AUTODESK UNIVERSITY

GEN20604

Intelligent AutoCAD Model Documentation Made Easy

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Learning Objectives

- Learn how to create base views and projected views from 3D models
- Learn how to create and control section and detail views
- Learn how to add annotations and adjust them when changes occur
- Learn how to document multipart assemblies

Description

Once you've created a 3D model, you still need to produce 2D drawings. In this class, you will learn how to use AutoCAD software's model documentation tools to create annotated orthographic, section, cutaway and detail views; how to control their scale and appearance; and how to ensure that views and annotations update properly if the 3D model changes. We will work with 3D models created in AutoCAD software, Inventor software, and other CAD programs, and see firsthand how to solve problems such as disassociated annotations. You will also learn how to create multi-sheet drawings documenting multipart assemblies. This class will help you develop best practices and timesaving techniques to capitalize on the power of AutoCAD software's model documentation tools to create smarter 2D drawings. This session features AutoCAD.
AIA Approved.

Your AU Expert

David Cohn is the senior content manager for CADLearning products at 4D Technologies, where he develops content standards and creates affordable training solutions for Autodesk, Inc. software, including AutoCAD software, AutoCAD LT software, and ReCap software. He has more than 30 years of hands-on experience with AutoCAD software and 15 years with Revit software as a user, developer, author, and consultant. He is an Autodesk Certified Professional for both AutoCAD software and Revit software. A contributing editor to *Desktop Engineering* magazine, he is also the former senior editor of *CADalyst* magazine, and the author of more than a dozen books about AutoCAD software. A licensed architect, Cohn was also one of the earliest AutoCAD third-party software developers, creating numerous AutoCAD add-on programs. As an industry consultant, Cohn has worked with many companies, including Autodesk. He has taught college-level AutoCAD courses, and has consistently been a top-rated speaker at Autodesk University.

Introduction

Two-dimensional drawings remain the most convenient way of conveying design information to those who will build the buildings or manufacture the products depicted in those drawings. But most designers take their 3D designs, mentally flatten them, and manually produce those 2D drawings. Happily, this is no longer necessary. AutoCAD provides a number of tools to help you create two-dimensional drawings based on three-dimensional models.

You can use the Drawing Views tools to create standard views such as top, front, and side views, as well as various isometric views, sections, and detail views directly from 3D models. These views are created as drawing view objects, which remain associative to the model from which they were created. Although you cannot select the drawing view geometry to modify it, if you make any changes to the 3D model, the drawing view geometry automatically updates.

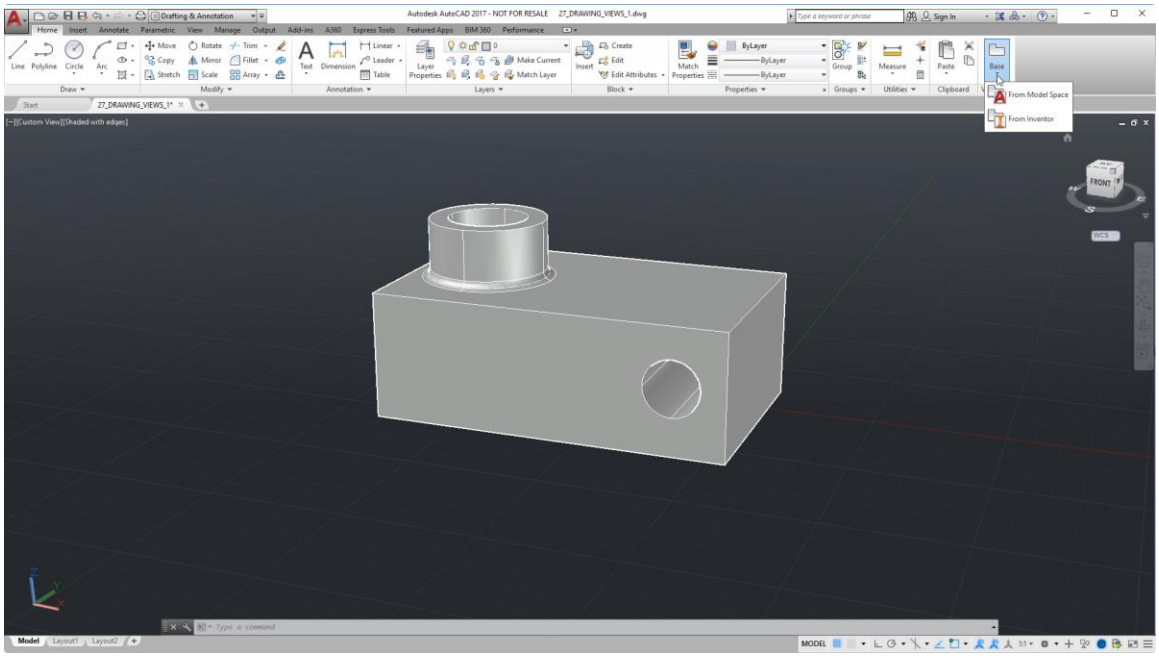
In this class, we will take a detailed look at all of these Drawing View tools.

