols dimension.



Once the dimension has been created, you must use the Properties menu (Dimension Line tab) to specify whether you want to use one or two-symbols.



After the Diameter Dimension creation, you can swap to Radius Dimension by using the contextual menu.

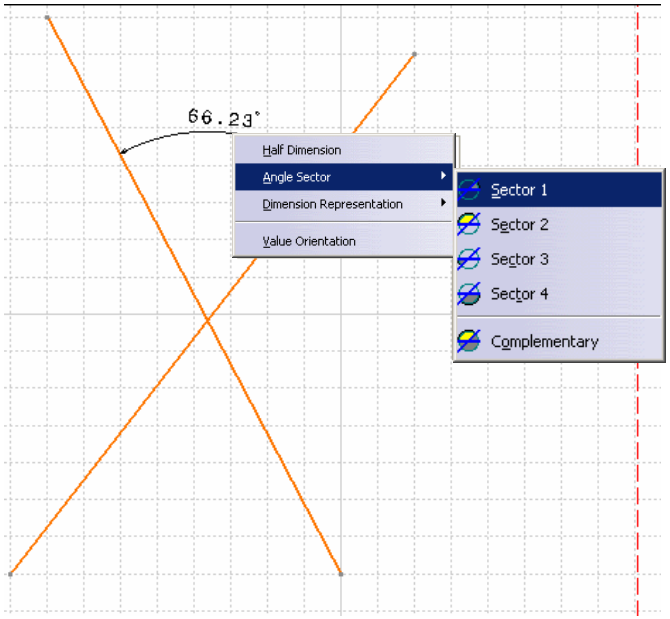


Hints & Tips on Dimension Commands (6/13)

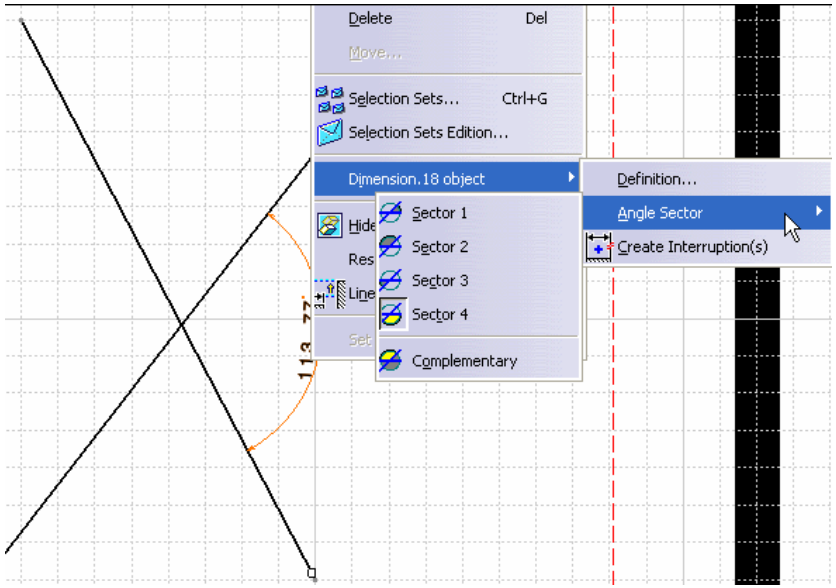
Student Notes:

6 The Angle dimensions Creation :

During angle dimension creation you can move the dimension to a new sector by using the *Angle Sector* contextual menu



You can also edit the angle sector of an existing angle dimension, by right-clicking the angle dimension and selecting the Dimension_name object -> *Angle Sector* command from the contextual menu.

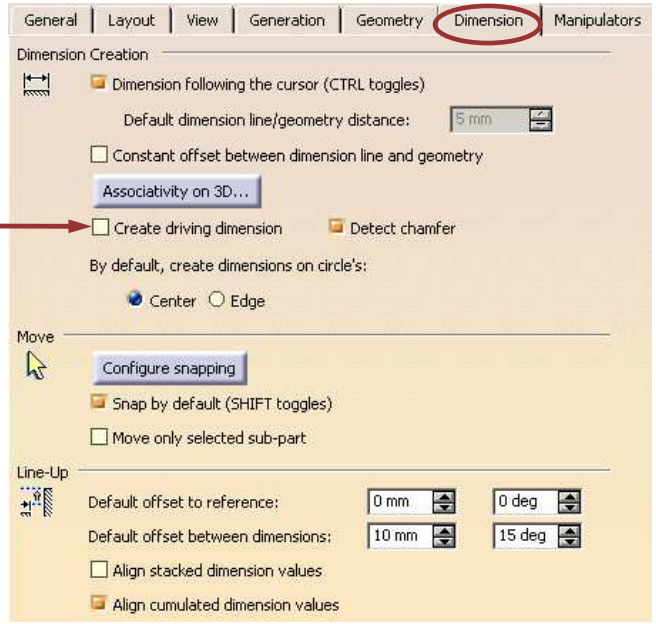


Student Notes:

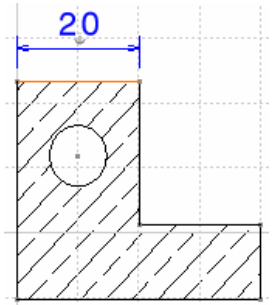
Hints & Tips on Dimension Commands (7/13)

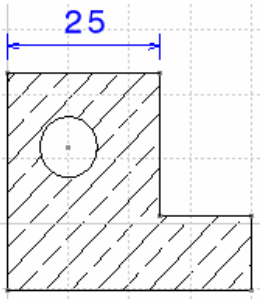
7 **The Create driving dimension Option :**

Select this Option if you want to create driving dimensions by default.



During the dimensions creation, you can set directly the value in the Tools Palette toolbar.

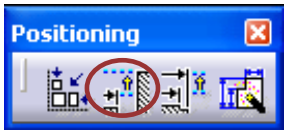




Student Notes:

Hints & Tips on Dimension Commands (8/13)

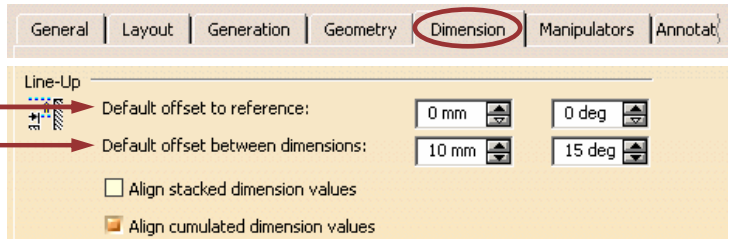
8 **The Line-Up Options:**

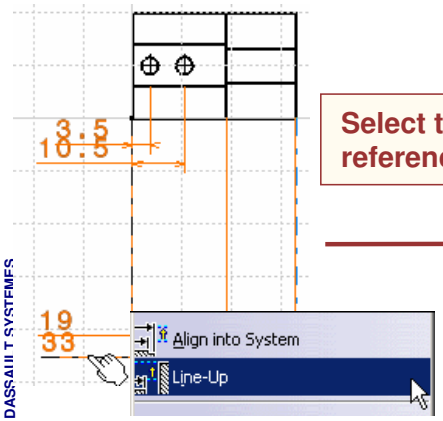


The *Line-Up* function allows you to position length, distance, radius, diameter an angle dimensions according to a given reference.

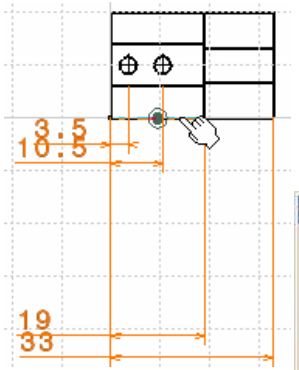
To define the distance and the angle in relation with the reference element.

To define the distance and the angle between each dimension you line up.

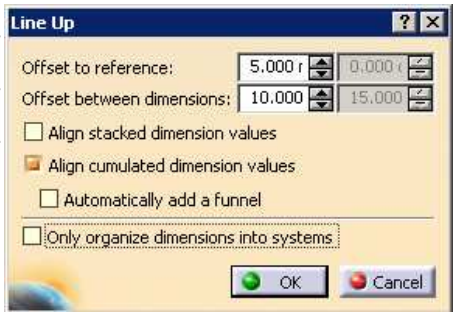


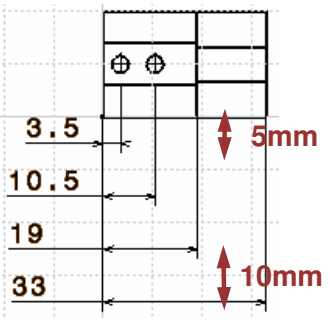


Select the reference element



Define de Line-up parameters

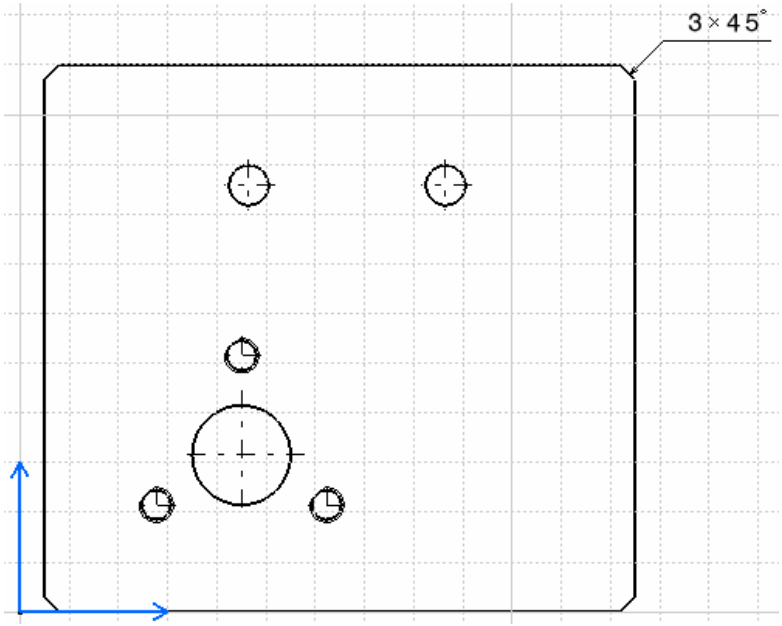




Hints & Tips on Dimension Commands (9/13)

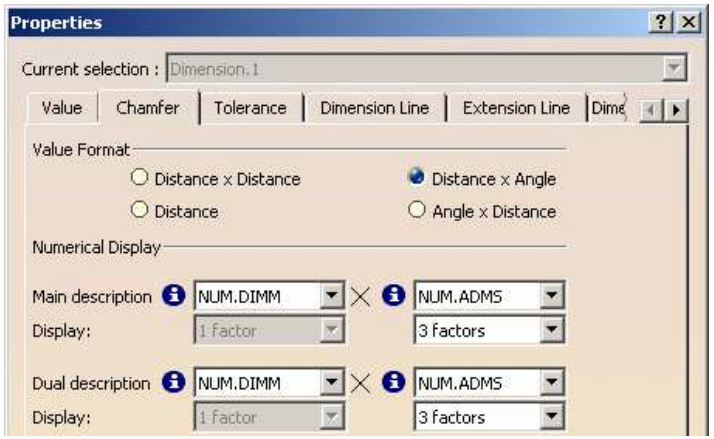
Student Notes:

9 Creating Chamfer Dimensions :



During the creation of the chamfer dimension, you can define the format of the dimension and the representation mode in the Tools Palette toolbar.

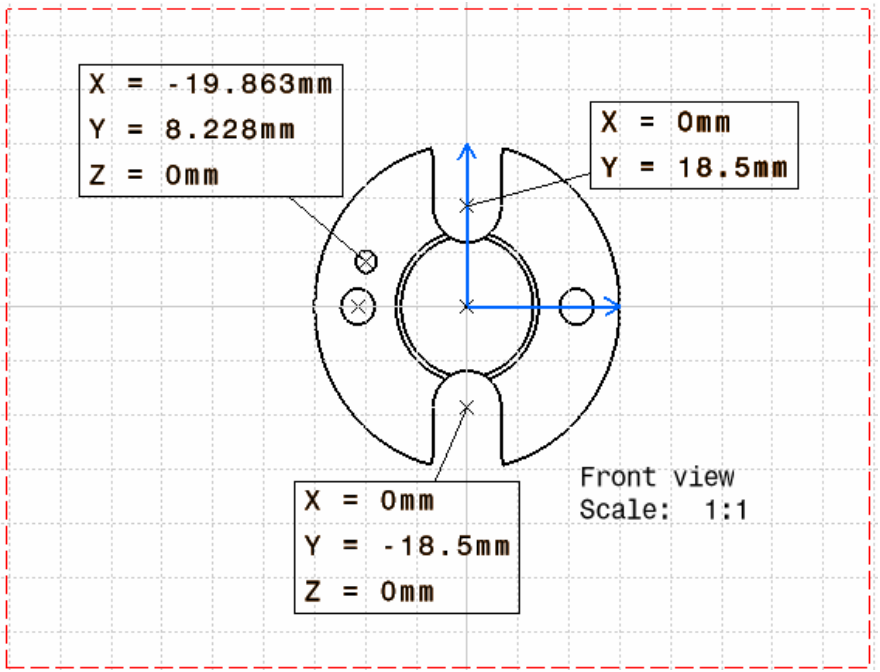
You can also modify those properties afterwards by accessing the Chamfer tab in the dimension properties.



Student Notes:

Hints & Tips on Dimension Commands (10/13)

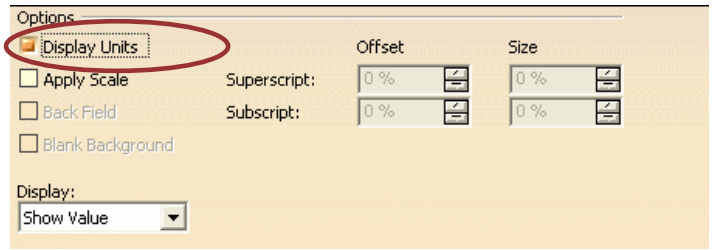
10 **Creating Coordinate Dimensions :**



The Tools palette appears with two options: 2D Coordinates lets you create 2D (x, y) coordinate dimensions for interactive geometry, 3D Coordinates lets you create 3D (x, y, z) coordinate dimensions for generative geometry.

Coordinates are relative to absolute axis system except for view created selecting a 3D local axis system.

If you need to hide the coordinate dimension's unit, you can do so by editing the properties of the coordinate dimension.

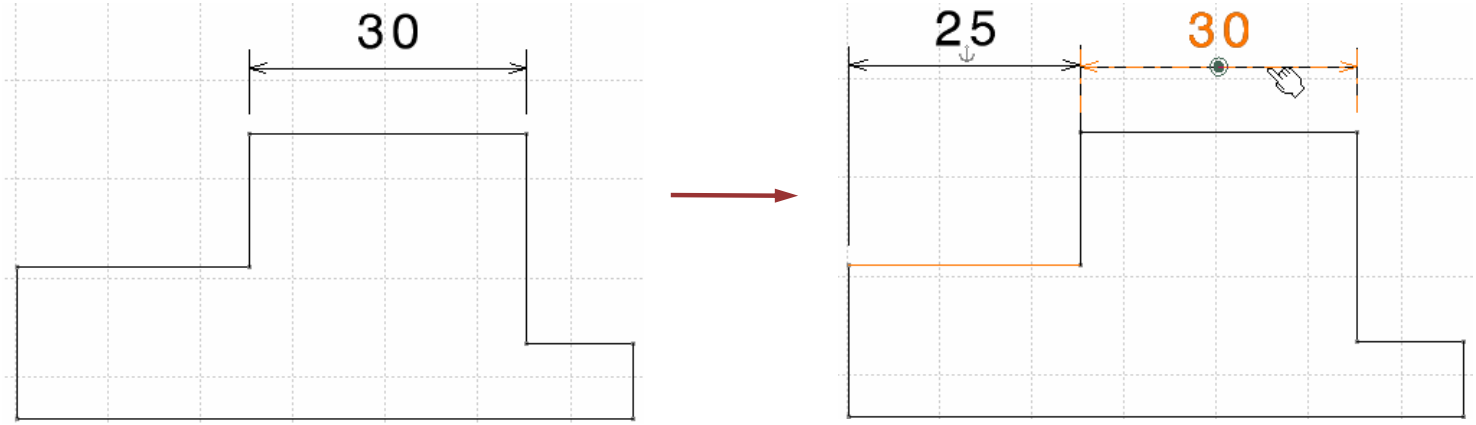


Student Notes:

Hints & Tips on Dimension Commands (11/13)

11 **How to align dimensions during the creation :**

Select the 30 dimension while you are creating the 25 dimension if you want to align the two dimensions.

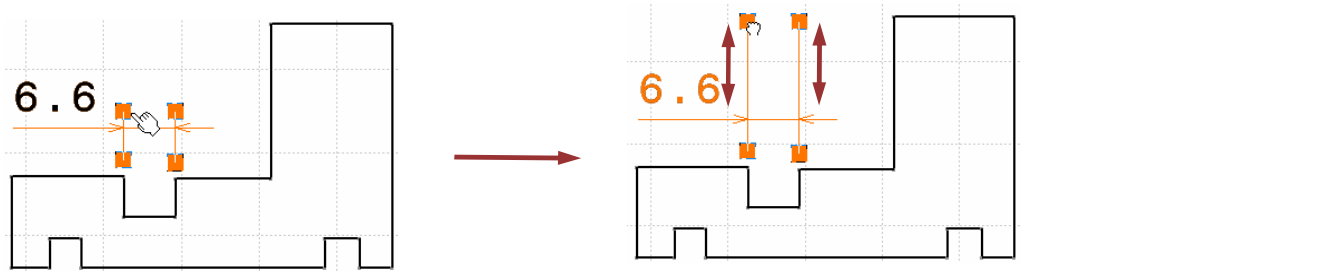


Student Notes:

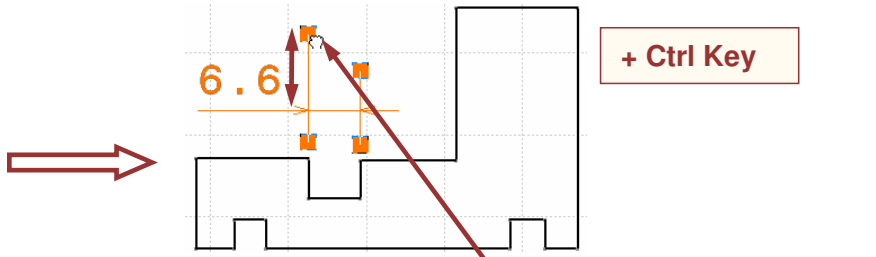
Hints & Tips on Dimension Commands (12/13)

12 **How to move extension lines :**

If you select a manipulator, you will move the both extension line with the same length.



You want to move just one extension line, you can do it by selecting the concerning manipulator and holding on the Ctrl Key while you are moving the manipulator.



+ Ctrl Key

You can Double Click the manipulator to set the Overrun value and you can choose to apply this value to one or both sides.

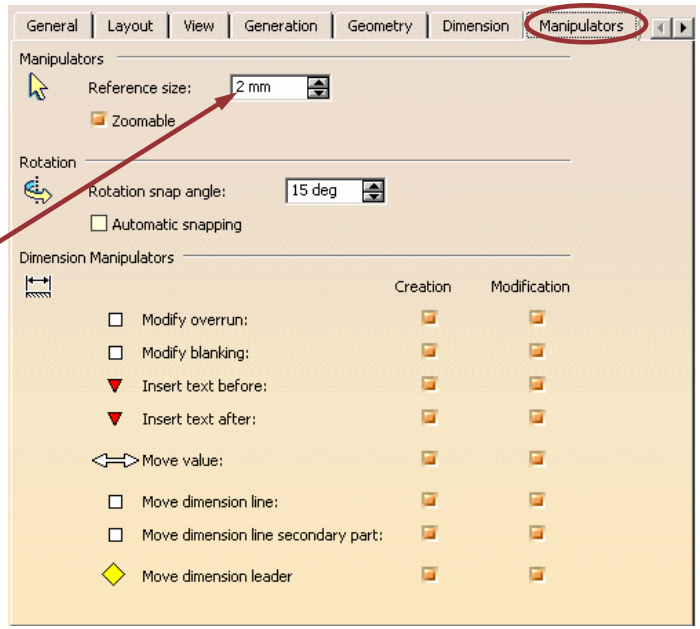
You can also use the Properties menu.

Student Notes:

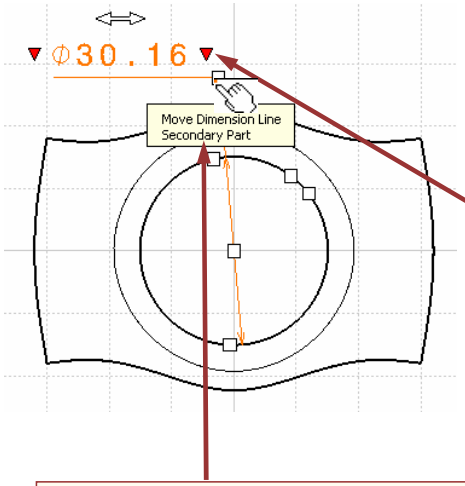
Hints & Tips on Dimension Commands (13/13)

13 **Using Manipulators to modify dimensions :**

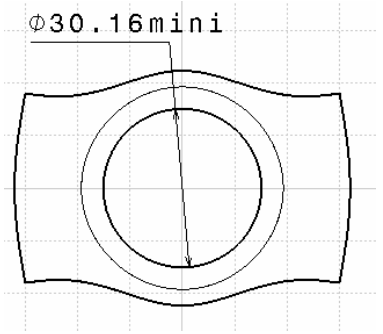
In the Tools Options Drafting panel, you select which Manipulators you want to display during the dimension creation and when you want to modify a dimension.



You can modify the size of all the Manipulators.



You can Click on the Manipulator to add directly Text After.

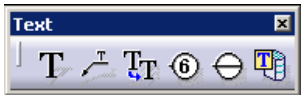


If you let the mouse cursor on a manipulator, it will appear a contextual help.

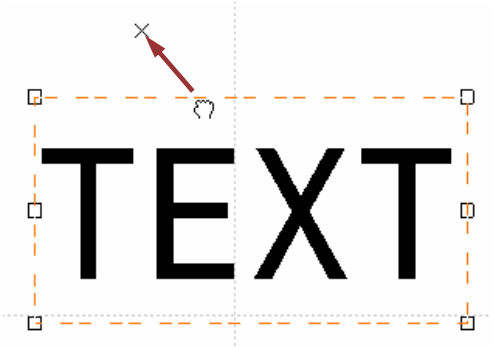
Student Notes:

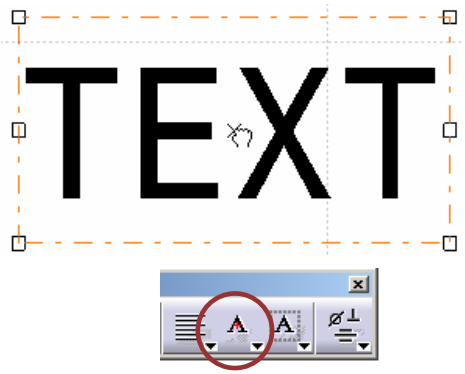
Hints & Tips on Text Commands (1/14)

1 How to snap a Text on a point :



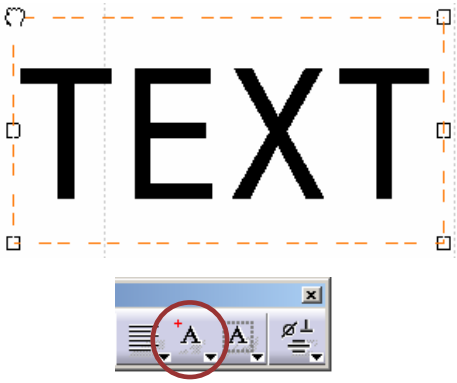
Select the Text and drag it so as to the mouse icon encounter the point.





The result depends on the Anchor point of the Text box.

If you modify the Anchor point of the selected text after having snapped it, the Text will move in function.



Student Notes:

Hints & Tips on Text Commands (2/14)

2 How to associate a Text to an existing element :

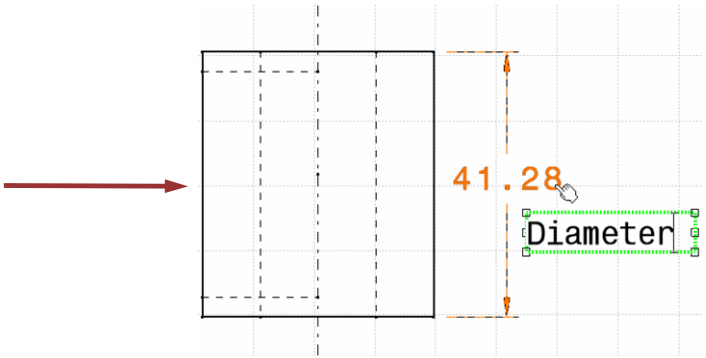
You have two ways to associate a Text to an existing element:

- by selecting the geometry at the creation
- by using the Positional link function

Remark: You can create a text in a view not up to date, but you can not associate it to any geometry.

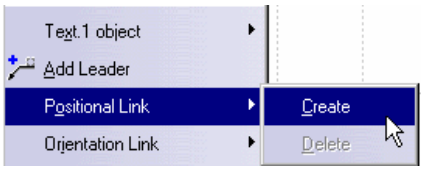
Click on the Text icon and select the dimension to which you want to associate the text.

If you move the dimension, the text will follow and keep the same relative position.



Or

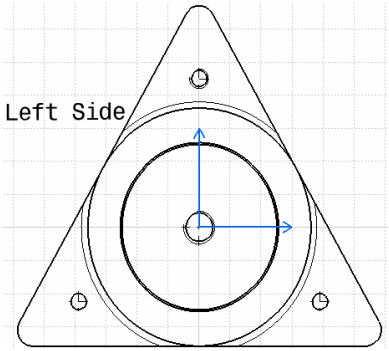
Select any part of a created text and use Positional Link – Create in the contextual menu to select the dimension to which you want to associate the text.



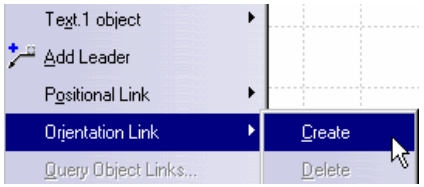
Student Notes:

Hints & Tips on Text Commands (3/14)

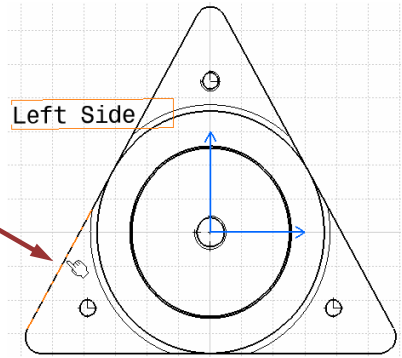
3 How to orientate a Text along an existing element :

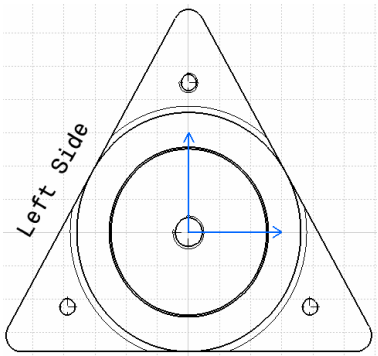


Select any part of a created text and use *Orientation Link – Create* command.



Select the line.



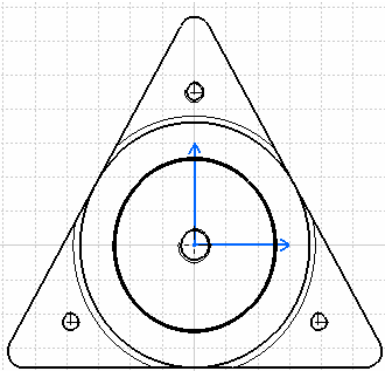


If you change the line orientation, the text will follow.

