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CATIA V5-6R2023
Functional Tolerancing & Annotation

Learning Guide

Edition 1.0

ASCENT - Center for Technical Knowledge®
CATIA V5-6R2023
Functional Tolerancing & Annotation
Edition 1.0

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AS-56CAT23-FTA1-SG // RS-56CAT23-FTA1-SG

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Preface

The *CATIA V5-6R2023: Functional Tolerancing & Annotation* learning guide has numerous practices that will help you acquire the skills to create and display engineering, manufacturing, and assembly information directly on the 3D part, assembly, or process model. This extensive hands-on guide will provide you with a thorough understanding of geometric tolerances, dimensions, notes, and other annotations critical to the accurate and cost-effective creation of mechanical parts and assemblies. This guide complies with the industry and government initiated American Society of Mechanical Engineers' (ASME) Y14.41 3D standards for the creation and submission of model only, paperless design applications.

Topics Covered

- Introduction to Functional Tolerancing & Annotation
- Workbench overview
- Annotation process
- Extracting 2D view from the 3D model
- Annotation planes and extraction views
- Construction geometry
- Semantic and non-semantic annotations
- Datum Reference Frames
- Tolerance Advisor
- Basic Dimensions
- Annotations: Text, Flag Notes, Datum Elements, Datum Targets, Roughness, Dimensions
- Restricted Areas
- Threads
- Annotation Visualization Tools: Query, Grouping, Leader Symbols, Annotation Mirror and Transfer, Filters
- Cameras and Captures
- Geometry Connection Management
- FT&A analysis and reporting
- Product Functional Tolerance and Annotation workbench

Prerequisites

- Access to the V5-6R2023 version of the software, to ensure compatibility with this guide. Future software updates that are released by Dassault Systèmes may include changes that are not reflected in this guide. The practices and files included with this guide might not be compatible with prior versions (e.g., V5-6R2022).
- Completion of the *CATIA V5-6 R2023: Introduction to Modeling* guide and a working knowledge of GD&T application are recommended.

Note on Software Setup

This guide assumes a standard installation of the software using the default preferences during installation. Lectures and practices use the standard software templates and default options for the Content Libraries.

Note on Learning Guide Content

ASCENT's learning guides are intended to teach the technical aspects of using the software and do not focus on professional design principles and standards. The practices aim to demonstrate the capabilities and flexibility of the software rather than following specific design codes or standards.

Lead Contributor: Mark Potrzebowski

Mark is a seasoned trainer and curriculum designer with more than 15 years of experience in the PLM industry. With a primary focus on CATIA, Creo, and PLM systems, Mark uses his Instructional Design skills and comprehensive CAD experience to develop training products at ASCENT, including learning content for print and web, as well as instructional videos and presentations.

Mark holds a bachelor's degree in Computer Graphics Technology from Purdue University, West Lafayette, Indiana.

Mark Potrzebowski is the Lead Contributor for *CATIA: Functional Tolerancing & Annotation*.

In This Guide

The following highlights the key features of this guide.

Feature	Description
Practice Files	The Practice Files page includes a link to the practice files and instructions on how to download and install them. The practice files are required to complete the practices in this guide.
Chapters	A chapter consists of the following: Learning Objectives, Instructional Content, and Practices. • **Learning Objectives** define the skills you can acquire by learning the content provided in the chapter. • **Instructional Content**, which begins right after Learning Objectives, refers to the descriptive and procedural information related to various topics. Each main topic introduces a product feature, discusses various aspects of that feature, and provides step-by-step procedures on how to use that feature. Where relevant, examples, figures, helpful hints, and notes are provided. • **Practice** for a topic follows the instructional content. Practices enable you to use the software to perform a hands-on review of a topic. It is required that you download the practice files (using the link found on the Practice Files page) prior to starting the first practice.
Appendices	Appendices provide additional information to the main course content. It could be in the form of instructional content, practices, tables, projects, or skills assessment.

Chapter 1

Introduction to FT&A

The goal is to teach you how to create tolerances and annotations to detail a model using the CATIA: Functional Tolerancing & Annotations workbench. You should already have a foundation level understanding of the rules, standards, and types of geometrical tolerances, dimensions, and annotations.

Learning Objectives

- Understand the usage of tolerancing and annotations.
- Understand the use of the FT&A Workbench.
- Recognize the FT&A interface and tools.
- Understand the annotation process.
- Create drawings from an annotated model.
- Learn how to access annotations and set annotation references.

1.1 Tolerancing and Annotations

Tolerances and annotations provide the details required to manufacture and fabricate a model designed in CATIA. Tolerances and annotations guide a Manufacturing group in the following ways:

- Details the acceptable standards.
- Provides the window for geometric variances and imperfections in parts and assemblies.
- Facilitates the production process and manufacturing standards.

Traditionally, these annotations are placed in the 2D drawing. Using the Functional Tolerancing & Annotation workbench, all annotations are stored directly in the 3D model. This eliminates the need to develop a drawing, and supports a paperless design and manufacturing environment. An example of an annotated part model is shown in Figure 1–1.

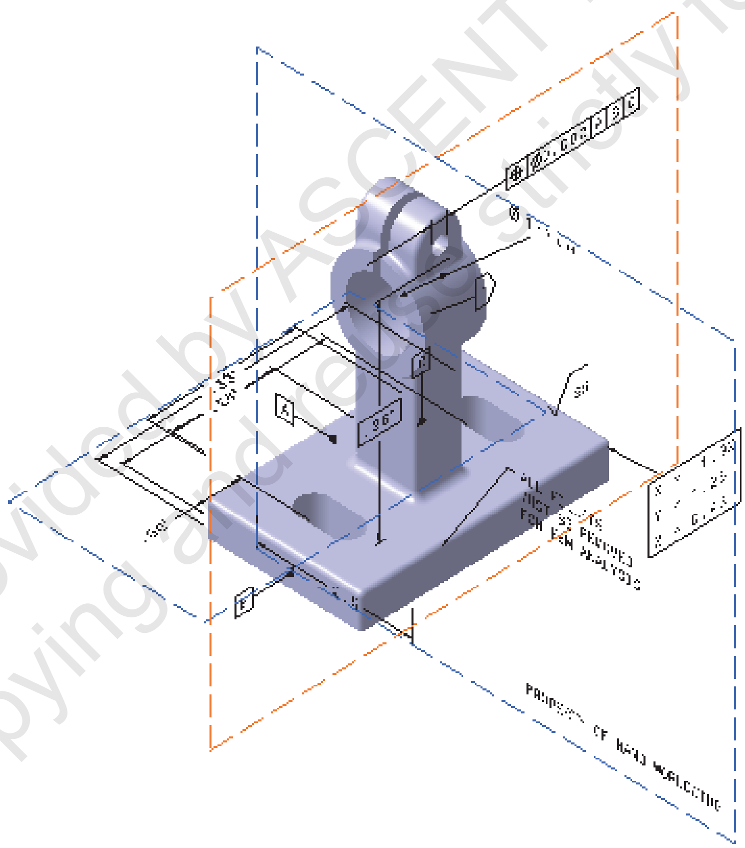


Figure 1–1

Placing the tolerances and annotations directly into the 3D design enables the manufacturing process to be considered throughout the design cycle, specifically in the initial stages. This helps avoid conflicts between design and manufacturing as early in the product life cycle as possible.

Annotations can be created in the following CATIA model types:

Model Type	File Extension	Workbench
Part	*.CATPart	Functional Tolerancing & Annotation
Assembly	*.CATProduct	Product Functional Tolerancing & Annotation
Process	*.CATProcess	Process Tolerancing & Annotation

Although all annotations can be added to parts or products after their designs are completed, certain annotations can be created during the design process.

All annotations are organized in specific branches of an Annotation Set in the specification tree. An example is shown in Figure 1–2.

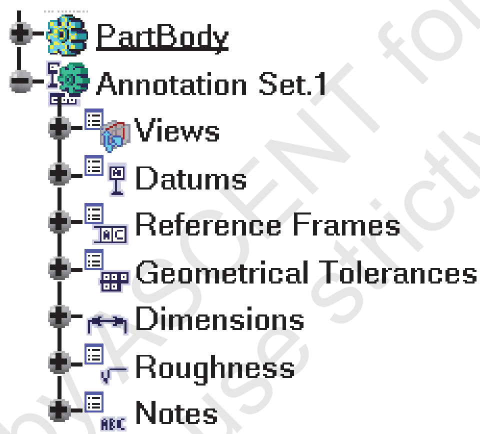


Figure 1–2

Annotation Standards

Similar to the drawing standards that are used to create 2D drawings, standards apply to the creation of annotations in a 3D model. For example, some annotations are created to match the ASME standard.

Since it is possible to break a rule of the applied standard, two annotation categories can be created:

- Semantic
- Non-Semantic

Semantic

This type of annotation always matches the rules, specifications, and designations assigned by the active standard. Semantic annotations are created using the Tolerance Advisor or Framed (Basic) Dimensions icons. The *Semantic Tolerance Advisor* dialog box is shown in Figure 1–3.

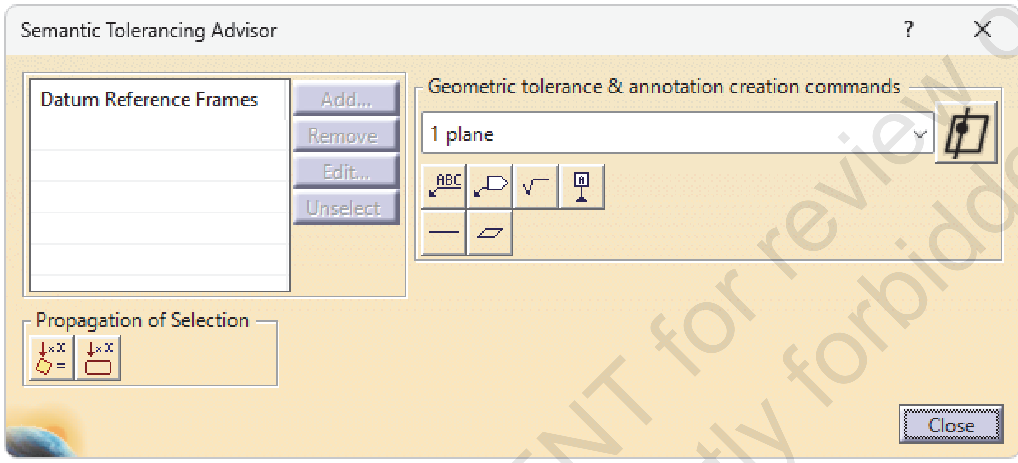


Figure 1–3

Non-Semantic

Non-semantic annotations might not necessarily match the active standard. Create this type of annotation when the active standard does not adequately represent the design intent of the model or the standards adopted by the designing company. Care should be taken when creating a non-semantic annotation to ensure that the end user of the annotation understands the designer's intent. Non-semantic annotations are created using the icons shown in Figure 1–4.

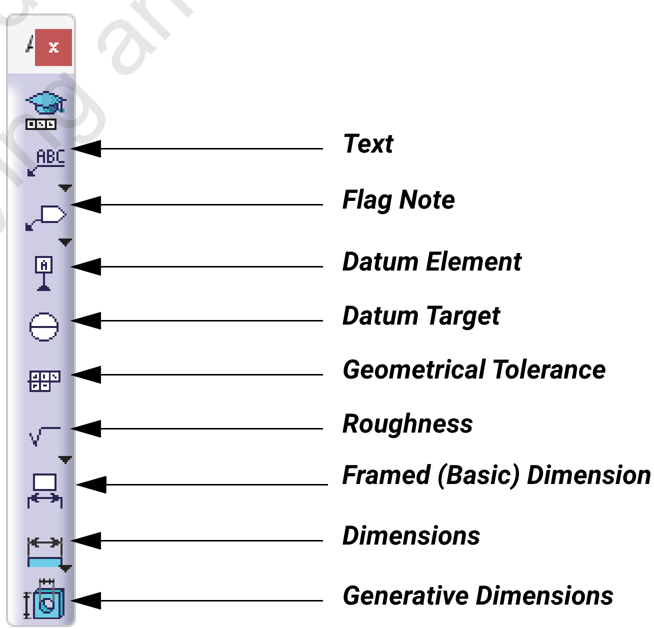


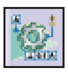
Figure 1–4

1.2 FT&A Workbenches

You can create 3D annotation on CATIA models in three modes:

- Part
- Assembly
- Process

Part

The majority of this learning material focuses on the creation of annotations and tolerances for part models in the Functional Tolerancing & Annotation workbench. To access this workbench, select **Start>Mechanical Design>Functional Tolerancing & Annotation**. The workbench symbol changes to .

Assembly

To access the Product Functional Tolerancing & Annotation workbench, select **Start>Mechanical Design>Product Functional Tolerancing & Annotation**.