Creating Base Views

In order to work with drawing views, you must first place a base view onto a paper space layout. A *base view* is the first view created in a drawing. Other views are then derived from the base view. The base view can include all of the visible solids and surfaces within model space, selected objects, or a model imported from Autodesk Inventor.

We will first look at how you would create a base view from an existing 3D AutoCAD model.

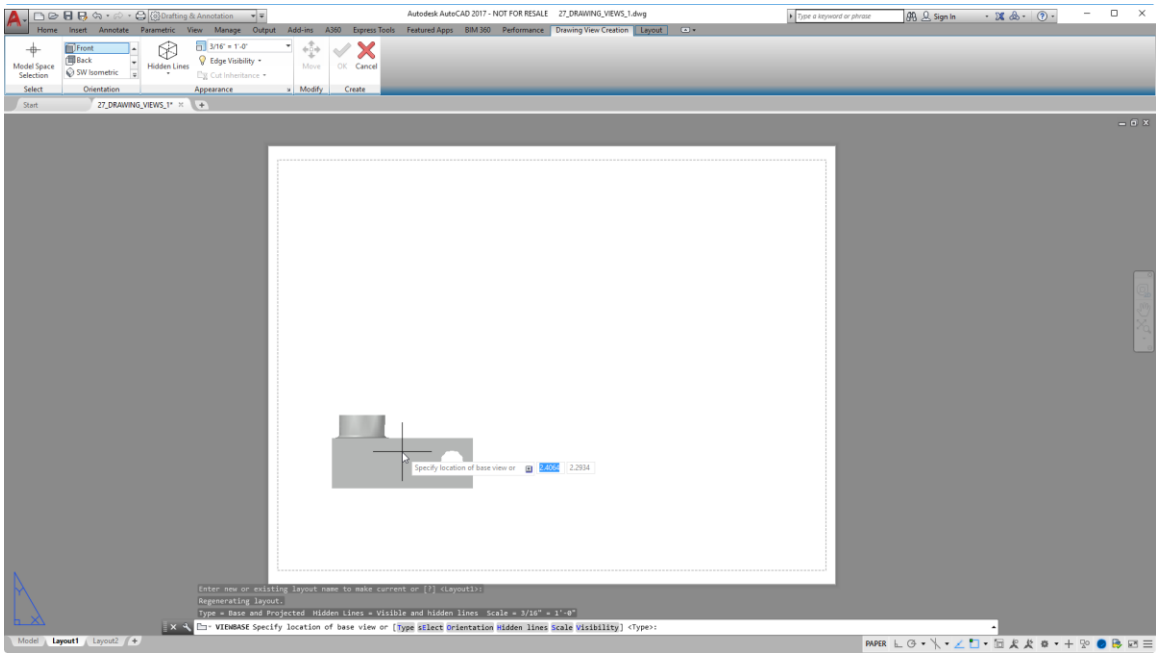
On the **Home** ribbon, expand the **View** panel and expand the **Base** button. There are actually two tools. The **Base View from Model Space** tool generates a base view of a model that already exists in model space, while the **Base View from File** tool generates the base view from a model that must first be imported from an Autodesk Inventor file. Click the **From Model Space** button. Note that if you were to click this button in an empty drawing, AutoCAD would prompt you to open an Inventor file. And if you were already working in a paper space layout, the program would simply prompt you to specify the location of the base view.



