AutoCAD Mechanical Design Concepts

This chapter describes the concepts of the different key areas of functionality that assist you in the creation and editing of 2D mechanical drawings.

Objectives

After completing this chapter, you will be able to:

- Describe AutoCAD® Mechanical concepts and key features.
Lesson: Concepts and Key Functionality

Overview

This lesson describes the concepts of the different key areas of functionality that assist you in the creation and editing of 2D mechanical drawings. This includes learning about the following:

- Mechanical structure.
- Using industry and custom standard settings.
- Different geometry creation tools to improve productivity.
- Adding annotation and dimensions that meet your standards.
- Creating detail views.
- Using standards-based features and parts.
- Adding and designing machinery-specific parts such as shafts, springs, belt drives, and cams.
- Checking your design against known stresses.
- Linking and documenting Autodesk® Inventor™ part and assembly files.

When you have a general understanding of the key areas of functionality and how they can help you, it is easier to learn about a command or option from one of these areas.

Objectives

After completing this lesson, you will be able to:

- Describe the purpose and benefit of Mechanical structure.
- Explain how mechanical standards impact the creation of drawing geometry.
- Describe the benefits of using design productivity tools.
- Explain the benefits of using AutoCAD Mechanical commands to add dimensions and annotation to your designs.
- Describe the benefits of using AutoCAD Mechanical commands to create detail views and add balloons, a parts list, a border, title block, and revision table.
- Define standard content and describe the benefits of using it in your designs.
- Explain what parts can be created with the different machinery generators.
- Describe how the calculation tools assist you when creating a design.
- Describe the purpose and benefit of linking Autodesk Inventor models into an AutoCAD Mechanical drawing.
- Explain the purpose of the Configuration and Setup Guide.
Lesson: Concepts and Key Functionality

About Mechanical Structure

When you understand Mechanical structure and how it can benefit you during the creation and modification of a 2D design, you can decide if you want to learn more about the commands and options for creating and leveraging Mechanical structure in your designs.

In the following illustration, two different views of a part were created and defined with Mechanical structure. Because it was defined with structure and the standards were followed, the specific geometry from the front view was easily referenced into a new drawing where it becomes one part in the overall assembly.

Definition of Mechanical Structure

Mechanical structure enables you to logically create and organize your 2D geometry into parts and assemblies. To help you access and reuse this logical organization of information, AutoCAD Mechanical displays the organizational structure in the browser in a hierarchical fashion. The drawing geometry associated with a part is further organized into specific folders for the different views of that part. In the browser, these view folders are nested below the part. Individual parts are nested below the overall assembly or under subassemblies. Using Mechanical structure, you create the same hierarchical part to assembly structure relationship during the design process that you intend to occur during the manufacture and assembly of the product.

After part and assembly data is defined as having Mechanical structure, you can quickly reuse different views in other designs and populate the data in parts lists and bill of materials (BOM). By adding or deleting instances of the same part or assembly to the design, the quantity in the BOM updates to reflect the correct current count.

When it comes time to modify a part or assembly, editing one instance updates all instances of that view. This saves you time from having to edit the part or assembly geometry in each location it is used in your design.

By having Mechanical structure in an assembly file and using associative hide, you can automatically hide the display of part geometry based on the position of other parts.
Example of Mechanical Structure

When creating a design of an assembly, Mechanical structure makes the task of organizing and manipulating the geometry that represents different parts in different views easier and more logical. In the following illustration, three views of an attachment you hook to a ladder to make a work platform are shown with the Mechanical browser. In the browser, the part titled Plank Support is selected. The geometry defined within its three views are shown in dashed lines in the drawing window to highlight them in the three views. This structure enables you to move and rotate each view as if all the objects in that view were a single part object while also enabling you to edit the individual objects in the view.

About Standards-Based Design

To assist in the communication of design data, different industry organizations have established different standards. By learning how to configure and use AutoCAD Mechanical, your drawings conform to these standards as well as any variations specified by your company. In the following illustration, the custom standard called COMPANY STANDARDS is being selected to make it the active standard. This custom standard is initially based on one of the industry standards.
Definition of Standards-Based Design

Standards-based design means you create geometry and annotation that meets industry-accepted standards like ANSI, ISO, and DIN to name just a few. It also means meeting any company-specific variation of those industry standards. Within a standard, there are multiple elements that you can edit so you can achieve the settings specific to your requirements. Customizing an existing standard can include but is not limited to changing what layer geometry is assigned, changing how dimensions are to appear, selecting what welding symbols can be added to the drawing, and defining what information is stored in the BOM. You activate or modify a standard on the AM:Standards tab of the Options dialog box.

