Autodesk® Inventor® 2018
Working with 3D Annotations & Model-Based Definition

Learning Guide
Mixed Units - 1st Edition
ASCENT - Center for Technical Knowledge
Autodesk® Inventor® 2018
Working with 3D Annotations & Model-Based Definition
Mixed Units - 1st Edition

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Preface

Autodesk® Inventor® 2018: Working with 3D Annotations & Model-Based Definition teaches experienced Autodesk Inventor users how to create 3D annotations to support the visual presentation of annotations in 3D PDF format and a Model-based Definition (MBD) workflow.

The geometry designed in a 3D CAD modeling environment is created perfectly. During the manufacturing stage, it is not possible to achieve the same perfection. Variations in size, feature location, and orientation are unavoidable. This learning guide instructs how to use the tools in Autodesk Inventor 2018 to create 3D annotations that communicate dimensional and GD&T data, hold/thread notes, surface texture requirements, and informational text-based annotations; all of which aim to improve manufacturing accuracy. Additionally, this learning guide explains how you can share your 3D annotated models as 3D PDFs, as STEP files for use by other software applications, or in 2D drawing views.

Knowledge of GD&T required.

The international GD&T standard, ASME Y14.5M-1994, governs how annotations should be added to clearly describe the model’s intent. This learning guide assumes that you know how the model is to be annotated and aims to only explain how they are added using the Autodesk Inventor software.

Topics Covered

- Creating dimensional annotations.
- Creating hole/thread note annotations.
- Creating surface texture annotations.
- Creating text-based annotations to a model to communicate additional modeling information.
- Creating tolerance features to a model.
- Using the Tolerance Advisor to review informational messages and warnings on the tolerance features in a model.
- Creating a general profile note annotation.
• Editing 3D annotations in a model.
• Working with Design Views to accurately present the 3D annotations.
• Exporting an annotated model as 3D PDF or STEP 242 files.
• Retrieving 3D model annotations into a 2D drawing view.

Note on Software Setup

This learning 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.

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Lead Contributor: Jennifer MacMillan

With a dedication for engineering and education, Jennifer has spent over 20 years at ASCENT managing courseware development for various CAD products. Trained in Instructional Design, Jennifer uses her skills to develop instructor-led and web-based training products as well as knowledge profiling tools.

Jennifer has achieved the Autodesk Certified Professional certification for Inventor and is also recognized as an Autodesk Certified Instructor (ACI). She enjoys teaching the training courses that she authors and is also very skilled in providing technical support to end-users.

Jennifer holds a Bachelor of Engineering Degree as well as a Bachelor of Science in Mathematics from Dalhousie University, Nova Scotia, Canada.

Jennifer MacMillan has been the Lead Contributor for Autodesk Inventor Working with 3D Annotations & Model-Based Definition since its initial release in 2017.
In this Guide

The following images highlight some of the features that can be found in this Learning Guide.

FTP link for practice files

Practice Files

The Practice Files page tells you how to download and install the practice files that are provided with this learning guide.

Learning Objectives for the chapter

Chapters

Each chapter begins with a brief introduction and a list of the chapter’s Learning Objectives.
1.3 Working with Commands

Starting Commands

The main way to access commands in the AutoCAD software is to use the Ribbon. Most of the sub-commands are available in the Ribbon. Some sub-commands are available in the Command Line or the toolbars or tool palettes. The AutoComplete option automatically completes the entry when you type the first few characters. You may use the AutoComplete feature to search for system variables or to set the delay response time, as shown in Figure 1-12. You can then select the tool and select a command.

You can also click (Command) to display the task settings for the AutoComplete feature.

If you need to store a command, press Enter to cancel. You might need to press F10 more than once.

Instructional Content

Each chapter is split into a series of sections of instructional content on specific topics. These lectures include the descriptions, step-by-step procedures, figures, hints, and information you need to achieve the chapter’s Learning Objectives.

Practice Objectives

Practice Objectives

Estimated time for completion: under 3 minutes.

In this practice you will open a drawing, save it, and modify the AutoLISP script, as shown in Figure 1-31.

1. Open Building Valley.dwg from your class files folder.
2. In the Quick Access Toolbar, click (Save). In the Command Line, the message displays indicating that the AutoCAD software has performed a quick save.
3. In the Application Menu, click (Open) to open the Options dialog box.
4. In the Open and Save tabs, change the time for Automatic save to 15 minutes.

