Student Notes:

CATIA V5 Training
Foils

3D Functional Tolerancing and Annotation

Version 5 Release 19
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About this course

Objectives of the course

Upon completion of this course you will be able to:

- Add 3D annotations to a part

- Manage and position these annotations

- Create and manage annotation planes and views

- Manage the 3D geometry associated to the 3D annotations

Targeted audience

Mechanical Designers

Prerequisites

Students attending this course should be familiar with

- Basics of Solids and Surfaces creation.

- Basics of Knowledgeware.

2 Days

Table of Contents (1/3)

Table of Contents (2/3)

Table of Contents (3/3)

Student Notes:

Introduction to FT&A

You will become familiar with the concepts behind 3D Functional Tolerancing & Annotation workbench

- **Why do we need Geometrical Tolerances and Annotations**
- **How to Generate Annotations**
- **Basic Concepts of FTA**
- **Unique Tolerancing Channel**
- **Getting Familiar With FTA Workbench**
- **To Sum Up**

Why do We Need Geometrical Tolerances and Annotations

Designers create parts that are generally required to create an assembly of a particular product. Each part will be engineered to perform a function and, most importantly, to assemble with a mating part.

Thus every part should EXACTLY FIT in the final assembly and answer the requested functions.

Parts cannot be manufactured to exact sizes because of natural imperfections in the world, including machine tools, part programs, tooling, and also human errors. So a plan is needed to allow the production process to accept the imperfections. This gave rise to the concept of "Tolerancing" or "Allowable Deviation".

A tolerance is the amount of deviation from the exact size allowed on a part. Any part within the tolerance will still be functional.

Designers need to set part tolerances that are large enough to keep manufacturing costs down and close enough to ensure that all parts will assemble with the mating part.

It is easy for the designer to use a close tolerance on features to reduce part dimension variables; however the part cost increases dramatically as tolerances are reduced.

How to Generate Annotations (1/2)

- **Identify the Geometrical Features of the mechanism**
- **Identify the 'Use Aptitude Conditions'**
    - **The 'Use Aptitude Conditions' are the functional requirement conditions identified during the functional analysis of the mechanism.**
    - **Here is a mechanism with 'Use Aptitude Conditions' to respect:**
        - **UAC 1: Radial gap > 0.1mm**
        - **UAC 2: Axial gap > 3mm**
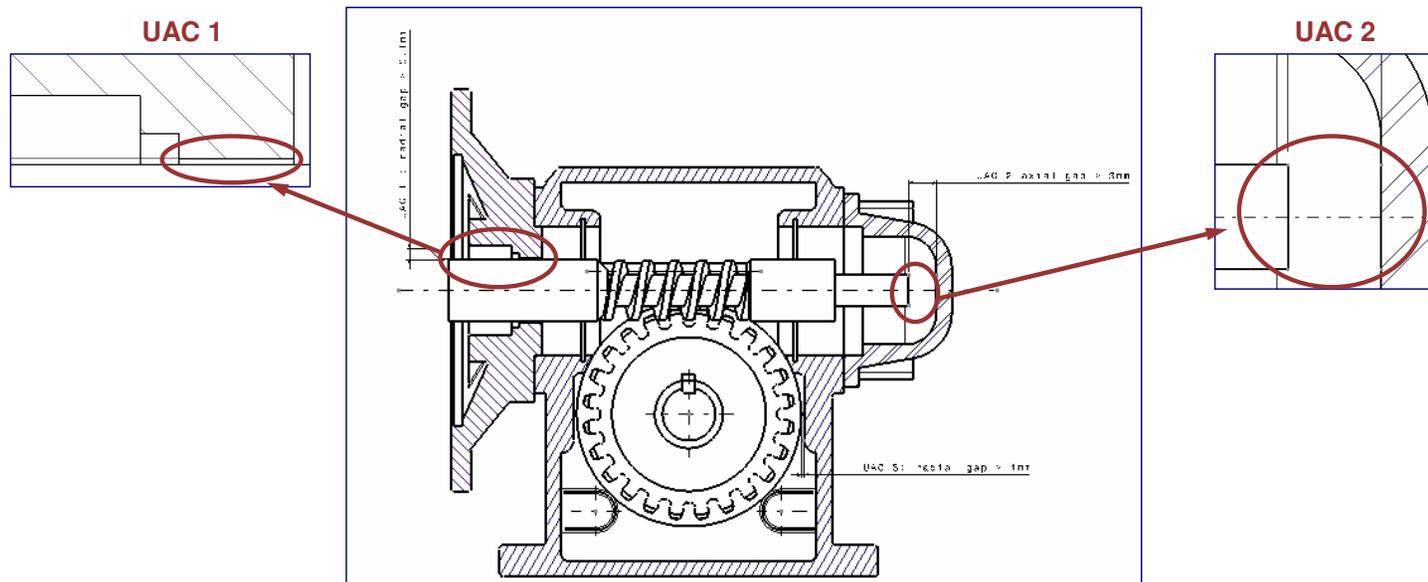
UAC 1

UAC 2

How to Generate Annotations (2/2)
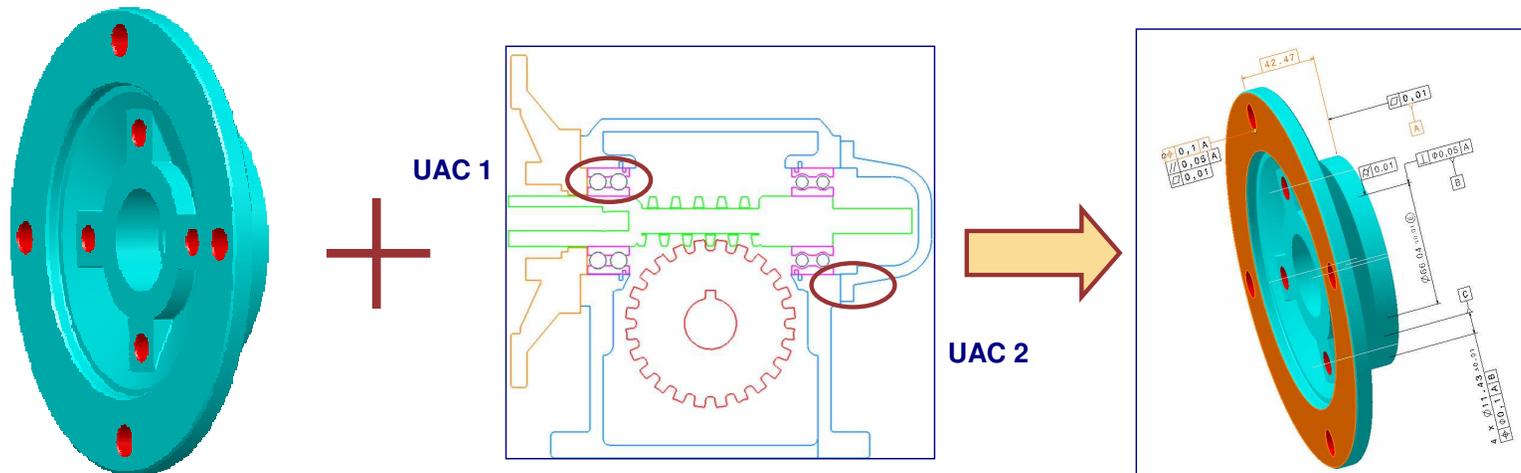
UAC 1

UAC 2

UAC 1	**Radial gap > 0.1mm**
UAC 2	**Axial gap > 3mm**

Basic Concepts of FTA

In this section, you will be introduced to the basic concepts of Functional Tolerancing and Annotations.

Virtual world

Real world
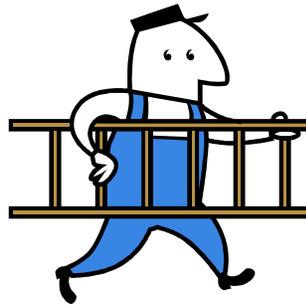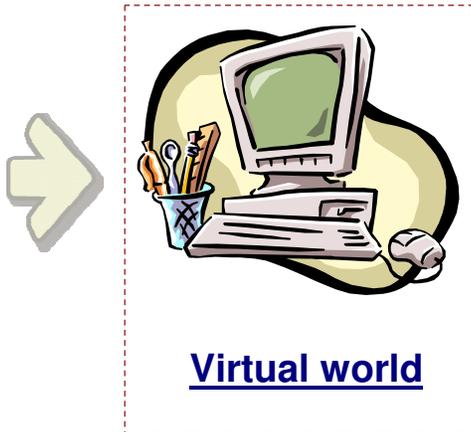
Industrial Objectives

'Invent & create innovative products which meet customers requirements, fit functions & can be produced with a high level of quality, best cost & within time.'

Geometrical variations
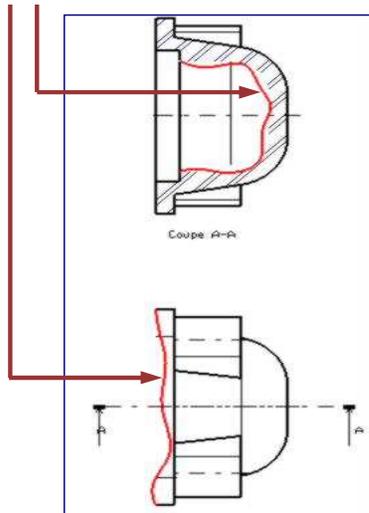
As Built   **Means of Production**   **As Designed**
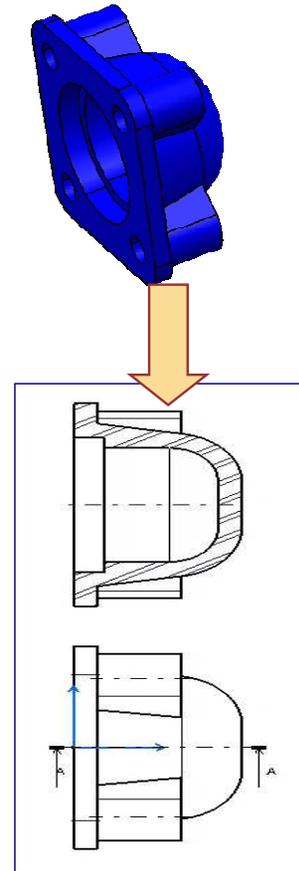
Tolerancing Purpose

"A bridge between the virtual & real world"

Virtual world

Real world

EX:

Need: Describe the non-ideal geometry
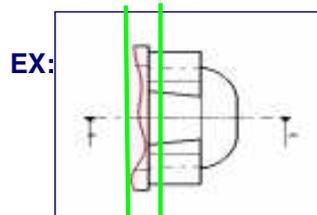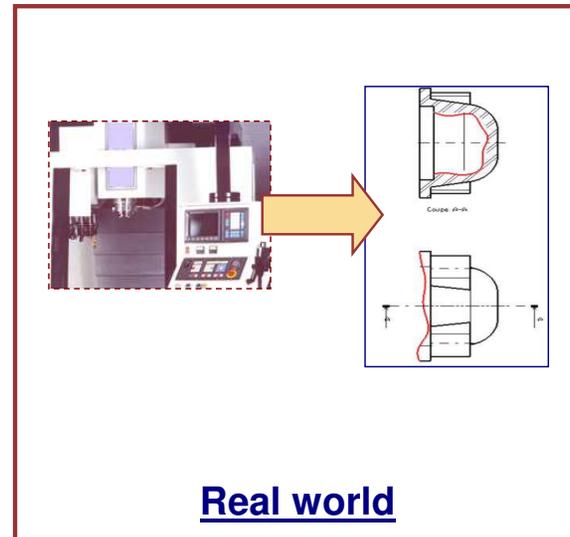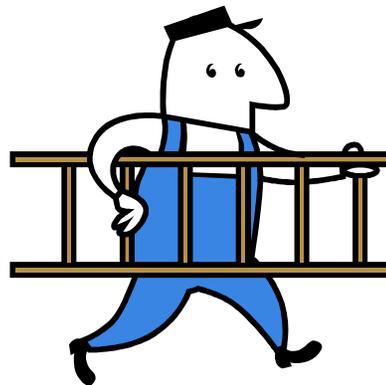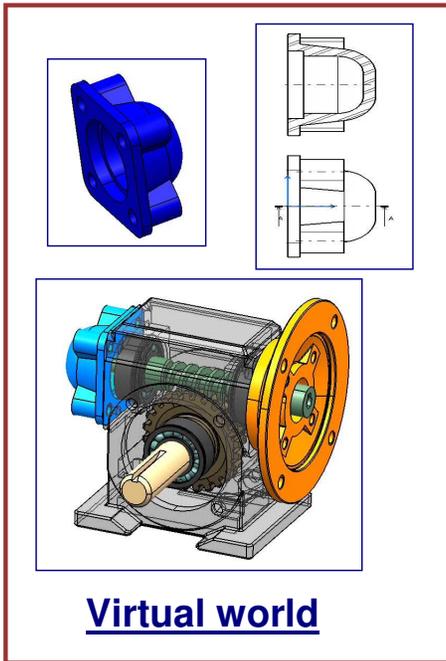
Acceptable Interval of tolerance

Tolerancing Scope

"Transversal topic: Dimensional Management in the Product Development Process"

Voice of Customer

Design

Quality
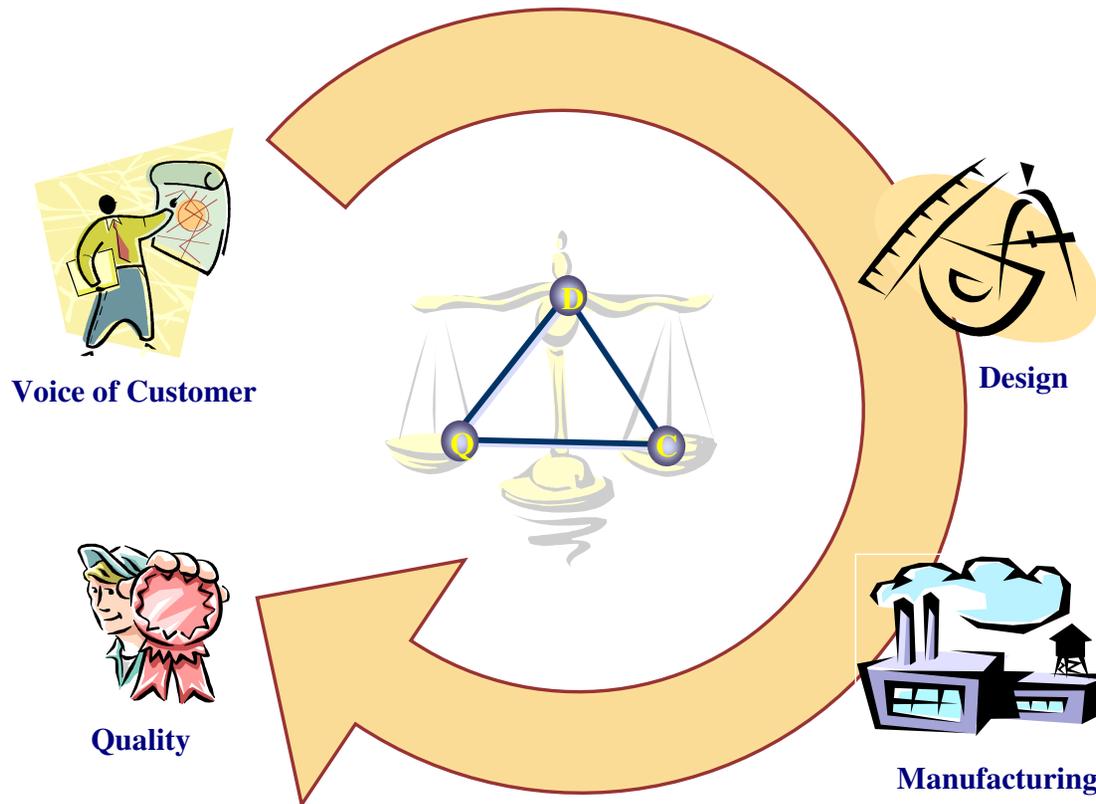
Manufacturing

Current Tolerancing Situation

Pains:

- **High cost of Manufacturing due to over quality.**

- **High rate of faulty parts produced.**

- **Products not complying with customer requirements.**

- **Major Engineering changes in the final stages of the project.**

Objectives:

- **Products which fit the customers' requirements and can be produced.**

- **Product and Process optimization.**

- **Facilitate transversal collaborative work.**

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Potential Tolerancing Improvements

"Create a unique Tolerancing channel"
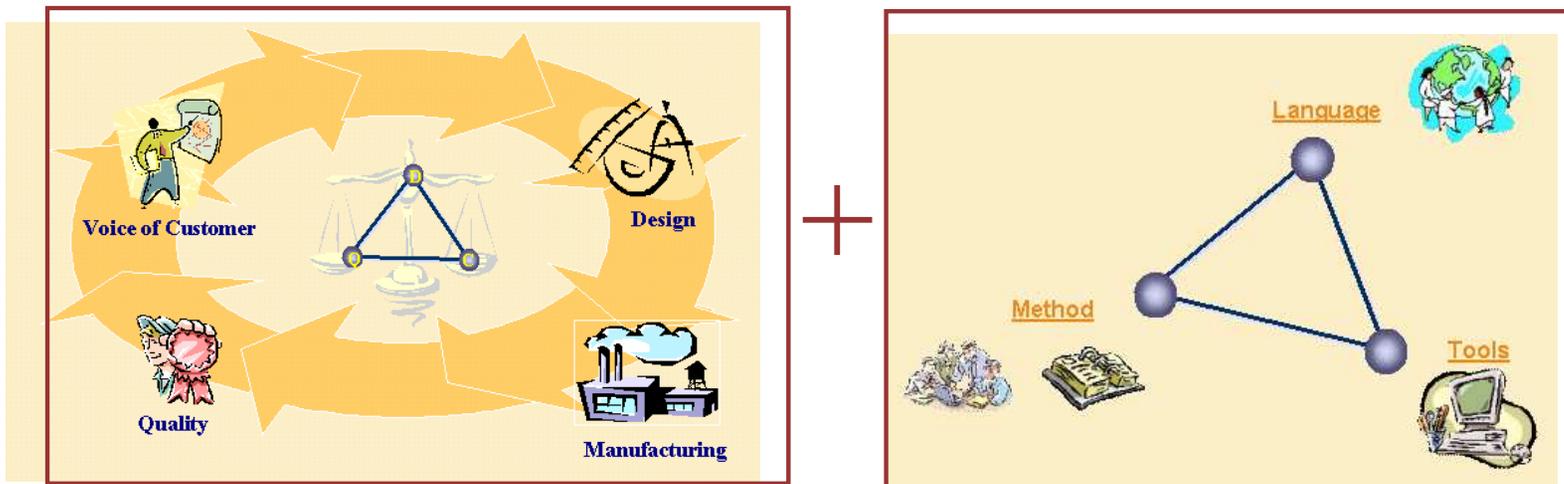
Unique Tolerancing Channel

This skillet explains the limitations in the current design process and provides with a value proposal and gives a strategy to implement it.
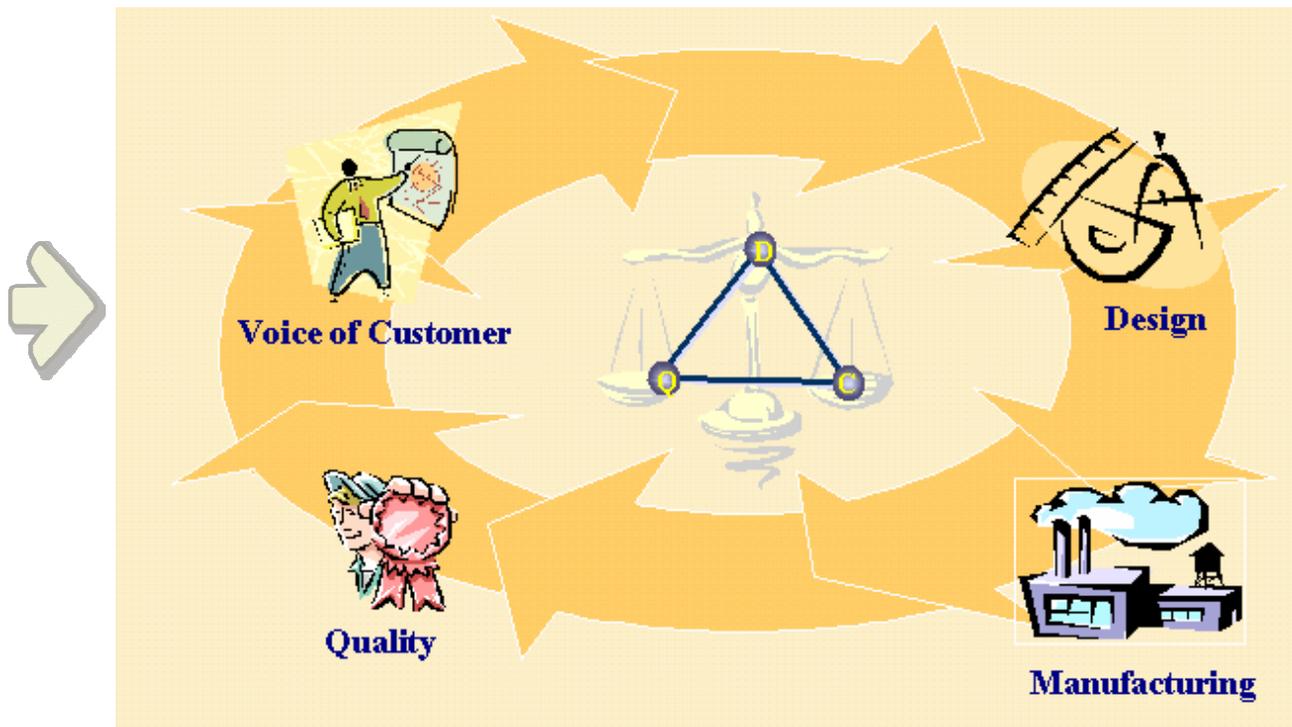
Transversal Topic:

Lets have a look for Design purpose
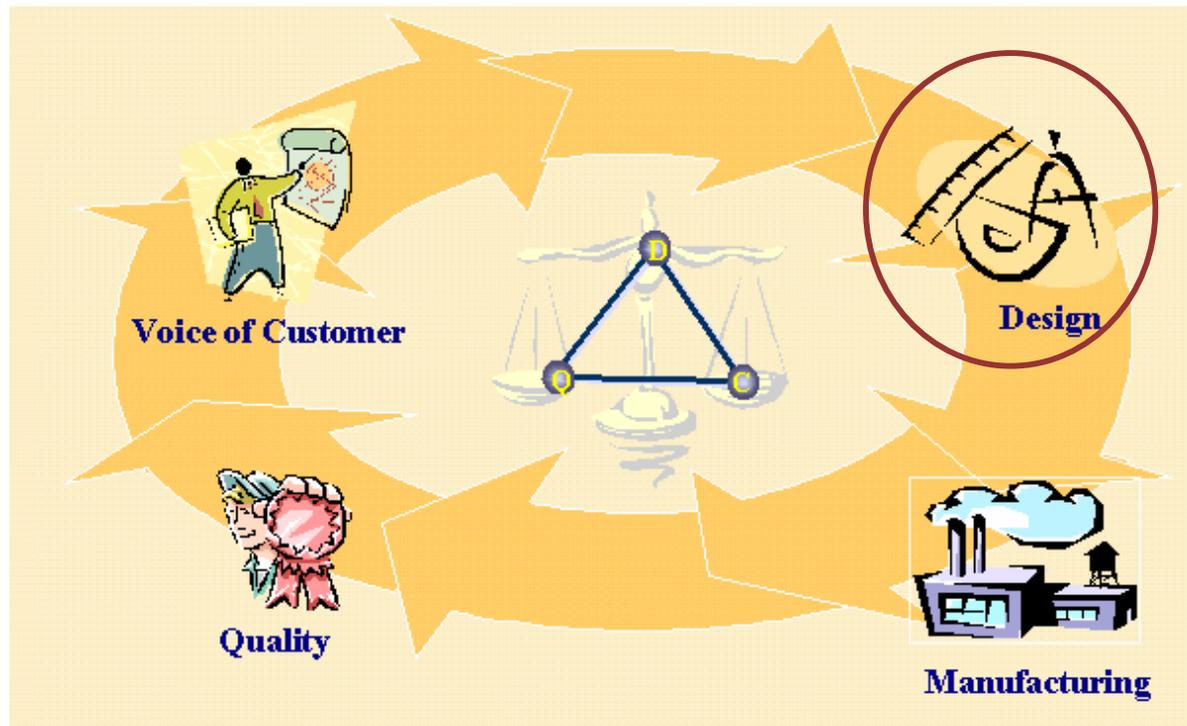
Student Notes:

Usual Design Process:

1 **Geometrical Definition**
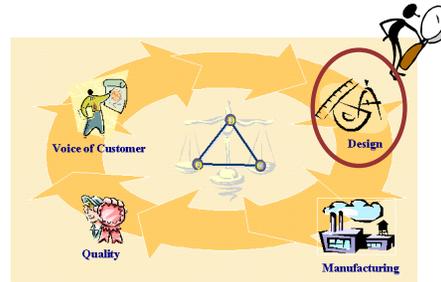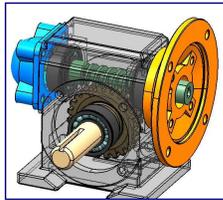
2 **Use Aptitude Conditions**
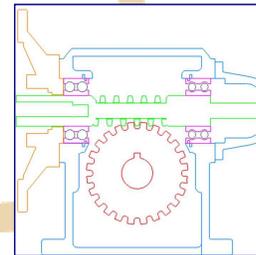
3 **Initial Drawings & write 2D annotations**
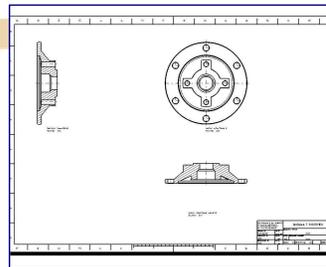
4 **Select & Validate Tolerancing schema**

5 **Validate Drawings**

Voice of Customer

Design

Quality

Manufacturing
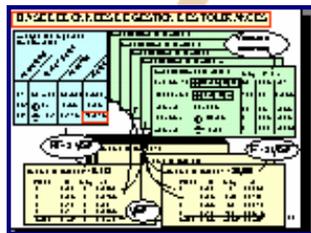
Student Notes:
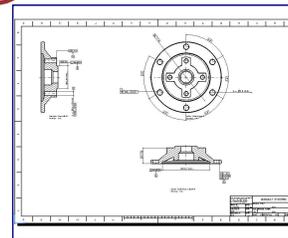
Highlights of Current Design Process:

Pains:

- **Drawing as master:**

  - ◆ **Annotations not linked with 3D geometry.**

  - ◆ **Consistency between 2D Drawings & 3D Geometry costs a lot.**

  - ◆ **Risk of mistakes, oblivion & misunderstanding.**

- **Tolerances and dimensions redefined & converted several times in the product Development process.**

Value Proposal:

- **Create & Manage 3D Annotations attached to the 3D geometry.**

- **Drawing as result, 3D as master.**

- **3D Annotations used and shared along the product life cycle.**

- **Added high value:**

  - ◆ **Capturing, sharing and re-applying corporate knowledge.**

New Design Process - FTA Way (1/4)

1 **Geometrical Definition**
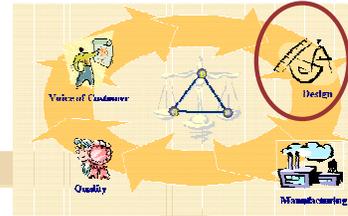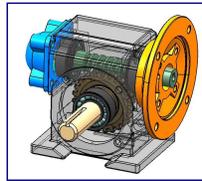
2 **Use Aptitude Conditions**
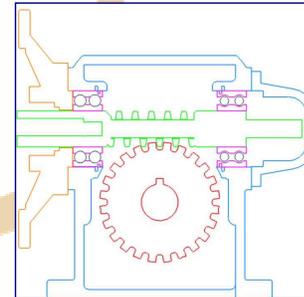
3 **Write 3D Annotations**
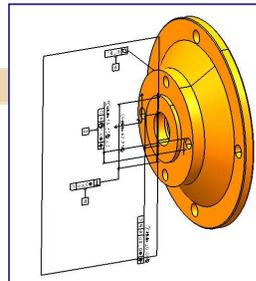
4 **Select & Validate Tolerancing schema**
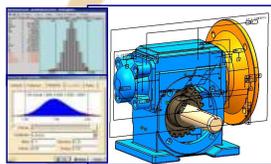
5 **Generate drawings & Look through 3D annotation during digital mock up review process**
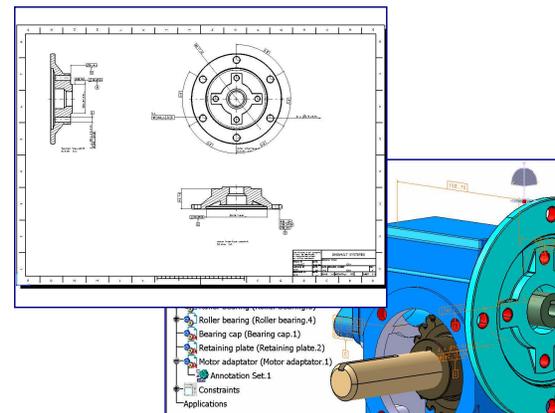
New Design Process - FTA Way (2/4)

- **Design Strategy**
    - **To allow 3D only process:**
        - **Specification: Dimensioning, Tolerancing, Notes,…**
        - **3D annotations Communication and Review**

    - **To improve Dimensional Quality:**
        - **Tolerance Analysis**
        - **Tolerance Synthesis**

    - **To be used by Downstream Applications:**
        - **Manufacturing (tolerance charting)**
        - **Assembly process planning**
        - **Inspection, Metrology**
        - **Company internal applications**

    - **To define and support new standards for 3D annotations:**
        - **ASME Y14.41-2003**
        - **ISO/TC 213/WG 14**

New Design Process - FTA Way (3/4)
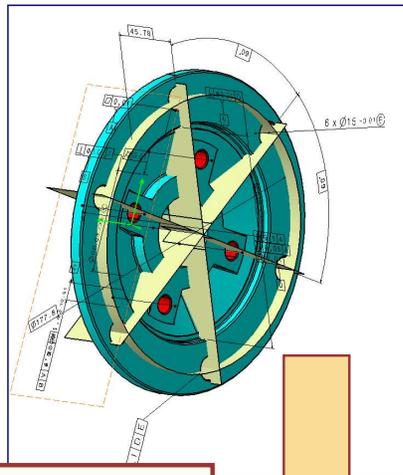
2D creation
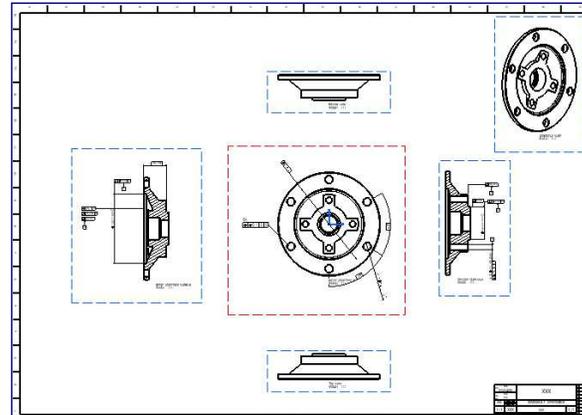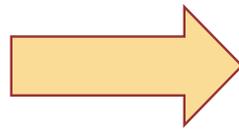
3D Geometrical modification
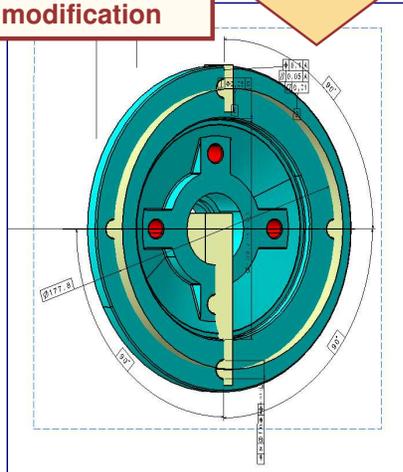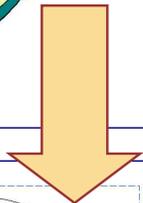
2D Update
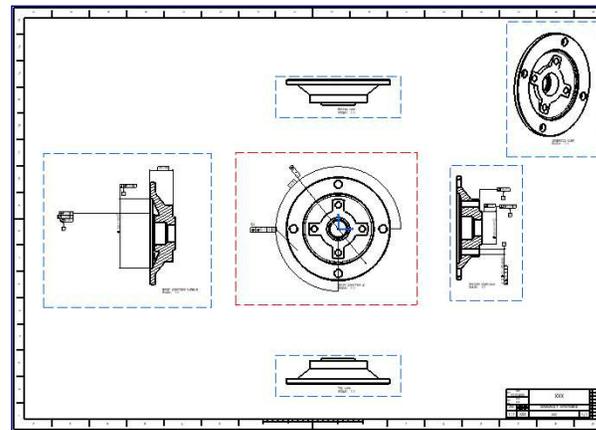
On Update: 2D geometry and annotations are modified

New Design Process - FTA Way (4/4)

🔹 **Main Characteristics**

   🔷 **Fundamentals (Editor)**

- Creation of annotations without semantic and syntactic control (Industry Standards)
- Linked to the 3D geometry
- All Interactive Drafting dress-up capabilities

   🔷 **Advanced (Advisor)**

- Proposal of applicable tolerance types regarding the selected surfaces
- Proposal of tolerance options when applicable
- Tolerancing rules verification
- Automatic support of annotation syntax (GD&T)
- GUARANTEES of semantic & syntactic (Industry Standards) validity of the tolerancing, through the part / assembly life cycle

Getting Familiar With FTA Workbench

You will become familiar with the User Interface of Functional Tolerancing & Annotations workbench.

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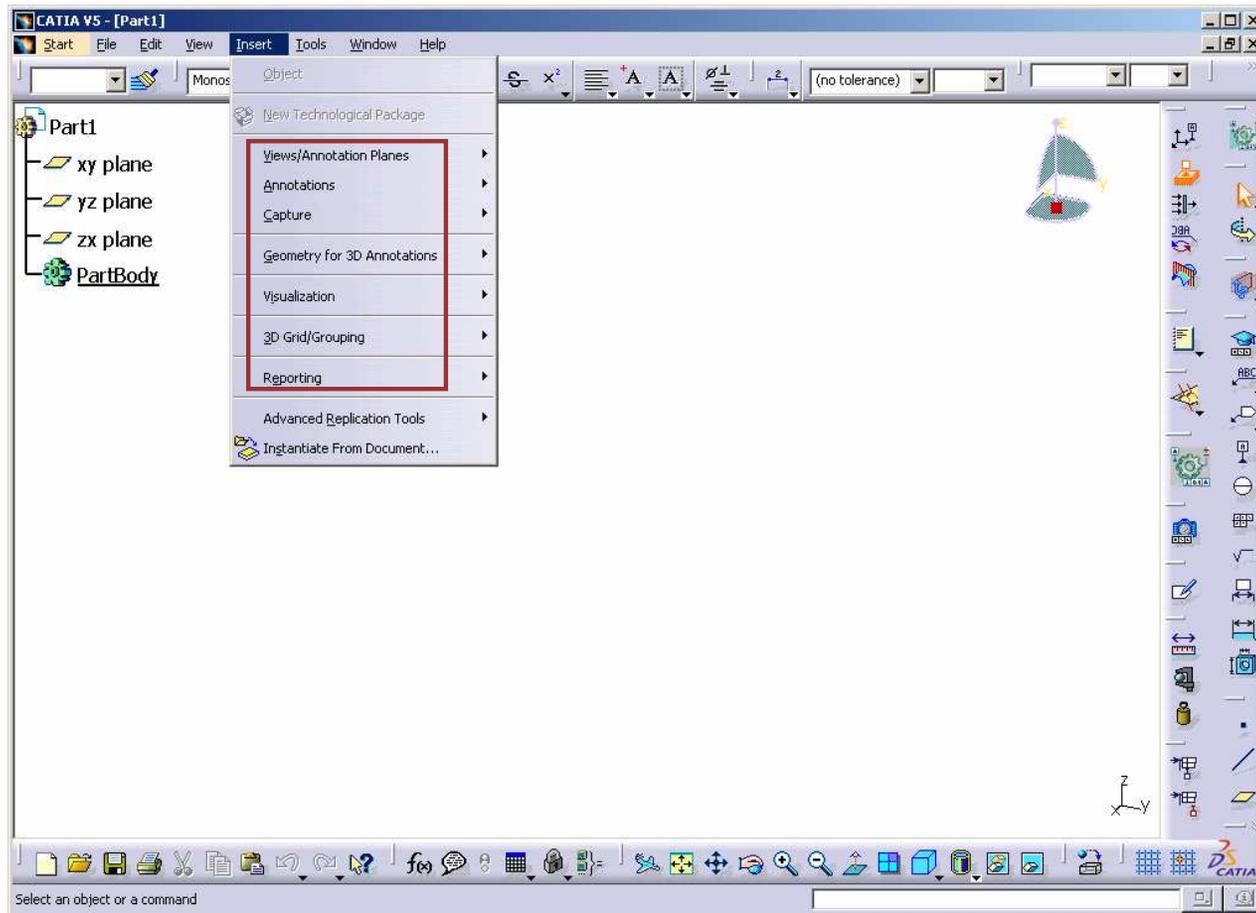
Scope of Functional Tolerancing and Annotations

3D Functional Tolerancing & Annotation workbenches allows you to define and manage 3D tolerance specifications and annotations directly on 3D parts or products.

As discussed earlier, FTA reduces the reliance on 2D drawings and considers 3D as the master representation. Thus, driving the engineering process from design phase to manufacturing phase.