To create design data that meets these standards, you use the AutoCAD Mechanical tools in place of the AutoCAD® drawing and modifying tools. You can apply your drawing standards to all new drawings that you create in AutoCAD Mechanical and to previously existing AutoCAD drawing (DWG™) files that are opened in AutoCAD Mechanical.

In the following illustration, the Layer/Objects dialog box shows the properties the different geometry has after you create it. Different objects are mapped to be put on different layers. Because the objects are mapped to a layer you configure to meet your company standards, you can focus on creating the design geometry and not on what layer you are creating the geometry on.
Example of Standards-Based Design

Using the standards-based drafting and design tools of AutoCAD Mechanical, the two views of a spacer plate for planting corn seed were created following both industry standards for notation and company standards for layer settings and use.

About the Design Productivity Tools

You use design productivity tools to focus on creating your design while spending less time focusing on routine drafting tasks such as creating hole patterns, adding centerlines, and adding section lines.

In the following illustration, the holes, centerlines, and section line were added to the part using different design productivity tools. Each of these objects were automatically placed on the correct layer.
**Definition of Design Productivity Tools**

Design productivity tools are commands and options that help you create geometry to represent mechanical designs. By using these tools, the geometry is created on the appropriate layers based on your standard, and your ability to edit the geometry is enhanced.

The design automation tools include tools for the creation of rectangles, hatches, fillets, chamfers, construction lines, symmetric lines, breakout lines, section lines, and centerlines. You also have power commands for dimensioning, creating views, snapping to geometry, copying geometry, editing geometry, and erasing geometry.

In the following illustration, construction lines are projecting from the front view and top view to assist in locating and sizing the geometry for the right side view.

![Construction Lines Illustration](image)

**Example of Design Productivity Tools**

A part of the centerline productivity tool enables you to define and create a pattern of holes. In the following illustration, the circular centerline was created based on the specified diameter and the number or holes dictated the position and angle of linear centerlines. Countersink holes based on industry sizes are now being defined to be positioned at the intersection of these centerlines.

![Centerline Productivity Tool](image)
About Dimensions and Annotation

An important but time-consuming task to completing a design is the adding of dimensions and other annotations. By learning about the benefits of using AutoCAD Mechanical commands to add dimensions and annotation, you can understand the importance of learning how to use those commands to decrease the time it takes you to dimension and annotate your designs.

In the following illustration, the different views of the part were dimensioned using Mechanical-specific dimensioning commands.

Definition of Dimensions and Annotation

The dimension and annotation commands follow industry standards for defining their configuration, contents, and appearance. The layer on which the dimension or annotation resides after it is created automatically follows the settings in the active standard regardless of what layer may be active prior to executing the command. The size of the symbols and text are based on the model scale factor set in the standard.

When you use the power dimensioning command, the type of dimension you create is based on the geometry you select. This enables you to add the various dimension types while using a single dimension command. For each dimension you create, you can quickly add special characters like square, countersink, centerline, depth, as well as tolerance or fit values.

The different dimension and annotation commands also enable you to add a set of baseline, chain, ordinate, symmetrical, and shaft diameter dimensions. You can also notate the position and size of holes by adding hole charts and lists to a drawing sheet. Feature control frames, datum identifiers, weld symbols, and surface texture symbols are a few of the other types of annotations you can add to your drawing. The display of each of these symbols is also controlled by the active standard.

By using one of the three text tools, you create the text using one of three set heights that automatically scale in the model space based on the standard’s model scale factor value.
Example of Dimensions and Annotation

In the following illustration, a drawing view of a cylindrical connecting part is shown with dimensions and other notations. The dimensions in this view specify the size of the part and location of some of the features. Some of the notations for the dimensions were modified from their default during their creation and others were modified after they were placed. The size of the text under the view was set by using two of the three text tools for sizing and creating text.
About Production Drawing Creation

To complete your design and create production-ready drawings, you often need to create detail views, add balloons to parts in an assembly, add a parts list relating to those balloons, insert and document revisions, and insert a border and title block. By having an understanding of how AutoCAD Mechanical assists you in these tasks, you identify the importance of learning more about the commands and options so you can use them to complete your production-ready drawings.

In the following illustration, a completed drawing sheet shows multiple drawing views of an assembly as well as a detail view for greater clarity, part balloons, and a parts list. The numbers in the balloons were automatically populated based on the selected parts. The item numbers in the parts list automatically match the balloon numbers.

![Diagram of a completed drawing sheet showing multiple drawing views of an assembly with balloons and a parts list.](image)

Definition of Production Drawing Creation

Using AutoCAD Mechanical commands, you can create detail views that enlarge small, hard-to-see areas of a drawing view. These detail views reflect the contents of the defined view area at a larger scale factor. Changes in the original view are reflected in the detail view. By using Mechanical structure, the detail views are associative to the view the detail is based upon. This means that when you edit the initial view, the detail view automatically updates to reflect these edits.