Chapter Review Questions

Chapter Review Questions

1. How do you switch from the drawing window to the text window?
   a. Use the icons in the Status Bar.
   b. Press (Esc).
   c. Press F2.
   d. Press the (pantyp) key.
   e. Press F10.
2. How do you cancel a command using the keyboard?
   a. Press F2.
   b. Press (Esc).
   c. Press (Ctrl).  
   d. Press (Delete).
3. What is the quickest way to repeat a command?
   a. Press (Esc).
   b. Press F2.
   c. Press (Ctrl).
   d. Press (Delete).
4. To display a specific Ribbon panel, you can right-click on the Ribbon and select the required panel in the contextual menu.
   a. True
   b. False
5. How are points specified in the AutoCAD Cartesian workspace?
   a. X value, Y value

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### Command Summary

The Command Summary is located at the end of each chapter. It contains a list of the software commands that are used throughout the chapter, and provides information on where the command is found in the software.

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Chapter 1

3D Annotations in Autodesk Inventor

This chapter introduces the concept of 3D annotations. It teaches you how to use the Annotation tools available in the Autodesk® Inventor® software to support the visual presentation of annotations in 3D PDF format, and how to use annotations to support a Model-based Definition (MBD) workflow.

Learning Objectives in this Chapter

- Describe how 3D annotations are used in a model to help visualize the design in a 3D configuration as opposed to a 2D drawing.
- Describe MBD and how adding 3D annotations to a model supports sharing data that is machine readable, enabling software applications to directly use the data.
- Navigate to and access the available annotation tools to review, edit, and create 3D annotations.
- Review the current active standard used in the part model when 3D annotations are added.
1.1 Introduction to Annotations

The ability to create annotations directly in a 3D model was implemented with the release of Autodesk Inventor 2018. The available tools enable you to incorporate and display valuable manufacturing information directly on the 3D model. The new Annotation functionality supports:

- The creation and display of 3D annotations that can be shared and viewed for visual purposes using a 3D PDF.
- The Model-based Definition (MBD) process for annotating a 3D model with the required design data for manufacturing. The export file (STEP 242) produces a machine readable file that enables software applications and CNC tools to directly use the data.

Figure 1–1 shows an example of an annotated 3D model.

Prior to adding 3D annotations, consider the following:

- Use appropriate dimensions and tolerances when creating the model geometry. Sketch and feature dimensions can be promoted and used as 3D annotations to reduce the need for creating new dimensions.
- Know the intent of the model and, if required, how it will interact with any top level assembly files. This understanding will help when creating GD&T annotations.
1.2 General MBD Overview

Model-based Definition (MBD) is an industry term that refers to the process of creating all design information required to fully and accurately describe a 3D model within its design file. The annotations added to the 3D model supplement the parametric modeling data to fully define how the model is to be manufactured. By including all manufacturing information in the design file, the need for a 2D drawing is eliminated.

The following terms are commonly used when discussing MBD:

- **Product Manufacturing Information (PMI):** The annotations and attribute data included along with the parametric modeling data to create the digital data set.

- **Model-based Engineering (MBE):** The modeling strategy that produces a single master file containing all of a model’s 3D design data.

An early design and documentation method uses a single file, a 2D Drawing, to show the part definition and geometric dimensioning & tolerancing information (GD&T) that is required to manufacture a model. Figure 1–2 shows a drawing containing dimensions and geometric tolerancing information.
Another earlier method, which is still commonly used today, is the two file system. In this method, a 3D file contains the model geometry and the 2D file contains the views, annotations, dimensions, and GD&T data, as shown in Figure 1–3.

Today, using an MBD workflow in an MBE modeling environment produces a single master file that contains all of a model’s 3D design data, as well as the PMI information required to manufacture it. MBE is being implemented in many industries to help get better products to market faster, with fewer costly errors. Benefits include the following:

- MBE reduces duplication of effort and prevents contradictory information from occurring between the 3D model and the 2D drawing. Ultimately, it can lead to paperless documentation.
- MBE reduces the amount of data created, stored, and tracked. Additionally, Product Lifecycle Management (PLM) is simplified by only having to manage a single file.
- Quality Control and Part Inspection information is easily relayed.
- MBE easily adapts to the First Article Inspection (FAI) process. This is the process of inspecting a manufactured part to the required GD&T data and then producing a documented report for quality control purposes.
• MBE provides for collaboration across an entire business: Engineering/Design, Manufacturing, Quality Control & Inspection, Project Management, etc., as shown in Figure 1–4. All of these groups (as shown in ) can easily view their required data.