- **In CATIA, Workbenches are dedicated for the following**
  - ◆ **Functional Tolerancing & Annotation**
    - ▪ **This Workbench is used for creating Tolerances on Parts in isolation**
  - ◆ **Product Functional Tolerancing & Annotation**
    - ▪ **This Workbench is used for creating Tolerances on Products**
  - ◆ **Process Tolerancing & Annotation**
    - ▪ **This Workbench is used for creating Tolerances on Processes**

Tolerances on Parts
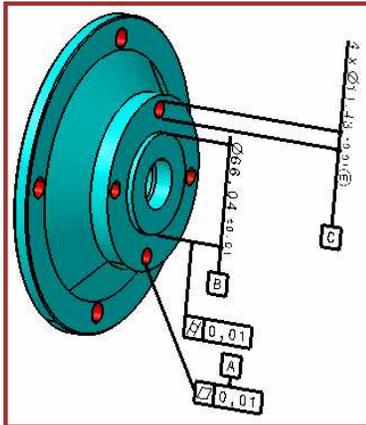
Tolerances on Products
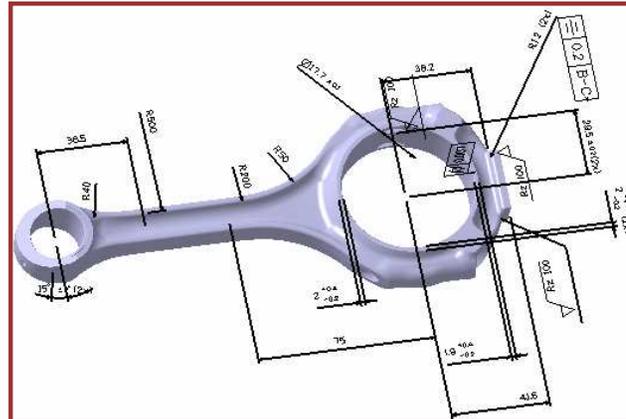
Accessing the Workbench

It is possible to work with FTA at Part, Product and Process level in any CATPart, CATProduct and CATProcess document respectively.

Part Level FTA
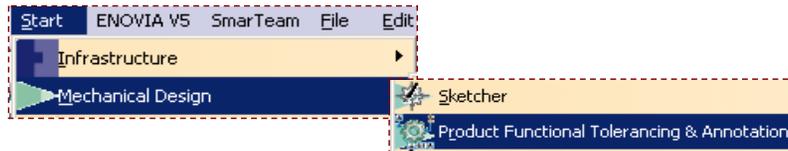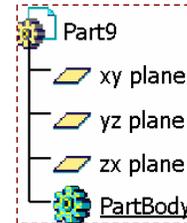
Product Level FTA
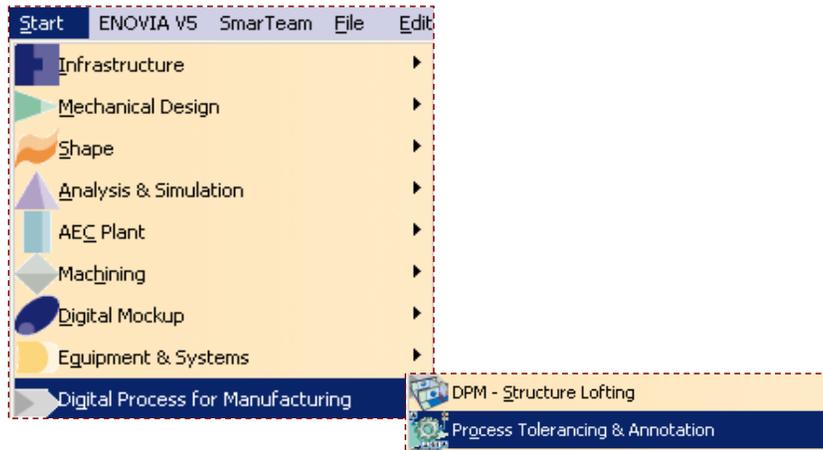
Process Level FTA
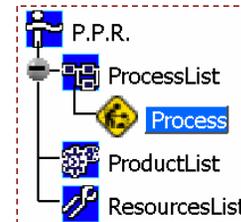
The User Interface – Part Level

You can access the following annotations tools through Insert Menu for Part level FTA
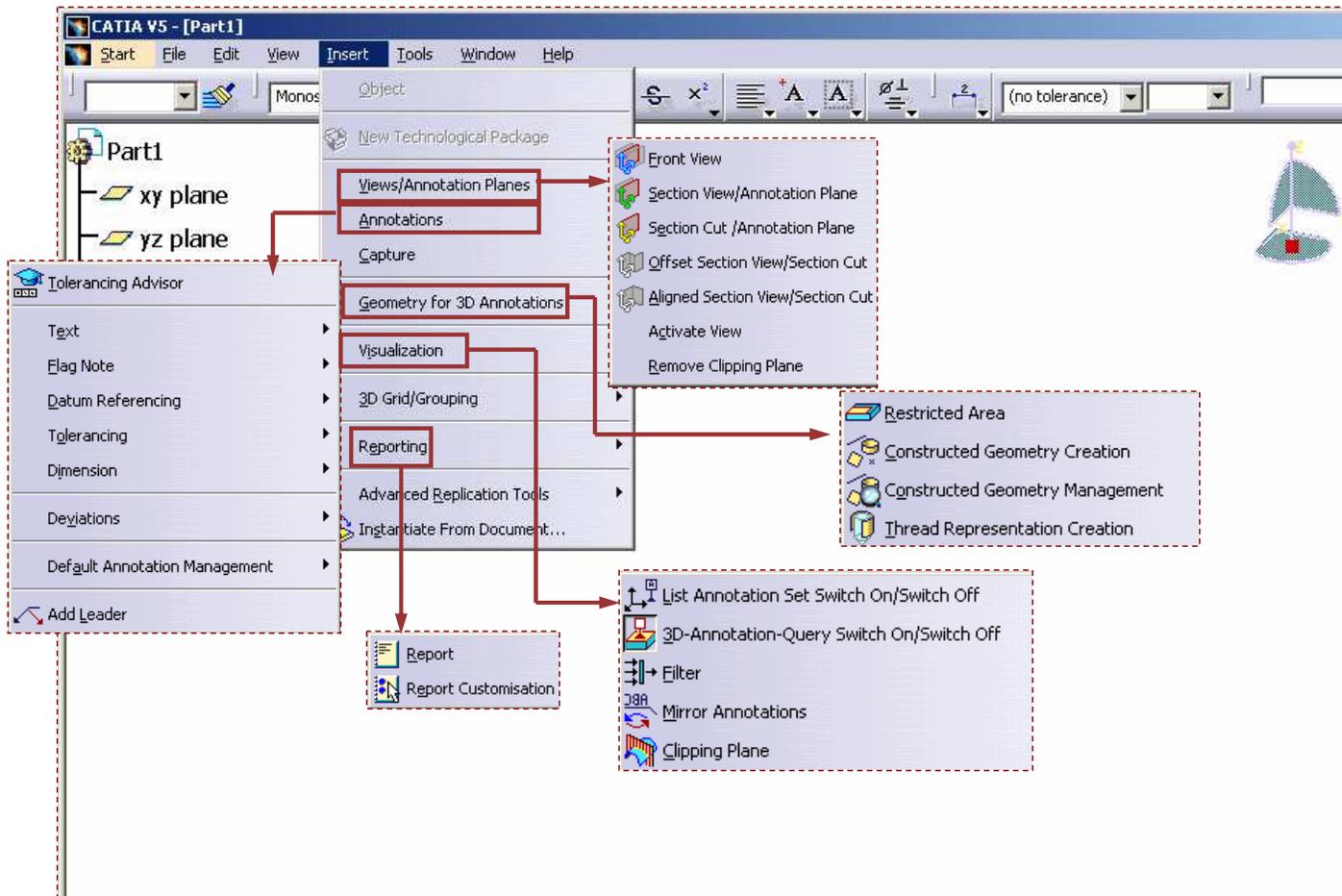
The FTA Toolbars

Specific Product Functional Tolerancing & Annotation and Functional Tolerancing & Annotation Toolbars

Annotation   **Note Object Attribute**   **Reporting**

Tolerancing Creation   **Tolerancing Analysis**

Views   **Grouping**   **Capture**   **Geometry for 3D annotations**

Visualization   **Reference Elements***

* Only available on FTA environment

Tolerancing Graphical Management   **Tolerancing Geometrical Management**

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To Sum Up

You have seen:

- **How FTA helps to bridge the gap between Real and Virtual world.**
- **The New Design process using FTA.**
- **The User interface of FTA workbench**

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The Tolerancing Advisor

In this lesson you will learn how the Tolerancing Advisor facilitates creation of Tolerances, Datum features, Semantic and Non- Semantic Annotations.

- **What is Tolerancing Advisor**
- **Different ISO Standards Rules**
- **Creating Datums**
- **Creating Semantic Annotations**
- **N Elements Tolerancing**
- **Creating Non-Semantic Annotations**
- **Creating Framed Dimensions**
- **To Sum Up**

What is Tolerancing Advisor?

The Tolerancing Advisor is a tool to specify 3D annotations on a product. It is a wizard that assists you to create permissible annotations according to the selected geometrical element or existing annotation.

- **It 'Proposes' all possible Annotations and Tolerances that can be applied on the selected surface.**
- **It helps to verify Tolerancing Rules.**
- **Guarantees you the correctness of all the annotations with the standards used using syntactic and semantic verifications.**

When this surface is selected...
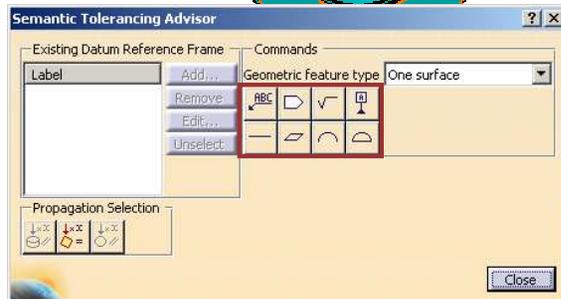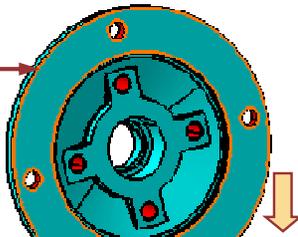
When this Edge is selected...
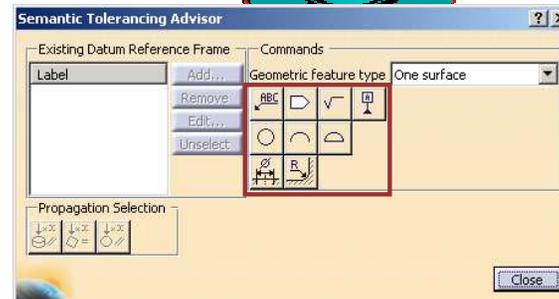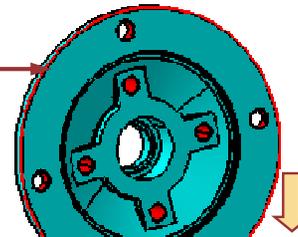
The example explains how the Dialog Box changes when the selection changes, displaying only permissible tolerances.

Different ISO Standards Rules (1/5)

International Organization For Standardization Rules Number 406.1987: These Standards are used to set Tolerances for linear and angular dimensions in technical drawings.

Elements	Type of Tolerances	Tolerancing characteristics	Symbols	
Isolated elements (1 to N Elements)	Dimensional	Linear		
		Angular		
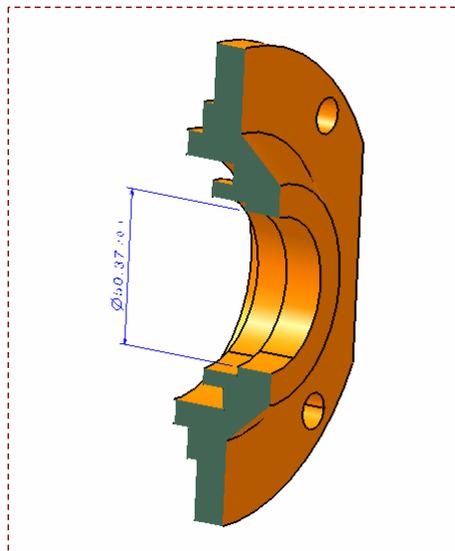
Different ISO Standards Rules (2/5)

International Organization For Standardization Rules Number 1101.1983:

Elements	Type of Tolerances	Tolerancing characteristics	Symbols
Isolated elements	Profile	Straightness	—
		Flatness	▱
		Circularity	○
		Cylindricity	⌀
Isolated or associated elements		Profile-of-Line	⌒
		Profile-of-Surface	⌓
Associated elements	Orientation	Parallelism	//
		Perpendicularity	⊥
		Angularity	∠
	Position	Position-with-DRF	⊕
		Concentricity	◎
		Symmetry	⩶
	Runout	Circular Runout	↗
		Total Runout	⌁

Different ISO Standards Rules (3/5)

International Organization For Standardization Rules Number 1101.1983:
Powertrain case Study

Runout specification

Profile specification

Orientation & position specifications
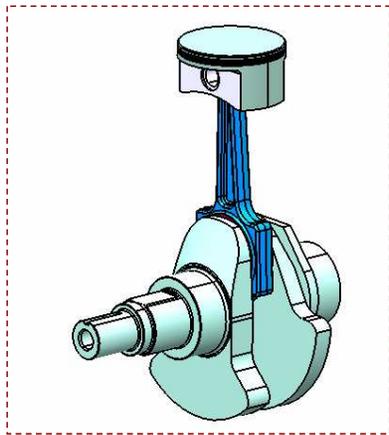
Context
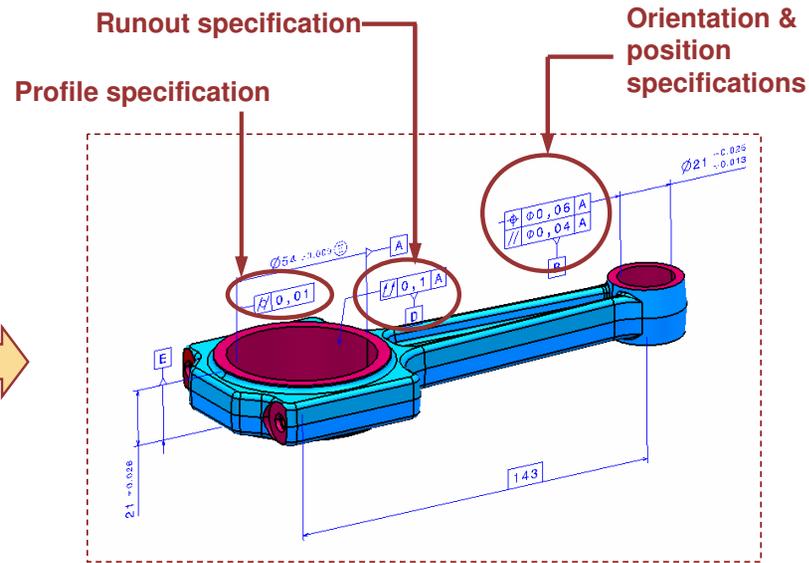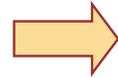
Functional Tolerancing for connecting rod

Different ISO Standards Rules (4/5)

International Organization For Standardization Rules Number 5459.1981:

Datum Frame:

Element not used as Reference Frame Datum, but needed to create Reference Frame Datums (see next page)

Reference Frame Datum
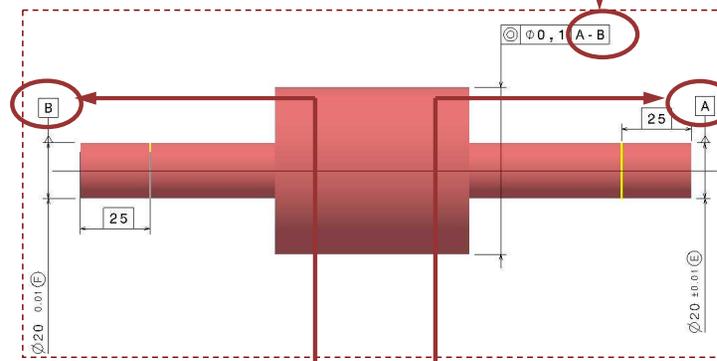
Datum Frame

Reference Frame Datum:

Reference Frame datum on an ideal surface	Real Surface	Real Surface on Marble Control
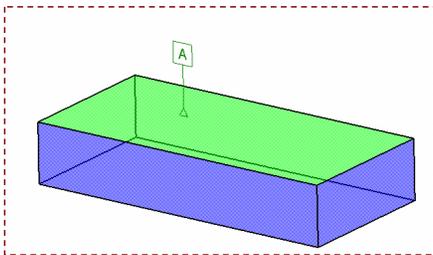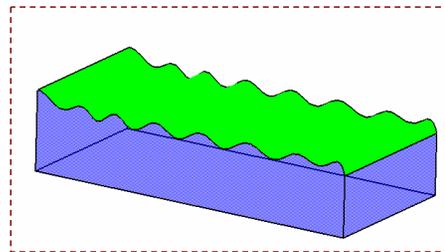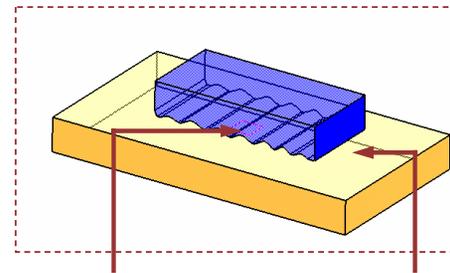
Reference Frame Datum: Plane set by marble

Reference Frame Datum simulated: Surface of marble

Different ISO Standards Rules (5/5)

International Organization For Standardization Rules Number 5459.1981:

There are 3 kinds of Reference Frame Datum:

Simple Reference Frame Datum (see previous slide)
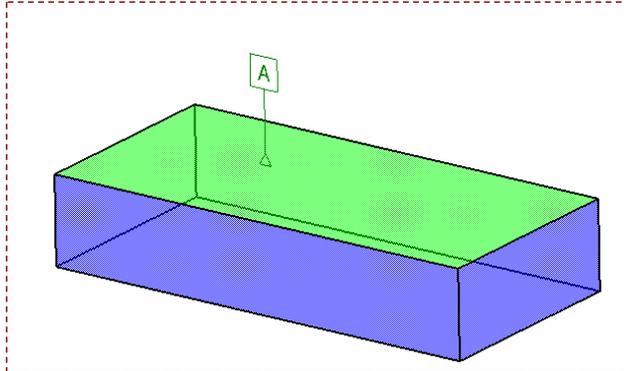
Specified Reference Frame Datum
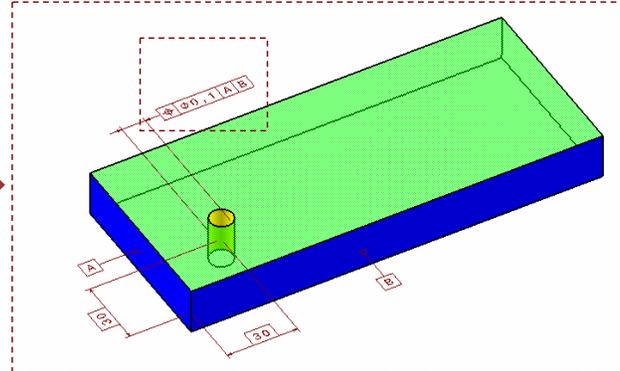
Common Reference Frame Datum
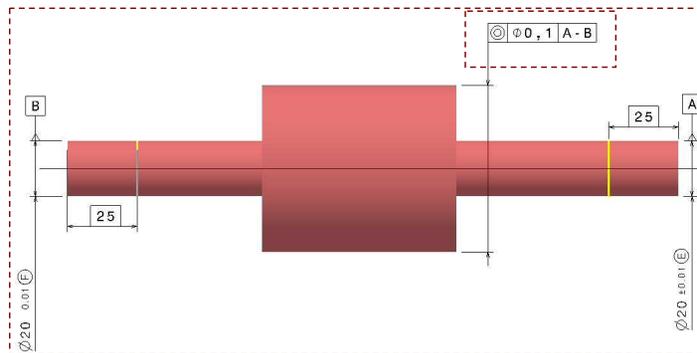
Creating Datums

In this skillet you will learn the basic principles behind creating Datum Elements and Datum System.
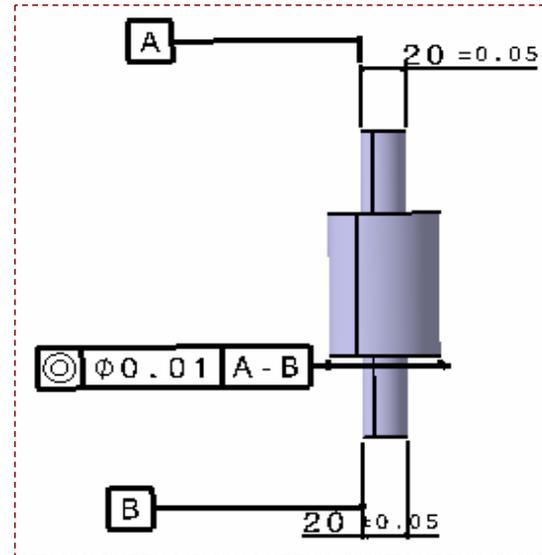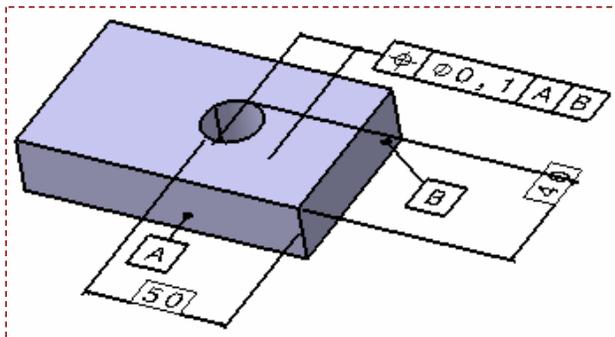
What is Datum (1/2)

- **Datums are used to identify the datum element in the tolerance frame, specified in case of geometrical tolerancing and on form tolerancing. The Datum element may be a face of a part. The face marked with Datum, indicates that the face will be a reference face and other faces will be machined with respect to this face. A capital letter is used to identify the datum element.**
- **You can specify Datums by following three methods:**
  - **Simple Datum elements**
  - **Specified Datum systems**
  - **Common Datum elements**

- **Datum System Composition**
  - **When only one identifier is specified in the tolerance frame, the datum is a single datum.**
  - **When the identifiers are specified separately in each frame of the tolerance frame, the datum elements represent a datum system. Reference A is the primary datum and reference B is the secondary datum. Using this datum system, fitting will be performed first on datum A, then on datum B, with respect to A.**
  - **When two identifiers separated by a dash are specified in the tolerance frame, the datum is a common datum. The two datum elements are to be considered simultaneously.**

What is Datum (2/2)
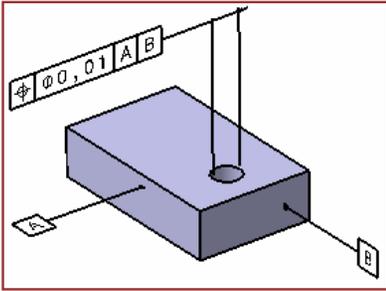
Simple Datum elements

The faces are Datum Faces and indicated with Simple Datum Elements. The Hole will be machined with Datum A and Datum B faces.
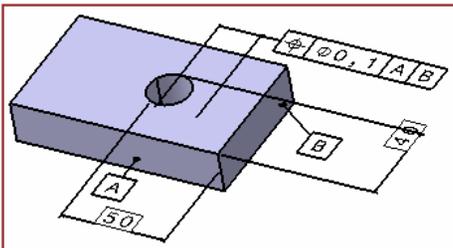
Specified Datum systems

Hole position with respect to the system composed of the face A and face B correctly positioned respectively to face A.
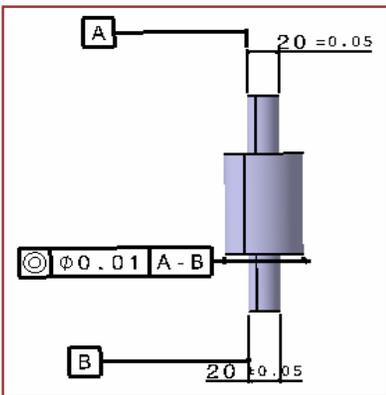
Common Datum elements

Middle portion of the shaft is concentric with Datum Faces A and B simultaneously.

How to Create Simple Reference Datum Element

You will use the Tolerancing Advisor to create simple Reference Datum

1 **Select the Tolerancing Advisor icon**

2 **Select the element that should be Datum. In this example, a face is selected**
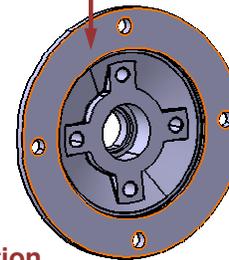
3 **The Semantic Tolerancing Advisor dialog box is displayed.Select the Datum icon.**
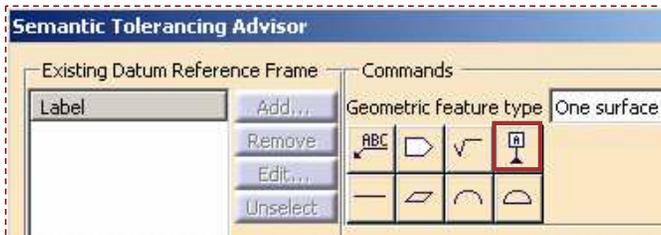
Click to create datum reference frame

4 **Enter the label & validate the creation**
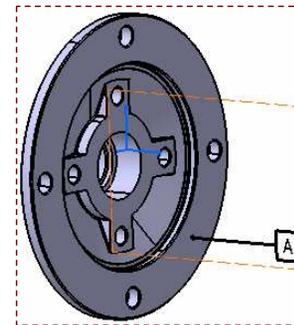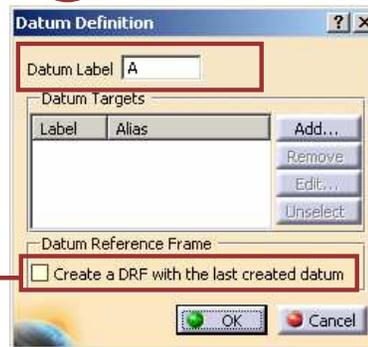
Observe the representation in the specification tree
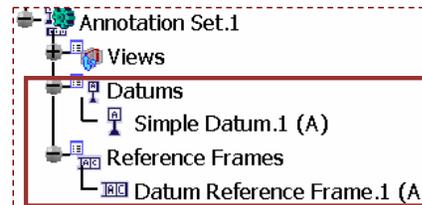
How to Create Specified Reference Datum Element (1/2)

In order to create Common Reference Frame Datum, Simple Reference Datums have to be created before.

1 **Select the Tolerancing Advisor icon**

2 **Click on 'Add' to create New Reference Frame Datum**

The Dialog Box shows previously created Reference Frames
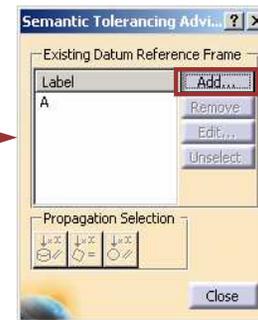
3 **Select the Primary Reference Datum**
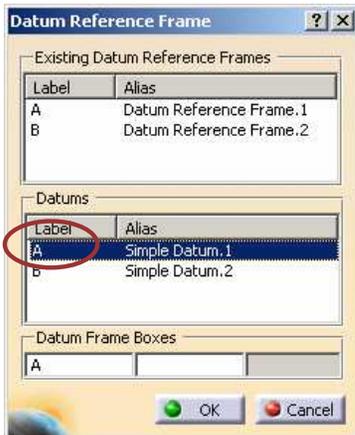
4 **Select the Secondary Reference Datum**
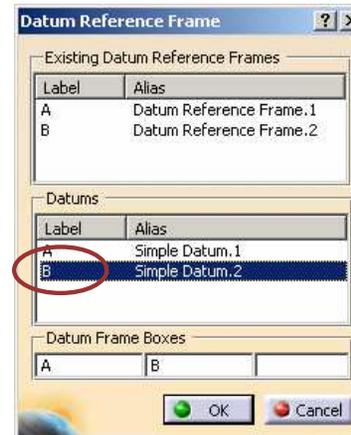
Repeat the step 4 for the tertiary reference, if needed.

How to Create Specified Reference Datum Element (2/2)

⑤   **The Datum reference is added to the list**
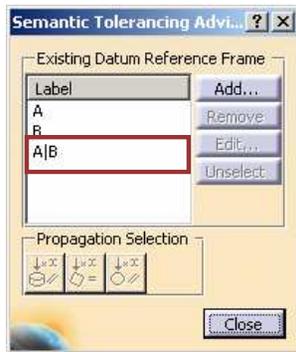
You can use these Datum Reference Frames to define Geometric tolerances
Interpretation of the following example: The Position of the hole with respect to the datum surfaces
with respect to Datum reference frames A and B should be within a tolerance zone of 0.01 mm.
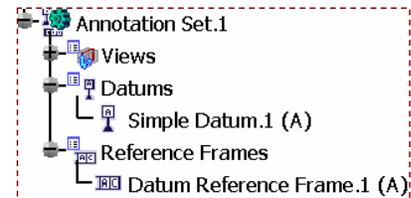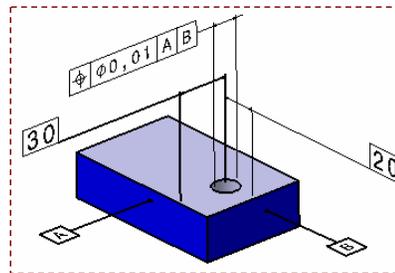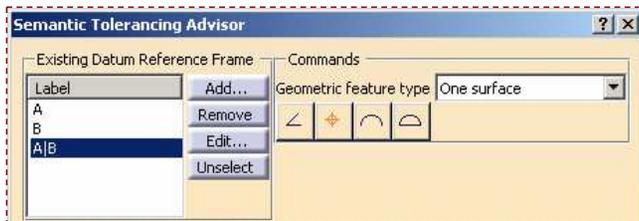
How to Create Common Reference Datum Element (1/2)

In order to create Common Reference Frame Datum, Simple Reference Datums have to be created before.

1 **Select the Tolerancing Advisor icon**

2 **Click on 'Add' to create a New Reference Frame Datum**
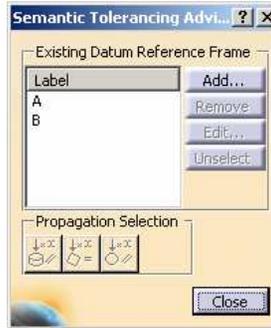
3 **Select the Primary and Common reference Datum in the dialog box**
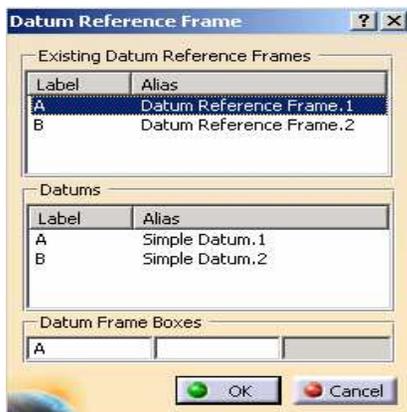
Click in the first Datum Frame Box

+
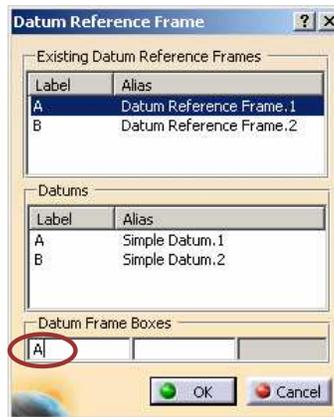
Select the Common Reference
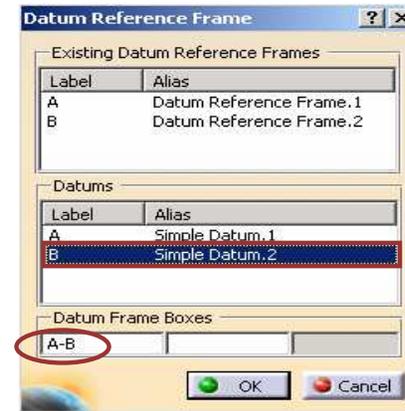
How to Create Common Reference Datum Element (2/2)

4   **Observe that the Datum Frame is added in the list**
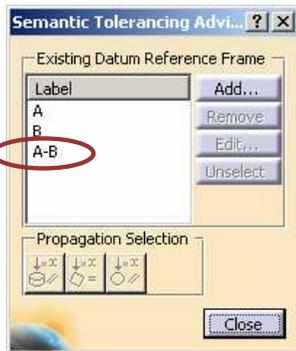
Interpretation: In the example below concentricity is maintained with respect to Datum A and B simultaneously.
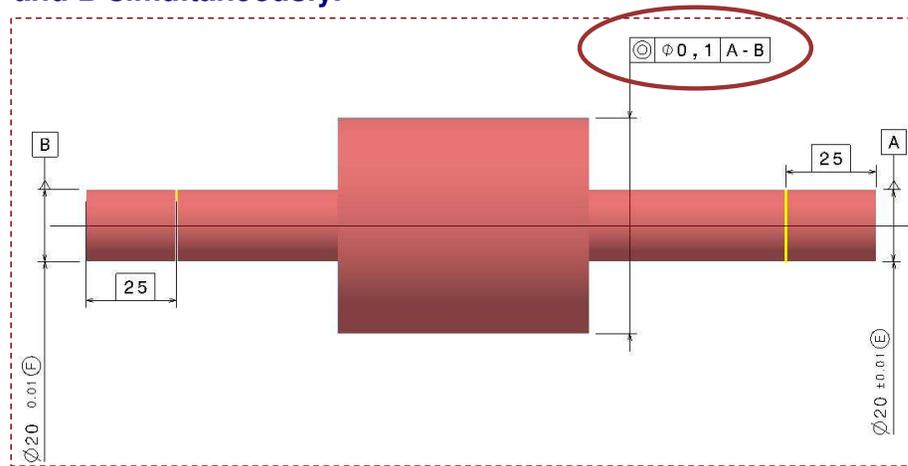
Creating Semantic Annotations

In this skillet you will learn what are Semantic Annotations and how to create them.
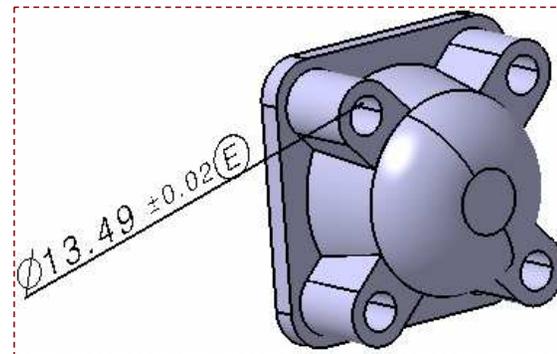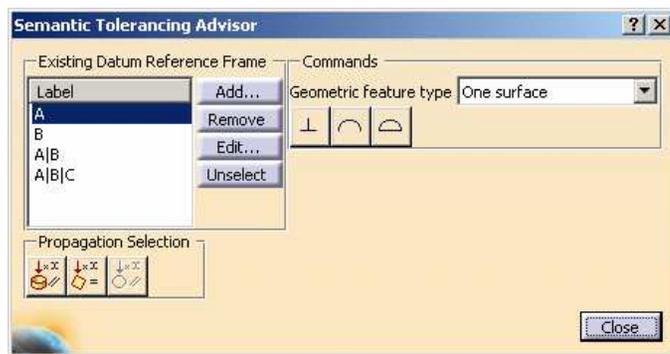
What are Semantic Annotations

Semantic Tolerances are CATIA Objects which automatically take into account the element to be toleranced.

They fully comply with the ISO or ASME/ANSI norms and the aim of FTA is to fully cover these definitions given in the standards.

Semantic Tolerances are created using Tolerancing Advisor and help to validate the consistency with the geometry.

The Semantic annotations can be re-used and can be interpreted by applications like tolerance analysis, inspection, manufacturing, assembly process etc.

Following are the Semantic Annotations:

Tolerancing Advisor

Text with Leader

Flag Note with Leader

Datum Element

Datum Target

Generative Dimension

Dimensions

Framed (Basic) Dimensions

Roughness

Geometric Tolerance
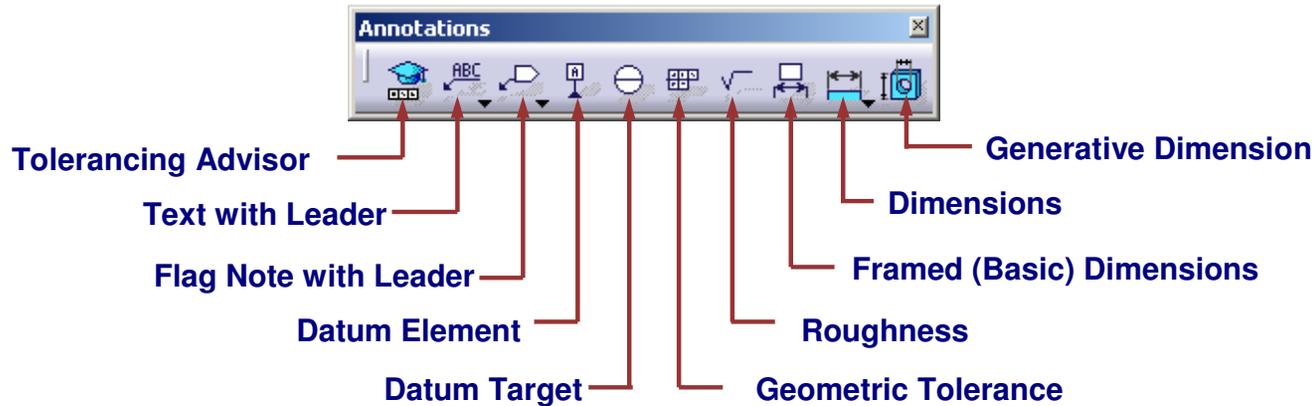
Semantic Toleranced dimensions cannot be faked

How to Create Semantic Dimensions

Here you will learn how to create Semantic Dimensions.

1   **Select the Hole feature to dimension and select Tolerancing Advisor icon.**
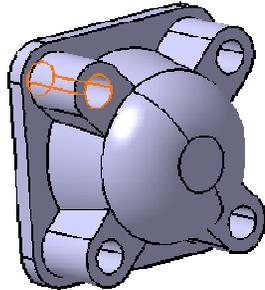
2   **Select Diameter Dimension. The command frame only display dimensions that correspond to the selected element.**

Semantic Tolerancing Advisor

Existing Datum Reference Frame | Commands

Label   Add...   Geometric feature type | One surface

Remove

Edit...
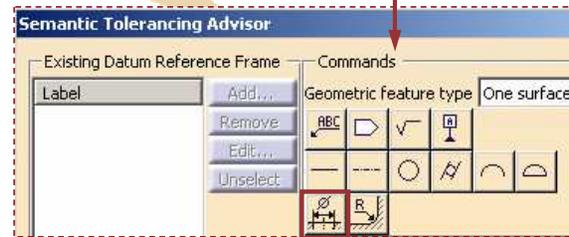
Unselect

As the selected element is a hole, the only available dimensions are Diameter and Radius.