Since the drawing already contains a 3D model, the program prompts you to select objects, and you can use any convenient object selection method. You can also simply press **ENTER** to select the entire model.

Next, the program prompts you to enter a new or existing layout name to make current. If you enter the name of an existing layout, the base view will be created on that layout, or you can enter a new layout name, in which case AutoCAD will first create a new layout. In this case, type the new layout name "**MyViews**" and then press **ENTER**.

Now you can see the new layout. The ribbon changes to the **Drawing View Creation** contextual ribbon, and AutoCAD prompts you to specify the location of the base view. There are also a number of options, many of which can be controlled using the tools in the ribbon.



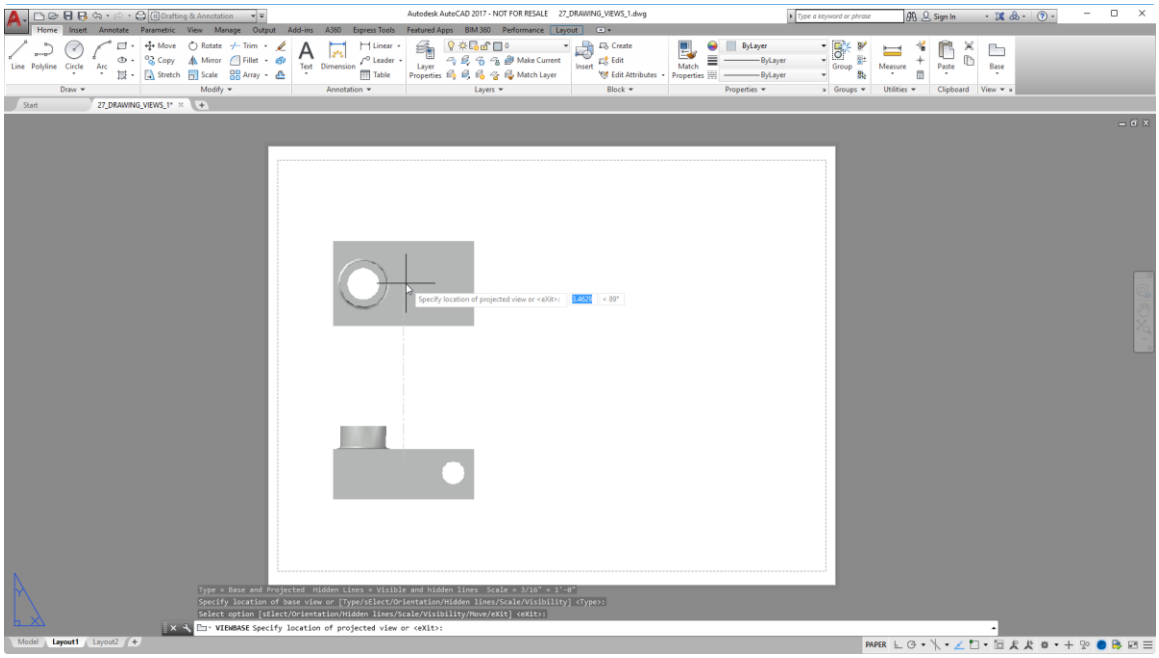
AutoCAD automatically sets an appropriate scale so that the view you are creating will fit onto the layout, but you can change the scale if you wish. You can also always change the scale of the view later, even if you have added other views and annotations.

By default, the base view orientation is the Front view, but you can select a different orientation from the **Orientation** drop-down.

As soon as you click to place the base view, AutoCAD displays a list of possible options. Generally, it is easier to simply place the view now and make any necessary changes later after you have placed other views. So simply press **ENTER** to complete the placement of the base view.

Next, the program prompts you to specify the location of a projected view. If you are ready to place other views projected from this base view, you can do so now. If you prefer, you can press **ENTER** one more time to exit from the command and then come back later to place other projected views.