Student Notes:

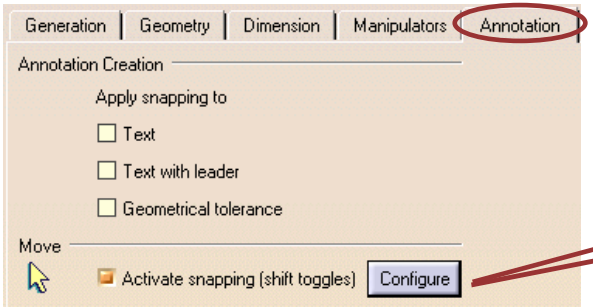
Hints & Tips on Text Commands (4/14)

3 How to orientate a Text along an existing element :

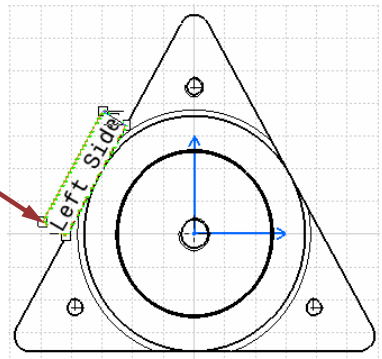


At the creation of the Text, you can also use the Shift Key to orientate directly the Text if you have selected the Activate snapping option in Tools Options.

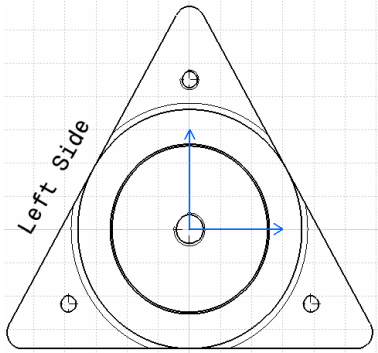
If you select Text in Apply snapping to Option, it's the contrary. The Text will be orientated along the geometry by default and you will have to press the Shift Key to orientate it horizontally.



Press the Shift Key when you select the line.



If you change the line orientation, the text will follow.

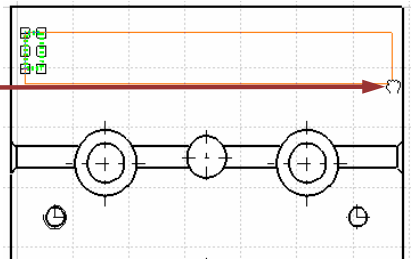


Student Notes:

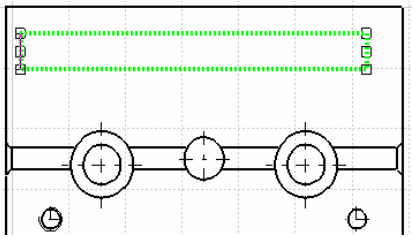
Hints & Tips on Text Commands (5/14)

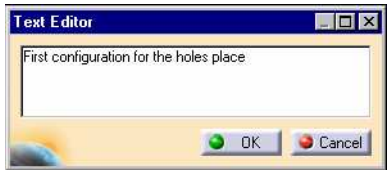
4 **You can predefine the width of the text box at the creation :**

Select the Text icon then draw a box using MB1.

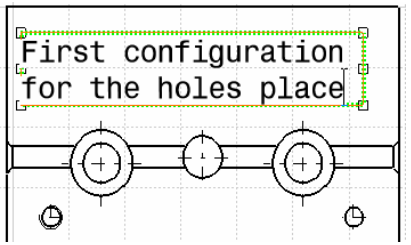


The width of the Text box is now defined.





If your text is too long, it will be added an other line automatically and the width will be kept.

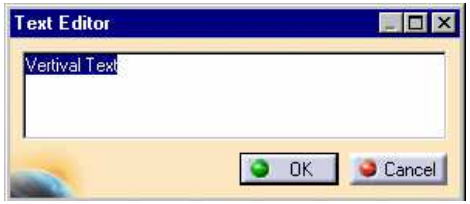


Student Notes:

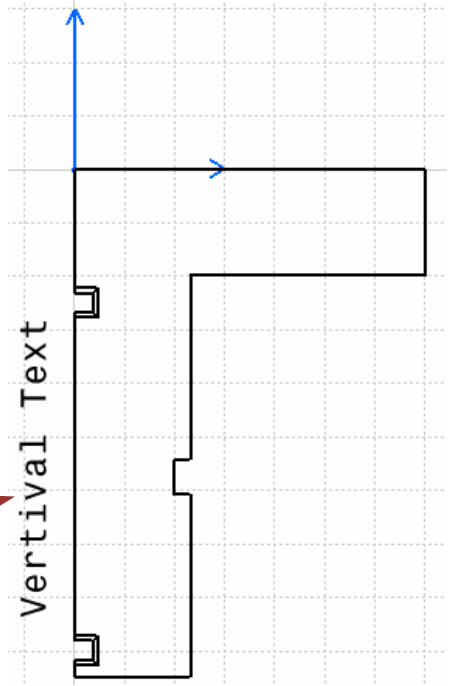
Hints & Tips on Text Commands (6/14)

5 You can create Texts vertically directly during the creation :

Select the Text function then press CTRL Key when you select the position of the Text.



You directly write the Text vertically.



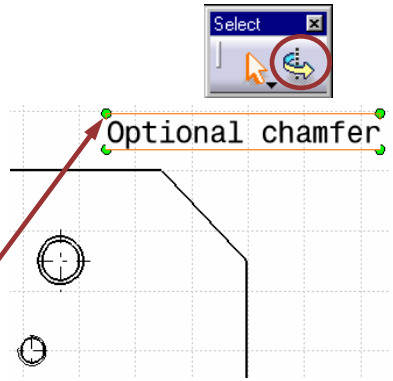
This is also valid for Text with Leader and Geometrical Tolerance.

Student Notes:

Hints & Tips on Text Commands (7/14)

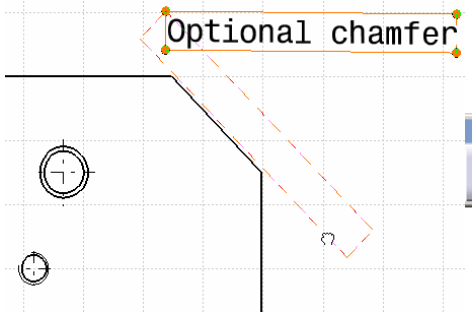
6 You can rotate the text box manually :

Select the Free Rotation icon in the Select toolbar then select the Text box.

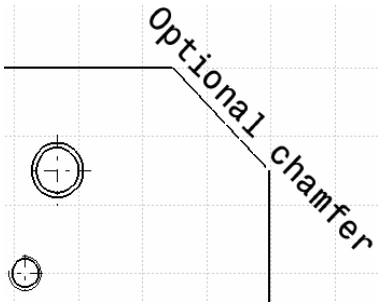


You can select any of the manipulators and perform rotation.

The center of rotation will be the Anchor point of the Text box.



You can change the center of rotation by modifying the Anchor point.



If you keep pressing the Shift Key during the rotation, you will rotate by step of 15 degrees.

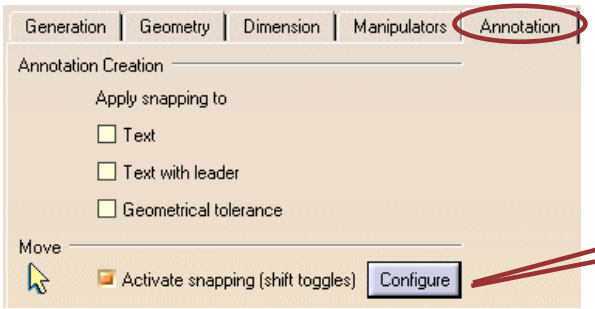
Student Notes:

Hints & Tips on Text Commands (8/14)

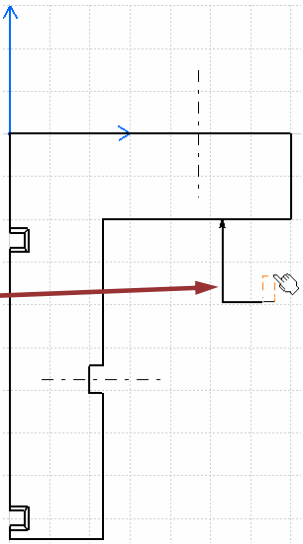
7 How to orientate Text with Leader or Geometrical Tolerance at the creation:

You can orientate directly a Text with Leader (or Geometrical Tolerance) perpendicularly to an element by using the Shift Key if you have selected the Activate snapping option in Tools Options.

If you select Text with Leader (or Geometrical Tolerance) in Apply snapping to Option, it's the contrary. The Text with Leader (or Geometrical Tolerance) will be orientated perpendicularly to the geometry by default and you will have to press the Shift Key to orientate it differently.



When you position the Text with Leader (or Geometrical Tolerance), press Shift Key to orientate it perpendicularly to the geometry.



If you move the geometry, this properties will be kept.

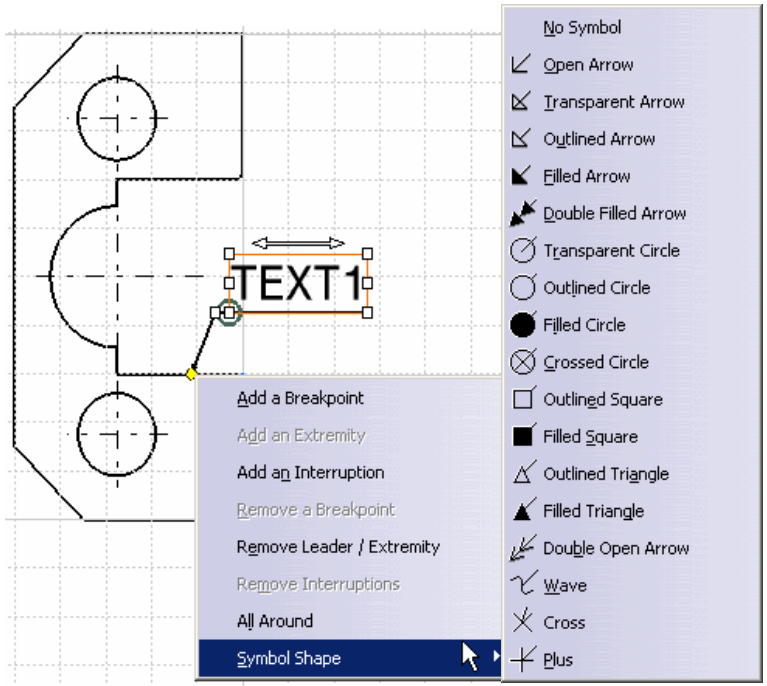
Student Notes:

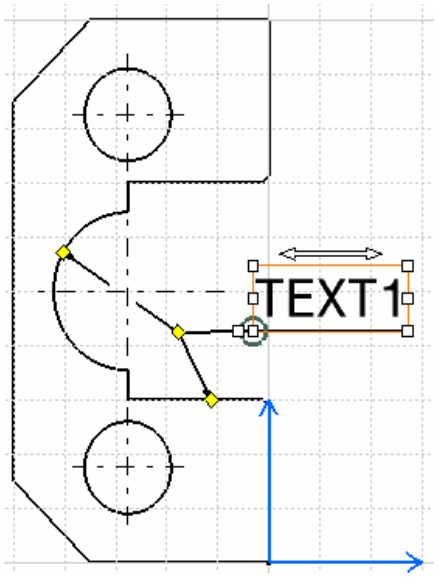
Hints & Tips on Text Commands (9/14)

8 How to modify Text with Leader :

These modifications are also valid for all annotations with Leader.

You can add/remove a Breakpoint by using the contextual menu on the yellow manipulator.





If you have selected the Snap to Point option, the new break point will be created on the grid and you will be able to move it only on the grid.

Use Shift Key to deactivate this option or snap on the grid if you haven't selected the option.

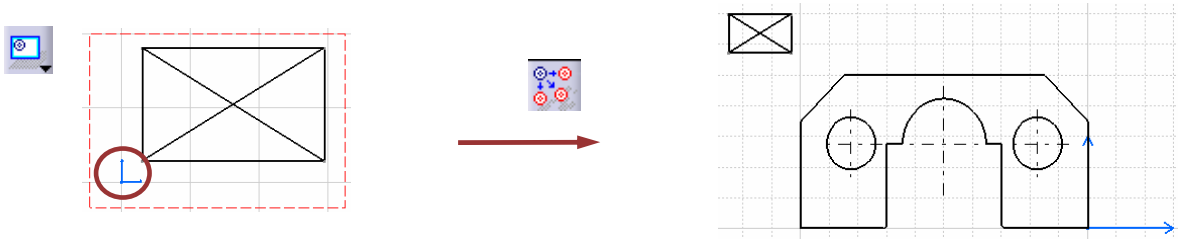
You can create/remove multiple leader extremities, create/remove interruptions. In those cases, by selecting a yellow manipulators, you will act on the ascendant branch.

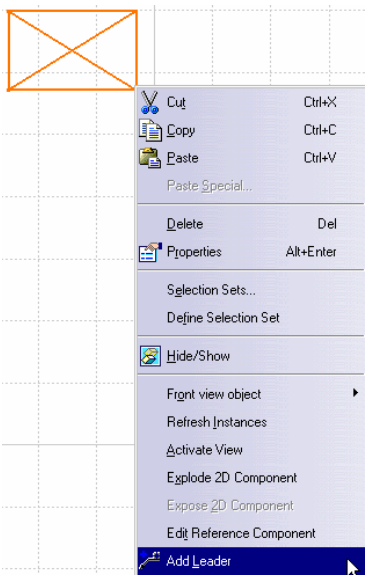
Student Notes:

Hints & Tips on Text Commands (10/14)

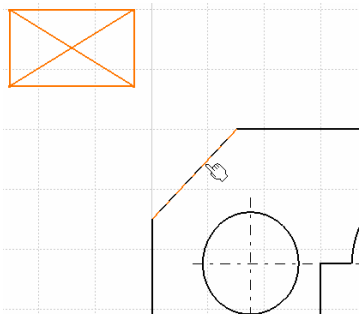
9 How to add leader on 2D Component :

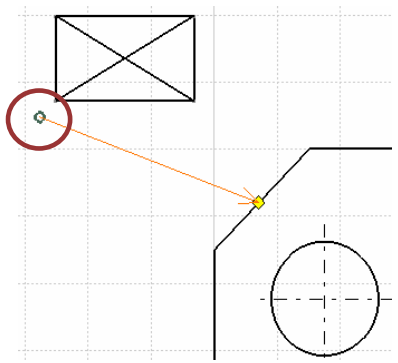
You have created a 2D Component in a Detail Sheet and you instantiate it in another sheet.





You can add a leader on the 2D Component by using the contextual menu and selecting Add Leader. Then select the geometry where you want the arrow to point.



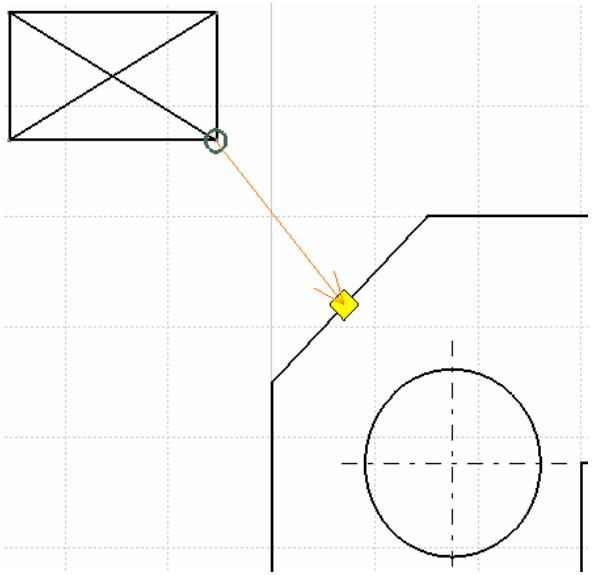


By default, the origin of the leader is at the origin point of the 2D Component.

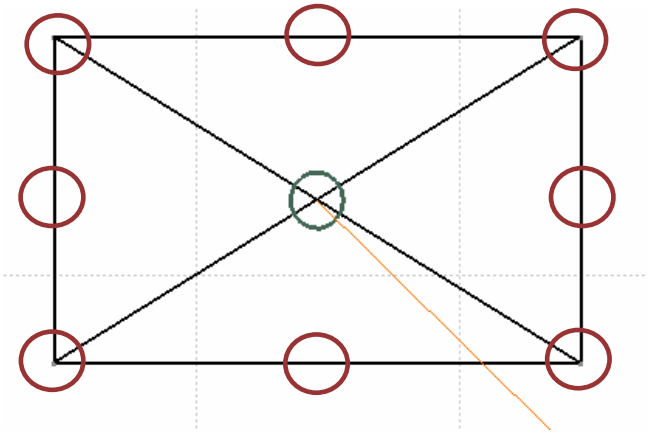
Student Notes:

Hints & Tips on Text Commands (11/14)

10 **How to modify the origin point of a leader on 2D Component:**

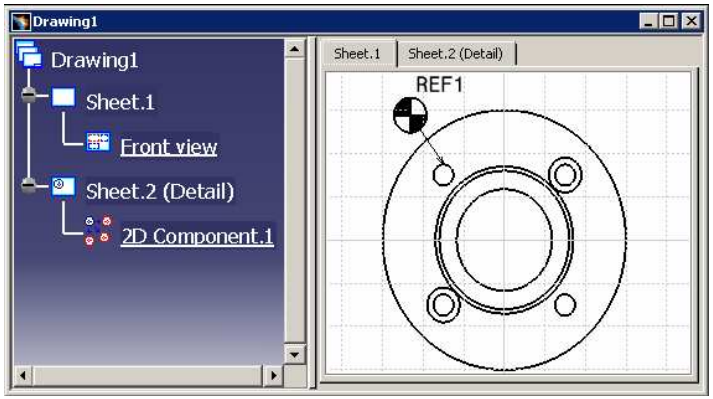


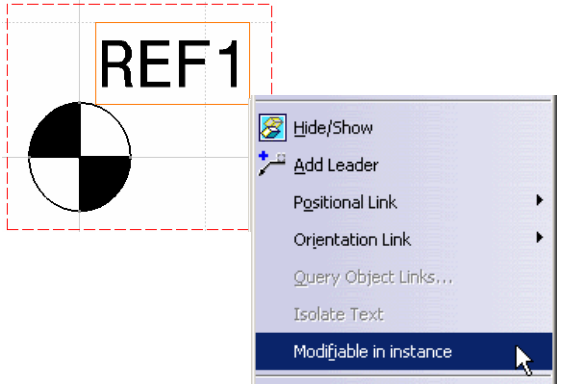
You have only some possibilities to change the origin point of the leader. It depends on the geometry of the 2D Component but it keeps quite similar to this example.



Hints & Tips on Text Commands (12/14)

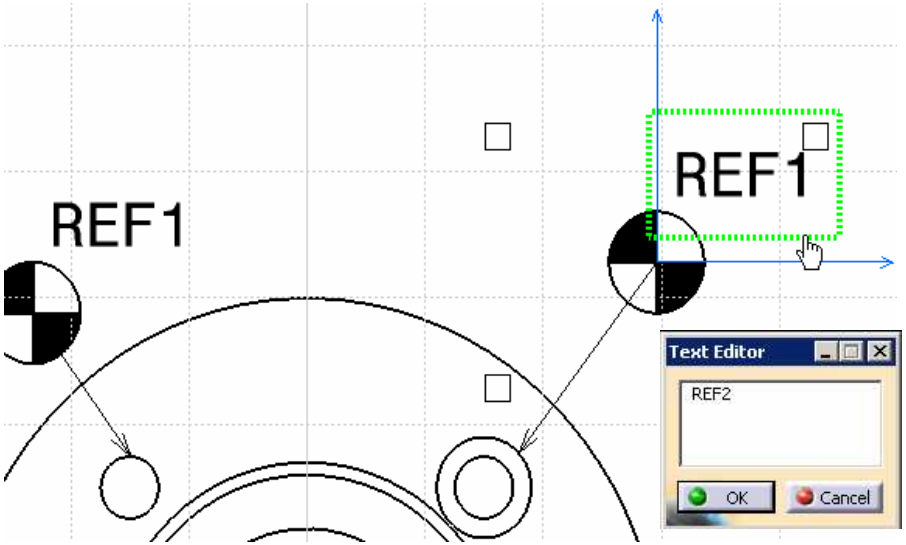
11 **How to modify text in 2D Component instances:**

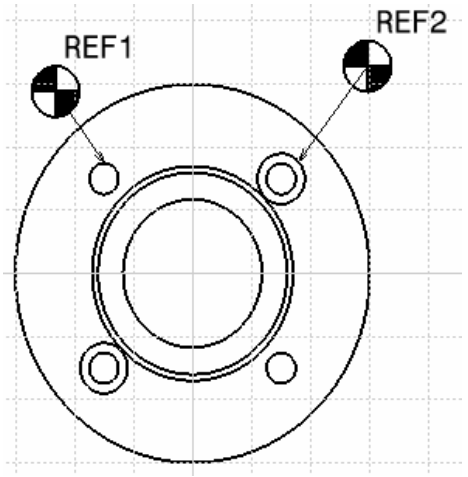




You can modify text string and properties for each instance by using the "Modifiable in instance" function in the contextual menu on 2D Component in the Detail sheet.

This operation is irreversible and all the 2D component instances texts become modifiable.



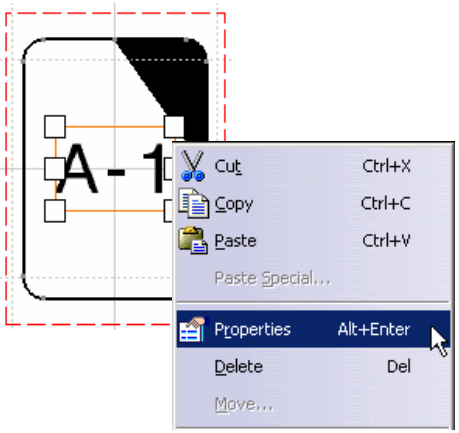


Student Notes:

Student Notes:

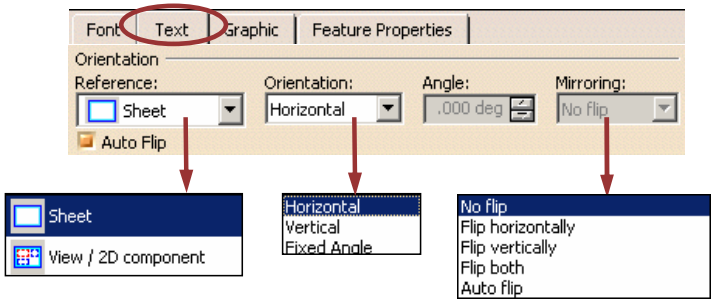
Hints & Tips on Text Commands (13/14)

12 How to fix the text orientation in 2D Component instances:



You can specify text orientation in the Properties menu of the 2D Component in the Detail sheet.

This operation concerns all the 2D component instances texts.



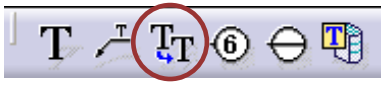
Here is a summary table

	Original orientation	Horizontal	Vertical	Flip Horizontal	Flip Vertical	Rotate
Text rotate with 2D component instance	V-101	V-101	V-101	V-101	V-101	V-101
Text orientation is fixed and independent from 2D component instance orientation	FE 101 / EQ-022	FE 101 / EQ-022	FE 101 / EQ-022	FE 101 / EQ-022	FE 101 / EQ-022	FE 101 / EQ-022

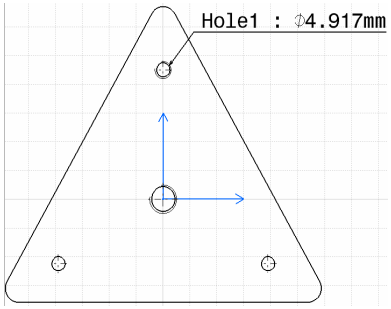
Student Notes:

Hints & Tips on Text Commands (14/14)

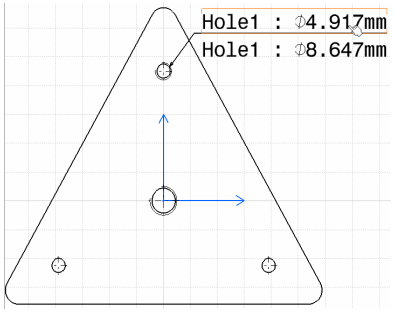
13 **How to use Text Replicate :**



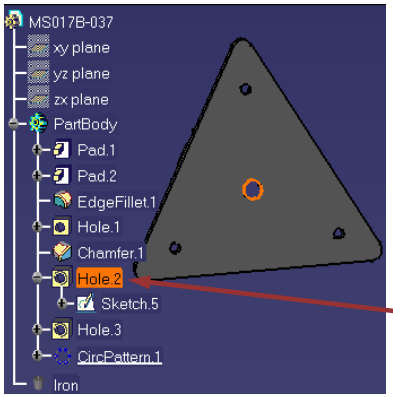
This function allows you to replicate a Text. You can also replicate a Text with Attribute.



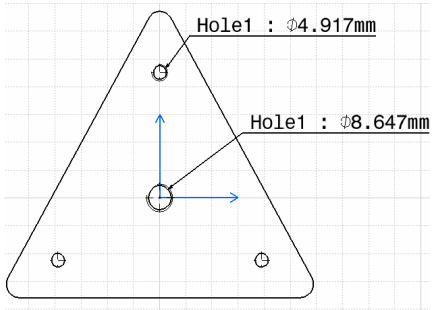
1 Here the Text is linked to the diameter value of the hole in the 3D Part.



3 The new replicated text automatically appears under the cursor then click where you want the new text to be positioned. You can add leader.



2 Select the hole in the 3D Part then enter the Text Replicate function.



STUDENT GUIDE

Student Notes:

Hints & Tips on Generative Drafting

You will become familiar with …

- Generating Specific Views
- Creating Section View/Section Cut with a Profile Defined in 3D
- Update Management
- Auto-Dimensioning
- Advanced Filtering Techniques
- Balloon Creation

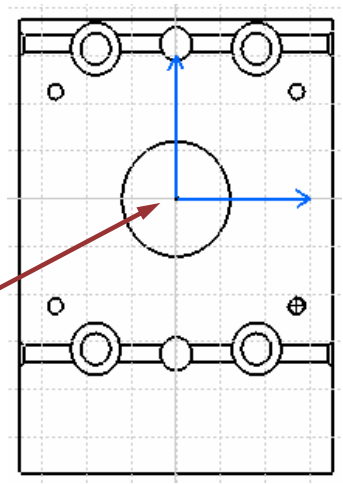
Generating Specific Views (1/3)

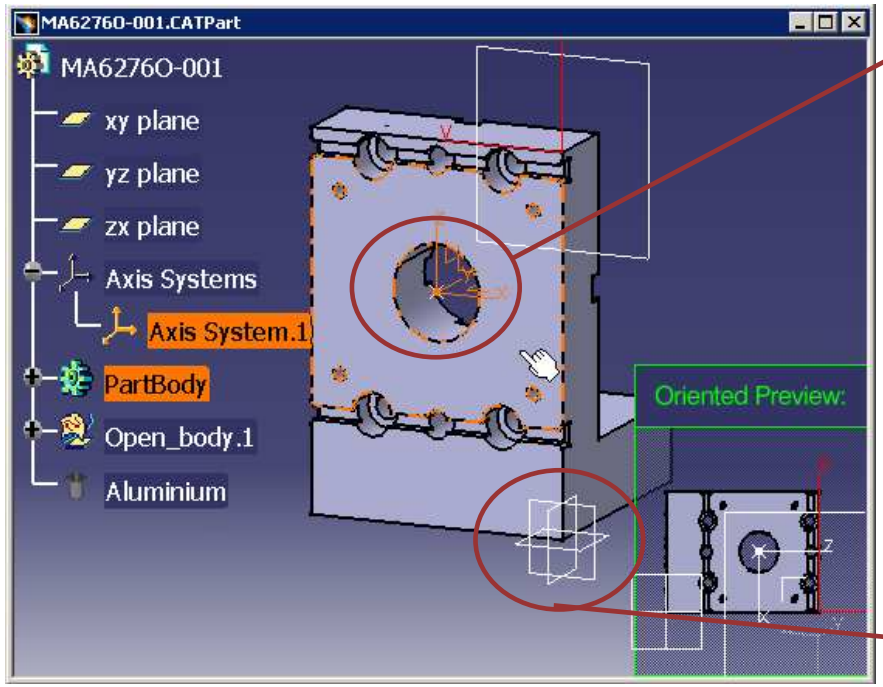
Student Notes:

You can use a 3D local axis as reference for the Front view in the view creation .