**Figure 1–4**

** Governing Standard:**

The American Society of Mechanical Engineer standard Y14.41 (ASME Y14.41) defines the requirements for Model-based Definition in a CAD software and for users creating products using 3D modeling tools.
1.3 The Annotate Interface

To add 3D annotations to an Autodesk Inventor part model, you can use the tools available in the Annotate tab. These tools (shown in Figure 1–5) enable you to create the following annotations to support both the inclusion of 3D annotations for visual purposes and an MBD workflow:

- GD&T Annotations (Tolerance Features)
- Dimensional Annotations
- Surface Texture Annotations
- Hole/Thread and General Notes

The annotation tools can only be accessed in the ribbon. There are no context menu options available to initiate their creation.

Model Browser

As 3D annotations are added to the model, they are listed in one of two folders that are automatically added to the Model Browser. These folders (shown in Figure 1–6) divide the dimensional, hole/thread, surface, and note based annotations into the Annotations folder, and the GD&T annotations into the Tolerance Features folder.
Context Menus

There are context menus available when right-clicking on any annotation or tolerance features in the Model Browser. Alternatively, similar context menus are available by right-clicking on an annotation in the graphics window. These menus provide access to additional options that can help place/orient the annotation, set the precision, add text, control its visibility, etc. Figure 1–7 shows the context menus available for radial dimension annotations.

Figure 1–7

Annotation Selection

The Select Annotations selection priority can also be set in the graphics window by pressing and holding the <Shift> key with the right mouse button (RMB).

The Select Faces and Edges selection priority is set. Press and hold <Ctrl> or <Shift> to select multiple annotations at once.

Use the Select Annotations selection priority in the Quick Access toolbar (shown in Figure 1–8) to enable annotation selection only. This also enables you to use the window selection method to select multiple annotations at once.

Figure 1–8
1.4 Active Standard

The Active Standard must be set in each file prior to adding any MBD annotations. This standard defines the units of measure that will be used when annotating the 3D model and the GD&T standards. If the model was created with a 2018 default template, the active style is preset; however, if the model was created in any pre-2018 software release, you must assign the active standard before adding any 3D annotations.

The two options for setting this standard in pre-2018 models are:

- In the Tools tab>Options panel, click (Document Settings). In the Document Settings dialog box, select the Standard tab and select an Active Standard from the drop-down list, as shown in Figure 1–9.

![Figure 1–9](image)

The default setting for a model created with the 2018 Metric part standard template is **ASME-mm**. The default setting for a model created with the 2018 Imperial part standard template is **ASME**.

- As soon as you access any commands in the Annotate tab, you are immediately prompted (as shown in Figure 1–10) to specify the annotation standard that will be used for the part document. Select the standard from the drop-down list and click **OK**.

![Figure 1–10](image)
Once annotations are added in the model, you can return to the Document Settings dialog box to change the annotation standard, if required. If changing from an ASME to an ISO or DIN standard, you are prompted with the Tolerance Standard Mismatch dialog box, as shown in Figure 1–11. This requires you to select whether to delete mismatched model annotations or not.

![Tolerance Standard Mismatch]

**Figure 1–11**

**TIP: 3D Annotation Color**

The active color scheme set for the software controls the color of any created annotations and the selection color when they are selected or modified.

**Supported Environments**

The Annotation tab and its tools are only available in the Part and Sheet Metal environments. Although not currently available for the Assembly environment, you can edit parts in the context of the top-level assembly and gain access to the annotation tools.
Practice 1a: Introduction to the Annotation Interface

Practice Objectives

- Review 3D annotations and become familiar with the annotation tools available in Autodesk Inventor part files.
- Select and edit 3D annotations.
- Relocate 3D annotations in an isometric view.
- Create a Leader Text annotation with parametric data.

In this practice, you will learn how to navigate to the annotation tools, access context menus, and how to select and review existing 3D annotations. To complete the exercise, you will edit an existing dimensional annotation and create a simple Leader Text Annotation. The completed part is shown in Figure 1–12.

Task 1 - Open an existing part file.