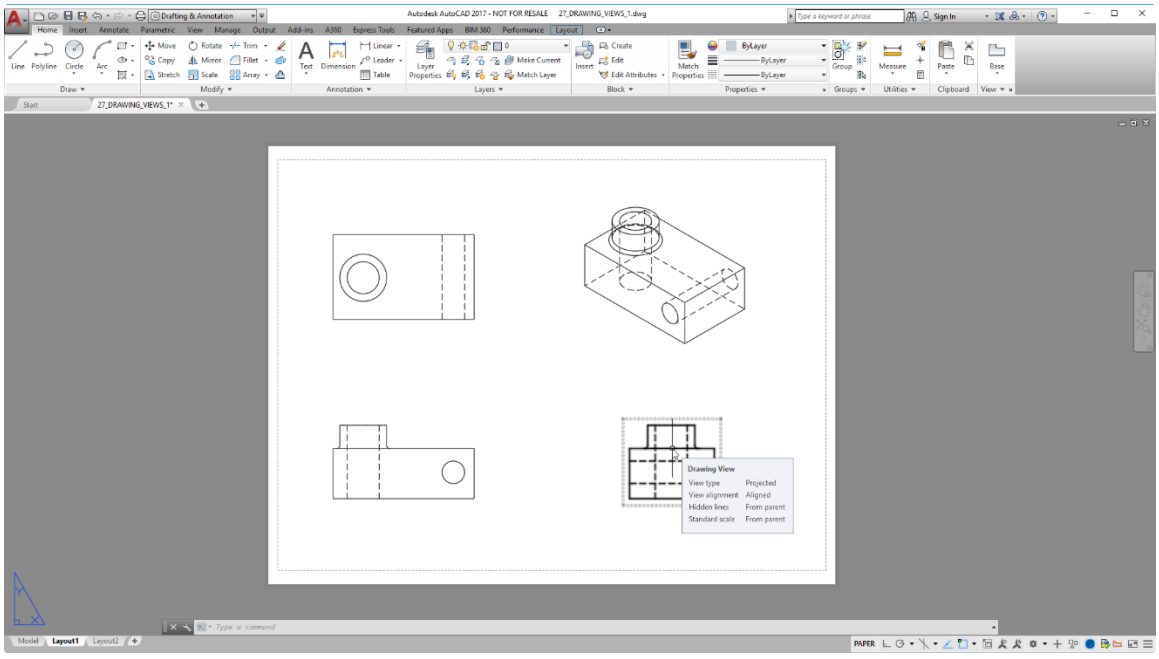
When you move the cursor above the base view, you can immediately see the projected view. You no longer see the **Drawing View Creation** contextual ribbon because projected views are child views of the parent base view, so by default, the projected view automatically takes on the properties of the base view. It will therefore have the same scale and view style as the base view. But you can always change the properties of projected views later.



Click to specify the location of the projected view. As soon as you do, you can see the top view. The command is still active, and AutoCAD is prompting you to specify the location of another projected view. Also note that the next projected view is still based on the original base view. Move the cursor to the right and place a right-side view.

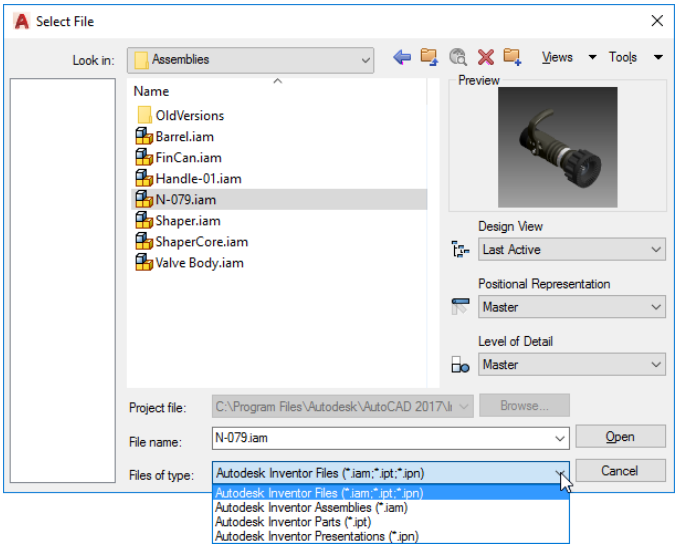
As soon as you do, the prompt repeats and you can place yet another projected view. This time, move the cursor off at an angle and create an isometric view. Notice that the isometric projection depends on where you move the cursor.

When you are finished placing projected views, you must press **ENTER** to exit from the command. As soon as you do, all the views update to reflect the view style.