3   **Enter the tolerance value and select the Enveloping condition.**

Limit of Size Definition

Tolerance
- General Tolerance
  - Standard :
  - Tolerance :
- Numerical values
  - Upper Limit : 0.02mm
  - ☑ Symmetric Lower Limit
  - Lower Limit : -0.09mm
- Tabulated values
  - H9
- Single limit
  - ● MIN ○ MAX
  - Delta / nominal : 0mm

Options
Envelop Condition : (E)

OK   Cancel
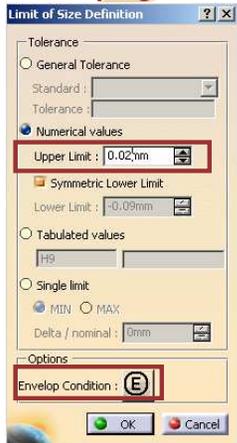
4   **Click OK in the Limit of Size Definition dialog box.**

$\emptyset 13.49 \pm 0.02$ (E)
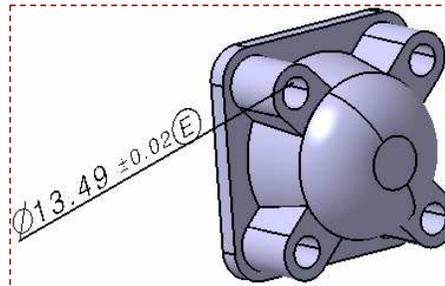
How to Create Semantic Form Tolerances

To create semantic form tolerances, select the element to tolerance and click the tolerancing advisor icon.The commands frame only displays form tolerances that correspond to the selected element

1 **Select the Face to be toleranced and Tolerancing Advisor tool**
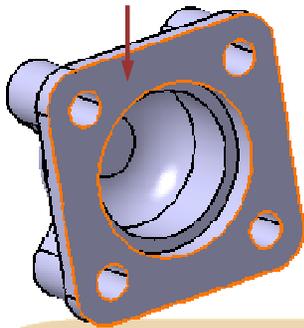
2 **Semantic Tolerancing Advisor displays corresponding Form dimensions. Select 'Flatness'**

Semantic Tolerancing Advisor

Existing Datum Reference Frame

Label

Add...
Remove
Edit...
Unselect

Commands

Geometric feature type | One surface

ABC
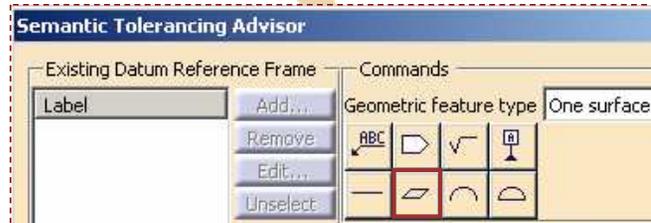
As the selected element is a plane surface, the only available tolerances are straightness, flatness, profile-of-line and profile-of-surface specifications.

3 **Enter the Tolerance Value**

Geometrical Specification | ? | X |

Tolerance value

⦿ Numerical: 0.5mm
◯ Tabulated:

⬭ | 0.5

General | Unit Basis | Tolerance Zone

Modifiers

Toleranced Element(s) Material Condition: Ⓜ Ⓛ
Non applicable

Free State Variation: Ⓕ

Tolerance Zone Direction

Plane direction: No selection | Unselect
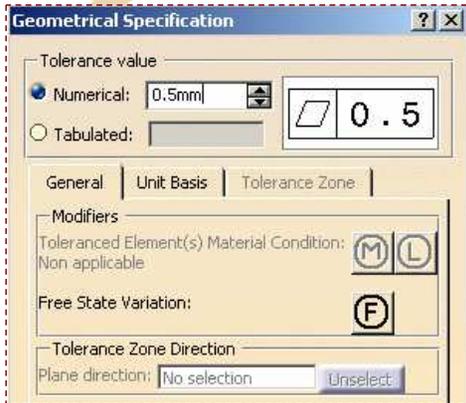
4 **Click OK in the Geometrical Specification dialog box.**

□ 0.5
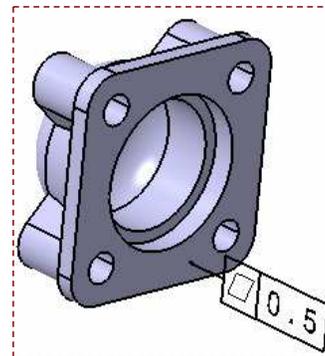
How to Create Semantic Orientation Tolerances (1/2)

To create orientation tolerances, you must have created a datum reference frame

1 **Datum Reference Frame A is already created.**
Select the Tolerancing Advisor tool
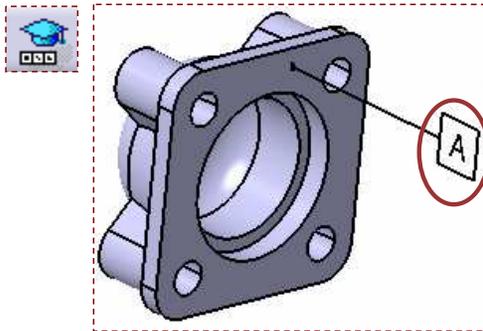
2 **Select the feature (Hole) to be toleranced**
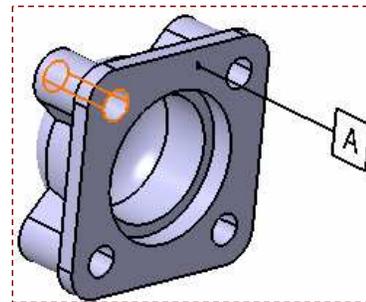
3 **Select the Datum Reference Frame A and also select 'Perpendicularity'**
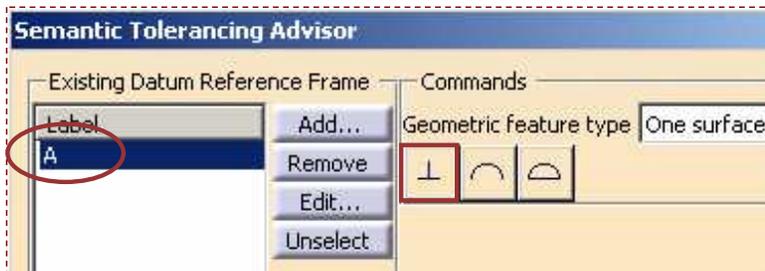
The commands frame only contains orientation tolerances that correspond to the selected element compared to the selected reference.

How to Create Semantic Orientation Tolerances (2/2)

4 **Enter Tolerance Values in the 'Geometrical Tolerance' Dialog box.**
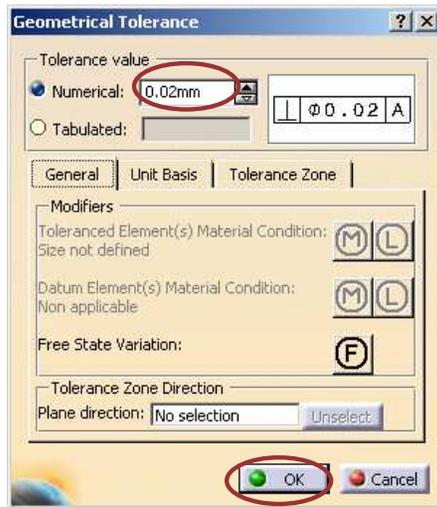
5 **Click OK in the 'Geometrical Tolerance' dialog box.**
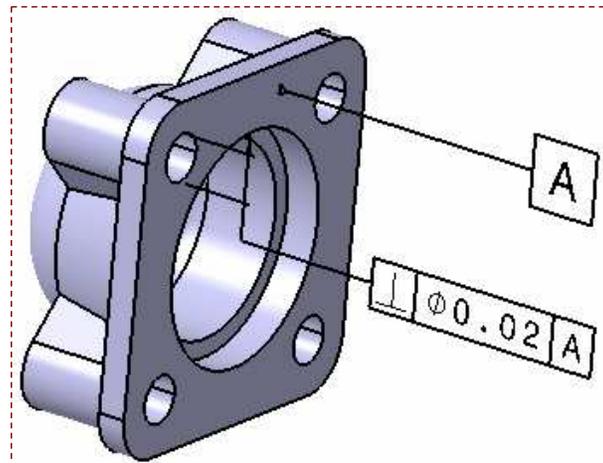
How to Specify Tolerance Zone Direction

Tolerance Zone direction can be specified for Form, Position, and Orientation type of tolerances. The tolerance zone direction is driven by functional needs.

1 When creating the following geometrical tolerance, click on 'Tolerance Zone Direction'.
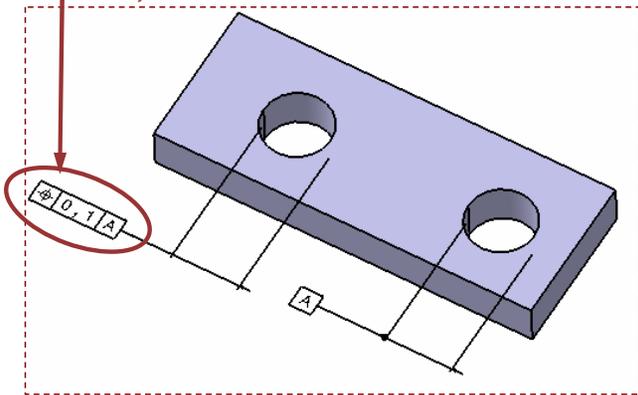
2 Select the Top edge of the pad as the direction.
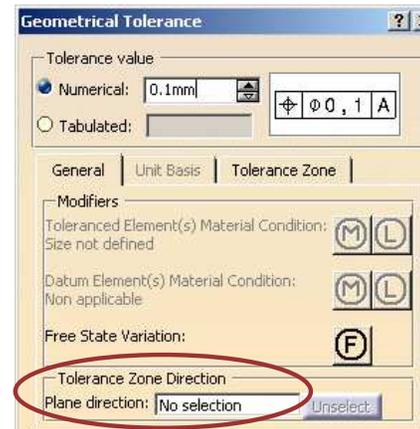
3 Click OK in the 'Geometrical Tolerance' dialog box.

Select this edge as direction
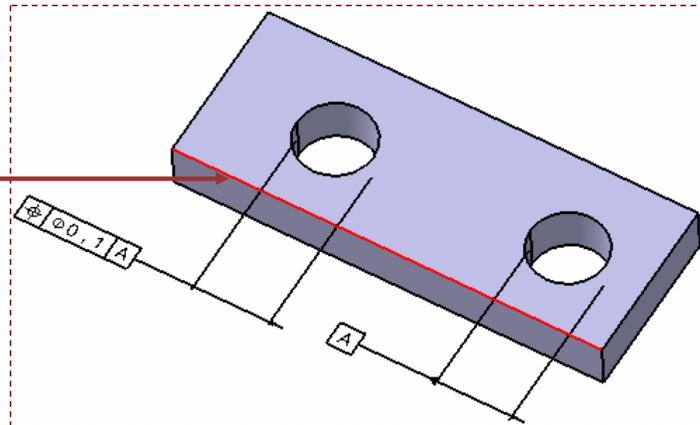
How to Create Semantic Position Tolerances

In order to create position tolerances, you must have created a datum reference frame

1 **Select the element to be toleranced, click the tolerancing advisor icon, and choose the datum reference frame from the list**
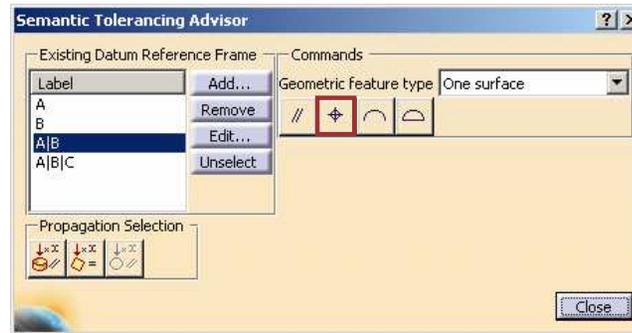
2 **Enter the Tolerance Value and Click OK.**
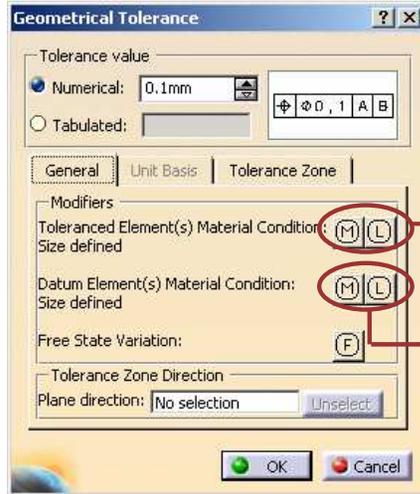
The commands frame only contains position tolerances that correspond to the selected element compared to the selected reference frame
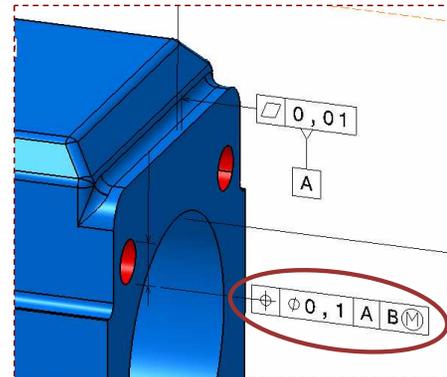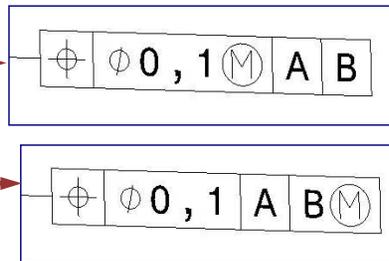
Angular Dimensions

You will see the display of the Upper Limit and Lower Limit value in the 'Limit of Size Definition' dialog box, same as that appears in 3D window.

Angular Dimension in 3D window.

Upper Limit
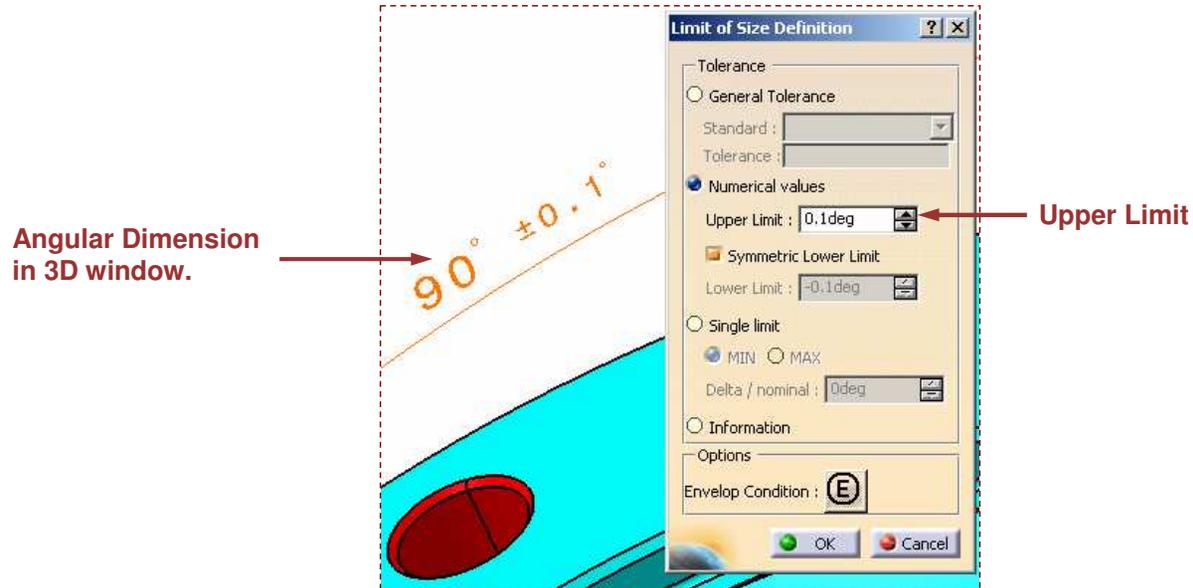
😎 **'Always try to create semantic tolerances and dimensions' option from Tools > Options > Mechanical Design > Functional Tolerancing and Annotation > Tolerancing > Semantic Control, should be selected.**

N Elements Tolerancing

In this skillet you learn N elements Tolerancing concept through various Illustrations.
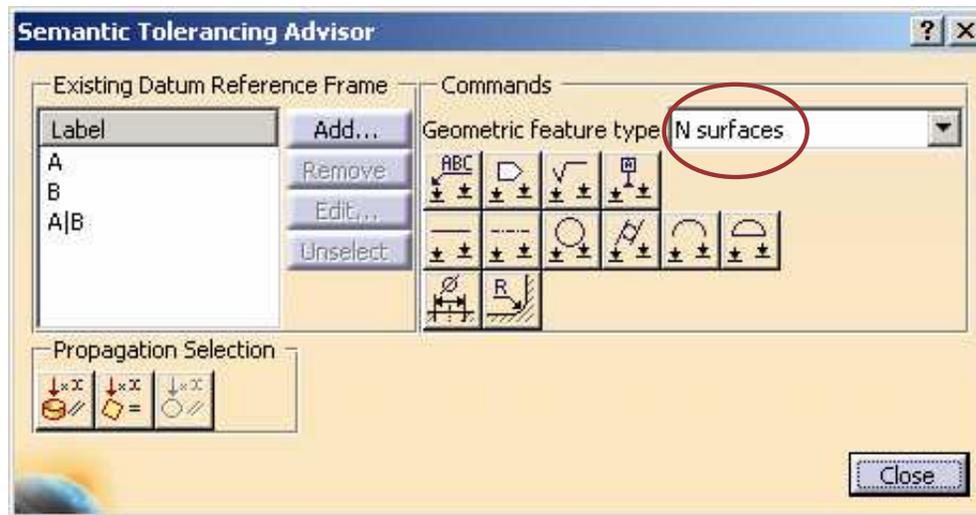
One Surface Illustration

- **N Elements Tolerancing**
  - ◆ **One Surface illustration:**
    - ▪ **Example**

Resulting Tolerancing Dialog Box

Selected Elements

Resulting 3D Annotations

N Surfaces Illustration (1/2)

- **N Elements Tolerancing**
    - **N surfaces illustration:**
        - **Common Zone**
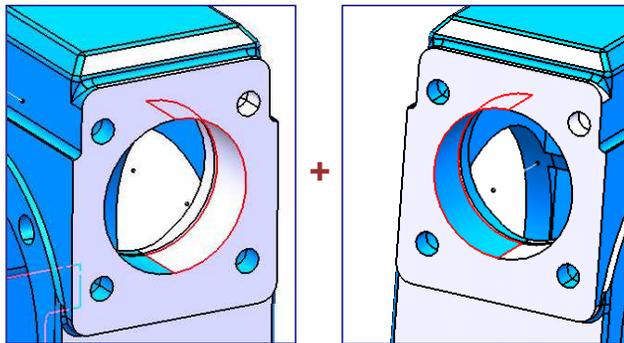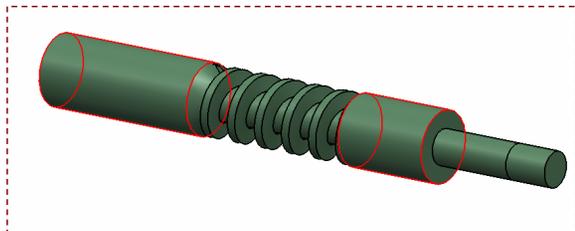
N surfaces symbol

Selected Elements
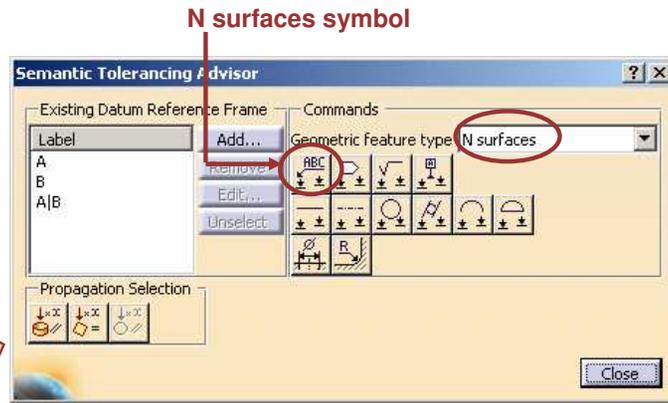
Resulting Tolerancing Dialog Box
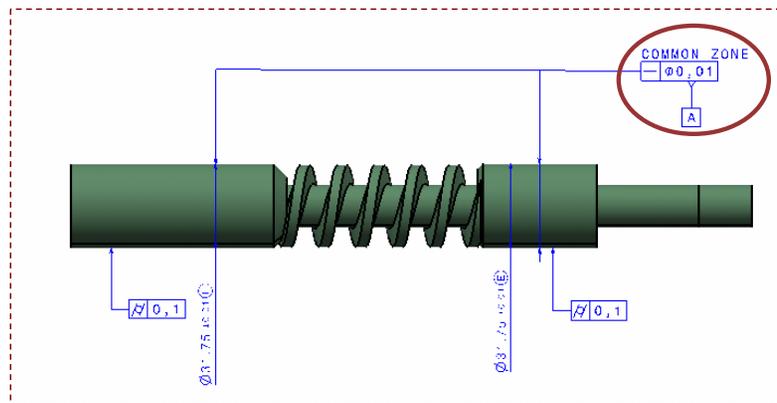
Resulting 3D Annotations

N Surfaces Illustration (2/2)

Using Propagation Selection in Semantic Tolerancing Advisor

1 **Select one of the faces of the cone shown below and click on Semantic Tolerancing Advisor icon.**
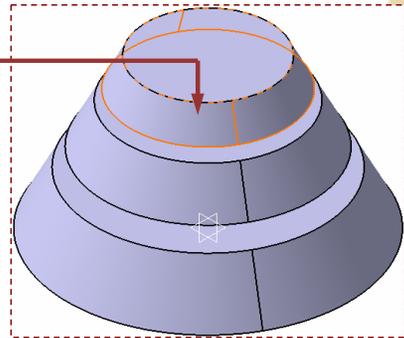
2 **Click on 'All same canonicity faces' icon.**

Face of cone
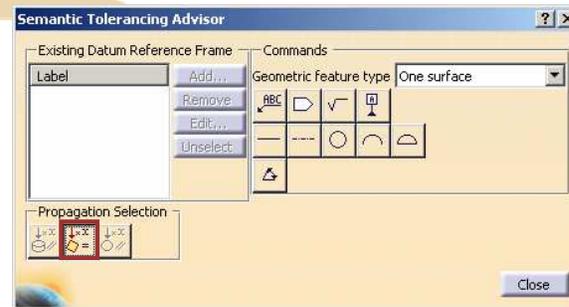
3 **Observe that the Geometric feature type has changed to 'N surfaces'.**
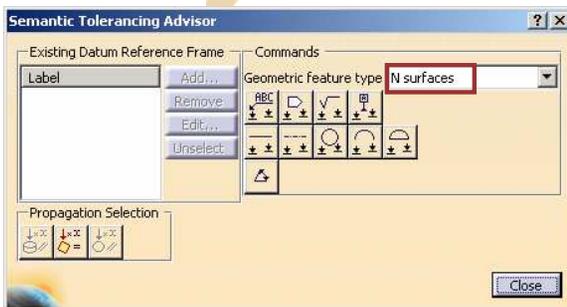
4 **Apply the appropriate tolerance to the multiple selected faces of the cone.**
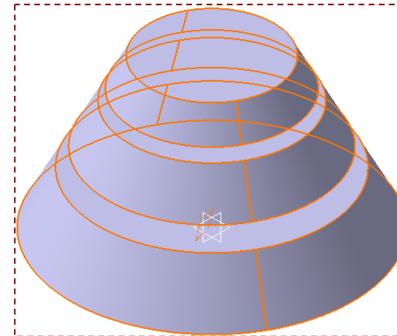
Tab/Slot Illustration

- **N Elements Tolerancing**
    - **Tab / slot illustration:**

Tab/ slot symbol

Resulting Tolerancing Dialog Box

Selected Elements

Resulting 3D Annotations

Pattern Illustration (1/3)

- **N Elements Tolerancing**
  - **Pattern illustration:**
    - **Example**

Pattern symbol

Resulting Tolerancing Dialog Box

Selected Elements

Resulting 3D Annotations

Pattern Illustration (2/3)

Using Propagation Selection in case of cylindrical features.

1 **Select one of the patterned hole feature and click on the Semantic Tolerancing Advisor icon.**
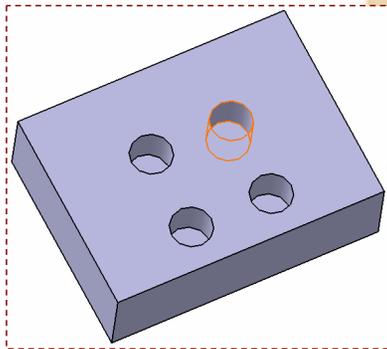
2 **Click on 'All same diameter parallel cylinder' icon**
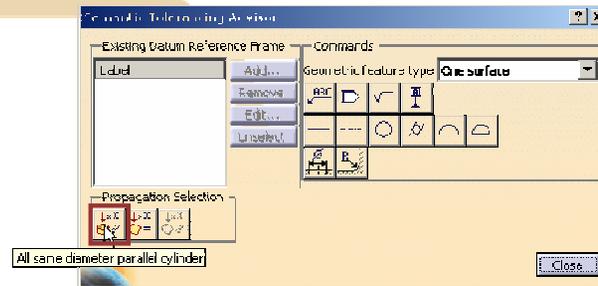
3 **Observe that the Geometric feature type has changed to 'Pattern'.**
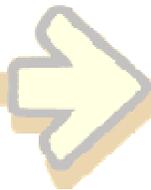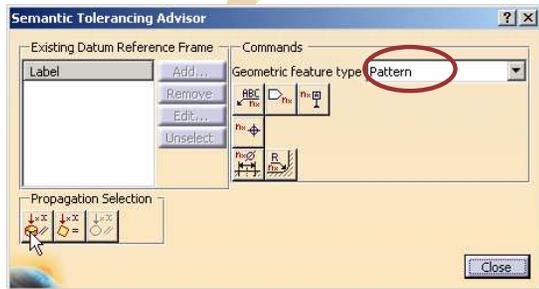
4 **Apply the appropriate tolerance to the multiple selected holes and Click Close.**
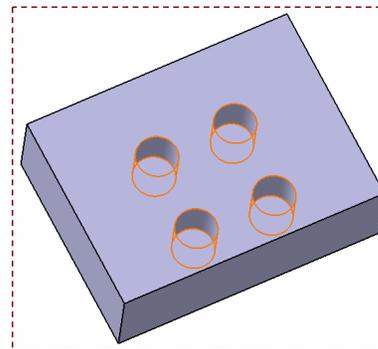
Pattern Illustration (3/3)

Using Propagation Selection in case of spherical feature.

1 **Select one of the patterned sphere feature and click on the Semantic Tolerancing Advisor icon.**
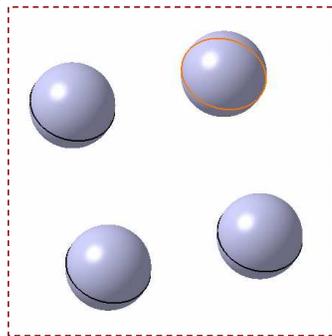
2 **Click on 'All same diameter parallel sphere' icon**
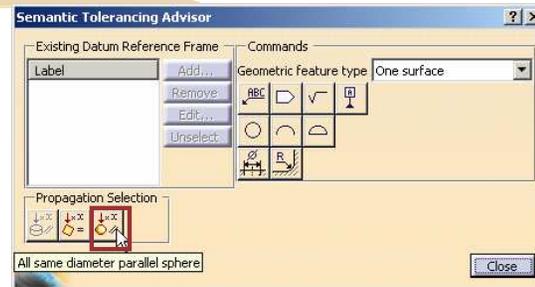
3 **Observe that the Geometric feature type has changed to 'Pattern'.**
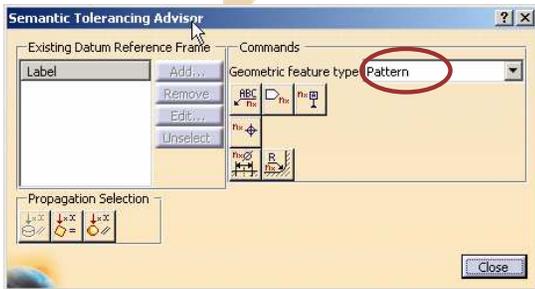
4 **Apply the appropriate tolerance to the multiple selected spheres and Click Close.**
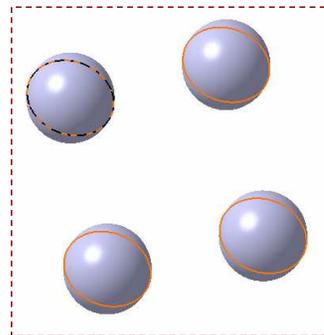
Creating Non-Semantic Annotations

In this skillet you will learn what are Non-Semantic Annotations and how to create them.
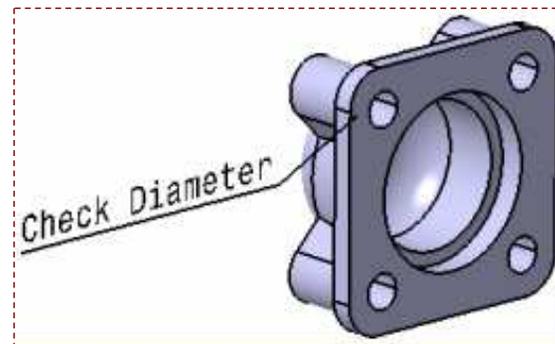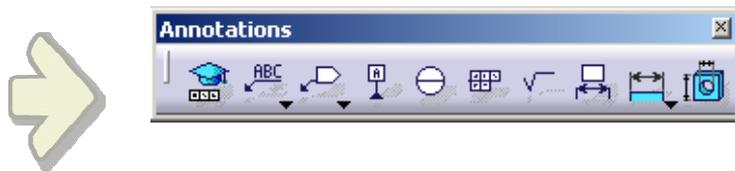
What are Non-Semantic Annotations

Non-Semantic Tolerances are CATIA Objects which do NOT automatically take into account the element to be toleranced and the context.

They are NOT defined in the ISO or ASME/ANSI standards. Non-Semantic 3D annotations can be used in case of company defined symbols and syntaxes that are not covered by standards.

When Non-Semantic Annotations are created:

- **Only graphical attributes are taken into account.**
- **There is no control over the attribute values.**
- **There is no control of consistency regarding the geometry on which it is applied.**

Non-Semantic Annotations can be created in the form of:

- **Text**
- **Flag Note**
- **Note Object Attribute (NOA)**

You can also create these annotations by using ' Tolerancing Advisor' tool.

This will be explained in the following slides.

How to Create Texts

Text creation is Non-Semantic type of Annotation

1 **Select the element the Annotate**
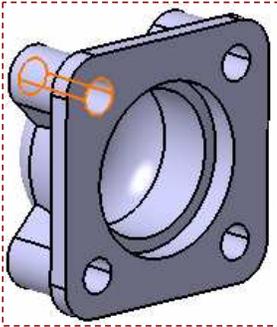
2 **Click on Tolerancing Advisor Icon and select the "Text with Leader" icon.**

Semantic Tolerancing Advisor

Existing Datum Reference Frame — Commands

Label   Add...   Geometric feature type   One surface

Remove   ABC

Edit...

Unselect

3 **Type the text**

Text Editor

Check Diameter

OK   Cancel

Check Diameter
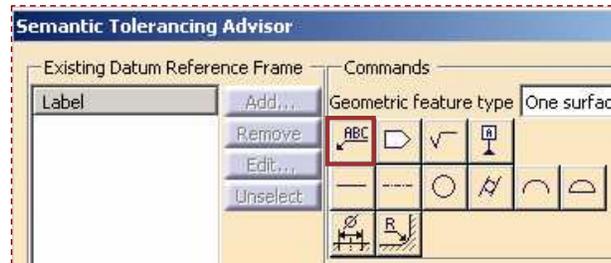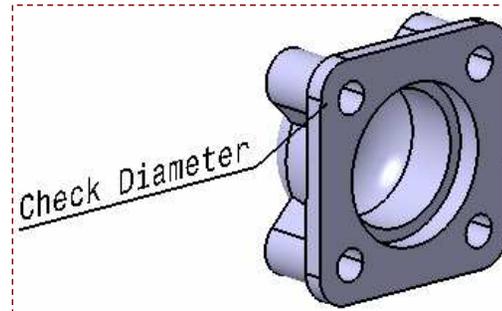
How to Create Flag Notes

Flag note creation is a Non-semantic Annotation. This function allows to add hyperlinks to the document

1 **Select the element to annotate**
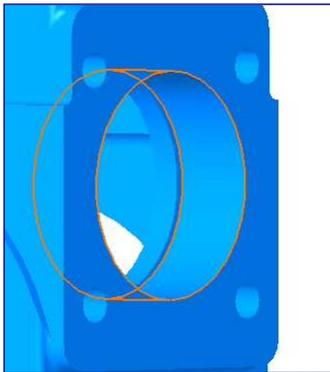
2 **Click the tolerancing advisor icon and select the "Flag Note with Leader" icon**

Semantic Tolerancing Advisor

Existing Datum Reference Frame

Label
A
B
A|B
A|B|C

Add...
Remove
Edit...
Unselect

Commands

Geometric feature type | One surface

Propagation Selection

Close

Hole Manufacturing

Click on the Note to open the attached document

3 **Type the name of the link and add the link**

Type the name of the link

Flag Note Definition

Definition
Name: Hole manufacturing
URL:
Browse...

Link to File or URL

Go to
Remove
Edit
More>>

OK   Cancel

Select the file to link to the element

Fraise à rainer extensible carbure

Pour l'usinage de :

- Bois
- Agglomérés
- Produits synthétiques

Jeu de 2 fraises à rainer extensible carbure
Ø 120 – Alésage 30H7.

e	z	Référence	Prix TTC
4 à 7,5	2 + 2	LB 001002	69.46 €
5 à 9,5	2+2	LB 001001	67.10 €
5 à 9,5	4+4	LB 001004	76.21 €

How to Apply Roughness

Roughness creation is a Non-Semantic Type of Annotation

1   **Select the element to annotate**
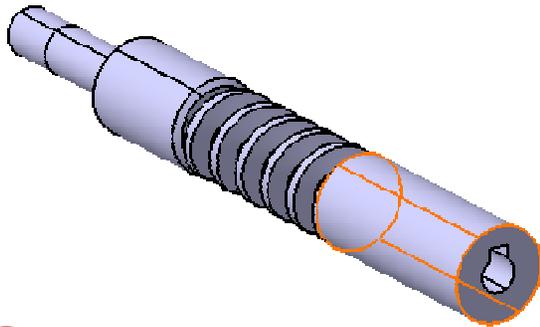
2   **Click the tolerancing advisor icon, select the "Roughness" icon, choose the type and the roughness value**
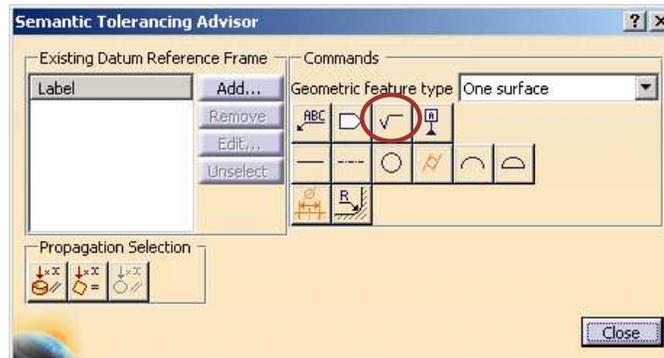
3   **Select appropriate values**
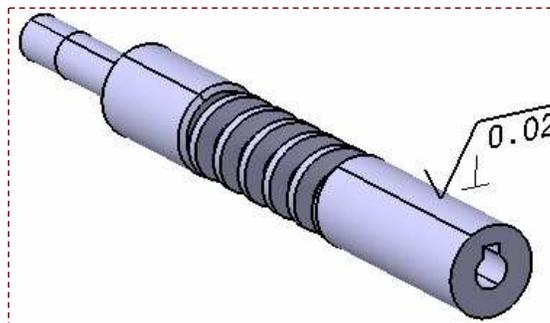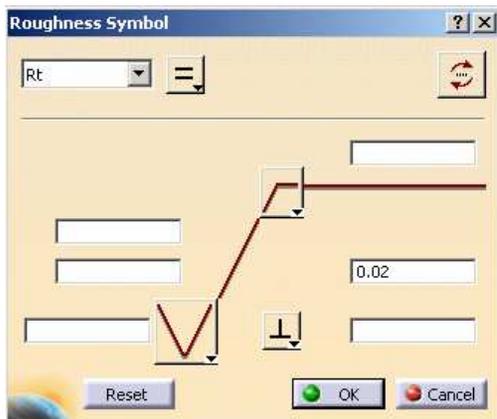
Creating Framed Dimensions

*In this skillet you will learn Framed Dimension creation*
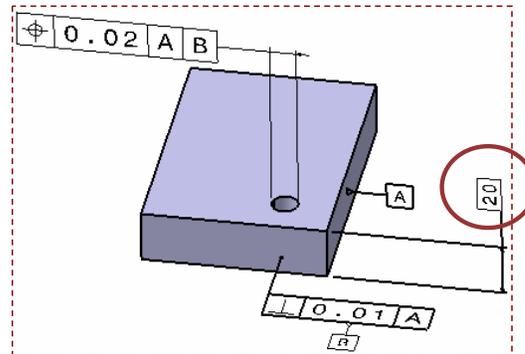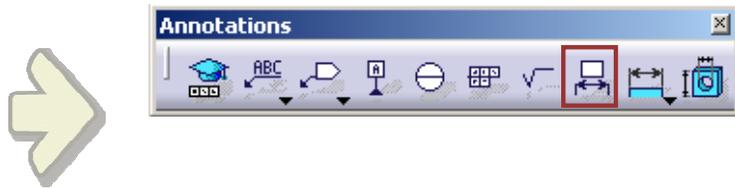
What are Framed Dimensions

Framed Dimensions are used to specify the location or size of a element. They must be linked to partial references, restricted areas, or one of the following tolerances.