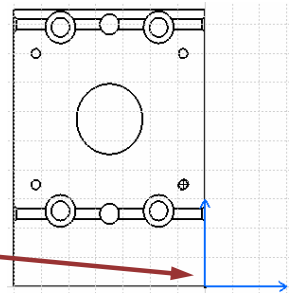
After entering the Front View function, you must select the Axis System in the Specifications Tree before selecting the reference view plane. This specification is propagated in projection, section, auxiliary and detail views.

It is not necessary that this Axis System is the current one.





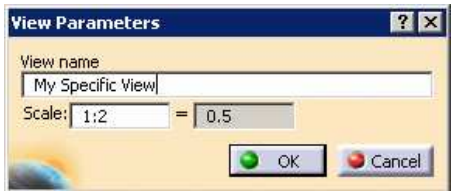
If you don't select a local Axis System, you will generate a Front view with a blue axis system which is the projection in the view plane of the 3D absolute axis system.

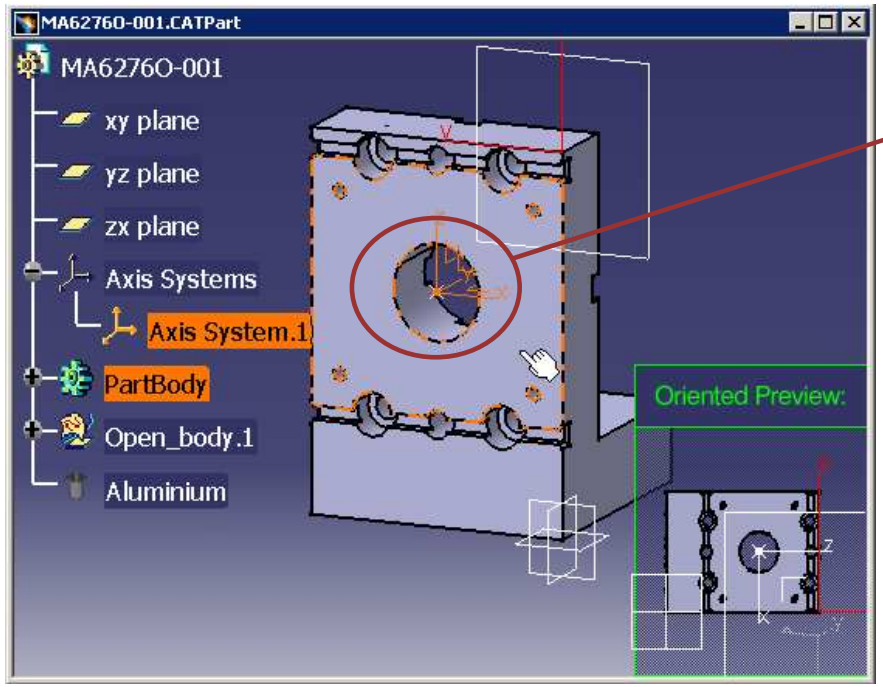


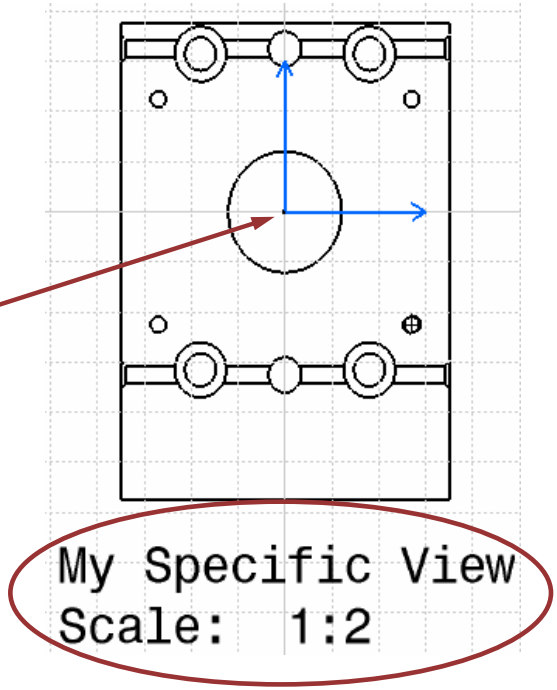
Student Notes:

Generating Specific Views (2/3)

The Advanced Front View command allows you to define view name and scale at the view creation. This command takes also into account local Axis System reference.

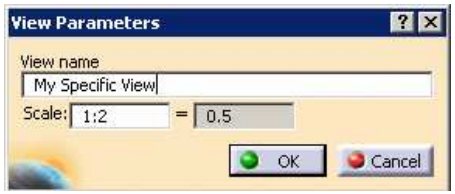


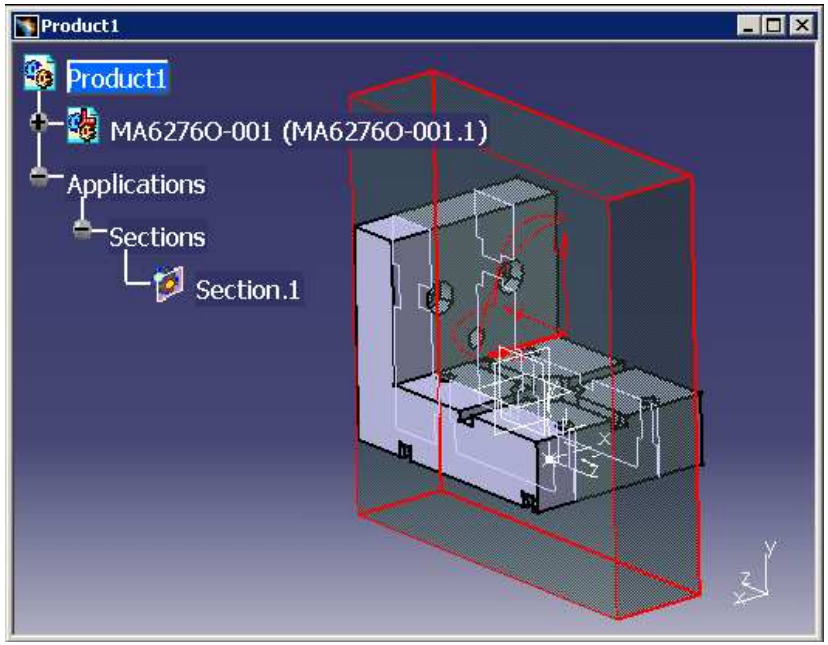


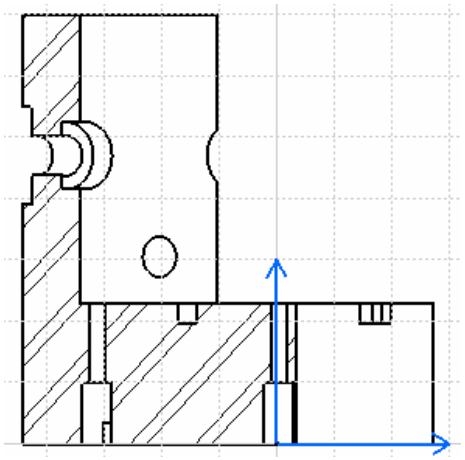


Generating Specific Views (3/3)

The Advanced Front View command allows you to use a DMU Section box to specify a 3D clipping view.







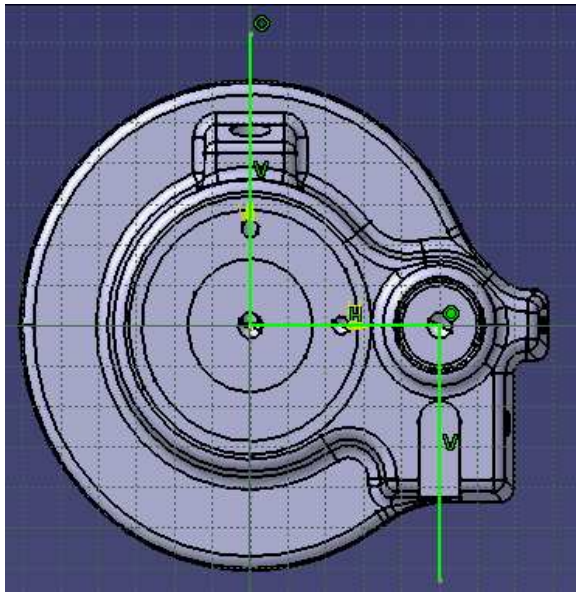
Select the Section Feature in the specifications tree then a plane.

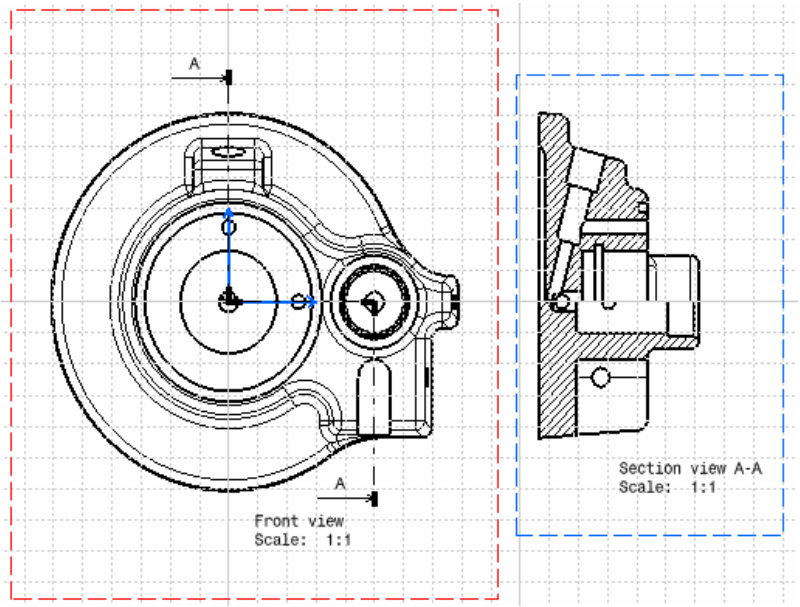
Student Notes:

Student Notes:

Creating a Section View/Section Cut with a Profile Defined in 3D

Defining a profile in 3D to create a section view / Section cut enables you to make the profile associative with the geometry, and therefore to drive this profile using dimensional constraints.





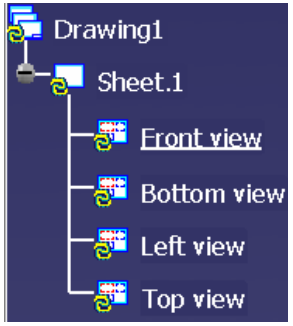
Depending on the type of profile you sketch and the type of section you want (offset or aligned), you can select different icons in the Sections Toolbar. If the 3D profile is not valid for generating the appropriate view, you will not be able to select it.

When editing a 3D profile, make sure that you modify it in accordance with the type of section (offset or aligned) you created. If an edited profile is invalid when you update a drawing, the associated section view/section cut will not be displayed (an error symbol will appear instead).

Student Notes:

Update Management

If you perform modifications in a 3D model which has associated drawings, an update will be requested in the drawings. You can update all views or a selection of views.



The Update icon is active in the Update toolbar when a drawing contains views that need to be updated. You can update all views in the active sheet by clicking this icon.

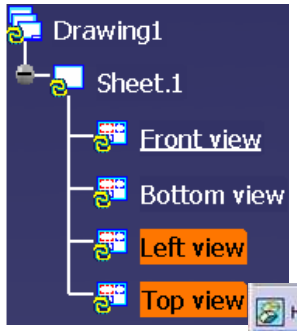
Updating views means:

-re-computing associative section/auxiliary view profiles

-re-generating the geometry

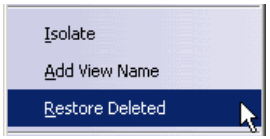
-re-computing any annotation/dimension/dress up element linked to the generated geometry

-taking into account deleted views or views that are graphically modified on the condition the view is up-to-date when delete or modify it.



An update symbol appears in the specification tree for the views that need to be updated. You can update a selection of views by selecting them and using the *Update Selection* contextual command

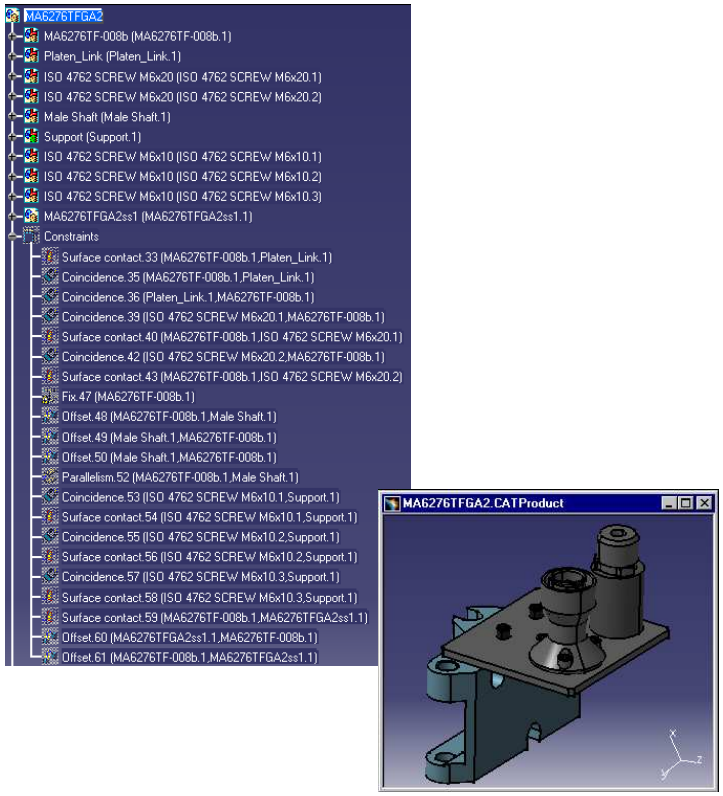
Update symbols also appear in the specification tree to indicate drawings and sheets containing views that need to be updated.

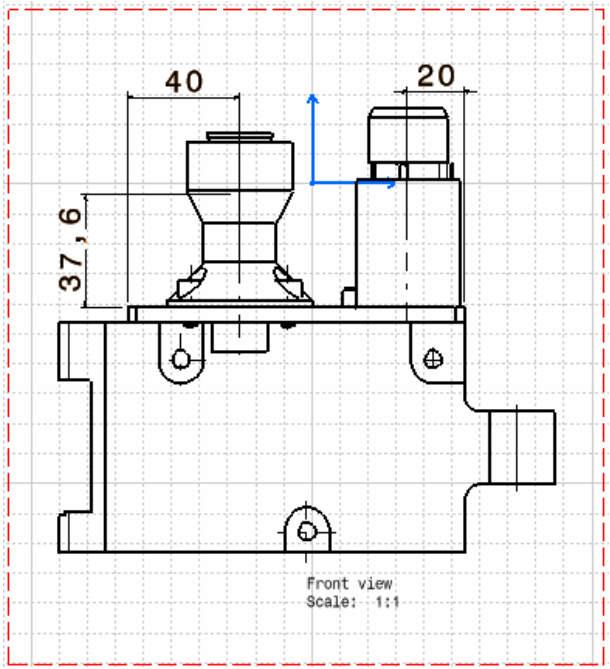


Remarks: you can restore deleted elements by selecting Restore Deleted into the contextual menu and then updating the view. You can either use the Update icon if you modify the 3D model or key in C:Force Update if you did not modify the 3D model.

Auto-Dimensioning (1/16)

If you use the Generating Dimensions function on a drawing generated directly from an assembly, you will have only dimensions from the assembly constraints and not from each Part. Moreover, the generated dimensions are only in relation to offset or angle constraints.





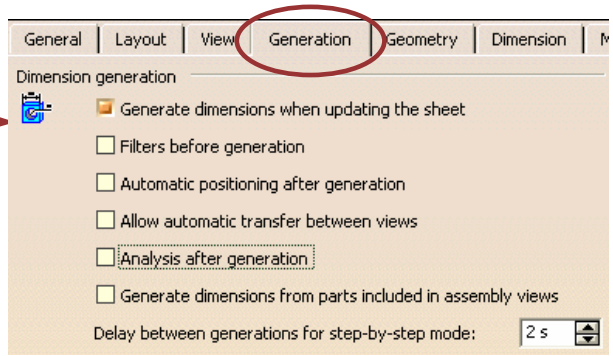
Student Notes:

Student Notes:

Auto-Dimensioning (2/16)

- **Dimension Generation settings: The way that you can generate and manage dimensions depends on the options that you have chosen in the Tools/Options/Drafting command, Generation tab**
  - **Generate dimensions when updating the sheet**

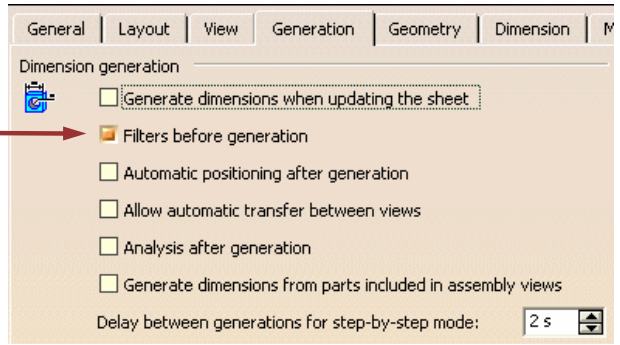
By selecting this Option, if you have already generated dimensions, new dimensions will be generated automatically if you update your drawing after a modification in the 3D.

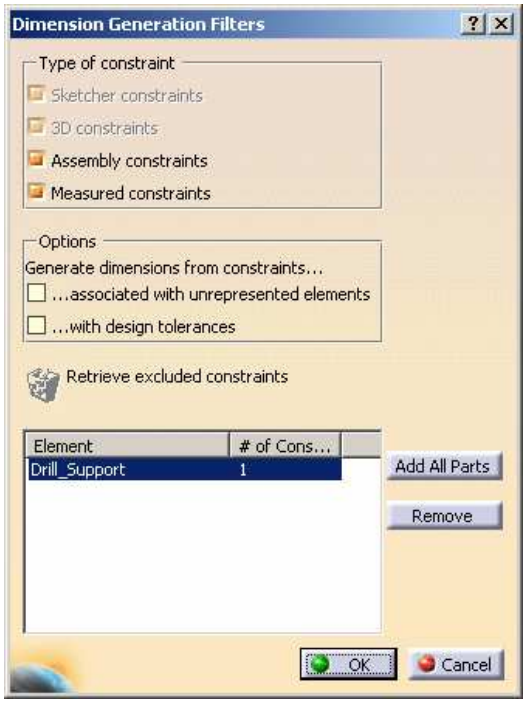


Auto-Dimensioning (3/16)

- The Filters before generation option

Select this option if you want to display the Dimension Generation Filters panel when you enter the Generating Dimensions function. This option allows you to select the view in which one you want to generate the dimensions.





If you don't select this option, all dimensions will be automatically generated when you select the Generating Dimensions function in all the views.

Before accessing the Generating Dimensions function, you can also use a Trap to define the views concerned by the dimensions generation.

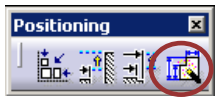
It's the same for the Generating Dimensions Step by Step function.

Student Notes:

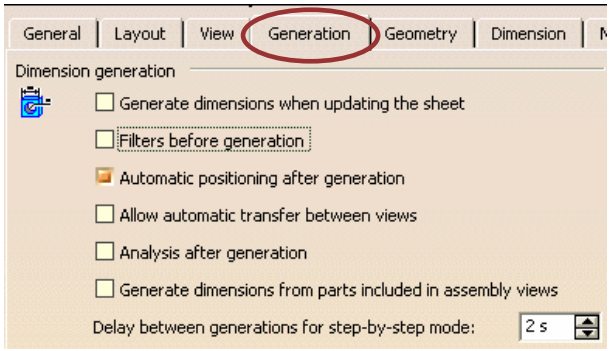
Student Notes:

Auto-Dimensioning (4/16)

- The Automatic positioning after generation option

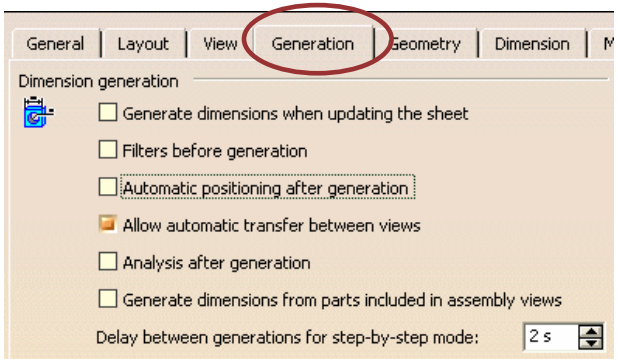


If you chose this option, the Dimension Positioning function will be automatically executed at the end of the dimensions generation.



- The Allow automatic transfer between views option

If you have already generated the dimensions on the drawing and you add an other view, when you repeat the Generating Dimensions function, some created dimensions will be automatically transferred into the new view if that improves the clearness of the drawing.

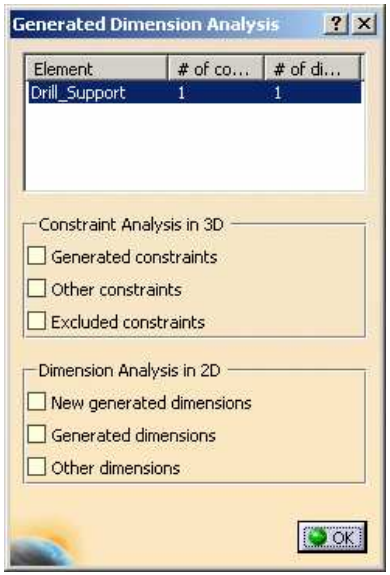


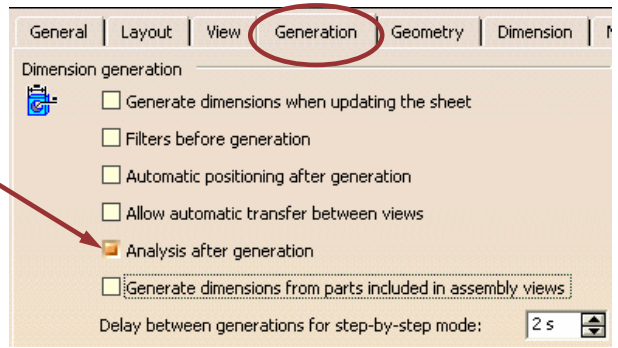
Student Notes:

Auto-Dimensioning (5/16)

- The Analysis after generation option

You have to chose this option if you want to display the Generated Dimension Analysis panel at the end of the dimensions generation.





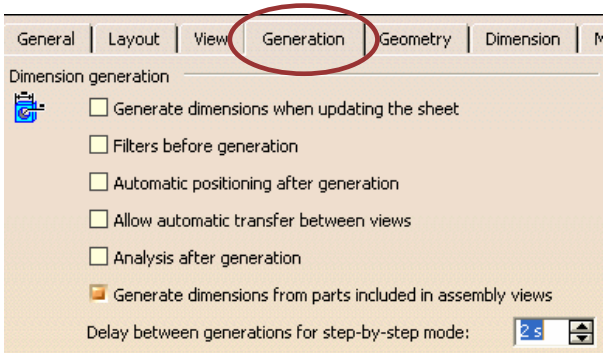
If you don't select this option, you won't have any information about the number of generated dimensions in relation to the number of constraints in the 3D .

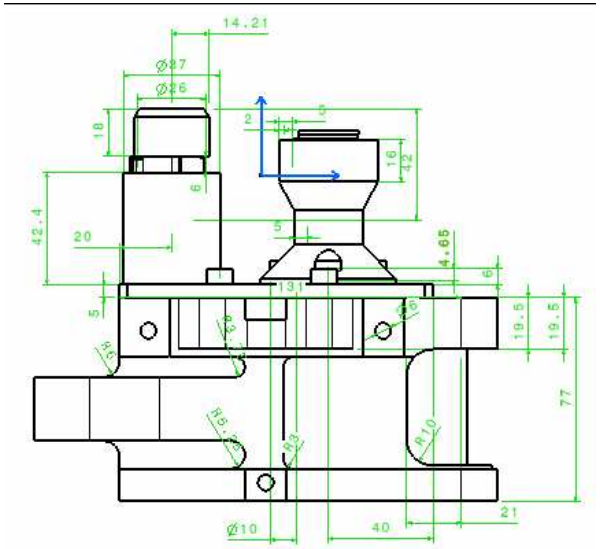
Student Notes:

Auto-Dimensioning (6/16)

- The Generate Dimensions from parts included in assembly views

Select this option to extract 3D part constraints (on top of assembly constraints) when generating product dimensions.
This option is particularly useful if you want to generate dimensions for all parts included in assembly or product views, without displaying the Dimension Generation Filters dialog box before dimension generation. Note that if you display the Dimension Generation Filters dialog box before generating dimensions, you will need to indicate for which parts you want to generate dimensions.

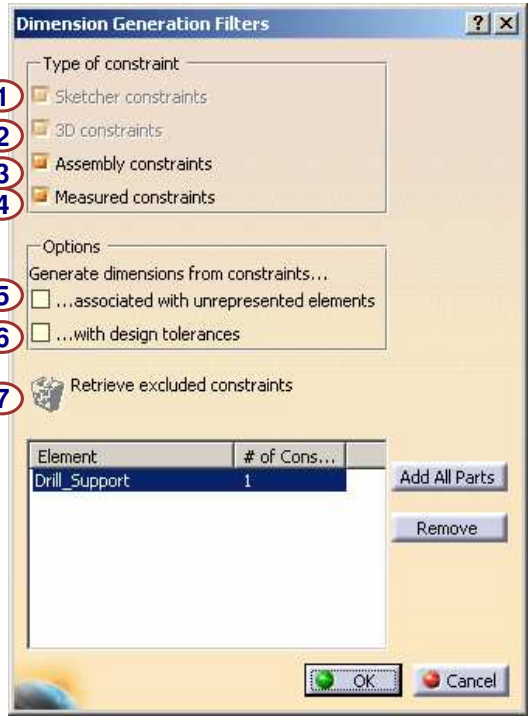




Student Notes:

Auto-Dimensioning (7/16)

- The Dimension Generation Filters panel
  - This panel gives you information about the total number of constraints in 3D and allows you to select the kind of constraints that will be taken into account for dimensions generation.



1 Generate all the dimensions from constraints created in the Sketcher workbench.

2 Generates dimensions from 3D constraints

3 Generates dimensions from assembly constraints. This option is active only in the case of products or assemblies.

4 Generates dimensions from measured constraints

5 Generates dimensions from constraints associated with elements that are not represented on the drawing (i.e. which are not visible in the various views your drawing may contain). In this case, the generated dimensions will appear as not attached to any element in the drawing.