Process

To access the Process Tolerancing & Annotation workbench, select **Start>Digital Process for Manufacturing>Process Tolerancing & Annotation**.

1.3 FT&A Interface

The Functional Tolerancing & Annotation workbench interface displays, as shown in Figure 1–5.

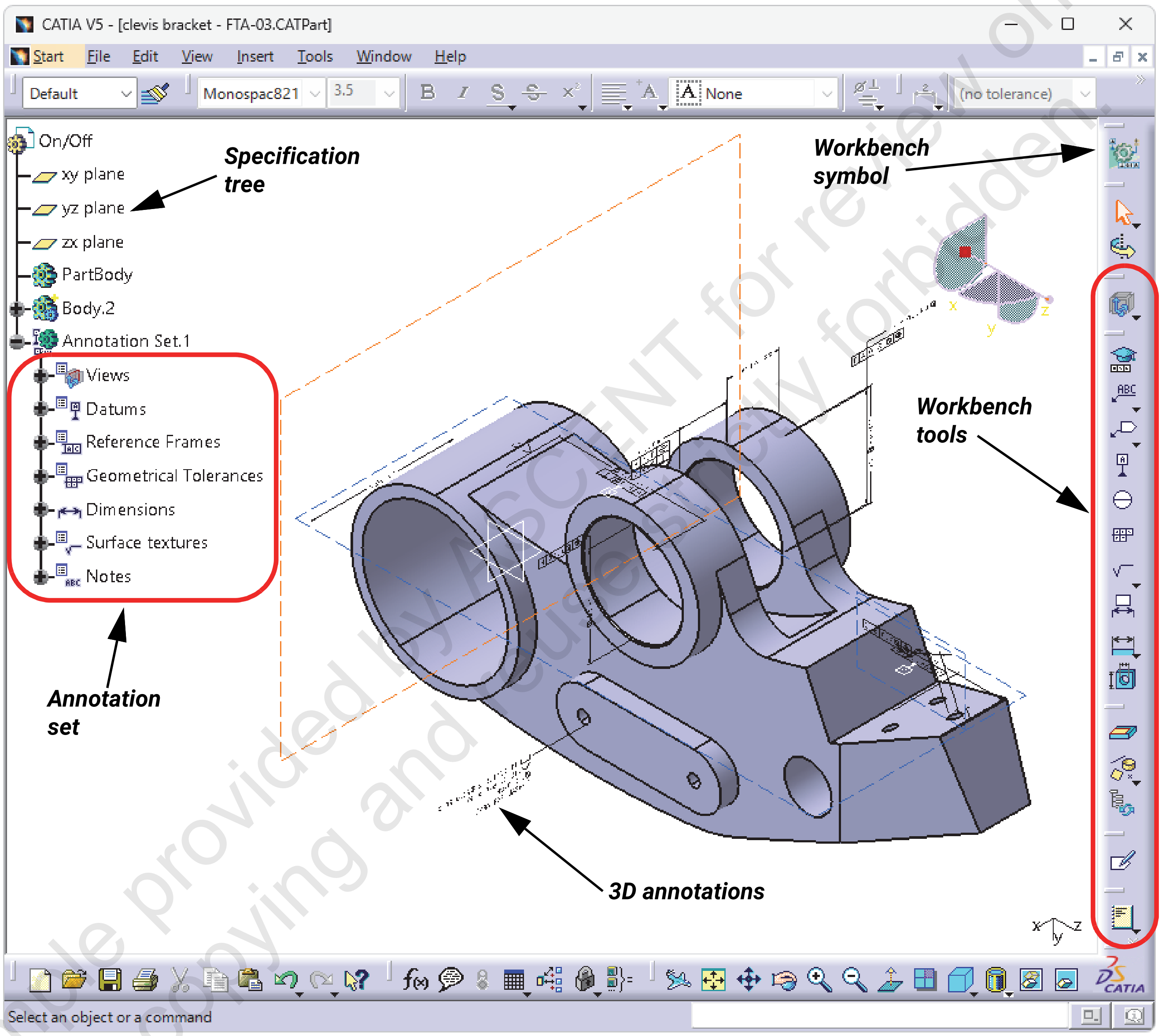
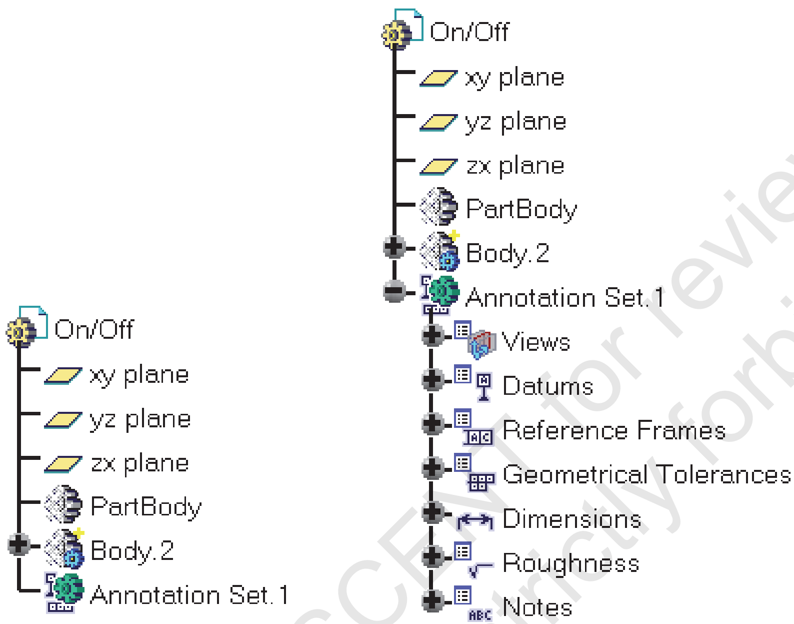


Figure 1–5

Annotation Switch On/Off

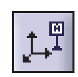
Since 3D annotations add visual complexity to the model, they can be toggled on and off to simplify the display. If you receive a model that has no annotations displayed, you must toggle on the annotation set. To toggle the annotations on and off, right-click on the annotation set and select **Annotation Set Switch On/Switch Off**. This is useful when viewing an annotated model in an assembly.

When an annotation set has been toggled off, the name of the annotation set displays in the specification tree, as shown in Figure 1–6. However, the branch cannot be expanded and no annotations display on the model.



Annotation Set toggled off ***Annotation Set toggled on***

Figure 1–6

The display status of an annotation set can also be controlled by clicking  (List Annotation Set Switch On/Switch Off) in the *Visualization* toolbar. The *Annotation Set Switch On/Switch Off* dialog box opens, as shown in Figure 1–7.

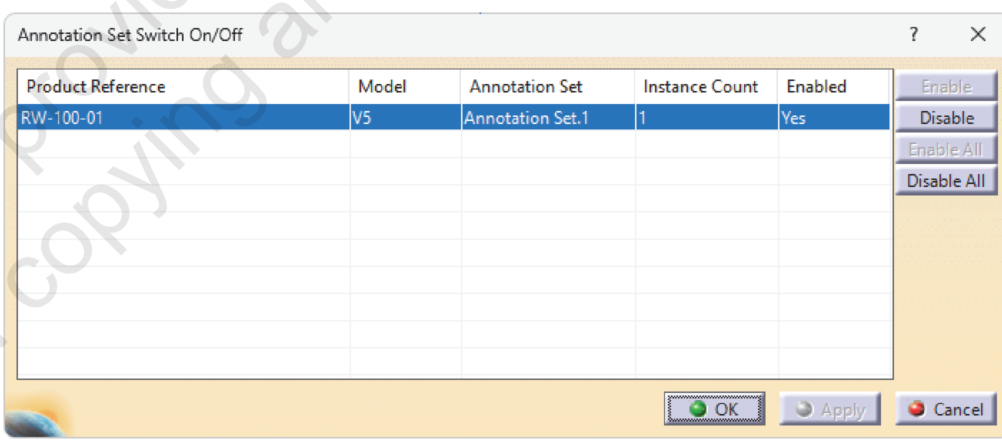


Figure 1–7

To control the display of an annotation set, select it in the list and use the buttons on the right side of the dialog box. This tool is very useful when working in the Product Functional Tolerancing & Annotation workbench. It enables you to control the display of multiple part-level annotation sets in a single operation.

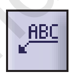
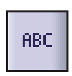
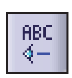
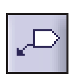
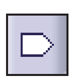
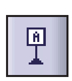
1.4 FT&A Tools

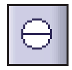
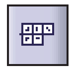
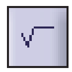
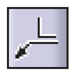
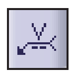
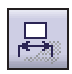
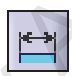
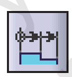
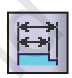
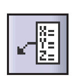
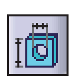
This section provides an overview of the different tools available in the Functional Tolerancing & Annotation workbench. The workbench consists of the following seven toolbars:

- Annotations
- Geometry for 3D Annotations
- Views/Annotation Planes
- Reporting
- Visualization
- Capture
- Grouping

Annotations

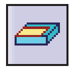
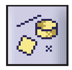
The *Annotations* toolbar contains all of the tools required to fully annotate and tolerance the part. The *Annotations* toolbar icons are as follows.

Option	Icon	Description
Tolerance Advisor		Creates semantic annotations based on the selected geometry.
Text flyout		
Text with Leader		Creates text with a leader to geometry in the annotation plane.
Text		Creates text in the annotation plane.
Text Parallel To Screen		Creates text referencing geometry or an annotation plane that is always parallel to the screen.
Flag Note flyout		
Flag Note with Leader		Creates a flag note with a leader to geometry in the annotation plane.
Flag Note		Creates a flag note in the annotation plane.
Datum Element		Creates a datum element from selected geometry.

Option	Icon	Description
Datum Target		Creates a datum target from selected geometry.
Geometrical Tolerance		Creates a geometrical tolerance for the selected geometry.
Symbols flyout		
Surface Texture		Specifies the surface roughness.
Edge of Undefined Shape		Specifies the conditions of an edge of an undefined shape.
Weld Feature		Specifies the conditions of a weld feature.
Framed (Basic) Dimensions		Creates a basic dimension for the selected geometry.
Dimensions flyout		
Dimensions		Creates dimensions for the selected geometry.
Cumulated Dimensions		Creates cumulated dimensions for the selected geometry.
Stacked Dimensions		Creates stacked dimensions for the selected geometry.
Coordinate Dimensions		Creates coordinate dimensions for the selected geometry.
Curvilinear Dimensions		Creates curvilinear dimensions for the selected geometry.
Generative Dimension		Creates selected model feature dimensions automatically.

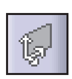
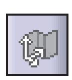
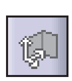
Geometry for 3D Annotations

The *Geometry for 3D Annotations* toolbar is used to create and manage constructed geometry, which is used to correctly reference and annotate the part. The *Geometry for 3D Annotations* toolbar icons are as follows.

Option	Icon	Description
Restricted Area		Creates a restricted area from the selected geometry.
Geometry for 3D Annotations flyout		
Constructed Geometry Creation		Creates construction geometry based on selected element(s).
Constructed Geometry Management		Shows details of constructed geometry references.
Thread Representation Creation		Creates thread representation construction geometry.
Geometry Connection Management		Shows details of constructed geometry connections.

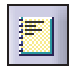
Views/ Annotation Planes

The *Views/Annotation Planes* toolbar contains the various view creation tools. These views are used to hold all annotations for the part, and can be extracted with their annotations to a 2D representation. The following describes the *Views/Annotation Planes toolbar* icons (flyout menu).

Option	Icon	Description
Principal Views		Creates the principal views (Front, Left, Right and so on).
View from Reference		Creates a view from the selected plane/planar surface.
Offset Section View/Section Cut		Creates an offset section view from the selected sketch.
Aligned Section View/Section Cut		Creates an aligned section view from the selected sketch.

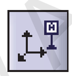
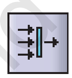
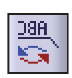
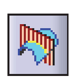
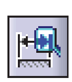
Reporting

The *Reporting* toolbar is used to evaluate your annotations against specific checks developed from tolerancing standards. Annotations that do not meet the check criteria can be identified for resolution. The following describes the *Reporting* toolbar icons (flyout menu).

Option	Icon	Description
Report		Generates a report based on annotations.
Report Customization		Customizes report generation options.

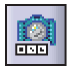
Visualization

The *Visualization* toolbar tools can be used to investigate part/annotation relationships, filter and adjust the display of annotations, and better display the part and its details. The following describes the *Visualization* toolbar icons.