Framed Dimensions represent the dimensions which will not be altered during Manufacturing.

Elements	Type of Tolerances	Tolerancing characteristics	Symbols
Isolated elements	Profile	Straightness	
		Flatness	
		Circularity	
		Cylindricity	
Isolated or associated elements		Profile-of-Line	
		Profile-of-Surface	
Associated elements	Orientation	Parallelism	
		Perpendicularity	
		Angularity	
	Position	Position-with-DRF	
		Concentricity	
		Symetry	
	Runout	Circular Runout	
		Total Runout	

How to Create Framed Dimensions

You will see how to create Framed Dimensions

1 **Select the Framed Dimension icon and select a annotation on which you can add framed dimension.**
In the example, Position annotation is selected

2 **Click 'Start creation mode'.**
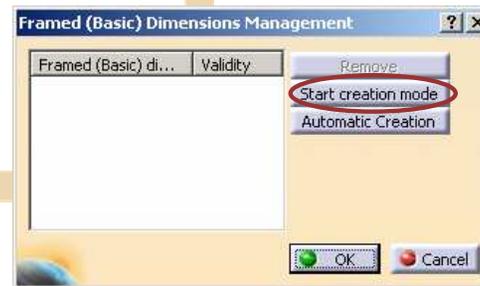
4 **Observe the created Framed Dimension.**
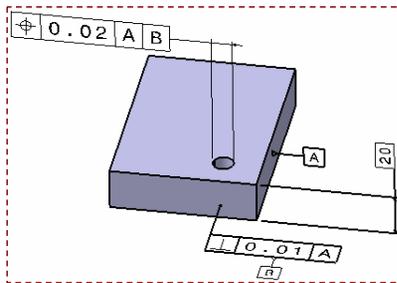
3 **Create the Dimension and click in open area.**
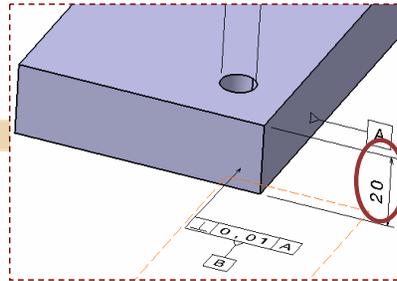
5 **Click 'End creation mode'.**

6 **Click OK to validate the Dimension.**
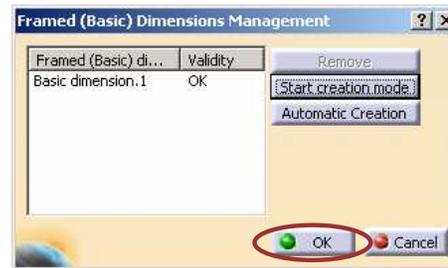
Automatic Framed Dimensions

We can create Automatic Framed Dimensions in the following cases:

- **Selection of a Datum Reference Frame**
- **Selection of one of the following types of geometrical tolerances with Datum Reference Frame:**
  - **Position (localization),**
  - **Profile of any line,**
  - **Profile of a surface,**
  - **Angularity.**
- **Selection of a position (localization) tolerance without datum reference frame applied to a pattern of cylindrical features.**
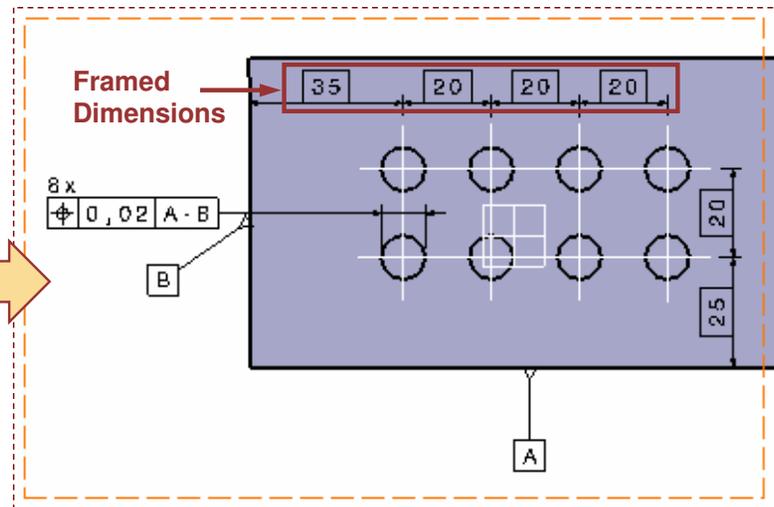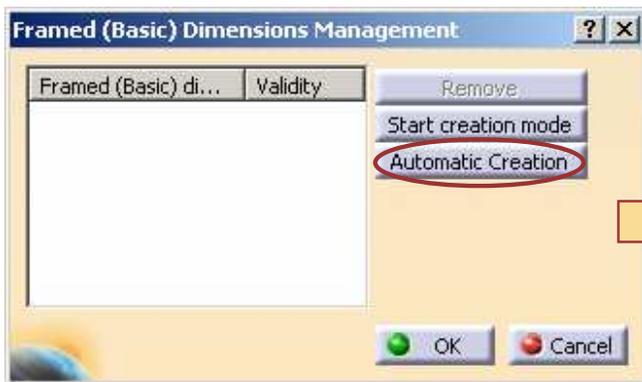
How to Create Automatic Framed Dimensions (1/2)

You will see how to create Automatic Framed Dimensions

1   **Click on 'Framed Dimension' icon and select an annotation on which you can add framed dimension.**
In the example, Position annotation is selected
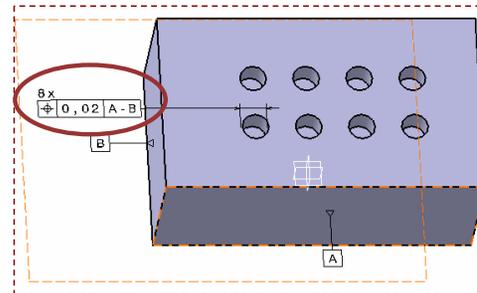
2   **Click on 'Automatic Creation'.**
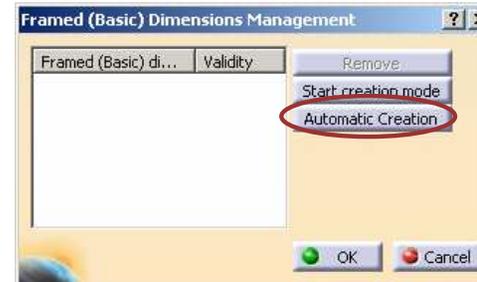
3   **Click on 'Start creation mode' to validate the Basic dimensions created.**
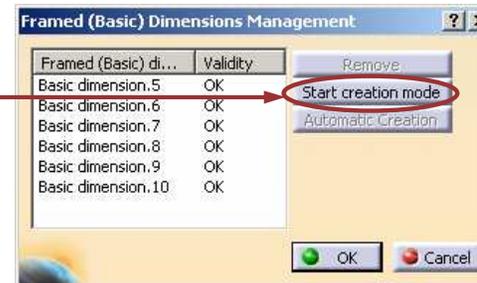
How to Create Automatic Framed Dimensions (2/2)

The framed dimensions are created for the holes with respect to the Datum Reference Frame.
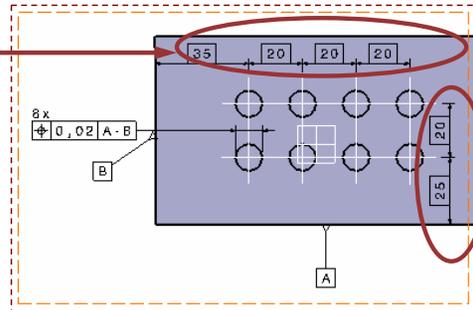
4 **Click on 'End creation mode'**
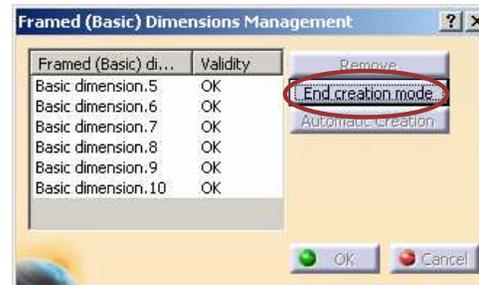
5 **Click 'OK' to validate the automatically created framed dimensions.**
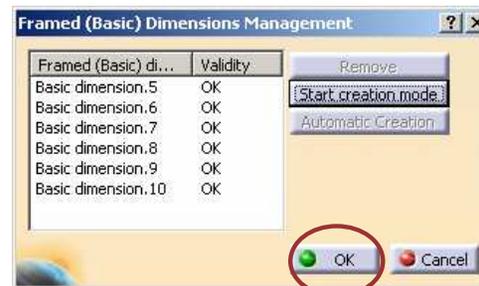
Student Notes:

To Sum Up

In this lesson you have seen how to:

- **Work with the Tolerancing Advisor**
- **Create Framed Dimensions**
- **Create Semantic and Non-Semantic Annotations**

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Creating & Managing FTA Annotations

You will learn to create different Annotation Planes and will learn tools to manage Annotations.

- **Creating View/Annotation Planes**
- **Aligned Section Views and Section Cuts**
- **Offset Section view / Section Cut**
- **Editing and Managing Annotations**
- **Managing Captures**
- **Generating Check Report**
- **To Sum Up**

Creating View/Annotation Planes

In this section you will learn to create different Annotation planes and use them to extract drafting views

Views

Front View

Annotation Plane

Section View

Annotation Plane

Section Cut

Annotation Plane

Offset Section View/Cut

Annotation Plane

Aligned View/Cut
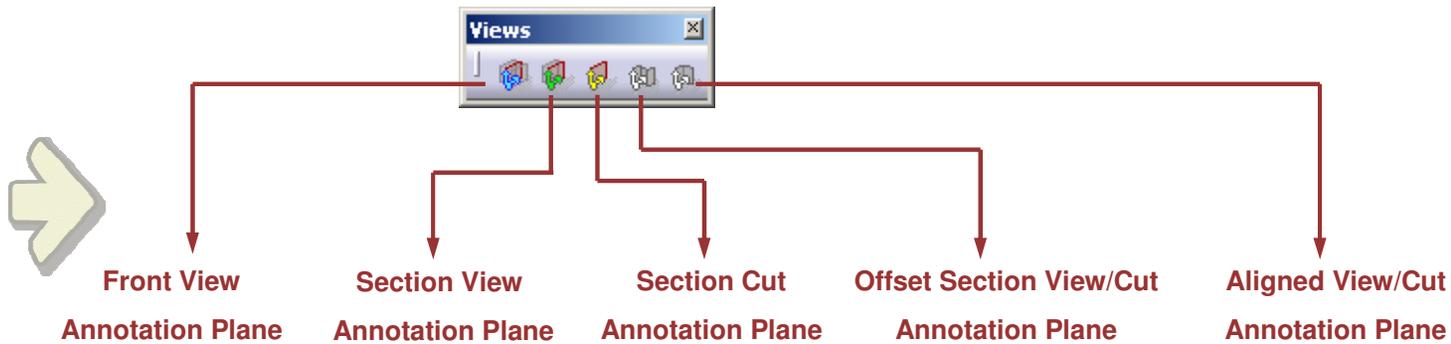
Annotation Plane

About Annotation/View Planes (1/3)

There are Three types of Annotation/View Planes

- **Front View Annotation Planes**
- **Section View Annotation Planes**
- **Section Cut Annotation Planes**

The views/annotation planes are displayed with a dashed frame in 3D along with the origin and the axis system. The frame gets automatically resized to accommodate new annotations. To create an annotation in a particular view plane you have to make it 'Active'. This activated plane is 'preferred' to receive new annotations.

CATIA checks whether the new annotations can be created in this 'activated' plane or not. If 'not', it prompts you for automatic creation of new view plane.

'Active' Front view

Frame

Front views: Represented by blue axis

Section views: Represented by Green axis

Section Cut views: Represented Blue axis

The normal axis is red until you create an annotation
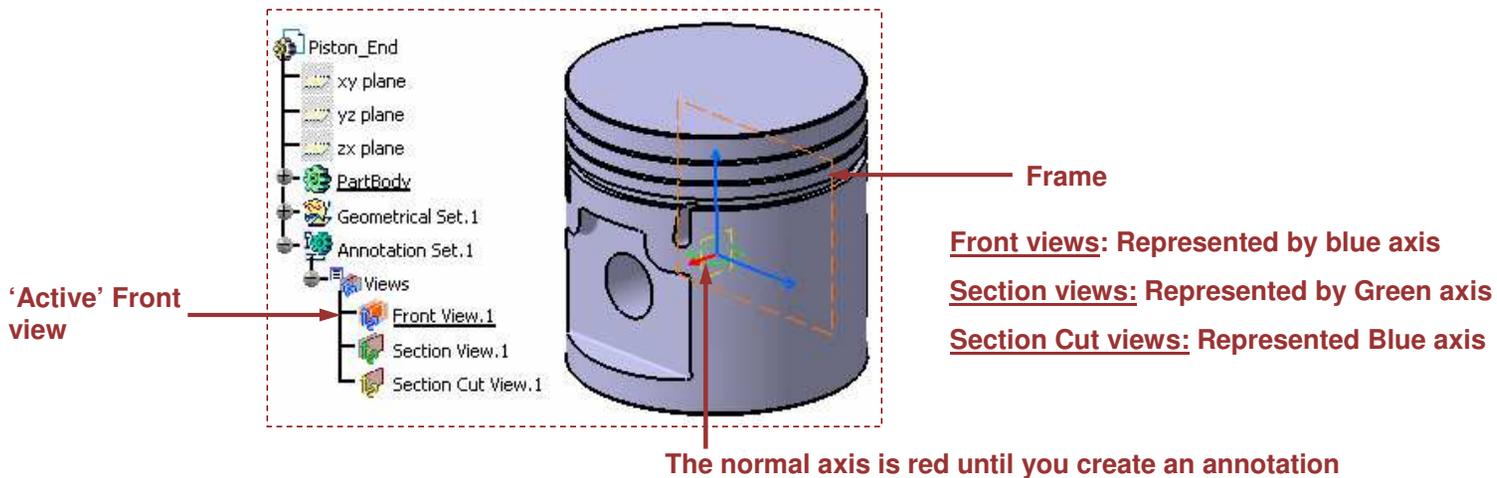
About Annotation/View Planes (2/3)

Why Annotation/View Planes?

Annotation planes are used to provide support to annotations. Whenever an annotation is created, it is always created on an Annotation Plane. Using these Planes, you can create different annotation or view planes. Any annotation (Datum, Text etc.) that you will create will lie on one of these planes.

Creating annotations on the view planes helps to transfer 3D annotations created on parts into the drawings. This is done by extracting 2D drawings using 'View from 3D' functionality in Drafting Workbench. The drawings generated will have these annotations embedded in 3D part.


Annotations are automatically created in the drawing

This is section View

Section View.1C-C
Scale: 3:1
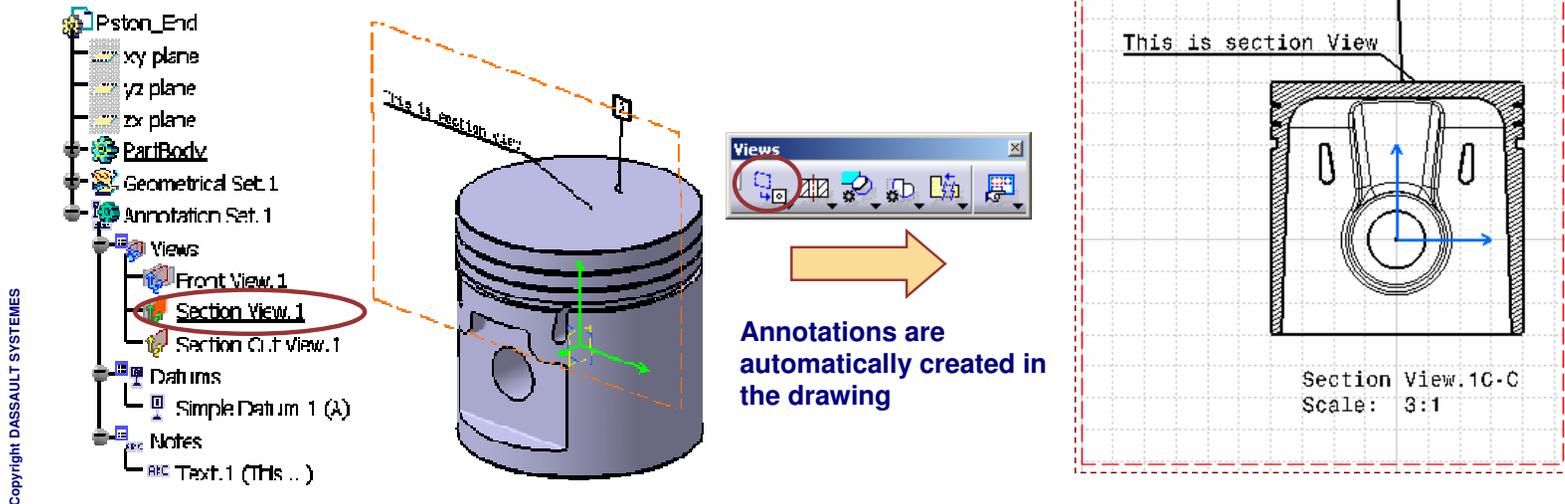
About Annotation/View Planes (3/3)

How Annotations will lie on a Particular Annotation Plane?

- **If the Annotations are created on the view plane itself.**
- **In case when creating front/projection views, If the annotations lie in the planes parallel to the view plane and are in foreground and background spaces.**
- **In case of creating of Section views, If the annotations lie in the planes parallel to the view plane and in the background space bounded by this view/annotation plane.**
- **The Annotation Plane should intersect the related geometry.**

How to Create Front View Annotation Plane

Front Views are represented by blue axis and identified as "Front View" in the specification tree

1 Access the Views Toolbar and select Front View/Annotation Plane

2 Select the Planar geometry element or an Axis element. Here ZX Plane is selected
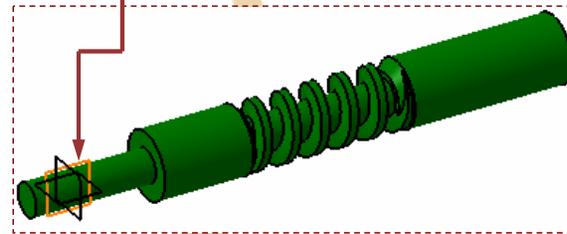
3 You can see a Annotation plane is created. You can use this plane to support annotations
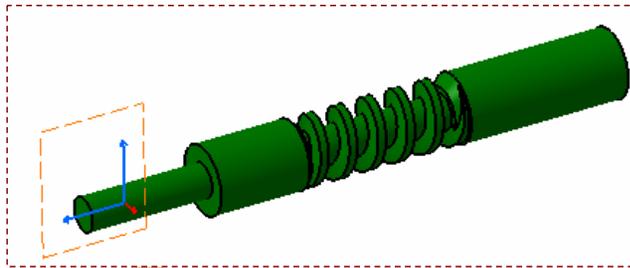
4 You can Extract a 2D drawing in the Drafting workbench. Use 'View from 3D' to extract the view

Front View.1
Scale: 1:1
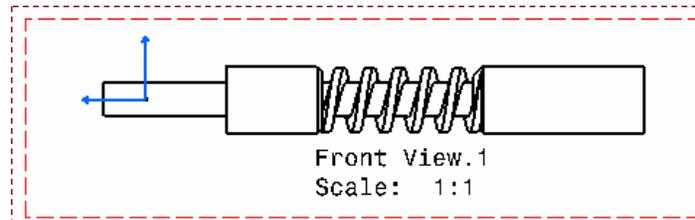
Front View is Extracted

General Process of Extracting 2D Views from 3D (1/3)

1 Open the model containing 3D annotations and set them all visible

2 Switch to Generative Drafting workbench

3 Access the 'View from 3D'

Projections

4 Go back to 3D model and select the annotation plane either from the Tree or from the 3D geometry
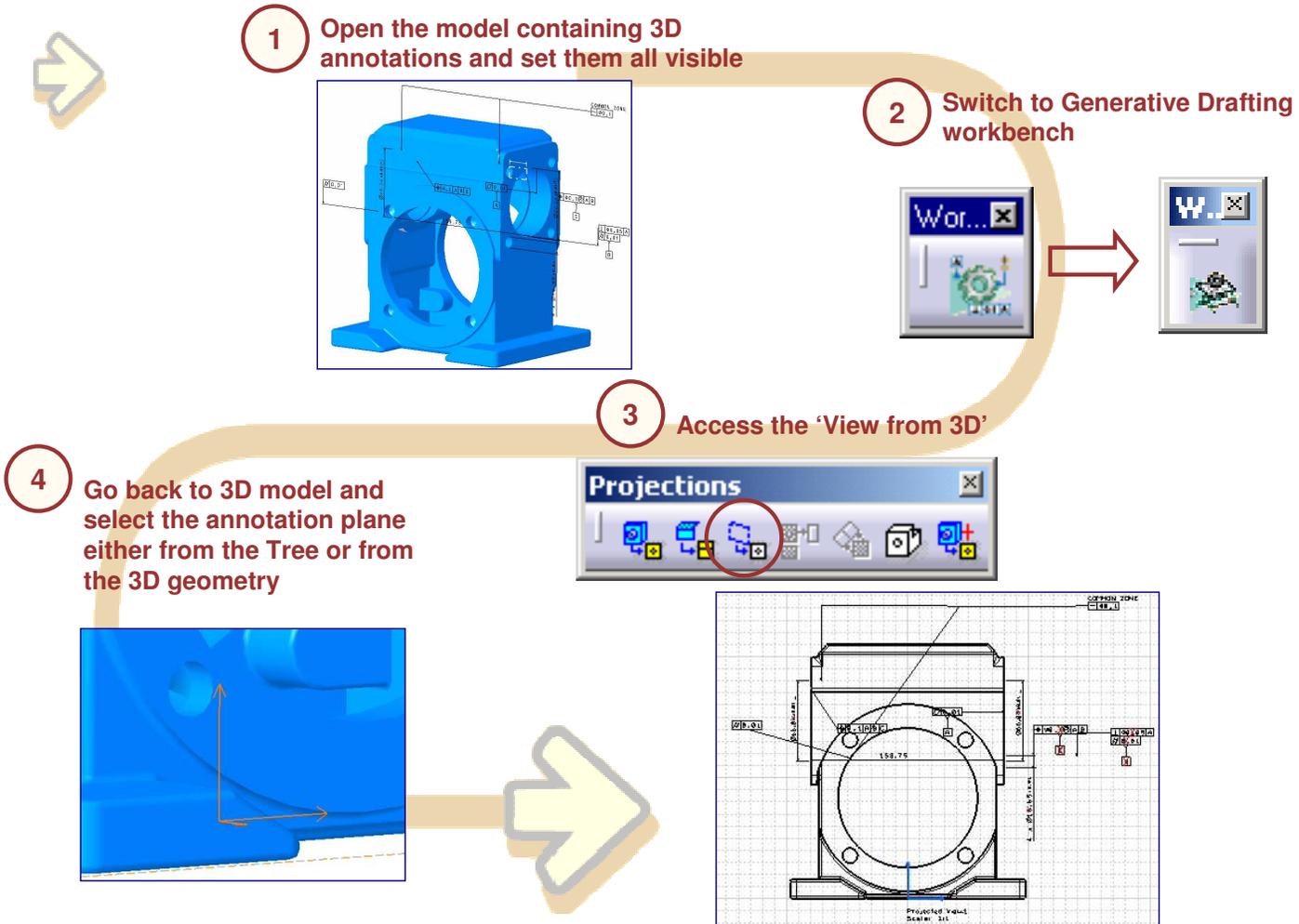
General Process of Extracting 2D Views from 3D (2/3)

Extracted View Position

When you extract a 2D view from 3D, the 2D view can be placed anywhere in the drafting
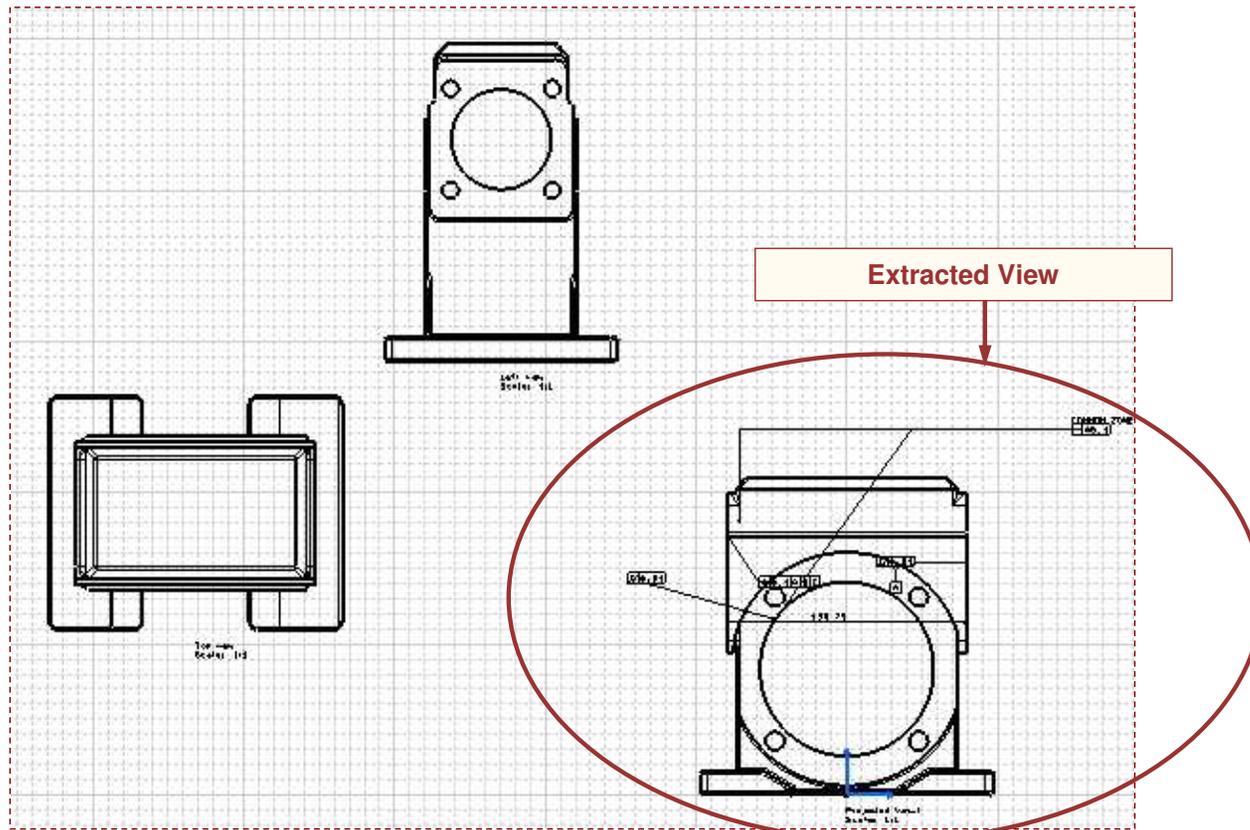
Extracted View

General Process of Extracting 2D Views from 3D (3/3)

Red crosses explanation

Red crosses mean that the geometrical element(s) associated to the annotation does not appear in the 2D view
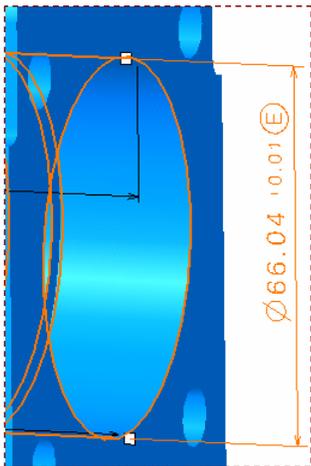
Figure 1
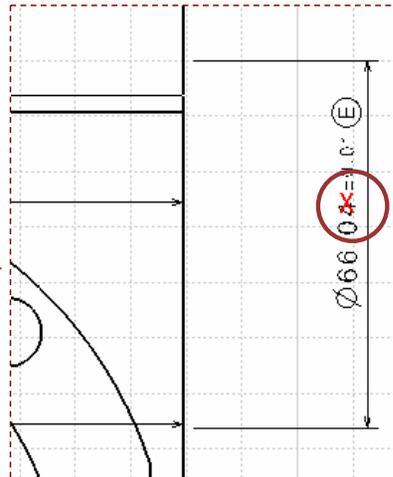
Figure 2

The dimension is on a hole that is not visible in this view.

The display of Red Cross can be avoided by changing the settings of the generated view. Right-click on the view > Properties > View From 3D > Generation Mode of Annotation. Deactivate 'Generate Red Cross on Annotation'.
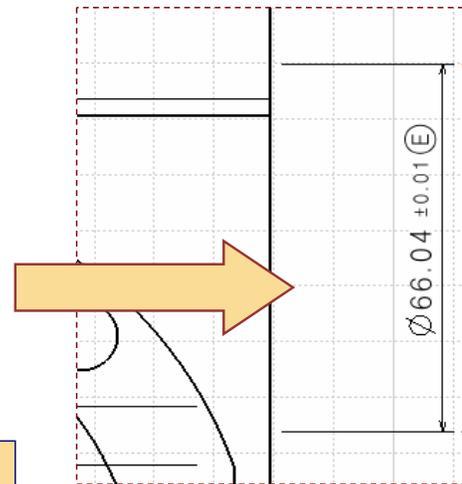
For more settings refer to 'The User Settings' Skillet > Generative Drafting Settings.

Figure 3

Aligned Section Views and Section Cuts

In this skillet you will learn how to create Aligned Section Views and Cuts
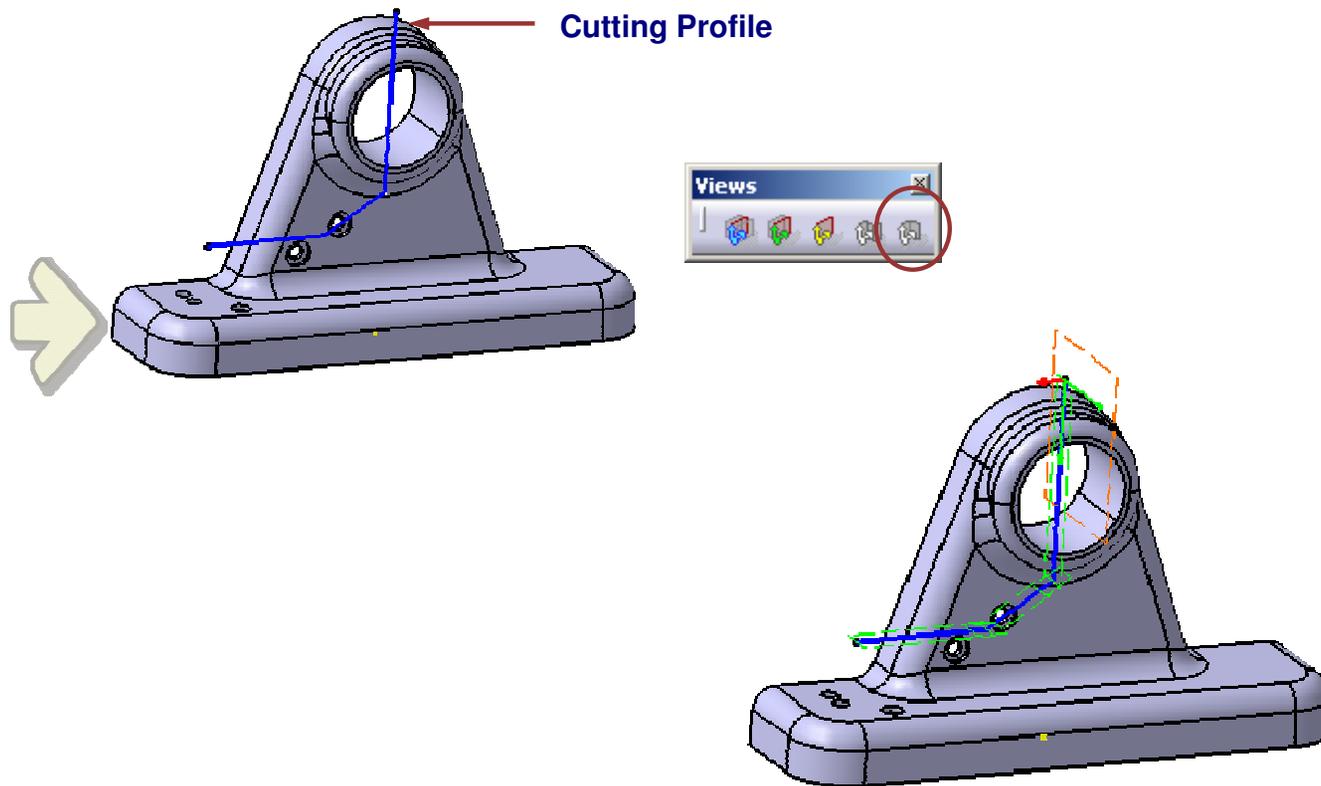
Cutting Profile

Creating Aligned Section view/Section Cut

- **An aligned section view/aligned section cut is created from a cutting profile defined from non-parallel planes. In order to include in a section, certain angled elements, (the cutting plane) may be bent so as to pass through the required features. The plane and feature are then imagined to be revolved into the original plane.**
- **Aligned section views are made up of several section views/annotation planes.**
- **Aligned section cuts are made up of several section cut views/annotation planes.**

Aligned Section views/cuts

The view is extracted using this plane
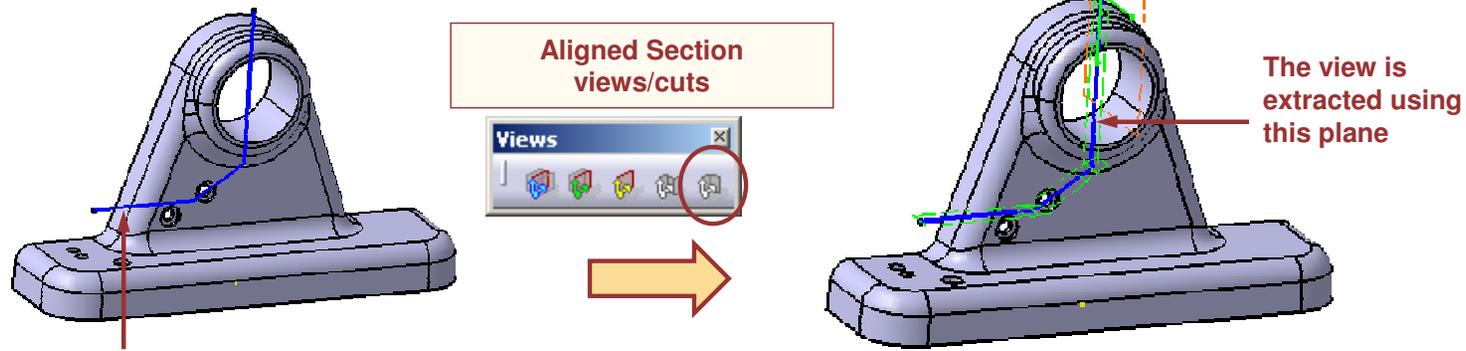
The Sketch represents a cutting profile. The sketch geometry contains lines which are not perpendicular to each other

Several annotation planes are created passing through the profile. You can use these planes to create aligned cuts/views

- **Using any one of these planes you can extract views.**
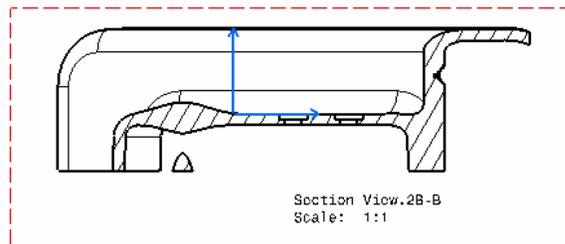
Section View.2B-B
Scale: 1:1

How to Create Aligned Section Views/Section Cuts

You will learn how to create Aligned Section Views/Section Cuts

1 **Create a profile (that only contains lines) in sketcher workbench**
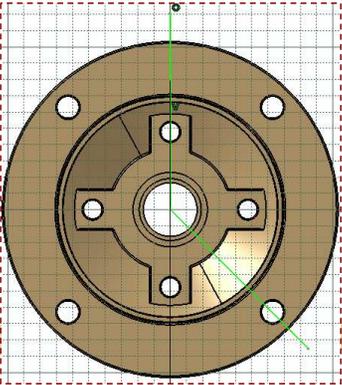
2 **Switch to Functional Tolerancing & Annotation workbench and select the Aligned Section Views/Section Cuts function**

3 **Select a profile (or create a new one) and the view type. Click to validate the view creation**

4 **Create Annotations and Extract the 2D View from a 3D Aligned View**
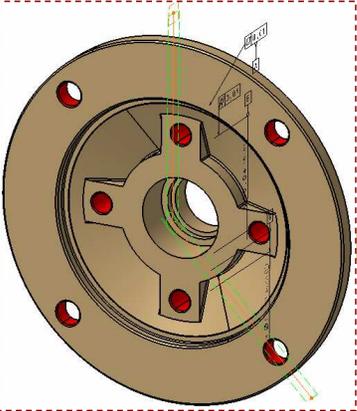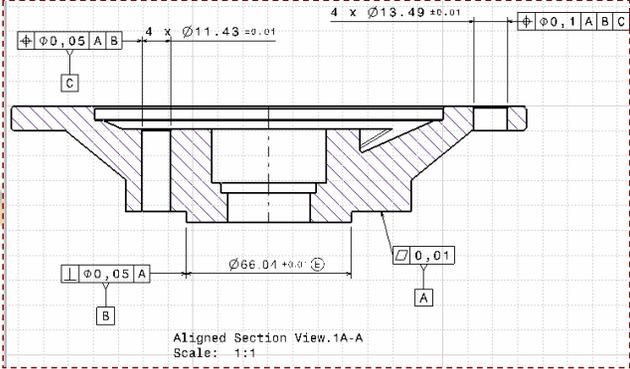
Aligned Section View.1A-A
Scale: 1:1