1. In the Get Started tab>Launch panel, click (Projects) to open the Projects dialog box.
2. Click **Browse**, navigate to `C:\Autodesk Inventor 2018 3D Annotations Practice Files` folder and select **Annotations.ipj**. Click **Open**. This project becomes the active one. Click **Done**.

3. Open **Flange_Bearing_Intro.ipt**. The model displays with 3D annotations already added, as shown in Figure 1–13.

![Figure 1–13](image)

**Figure 1–13**

### Task 2 - Review the active standard.

In this task, you will review the current Active Standard to ensure that it is set to **ASME**.

1. In the **Tools** tab>Options panel, click **(Document Settings)**. In the Document Settings dialog box, select the **Standard** tab and note that the Active Standard is set as **ASME**, as shown in Figure 1–14. This was preset because the 2018 standard Imperial part template was used to create the model.

![Figure 1–14](image)

2. Close the Document Settings dialog box without making a change to the Active Standard.

The default setting for a model created with the 2018 Metric part standard template is **ASME - mm**.
Task 3 - Review the annotation interface.

In this task, you will review the tools on the Annotate tab, review the automatic annotation scale value, and become familiar with locating annotations in the Model Browser.

1. Select the Annotate tab. The options in this tab are subdivided into panels to help you find commands associated with creating GD&T annotations (Geometric Annotation panel), as well as dimensional, hole/thread notes, and surface textures (General Annotation panel). Additional text-based annotations can be created using the Notes panel.

2. In the Manage panel, note that the annotation scale is set to an automatic value, as shown in Figure 1–15. This value is preset based on the current model size.

![Figure 1–15](image)

3. Use standard orientation commands (ViewCube, Pan, Zoom) to manipulate the orientation and the zoom level of the model. The annotations move with the model and the annotation scale remains unchanged.

4. Return the model to its default Home View using the ViewCube.

5. In the Model Browser, expand the Annotations and Tolerance Features folders, as shown in Figure 1–16.

   - All dimensional, hole/thread note, and surface texture annotations created in the model are listed in the Annotations folder. In this case, seven linear, and a single radial and diameter dimension annotation exist in the model.
   - All GD&T annotations are added to the Tolerance Features folder. Currently, there are five in the model.
6. Hover the cursor over the dimensional entries in the Model Browser to highlight them in the model.

7. In the Tolerance Features folder, hover the cursor over the Planar Surface 1 node. This was the first GD&T annotation added to a planar surface in the model. Note that the surface selected as the planar reference for this GD&T highlights in the model.

8. Expand Planar Surface 1. Hover the cursor over the Feature Control Frame 1 and the Datum Identifier 1 (A) entries. These highlight on the model to identify them.

Task 4 - Review the selection priority options for working with 3D annotations.

3D annotations can be selected directly in the graphics window or they can be selected in the appropriate folder in the Model Browser. When selected in one location, the annotation also highlights in the other to help identify it. In this task, you will select annotations and review the selection priority option, which enables you to easily select multiple annotations.
1. In the Quick Access Toolbar, ensure that (Select Face and Edges) is active as the selection priority. This is generally the default selection priority that is active. This setting also enables you to select 3D annotations.

2. Select any annotation in either of the folders in the Model Browser. Once selected, the annotation is highlighted in the graphics window.

3. To select multiple annotations, press and hold <Ctrl> or <Shift> while selecting.

4. Select in the background in the graphics window to clear the selection.

5. Attempt to drag a selection window around multiple annotations in the graphics window. This cannot be done while the (Select Face and Edges) selection priority is set.

6. In the Quick Access Toolbar, select (Select Annotations) from the drop-down list, as shown in Figure 1–17.

![Figure 1–17](image)

7. Return to the graphics window and drag a selection window around multiple 3D annotations. Using the Select Annotations selection priority, annotations are now selected.

8. Note that with the Select Annotations selection priority active, you cannot select any model geometry. This option is great when manipulating the display of annotations in various Design Views, but should be left at (Select Face and Edges) otherwise.

9. Return the selection priority to the (Select Face and Edges) option.

The Select Annotations selection priority can also be set using a context menu in the graphics window by pressing and holding <Shift> + right mouse button (RMB).

Design Views are discussed in Chapter 4.
Task 5 - Modify the location of annotations.

In this task, you will review the context menus and you will move and reposition existing annotations.