Option	Icon	Description
List Annotation Set Switch On/Switch Off		Enables annotation sets to be switched on and off.
3D-Annotation-Query Switch On/Switch Off		When active, selecting an element also highlights all associated geometry and annotations.
Filter		Enables annotations to be filtered and displayed.
Mirror annotations		Mirrors direction of text in the annotation plane.
Clipping Plane		Cuts part by selected annotation plane.
Analysis Display Mode		When enabled, it will color code dimensions that are invalid, linked to deleted geometry, or linked to unloaded files.
Annotation Blanking		Displays annotations with a blank background

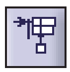
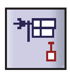
Capture

The *Capture* tool is used to create snapshots of your model. Capture enables you to orient and display the part in a specific way, and to hide and show required annotations to clearly display the required information. The following describes the **Insert>Capture>Capture** tool.

Option	Icon	Description
Capture		Creates a capture in the Tolerancing Capture workbench.

Grouping

The Grouping tools can be used to clean up and organize the display. You can manually or automatically group annotations. The following describes the Grouping toolbar icons.

Option	Icon	Description
Automatic Grouping		Automatically groups selected annotations together.
Manual Grouping		Manually groups selected annotations together.

1.5 Annotation Process

General Steps

Use the following general steps to add annotations and tolerances to a model:

1. Define the tolerancing standard.
2. Define the annotation planes.
3. Create dimensions and tolerances.
4. Create datum reference frames.
5. Create geometrical tolerances.
6. Create basic dimensions.
7. Create additional annotations.
8. (Optional) Extract 2D drawing views.

Step 1 - Define the tolerancing standard.

The tolerancing standard defines the properties to which the annotations must conform. To define the standard, select **Tools>Options>Mechanical Design>Functional Tolerancing & Annotation** and select the *Tolerancing* tab. Select an option in the **Default standard at creation** drop-down list, as shown in Figure 1–8.

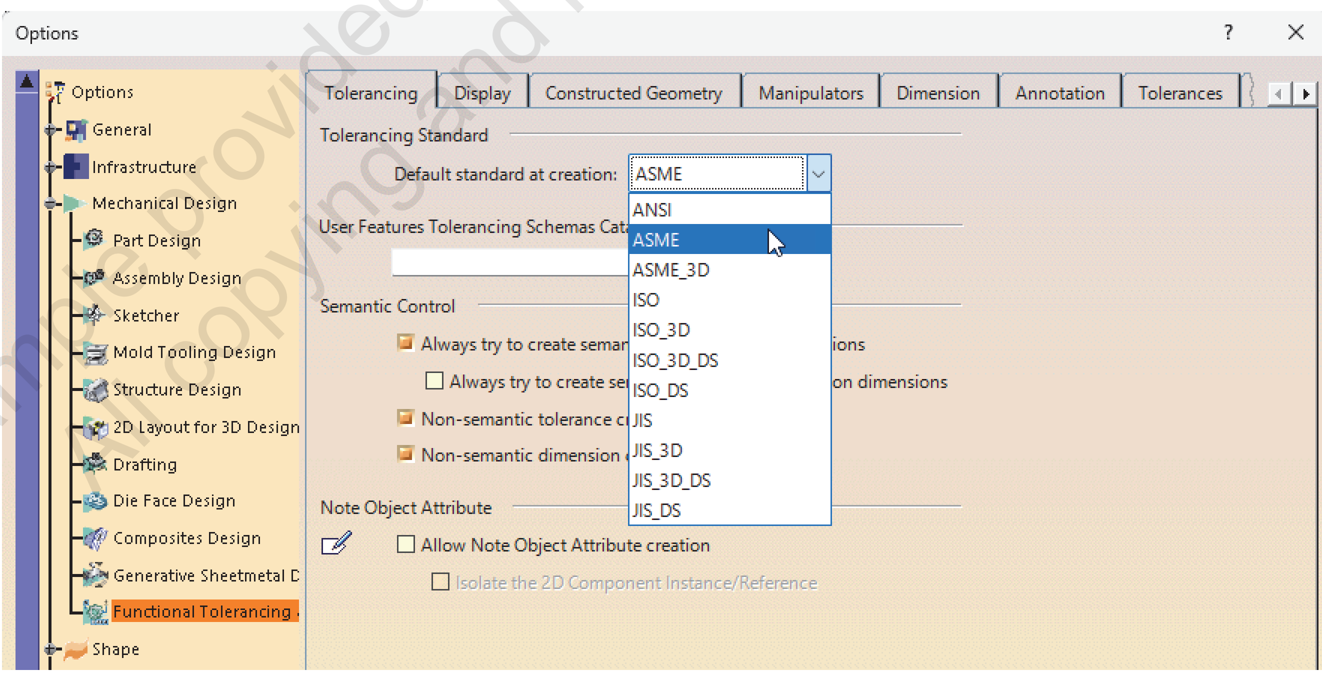


Figure 1–8

New models are created using the default standard. To change the standard for existing annotation sets, right-click on the **Annotation Set.1** branch of the specification tree and select **Properties**. Use the drop-down list in the *Standards* tab of the *Properties* dialog box to change the standard for a model, as shown in Figure 1–9.

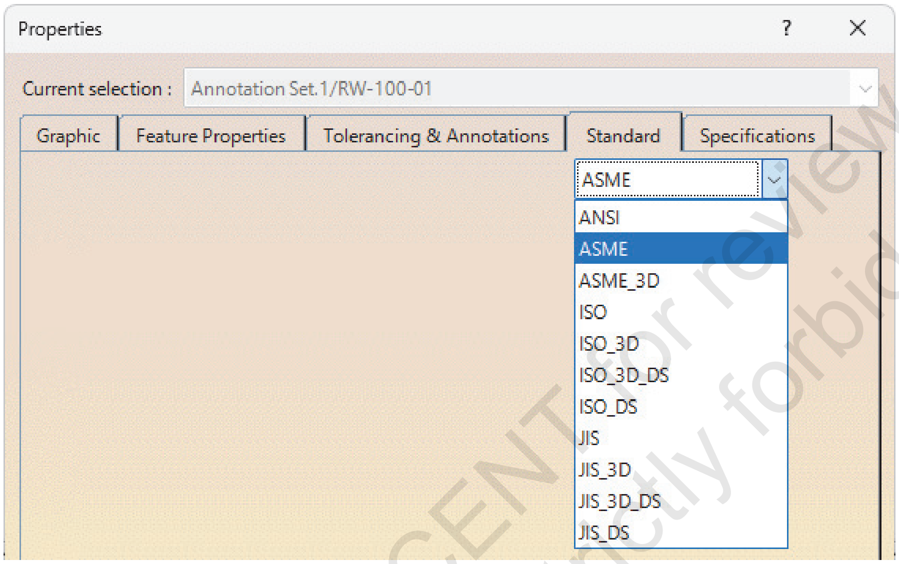


Figure 1–9

Step 2 - Define the annotation planes.

Before creating any annotations, define an annotation or view plane by referencing any planar face or reference plane in the model. The view plane provides an orientation reference for the annotation. When creating new annotations, they are associated with the active view plane. This enables annotations to be selected based on the view plane that was active during their creation.

The top face of the part shown in Figure 1–10 has been defined as an annotation plane.

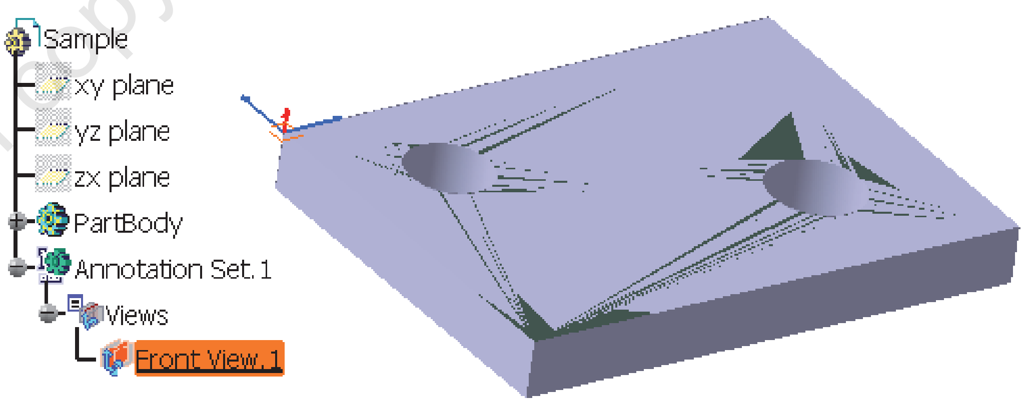


Figure 1–10

Active View

The active view plane is identified by a red border on the model. In the specification tree, the active view displays with a red plane and it is underlined, as shown in Figure 1–11. If no view planes exist, one is created that accommodates the annotation being defined.

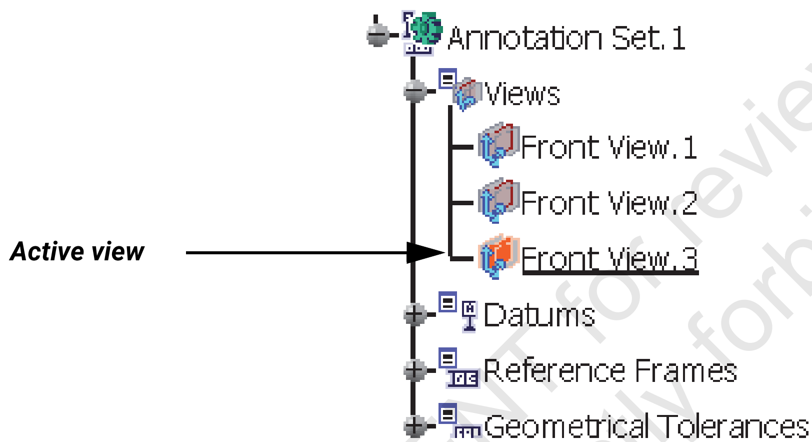


Figure 1–11

To activate a view plane, right-click on the view in the specification tree and select **Activate View**. You can also activate the view by double-clicking the view frame on the model.

Step 3 - Create dimensions and tolerances.

Dimensions and tolerances should be added as early as possible to support the datum reference frames and geometrical tolerances that are added to the model.

Dimensions can be toleranced using one of the following methods:

- General Tolerance (e.g., 90)
- Numerical Values (e.g., 90 ± 0.1)
- Tabulated Values (e.g., 90 H7)
- Single Limit (e.g., 90 MIN)

Some dimensions and tolerances have been added to the part, as shown in Figure 1–12.

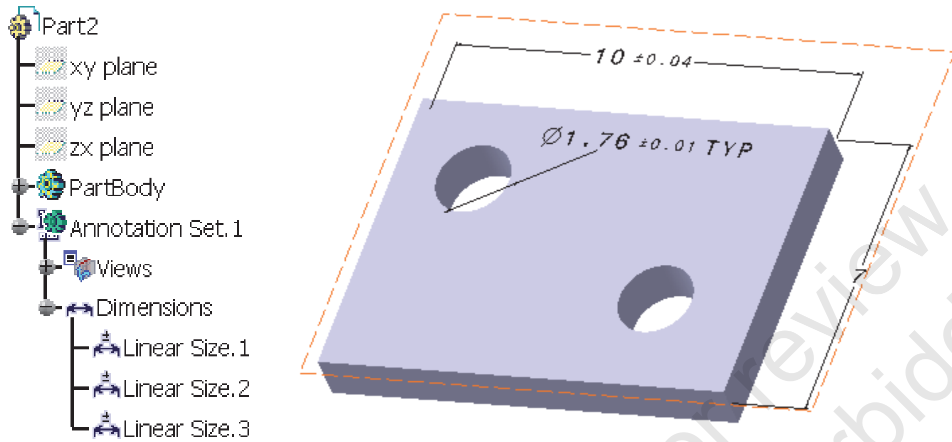


Figure 1–12

Step 4 - Create datum reference frames.

The next step is to define the semantic datums for the model. They can be defined on planar faces or reference planes in the model. If a non-planar surface is selected, datum target points must be defined to establish the datum.

Datum reference frames (DRF) provide the basis for all geometrical tolerancing in the model. The DRF's are developed from a selection of semantic datums. Once created, DRF's are added to the specification tree and the *Semantic Tolerancing Advisor* dialog box.

> ***Note:** In CATIA, Datum reference frames are also called Datum systems. The terms are used interchangeably.*

Three semantic datums (A, B, and C) and a datum reference frame (A|B|C) have been defined for the part model, as shown in Figure 1–13.