More on Aligned Section Views/Section Cuts

Section Cuts are represented by yellow axis and identified as "Section Cut" in the specification tree

Section Views are represented by green axis and identified as "Section View" in the specification tree
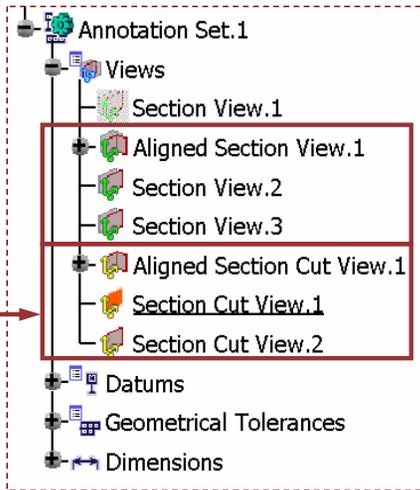
In the specifications tree, The Section Views/Cuts are ordered as children of the Aligned View but are also created independently
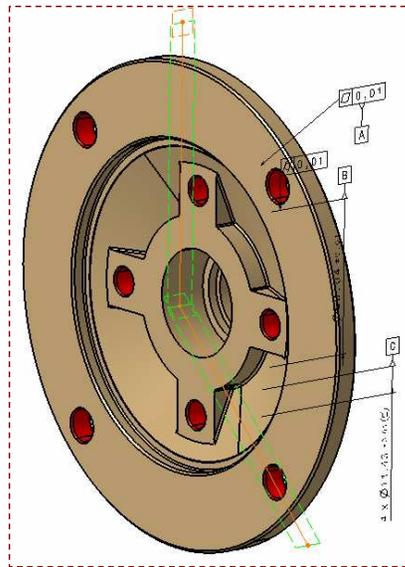
Aligned Section View Planes (Green color)
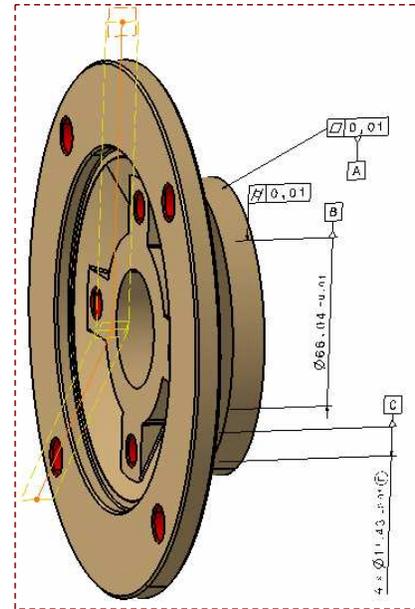
Aligned Section Cut Planes (Yellow color)

Offset Section View/Section Cut

In this skillet you will learn how to create Offset Section Views and Cuts

Cutting profile
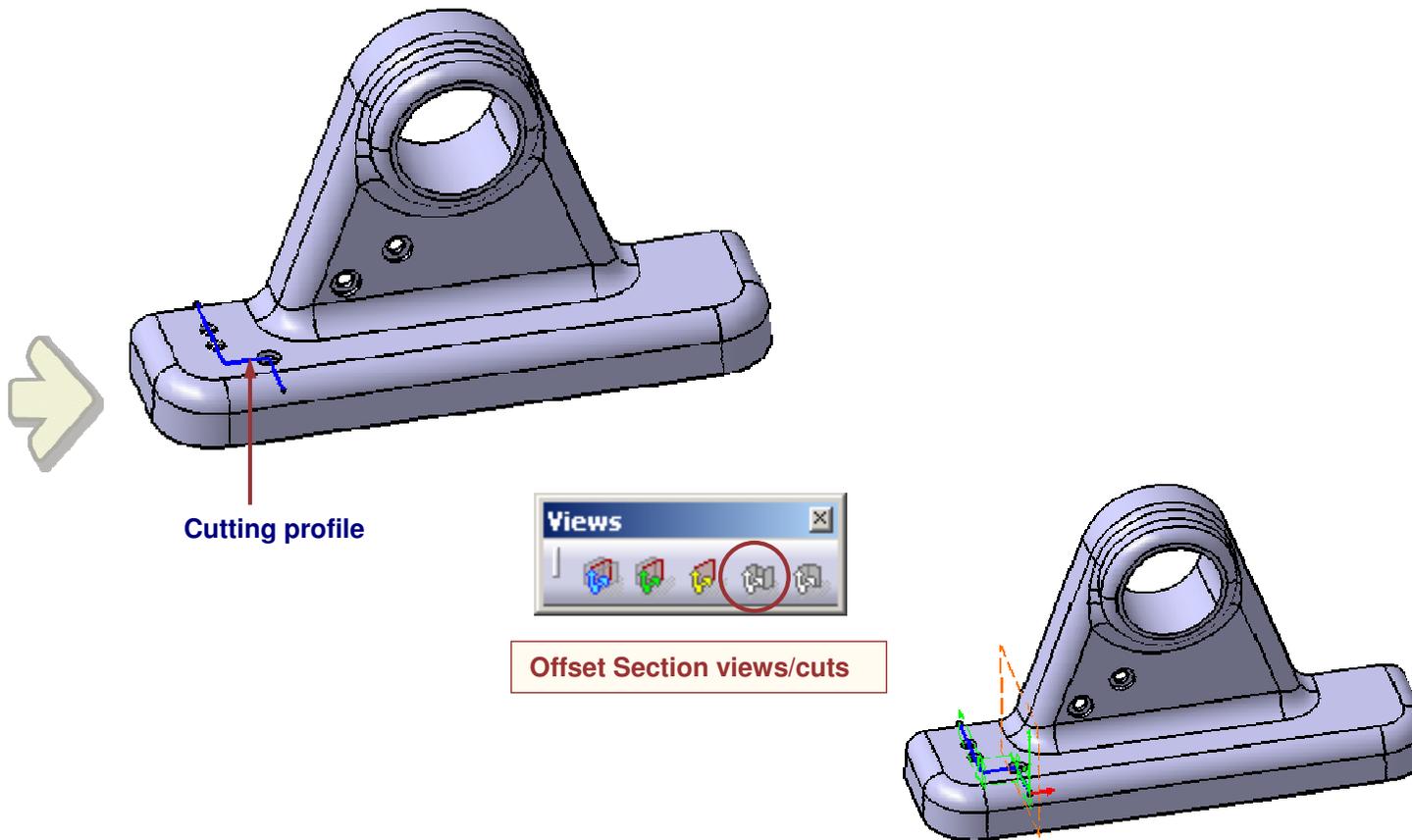
Offset Section views/cuts

Creating Offset Section View/Section Cut

- **Offset section views/offset section cuts let you show several features that do not lie in a straight line by offsetting or bending the cutting plane, which is often desirable when sectioning through irregular objects.**
- **Offset section views are made up of several section views/annotation planes.**
- **Offset section cuts are made up of several cuts views/annotation planes.**
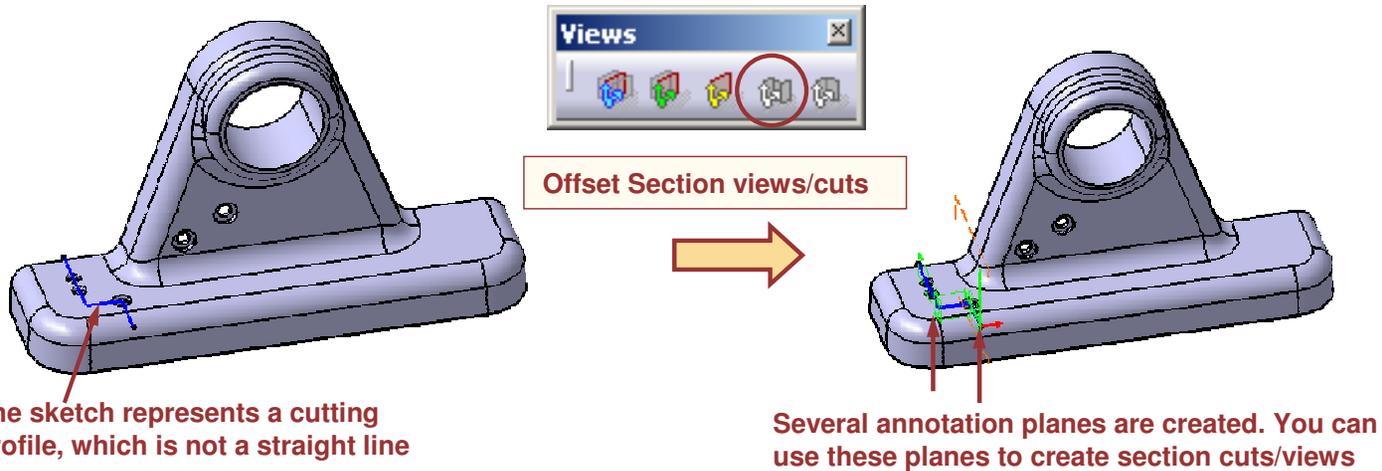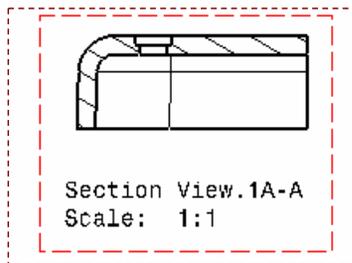
Offset Section views/cuts

The sketch represents a cutting profile, which is not a straight line

Several annotation planes are created. You can use these planes to create section cuts/views

- **Using any one of these planes you can extract views.**

Section View.1A-A
Scale: 1:1

How to Create Offset Section Views/ Section Cuts

You will learn how to create Offset Section Views / Section

View Type
⦿ Section ○ Section Cut
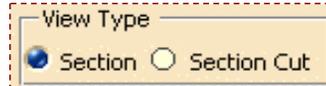
1 **Create a profile (that only contains perpendicular or parallel lines) in sketcher workbench. (The starting and ending profile segments must be parallel.)**
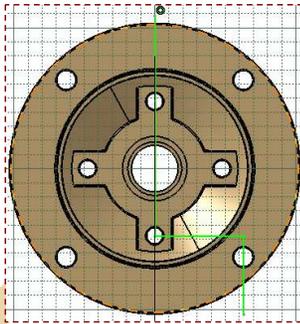
2 **Switch to Ft&A workbench and select the Offset Section Views / Section Cuts function**

Views

3 **Select a profile (or create a new one) and the view type. Click to validate the view creation**

Offset View Creation **? X**

Offset View Type
⦿ Section View ○ Section Cut

Profile
No selection

Invert Normal

⦿ OK ⦿ Cancel

4 **Create Annotations and Extract the 2D View from a 3D Offset View**

4 x Ø11.43 ±0.01

⊕ Ø0,05 A B

4 x Ø13.49 ±0.01

⊕ Ø0,1 A B C

C

⌲ 0,01

Ø66.04 ±0.01 Ⓒ

⊥ Ø0,05 A

A

B
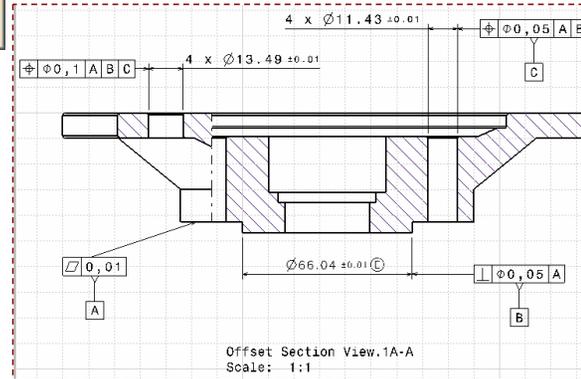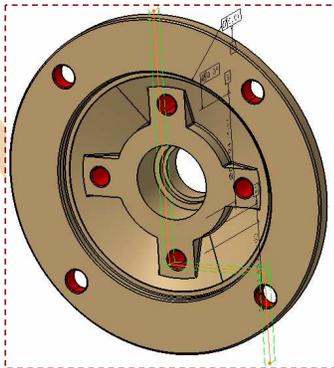
Offset Section View.1A-A
Scale: 1:1

More on Offset Section Views/Section Cuts

Section Cuts are represented by yellow axis and identified as "Section Cut" in the specification tree

Section Views are represented by green axis and identified as "Section View" in the specification tree.
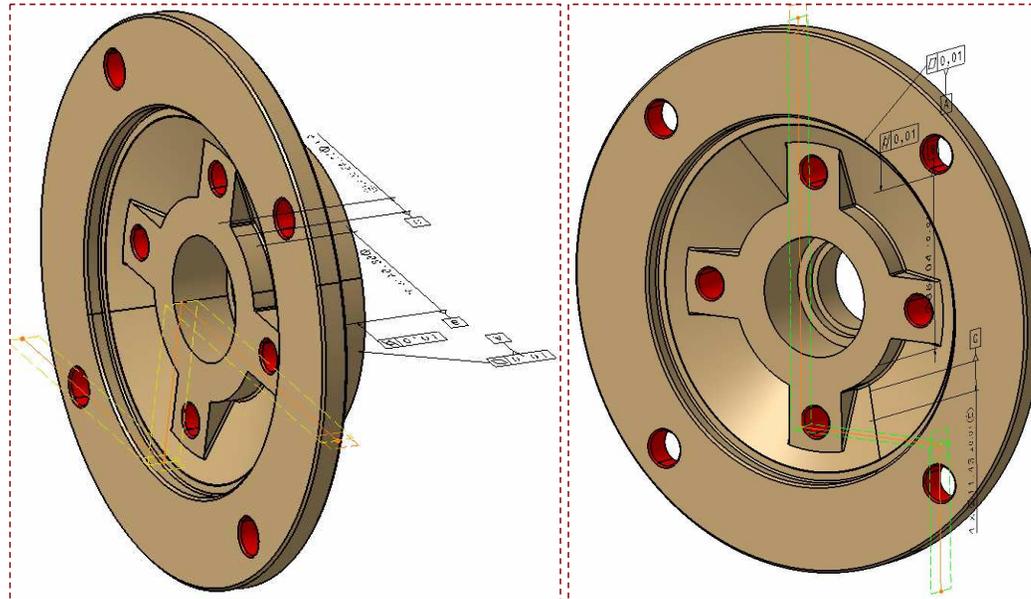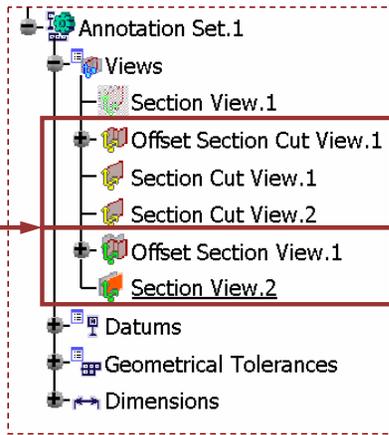
In the specifications tree, The Section Views/Cuts are ordered as children of the Offset View but are also created independently

Editing and Managing Annotations

In this skillet you will learn how to manage Annotations by transferring and filtering Annotations

How to Activate Annotation Planes

**It is possible to Activate a Annotation Plane and select every annotation created
on that plane.**

1   **Select the Annotation Plane**
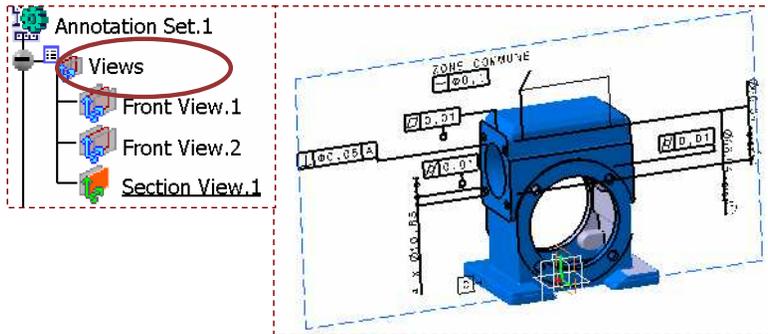
2   **Right-click the annotation plane and
choose "Select Annotations"**
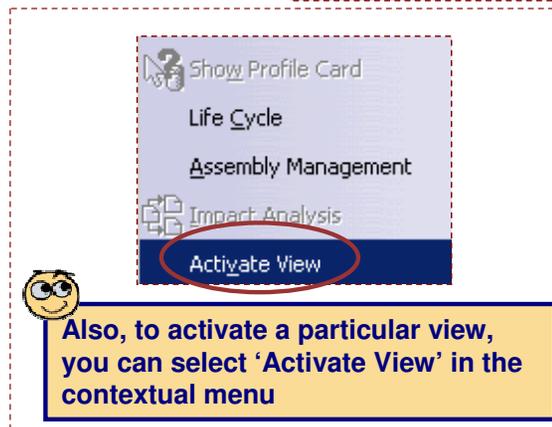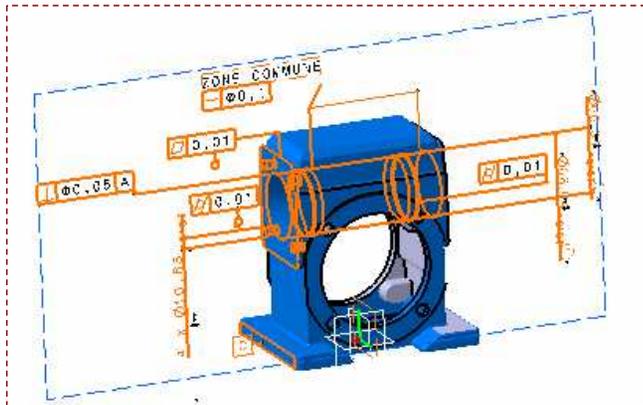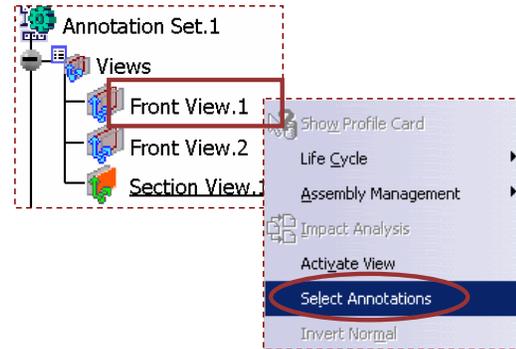
**Observe that all Annotations are selected on that
particular Annotation plane**

**Also, to activate a particular view,
you can select 'Activate View' in the
contextual menu**

How to Transfer Annotations

It is possible to transfer annotations from one Annotation plane to another

1 **Select the Annotation(s) to transfer.
Select Datum A lying on front View.1**
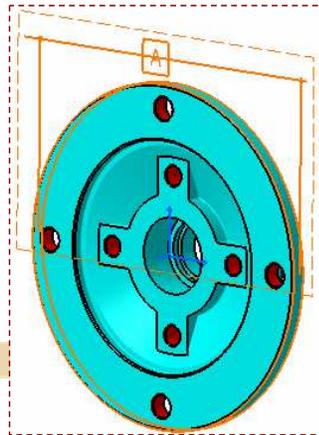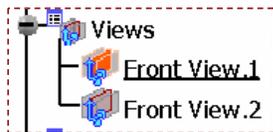
2 **Select "Transfer To
View/Annotation Plane"
command in the contextual menu**

Associated Geometry

Select Views/Annotation Plane

Transfer To View/Annotation Plane
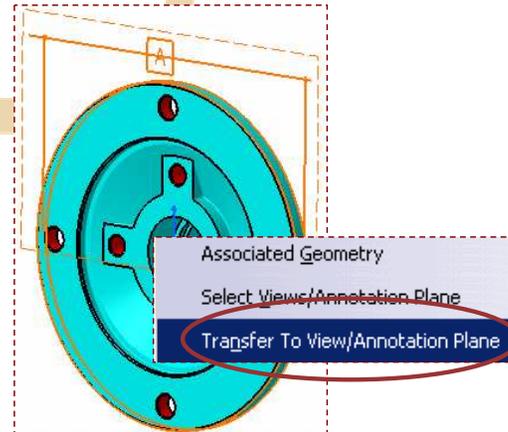
3 **Select another Annotation plane**
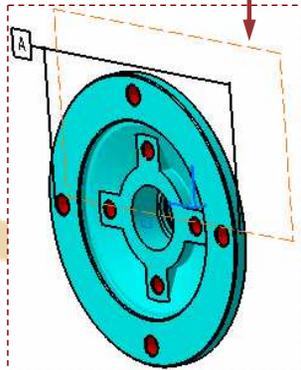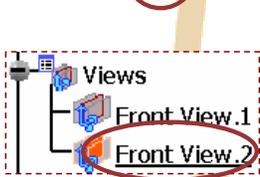
**You will observe that the Annotation
is transferred to "Front View.2" view.**

How to Filter Annotations

You will now learn to filter annotations.

1 **Select the filter Icon**

Visualization
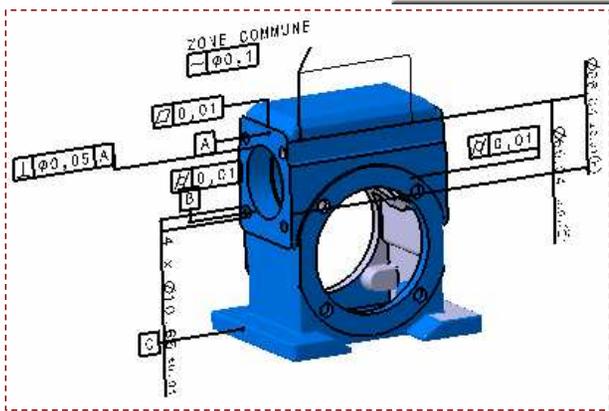
2 **Make your Filter choice.**
In the example, filter is applied on 'Datum'

Filter

Definition
Current filter: Sub-Type: Simple Datum
Filter choice: By sub-type | Simple Datum | 0.01
☐ Refine filter   ☐ Show geometry attachments

Results
Tolerances in the document: 14
Number of selected tolerances: 3

OK   Apply   Cancel

All
None
By sub-type
By type
By feature
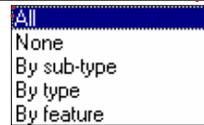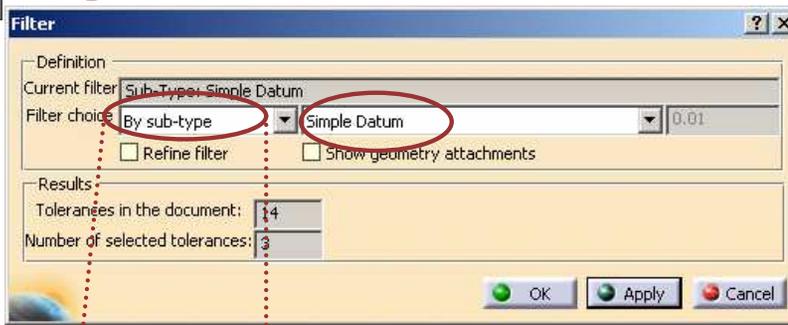
3 **Click OK to validate**
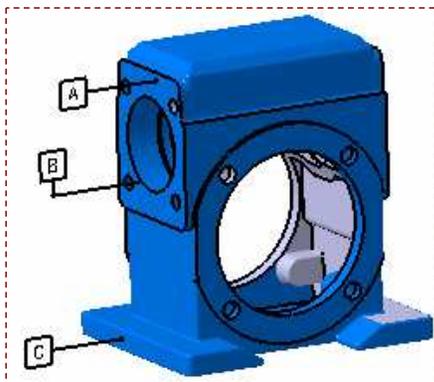
You can filter the display of annotations in the 3D viewer using the following criteria:

- by type (non-semantic)

- by sub-type (text)

- Datums, Datum targets, Geometrical Tolerances by feature or geometrical element, by annotation plane. In the Results window, you can see some indications resulting of applied filter. The default filter is ALL to display all the FT&A annotations.

FT&A objects are accessible by type and attributes using Edit+Search capabilities

Managing Captures

In this skillet you will learn how captures can be helpful in managing parts overloaded with annotations
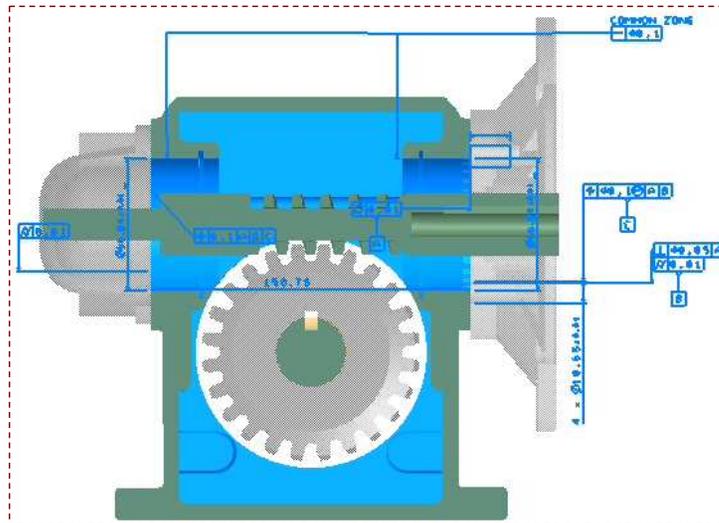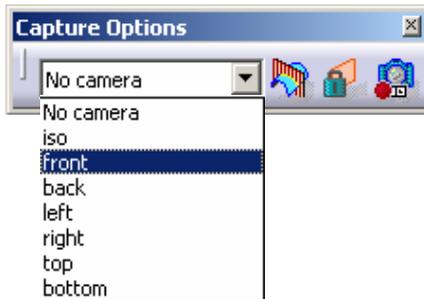
General Process of Creating Captures

Sometimes, models are overloaded with annotations. Captures allow you to create pre-defined views, with only pre-selected annotations

1 **Click on the Capture icon.**

2 **Define the view**

3 **Define the capture properties**

4 **Exit the workbench and display the capture**
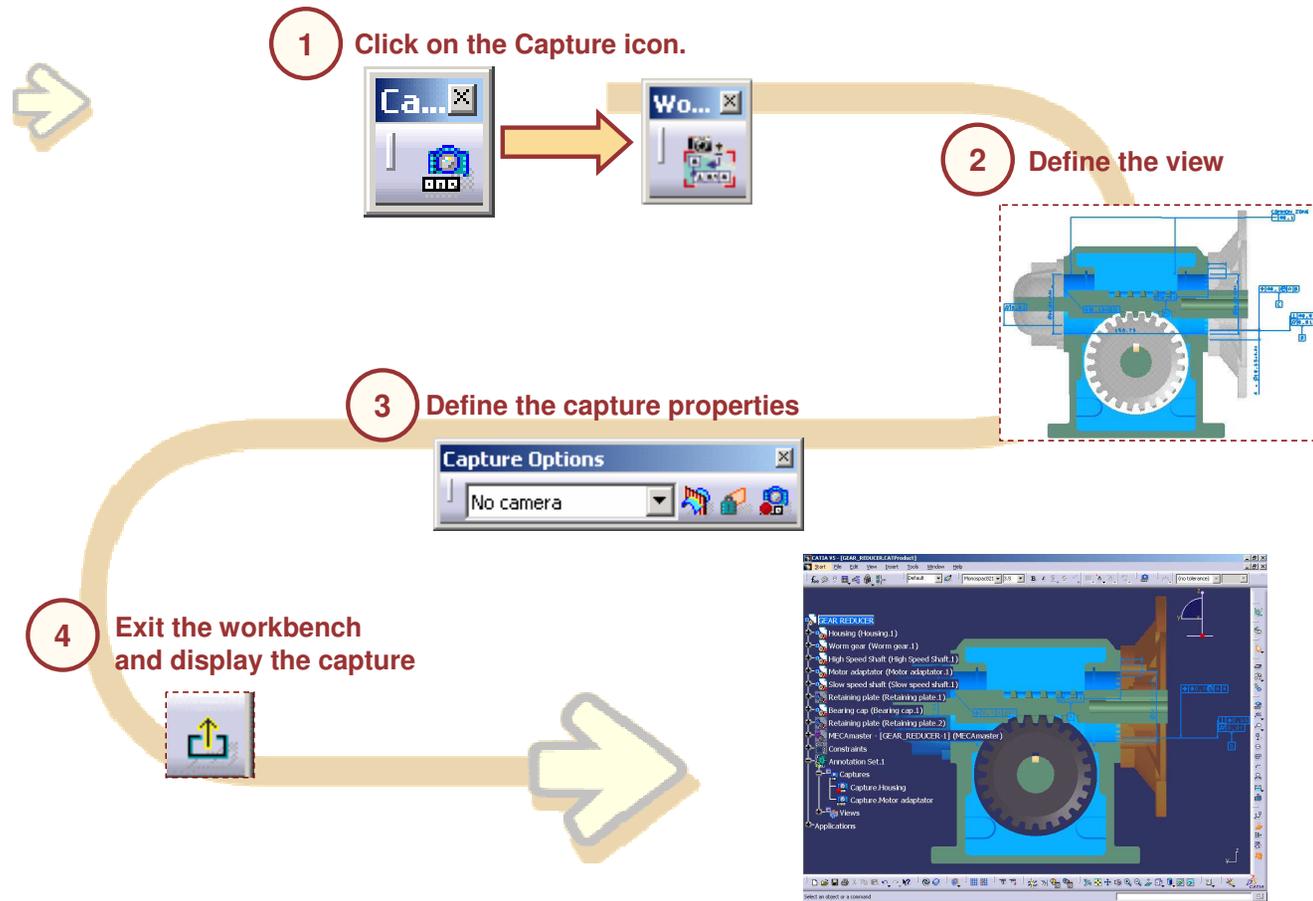
How to Create a Capture (1/3)

You will now learn to create a Capture.

1 **Access the Capture tool**

2 **Specify the Name of capture. By creating a new capture, you automatically access to the Tolerancing Capture Workbench**

Capture Definition ? X
Capture.1
☐ Display in 3D
More>>
● OK ● Cancel
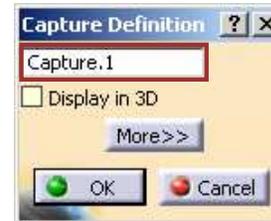
3 **Display the model as you want it to be in the capture. For example, you can display the model normal to an existing annotation plane**
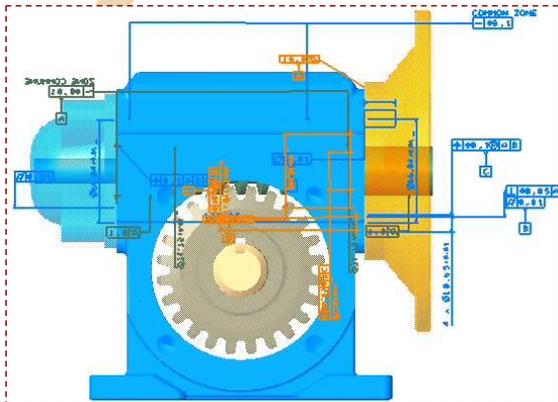
You can also create a Named View to orient your view

The ambiguity due to same Capture names can be avoided. Refer to the User Settings skillet > The Infrastructure Settings

How to Create a Capture (2/3)

(4) **You can now set various capture options**

Activate the associated view when capture is displayed

Select the previously created camera to associate it with the capture
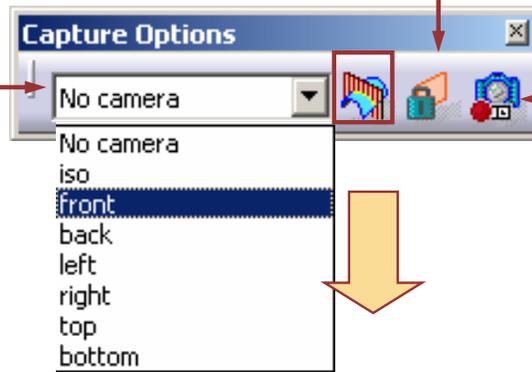
Activating this function will add every new annotations to the capture

Capture Options

No camera

No camera
iso
front
back
left
right
top
bottom
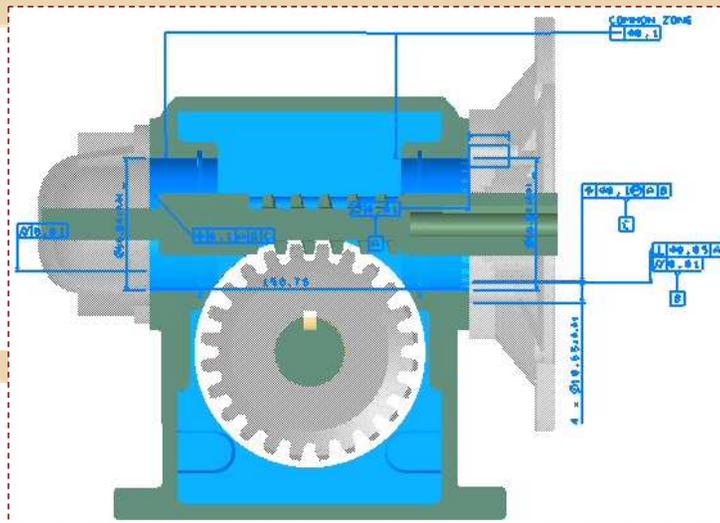
Clipping plane automatically activate the view state

How to Create a Capture (3/3)

5 **Exit the Tolerancing Capture workbench before displaying**

6 **To display a capture, Right-click on it in the specifications tree and choose "Display Capture"**

Annotation Set.1

Captures

Capture.1

Show Profile Card

Life Cycle ▶

Assembly Management ▶

Impact Analysis

Display Capture
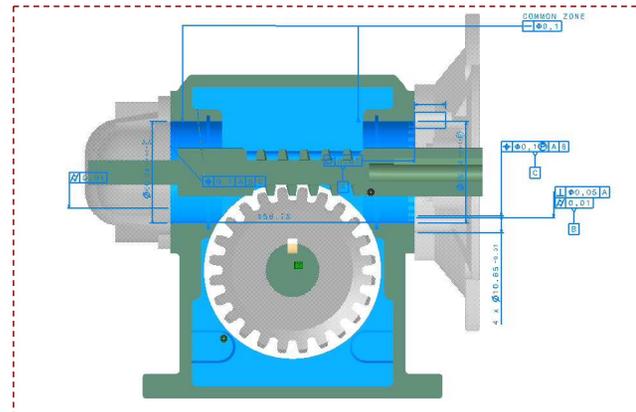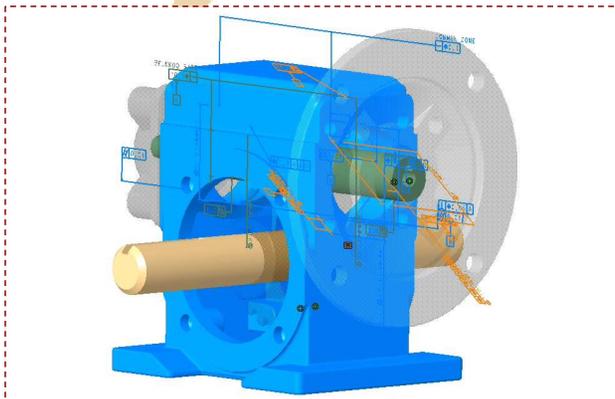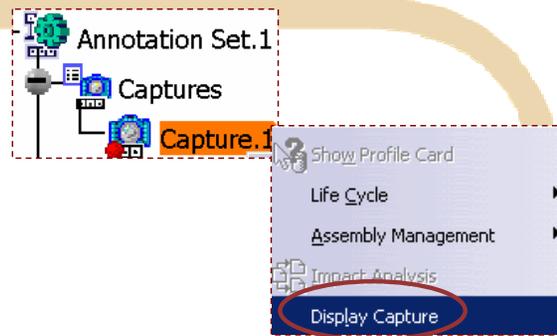
More About Creating Captures (1/2)

Use the annotations filter and hide/show command to select the annotations you want to see in the capture
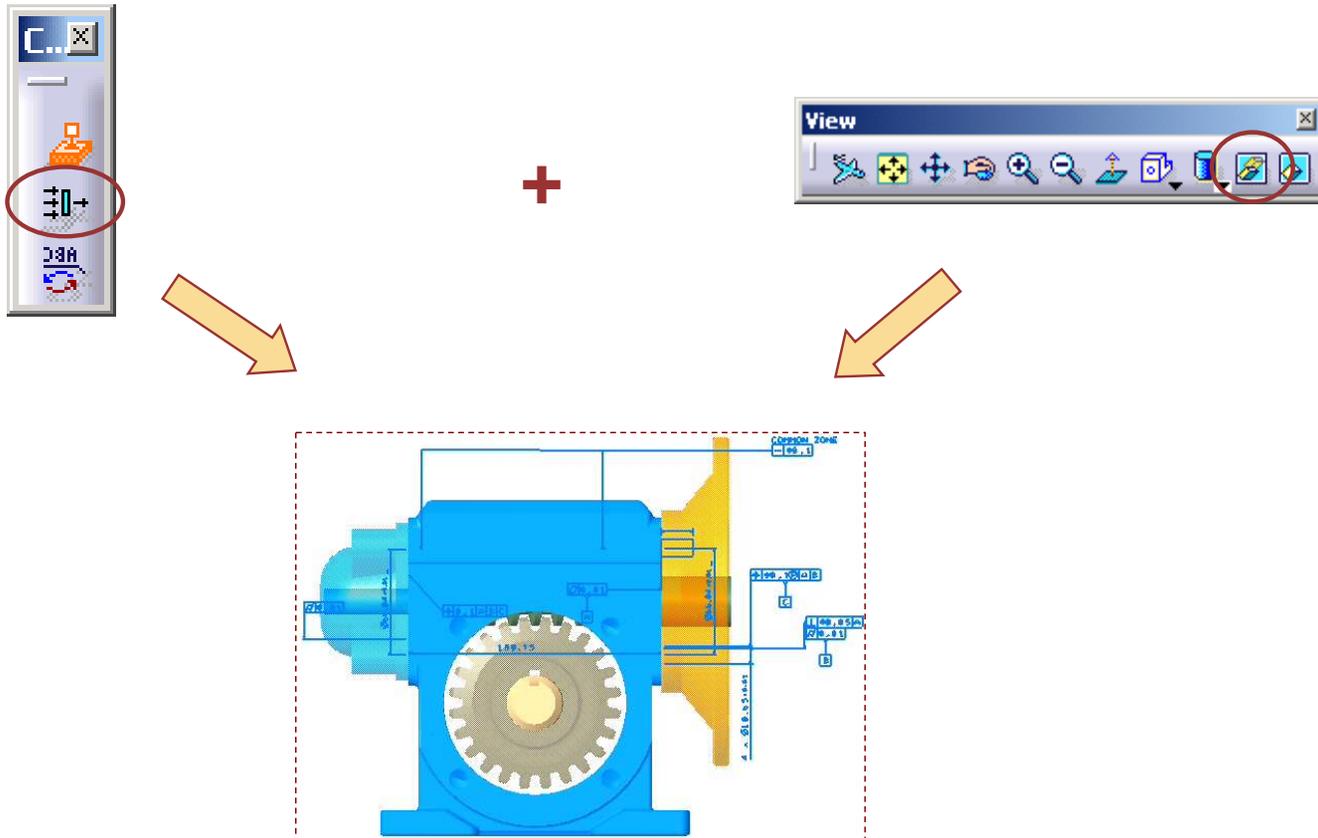
More About Creating Captures (2/2)

When you have more than one capture, it is possible to set any capture as current capture, without editing it. New annotations will be added to it.