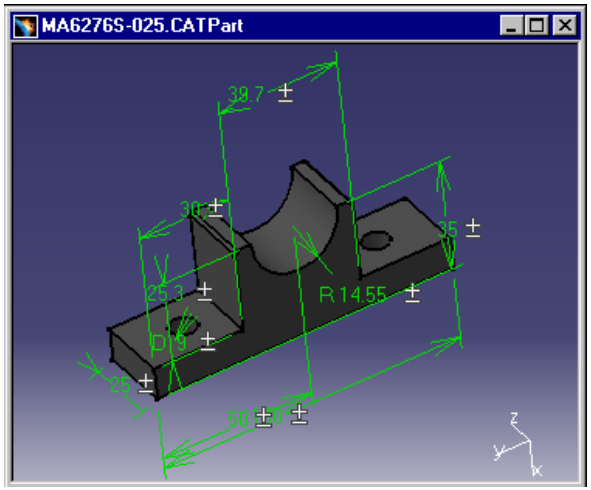
6 Generates dimensions from constraints with design tolerances, and applies the constraint tolerance to the corresponding generated dimension.

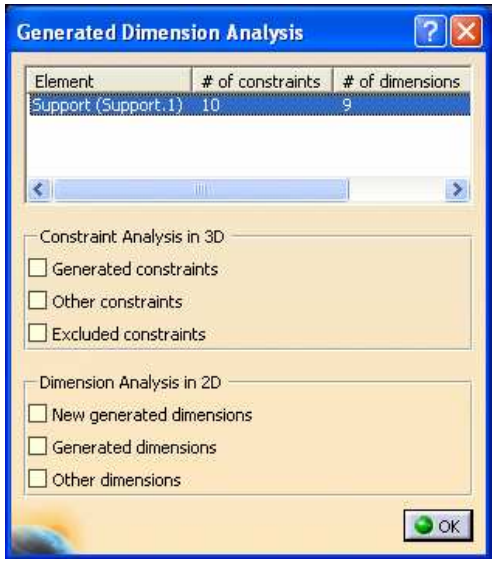
7 Retrieves all the constraints which you previously excluded (either using the Not Generated icon when performing a step by step generation, or by removing a generated dimension from the drawing), and re-generates the dimension. This icon is active only when there are dimensions to retrieve

Student Notes:

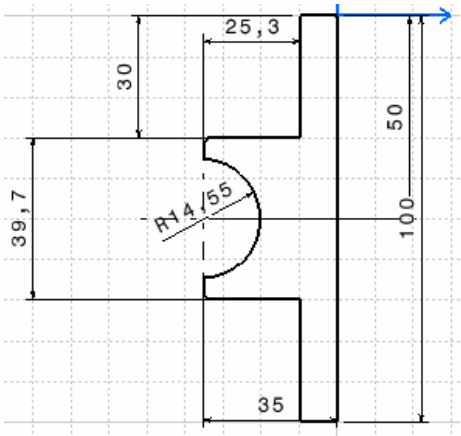
Auto-Dimensioning (8/16)

- The Generated Dimensions Analysis panel
  - The Generated Dimension Analysis dialog box displays the number of constraints available in the 3D, as well as the number of dimensions generated on the drawing, for each part or product in the drawing (in this case, there is only one part). You can use the options in this dialog box to highlight the dimensions in the drafting sheet as well as the associated 3D constraints you can visualize in your Part or Product document.





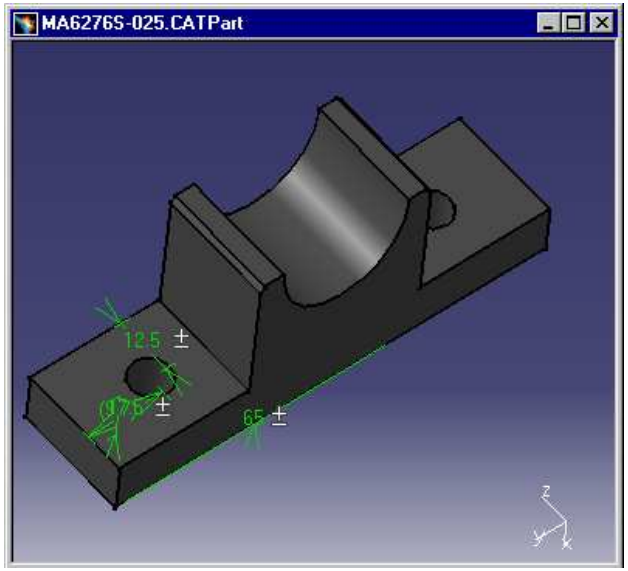
Be careful, this panel gives only information from the last dimensions generation.

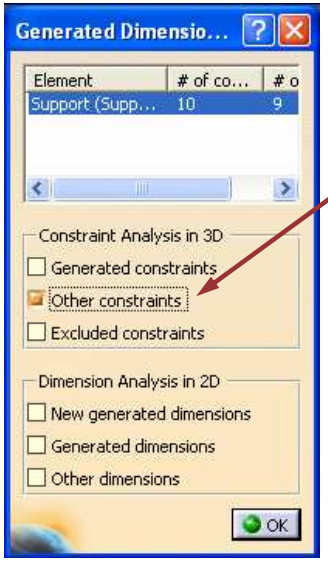


Student Notes:

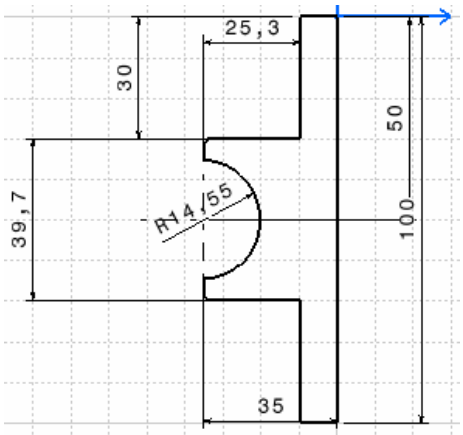
Auto-Dimensioning (9/16)

- The Generated Dimensions Analysis panel





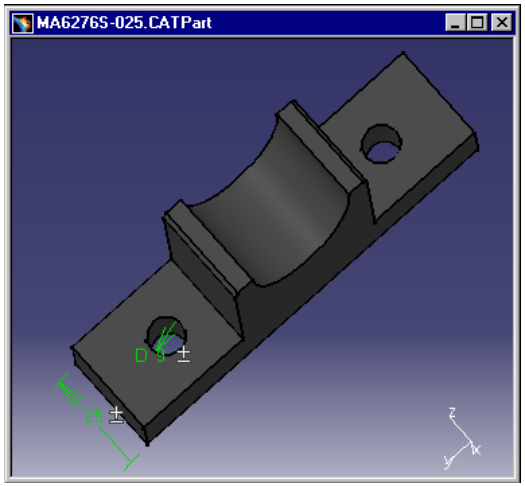
Displays all the constraints for which dimensions have not been generated on the drawing.

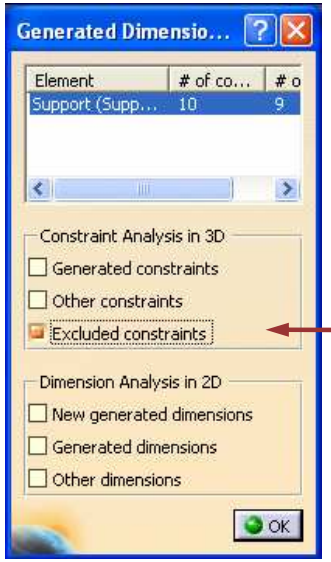


Student Notes:

Auto-Dimensioning (10/16)

- The Generated Dimensions Analysis panel





Displays all the constraints which have not been taken into account during the dimension generation (this can be because you previously excluded them using the Not Generated icon when performing a step by step generation, or because you removed a generated dimension from the drawing

Student Notes:

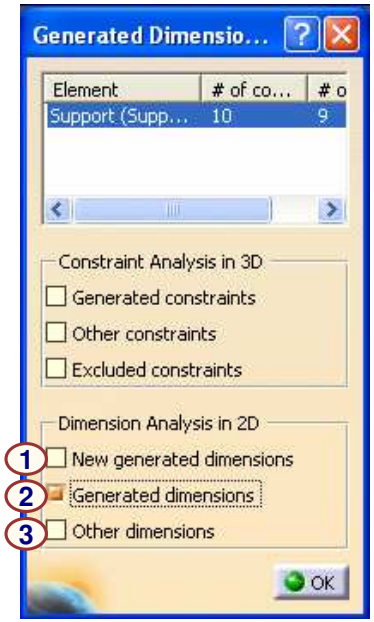
Auto-Dimensioning (11/16)

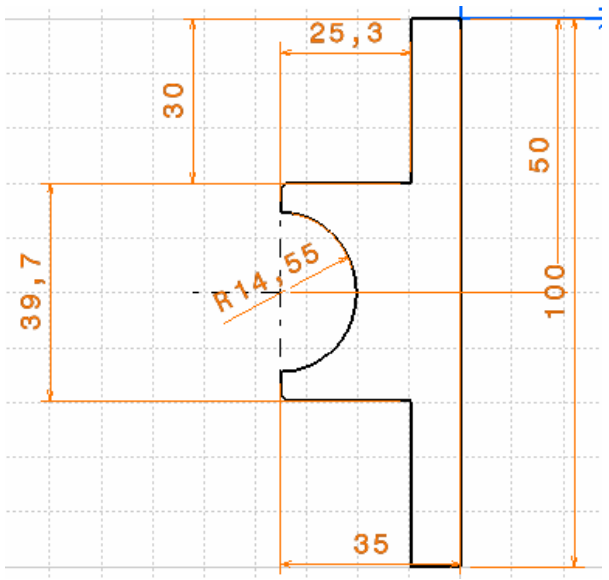
- The Generated Dimensions Analysis panel

1. Highlights in the drawing the new generated dimensions since your last dimension generation.

2. Highlights the dimensions created manually via the Interactive Drafting workbench

3. Highlights in the drawing all generated dimensions.



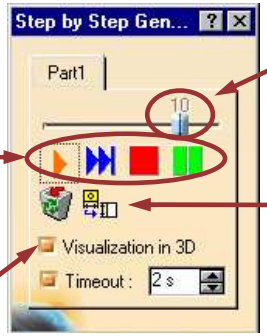


Student Notes:

Auto-Dimensioning (12/16)

- The Step by Step Generating Dimensions panel
  - The Step by Step Generating Dimensions function is interesting if you know which dimensions you want to generate, if you have finished the drawing layout (all the views are fixed) and if there is not to much dimensions to generate.

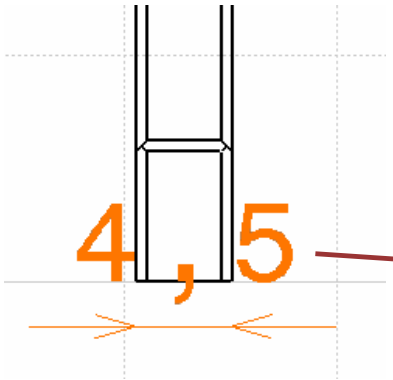
Buttons allowing you to generate the next dimension, to stop the step by step generation and to generate the remaining dimensions in one step, to abort the process and to make a pause.



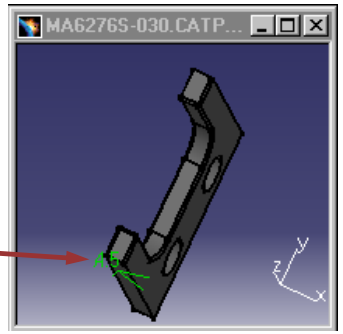
You will display in the drawing all generated dimensions.

You can transfer a dimension to another selected view.

You can visualize in the 3D the constraints corresponding to the dimensions to be created on the views (zoom automatic).

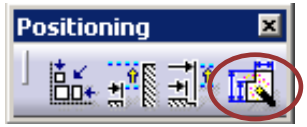


You can modify the dimension.

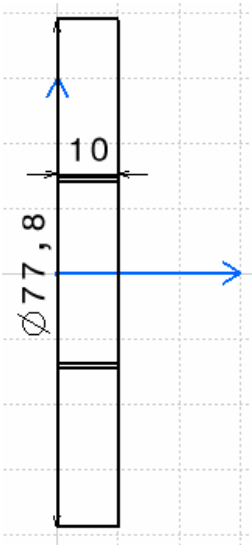


Student Notes:

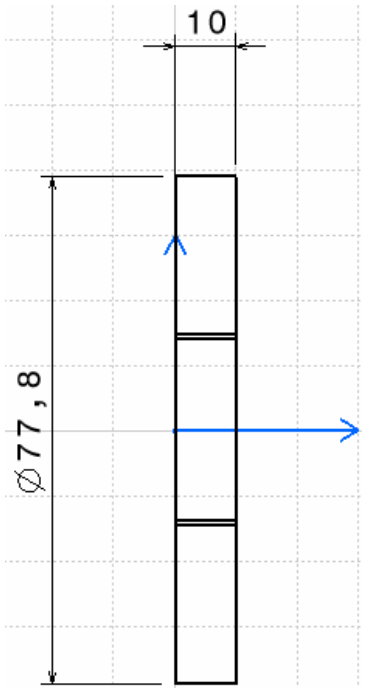
Auto-Dimensioning (13/16)



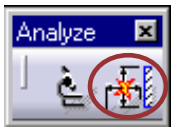
- **The Dimension Positioning function**
  - **This function will position distance and length dimensions (either generated dimensions or interactive dimensions) in a better way. These dimensions will be positioned on the active view exclusively.**



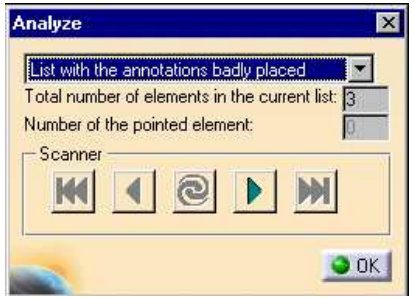
This function takes into account only distance & length dimensions and diameter dimensions if they are represented as side dimensions.

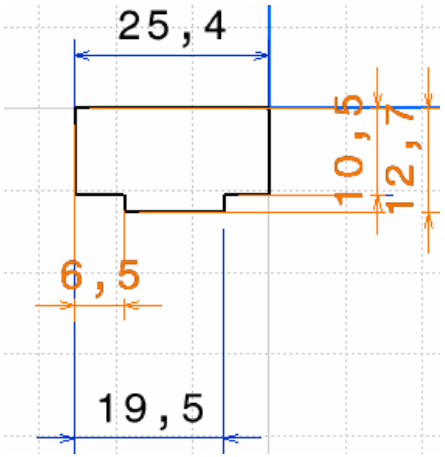


Auto-Dimensioning (14/16)

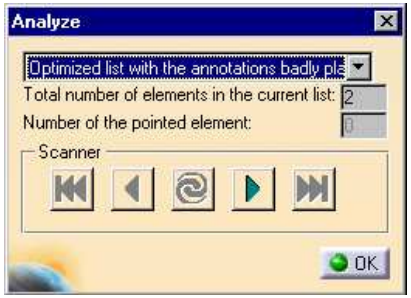


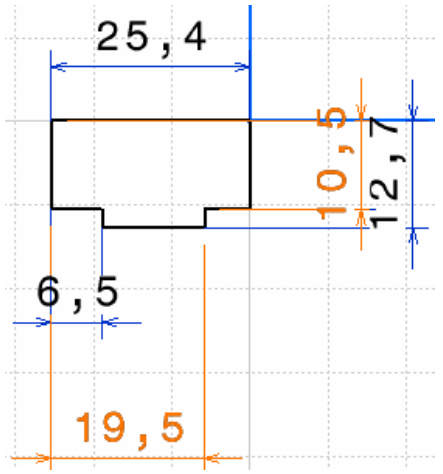
- The Dimension Analysis function
  - This function will analyze the dimensions generated from 3D and those created manually and will indicate if there are any interferences between them.





You can choose to have either a whole or a filtered list with the interfering elements. The total number of interfering elements is automatically updated according to your choice.





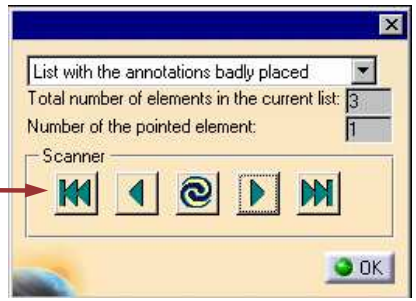
Student Notes:

Student Notes:

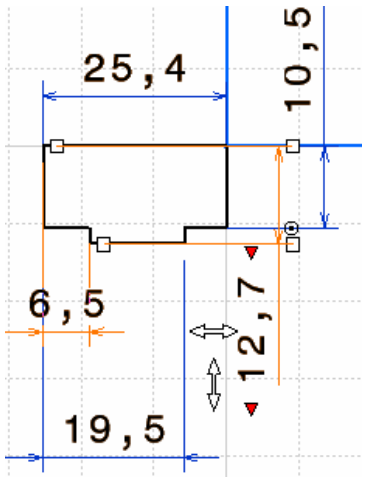
Auto-Dimensioning (15/16)

- The Dimension Analysis function

Use the scanner to navigate among the list of the interfering dimensions.



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

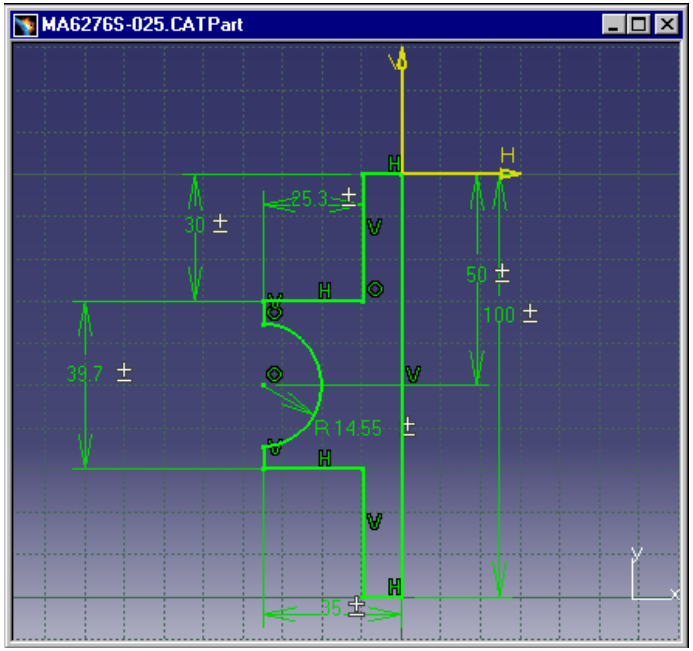


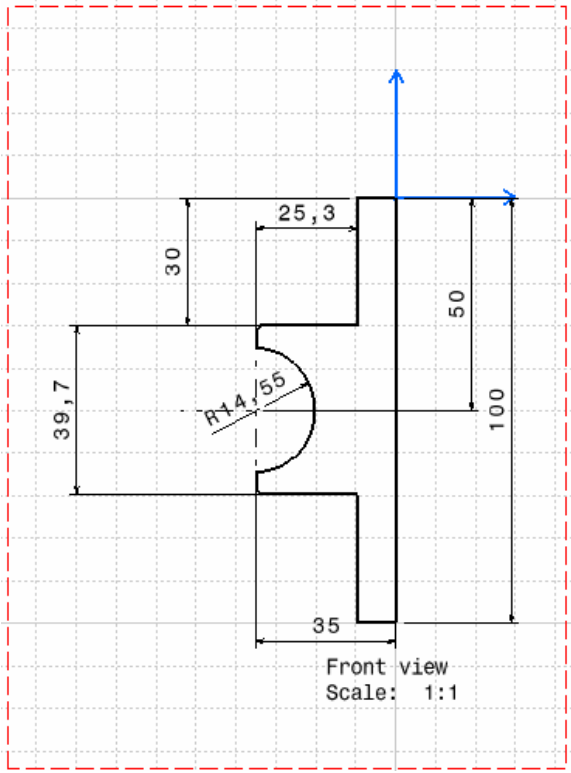
Student Notes:

Auto-Dimensioning (16/16)

- Hints & Tips

The positioning of the generated dimensions depends directly on the positioning of the constraints in the Sketch. So, if you want to optimize the clearness of the drawing, try to create your sketch constraints properly



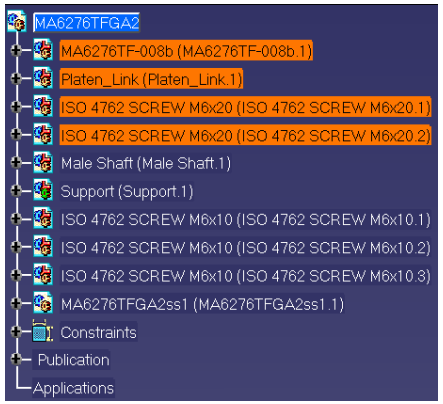


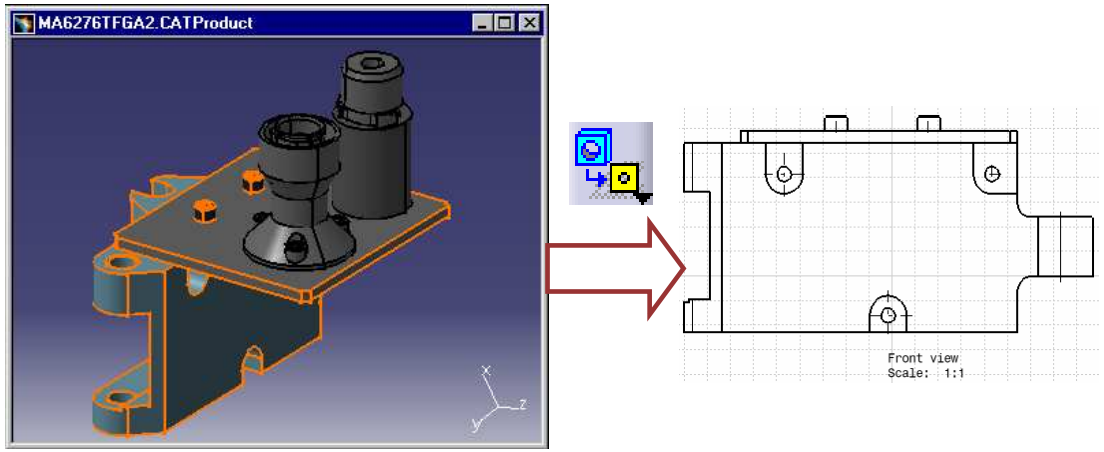
Student Notes:

Advanced Filtering Techniques (1/9)

- 3D components selection
  - In a Product (or a Part), you can select only the Parts (or Bodies) you want to show in the drawing.

Enter the Front view function then select in the 3D document all the parts/products/bodies you want to see in the view



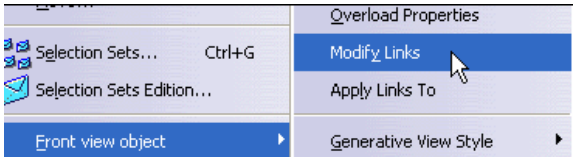


Student Notes:

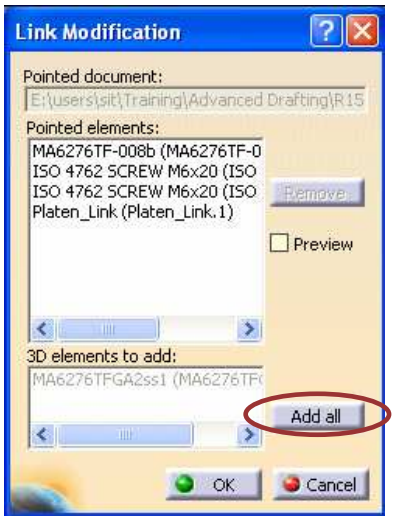
Advanced Filtering Techniques (2/9)

- After the view creation, you have the ability to modify the list of elements to be taken into account in an existing view, by adding or removing sub-products/parts/bodies.

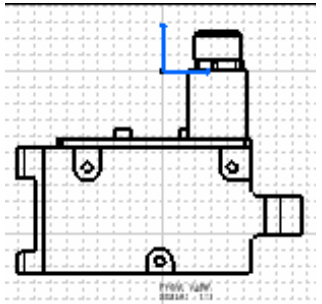
1. Right click and select *Front view object/ Modify Links* in the contextual menu. Link Modification dialog box is displayed.



2. In the 3D product, select some new parts/products and click *Add all* button.



3. After drawing update, the selected parts/products are projected in the front view

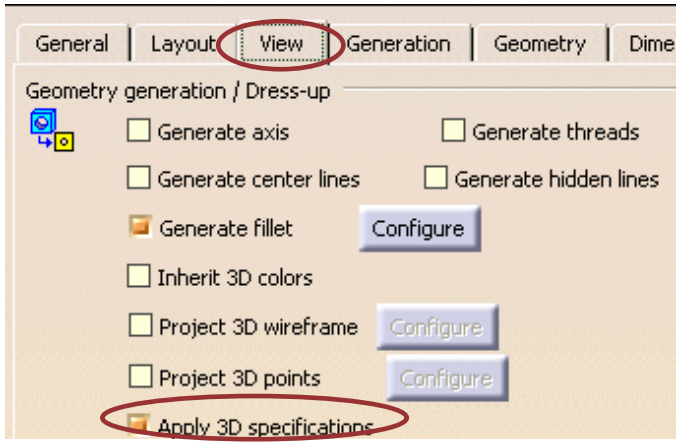


Advanced Filtering Techniques (3/9)

Student Notes:

- **The Drafting Properties panel**

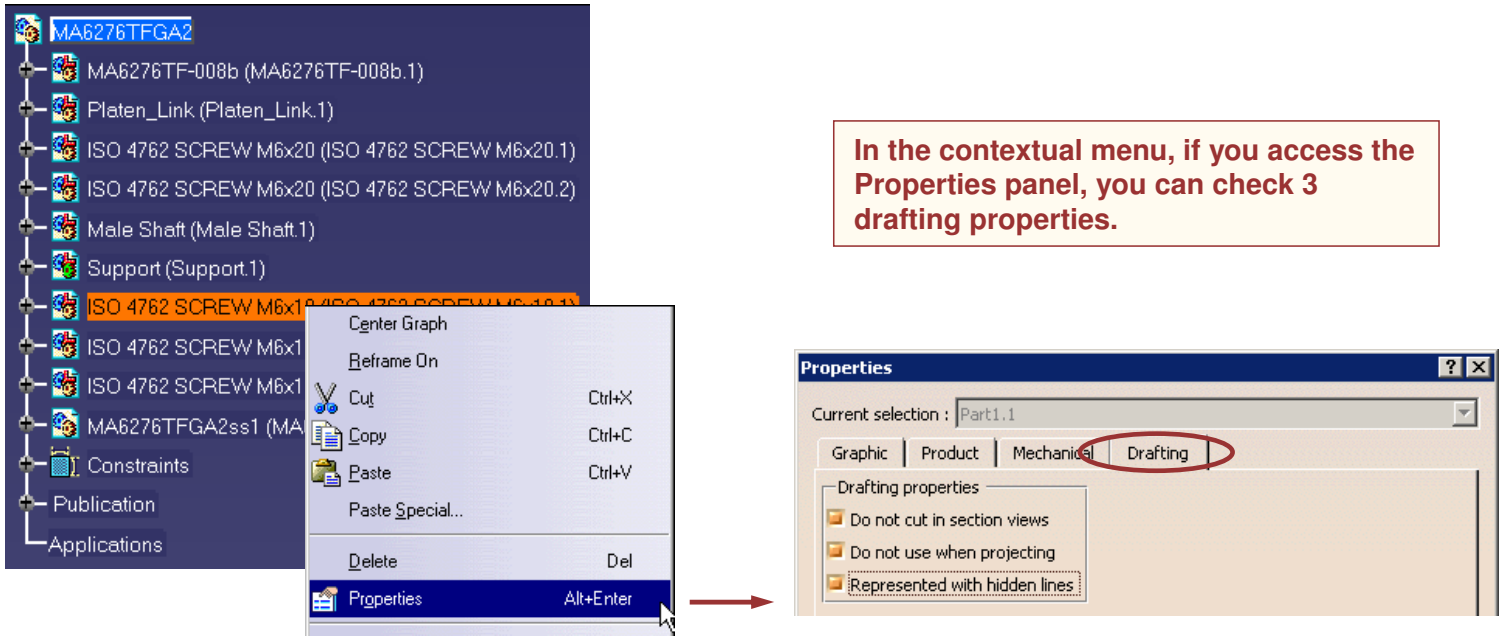
  You have the possibility to apply filters on a Product or a Part in relation to the type of view you want to generate. To enable this function, you must first check the *Apply 3D specifications* drafting setting



Student Notes:

Advanced Filtering Techniques (4/9)

- The Drafting Properties panel



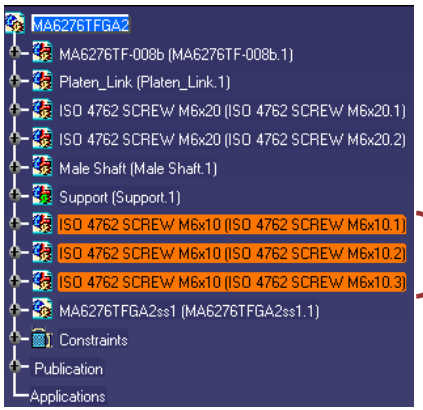
In the contextual menu, if you access the Properties panel, you can check 3 drafting properties.