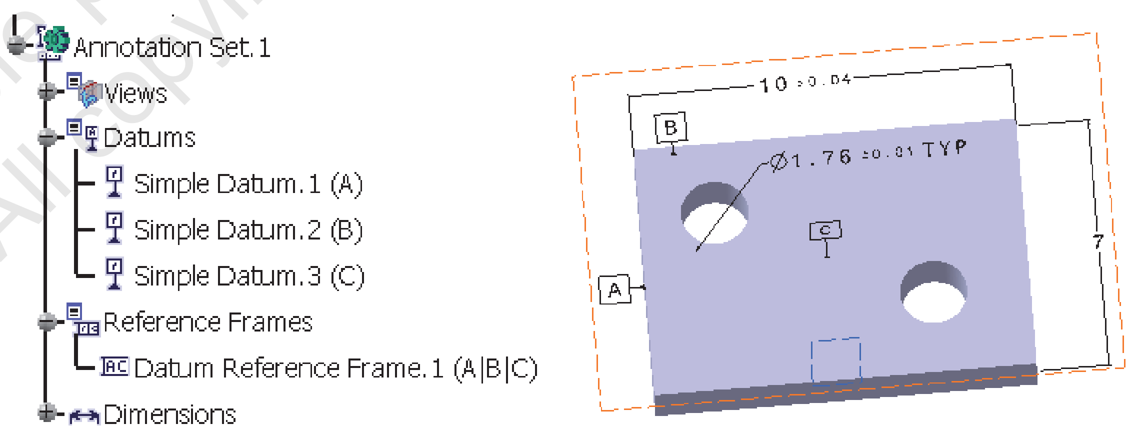


Figure 1–13

Step 5 - Create geometrical tolerances.

Since the geometrical tolerance must be constructed from one of the existing datum reference frames, they can only be added once the DRF's have been defined. A variety of geometrical tolerance types and conditions can be created in both semantic and non-semantic formats. A position tolerance has been added to the diameter dimension, as shown in Figure 1–14.

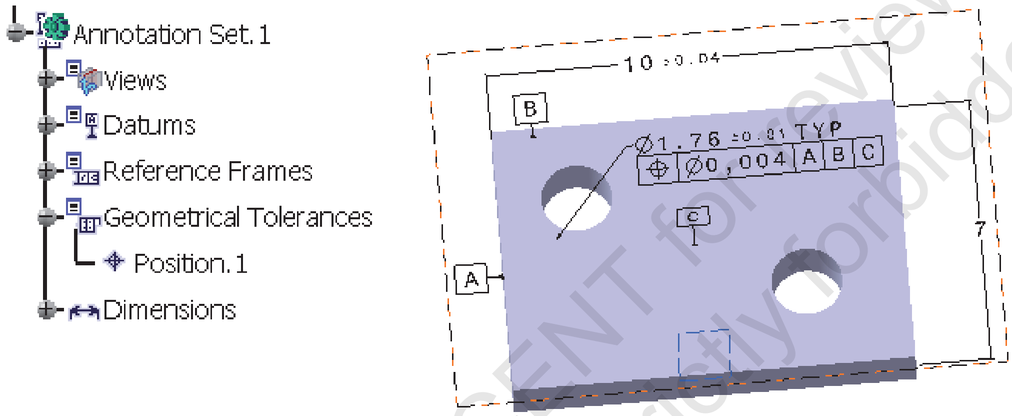


Figure 1–14

Step 6 - Create basic dimensions.

Basic dimensions are indicated by a frame or box around the dimension value. Since a basic dimension specifies a theoretically exact value, it must be associated with a datum reference frame to be created. A basic dimension has been added to the model, as shown in Figure 1–15.

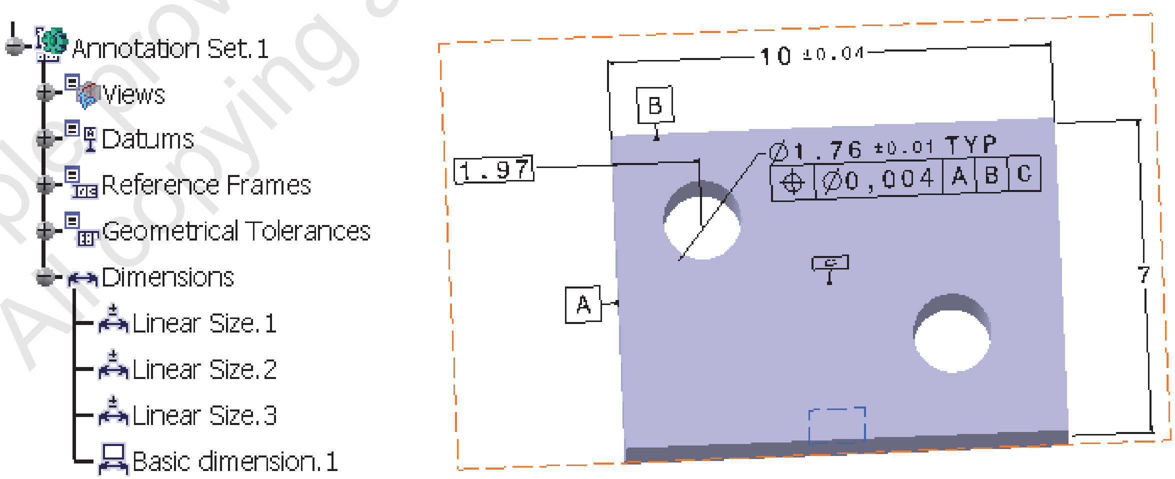


Figure 1–15

Step 7 - Create additional annotations.

The last step of the process is to complete the annotation of the model. The following list indicates some of the different types of additional annotations that can be defined:

- Text, including notes with leaders and title block information
- Flag notes
- Roughness symbols

A completely annotated model displays, as shown in Figure 1–16.

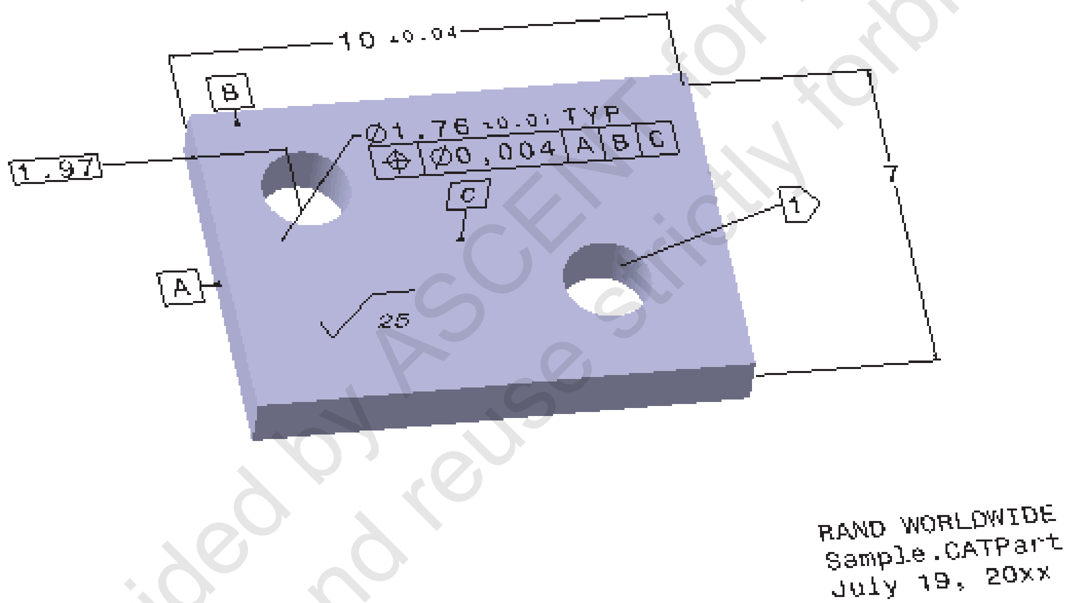


Figure 1–16

Step 8 - (Optional) Extract 2D drawing views.

Once the model has been fully annotated, you can extract drawing views directly from the annotation views.

1.6 Drawing Creation

Using the FT&A workbench simplifies the creation of drawing views. In the same way that drawing views can be extracted directly from the model, annotated views can also be taken directly from the model into the 2D drawing. This ability ensures that all engineering information is stored and extracted directly from the 3D model. It also enables the design and manufacturing process to minimize its reliance on the drawing for output.

General Steps

Use the following general steps to create a drawing using an annotated model.

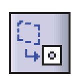
1. Create a new drawing.
2. Extract a view from the 3D model.
3. Align the drawing views.

Step 1 - Create a new drawing.

With the annotated model open, create a new drawing. The tolerance standard and drawing standard must be the same to display an annotated view from a model.

Step 2 - Extract a view from the 3D model.

Before creating any drawing views, tile the windows (recommended) so that selections between the model and drawing can be facilitated. Select **Window>Tile Horizontally**.

In the *Projections* toolbar of the Drafting workbench, click  (View from 3D). Expand the specification tree for the annotated model and select an annotation plane in the Views branch. The system previews the annotated view in the drawing window. Position and orient the view using the drawing compass and then select anywhere in the drawing background to generate the view.

A drawing with multiple annotated views displays, as shown in Figure 1–17.

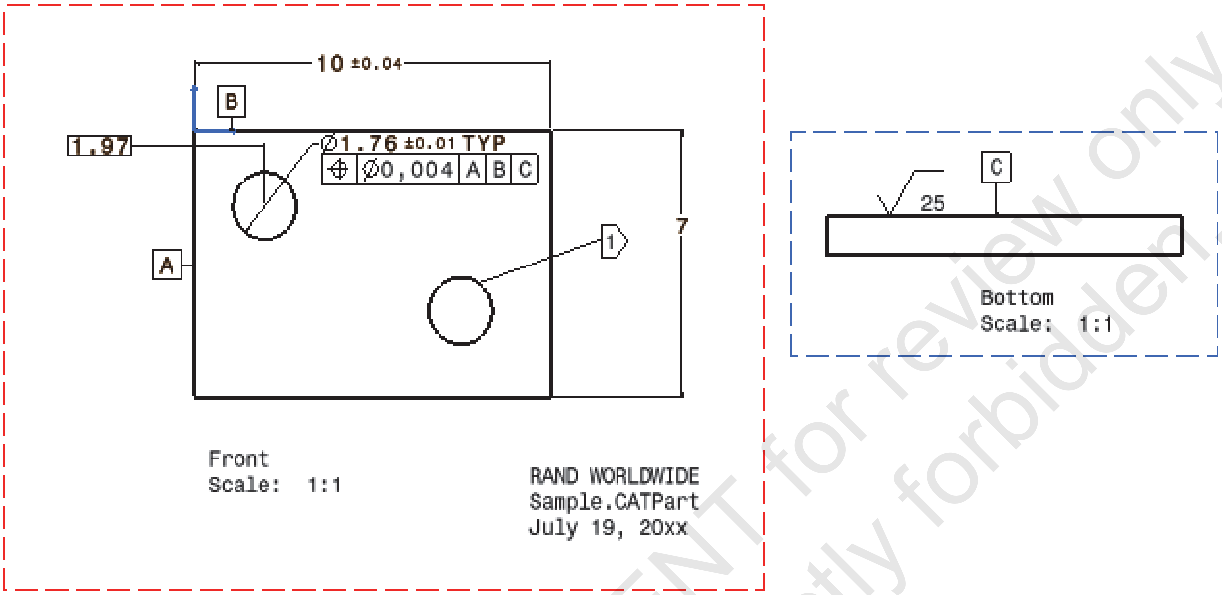


Figure 1–17

Step 3 - Align the drawing views.

By default, drawing views added from the model are not aligned correctly.

How To: Use the following steps to align the views:

1. Right-click on the border of the drawing view that moves into an aligned position and select **View Positioning>Align Views Using Elements**.
2. Select an element from each view to be aligned. For example, two edges are selected from the drawing views, as shown in Figure 1–18.

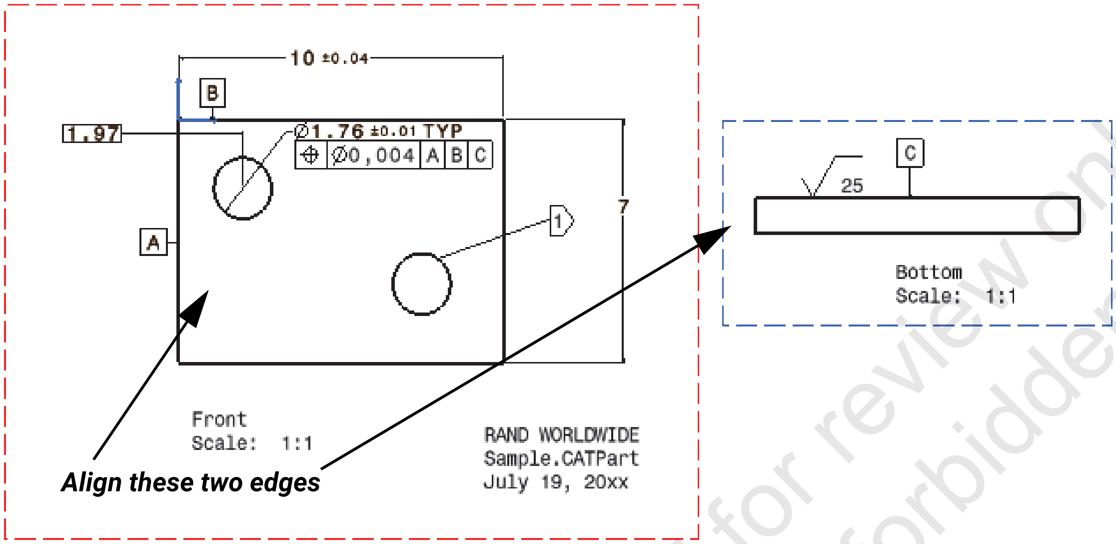


Figure 1–18

The aligned views display, as shown in Figure 1–19.