Right-click on the capture and select "Set Current". To unset the capture, right-click on it and select "Unset Current"
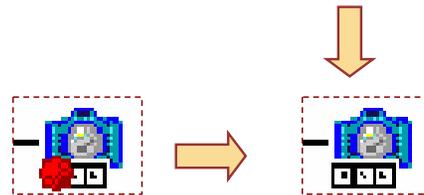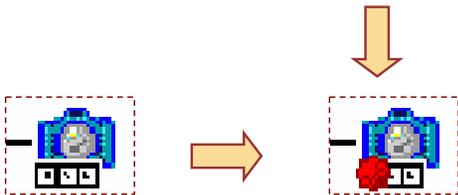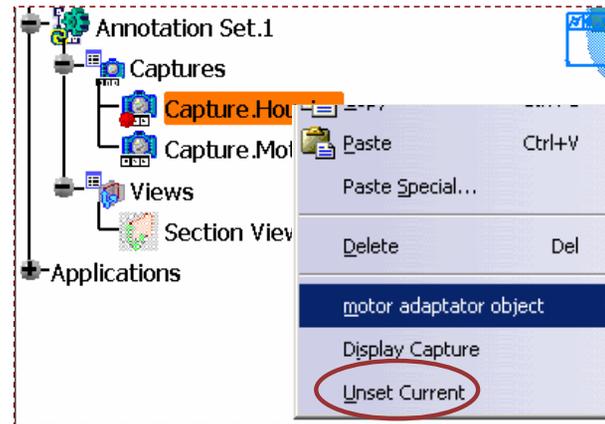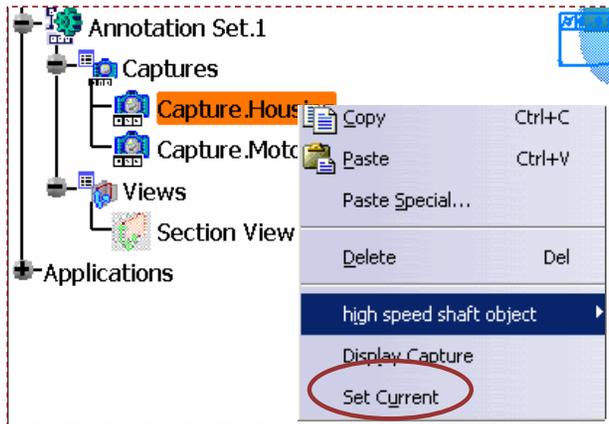
Generating Check Report

In FTA workbench it is possible to generate a report to check whether tolerancing rules are respected or not. These rules depend on standards used.

Click the Report icon

The application generates the report in the browser you use and displays it on screen using the options as specified in the Report Customisation command.

Activate this function to generate a check report.

It indicates which rules are broken

You can generate different settings in order to generate a check report: a set of rules extracted from tolerancing standards is checked for each datum and tolerance specification.

List of Expert Check :

Validity	Hierarchy	Check Name	Percent of Success	Help	Correct Function
		datum label unicity	Not Pertinent	A datum label shall be unique in the tolerancing set	Edit the datum label and replace it with a non-already used label
		datum label capital letter	Not Pertinent	A datum label shall only contain capital letter(s) (upper case)	Edit and Replace the datum-label small letter(s) with the corresponding capital letter (s)
		authorized or proposed datum label letter	Not Pertinent	A datum label should not contain the capital letters I, O or Q. according to ASME Y14.5M-1994 standard.	Edit and Replace the datum label
		identical datum label letter	Not Pertinent	A datum label shall be composed with the same letter repeatingly, when necessary (ISO standard)	Edit the datum label and check the character repetition
✔		datum label length	100%	A datum label shall not be composed with more than two letters (ASME standard)	Edit and check the datum label length
		datum target label ended by a positive number	Not Pertinent	A datum target label shall be ended by a positive number	Edit and check the datum target label numbering
		datum target label referencing to an existing datum label	Not Pertinent	A datum target label shall reference to an existing datum label	Edit the datum target label and check the datum label reference
		sequential datum target numbers	Not Pertinent	The numbers identifying datum targets shall be sequential and begin with 1	Check the datum target label sequence beginning with 1
		consistence between flatness specification and specified geometry	Not Pertinent	The Flatness specfication shall be applied to surfaces of the planar class of surface	Specify a correct form tolerance
		Modifiers on Toleranced Element	Not Pertinent	- For any form specification, the Free State Symbol may be applied and specified alone. - For any Form specification, such as linear profile, planar and position specifications, the Maximum/Least Material Condition (M/LMC) may be applied and specified alone. - For any Orientation, Location and Runout specification, several modifiers are allowed on the tolerance zone with following restrictions: * first, MMC or LMC or S (ASME standard only) conditions * second, the free state symbol, if necessary * third, the projected tolerance zone symbol, if necessary * fourth, the statistical tolerancing symbol, if necessary - Only MMC or LMC or S (ASME standard only) conditions shall be specified on datum elements	Edit the GDT and adjust the modifier(s) specification

To Sum Up

You have seen examples of collaborative work with CATIA knowledge tools:

- **How to create various annotation planes**
- **How to create and manage annotations**
- **How to create captures in case of document loaded with annotations**
- **How to generate a report**

Geometry for 3D Annotations

In this lesson you will learn how to add new geometries, replace geometries and reconnect their tolerances.

- **Creating Constructed Geometry**
- **Geometry Connection Management**
- **To Sum Up**

Creating Constructed Geometry

In this skillet you will learn to construct geometry (such as center point, axis, median plane, gage plane, etc) often used to define the theoretical dimensions of parts or products.

What is Constructed Geometry for 3D Annotations

Constructed geometry (such as center point, axis, median plane etc) is used in order to define the theoretical dimensions of parts or products (framed dimensioning).

These constructed elements represent the tolerancing feature and are used to define the tolerance zone position of:

- **Geometrical tolerances,**
- **Related position of the Datums of a Datum reference frame,**
- **Size and position of a partial surface**
- **or a datum target.**

The capability allows either to manage constructed geometry that has been manually created by the user.

The existing geometry is the represented geometry, the constructed geometry is the representing geometry.
It is possible to automatically construct geometrical elements like points, Axis, Plane,Cylinders, Sphere.
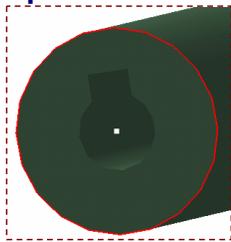
Points & Axis	Circles	Cylinders
For a circle, the constructed geometry is its center point	**For a cone, the constructed geometry can be a circle (and a plane)**	**For a circular pattern, the constructed geometry is a cylinder**
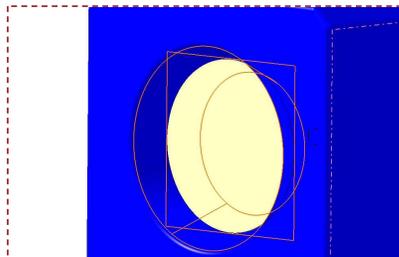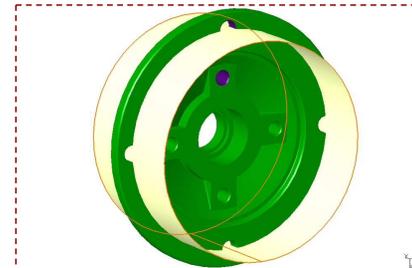
How to Create a Constructed Geometry (1/3)

Following process explains how to create Constructed Geometry for one selected element

1 Select a geometrical element.

2 Use the "Constructed Geometry Creation" function.

Geometry ...

3 Check the geometrical element as Axis in the creation box and Click OK.

Constructed Geomet...
Axis :
OK   Cancel

For a cylinder, the constructed geometry is its axis
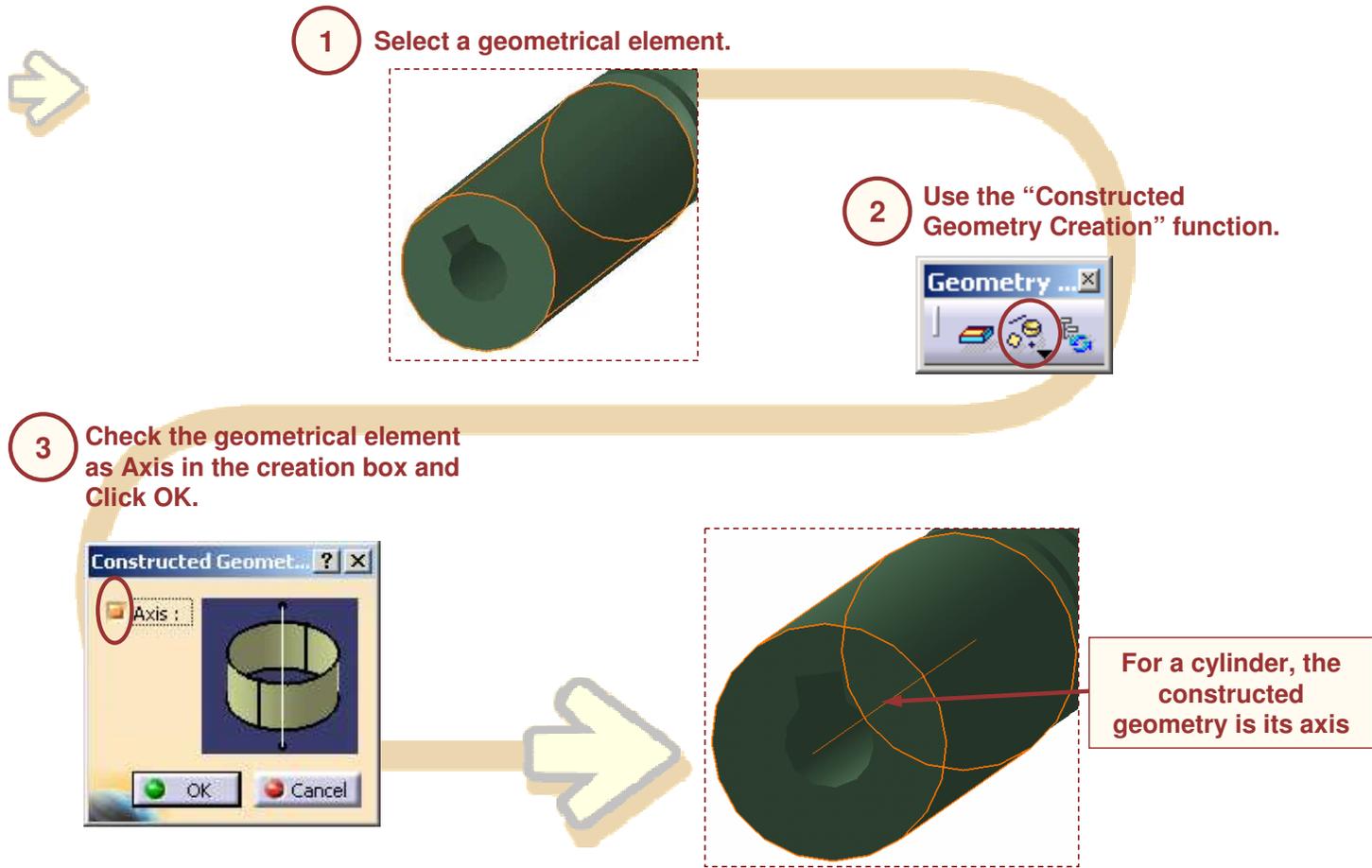
Copyright DASSAULT SYSTEMES

How to Create a Constructed Geometry (2/3)

Following process explains how to create Constructed Geometry for several elements

1 Use the "Constructed Geometry Creation" function

Geometry ...

2 Select the first element

3 Select the second element (and following...) WITHOUT Using Ctrl Key

For two selected planes, the constructed geometry is a median plane

4 Check the geometrical element creation box and Click OK.

Constructed Geometr...

Plane :

OK   Cancel
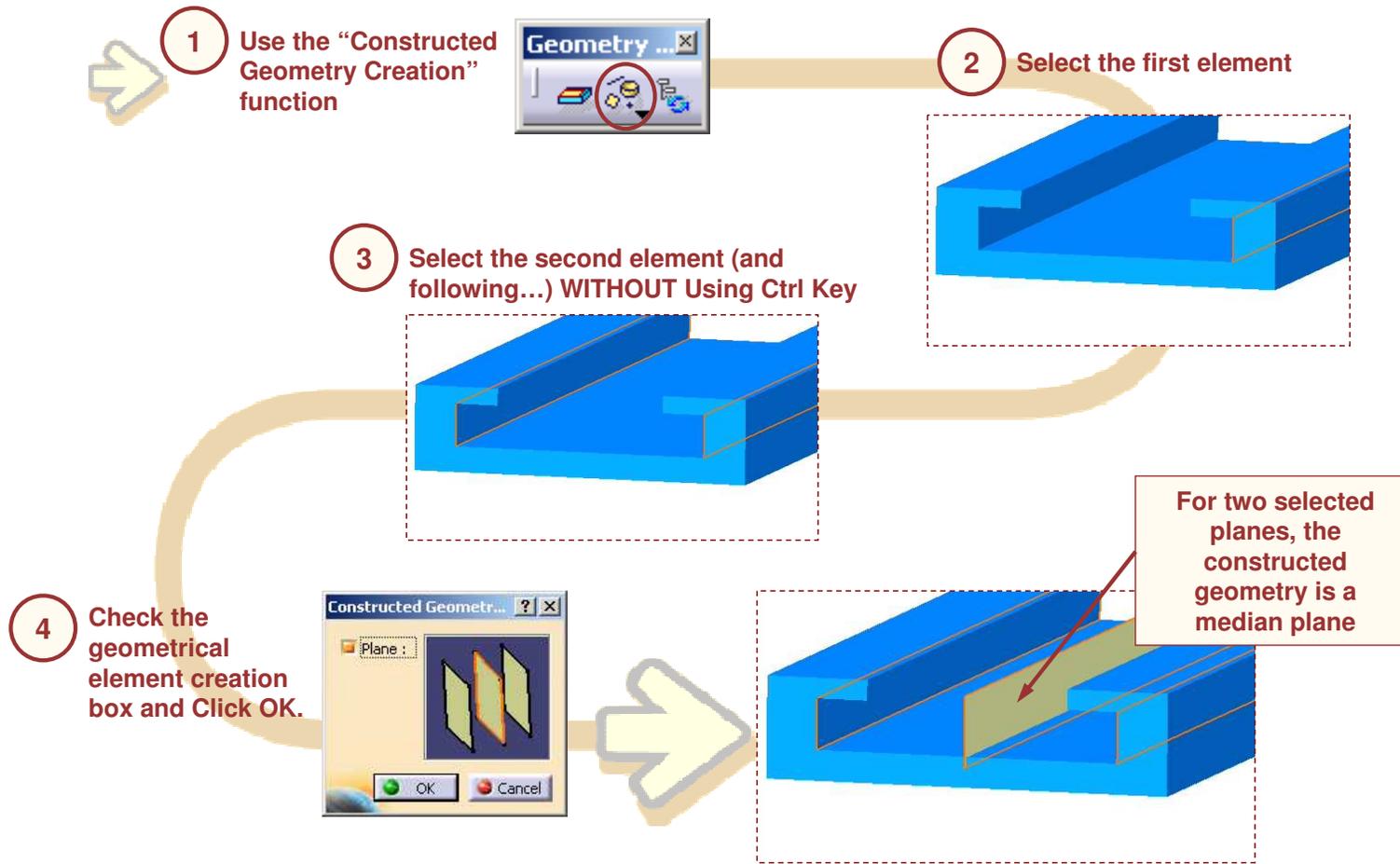
How to Create a Constructed Geometry (3/3)

The constructed geometry is placed in a specific node of the annotation set.

This geometry can also be deleted with the constructed geometry creation function

PartBody
Geometrical Set.1
Annotation Set.1
Views
Datums
Reference Frames
Geometrical Tolerances
Notes
Partial Areas
Restricted Area.1
Construction geometries
Angle Plane.1
Cylinder Axis.1

Geometry ....

Constructed Geomet...
Axis :
OK   Cancel
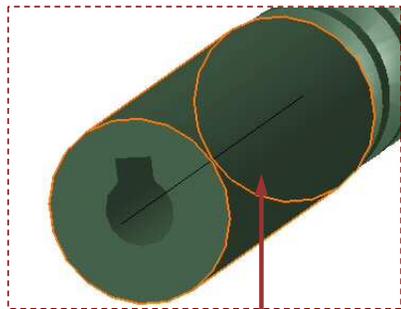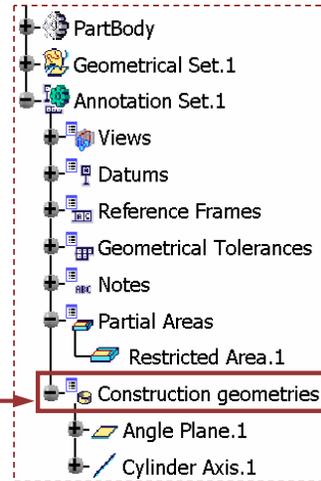
Selected element
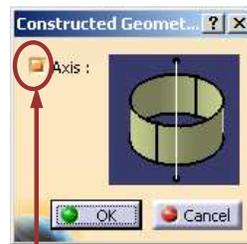
Uncheck the Axis box

The constructed geometry is automatically removed

The constructed geometry is useful to create Framed dimensions

Geometry Connection Management

In this skillet you will learn how to add new Geometries, replace Geometries and re-establish their tolerances.
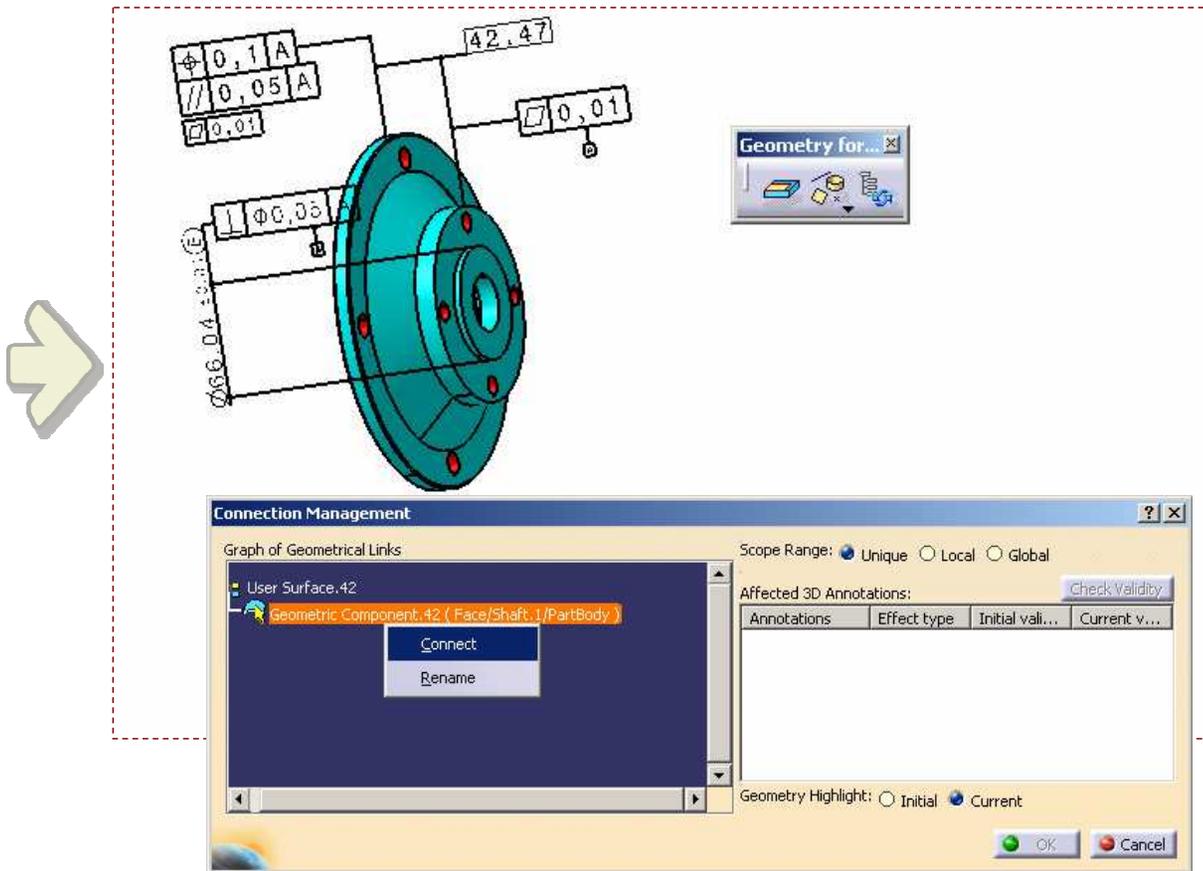
What is Geometry Connection Management (1/2)

Manage annotation connection offers to create, delete, modify or rename geometrical elements or user surfaces of an existing annotation.

Using Geometry connection Management it is possible to connect a new feature in the existing group of elements to be toleranced.

Also, using Geometry connection Management you can replace a feature from a group with a new feature.

Some geometrical modifications don't need any user intervention for the annotations to be update (like translation, distance or diameter value modification…)

Element definition modification

Distances values and diameter modification

Automatic update
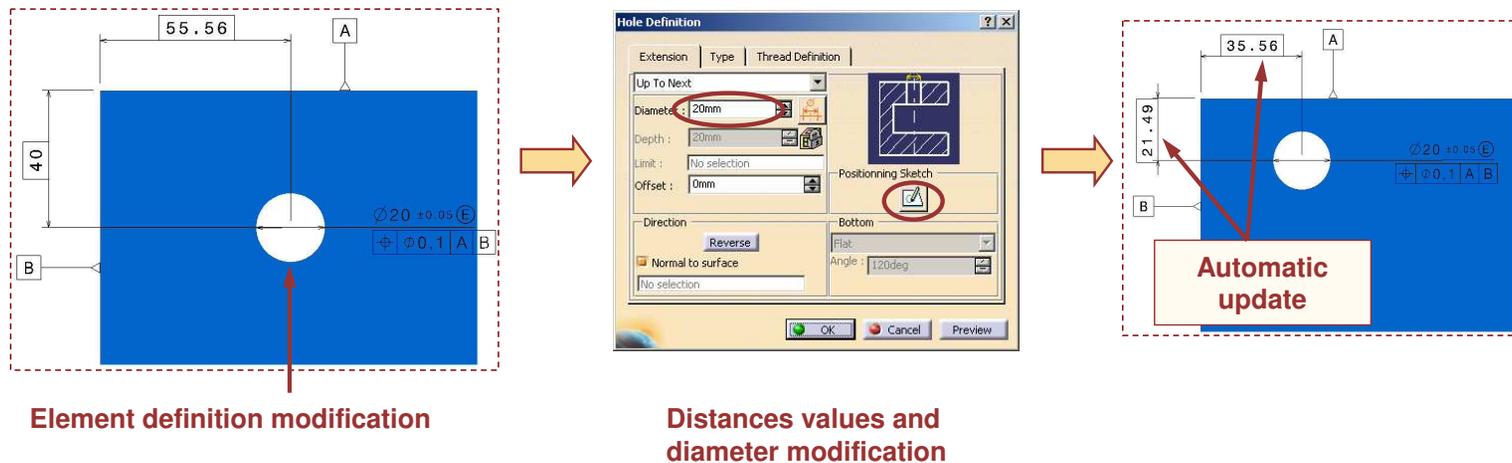
What is Geometry Connection Management (2/2)

Use the tool shown below to connect new geometry to an annotation. In case of major geometrical modifications (adding a new element, replacing a hole by a cone…)

Tool

Modify the Geometry by modifying the hole
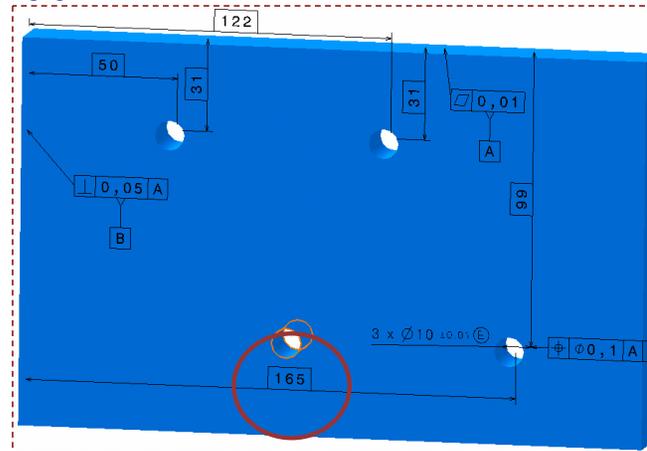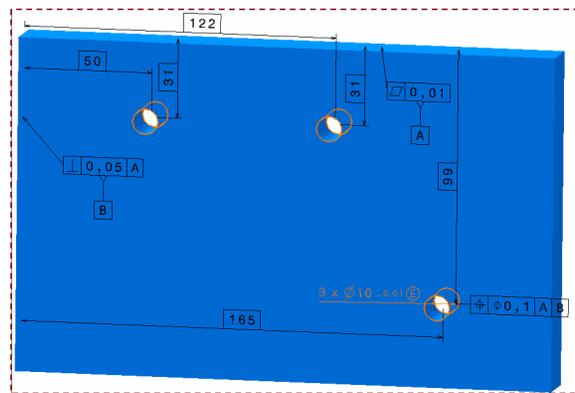
To include the new hole in the pattern annotations, we use the Geometry Connection Management Function

Accessing Geometry Connection Management

You can also Access Geometry Connection Management using contextual menu. Right-click on an desired annotation and select Associated Geometry >Geometry Connection Management
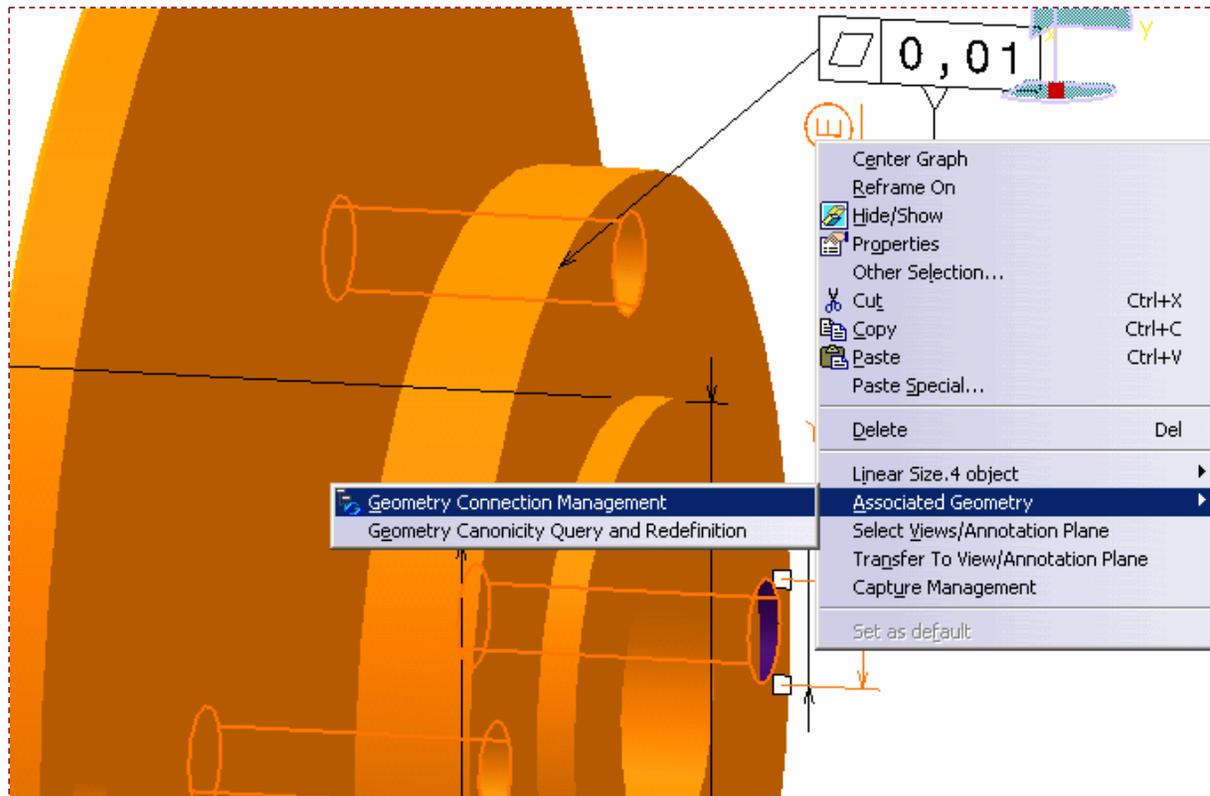
How to Connect a New Geometry to an Annotation (1/2)

Here we will learn how to connect a new geometry to an annotation

1 **Select the tolerance(s) to modify**
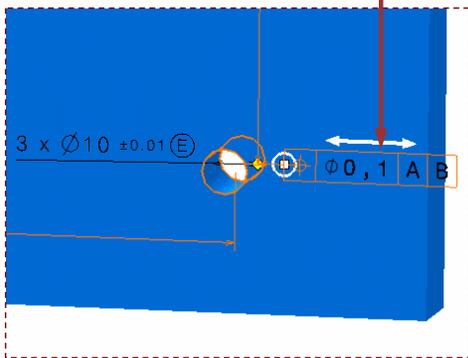
2 **Select the geometry connection management function**

3 **Right-click on "Group of Surface" and choose "Add Component"**
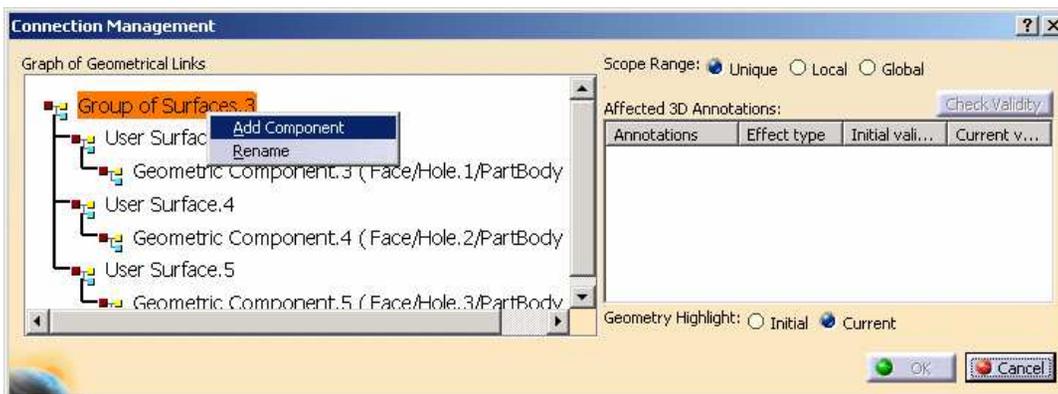
How to Connect a New Geometry to an Annotation (2/2)

4 **Select the geometrical element to add to the position tolerance**

3 x Ø10 ±0.01 Ⓔ

165
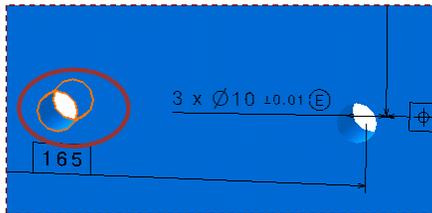
5 **Check Validity, and if it is OK, validate the annotation modification**
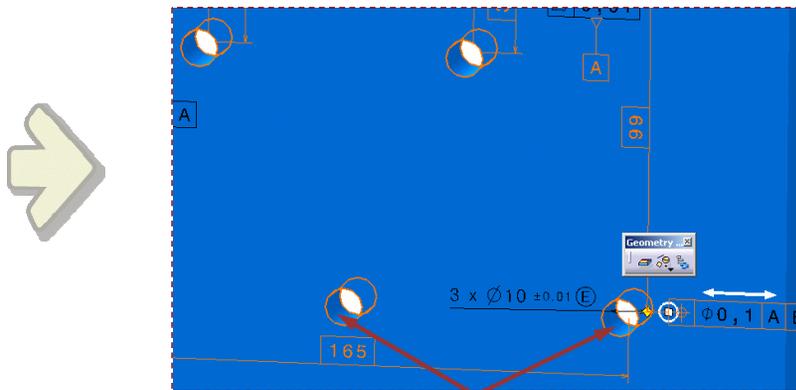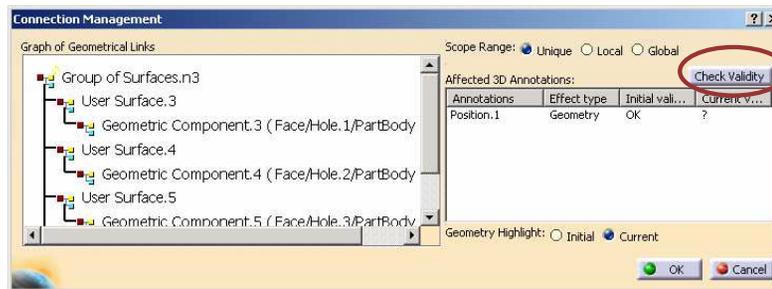
The position tolerance is now also connected to the new hole

How to Replace a Connected Element by a New One

Geometry connection management allows you to replace a feature by a new one

1 **Select the upper face or one of the annotations connected to it and select the Geometry Connection Management function**

80

⌀ 0,05 A

0,01

0,01

A

+
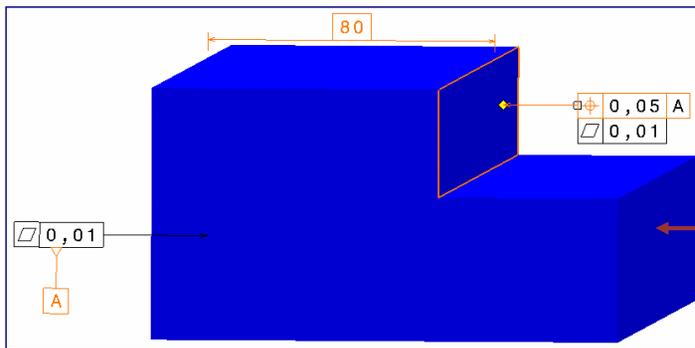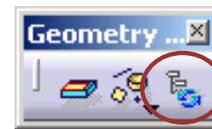
We want the annotations not to be connected to the upper right face but to the bottom right face

2 **Right-click on "Geometric Component" and select "Connect", then select the inferior face**

3 **Check validity and click OK to modify the connection**

Connection Management

Graph of Geometrical Links

■ User Surface.2
    ■ Geometric Component.2 (Face/Pad.1/F
        Connect
        Rename

130

0,01

A

⌀ 0,05 A

0,01
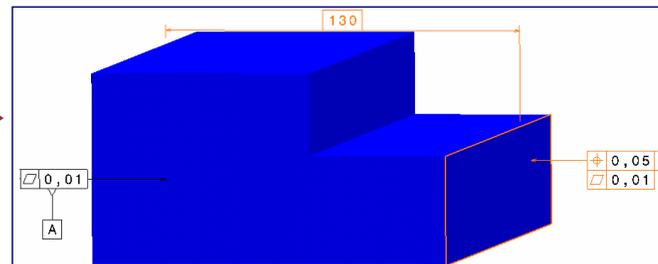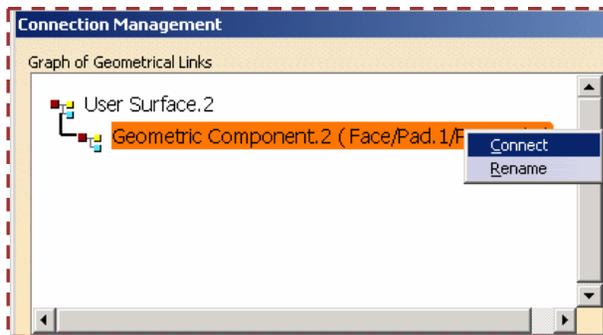
Using Scope Range in Connection Management (1/3)

Scope Range in the Connection Management tool gives information on Tolerances and Annotations which will be affected after transferring a particular tolerance i.e it gives an idea whether Annotations (which depend on Tolerance to be transferred) will be successfully transferred or not on to the new Geometry. Thus, it also checks the validity of the reconnected elements.

There are three options in the scope range:

- **Unique: Only the selected tolerance will move when it is transferred from one geometry to another.**
- **Local: Every tolerance connected to the feature on which it was initially placed will be transferred.**
- **Global: Every tolerance that are directly connected or indirectly applied to the feature on which the selected tolerance will be transferred**
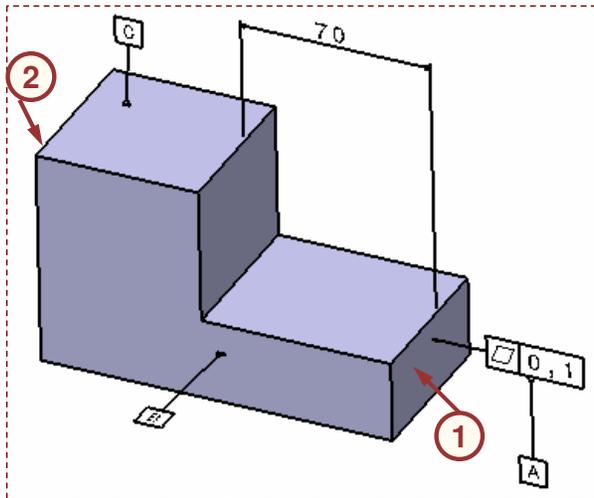
The Flatness tolerance is connected to face '1' as shown. Now you want to transfer it to face '2' using 'Geometry connection Management' tool. You will use three options in 'scope range' and study effect in each case.