You can use the Multi-selection to apply the same specifications to several Parts of a Product.

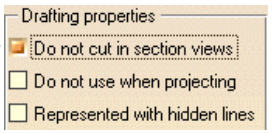
Student Notes:

Advanced Filtering Techniques (5/9)

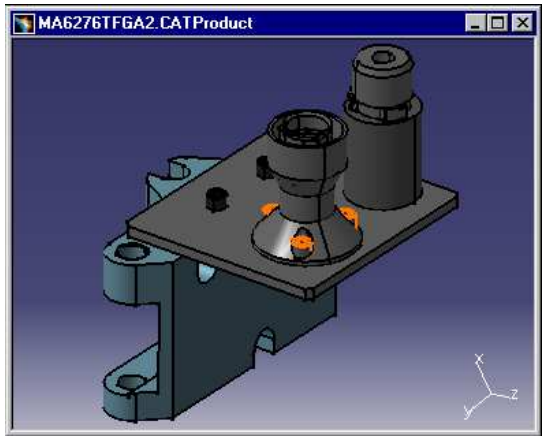
The Drafting Properties panel



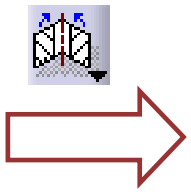
Multi-select the screws of the assembly and apply them the Do not cut in section views option.

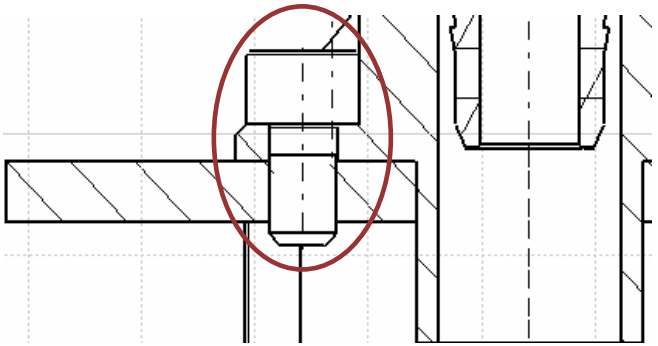


This option is not available on a Product.



If you create a section view going through the center of the screw, you can see that it is not cut.

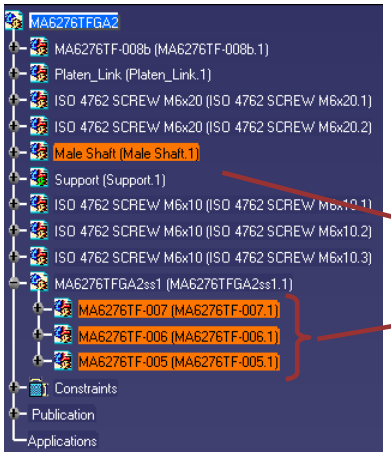




Student Notes:

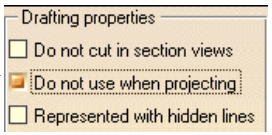
Advanced Filtering Techniques (6/9)

The Drafting Properties panel

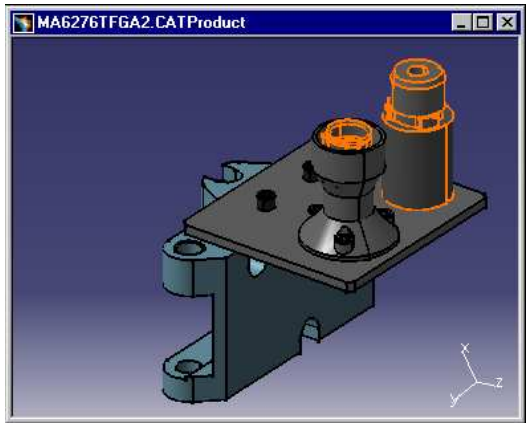


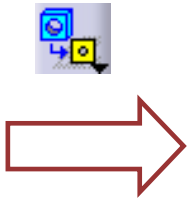
Multi-select the parts of the assembly you don't want to see in the drawing view and apply them the Do not use when projecting option.

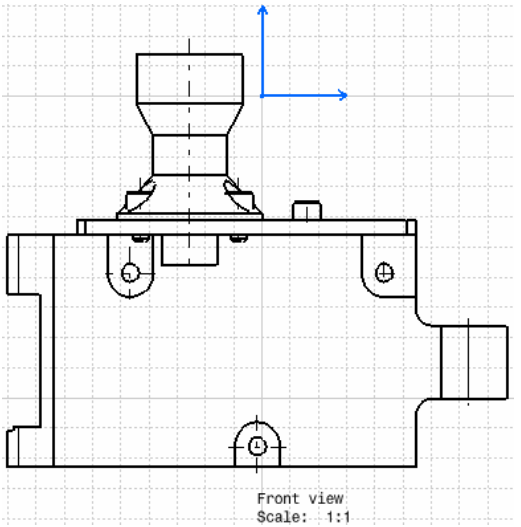
This option is not available on a Product.



If you create a front view of the general assembly, you won't see the selected Parts.



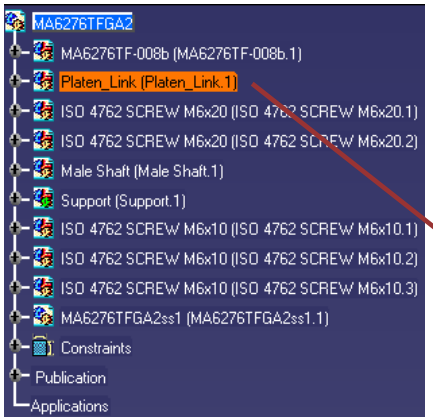




Student Notes:

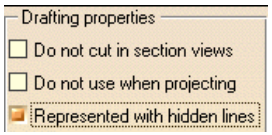
Advanced Filtering Techniques (7/9)

The Drafting Properties panel

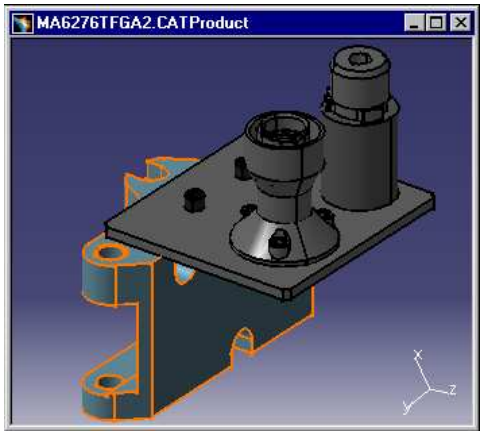


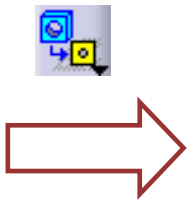
Select the parts you want to project in the drawing view with hidden lines and check the *Represented with hidden lines* option.

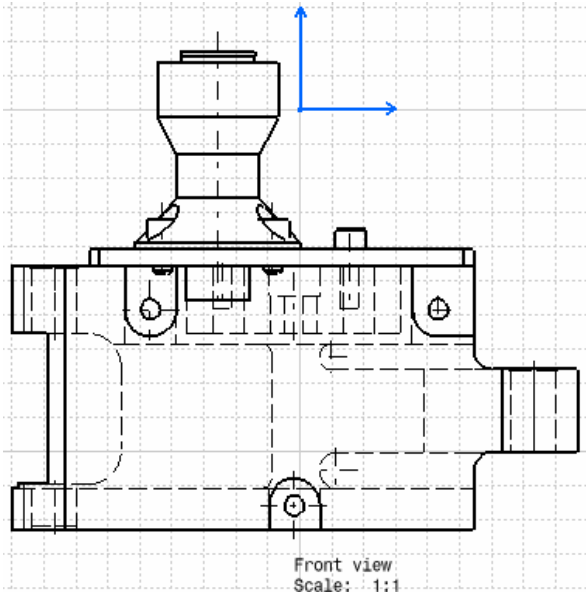
This option is not available on Products.



If you create a front view of the general assembly, you will see the selected parts displayed with hidden lines.



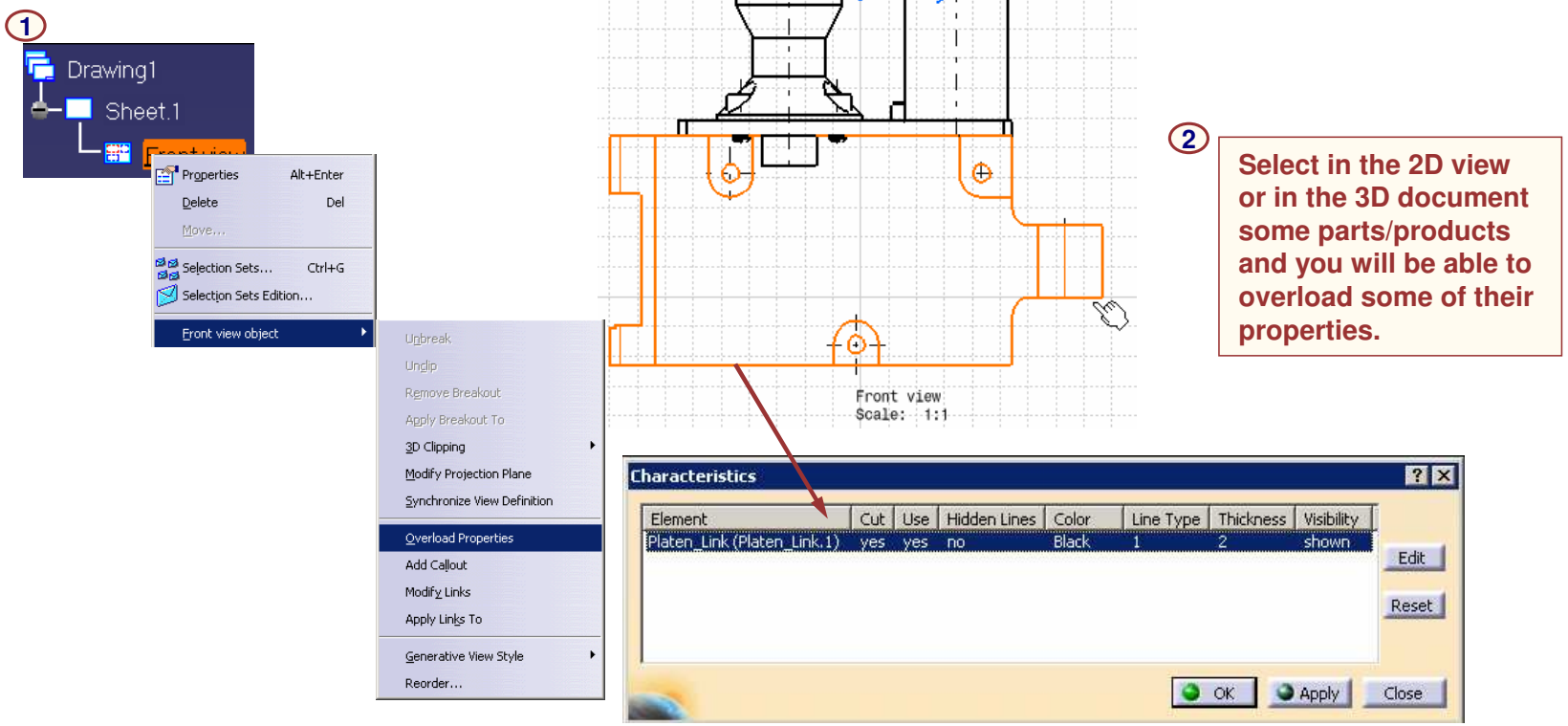




Student Notes:

Advanced Filtering Techniques (8/9)

- Overload properties
  - After view creation you can overload the properties of elements generated from a CATProduct.
    - Show/Noshow
    - Use/Unuse
    - Cut/Uncut
    - Color



2

Select in the 2D view or in the 3D document some parts/products and you will be able to overload some of their properties.

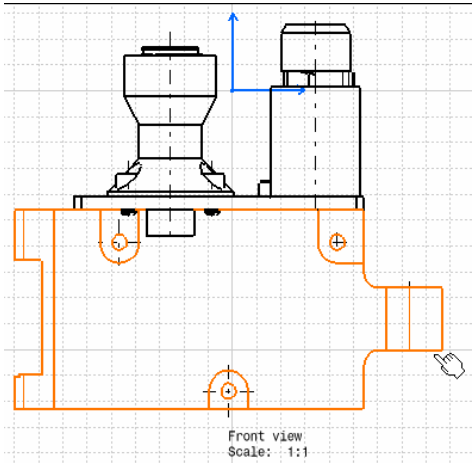
Advanced Filtering Techniques (9/9)

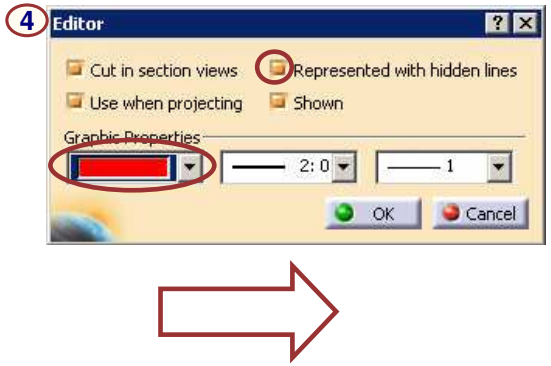
Student Notes:

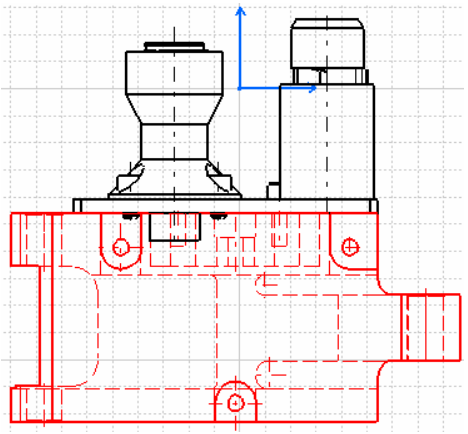
Overload properties

3 Click the Edit button to display the editor panel. Modify what is needed.

You can multi-select different parts in the 2D document or in the 3D document. A sub-product can also be selected in the 3D document.





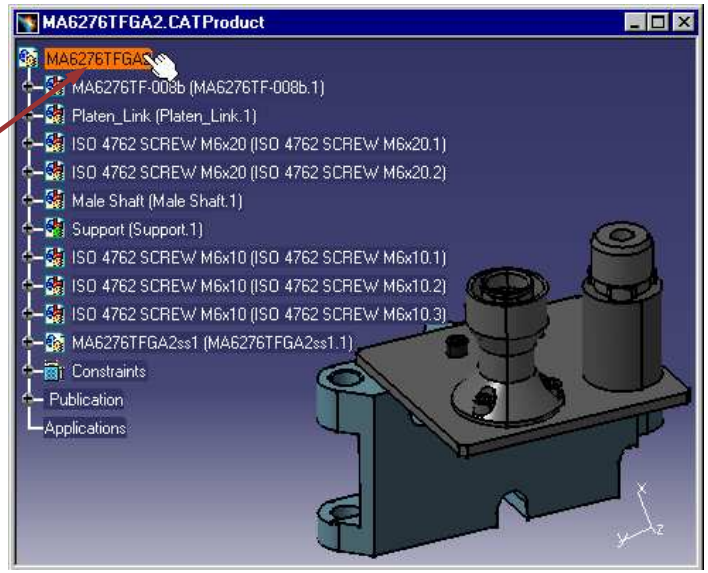


Student Notes:

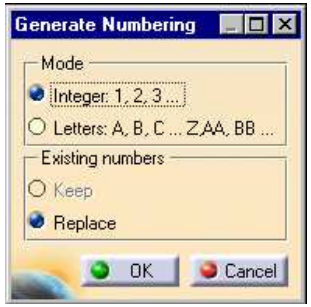
Balloons Creation (1/2)

- Generate numbering in assemblies 
  - You can generate in the active view balloons corresponding to references defined on the different parts of an assembly.
  - First of all you must generate the numbering in the CATProduct.

1 Enter the Generate Numbering function then select the assembly.



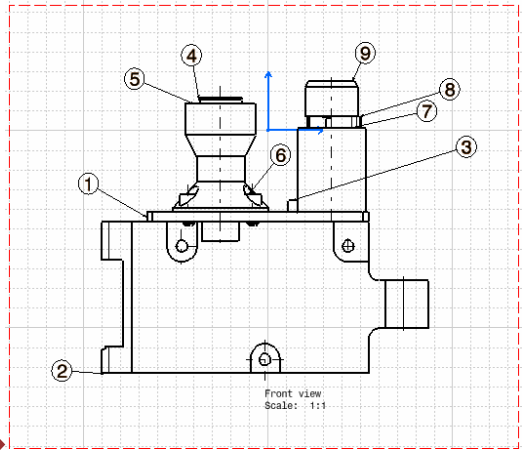
2 You can choose between Integers or Letters.



Balloons Creation (2/2)

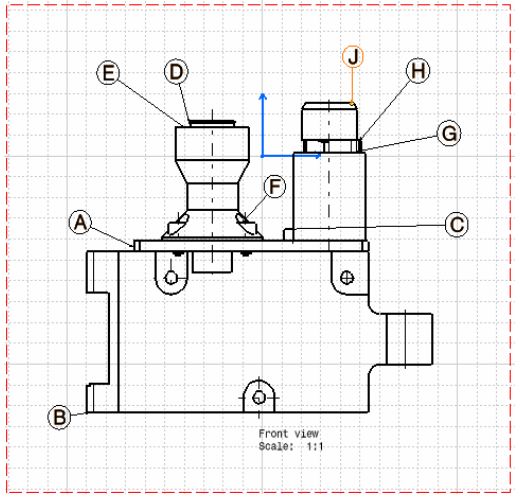
- Generate balloons

3 Active the view where you want to add balloons and select the *Generate Balloons* icon.



4 If needed, multi-select these balloons and modify their font size using the Text Properties toolbar.

You can also align them using *Element positioning* function.

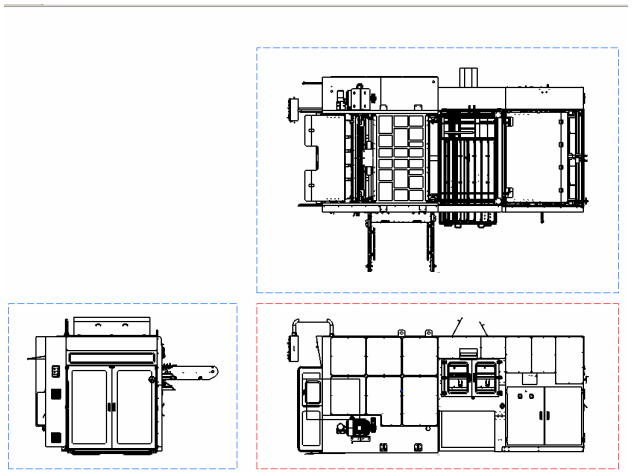


Student Notes:

Student Notes:

Drawing Generation of Large Assemblies

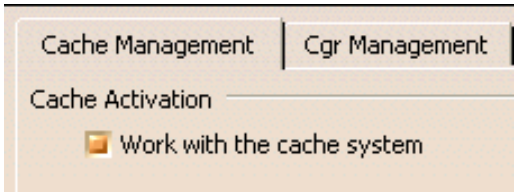
- General Recommendations
- Approximate generation mode



Student Notes:

General recommendations (1/2)

- Always use « Cache management » option
  - CATIA loads the appropriate 3D data when needed



- Adapt view generation mode to your needs
  - Exact/CGR/Approximate/Raster
- Use Occlusion Culling option
  - The Occlusion Culling option activates a pre-processing during view update, in order to determine which parts in the assembly will be hidden in the view.
  - This allows to avoid un-necessary operations during view update.
  - In Exact projection mode, it avoids loading geometry of hidden parts (they stay in visualization mode).
  - In All projection modes, it avoids computing the projection and HiddenLineRemoval operation for these hidden parts.
  - This options leads to Memory and CPU gains.
  - This option is available as a property of views and in Tools/options for the default value.

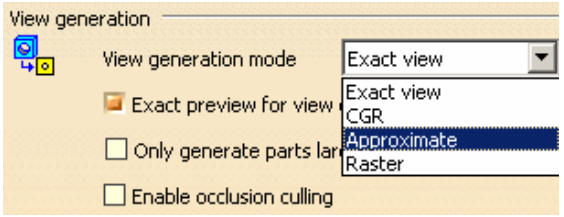
Student Notes:

General recommendations (2/2)

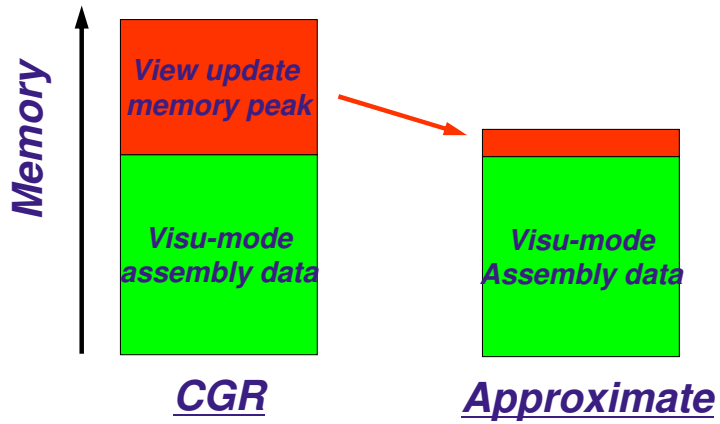
- Use Only generate parts larger than option:
  - Remove parts that are under a size defined in Tools/Options.
  - This parameter works for all generation modes (exact, CGR, Approximate, Raster).
  - This option is available as a property of views and in Tools/options for the default value.

Student Notes:

Approximate generation mode (1/3)



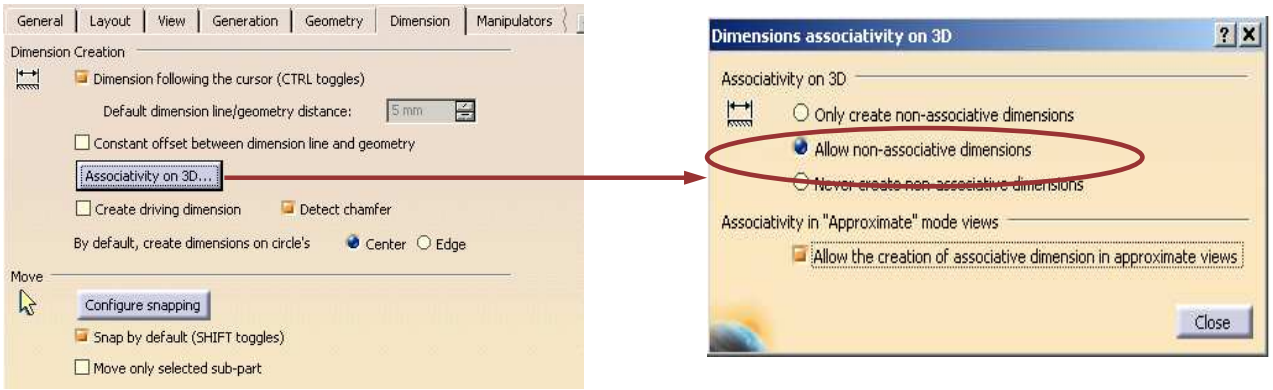
- "Approximate" generation mode
  - Similar to CGR generation mode
  - Generation mode available in Drafting settings
  - Works on visu-mode assembly data (cache mode active)
- Uses specific algorithm for projecting geometry
  - Reduces memory needed to update the view
    - Memory peak is less than 10% of assembly data size in memory
    - Memory allocation during update 10 times less compared to CGR mode



Student Notes:

Approximate generation mode (2/3)

- **Dimensions and dress-up elements**
  - **By default dimensions and dress-up elements are not associative to 3D. A specific setting must be checked.**
  - **The minimal geometrical information needed to create the associativity will be loaded (Selective Loading protocol).**
  - **The dimensions will measure the 3D elements, not the Approximated projected 2D elements.**



Student Notes:

Approximate generation mode (3/3)

- Limitations
  - No Thread generation
  - No Dimension generation
  - No Axis lines and Center-lines
  - No Fillet representation
  - No Detail Views
  - You cannot project 3D elements such as wireframe, points
  - Breakout profile has to be parallel to projection plane (no application of existing breakout on isometric views)

STUDENT GUIDE

Student Notes:

Administration tasks

Administrators can manage and customize standards such as ISO, JIS, ANSI, ASME, etc. or company standards. The Standards Editor lets administrators set the standards used for dress-up, dimensions, annotations, etc. as well as set the styles that will be used as defaults for element properties in the Interactive Drafting workbench.

In the Generative Drafting workbench, administration tasks consist in managing and customizing the styles that will be used to generate views using the standard Editor (generative view styles)

- About Standards and generative View styles
- Administration of Generative View Styles and Standards
- Setting standard parameters
- Setting standard styles
- Setting Generative View Style parameters
- Administration Settings

Student Notes:

About Standards and generative View styles (1/2)

- When users modify the properties of an element in the Interactive Drafting workbench, the modifications are only applied to the selected element, in the current drawing. Standard files let administrators set the properties of an element so that they will be applied to all elements of the same type in a drawing, as well as in all drawings which use a given standard.

  A standard file is an XML file which makes it possible to customize globally, for a CATDrawing, the appearance and behavior of drafting elements.

  With standard files, administrators can:
  - set standard styles that will be used as default values when creating new elements, i.e.:
    - define sheet styles, geometry styles, annotation styles, dimension styles, dress-up and dress-up symbols styles, callout styles.
  - set standard parameters, i.e.:
    - control the user interface with general parameters to restrict the values of some element properties, customize dimensions, annotations, dress up elements, dimension tolerance formats, dimension value formats, dress up elements, etc.

Student Notes:

About Standards and generative View styles (2/2)

- Generative view styles let you customize the appearance and behavior of a generated view via a set of pre-defined parameters and options.

  Administrators can create one or several generative view styles from which users can choose when creating a generative view.

  Generative view styles are defined in an XML file.

- By default, a pre-defined generative view style file is delivered. This file is located in install_root/resources/standard/generativeparameters/DefaultGenerativeStyle.xml.

  Administrators can customize this file to define their default generative view styles. They can also use this file as a template for creating new generative view styles.

  They can add as many generative view style files as needed. Refer to Administering Generative View Styles for more information.