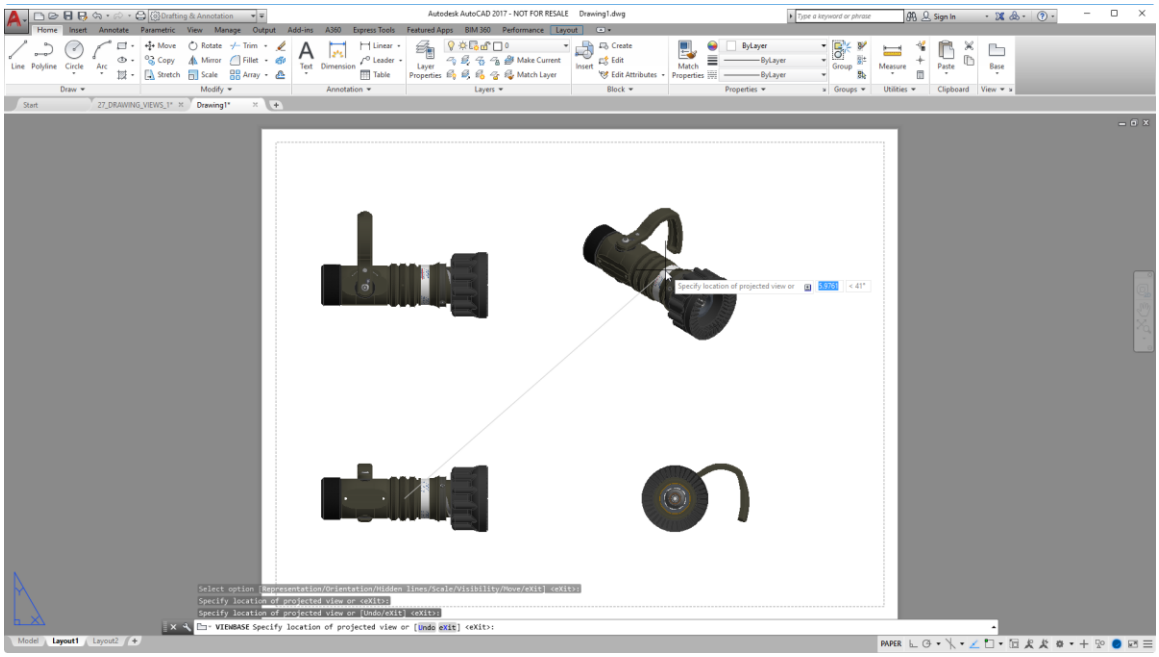


Note that what you see are not viewports, but rather a special type of object called a drawing view. If you move the cursor over a view, you can see in the tooltip that this is a drawing view.

You can also import an Autodesk Inventor model and use it as the basis for creating drawing views. To illustrate this, in a blank drawing, on the **Home** ribbon, expand the **Base** button in the **View** panel, and click the **From Inventor** button. AutoCAD displays a **Select File** dialog. You can choose an Inventor assembly, part, or presentation file. When you select the file, you see a preview of the model. Click **Open**.



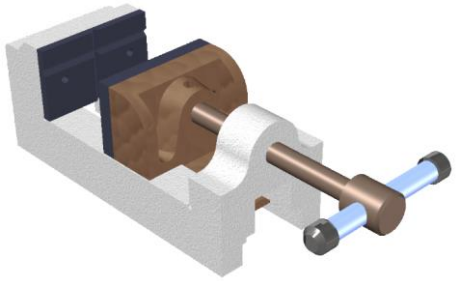
Next, the program prompts you to enter a new or existing layout name to make current. From this point forward, the process is exactly the same as when creating a base view and projected views from a 3D model that already exists in model space.



Documenting Multipart Assemblies

When working in a drawing that contains more than one 3D model, you can create drawing views of all of the parts in the model or create drawing views of individual parts.