Using Scope Range in Connection Management (2/3)

- **First Select 'Flatness' Tolerance**
- **Access 'Geometry Connection Management'**
- **Select the New Face where the tolerance should be transferred**
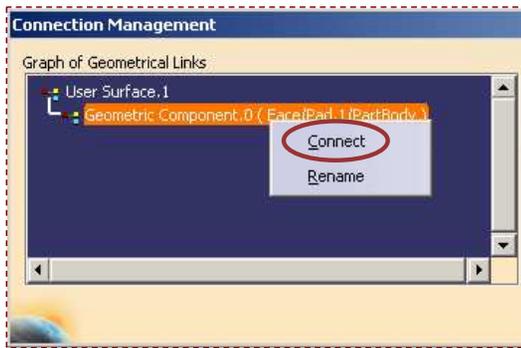
Select this face
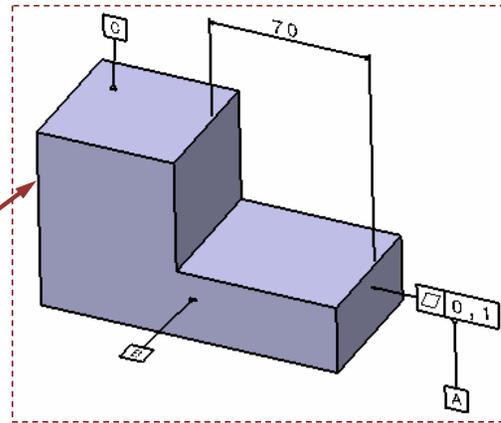
When Using 'Unique' Scope Range: ONLY Flatness tolerance is transferred on to the new face
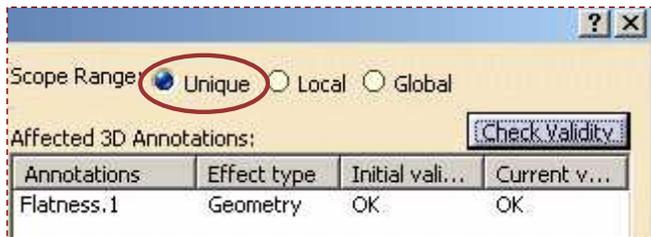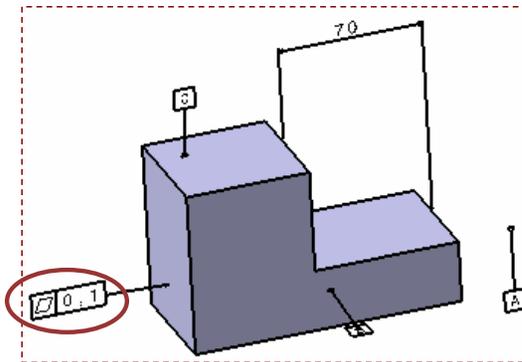
Observe that ONLY selected tolerance is transferred

All Validity checks are passed

Using Scope Range in Connection Management (3/3)

When Using 'Local' Scope Range: Every tolerance connected to the feature on which it was initially placed will be transferred. In this case, Flatness tolerance along with Datum 'A' & Datum Reference Frame will be transferred.
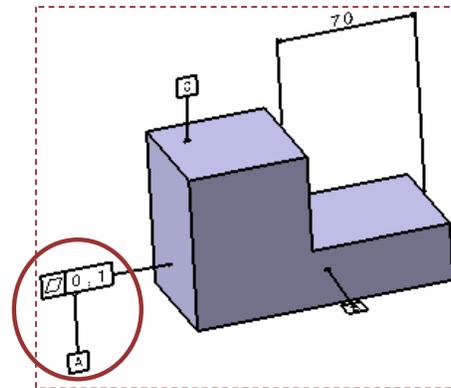
All Validity checks are passed

When Using 'Global' Scope Range: Every tolerance that are directly connected or indirectly applied to the feature on which the selected tolerance will be transferred.In this case, In this case, Flatness tolerance along with Datum 'A' & Datum Reference Frame and the Linear Dimension will be transferred.
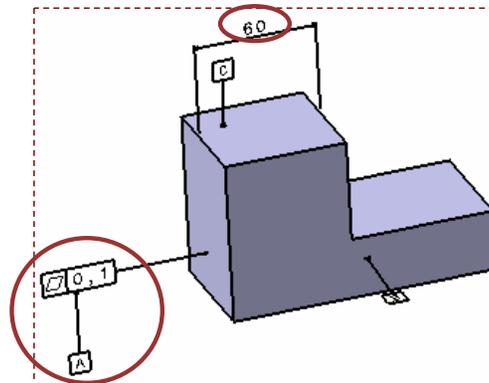
All Validity checks are passed

To Sum Up

In this course you have seen :

- **Constructed Geometry creation**
- **Geometry Connection Management**

Advanced Functions

In this lesson you will learn Advanced functions to create Restricted Areas, Datum targets.
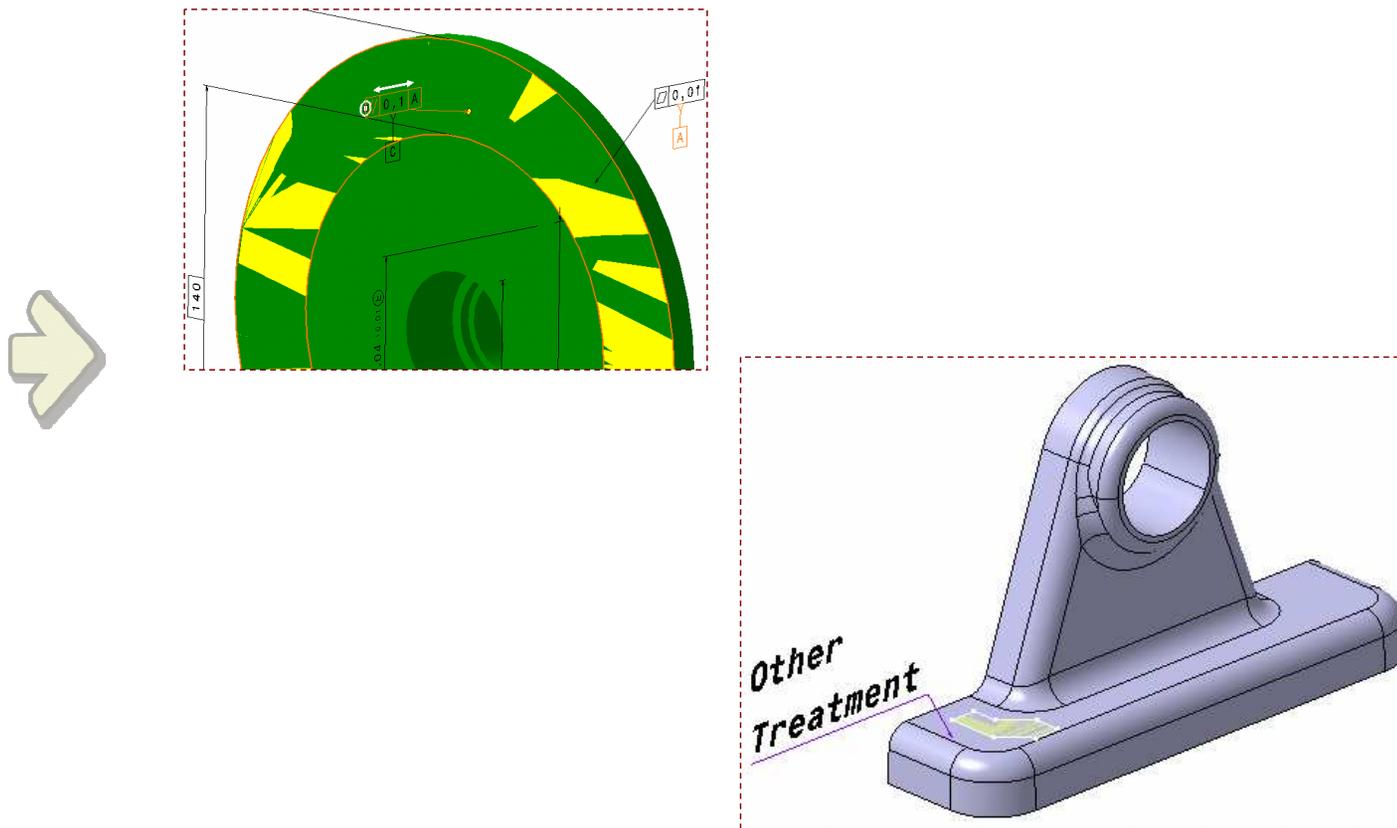
What are Restricted Areas

Restricted Areas

When the interface between two elements (or more) is done partially, the restricted area has to be used to specify the kind of contact (surface, circle, line,…) and tolerancing in a way to ensure that the function realized

Example:
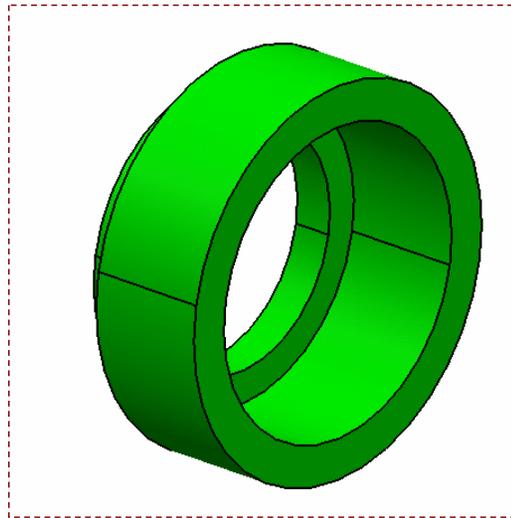
Part 1
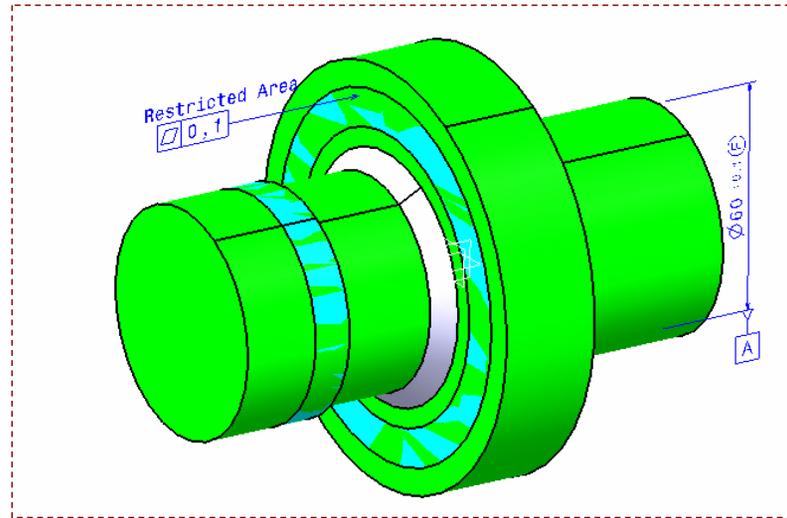
Part 2

How to Create Restricted Areas (1/2)

Here you will learn how to create Restricted Areas.

1 **Select "Restricted Area" function**
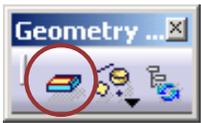
2 **Select the Restricted and the Restricting Area**

The surface to be used should be already created in Generative Shape Design workbench
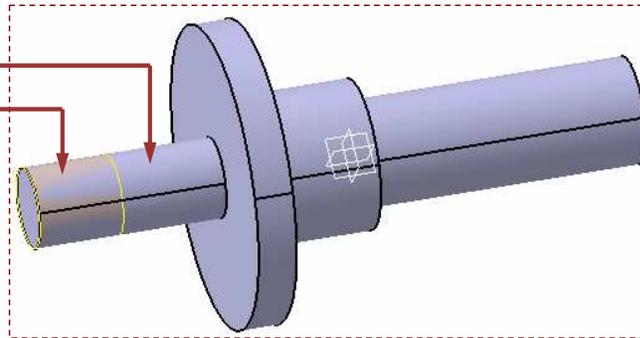
3 **Select Dimension icon. Create dimension between the edge of restricted area and edge shown.**
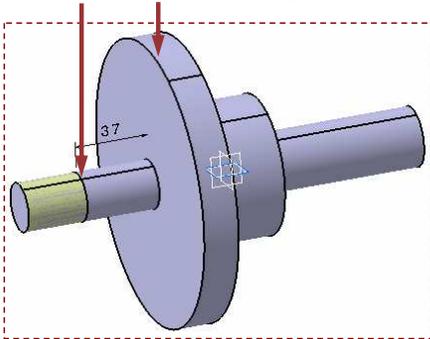
4 **Create Basic Framed Dimension between the edge of restricted area and edge shown**
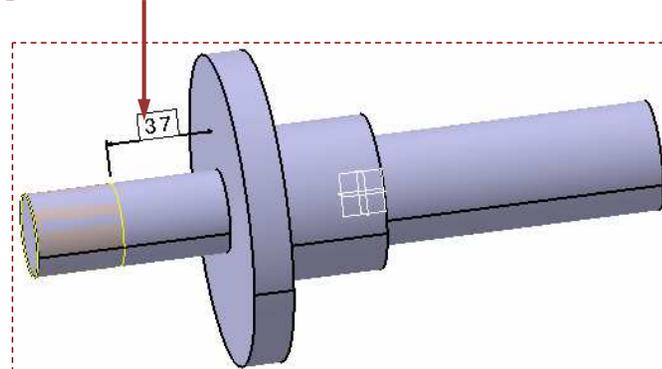
How to Create Restricted Areas (2/2)

5 Select "Tolerancing Advisor" command

Select these two faces
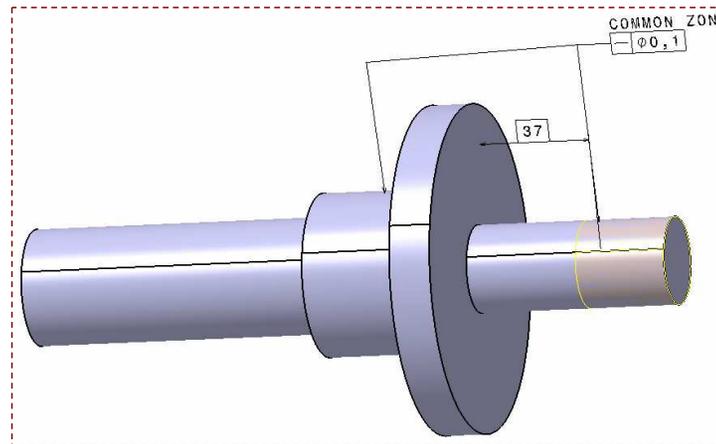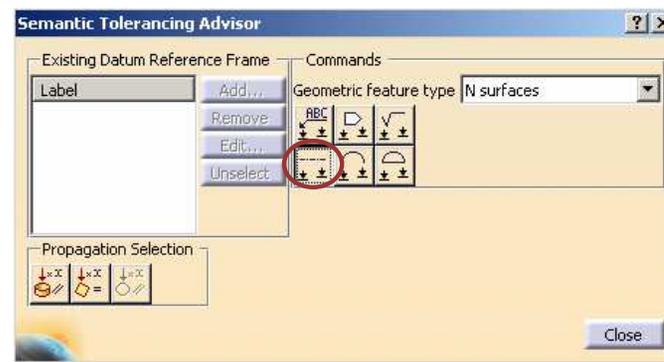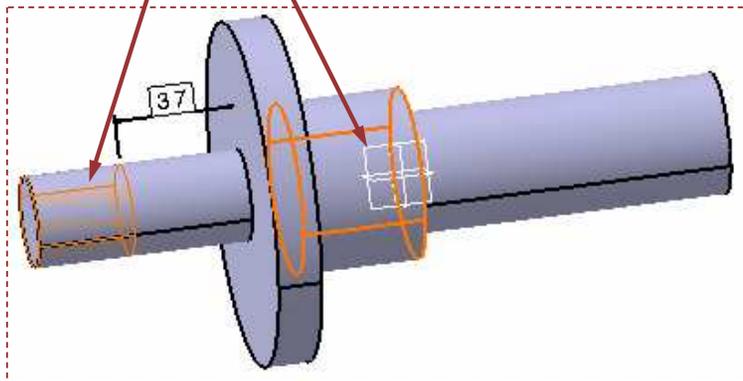
How to Create Datum Targets (1/2)

Here you will learn how to create Datum Targets

1 **Create Points on surface (with Generative Shape Design)**
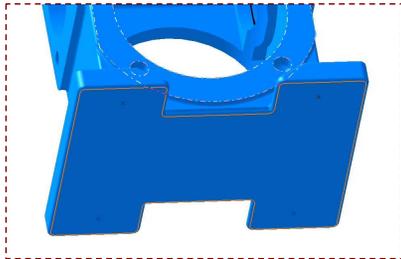
2 **Switch to FT&A workbench and create a semantic datum on the surface**

Semantic Tolerancing Advisor

Existing Datum Reference Frame — Commands
Label — Geometric feature type | One surface
A
B
A|B
A|B|C

Propagation Selection

Close

3 **Select "Add" in datum definition dialog box and choose the target type**

Datum Target

☐ Automatic naming
Label | D1
● Point only
○ Circular
○ Rectangular
0mm X 0mm

OK ● Cancel

4 **Select the points supporting the targets**

D1
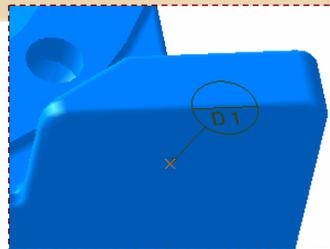
5 **Validate Datum Targets creation**

Datum Definition

Datum Label | D
Datum Targets

Label	Alias
D1	Datum Target.1
D2	Datum Target.2
D3	Datum Target.3
D4	Datum Target.4

Add...
Remove
Edit...
Unselect

Datum Reference Frame
☐ Create a DRF with the last created datum

OK ● Cancel

How to Create Datum Targets (2/2)

Datum Target Types
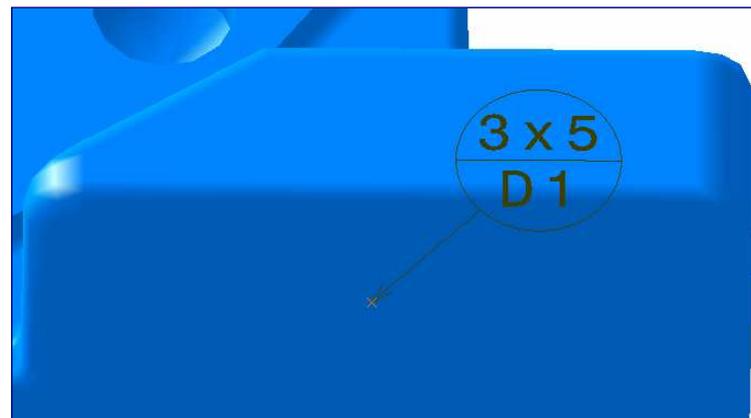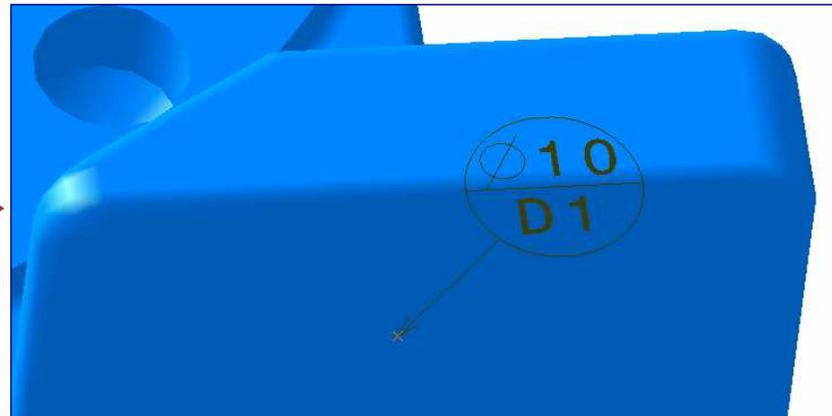
Creating Thread Representations

You can represent threads for a better understanding of the model.

You can represent all threaded elements of a model or only selected ones.

Select he thread icon

To create every thread representations of a model, check "All threads" command
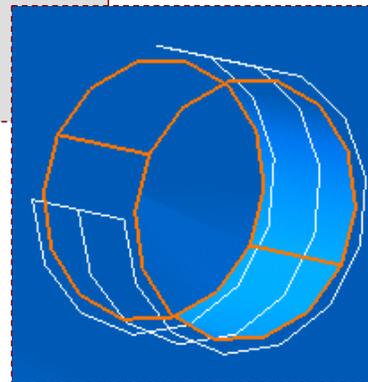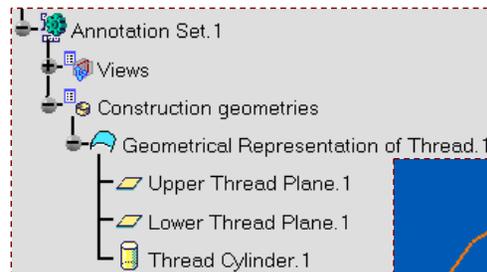
Only usable if you have threaded the hole in Part Design

How to Create Thread Representation

Here you will see how to define annotations associated to a selected thread.

1 **Select the median 3/4 circle arc which symbolizes the thread helical surface**
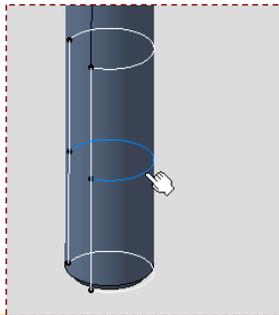
2 **Create a semantic thread dimension**
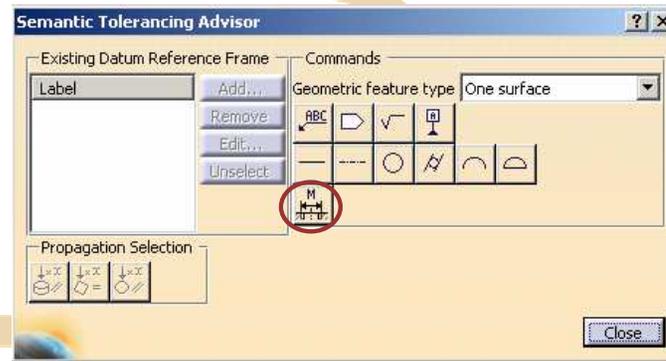
3 **Select both the Pitch and the Tolerance class options and Click OK.**

M20 x 1.5 - 6g
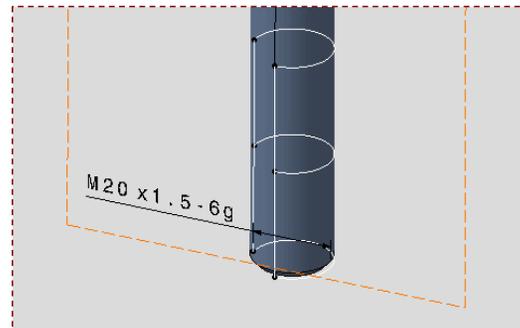
How to Create Knowledge Formulas on Tolerances (1/3)

In FTA it is possible to manage tolerance values by using formulas and knowledgeware elements or tools or workbench.

1   **Double-click to edit the Tolerance**
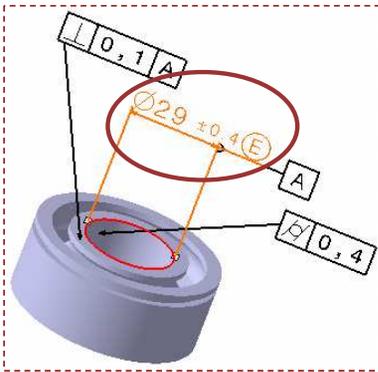
2   **Type the tolerance value for the Upper Limit and Click OK.**
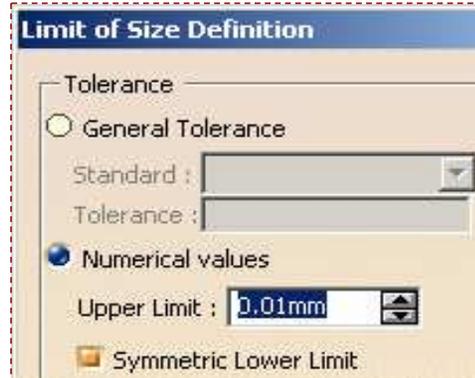
3   **Now, Double-click to edit the cylindricity Tolerance**
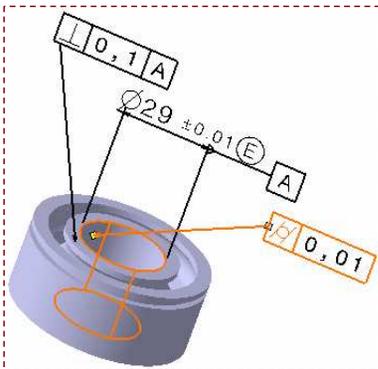
4   **Now, Right-click to add a formula. Link it with the upper limit values for the diameter**
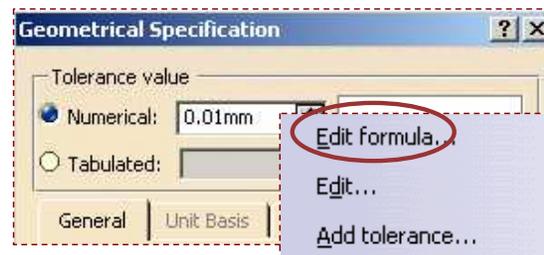
How to Create Knowledge Formulas on Tolerances (2/3)

5 **Now, apply the formula as shown.**
The cylindricity tolerance will change whenever Upper Limit of the Linear Size is changed.
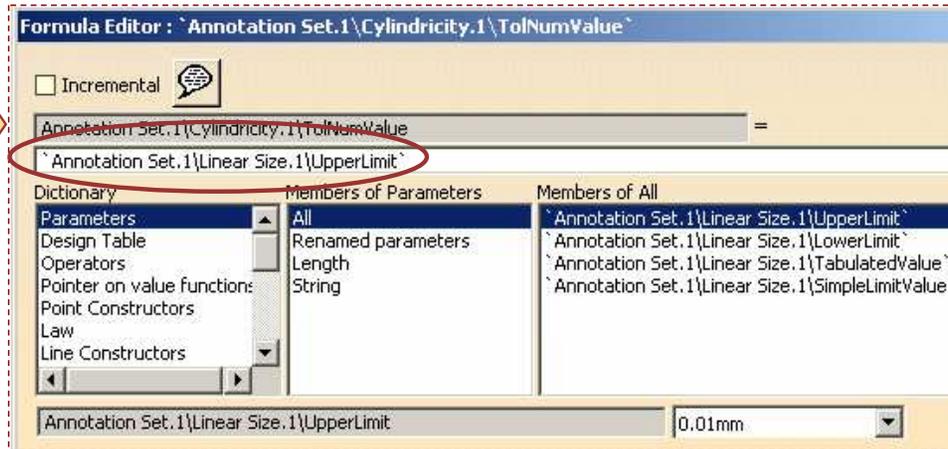
6 **Now, change the Upper Limit from 0.01 to 0.3**
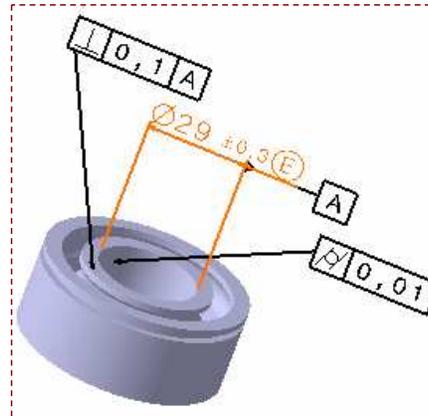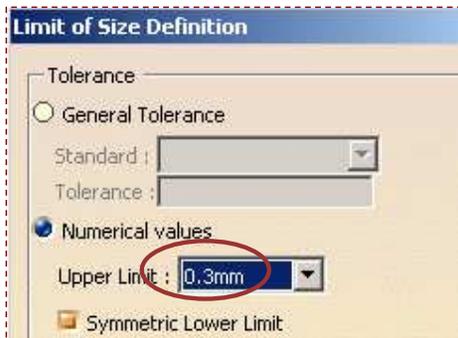
How to Create Knowledge Formulas on Tolerances (3/3)

7   **Observe that the value for cylindricity changes as formula is applied. Update the part.**
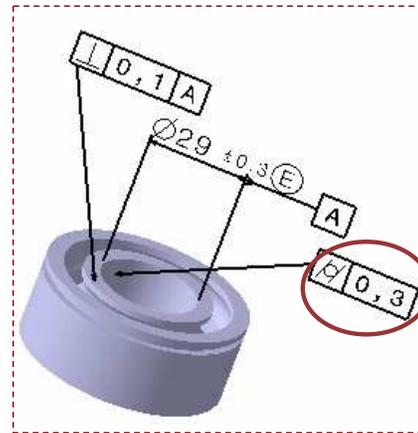
To see the 'Parameters and relations' node in the tree, go to Tools > options > Infrastructure > Part Infrastructure > Display tab
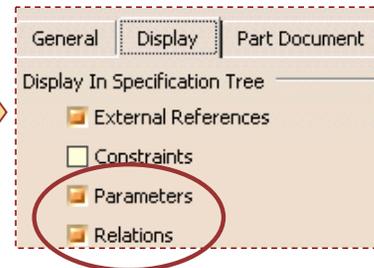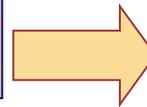
Student Notes:

To Sum Up

In this Lesson you have seen:

- **How to create Restricted Areas**
- **How to create a Datum Targets**
- **How to Represent Threads**
- **How to drive Tolerance values using formulas**

The User Settings

In this lesson you will learn some common settings to work efficiently with FTA workbench

- **FTA Settings - Tolerancing**
- **FTA Settings - Display**
- **FTA Settings - Constructed Geometry**
- **FTA Settings - Manipulators**
- **FTA Settings - Dimension**
- **FTA Settings - Annotation**
- **FTA Settings - Tolerances**
- **FTA Settings - View/Annotation Plane**
- **The Infrastructure Settings**
- **Generative Drafting Settings**
- **Co-ordinate Dimension Display Settings**
- **To Sum Up**

The User Settings

In this lesson you will learn various FTA Settings and effectively use them to suit to your working style.
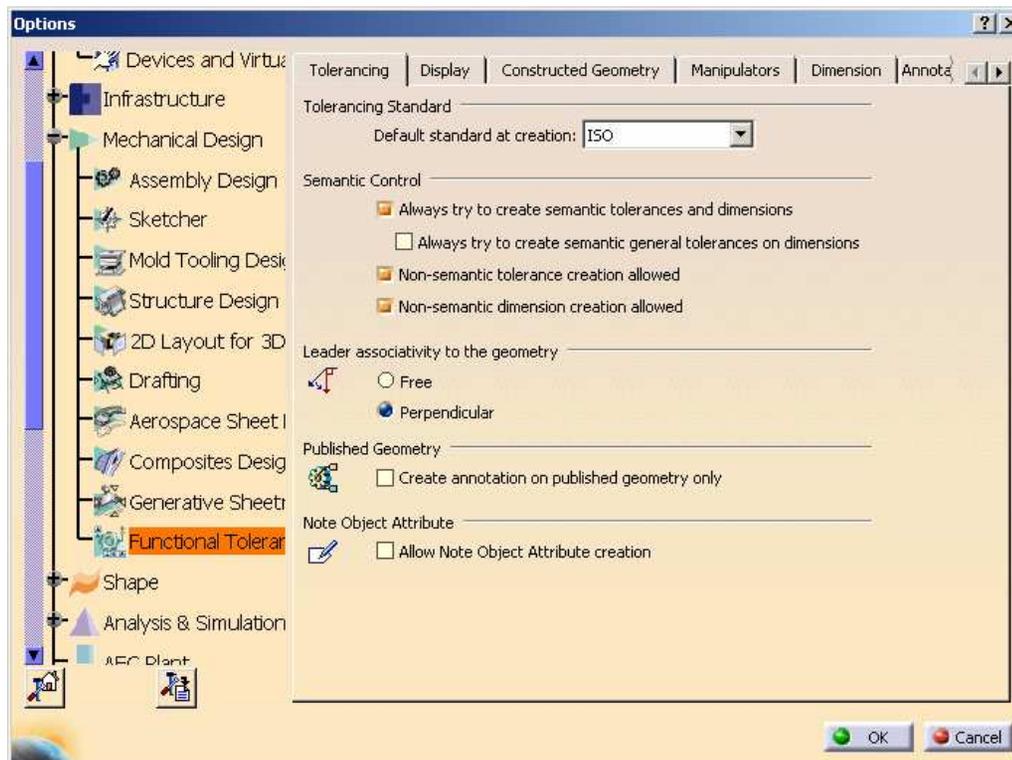
FTA Settings - Tolerancing

Access Tools > Options > Mechanical Design > Functional Tolerancing & Annotation > Tolerancing

| Tolerancing | Display | Constructed Geometry | Manipulators | Dimension | Annota |

Tolerancing Standard
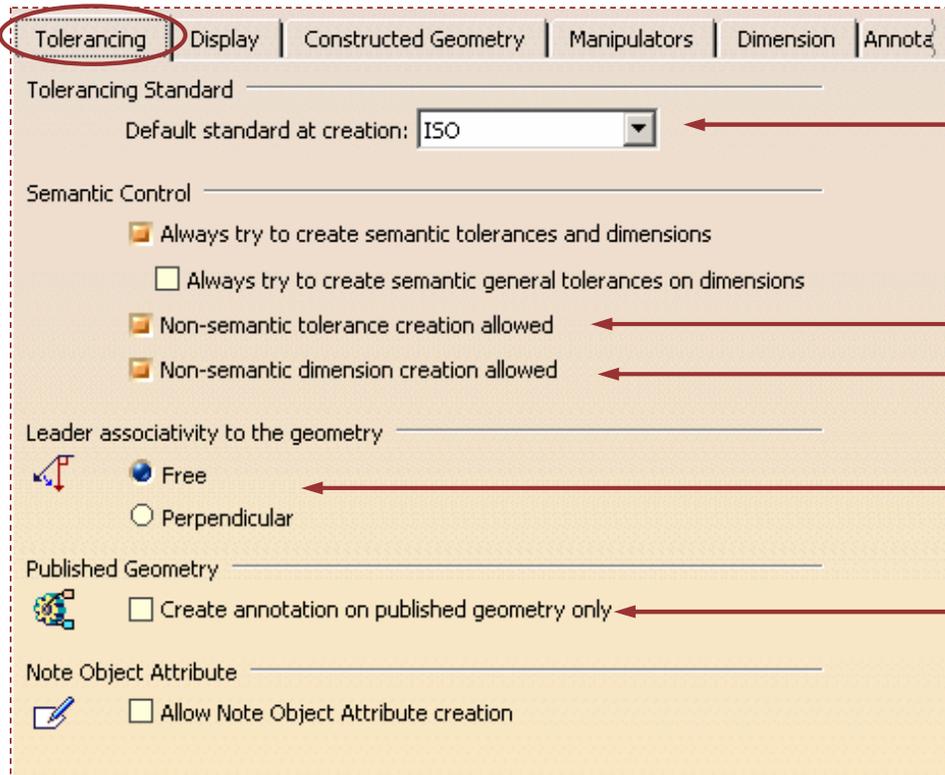
Default standard at creation: ISO ▼ ← **The default standard at creation provides: Three conventional standards (ANSI – ISO – JIS) and three CATIA-CADAM standards (CCDANSI – CCDISO – CCDJIS).**

Semantic Control

☐ Always try to create semantic tolerances and dimensions

☐ Always try to create semantic general tolerances on dimensions

☐ Non-semantic tolerance creation allowed ← **Whether non-semantic tolerances creation is allowed or not.**

☐ Non-semantic dimension creation allowed ← **Whether non-semantic dimensions creation is allowed or not.**

Leader associativity to the geometry

◉ Free ← **Whether leader annotations are perpendicular to their geometrical elements or not.**

○ Perpendicular

Published Geometry

☐ Create annotation on published geometry only ← **Defines whether annotations are created on published geometry only. When this option is selected, the Forbidden pointer is displayed over non-published geometry:**

Note Object Attribute

☐ Allow Note Object Attribute creation

FTA Settings - Display (1/3)

Access Tools > Options > Mechanical Design > Functional Tolerancing & Annotation > Display
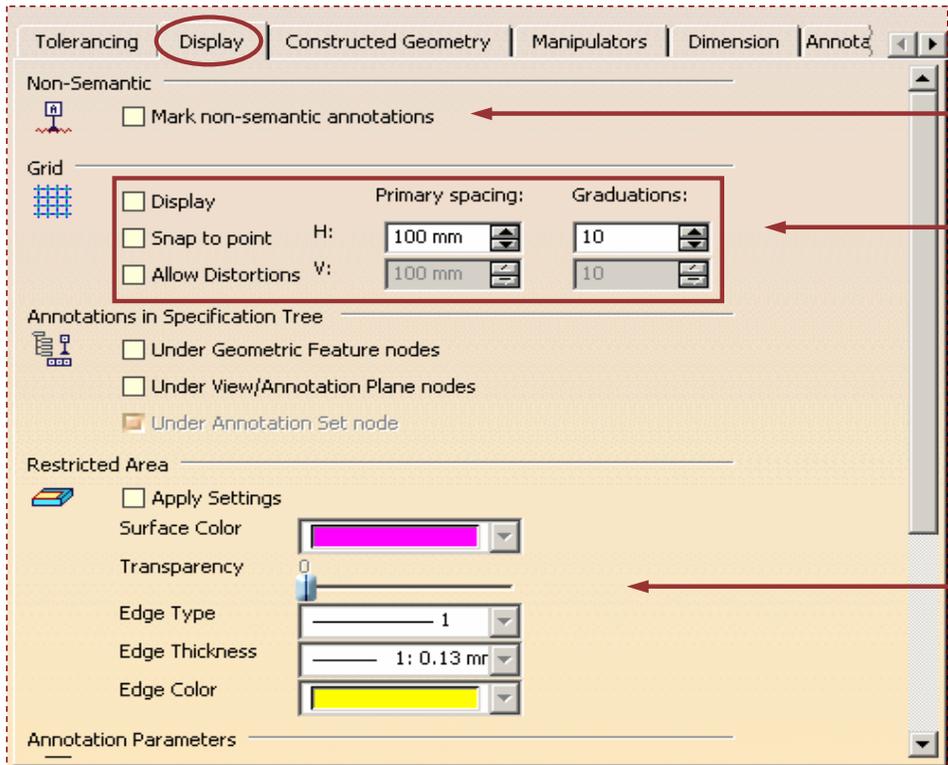
Mark with a wavy red line, in the specification tree and the geometry, non-semantic annotations

- Display the grid

- Snaps annotation to the grid's point

- Defines whether grid's spacing and graduations are the same horizontal and vertical

- Defines whether the following settings are applied while creating a partial surface feature

- Defines whether the next settings will be applied

- Defines the surface color of the partial surface

- Defines the edge type, the edge thickness and the edge color of the partial surface's border

FTA Settings - Display (2/3)

Access Tools > Options > Mechanical Design > Functional Tolerancing & Annotation > Display
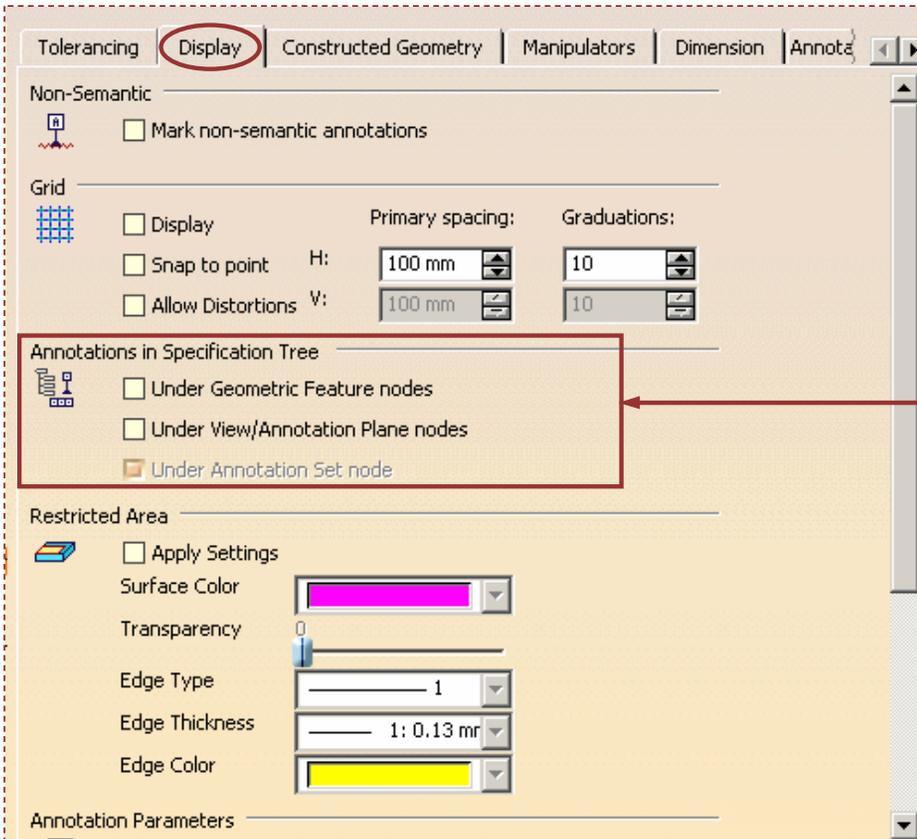
- Defines that 3D annotations should be displayed under the geometric feature nodes in the specification tree. This lets you view 3D annotations under the Part Design or GSD feature nodes to which they are applied.