Administration of Generative View Styles and Standards (1/5)

- **Location of Standards files: The location of the Standards files is defined by two environment variables which can be set during installation or modified afterwards (need to be logged as administrator and need to start V5 in administration mode):**
  - ***CATDefaultCollectionStandard:*** **Path and name of the directory (or directories) which contains:**
    - **the generativeparameters sub-directories (which contain the predefined generative view styles delivered by Dassault Systemes).**
    - **the drafting sub-directories (which contain the predefined drafting standards delivered by Dassault Systemes).**
    - **The default location for this directory (set during the installation process) is the installation directory install_root\resources\standard.**
  - ***CATCollectionStandard:*** **Path and name of the directory (or directories) which contains:**
    - **the generativeparameters sub-directories (which contain the customized generative view styles). It is in these generativeparameters sub-directories that you should add the generative view styles customized for a company, project or user.**
    - **the drafting sub-directories (which contain the customized drafting standards). It is in these drafting sub-directories that you should add the drafting standards customized for a company, project or user.**

Student Notes:

Student Notes:

Administration of Generative View Styles and Standards (2/5)

- Setting the location of generative view style files

There are two possibilities:

- If you want to place all customized generative view styles in a custom directory, named mydirectory for example, you need to proceed as follows:

1. Create a directory named as you like (mydirectory, for example).
2. Create a sub-directory under this directory, which needs to be named generativeparameters.
3. Place the XML files containing your customized generative view styles in mydirectory\generativeparameters.

- If you have not yet customized your XML generative view style files, then proceed as follows:

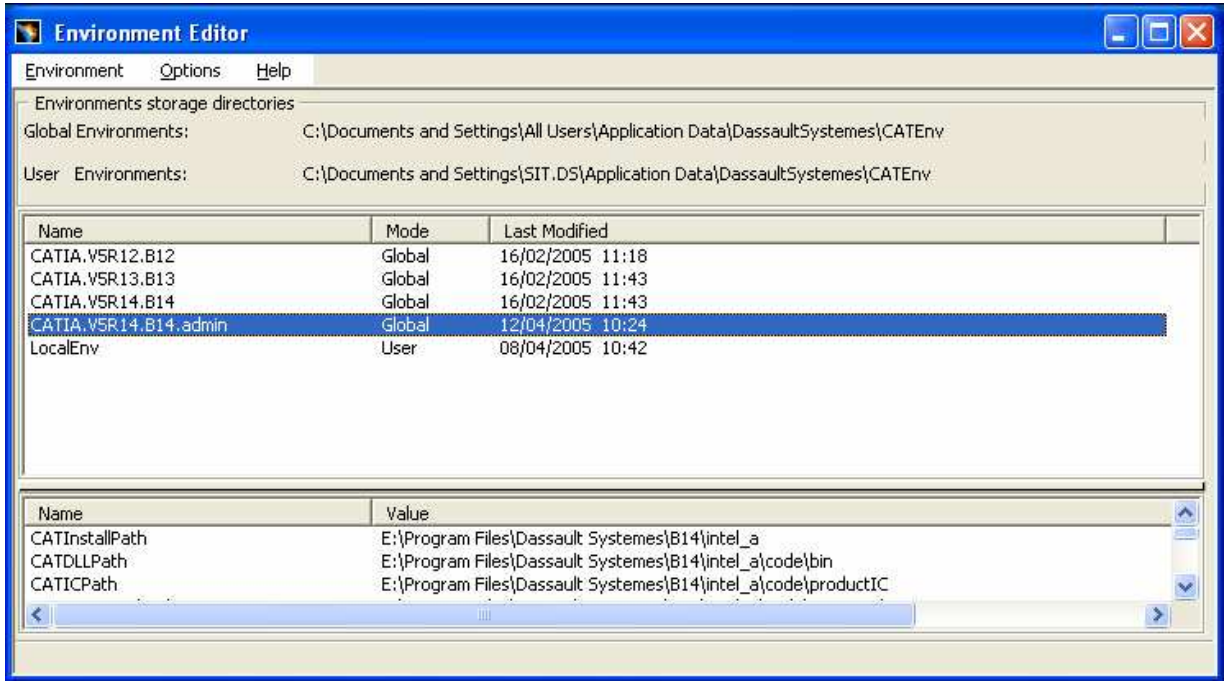
1. Create a directory named as you like (mydirectory, for example).
2. Create a sub-directory under this directory, which needs to be named generativeparameters.
3. Set the CATCollectionStandard variable to mydirectory. After you have customized the XML generative view style files, the standard editor will then save them in mydirectorygenerativeparameters.

- If the CATDefaultCollectionStandard and the CATCollectionStandard variables both contain an identically-named generative view styles, it is always the style found in CATCollectionStandard which will be used.

- If two directories referenced by the CATCollectionStandard and/or CATDefaultCollectionStandard variables contain identically-named generative view style files, it is always the style in the directory listed first which will be used.

Administration of Generative View Styles and Standards (3/5)

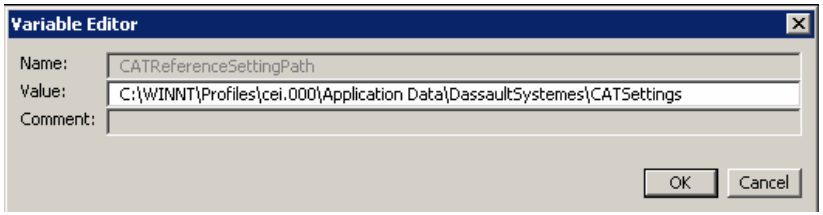
- **Customizing and defining Standards (1/3)**
  - **Editing and saving Standards files in Tools/Standards is only allowed when you are running CATIA V5 session in administrator mode (-admin).**
  - **The recommended method for customizing Standards files is the following:**
    - **launch the Environment Editor tool and create a new Environment called CATIA.V5R18.B16.admin (Global Mode)**



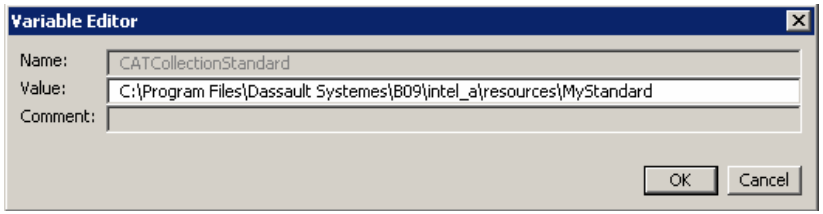
Student Notes:

Administration of Generative View Styles and Standards (4/5)

- **Customizing and defining Standards (2/3)**
  - **set up the CATReferenceSettingPath variable (use contextual menu on the variable)**



  - **set up the CATCollectionStandard variable (use contextual menu on the variable)**

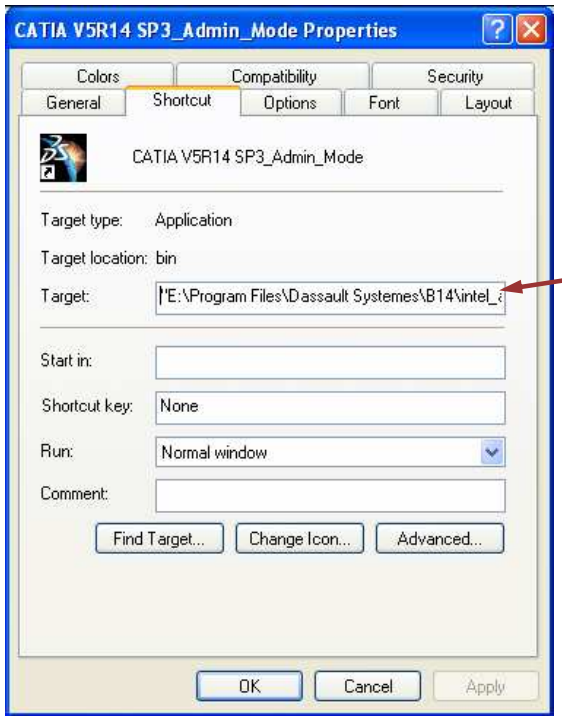


Student Notes:

Student Notes:

Administration of Generative View Styles and Standards (5/5)

- Customizing and defining Standards (3/3)
  - duplicate and/or modify the CATIA V5 shortcut icon using this new Environment in administrator mode (-admin). Use Contextual menu on the icon.



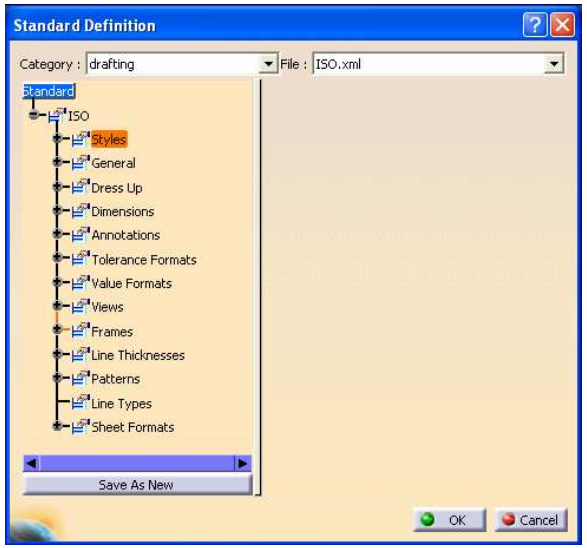
"C:\Program Files\Dassault Systemes\B18\intel_a\code\bin\CNEXT.exe" -admin -env CATIA.V5R18.B16.admin -direnv "C:\Document and Settings\All Users\Application Data\DassaultSystemes\CATEnv"

  - then use this new shortcut to launch CATIA V5 Session in administrator mode in order to customize Standards.

Student Notes:

Setting standard parameters (1/3)

- Structure of the Standard
  - A drafting standard file is structured as a tree, as it appears in the Standards Editor (available via Tools/Standards). It contains several main sections, each dealing with a specific aspect of drafting customization:
  - Styles
  - General parameters
  - Dress-up parameters
  - Dimension parameters
    - Company-defined dimension tolerance formats
    - Company-defined dimension value display formats
    - Pre-defined formats for tolerance and dimension values
  - Annotation parameters
  - Company-defined view generation
  - Company-defined frame formats
  - Company-defined line thickness
  - Company-defined patterns
  - Company-defined line-types
  - Company-defined sheet format

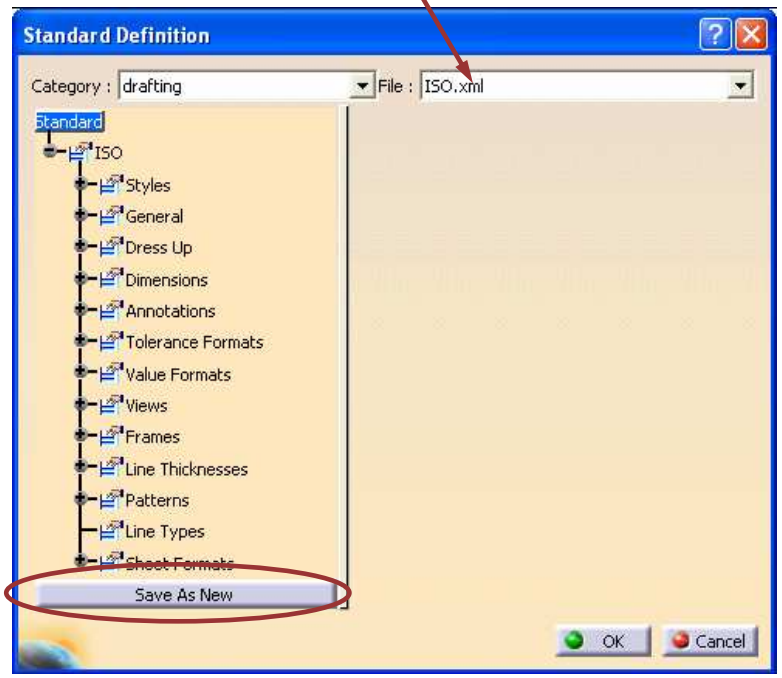


Student Notes:

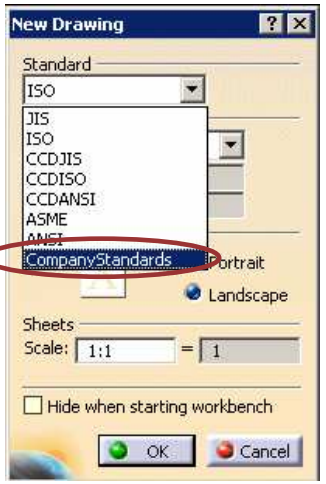
Setting standard parameters (2/3)

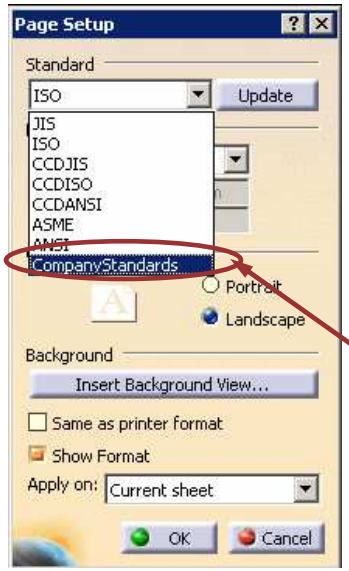
- Creating a new company standard
  - Using Tools/Standard command you can create your own standard based on one of the default ones (ISO, ANSI, ASME or JIS)
  - This will create a new XML file in the install_directory\B16\\intel_a\resources\standard\drafting directory:

1 Use *Tools/Standard* command and select one of the international standards. Make your modifications and click *Save As New* button to save your new standard



2a When you enter the Drafting Workbench, choose your company Standard from the list.





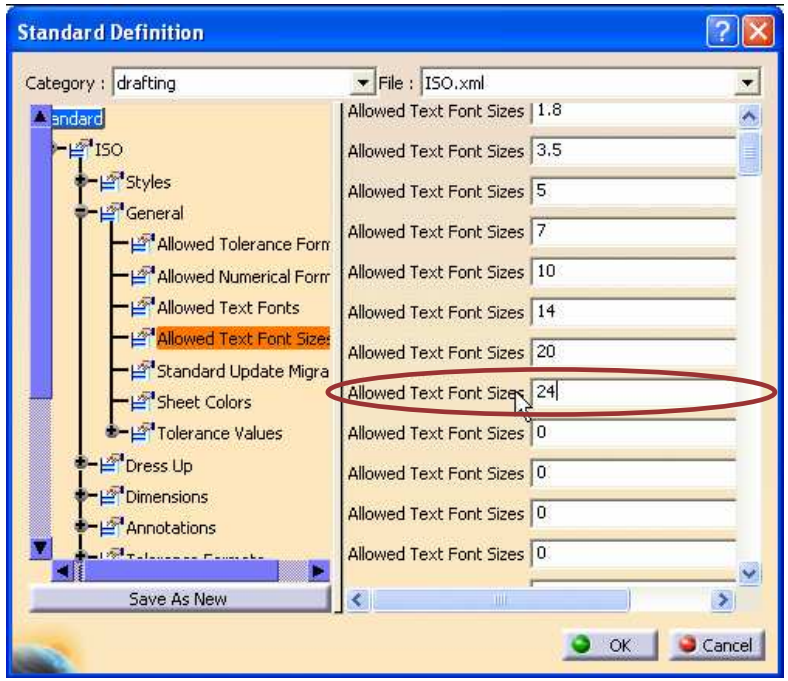
2b You can update an existing drawing with your company standard using *File/Page Setup* command.

Setting standard parameters (3/3)

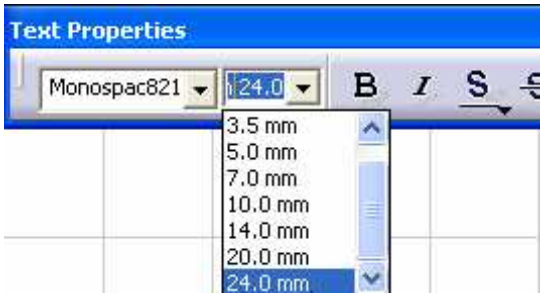
Student Notes:

Customizing standard parameters

1. Select *Tools/Standards* to launch the standards editor. Choose the drafting category, and then the ISO.xml file. Find the parameter to modify. You will find the detailed description of each parameter in the CATIA V5 Online Documentation, in Mechanical Design/ Interactive Drafting/Administrations Tasks/Setting Standard Parameters, e. g. add a new allowed text font size of 24mm.



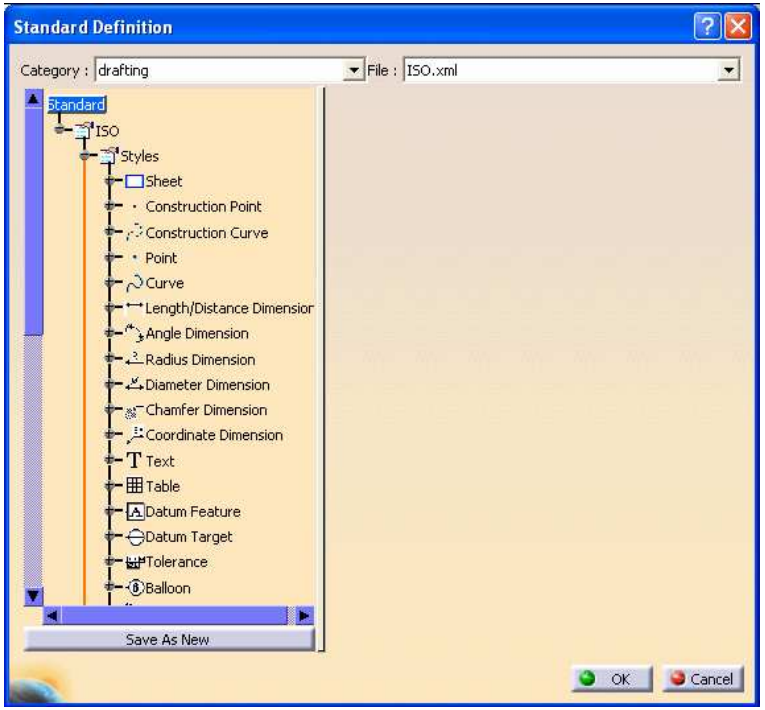
2. Create a new ISO drawing. The new allowed text font size will appear in text font sizes combo box.



Student Notes:

Setting standard styles (1/2)

- About Styles
  - Styles enable administrators to set the default values that will be applied to all properties of such elements as sheets, geometry, annotations, dimensions, dress-up and dress-up symbols, callouts, etc.
  - The default values are defined and stored in the standard XML file, where a set of new parameters are defined, one parameter for each element property whose default value can be set.
  - Default values are applied to elements as they are created. After creation, the user can modify element values as required.
  - If you modify styles in the standard itself and then update the standard file used by the drawing, the elements which have already been created will NOT be modified (i.e. their default values will remain as previously). Updating the standard will only have an impact on the next elements to be created.
  - By default, one style named Default is predefined in the standard files for each type of element.

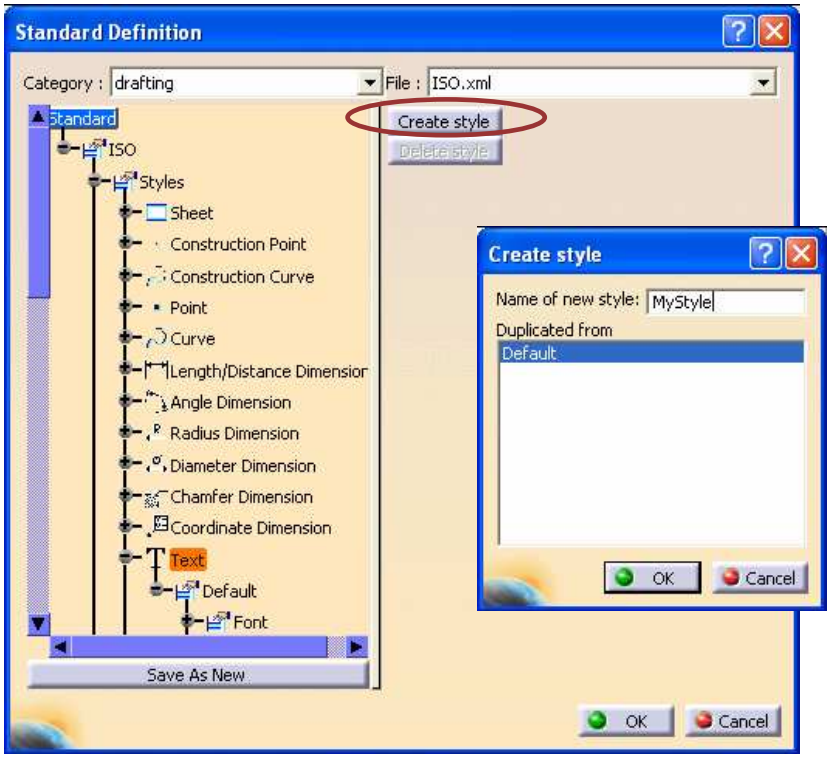


Student Notes:

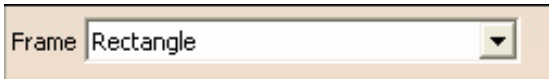
Setting standard styles (2/2)

- **Customizing styles**
  - You will see an example of how to create a new style for Texts. Note that a new style is always based on an existing style.

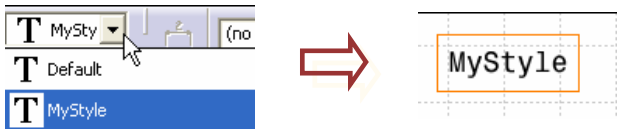
1. In the Style node select the Text node and click *Create Style* button. Type the name of the new style. A new style called "MyStyle" is added under the Text node in the editor.



2. Expand the Text node in the editor, and then select the Frame node. Choose Rectangle from the list



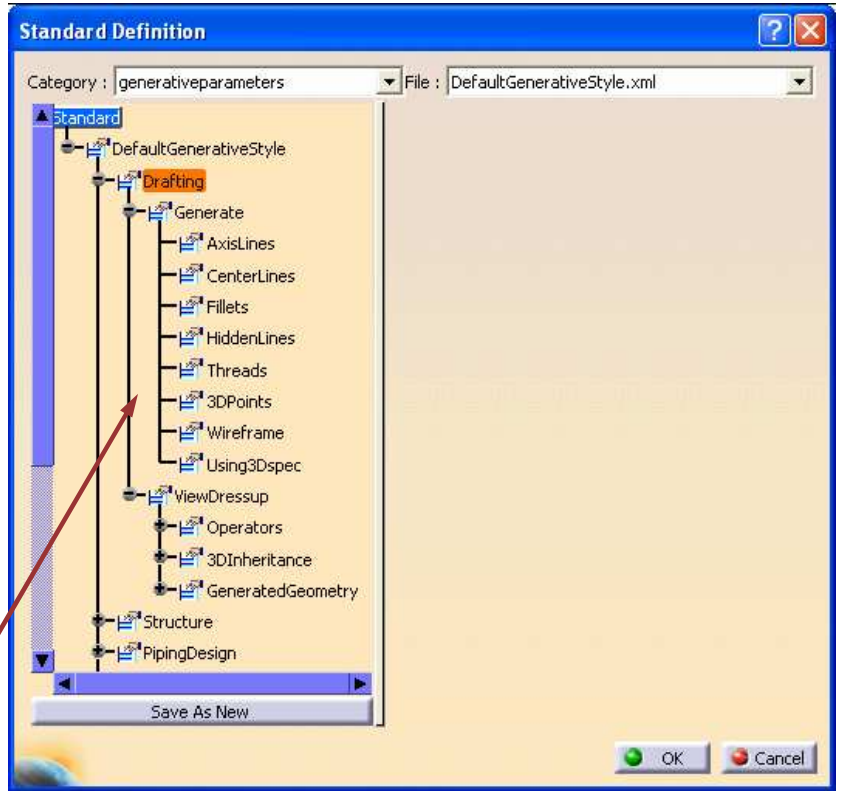
3. Start creating a new text in a sheet. In the Style toolbar, you can notice that two styles are now available: *Default* and *MyStyle*.



Student Notes:

Setting Generative View Styles (1/3)

- Structure of the Generative View Styles
  - A generative view style file is structured as a tree, as it appears in the Standard Definition dialog box (available via Tools/Standards). It contains one section dedicated to generative drafting customization called DefaultGenerativeStyle/ Drafting
  - You will find two sub-sections:
    - Generate parameters: specifies whether the elements will be projected in the view or not.
    - View dress-up parameters: defines the style of the various parameters which deal with the dress-up of the view.



You will find the detailed description of each generative view style parameter in the CATIA V5 Online Documentation, in Mechanical Design/ Generative Drafting/Administrations Tasks/Setting Generative View Style Parameters

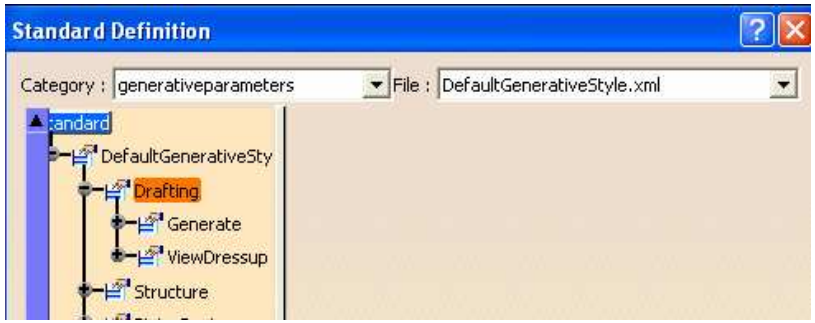
Student Notes:

Setting Generative View Styles (2/3)

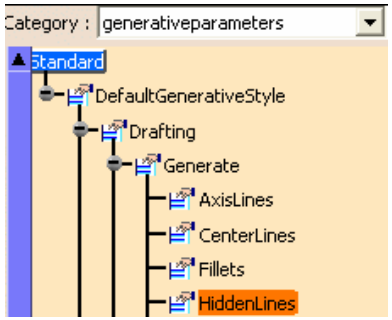
- **Customizing Generative View Styles**
  - You will learn how to customize generative view style parameters using a specific example. The pre-defined DefaultGenerativeStyle.xml file specifies that hidden edges are not visible and are in black color. You will modify these parameters so that hidden edges are always visible and displayed in orange.

1. Go to *Tools/Options/ Mechanical Design/ Drafting/ Administration* tab, and uncheck the *Prevent generative view style creation* option. This activates the generative view style functionalities.

2. Select *Tools/ Standards* to launch the Standard Definition dialog box. Choose the generativeparameters category, and then open the DefaultGenerativeStyle.xml file from the list.



3. Expand the node *DefaultGenerativeStyle/Drafting/Generate/HiddenLines* and set the parameter value to Yes

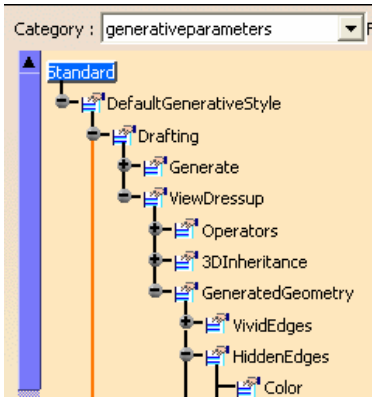


Student Notes:

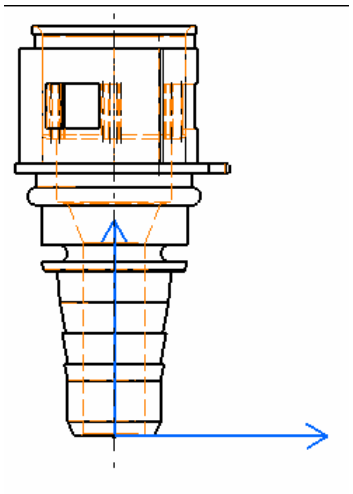
Setting Generative View Styles (3/3)

Customizing Generative View Styles

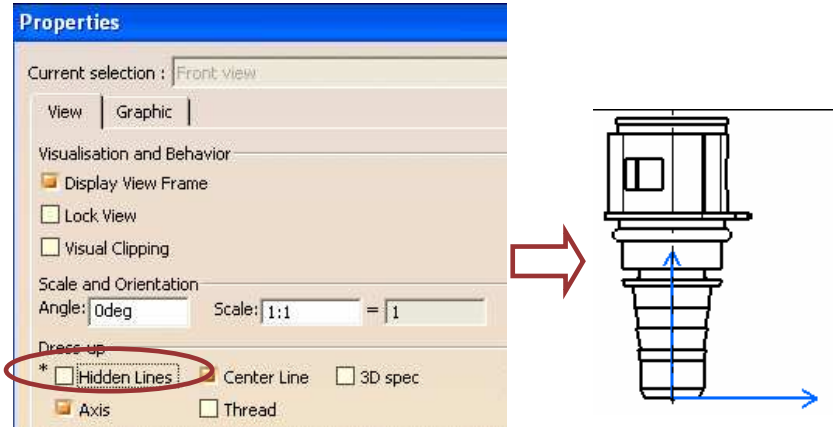
4 Expand the node. *DefaultGenerativeStyle/Drafting/View Dressup/GeneratedGeometry/HidenEdges* and set the color parameter to orange.