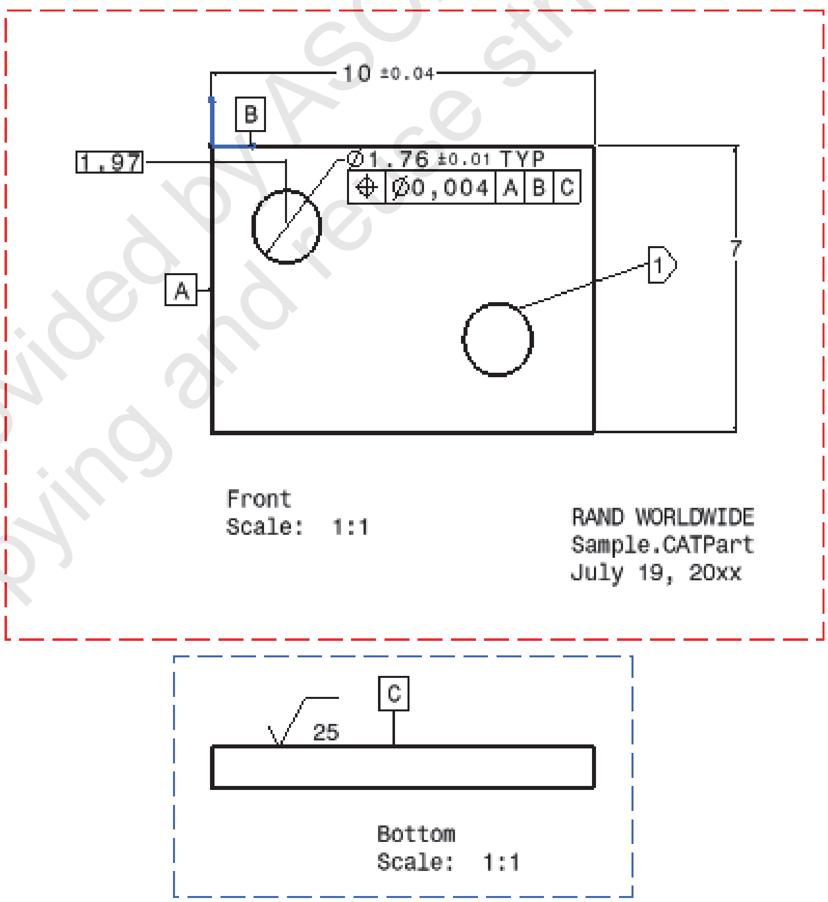


Figure 1–19

1.7 Accessing Annotations

The annotation set that is created to store all elements created in the Functional Tolerancing & Annotation workbench is viewable outside this workbench. Therefore, only the designers that are applying annotations require a license for FT&A. Anyone that is reviewing annotations and tolerances can do so using a Digital Mock-Up (DMU) license or can access them through another workbench that enables you to view CATPart and CATProduct files.

If you are using DMU to review part level annotations, you have to add the part to an empty CATProduct file. CATPart files cannot be directly opened in DMU. An example of an annotated model that has been inserted into a product in the DMU Navigator workbench displays in Figure 1–20.

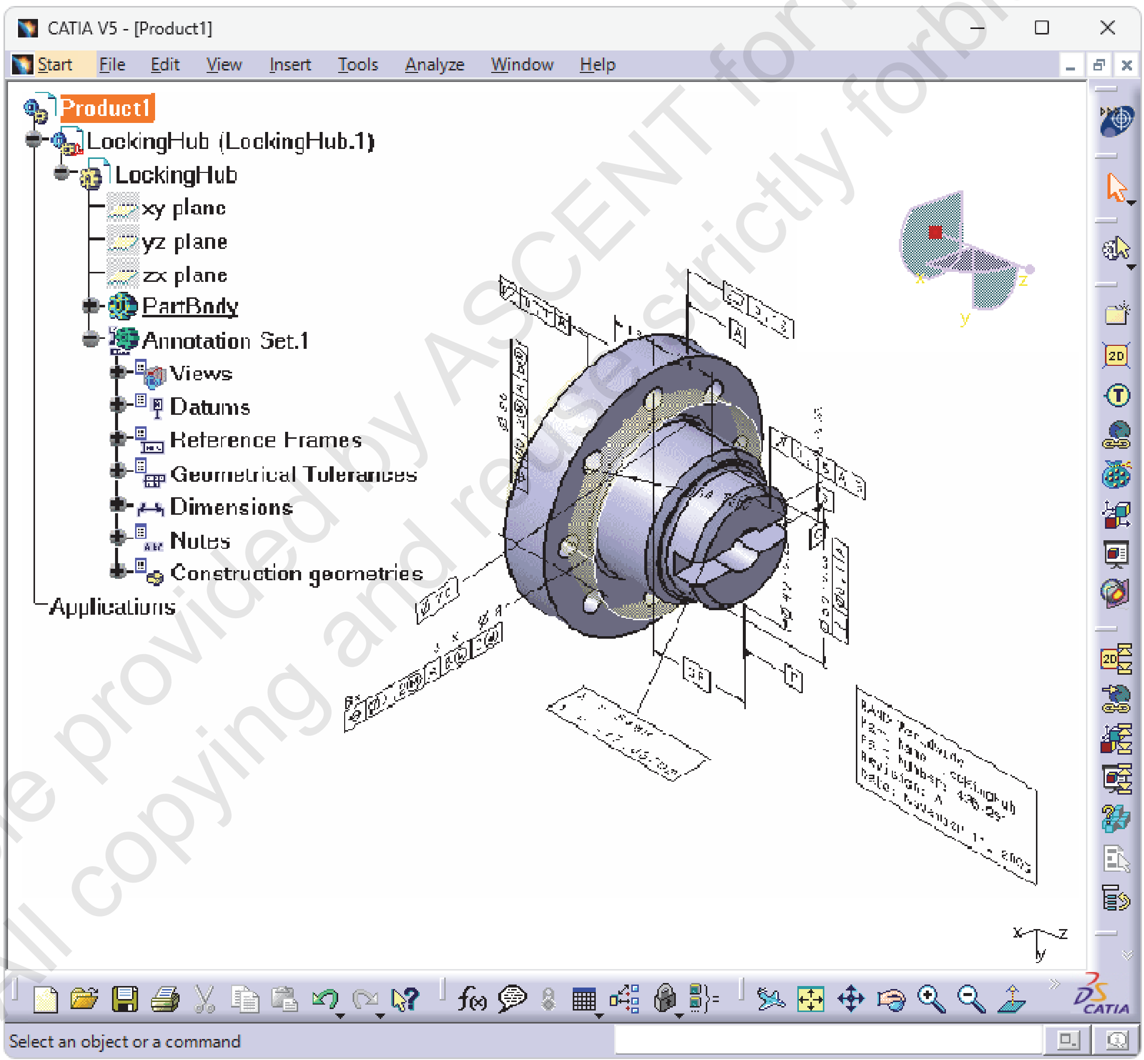


Figure 1–20

Remember that you might have to toggle on the annotation set before it can be viewed in the model. This is done by right-clicking on the annotation set and selecting **Annotation Set Switch On/Switch Off**.

1.8 Annotation References

You must select references when creating annotations. For example, the hole feature is selected to create the dimension and geometrical tolerance annotation shown in Figure 1–21. A parent-child relationship now exists between the hole and the annotation so that the annotation would fail if the hole were deleted.

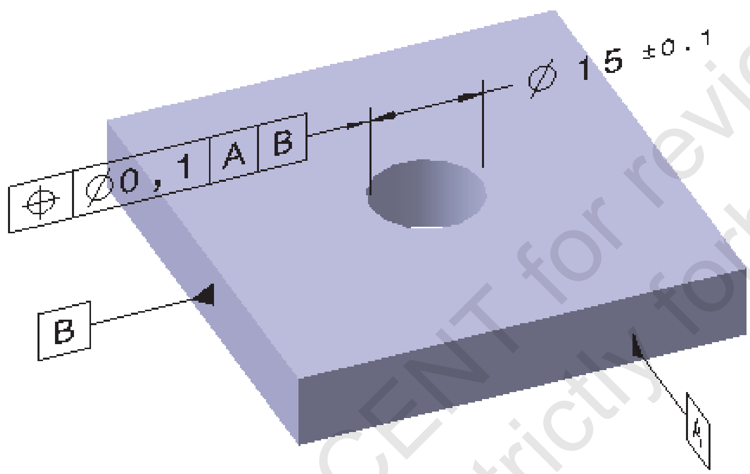


Figure 1–21

3D Annotation Query

The 3D Annotation Query tool () is used to identify the connection between a selected annotation and the rest of the model by highlighting elements on the model and in the specification tree when enabled. For example, the system highlights the hole feature if the toleranced dimension is selected, as shown in Figure 1–22.

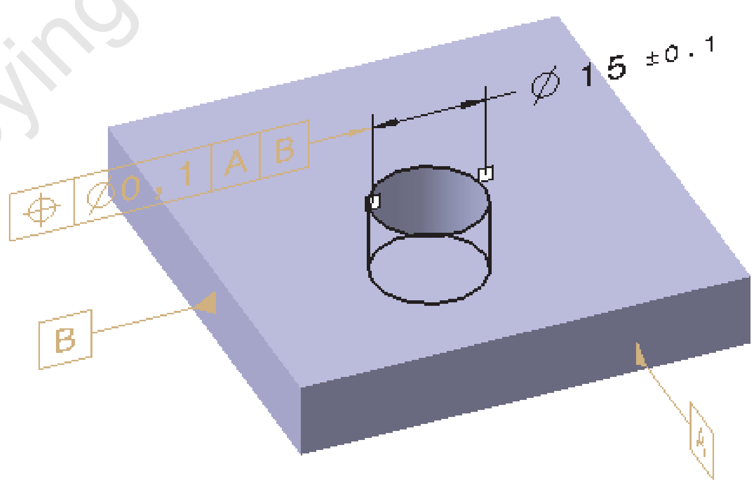


Figure 1–22

The 3D Annotation Query also considers the relationships between annotations. For example, if the geometrical tolerance is selected, the system highlights the hole feature and the Datum A and B annotations that are referenced by the geometrical tolerance.

By default, the **3D Annotation Query** tool is active. The icon is toggled on and off by selecting it in the *Visualization* toolbar, as shown in Figure 1–23.

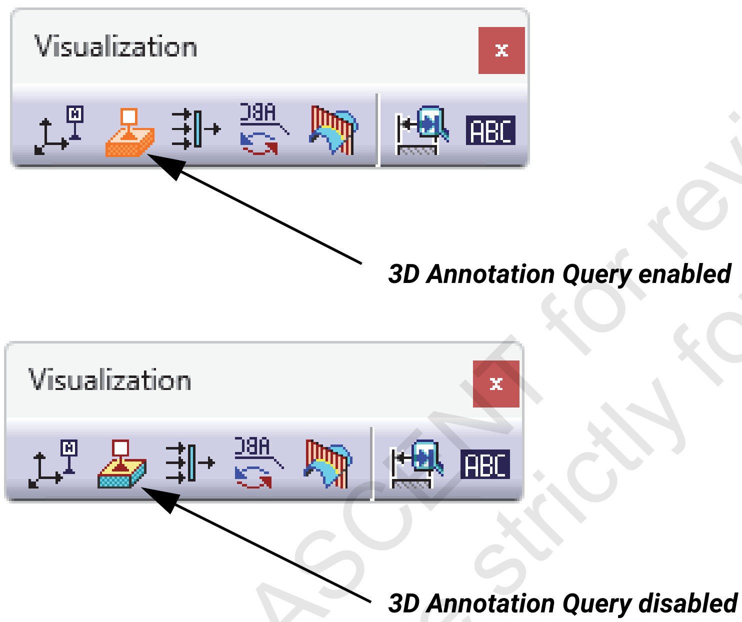


Figure 1–23

Practice 1a
FT&A Overview

Practice Objectives

- Review an annotated model.
- Activate a view.
- Create a note.
- Create annotated drawing views from a 3D model.

In this practice, you will review a model that contains a variety of tolerances and annotations. The intent of this practice is to become familiar with the FT&A workbench and the different types of annotations that can be created.

After reviewing the model, you create a new drawing file and extract annotated views from the 3D geometry. The intent is to reinforce the concept that all engineering information must be stored and extracted directly from the 3D model. This enables the design and manufacturing process to minimize its reliance on the drawing for output.

The model and completed drawing displays, as shown in Figure 1–24.