When you create an assembly, you typically want to list the parts and subassemblies, the number of them, and their specific properties in a table or chart format. The table you place on the drawing that lists this information is referred to as a parts list. To identify the different items in the drawing views, you add balloons with item numbers that match the item numbers in the parts list. The information for the balloons and parts lists come from a bill of materials (BOM) that is stored in the assembly file. The BOM often contains more information than you want to display on the drawing sheet. The BOM then becomes a central location for entering and editing data and can be exported for use in other software applications.
You have three methods for populating the data in the BOM. One method is to enter all of the data for the entire assembly manually. A second method is to enter the data associated with a part or subassembly into part references. The third method involves entering the values into the component properties for mechanically structured parts and subassemblies.

The second and third methods for populating BOM data are similar. In each of these methods you enter the properties for each unique part or subassembly and then the BOM is automatically populated with that information. By entering the data in the part reference or structured component properties, you can enter the data once and reuse it in multiple assemblies. For all the different assemblies where the part or assembly is used, the BOM is automatically populated with that information. The quantity value in the BOM is then based on the number of instances of a part or subassembly in the overall assembly.

To present your design information in a professional manner, you add a border along the outside edge of the sheet and a title block in one of the corners. The title block communicates specifics about the design and sheet. By using AutoCAD Mechanical commands to place the border and title block, you can easily add data to the title block or change the title block or border to a different size or appearance.

Adding a revision table that follows the active standard also helps communicate when changes to the design have occurred and what those changes entailed.

**Example of Production Drawing Creation**

Drawings used in the production of a design can vary based on the purpose of the drawing. In the following illustration, part of a drawing sheet shows two views of an assembly with some ballooned parts and a section of the parts list. After this drawing is completed, this type of drawing is beneficial to anyone who needs to identify the parts for the purpose of assembly or the ordering of replacement parts.
About Standard Content

When you create a design that includes holes, slots, bolts, or screws, you do not want to spend your time drafting these items. It is a lot more efficient to select them from a list and place them in your drawing. The standard content in AutoCAD Mechanical enables you to do just that.

In the following illustration, the Standard Parts Database dialog box is listing and showing some of the standards and categories of geometry you can select from to include in your design. That geometry can be features for a part you are designing or an entire part that you want to use in an assembly.

Definition of Standard Content

Standard content refers to the 600,000 plus parametric standard parts and pre-drawn features that you can select and use in your assembly and part designs. When using this content, you control the size of parts and features by setting length, width, or other size parameters during their placement.

Part content includes various types of screws, bolts, washers, nuts, and cotter pins. The different features you can add to a part include holes, external threads, and slots. The available sizes and shapes for each of these parts and features are based on common industry standards.

Because you are drafting in 2D, you specify the direction you wish to view the part or feature during the process of selecting and placing it in a drawing view. After you have added standard content to a view, you can project the different views of that standard content to other orthographic or auxiliary views of your design.

You can cycle the display of standard content between three different settings for the purpose of adding or removing detail from the display of the part or feature. When you are using Mechanical structure, you can control what is displayed as visible and hidden geometry by establishing an associative hide condition between parts in the assembly or geometry in the part. If you are not using structure, you can still establish hidden display for a part or feature by using 2D hide. This hide calculation method just does not update automatically like associative hide does.
In the following illustration, one of the standards has been expanded and the Features standard content has been selected. The preview buttons on the right show the types of features you can add to your part designs.

**Example of Standard Content**

Standard content makes the process of adding geometry to a design easier and quicker than creating the geometry from scratch. In the following illustration, bolted connection is shown in the assembly with two different appearances. For bolted connections, not only do you gain by being able to select and place industry standard parts, but you also gain by being able to set how you want the geometry to display relative to the other geometry in the assembly.
About Machinery Generators

When designing machinery, you often need to include shafts, shaft parts, springs, sprockets, belt or chain drives, or cams in that design. When you learn how you can create these items with AutoCAD Mechanical, you can use its capabilities when your designs require you to create such parts.

In the following illustration, the shaft view was created as it was being designed using the shaft generator. The end view was then created from the same shaft design by the shaft generator.

Definition of Machinery Generators

The different commands in AutoCAD Mechanical that create and add shafts, shaft parts, springs, sprockets, belt or chain drives, and cams to a design are referred to as machinery generators. During the creation of each of these parts, you select and enter sizes and values based on industry standards. For some of the generators, you also enter stresses, loads, and other design data to identify the correct part for your design requirements.

In the following illustration, the appropriate extension spring is in the process of being determined and created based on specified design criteria.