1. In the graphics window, select one of the 4.75 dimensional annotations and drag it into a more convenient location. Select in the background to clear the annotation's selection.

2. Move the other 4.75 dimensional annotation. Once moved, the two dimensional annotations should display similar to that shown in Figure 1–18.

3. In the Model Browser, select Linear Dimension 1 and note that one of the 4.75 dimensional annotations is selected. In the Model Browser, right-click on Linear Dimension 1 and select Toggle Alignment, as shown in Figure 1–19. The alignment of the dimension value flips.
4. Right-click on the other 4.75 dimensional annotation directly in the graphics window. Note that the same context menu options are available. Select **Toggle Alignment** for this annotation. The two dimensional annotations should display similar to that shown in Figure 1–20.

![Figure 1–20](image1.png)

5. Modify the locations of the remaining annotations in the model such that they display similar to that shown in Figure 1–21. To relocate annotations, select them and drag the green dots that display.

![Figure 1–21](image2.png)
Task 6 - Modify and create an annotation.

In this task, you will learn how to make changes using the Edit command and how to create an annotation. This task briefly explains the creation and editing interfaces. These topics will be discussed in more depth later chapters.

1. Edit the **R.500** Radial Dimension Annotation using one of the following methods. The annotation will display as shown in Figure 1–22.
   - In the Model Browser, right-click on **Radial Dimension 1** and select **Edit**.
   - In the graphics window, select, right-click on the **R.500** radial dimension annotation, and select **Edit**.
   - Alternatively, double-click on the annotation in either location.

![Figure 1–22](image)

2. Place the cursor at the beginning of the **Edit Dimension Text** field, as shown in Figure 1–23.

![Figure 1–23](image)

3. Enter **4 X** at the beginning of the field, as shown in Figure 1–24. Include a space before and after the X.

![Figure 1–24](image)

4. Click **✓** to complete the edit and update the annotation.
5. In the Annotate tab>Notes panel, click (Leader Text).

6. Select the top-most circular face on the model as the placement face and locate the annotation, as shown in Figure 1–25.

7. In the Format Text dialog box, enter the text shown in Figure 1–25. Note that the parameter <Material> is added by assigning the Type field to Properties - Model and the Property field to MATERIAL. Once selected, click to add the parameter to the note.

8. Click OK to complete the Leader Text Annotation.

9. Review the Model Browser and note that it was added to the list in the Annotations folder.

10. In the Quick Access Toolbar, change the material to Steel, Carbon and notice that the note updates to display the newly assigned material.

11. Save the model and close the window.
Chapter Review Questions

1. Which of the following statements are true regarding the process of MBD? (Select all that apply.)
   a. MBD is a process used to document a 3D model in a single 2D drawing file.
   b. MBD allows for the creation of Product Manufacturing Information (PMI) that details the requirements for manufacturing.
   c. The American Society of Mechanical Engineer standard Y14.41 (ASME Y14.41) defines the requirements for MBD.
   d. The Step format 203 supports the export of the required PMI data from a model.

2. All of the 3D annotations options available in the Annotation tab can also be accessed in the right-click context menu.
   a. True
   b. False

3. Which of the following are valid 3D annotations that can be created in an Autodesk Inventor part file? (Select all that apply.)
   a. Hole/Thread Notes
   b. Chamfer Notes
   c. Surface Textures
   d. Welding Symbols

4. Which of the following selection priority options enables you to select annotations in your model by dragging a selection window?
   a. 3D Annotations
   b. Select Features
   c. Select Face and Edges
   d. Select Annotations
5. In which of the following locations can you set the Active Standard that defines the standard used to create 3D annotations in a model?
   a. File Menu
   b. Application Options
   c. Document Settings
   d. Projects

6. Part files created prior to Autodesk Inventor 2018 are automatically assigned an Active Standard.
   a. True
   b. False
## Command Summary

<table>
<thead>
<tr>
<th>Button</th>
<th>Command</th>
<th>Location</th>
</tr>
</thead>
<tbody>
<tr>
<td><img src="image" alt="Document Settings" /></td>
<td>Document Settings</td>
<td>• Ribbon: <em>Tools</em> tab&gt;Options panel</td>
</tr>
</tbody>
</table>
| N/A    | Select Annotations | • Quick Access Toolbar: Selection Priority drop-down list  
|        |                   | • Context Menu: pressing and holding the <Shift> key with the RMB in Graphics Window |