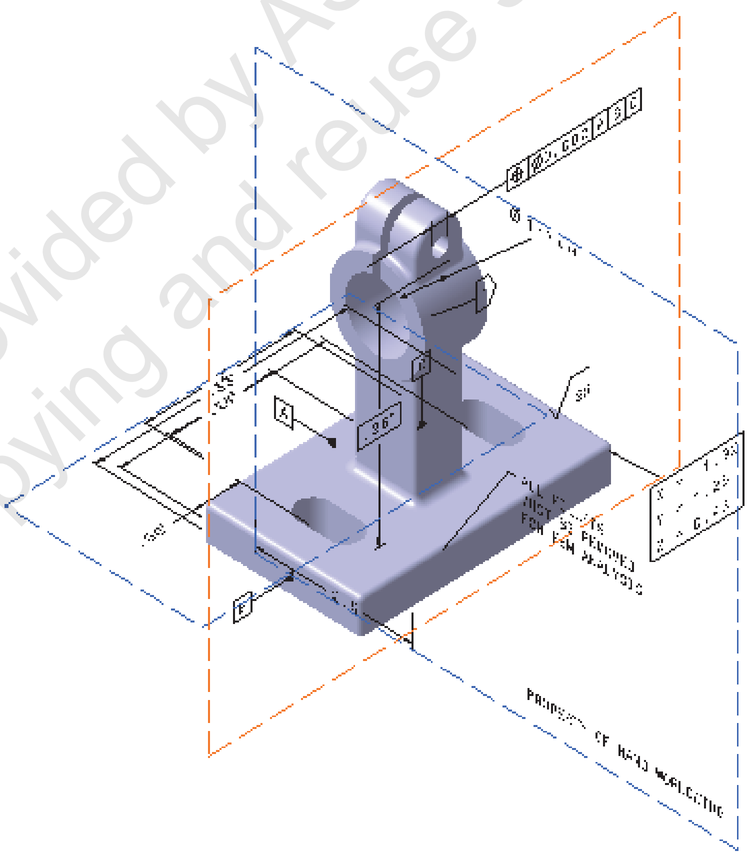
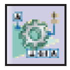


Figure 1–24

Task 1: Open a part file.

1. Open **FlangeLock.CATPart**.
2. Verify that you are in the Functional Tolerancing & Annotation workbench. The workbench icon should be . To access the workbench, select **Start>Mechanical Design>Functional Tolerancing & Annotation**.
3. Select **Tools>Options>General>Parameters and Measure** and select the *Units* tab.
4. Select **Inch (in)** from the *Length* drop-down list, as shown in Figure 1–25.

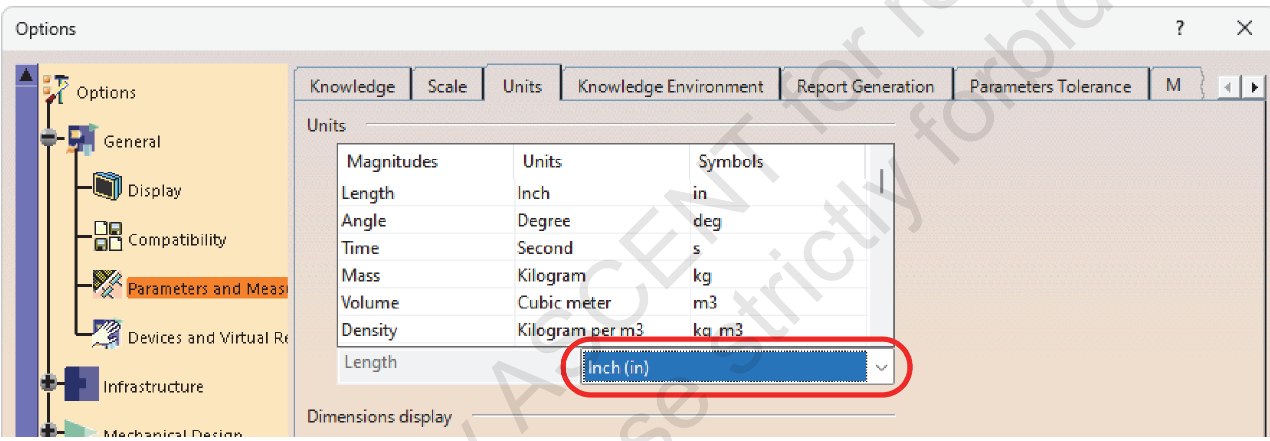
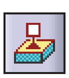


Figure 1–25

5. Click **OK**.
6. By default, the 3**D-Annotation-Query Switch On/Switch Off** icon is enabled. This icon highlights geometry and reference frames that are associated to the selected annotation. If enabled, the icon highlights in orange. For now, disable this option by clicking  (3D-Annotation-Query Switch On/Switch Off) in the *Visualization* toolbar.

Task 2: Review the annotations.

The presence of FT&A annotations in the model can be detected by the Annotation Set branch in the specification tree. You can toggle the display of annotations on and off to simplify the display of the model. Currently, the display of annotations is toggled off.

1. Right-click on **Annotation Set.1** in the specification tree and select **Annotation Set Switch On/Switch Off**. Annotations display on the model, as shown in Figure 1–26.

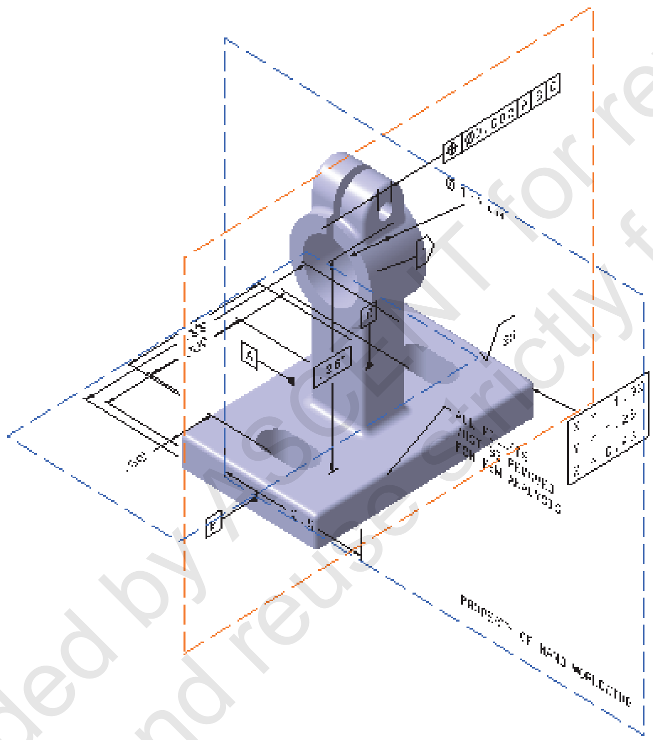


Figure 1–26

2. Expand the **Annotations** branch. A variety of annotations have been created in the model. Each annotation is organized in the tree by type, as shown in Figure 1–27.

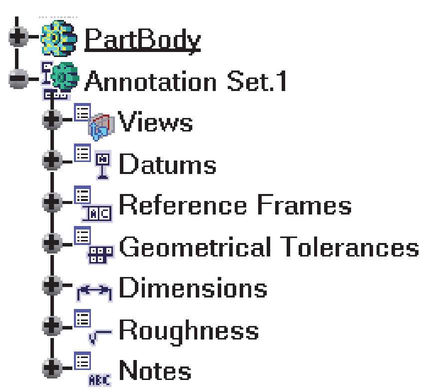


Figure 1–27

3. Expand the different branches of the specification tree and select a variety of annotations to become familiar with the different types. As tolerances are selected in the tree, they highlight on the model.

4. Click  (3D-Annotation-Query Switch On/Switch Off) so that it is enabled.

5. Select the **Position.1** tolerance, as shown in Figure 1–28. It also highlights the geometry associated to the tolerance (Datum Reference Frame A|B|C and Hole.2).

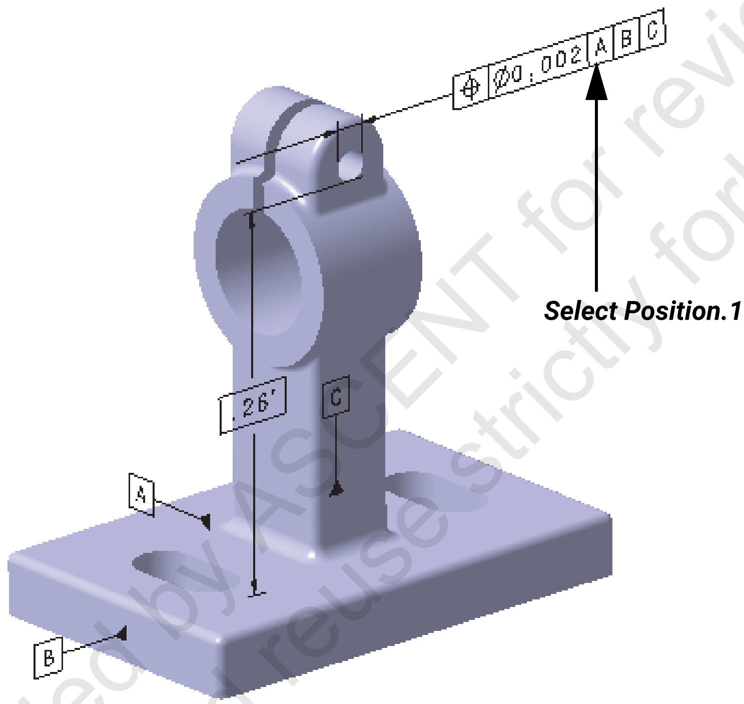


Figure 1–28

Note: *The other annotations are hidden in Figure 1–28 to simplify the display of the model.*

Task 3: Modify the model.

The dimensional annotations are associative to the geometry on which they are based. In this task, you modify the Multipad base feature and then update the model to view changes to the associated annotations.

1. In the specification tree, expand **PartBody** and then **Multipad.1**. Double-click on **Sketch.1**. The sketch for the multipad feature displays.

2. Double-click on the **2.5** dimension, as shown in Figure 1–29.

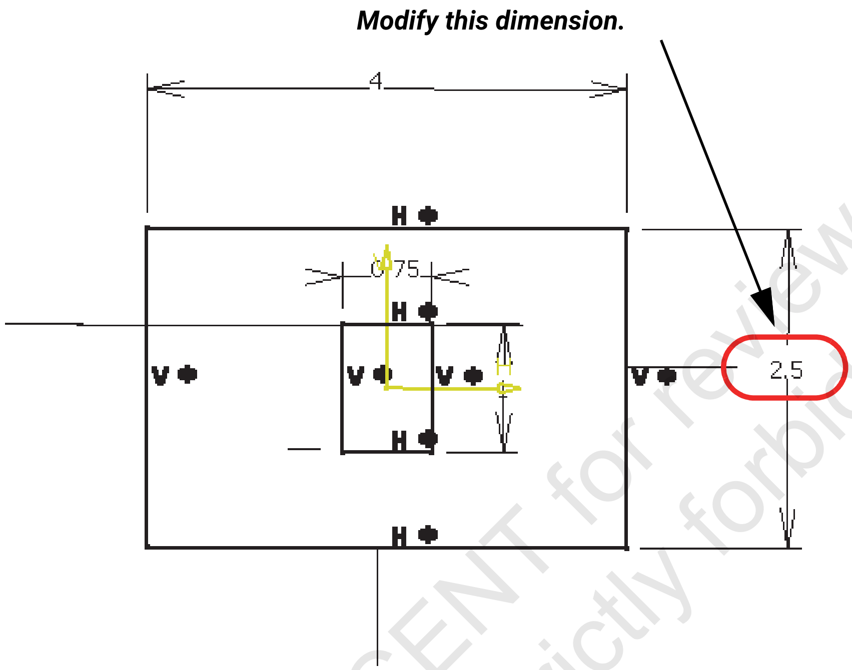
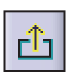


Figure 1–29

3. Enter a new value of **2.6** and click **OK**.

4. Click  (Exit workbench) to exit Sketcher. The system updates all geometry and annotations based on the design change, as shown in Figure 1–30.

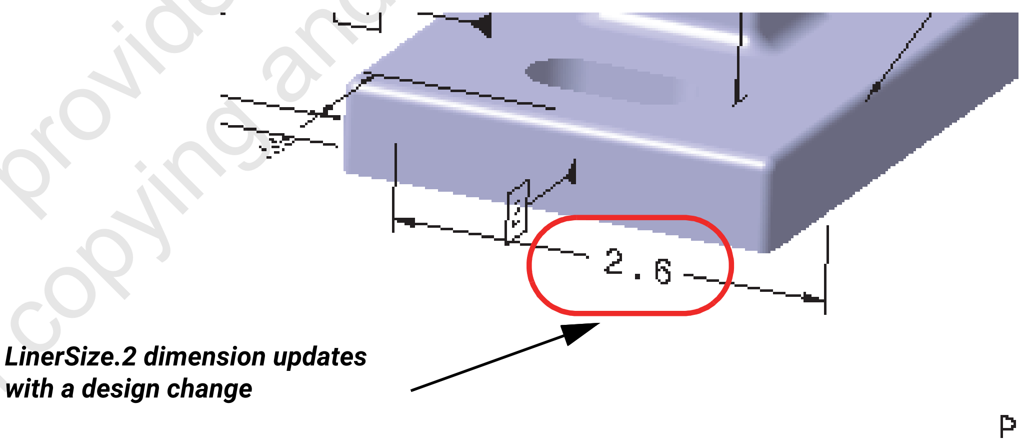
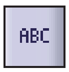


Figure 1–30

Task 4: Activate a view and add a note.