- Defines that 3D annotations should be displayed under the view/annotation plane nodes in the specification tree. This lets you view 3D annotations under the view node to which they are linked.

- Defines that 3D annotations should be displayed under the annotation set node in the specification tree.

Student Notes:

FTA Settings - Display (3/3)

Access Tools > Options > Mechanical Design > Functional Tolerancing & Annotation > Display
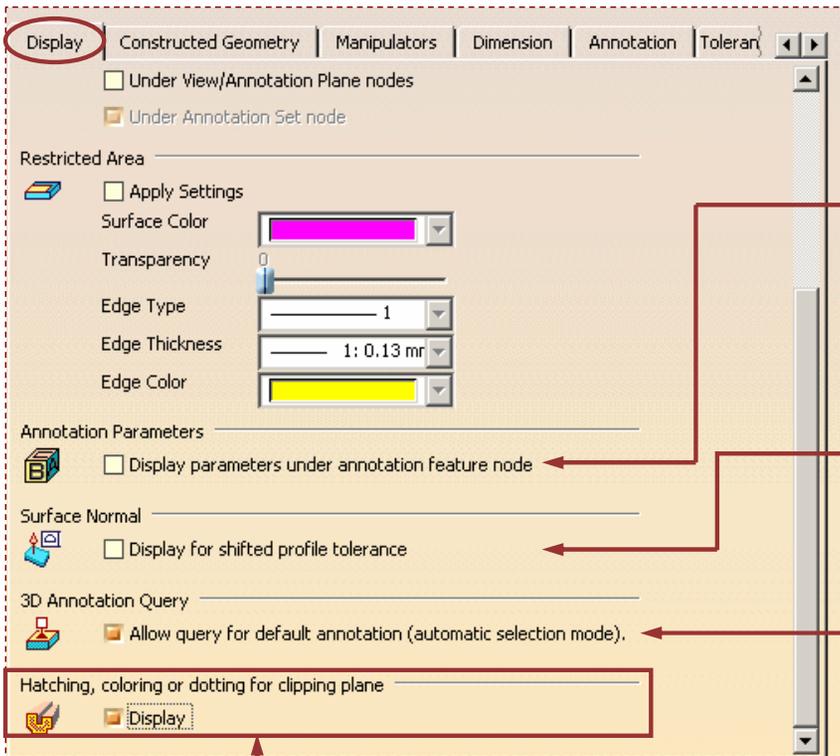
Defines that knowledge parameters (such as tolerance values, datum label, etc.) of annotations should be displayed under the annotation feature node in the specification tree; also defines that feature parameters of dimensions (accessible through the Edit Generative Parameter command) should be displayed under the dimension feature node in the specification tree.

Defines whether the normal of all the selected surfaces are displayed, or not, when a shifted profile tolerance is specified or queried.

Defines whether the query for default annotation is allowed, or not. This option allows you to highlight the related annotations or geometrical elements with the selected annotation or the related annotations with the selected geometrical element

Defines that, the section will be displayed properly, so that there will be no confusion in visualization of the part.

FTA Settings - Constructed Geometry

Access Tools > Options > Mechanical Design > Functional Tolerancing & Annotation > Constructed Geometry
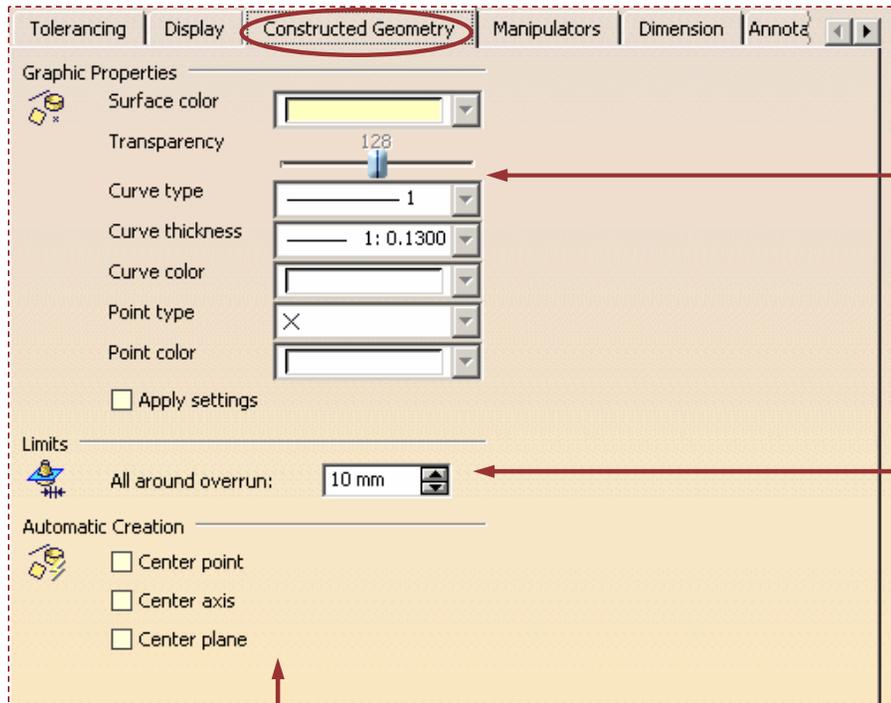
- Defines the surface color of the partial surface

- Defines the curve type, the curve thickness and the curve color of the partial surface's border

- Defines the point type and the color of the point

- Defines whether the previous settings will be applied

- Defines the minimal limit between the constructed geometry and its related geometry

- Defines whether all the center point's constructed geometry is automatically created or not, for circle center, sphere center.

- Defines whether all the center axis's constructed geometry is automatically created or not, for cylinder, cone.

- Defines whether all the center plane's constructed geometry is automatically created or not, for slot

FTA Settings - Manipulators

Access Tools > Options > Mechanical Design > Functional Tolerancing & Annotation > Manipulators
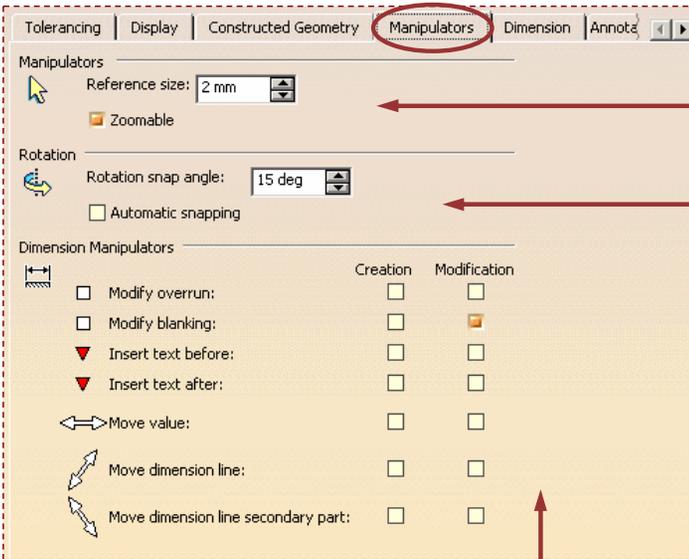
- Defines the annotation manipulator's size.

- Defines whether the annotation manipulator is zoom able or not.

- An angle value for rotating elements (this option is used to rotate text elements (text, frame, or leader)

- Whether the rotation will be snapped to the angle value or not.

- Defines whether overrun extension lines can be modified during creation or modification, or not.

- Defines whether blanking can be modified during creation or modification, or not.

- Defines whether a text before can be inserted during creation or modification, or not.

- Defines whether a text after can be inserted during creation or modification, or not.

- Defines whether only the value can be moved during creation or modification, or not.

- Defines whether only the dimension line can be moved during creation or modification, or not.

- Defines whether only the dimension line secondary part can be moved during creation or modification, or not.

FTA Settings - Dimension (1/2)

Access Tools > Options > Mechanical Design > Functional Tolerancing & Annotation > Dimension
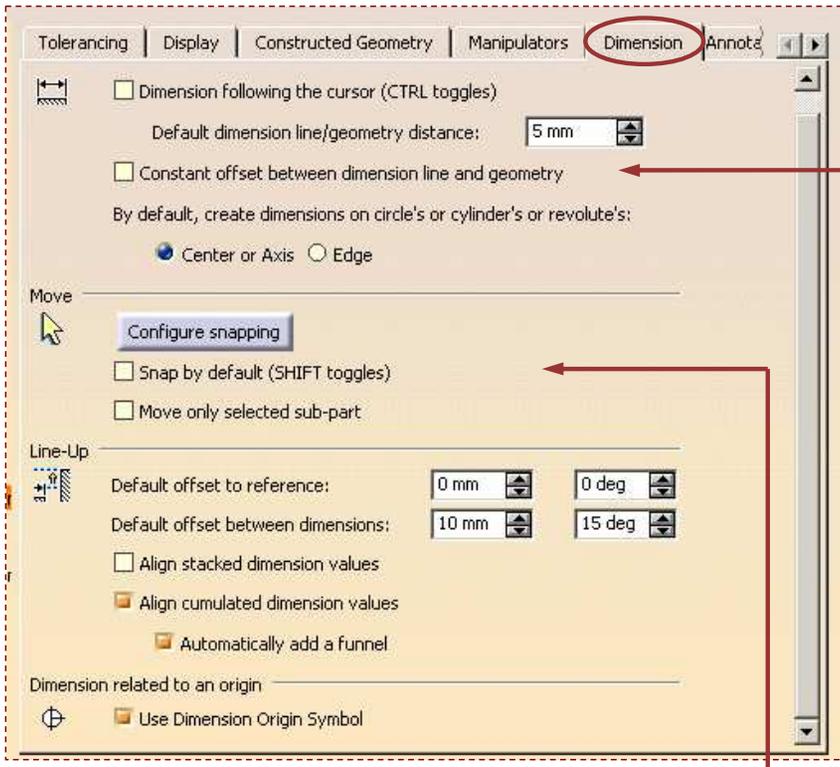
- **Defines whether the dimension line is positioned according to the cursor, following it dynamically during the creation process or not**

- **Defines whether a dimension aligned to another automatically ends the command or not**

- **Defines whether the distance between the created dimension and the geometry remains the same when you move the geometry or not**

- **Defines the value at which the dimension is created from the geometry**

- **Defines the dimension you will create between a circle and another element will be either on the circle center or on the circle edge**

- **Defines whether the dimension will be snapped on the grid and/or the dimension value will be located at its default position between symbols (it will work only if the cursor is between the symbols) or not**

- **Defines whether only a dimension sub-part (text, line, etc...) will be moved or not.**

FTA Settings - Dimension (2/2)

Access Tools > Options > Mechanical Design > Functional Tolerancing & Annotation > Dimension
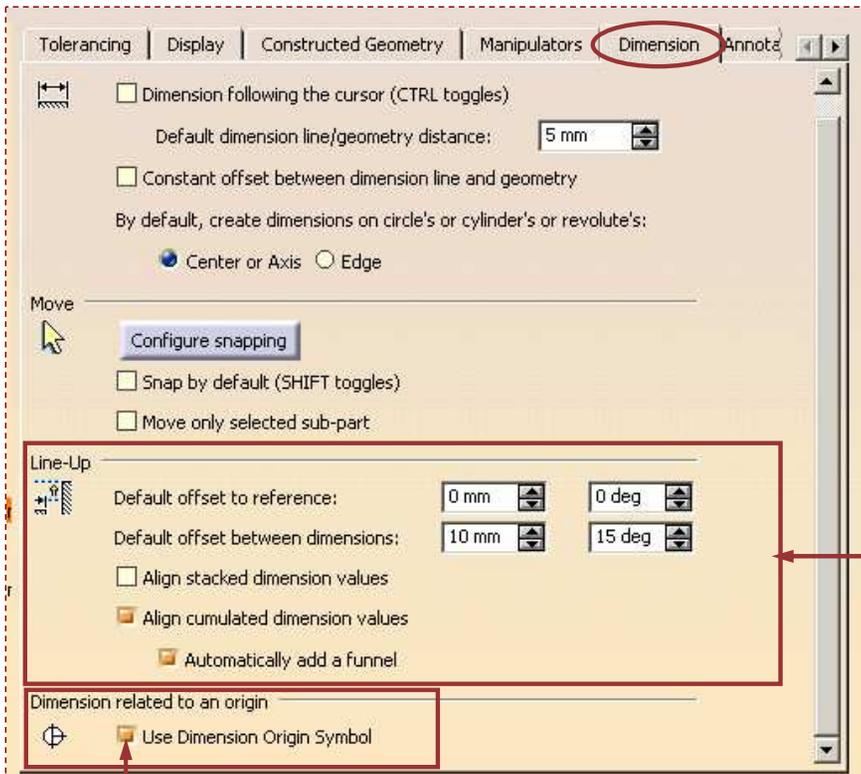

- **Offset to reference: Defines distance or angle between the selected reference.**

- **Offset between dimensions: Defines distance or angle between dimensions that can be organized as cumulated or stacked dimensions. (For both the above options, two fields are available. The first field is dedicated to length, distance and angular dimensions and the second field is dedicated to radius and diameter dimensions.)**

- **Align stacked dimension values: Defines whether the values of a group of stacked dimensions are aligned on the value of the smallest dimension of the group or not.**

- **Align cumulated dimension values: Defines whether the values of a group of cumulated dimensions are aligned on the value of the smallest dimension of the group or not.**

- **Automatically add a funnel: Defines whether the value of a cumulated dimension requires a funnel added automatically to be displayed correctly or not.**

- **Defines whether the dimension origin symbol is used or not**

FTA Settings - Annotation

Access Tools > Options > Mechanical Design > Functional Tolerancing & Annotation > Annotation

| | Display | Constructed Geometry | Manipulators | Dimension | Annotation | Toleran |

Annotation Creation
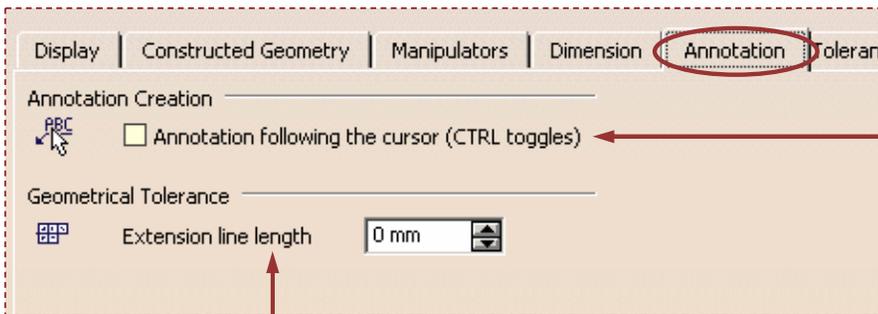
ABC [ ] Annotation following the cursor (CTRL toggles) ◄───── **- Defines whether the annotation is positioned according to the cursor, following it dynamically during the creation process or not.**

Geometrical Tolerance

Extension line length [0 mm] ⬍

- Defines the extension line length between the geometrical frame and its leader.

FTA Settings - Tolerances (1/2)

**Access Tools > Options > Mechanical Design > Functional Tolerancing & Annotation >
Tolerances**
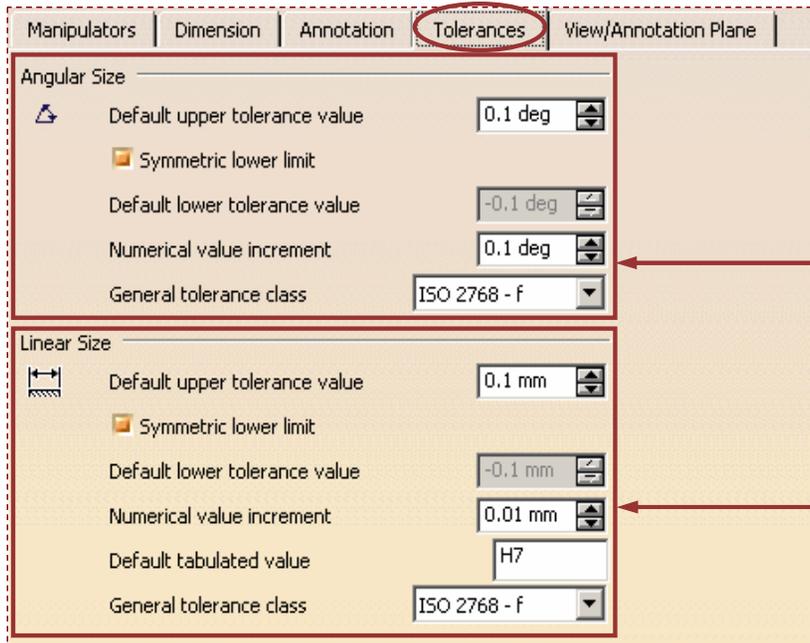
- Defines the default upper tolerance value for angular size.

- Defines whether the default lower tolerance value is symmetric in relation to the default upper tolerance value.

- Defines the default lower tolerance value for angular size, disable when Symmetric lower limit is checked and the increment for angular size numerical value.

- Defines the default upper tolerance value for linear size.

- Defines whether the default lower tolerance value is symmetric in relation to the default upper tolerance value.

- Defines the default lower tolerance value for linear size, disable when Symmetric lower limit is checked.

- Defines the increment for linear size numerical value and the default tabulated for linear size.

FTA Settings - Tolerances (2/2)

Access Tools > Options > Mechanical Design > Functional Tolerancing & Annotation > Tolerances
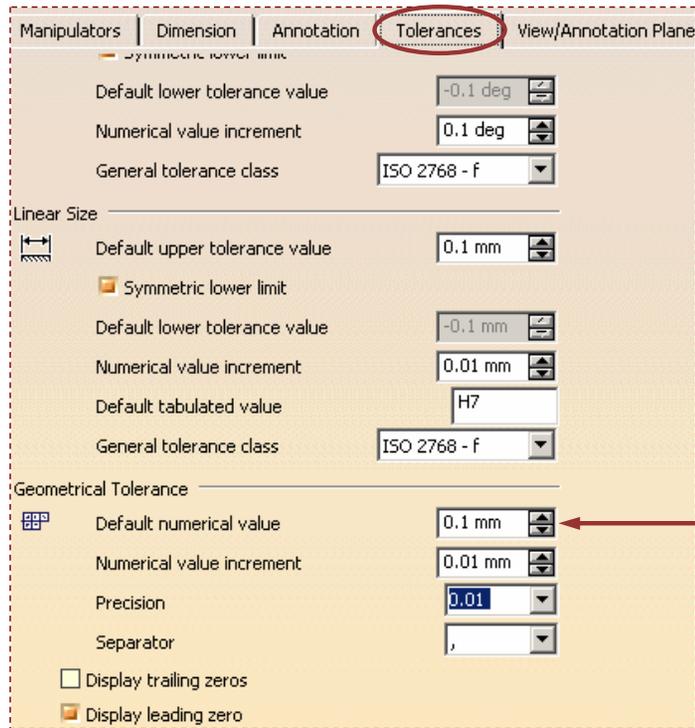
-**Defines the default numerical, the increment and the separator's symbol for geometrical tolerance.**

- **Defines whether '0's complete the number of digit displayed after the separator, according to the precision, or not.**

-**Defines whether the '0' before the numerical separator, when value is less than 1, is displayed or not.**

FTA Settings - View/Annotation Plane (1/2)

Access Tools > Options > Mechanical Design > Functional Tolerancing & Annotation > View/Annotation Plane

Manipulators | Dimension | Annotation | Tolerances | View/Annotation Plane | ◄ ►

View/Annotation Plane Associativity

☐ Create views associative to geometry

View/Annotation Plane Display

View axis display
○ Always hidden
○ Displayed for current view only
● Displayed for last selected view

☐ Zoomable

☐ Visualization of the profile in the current view
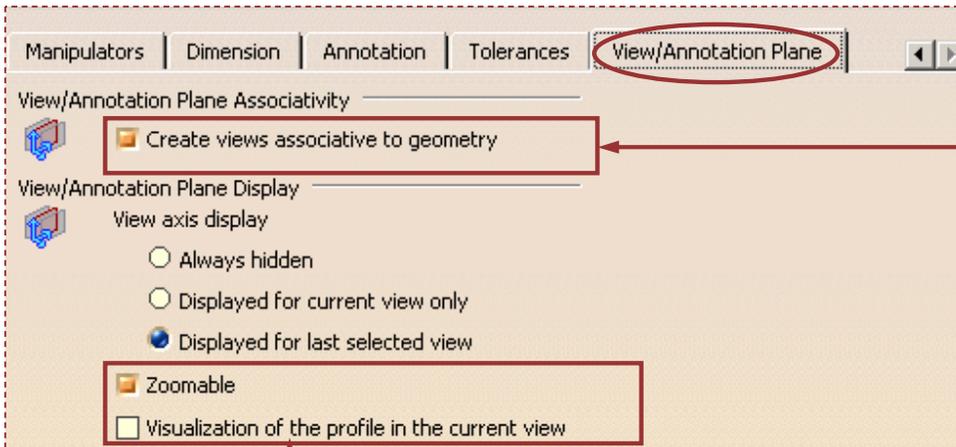
Creates views associative to the geometry, so that views and their annotations are automatically updated when the geometry is modified.

- Zoom able
Defines whether the annotation plane axis is zoomable.

- Visualization of the profile in the current view
Defines whether the view/annotation plane profile on the part/product is displayed or not.

FTA Settings - View/Annotation Plane (2/2)

Access Tools > Options > Mechanical Design > Functional Tolerancing & Annotation > View/Annotation Plane
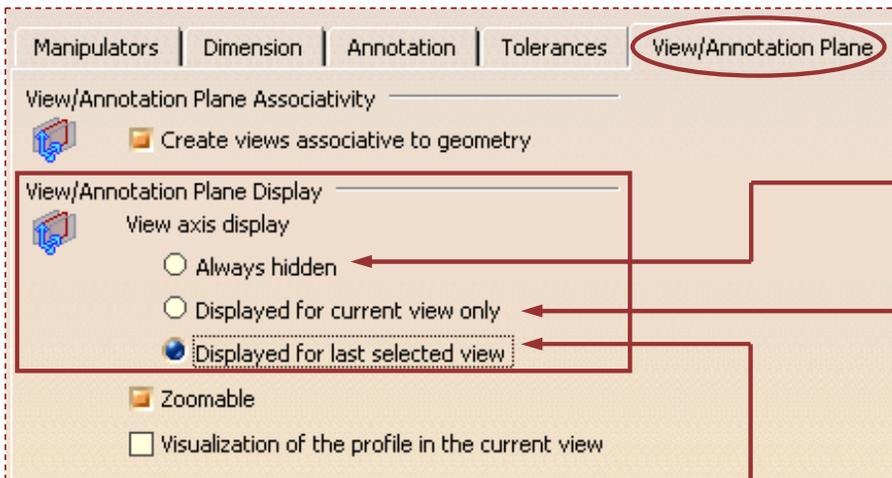
This option allows you to always keep the View Axis hidden.

This option allows you to display the View Axis for the last current view only.

This option allows you to display the View Axis for the last selected view.

The Infrastructure Settings (1/3)

Access Tools > Options > Infrastructure > Product Structure > CGR Management
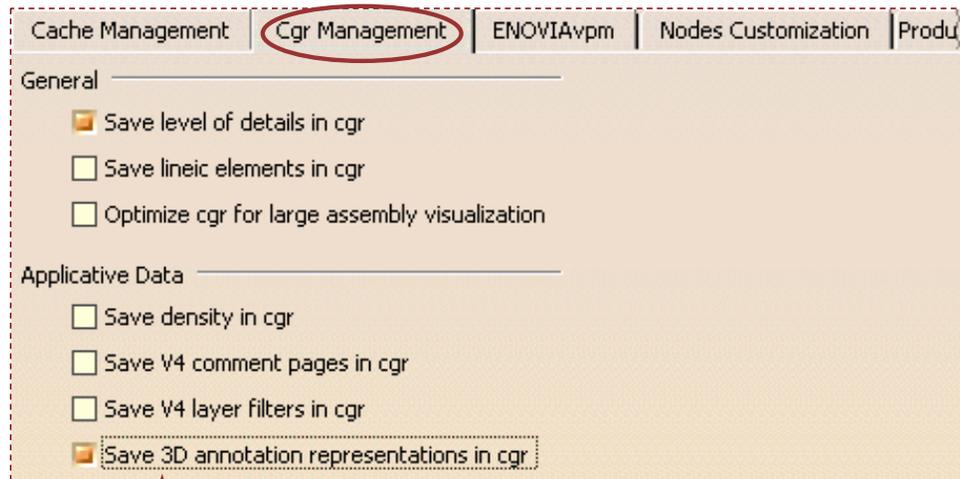
- You need to select this option to add the 3D annotations representation contained in a CATProduct or a CAT Process document to the generated CGR documents.

This option is taken into account when the Cache Activation option is selected only.

The Infrastructure Settings (2/3)

Access Tools > Options > Infrastructure > Part Infrastructure > General
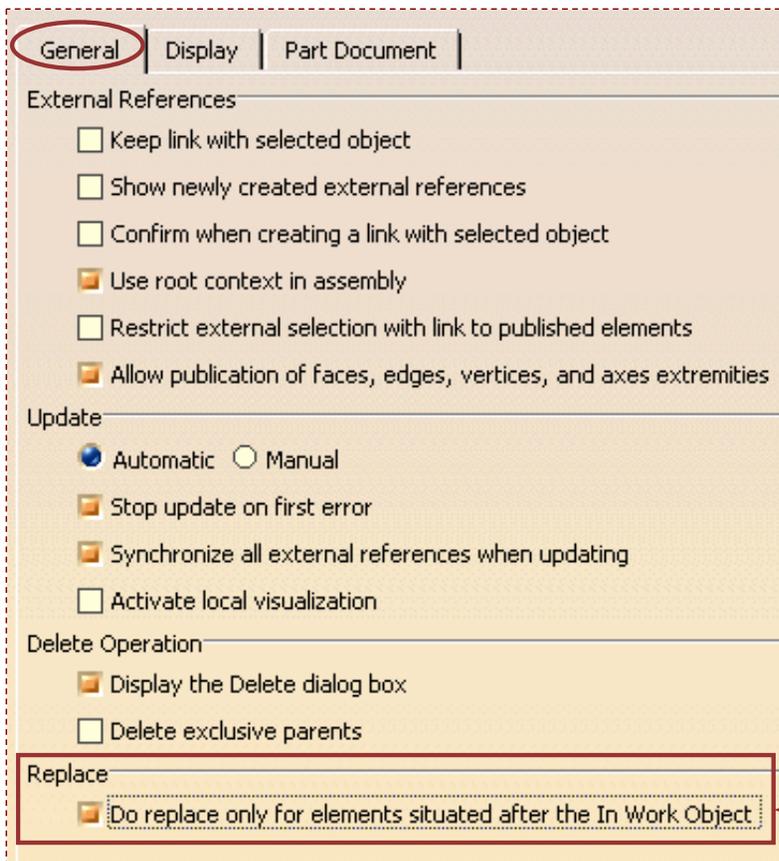
This option will do the replace, only for the elements after the 'In Work Object' from the specification tree.

The Infrastructure Settings (3/3)

Access Tools > Options > Infrastructure > Part Infrastructure > Display
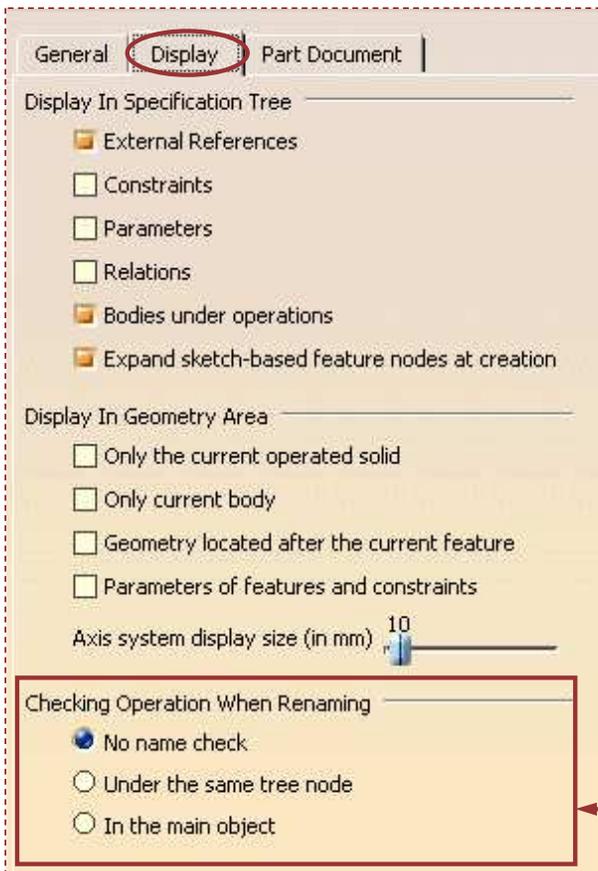
- **No name check:**
 Defines that, no warning dialog box will appear and capture will be created with the given name.

- **Under the same tree node:**
 Defines that, it will check if the name given to the capture is already used by other capture in the current annotation set node.

- **In the main object:**
 Defines that, it will check if the name given to the capture is already used by other features of the active part, product or process.

Generative Drafting Settings

Access Tools > Options > Mechanical Design > Drafting > View
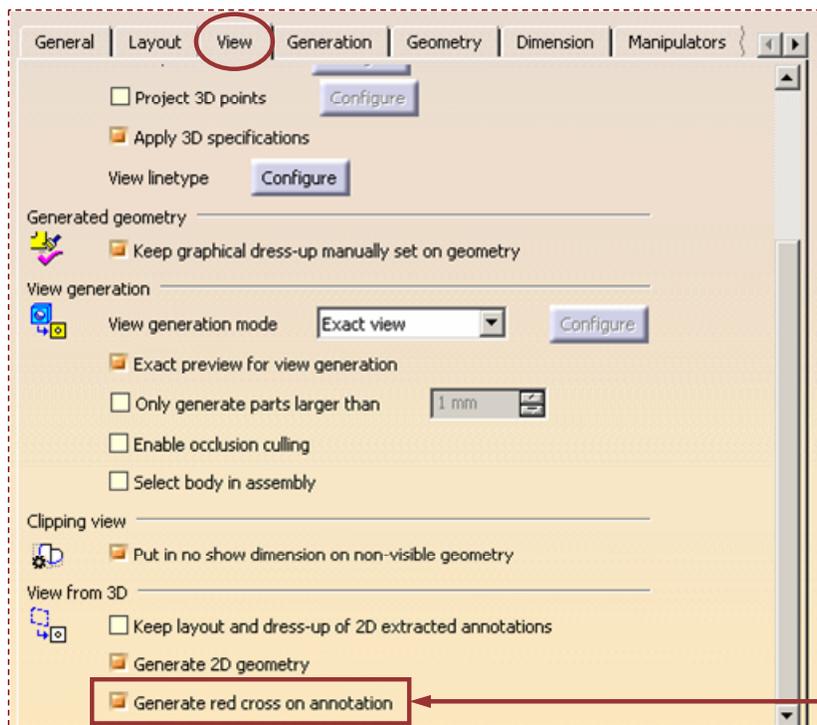
Activating this option will generate red cross on annotations (i.e annotations representing the features that are not in view).

Deactivating this option will avoid generation of red cross on annotations (i.e annotations representing the features that are not in view).

Co-ordinate Dimension Display Settings

Select the Co-ordinate Dimension > Right-click on the selection > Properties > Numerical
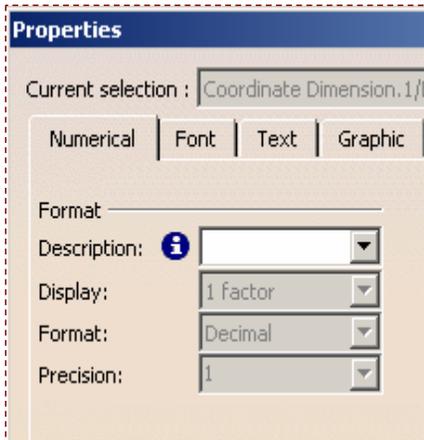
Default setting

Results in
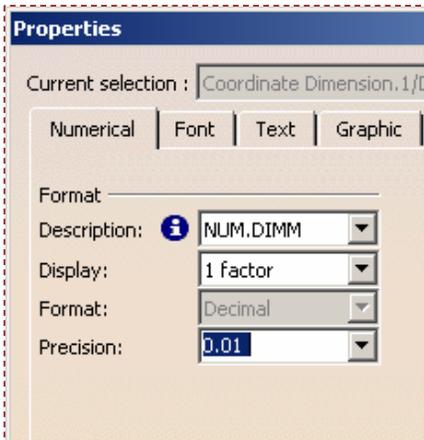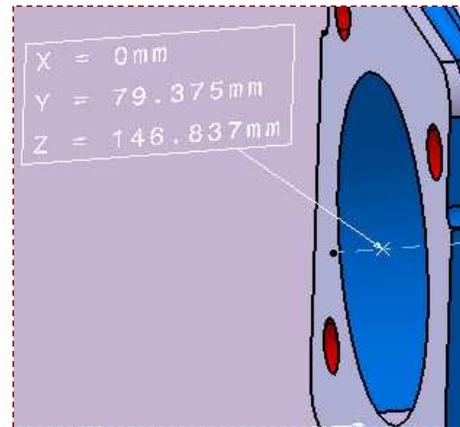
Changed setting

Results in
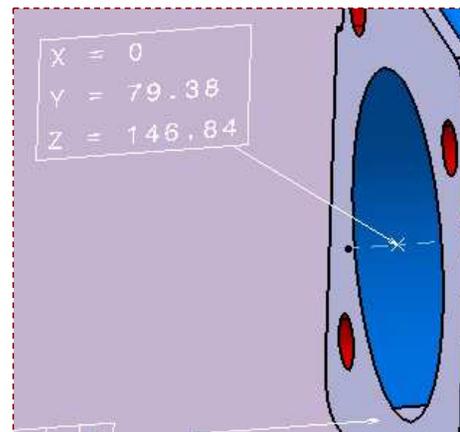
Student Notes:

To Sum Up

In this Lesson you have seen:

- **Various User settings to enable to you customize FTA to your style of working**
- **Infrastructure Settings**

Summary

In this course you have learnt:

- **How to Create Semantic Annotations using Tolerancing Advisor**
- **How to Manage Annotations**
- **How to link Annotations with Geometry**
- **Some Advanced functions like Restricted areas and Datum targets**
- **Various User Settings**