5 Create a front view. The view is generated, taking into account the new parameters you have defined.



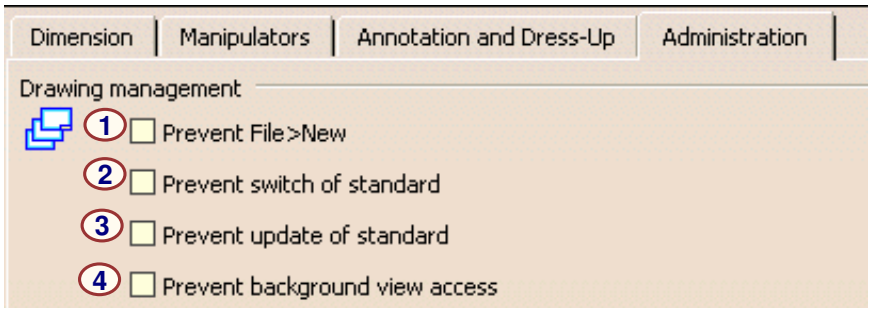
6 After the view creation you can still overload the generative view style in the Properties panel. In this case a star character is displayed.



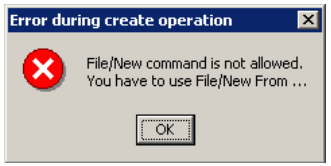
Student Notes:

Administration Settings (1/4)

- **Drawing management**

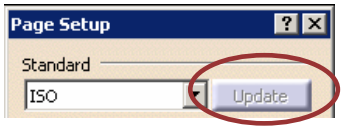


1. **Select this option to make it impossible to create drawings using the File > New command. All drawings will be created using the File > New From... command instead.**



2. **Select this option to make it impossible to change standards, i.e. to use a standard other than the one currently defined in the Page Setup dialog box.**

3. **Select this option to make it impossible to update standards for the current document in the Page Setup dialog box.**

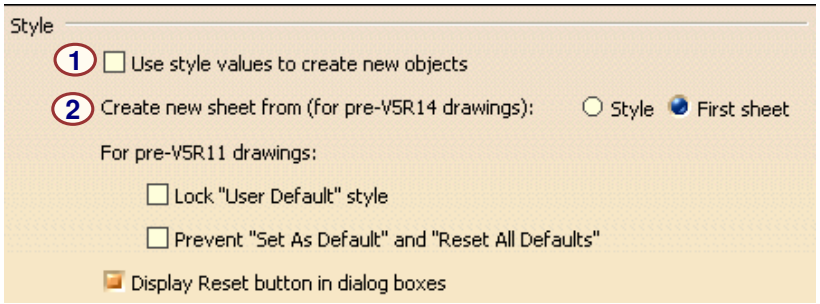


4. **Select this option to make it impossible to access the background view.**

Student Notes:

Administration Settings (2/4)

- **Style**



1 Select this option if you want dialog boxes, Properties toolbars and the Tools Palette to be pre-filled with custom style values (as defined in the Standards Editor) when creating new annotations. In this case, Properties toolbars and the Tools Palette will be disabled during the creation of the annotation. If you leave this box unchecked, annotation dialog boxes, Properties toolbars and the Tools Palette will be pre-filled with the last entered values (except for Texts, Texts with leader, Balloons and Datum features). In this case, Properties toolbars and the Tools Palette will be active during the creation of the annotation. If you select this option, you will be able to reset the current style values in dialog boxes at any time using the Reset button unless it is disabled.

2 This option lets you specify if the properties used for creating new sheets should be those defined in the standards or those defined in the first sheet of a drawing. These properties are the scale and the projection method (first or third angle).

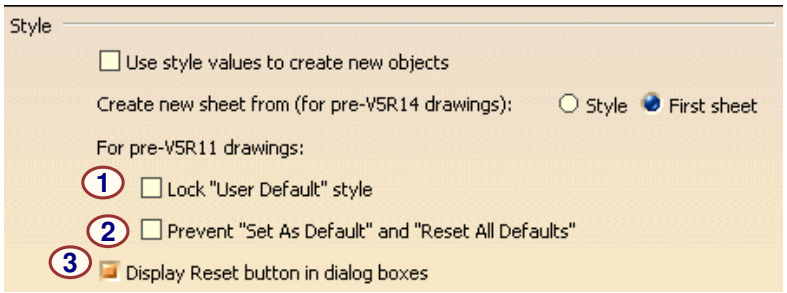
Select Style if you want the sheet to use the style defined in the standards (in Tools -> Standards -> Drafting -> [StandardName] -> Styles -> Sheet).

Select First sheet if you want the sheet to use the properties defined in the first sheet of a drawing. For example, you can use this option if you use an existing drawing to create a new one (i.e. when you want the new drawing to have the same properties as the existing drawing).

Student Notes:

Administration Settings (3/4)

Style



1. Select this option to make it compulsory to use User Defaults (i.e., user-defined values set as default). The Styles drop-down list will be set to Only User Defaults and will be inactive so that Original Defaults or User Defaults cannot be selected.

   This option applies only to drawings created with versions up to V5 R10 whose standard has NOT been updated or changed in V5 R11 and later.

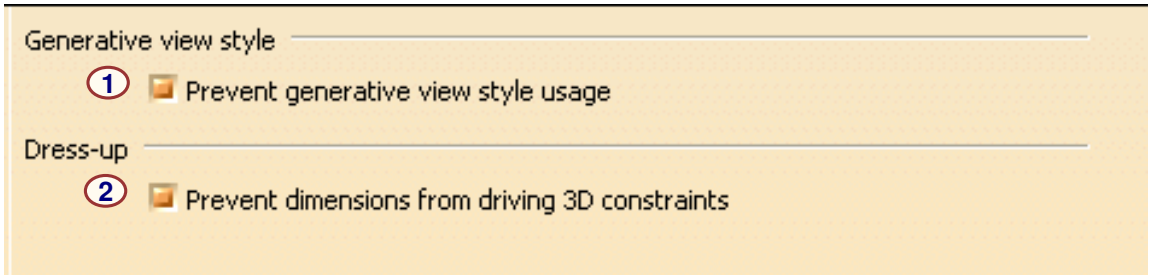
2. Select this option to use the current defaults and to make it impossible to create, change and reset user defaults (i.e. user-defined values). This disables the Set as Default and the Reset All Defaults commands.

   This option applies only to drawings created with versions up to V5 R10 whose standard has NOT been updated or changed in V5 R11 and later.

3. Select this option to display the Reset button in dialog boxes. Deselecting this option hides the Reset button in dialog boxes and disables the Reset functionality.

Student Notes:

Administration Settings (4/4)



1 Select this option if you do not want to use generative view styles when creating or updating views.

Note that this option also drives view generation parameters when updating views, whether the view was created using generative view styles or not. In this case, there are two possibilities:
• If you created the view using generative view styles (i.e. with this option selected) and then de-select this option, then, at the next update, the view properties (available via Edit -> Properties) will be used for all overloaded parameters (an asterisk * appears in the Properties dialog box in front of the parameters which are considered as being overloaded) and the view settings (available via Tools -> Options -> Mechanical Design -> Drafting - > View tab) will be used for the others.
• If you created the view without generative view styles (i.e. without this option selected) and then select this option, then, at the next update, all parameters available in the view properties will be used. To make sure that generative view styles are used in this case, click the Reset to style values button in the Properties dialog box.

2 Select this option to make it impossible to modify a 3D constraint via a 2D dimension that was generated from it.

Student Notes:

Increasing Productivity

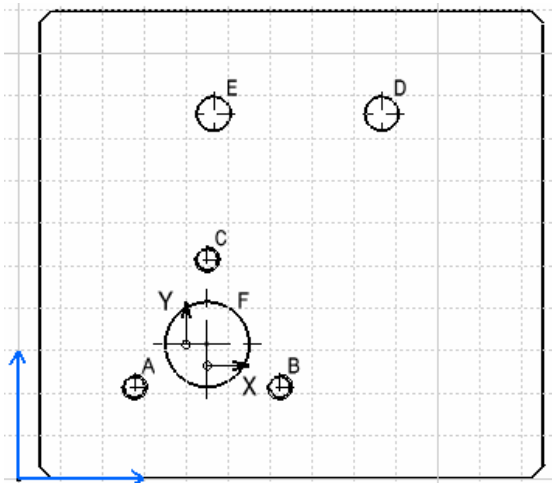
You will become familiar with …

- Generating Hole Dimensions Tables
- Creating Point Coordinates Table
- Creating a Table

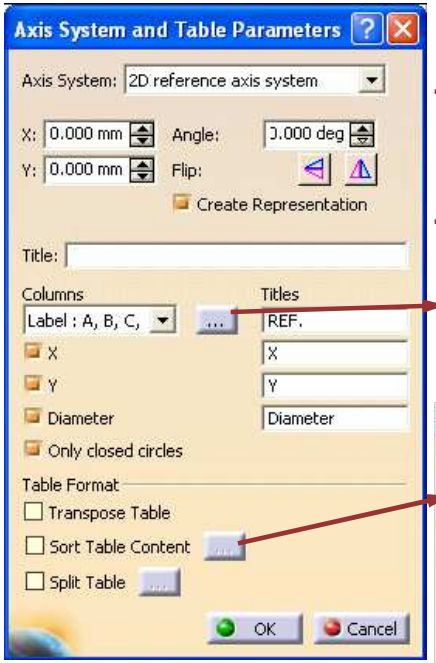
Student Notes:

Generating Hole Dimensions Tables

- You have the capability to generate a table which contains holes coordinates according to a specific origin.

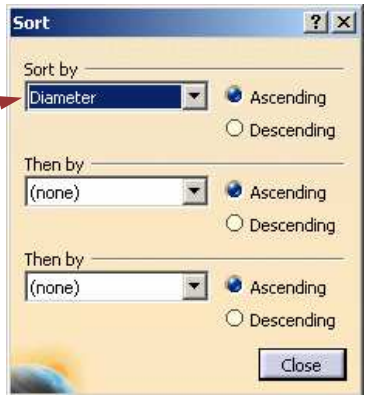


Holes Coordinates Table			
REF.	X	Y	Diameter
A	-17.32	-10	4.92
B	17.32	-10	4.92
C	0	20	4.92
D	41.5	54.26	8
E	1.5	54.26	8
F	0	0	20



You can choose the 2D axis system of the view or define an other one interactively by selecting a point, two lines or indicating a point by clicking in the view.

You can set the first label value used to reference the holes.



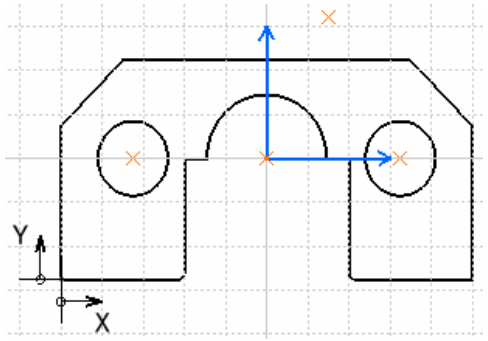
You can specify the way to sort the table.

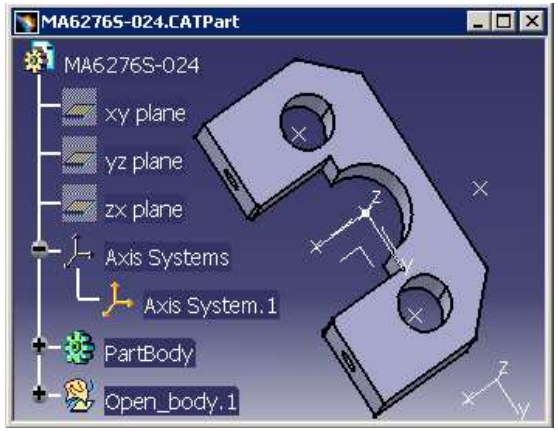
For threaded holes, the diameter is the hole diameter and not the thread diameter.

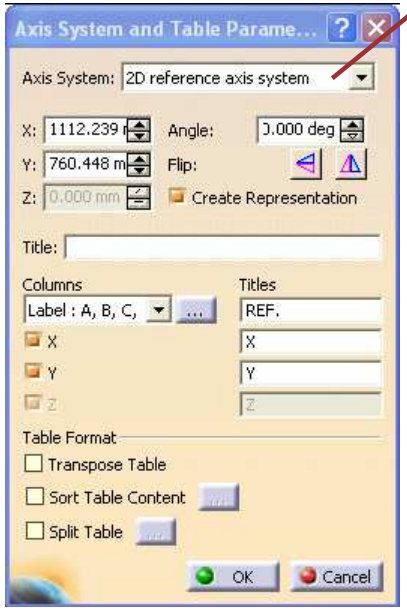
There is no associativity with the 3D data, if you make modification on the holes, you need to delete the table and regenerate a new one.

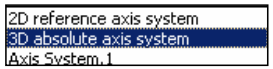
Creating Point Coordinates Tables

- You have the capability to generate a table which contains points coordinates according to a specific origin.









In this case, you can also choose the 3D axis system (in this case it is the absolute axis of the 3D model), or if the model is a single part, you can choose a local axis.

Points Coordinates Table

REF.	X	Y
A	50	27.63
B	65	60
C	17.5	27.63
	82.5	27.63
	50	27.63

2D reference axis system
3D absolute axis system
Axis System.1

Points Coordinates Table

REF.	X	Y	Z
A	-60	65	35
B	-27.63	17.5	0
	-27.63	82.5	0
	-27.63	50	25

2D reference axis system
3D absolute axis system
Axis System.1

Points Coordinates Table

REF.	X	Y	Z
A	-32.37	15	10
B	0	-32.5	-25
	0	32.5	-25
	0	0	0

2D reference axis system
3D absolute axis system
Axis System.1

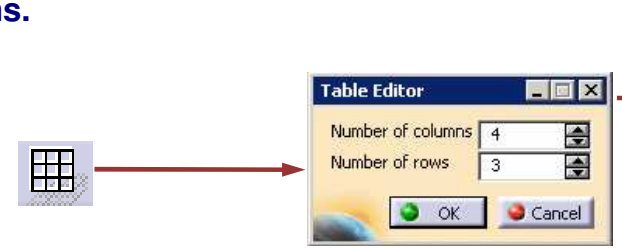
There is no associativity with the 3D document. If you make modification on the holes, you need to delete the table and regenerate it.

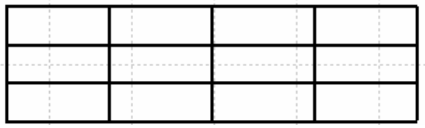
Student Notes:

Student Notes:

Creating Tables

- You can create a Table and add in it texts and views. You will be able to insert columns, rows, merge cells, invert lines, invert columns or switch lines and columns.

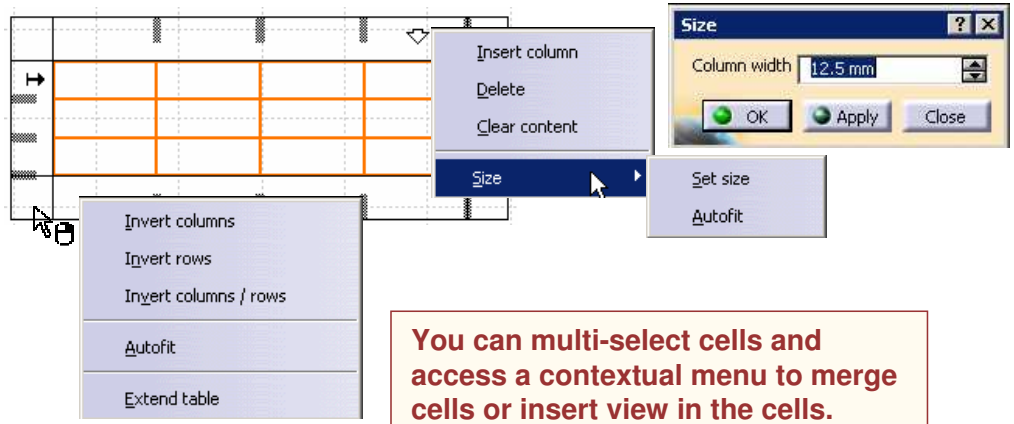




Line Height depends on the text size set in the Text Properties toolbar and Line Width is 5 times ones text size.

You can double-click the table to edit it. You can select rows and columns and modify them by using the contextual menu.

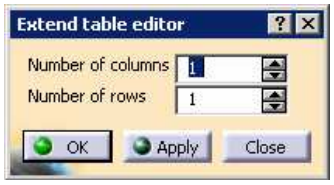
When the table is in edition mode, you can not move it anymore.

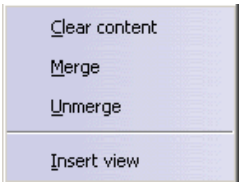


Remark1: you retrieve the same properties than the Text concerning snapping and positional capabilities.

Remark2: you can import CSV Table.

You can multi-select cells and access a contextual menu to merge cells or insert view in the cells.





Student Notes:

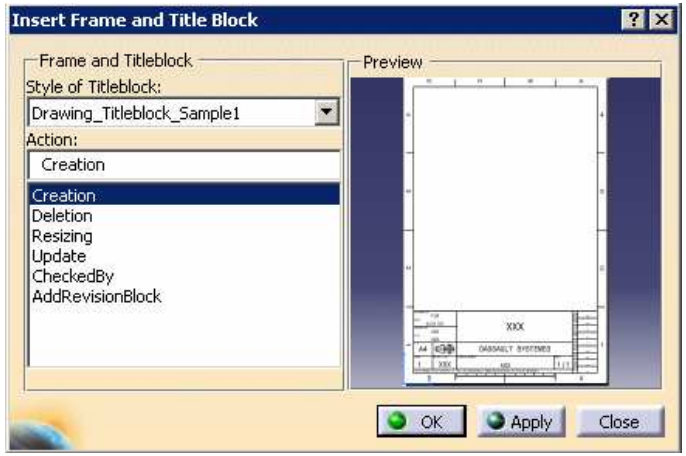
Creating Frames & Title Blocks with a Macro

You will become familiar with ...

- Defining Frames
- Filling in the Title Block

Defining Frames (1/14)

- **Standard title block macro**
  - You have the possibility to use a VBscript macro to generate Title Blocks automatically adjusted to your drawings formats.
  - You can access different styles of Title Blocks using the Frame Creation function.
  - This macro allows you to create, delete, resize or update title Blocks and to add some information like the name of the person who checks the drawings and the revision blocks.



Student Notes:

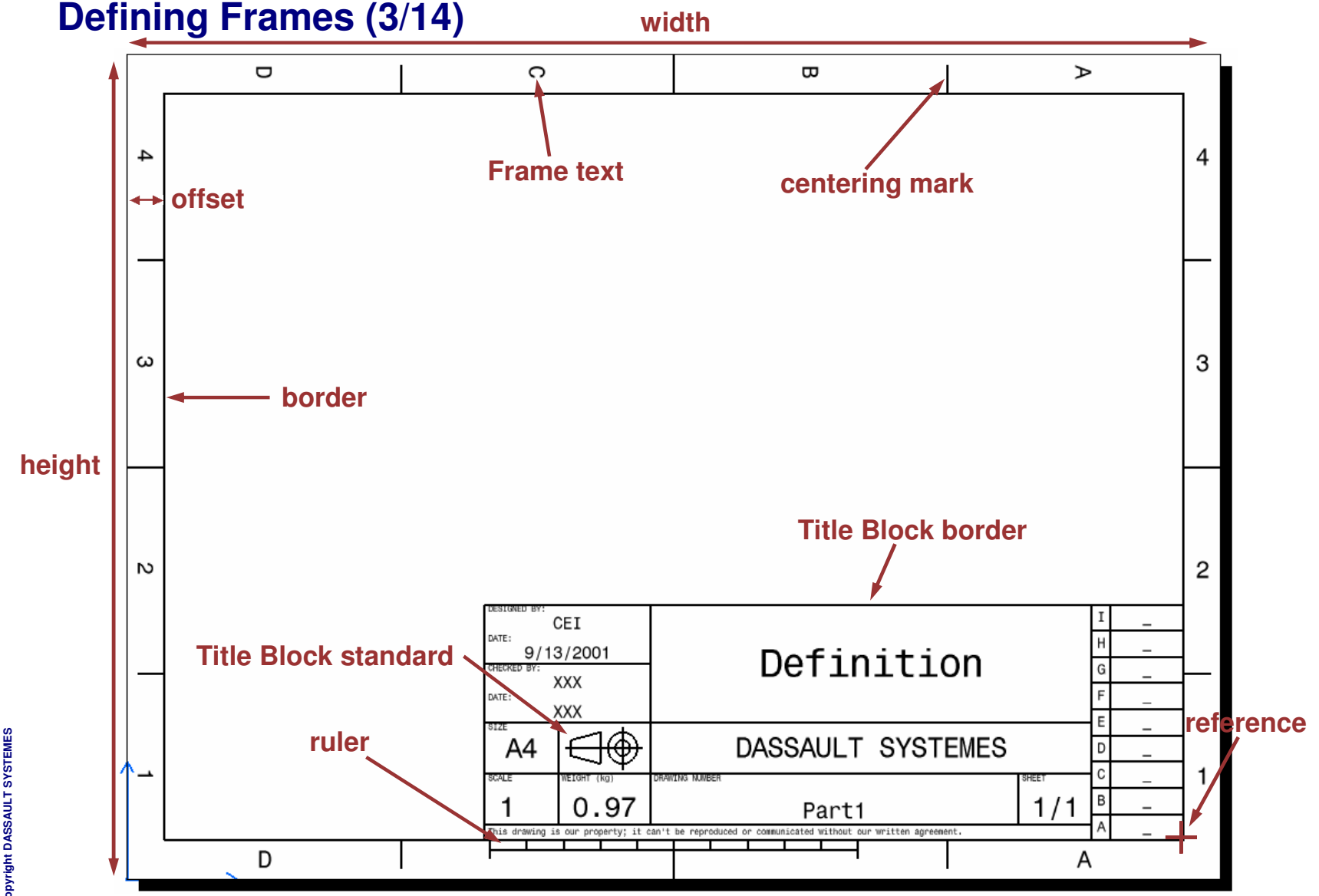
Student Notes:

Defining Frames (2/14)

- The title block macro is delivered with CATIA V5. You will find it in the install_root/intel_a/VBScript/FrameTitleBlock directory.
- We will explain in the following pages the macro instructions that generate the frames

```
Sub CATDrw_Creation()
  '-------------------------------------------------------------------------
  'How to create the FTB
  '-------------------------------------------------------------------------
  CATInit                          'To init public variables
  If CATCheckRef(1) Then Exit Sub 'To check whether a FTB exists already
  CATStandard                      'To compute standard sizes
  CATReference                     'To place on the drawing a reference point
  CATFrame                         'To draw the frame
  CATTitleBlock                    'To draw the TitleBlock and fill in it

End Sub
```

Defining Frames (3/14)



Student Notes:

Student Notes:

Defining Frames (4/14)

- How to init the dialog and create main objects

Here is the way to define main variables.

```
Public DrwDocument  As DrawingDocument
Public DrwSheets    As DrawingSheets
Public Selection    As Selection
Public DrwSheet     As DrawingSheet
Public DrwView      As DrawingView
Public DrwTexts     As DrawingTexts
Public Fact         As Factory2D
Public GeomElems    As GeometricElements
```

First of all, you need to activate the sheet and the view of your drawing if you want to create here the Title Block.

```
Sub CATInit()
   '-------------------------------------------------------------------
   'How to init the dialog and create main objects
   '-------------------------------------------------------------------
   Set DrwDocument = CATIA.ActiveDocument
   Set DrwSheets   = DrwDocument.Sheets
   Set Selection   = DrwDocument.Selection
   Set DrwSheet    = DrwSheets.ActiveSheet
   Set DrwView     = DrwSheet.Views.ActiveView
   Set DrwTexts    = DrwView.Texts
   Set Fact        = DrwView.Factory2D
   Set GeomElems   = DrwView.GeometricElements

End Sub
```

Student Notes:

Defining Frames (5/14)

- How to define frames overall dimensions

```
Sub CATStandard()
  '-------------------------------------------------------------------------------
  'How to compute standard values
  '-------------------------------------------------------------------------------
  Height = DrwSheet.GetPaperHeight
  Width  = DrwSheet.GetPaperWidth

  Offset = 10.*mm 'Offset default value = 10
  If DrwSheet.PaperSize = CatPaperA0 Or _
     DrwSheet.PaperSize = CatPaperA1 Or _
     DrwSheet.PaperSize = CatPaperUser And _
     (DrwSheet.GetPaperWidth > 594.*mm Or DrwSheet.GetPaperHeight > 594.*mm) Then
    Offset = 20.*mm
  End If

  OH = Width - Offset
  OV = Offset

End Sub
```

```
Public Height       As Double           'Sheet height
Public Width        As Double           'Sheet width
Public Offset       As Double           'Distance between the sheet edges and the frame borders
Public OH           As Double           'Horizontal origin for drawing the titleblock
Public OV           As Double           'Vertical   origin for drawing the titleblock
```

Student Notes:

Defining Frames (6/14)

- How to define a reference text and check that the called macro is the right one

You have to create a reference Text which will be used to check if you can make some actions like delete, update or resize on the existing Title Block. This Reference Text will identify the macro used to create the existing Title Block of your drawing.

If you want to add a new macro to create your customized Title Block, you have to rename the MacroID variable like the macro name.

```
Sub CATReference()
  '-------------------------------------------------------------
  'How to create a reference text
  '-------------------------------------------------------------
  Set Text = DrwTexts.Add("", Width - Offset, Offset)
  Text.Name = "Reference_" + MacroID

End Sub
```

```
Const MacroID       = "Drawing_Titleblock_Sample1"
```

When you want to make an action on an existing macro, a test will look for the using Reference Text.

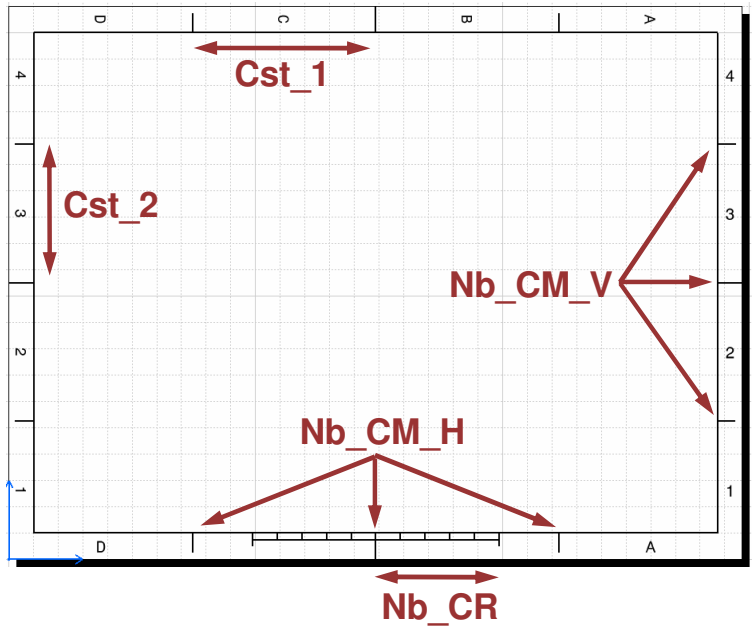
```
Function CATCheckRef(Mode As Integer) As Integer
  '-------------------------------------------------------------------------------
  'How to check that the called macro is the right one
  '-------------------------------------------------------------------------------
  nbTexts = DrwTexts.Count
  i = 0
  notFound = 0
  While (notFound = 0 And i<nbTexts)
    i = i + 1
    Set Text = DrwTexts.Item(i)
    WholeName = Text.Name
    leftText = Left(WholeName, 10)
    If (leftText = "Reference_") Then
      notFound = 1
      refText = "Reference_" + MacroID
      If (Mode = 1) Then
        MsgBox "Frame and Titleblock already created!"
        CATCheckRef = 1
        Exit Function
      ElseIf (Text.Name <> refText) Then
        MsgBox "Frame and Titleblock created using another style:" + Chr(10) + "         " + MacroID
        CATCheckRef = 1
        Exit Function
      End If
    End If
  Wend
  CATCheckRef = 0
```

Student Notes:

Defining Frames (7/14)

- How to create the frame

The frame is composed of a border, some centering marks, letters and a ruler.