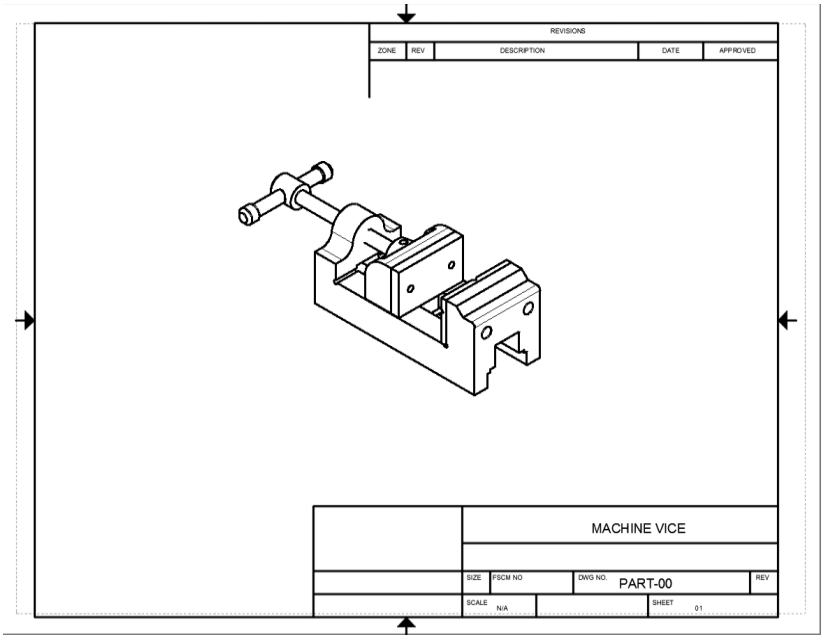
As when working with a single 3D model, you must first place a base view onto a paper space layout and then derive other views from that base view.



For example, this model of a machine vise consists of nine separate parts. Suppose you wanted to create separate drawings to document each part. On the **Home** ribbon, in the **View** panel, expand the **Base** button, and select **Base View from Model Space**. AutoCAD prompts you to select objects. Since the first sheet will consist of a view showing the entire vise, simply press **ENTER** to select the entire model. Next, the program prompts you to select the layout onto which you want to place the view. In this case, the drawing already includes a number of layouts, each of which already includes a border and title block. Type "**SHEET-01**" and press **ENTER**.

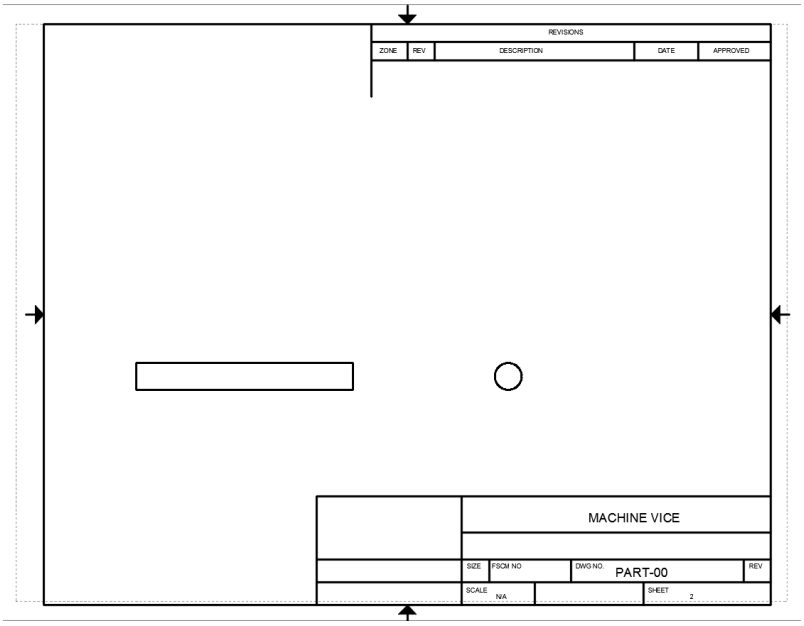
AutoCAD switches to that layout and displays the **Drawing View Creation** contextual ribbon. Expand the **Orientation** list and choose the **NE Isometric**. If the drawing view is too small, expand the **Scale** list and select a more appropriate scale, such as **1:2**. You can also expand the **Hidden Lines** split button and choose **Visible Lines**, so that the resulting view shows only the visible lines.

Since this is the only view you will place on this sheet, right-click, choose the **Type** option, and select **Base Only**. Then, click to place the isometric view and press **ENTER** to end the command. The isometric view is immediately placed onto the layout as a base view and the command ends.