The frame of the currently selected view highlights in orange on the model and in the specification tree. Any annotations created are automatically associated with the activated view.

In this task, you add a text annotation to the model to demonstrate the properties of the active view.

1. Expand the **Annotation Set.1>Views** branch of the specification tree. The active view, **Front View.2**, is indicated by a orange plane in the specification tree and on the model.
2. Double-click on **Front View.1** in the specification tree to activate the view.
3. Click  (Text) and select a location close to the one shown in Figure 1–31.

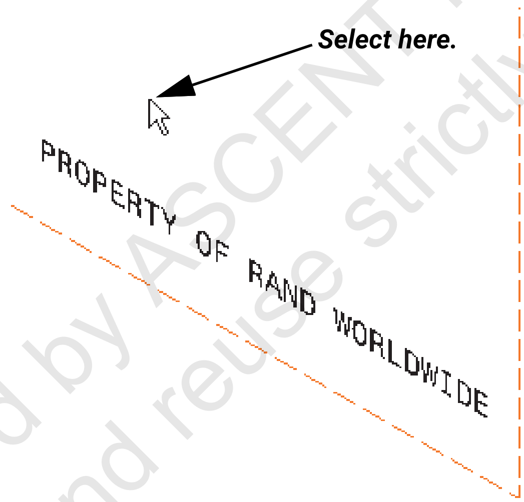


Figure 1–31

The *Text Editor* dialog box opens as shown in Figure 1–32.

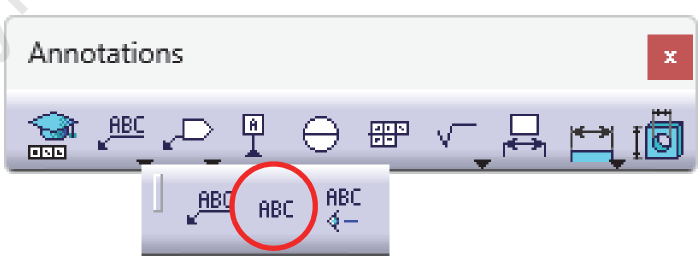
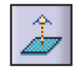


Figure 1–32

4. Enter the text **Material: Aluminum** and click **OK**. The annotation is added as **Text.3** to the **Notes** branch of the specification tree.

5. To orient the model using the view plane, click  (Normal View) and then select **Front View.1** on the model or in the specification tree.
6. Position the text annotation, as shown in Figure 1–33.

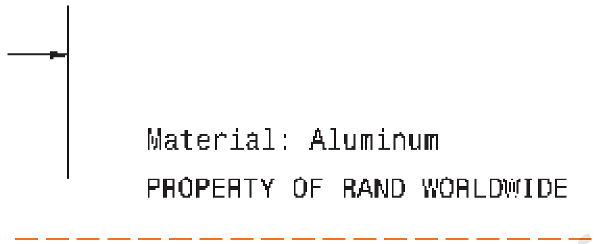


Figure 1–33

Task 5: Create a drawing.

1. Select **Start>Mechanical Design>Drafting**. The *New Drawing Creation* dialog box opens, as shown in Figure 1–34.
2. Create the drawing using the following parameters:
   - *Automatic Layout:* **Empty sheet**
   - *Standard:* **ASME**
   - *Sheet Style:* **C ANSI**

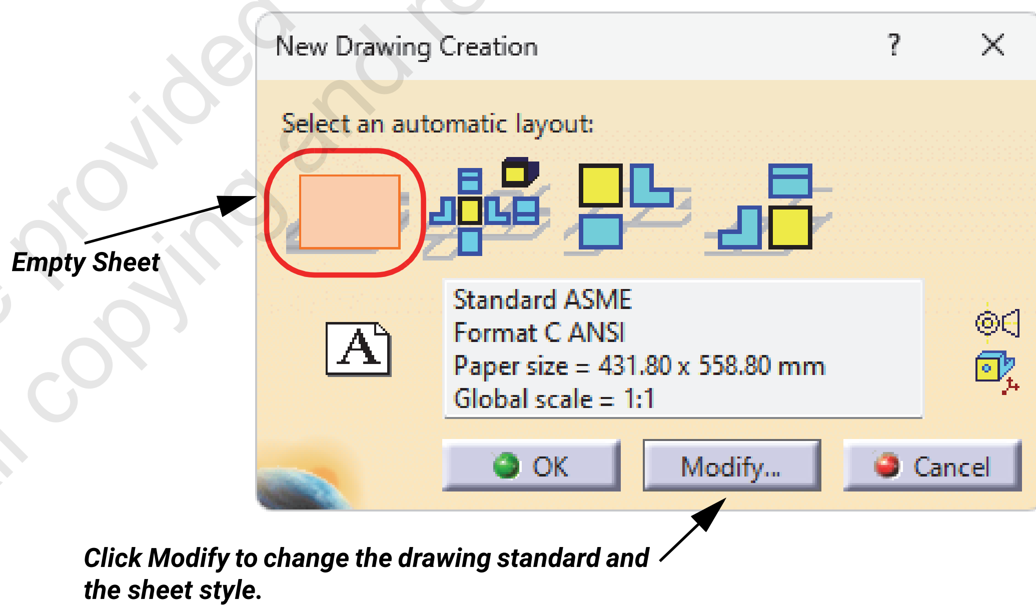
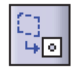


Figure 1–34

3. Click **OK** to create the new drawing.

4. Select **Window>Tile Horizontally** to see the model and drawing simultaneously.

5. In the *Projections* flyout of the *Views* toolbar, click  (View from 3D).

6. Select **Front View.1** in the specification tree in the *FlangeLock.CATPart* window. The system displays this view and all annotations that are associated with it, as shown in Figure 1–35.

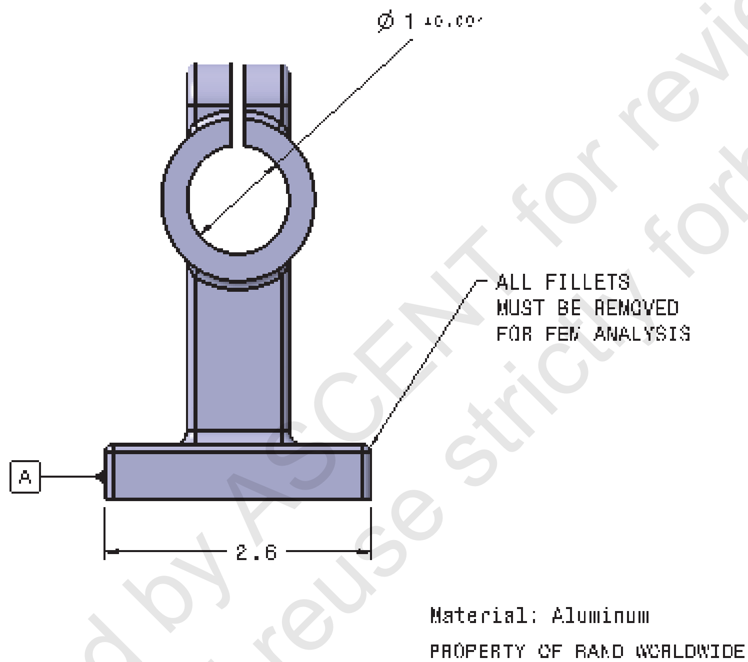
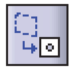


Figure 1–35

7. Click anywhere in the background of the drawing window to complete the view creation.

8. Create two more views using  (View from 3D), selecting **Front View.2** and **Front View.3**. Position the drawing views as shown in Figure 1–36.

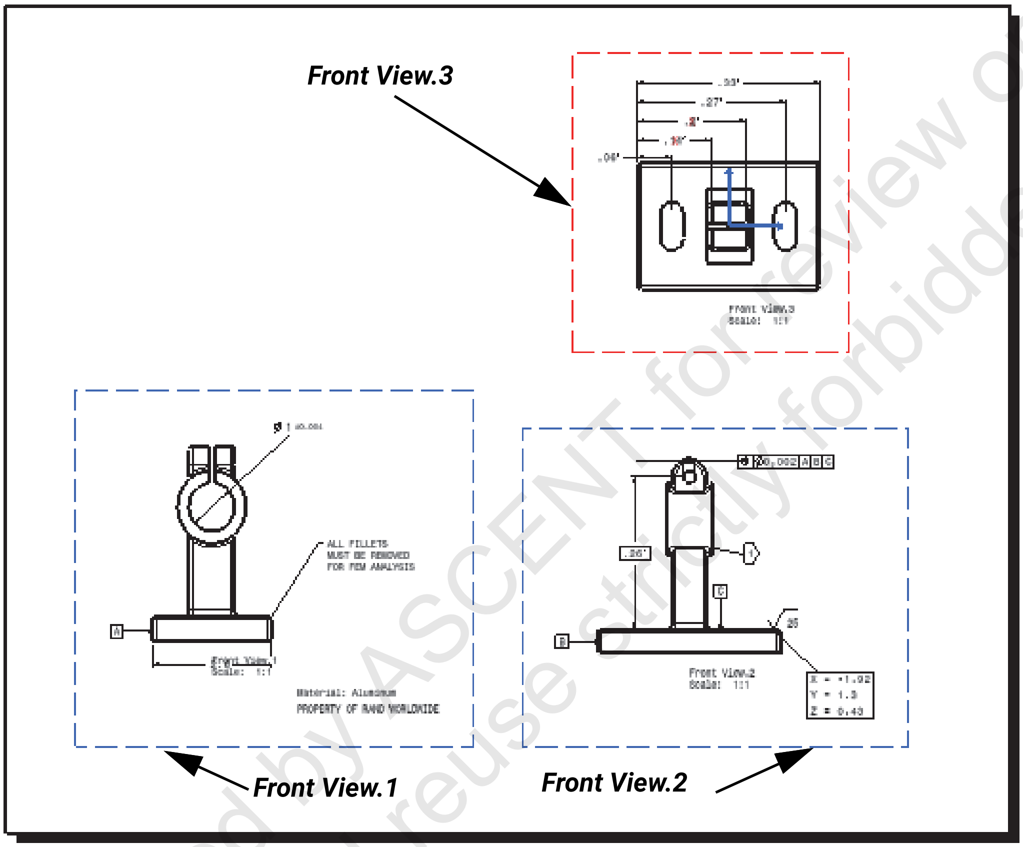


Figure 1–36

***Note:** You can also correct this issue by recreating the dimensions in the 3D model using geometry that displays in the drawing view.*

Two of the dimensions in **Front View.3** have a red x over them. This is because their leaders are linked to geometry that is hidden in the current view. To correct this issue, you display hidden lines in the drawing view.