Standard values.

```
Sub CATFrame()
  '-------------------------------------------------------------------
  'How to create the Frame
  '-------------------------------------------------------------------
  Dim Cst_1   As Double  'Length (in cm) between 2 horinzontal marks
  Dim Cst_2   As Double  'Length (in cm) between 2 vertical marks
  Dim Nb_CM_H As Integer 'Number/2 of horizontal centring marks
  Dim Nb_CM_V As Integer 'Number/2 of vertical centring marks
  Dim Ruler   As Integer 'Ruler length (in cm)

  CATFrameStandard      Nb_CM_H, Nb_CM_V, Ruler, Cst_1, Cst_2
  CATFrameBorder
  CATFrameCentringMark  Nb_CM_H, Nb_CM_V, Ruler, Cst_1, Cst_2
  CATFrameText          Nb_CM_H, Nb_CM_V, Ruler, Cst_1, Cst_2
  CATFrameRuler         Ruler, Cst_1

End Sub
```

Student Notes:

Defining Frames (8/14)

- How to define standard values in relation to the sheet format

This subroutine allows to compute the standard values in relation to the drawing format and orientation.

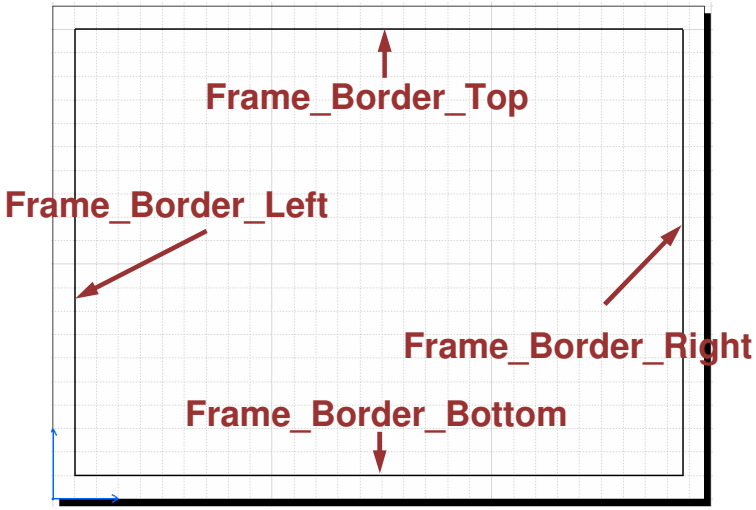
```
Public Text          As DrawingText
Public Point         As Point2D
Public Line          As Line2D
Public Circle        As Circle2D
```

```
Sub CATFrameStandard(Nb_CM_H As Integer, Nb_CM_V As Integer, Ruler As Integer, _
                     Cst_1 As Double, Cst_2 As Double)
  '-------------------------------------------------------------------------------
  'How to compute standard values
  '-------------------------------------------------------------------------------
  Cst_1 = 74.2*mm '297, 594, 1189 are multiples of 74.2
  Cst_2 = 52.5*mm '210, 420, 841  are multiples of 52.2
  If DrwSheet.Orientation = CatPaperPortrait And _
     (DrwSheet.PaperSize = CatPaperA0 Or _
      DrwSheet.PaperSize = CatPaperA2 Or _
      DrwSheet.PaperSize = CatPaperA4) Or _
      DrwSheet.Orientation = CatPaperLandscape And _
     (DrwSheet.PaperSize = CatPaperA1 Or _
      DrwSheet.PaperSize = CatPaperA3) Then
     Cst_1 = 52.5*mm
     Cst_2 = 74.2*mm
  End If

  Nb_CM_H = CInt(.5 * Width / Cst_1)
  Nb_CM_V = CInt(.5 * Height / Cst_2)

  Ruler   = CInt((Nb_CM_H - 1) * Cst_1 / 50) * 100 'maximum ruler length
  If RulerLength < Ruler Then
    Ruler = RulerLength
  End If

End Sub
```

Student Notes:

Defining Frames (9/14)

- How to draw the frame border

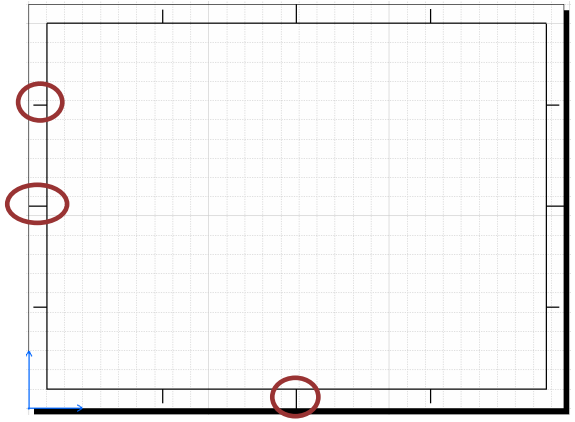


All the lines are created by using the reference as origin point.

```
Sub CATFrameBorder()
  '-------------------------------------------------------------------------
  'How to draw the frame border
  '-------------------------------------------------------------------------
  On Error Resume Next
  Set Line = Fact.CreateLine(OV, OV              , OH, OV              )
  Line.Name = "Frame_Border_Bottom"
  Set Line = Fact.CreateLine(OH, OV              , OH, Height - Offset)
  Line.Name = "Frame_Border_Left"
  Set Line = Fact.CreateLine(OH, Height - Offset, OV, Height - Offset)
  Line.Name = "Frame_Border_Top"
  Set Line = Fact.CreateLine(OV, Height - Offset, OV, OV              )
  Line.Name = "Frame_Border_Right"
  If Err.Number <> 0 Then
    Err.Clear
  End If

End Sub
```

Defining Frames (10/14)

- How to draw centering marks

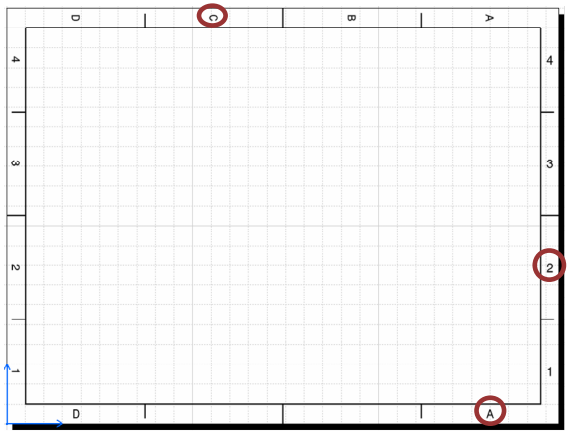


```
Sub CATFrameCentringMark(Nb_CM_H As Integer, Nb_CM_V As Integer, _
                         Ruler As Integer, Cst_1 As Double, Cst_2 As Double)
  '-------------------------------------------------------------------------
  'How to draw the centring marks
  '-------------------------------------------------------------------------
  On Error Resume Next
    Set Line = Fact.CreateLine(.5 * Width    , Height - Offset, .5 * Width, Height     )
    Line.Name = "Frame_CentringMark_Top"
    Set Line = Fact.CreateLine(.5 * Width    , OV             , .5 * Width, .0         )
    Line.Name = "Frame_CentringMark_Bottom"
    Set Line = Fact.CreateLine(OV            , .5 * Height    , .0        , .5 * Height)
    Line.Name = "Frame_CentringMark_Left"
    Set Line = Fact.CreateLine(Width - Offset, .5 * Height    , Width     , .5 * Height)
    Line.Name = "Frame_CentringMark_Right"

    For i = Nb_CM_H To Ruler/2/Cst_1 Step -1
      If (i * Cst_1 < .5 * Width - 1.) Then
        Set Line  = Fact.CreateLine(.5 * Width + i * Cst_1, OV, .5 * Width + i * Cst_1, .25 * Offset)
        Line.Name = "Frame_CentringMark_Bottom"
        Set Line  = Fact.CreateLine(.5 * Width - i * Cst_1, OV, .5 * Width - i * Cst_1, .25 * Offset)
        Line.Name = "Frame_CentringMark_Bottom"
      End If
    Next

    For i = 1 To Nb_CM_H
      If (i * Cst_1 < .5 * Width - 1.) Then
        Set Line  = Fact.CreateLine(.5 * Width + i * Cst_1, Height - Offset, .5 * Width + i * Cst_1, Height - .25 * Offset)
        Line.Name = "Frame_CentringMark_Top"
        Set Line  = Fact.CreateLine(.5 * Width - i * Cst_1, Height - Offset, .5 * Width - i * Cst_1, Height - .25 * Offset)
        Line.Name = "Frame_CentringMark_Top"
      End If
    Next
```

Student Notes:

Defining Frames (11/14)

- How to add texts around the border frame




```
Sub CATFrameText(Nb_CM_H As Integer, Nb_CM_V As Integer, Ruler As Integer, _
                 Cst_1 As Double, Cst_2 As Double)
  '-------------------------------------------------------------------------------
  'How to create coordinates
  '-------------------------------------------------------------------------------
  On Error Resume Next
    For i = Nb_CM_H To (Ruler/2/Cst_1 + 1) Step -1
      Set Text = DrwTexts.Add(Chr(65 + Nb_CM_H - i) , .5 * Width + (i - .5) * Cst_1, .5 * Offset)
      CATFormatFText "Frame_Text_Bottom", 0
      Set Text = DrwTexts.Add(Chr(64 + Nb_CM_H + i) , .5 * Width - (i - .5) * Cst_1, .5 * Offset)
      CATFormatFText "Frame_Text_Bottom", 0
    Next

    For i = 1 To Nb_CM_H
      Set Text = DrwTexts.Add(Chr(65 + Nb_CM_H - i), .5 * Width + (i - .5) * Cst_1, Height - .5 * Offset)
      CATFormatFText "Frame_Text_Top", -90
      Set Text = DrwTexts.Add(Chr(64 + Nb_CM_H + i), .5 * Width - (i - .5) * Cst_1, Height - .5 * Offset)
      CATFormatFText "Frame_Text_Top", -90
    Next

    For i = 1 To Nb_CM_V
      Set Text = DrwTexts.Add(CStr(Nb_CM_V + i)     , .5 * Offset        , .5 * Height + (i - .5) * Cst_2)
      CATFormatFText "Frame_Text_Left", -90
      Set Text = DrwTexts.Add(CStr(Nb_CM_V - i + 1) , .5 * Offset        , .5 * Height - (i - .5) * Cst_2)
      CATFormatFText "Frame_Text_Left", -90
      Set Text = DrwTexts.Add(CStr(Nb_CM_V + i)    , Width - .5 * Offset, .5 * Height + (i - .5) * Cst_2)
      CATFormatFText "Frame_Text_Right", 0
      Set Text = DrwTexts.Add(CStr(Nb_CM_V - i + 1), Width - .5 * Offset, .5 * Height - (i - .5) * Cst_2)
      CATFormatFText "Frame_Text_Right", 0
    Next

  If Err.Number <> 0 Then
    Err.Clear
  End If

End Sub
```

Student Notes:

Defining Frames (12/14)

- How to create a ruler

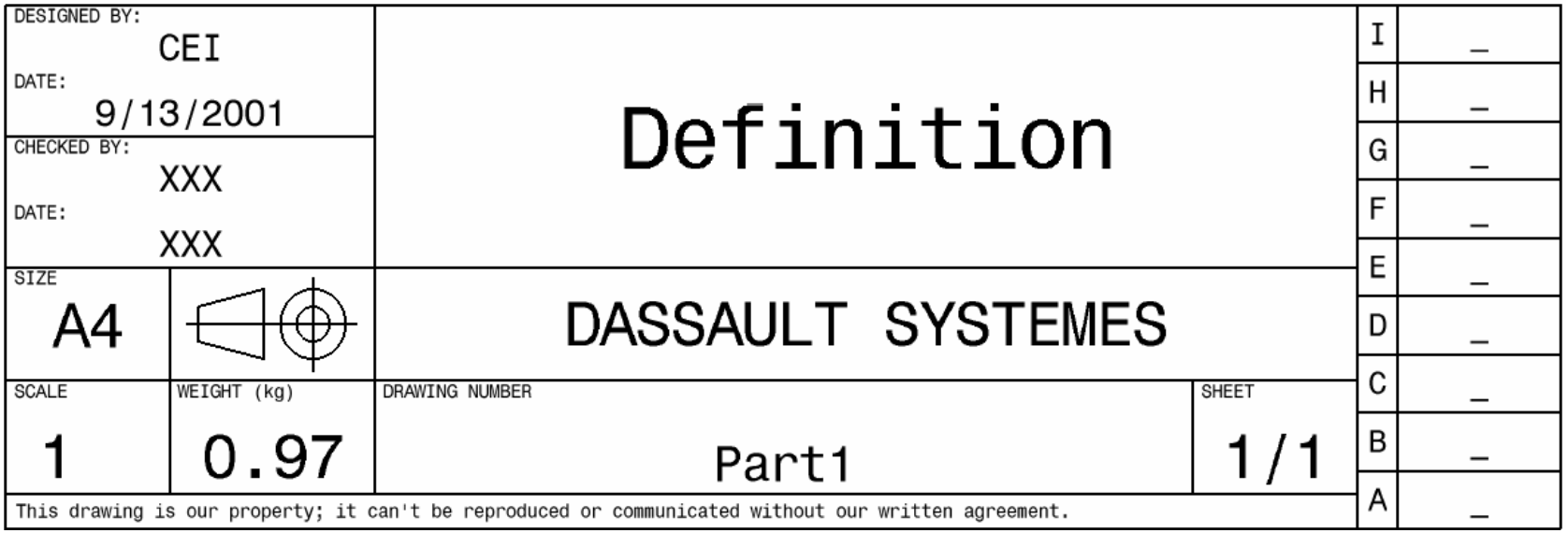

```
Sub CATFrameRuler(Ruler As Integer, Cst_1 As Single)
  '-------------------------------------------------------------------------------
  'How to create a ruler
  '-------------------------------------------------------------------------------
  'Frame_Ruler_Guide ------------------------------------------------
  'Frame_Ruler_1cm    | | | | | | | | | | | | | | | | | | | | | | |
  'Frame_Ruler_5cm    |         |         |         |         |

  On Error Resume Next
    Set Line = Fact.CreateLine(.5 * Width - Ruler/2 , .75 * Offset, .5 * Width + Ruler/2, .75 * Offset)
    Line.Name = "Frame_Ruler_Guide"

    For i = 1 To Ruler/100
      Set Line = Fact.CreateLine(.5 * Width - 50 * i, OV, .5 * Width - 50 * i, .5 * Offset )
      Line.Name = "Frame_Ruler_5cm"
      Set Line = Fact.CreateLine(.5 * Width + 50 * i, OV, .5 * Width + 50 * i, .5 * Offset )
      Line.Name = "Frame_Ruler_5cm"
      For j = 1 To 4
        Set Line = Fact.CreateLine(.5 * Width - 50 * i + 10 * j, OV, .5 * Width - 50 * i + 10 * j, .75 * Offset)
        Line.Name = "Frame_Ruler_1cm"
        Set Line = Fact.CreateLine(.5 * Width + 50 * i - 10 * j, OV, .5 * Width + 50 * i - 10 * j, .75 * Offset)
        Line.Name = "Frame_Ruler_1cm"
      Next
    Next

  If Err.Number <> 0 Then
    Err.Clear
  End If

End Sub
```

Student Notes:

Defining Frames (13/14)

- How to create the title block

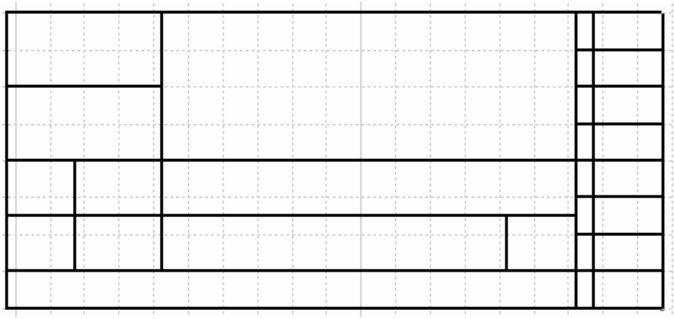
```
Sub CATTitleBlock()
  '-------------------------------------------------------------
  'How to create the TitleBlock
  '-------------------------------------------------------------
  CATTitleBlockFrame    'To draw the geometry
  CATTitleBlockStandard 'To draw the standard representation
  CATTitleBlockText     'To fill in the title block

End Sub
```



Student Notes:

Defining Frames (14/14)

- How to draw the title block frame



```
Public Col(6)       As Double           'Columns coordinates
Public Row(6)       As Double           'Rows    coordinates
```

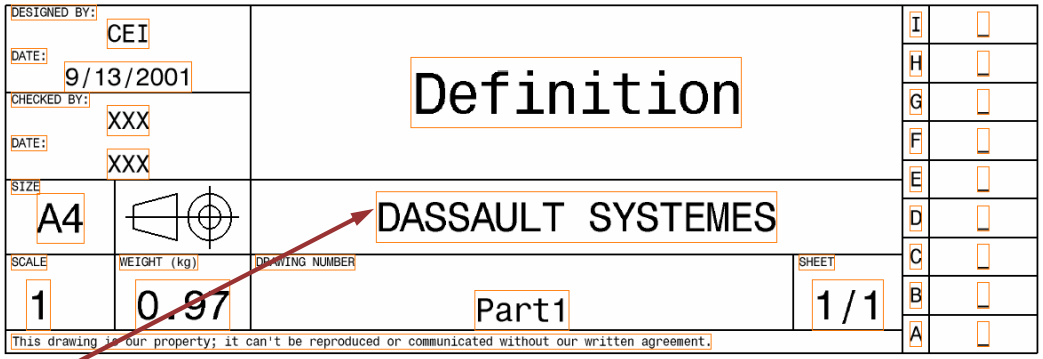
```
Sub CATTitleBlockFrame()
  '-------------------------------------------------------------------------------
  'How to draw the title block geometry
  '-------------------------------------------------------------------------------
  Col(1) = -190*mm
  Col(2) = -170*mm
  Col(3) = -145*mm
  Col(4) = - 45*mm
  Col(5) = - 25*mm
  Col(6) = - 20*mm
  Row(1) = +  4*mm
  Row(2) = + 17*mm
  Row(3) = + 30*mm
  Row(4) = + 45*mm
  Row(5) = + 60*mm

  On Error Resume Next
    Set Line    = Fact.CreateLine(OH + Col(1), OV          , OH          , OV          )
    Line.Name   = "TitleBlock_Line_Bottom"
    Set Line    = Fact.CreateLine(OH + Col(1), OV          , OH + Col(1), OV + Row(5))
    Line.Name   = "TitleBlock_Line_Left"
    Set Line    = Fact.CreateLine(OH + Col(1), OV + Row(5), OH          , OV + Row(5))
    Line.Name   = "TitleBlock_Line_Top"
    Set Line    = Fact.CreateLine(OH         , OV + Row(5), OH          , OV          )
    Line.Name   = "TitleBlock_Line_Right"
    Set Line    = Fact.CreateLine(OH + Col(1), OV + Row(1), OH + Col(5), OV + Row(1))
    Line.Name   = "TitleBlock_Line_Row_1"
    Set Line    = Fact.CreateLine(OH + Col(1), OV + Row(2), OH + Col(5), OV + Row(2))
    Line.Name   = "TitleBlock_Line_Row_2"
    Set Line    = Fact.CreateLine(OH + Col(1), OV + Row(3), OH + Col(5), OV + Row(3))
    Line.Name   = "TitleBlock_Line_Row_3"
    Set Line    = Fact.CreateLine(OH + Col(1), OV + Row(4), OH + Col(3), OV + Row(4))
    Line.Name   = "TitleBlock_Line_Row_4"
    For i = 1 To (NbOfRevision-1)
      Set Line  = Fact.CreateLine(OH + Col(5), OV+Row(5)/NbOfRevision*i, OH, OV+Row(5)/NbOfRevision*i)
      Line.Name = "TitleBlock_Line_Row_5"
    Next
```

Student Notes:

Student Notes:

Filling in the Title Block (1/4)

- How to add texts



Here the example of a subroutine just for adding a text to indicate the company name.

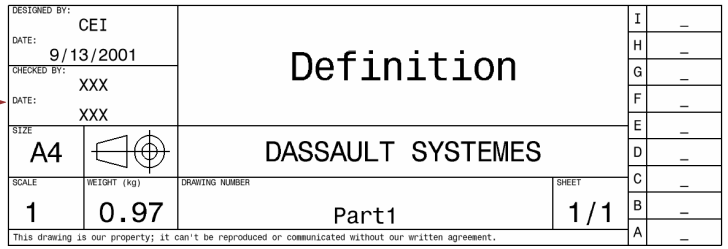
```
Sub CATTitleBlockText()
   '-------------------------------------------------------------------------------
   'How to fill in the title block
   '-------------------------------------------------------------------------------

   Text_11 = "DASSAULT SYSTEMES"

   Set Text      = DrwTexts.Add(Text_11, OH + .5*(Col(3)+Col(5)), OV + .5*(Row(2)+Row(3)))
   CATFormatTBText "TitleBlock_Text_Company"      , catMiddleCenter, 5


End Sub
```

```
Sub CATFormatTBText(textName As String, anchorPosition As String, fontSize)
   '-------------------------------------------------------------------------------
   'How to format the texts belonging to the titleblock
   '-------------------------------------------------------------------------------
   Text.Name           = textName
   Text.SetFontName      0, 0, "Courrier 10 BT"
   Text.AnchorPosition = anchorPosition
   Text.SetFontSize      0, 0, fontSize

End Sub
```

Filling in the Title Block (2/4)

- How to add texts linked to 3D information

The subroutine which adds Text Boxes in the Title Block can directly fill in several information from the 3D like the Definition, the PartNumber, the weight, etc…, or from the drawing like the size, the scale, the sheet number or the format.




```
Text_10 = "A" + CStr(DrwSheet.PaperSize - 2)

Set Text    = DrwTexts.Add(Text_10, OH + .5*(Col(1)+Col(2)), OV + Row(2) + 2         )
CATFormatTBText "TitleBlock_Text_Size_1"       , catBottomCenter, 5
```

```
Sub CATLinks()
  '-------------------------------------------------------------------------------
  'How to fill in texts with data of the part/product linked with current sheet
  '-------------------------------------------------------------------------------
  On Error Resume Next
    Dim ProductDrawn As ProductDocument
    Set ProductDrawn = DrwSheet.Views.Item("Front view").GenerativeBehavior.Document
    If Err.Number = 0 Then
      DrwTexts.GetItem("TitleBlock_Text_Number_1").Text = ProductDrawn.PartNumber
      DrwTexts.GetItem("TitleBlock_Text_Title").Text    = ProductDrawn.Definition
      Dim ProductAnalysis As Analyze
      Set ProductAnalysis = ProductDrawn.Analyze
      DrwTexts.GetItem("TitleBlock_Text_Weight_1").Text = FormatNumber(ProductAnalysis.Mass,2)
    End If
    Err.Clear

    For i = 1 To DrwSheets.Count
      DrwSheets.Item(i).Views.Item("Background View").Texts.GetItem("TitleBlock_Text_Sheet_1").Text = CStr(i) & "/" & CStr(DrwSheets.Count)
    Next

End Sub
```

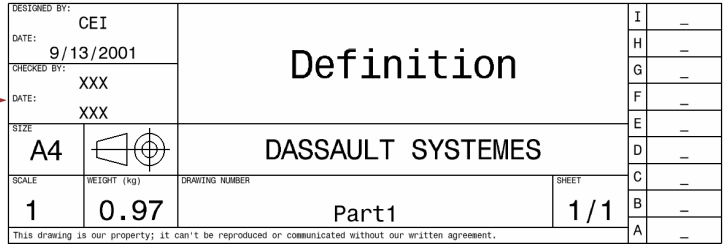
Student Notes:

Student Notes:

Filling in the Title Block (3/4)

- How to add texts linked to the operating system

The subroutine which add Text Box in the Title Block can directly fill in information from the operating system like the creation date or the user name.



If you want, you can always modify these information by editing the text box in the background view.

```
Text_15 = CATIA.SystemService.Environ("USERNAME")

Set Text      = DrwTexts.Add(Text_15, OH + Col(2) + 2.5       , OV + .5*(Row(4)+Row(5)))
CATFormatTBText "TitleBlock_Text_Designer_1"   , catBottomCenter, 3


Set Text      = DrwTexts.Add(Date    , OH + Col(2) + 2.5       , OV + Row(4)             )
CATFormatTBText "TitleBlock_Text_DDate_1"      , catBottomCenter, 3
```

Be careful, USERNAME is only a windows environment variable.

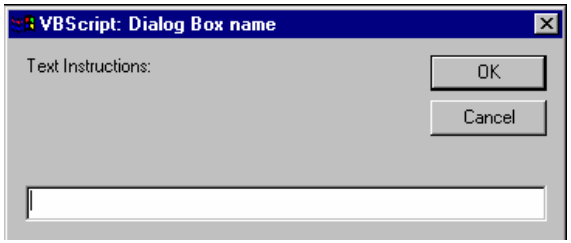
Student Notes:

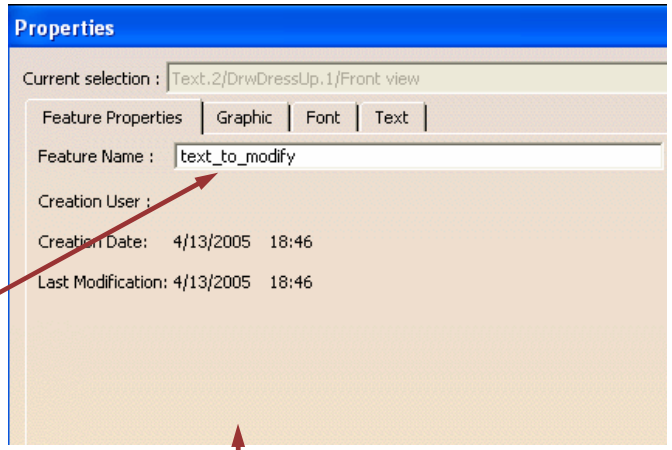
Filling in the Title Block (4/4)

- How to add texts using inputbox function

You can add title box texts using the VBscript function Input box which asks user to key in a value in a specific panel.

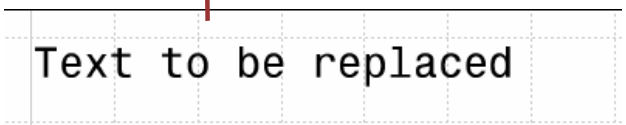
You need to identify the text id you want to modify then you will replace its value by the value specified in the input box panel.






```
Dim NewText As DrawingTexts
Dim Variable As String

Set NewText = DrwTexts.GetItem("Text_to_modify")

Variable = InputBox("Text Instruction:", "Dialog Box name", Variable)

NewText.Text = Variable
```

Properties



Student Notes:

To Sum Up

In this course you have seen :

- Hints & Tips on Dimension commands
- Hints & Tips on Text commands
- Drawing Generation from large assemblies
- Managing Standards
- Generating Specific Views
- Generating Dimensions
- Filtering Techniques in Generative Drafting
- Using VBScript macro to generate Title Blocks