Example of Machinery Generators

The designing of machines typically includes shafts, bearings, gears, cams, belts, and chains. Creating this geometry from scratch or building and manipulating a library of industry-accepted parts is time consuming. In the following illustration, the sprockets, chain, chain length, shafts, bearings and gears
were all added to the drawing after selecting or specifying specific criteria in their corresponding generators. The hidden line calculation and selection of what hidden lines to display in these views was accomplished using the Associative Hide functionality.

About the Calculation Tools

During the process of creating a design, you come across situations where you are not sure what size standard content should be used or you need to calculate and report the results of different scenarios of your design. With an understanding of the available calculation tools, you can identify those situations where a calculation tool can help you to accomplish these tasks.

In the following illustration, different finite element analysis (FEA) results show and report what happens to this part when it is restrained in a certain way and specific forces are applied.
Definition of the Calculation Tools

You use the calculation tools in one of two ways. One way is within the standard content and machinery generator commands to help select parts that meet your design criteria. The other way is to enter a design scenario to calculate results and generate a report. Both uses of the calculation tools apply accepted engineering formulas when calculating the results.

You can calculate results and generate reports for the following areas:

- Finite Element Analysis.
- Screw Calculation.
- Shaft Calculator.
- Moment of Inertia for irregular shapes and predefined cross sections.
- Moments and Deflection of a Beam.
- Bearing Calculator.

Example of the Screw Calculation Tool

As you create your designs, you often fasten different parts together using screws or bolts. To ensure that you selected a large enough fastener for the conditions it will be under, you can calculate various factors using Screw Calculation. In the following illustration, the calculated results were added to the drawing for future reference. In this manner, part of the design knowledge and intent is being captured with the design to be readily available in the future.

About Autodesk Inventor Linked Models

If you are working with others who are creating 3D part and assembly models in Autodesk Inventor, you can create production-ready drawings from those parts and assemblies for them while using AutoCAD Mechanical. This enables you to assist in the completion of a project while using the software you are most familiar with.
Definition of Autodesk Inventor Linked Models

Working with linked Autodesk Inventor models means that you use AutoCAD Mechanical to create drawing views of Autodesk Inventor assembly (IAM) and part (IPT) files. After you have generated the drawing views in AutoCAD Mechanical, you then add dimensions and other annotation to create production-ready drawings. Because the Autodesk Inventor file is linked, any changes to the assembly or part can update the drawing and dimensions in the AutoCAD Mechanical drawing.

After you have created a view of the part or assembly, you can then create any of the following views:
- Orthographic
- Auxiliary
- Isometric
- Detail
- Broken
- Section

To link Autodesk Inventor models into an AutoCAD Mechanical drawing file, you do not have to have Autodesk Inventor installed on your computer. AutoCAD Mechanical is able to read and interpret those model files.

Example of an Autodesk Inventor Linked Model

In the following illustration, an Autodesk Inventor part was linked into an AutoCAD Mechanical drawing file. After establishing the link, the three orthographic views and the isometric view were created. The dimensions were added to the drawing views as the next step in creating a finalized drawing for this part.
About the Configuration and Setup Guide

As you begin to learn and use AutoCAD Mechanical, there is a good chance you will apply your knowledge and experience of standard AutoCAD commands and methods during the creation of your design in AutoCAD Mechanical. Although in some cases in AutoCAD Mechanical you use the same commands as AutoCAD, there are other cases when you can use commands or workflow unique to AutoCAD Mechanical. By using the AutoCAD Mechanical commands and workflow, you can achieve the best possible benefit and productivity you can. To help you learn AutoCAD Mechanical, you can access a help file that compares the commands and workflow of AutoCAD to AutoCAD Mechanical.

Definition of the Configuration and Setup Guide

The Configuration and Setup Guide is a help file created for the purpose of teaching you how to configure your AutoCAD Mechanical drawings and to compare the workflow and commands of AutoCAD to AutoCAD Mechanical.

You access the Configuration and Setup Guide from the Getting Started Launchpad window. When set, this window displays automatically each time you start AutoCAD Mechanical. To display it at any time, click Help menu > Launchpad. In the following illustration, the upper part of the Getting Started Launchpad window shows the link to the Configuration and Setup Guide.
Example of the Configuration and Setup Guide

You navigate through the contents of the guide in the same manner that you do any other help file. Typically, three tabs will be displayed to enable you to learn more about the subject. The Concept tab displays general ideas about the subject. The Procedure tab displays links to specific steps to help you accomplish your tasks. The Quick Reference tab shows access to the command and explains the options available.
Chapter Summary

In this chapter, you learned about the concepts of the different key areas of functionality that assist you in the creation and editing of 2D mechanical drawings.

Having completed this chapter, you can:

- Describe AutoCAD Mechanical concepts and key features.