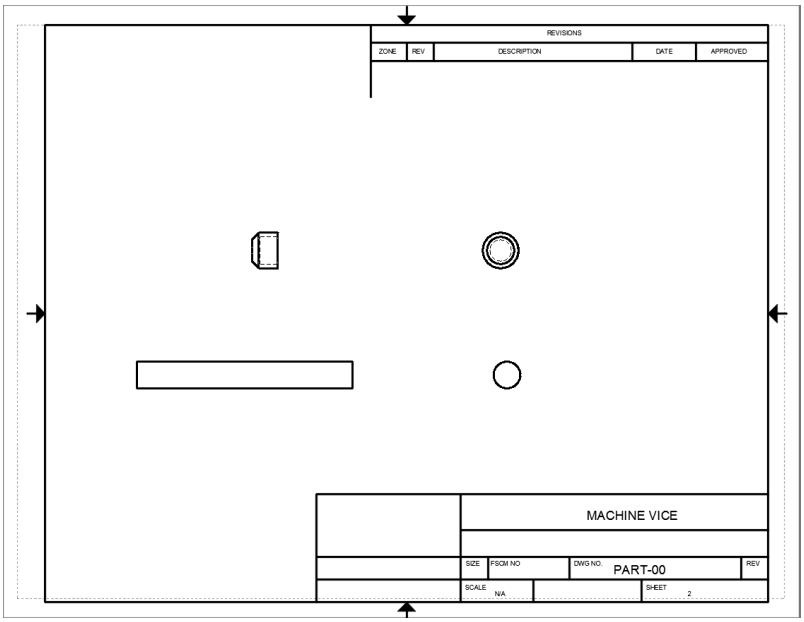
Next, create a base view of one of the other parts of the vise. Click the **Model** tab to switch back to model space. Then, on the **Home** ribbon, expand the **View** panel, expand the **Base** split button, and select **Base View from Model Space**. AutoCAD prompts you to select objects. You can use any convenient object selection method. Select the handle and then press **ENTER**.

AutoCAD prompts you to enter a new or existing layout name. Type "**SHEET-02**" and press **ENTER**. You are immediately taken to that sheet, and you can see a front view of the handle. Since that is not the view you want, expand the **Orientation** list and choose the **Right** view. Now you can see the right-side view of the handle. Notice that the scale has changed. Right-click and choose the **Type** option. AutoCAD remembered that the last time you placed a base view, you placed the base view only. Change this back to **base and Projected**. Then, click to place the base view, and press **ENTER**. Since you changed the Type option, the program prompts you to place a projected view. Place the view to the right. Then, press **ENTER** to exit from the command. AutoCAD immediately completes the creation of the two views.



Since this is a small part, you can place views of one of the other parts on this same sheet. On the **Layout** ribbon, in the **Create View** panel, expand the **Base** split button and select **Base View from Model Space**. When you start this tool while working in a layout, AutoCAD assumes that you want to place a view of the entire model. Since that is not what you intend, however, click the **Model Space Selection** button in the ribbon. AutoCAD switches back to model space and you can see that the entire model is selected. But notice that there is a **Remove** option. When you choose that option, AutoCAD prompts you to select objects to remove. You can use any convenient object selection method to remove everything except for one of the handle end caps. Then, press **ENTER**.

AutoCAD immediately switches back to the previous layout, and you can see a preview of the base view of the cap. If that is not the view you want, open the **Orientation** list and choose the **Right** side view. If the view is not at the desired scale, expand the **Scale** drop-down and choose **1:1**. Then, click to place the base view and press **ENTER**. Then, place a projected view to the right. When you are finished, press **ENTER** to exit from the command. AutoCAD immediately completes the creation of the two views.