9. Right-click on the border of **Front View.3** and select **Properties**.

10. In the *Properties* dialog box, in the *View* tab, select **Hidden Lines**, as shown in Figure 1–37.

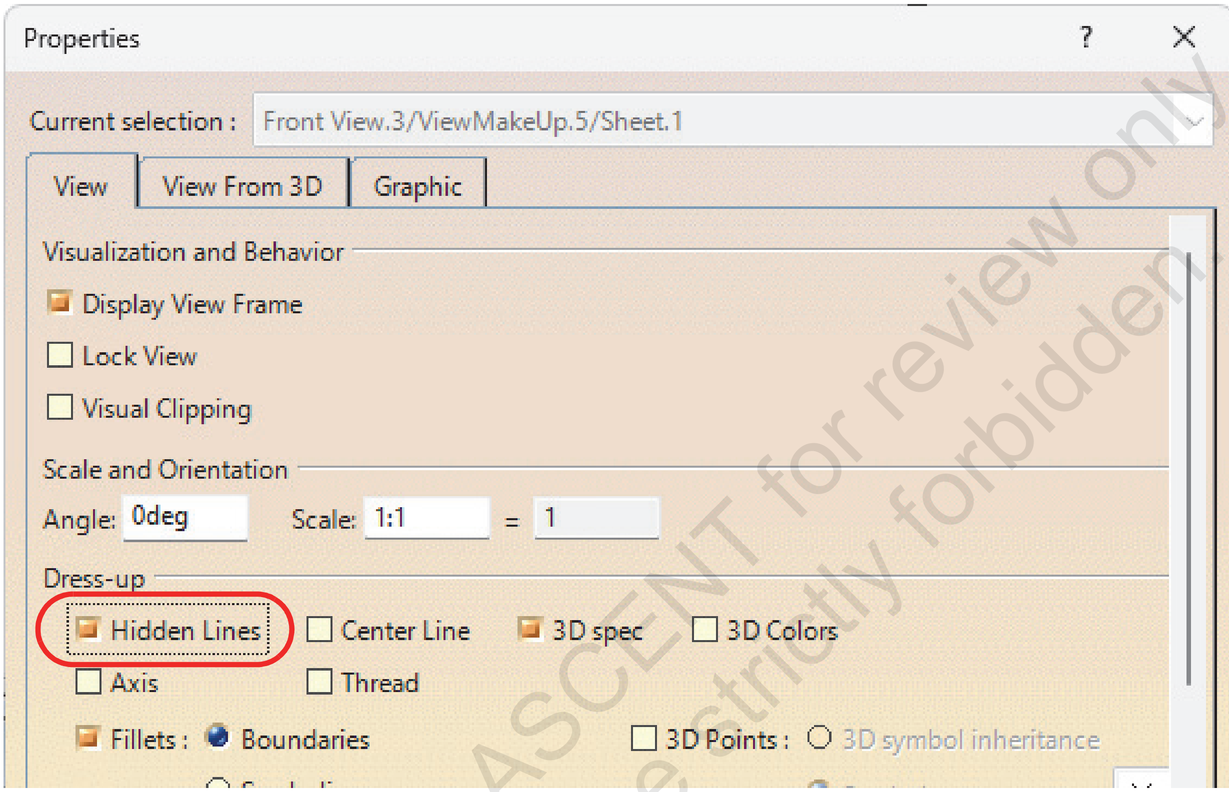


Figure 1–37

11. Click **OK** to close the *Properties* dialog box. The view is updated and the x in both dimensions are now hidden.

Task 6: Align the drawing views.

1. Right-click on the border of **Front View.3** and select **View Positioning>Align Views Using Elements**.
2. Select the entities from **Front View.2** and **Front View.3**, as shown in Figure 1–38.

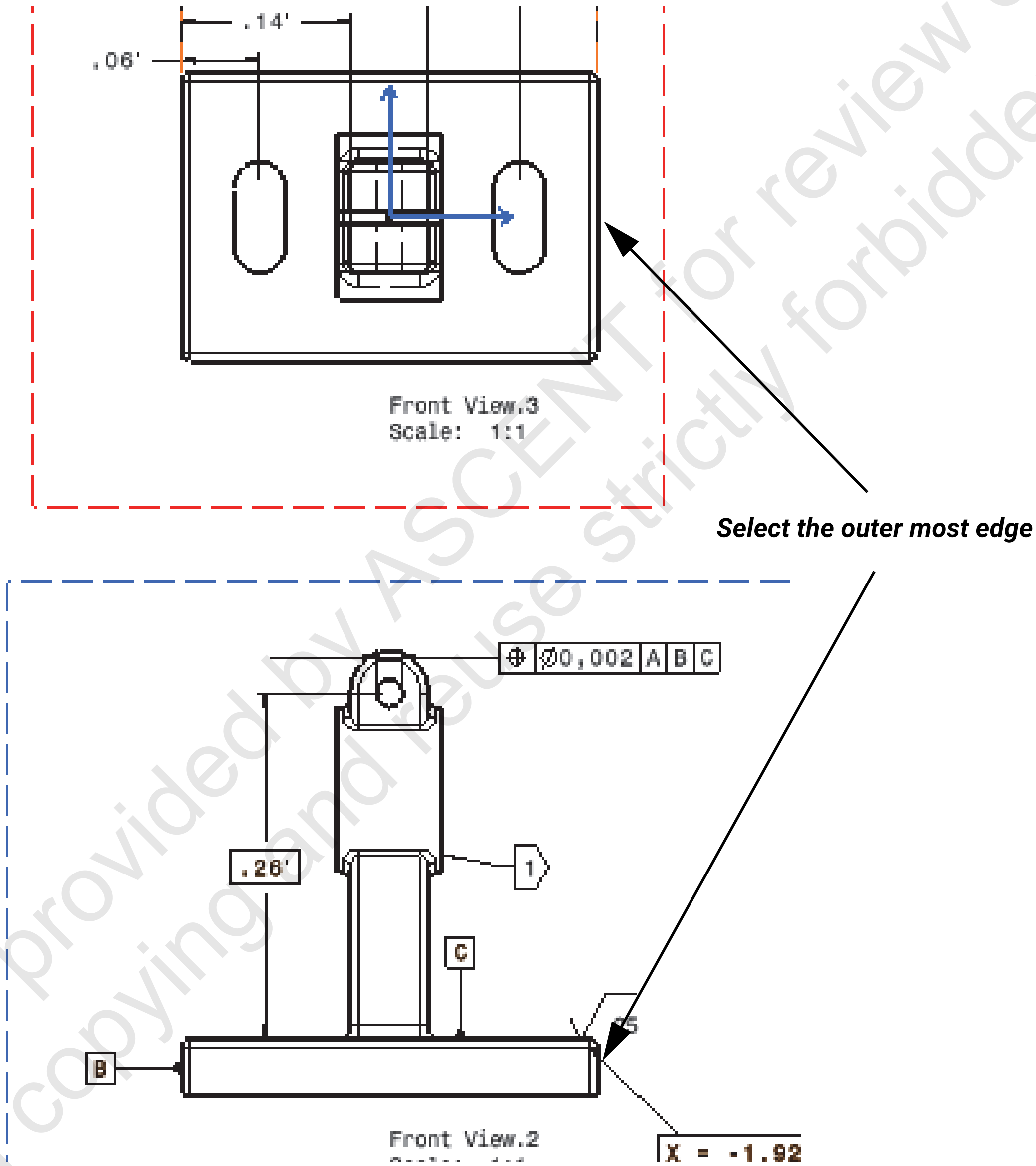


Figure 1–38

3. Repeat this operation to align **Front View.1** and **Front View.2**. The drawing displays, as shown in Figure 1–39.

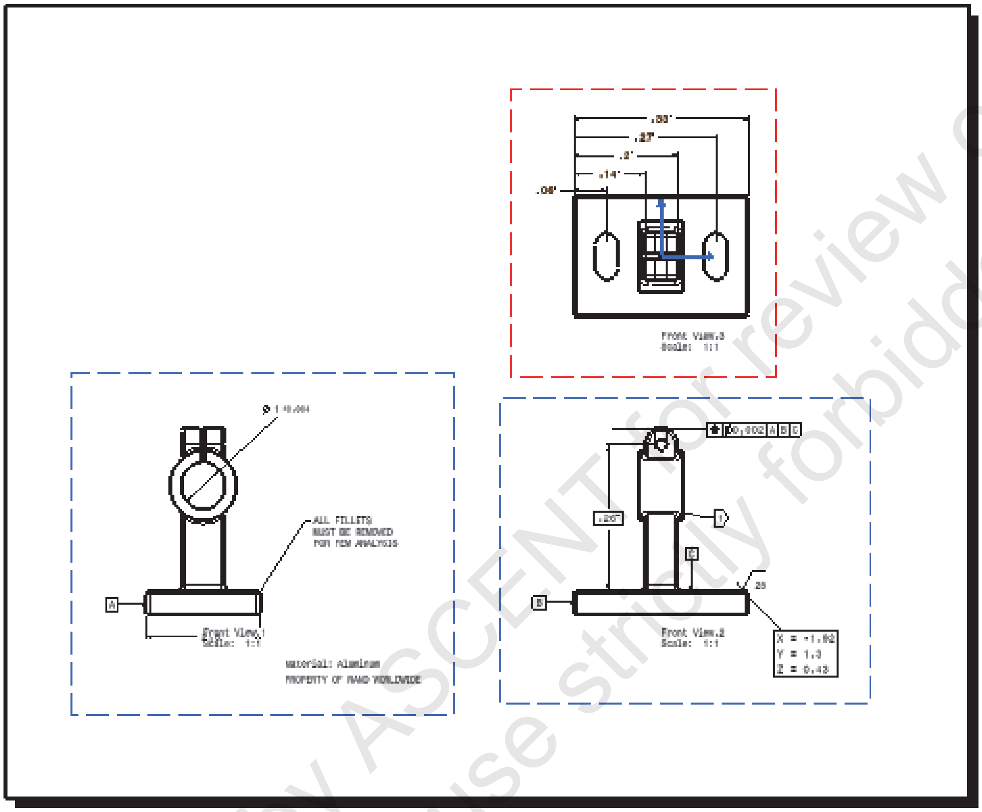


Figure 1–39

Task 7: Transfer an annotation to another view.

1. Activate the **FlangeLock.CATPart** window and maximize the window.
2. Right-click on the **2.6in** dimension and select **Transfer To View/Annotation Plane**, as shown in Figure 1–40.

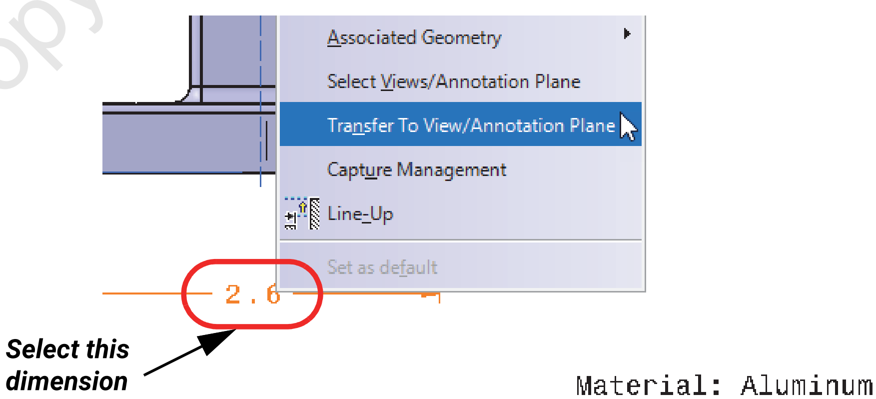


Figure 1–40

3. Select **Front View.3** in the specification tree. The system transfers the **2.6in** dimension to this view plane.
4. Drag the dimension to an appropriate location.
5. Select **Window>Drawing1**.
6. Update the drawing to reflect the modification made to the **2.6in** dimension. **The Front View.3** drawing view displays, as shown in Figure 1–41.

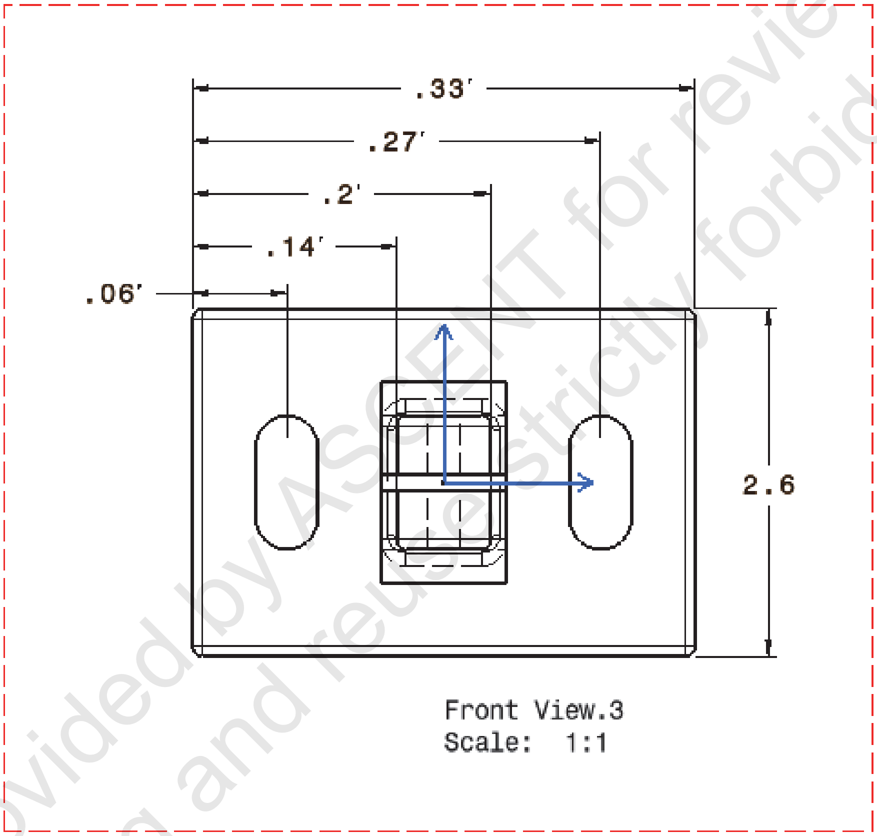


Figure 1–41

7. Save the model and drawing and close all windows.

End of practice