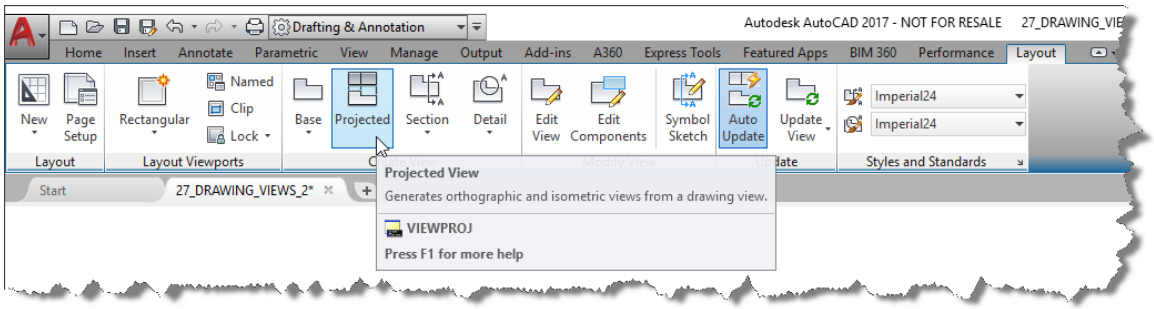


You can use these same methods to place views of the other parts of the machine vise onto other layout sheets.

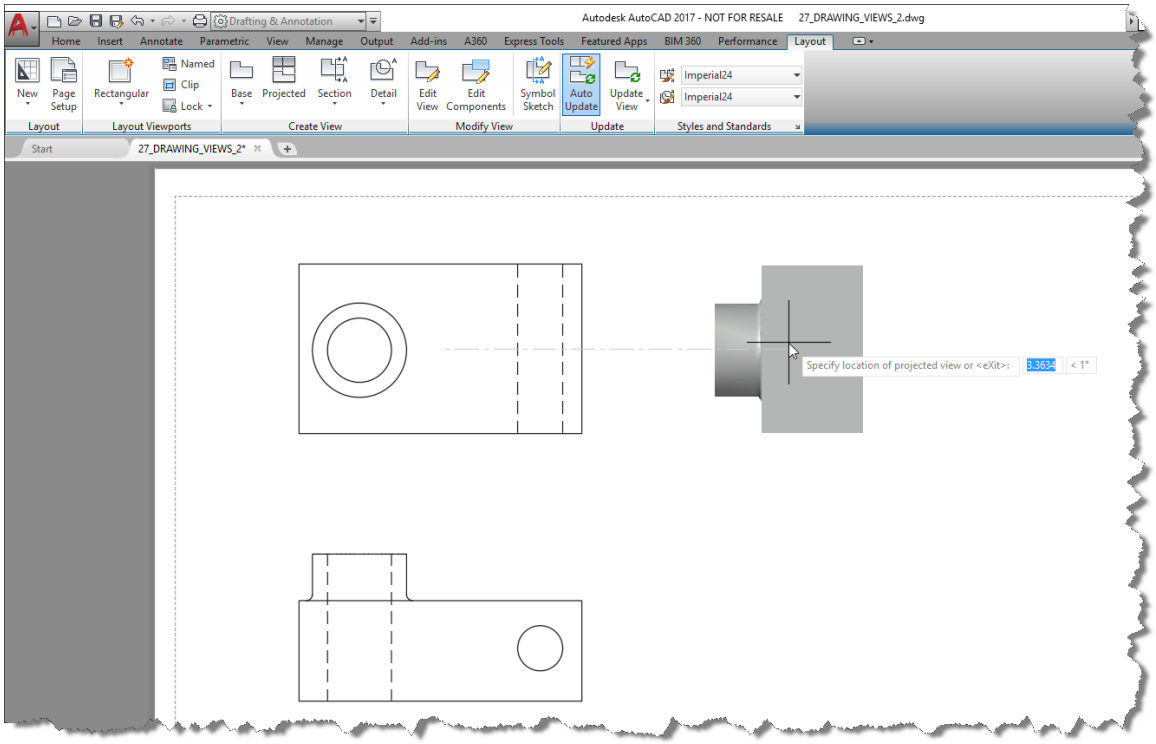
Creating Projected Views

Once you have placed drawing views in a layout, you can create additional projected views based on any of those views.

For example, suppose you have already placed several drawing views in a layout and you would like to create some additional views. On the **Layout** ribbon, in the **Create View** panel, click the **Projected View** tool.



AutoCAD prompts you to select a parent view. You can select any of the views that have already been placed in the layout. Whichever view you choose becomes the new parent view from which the new view will be projected, even if the view you choose is itself a child of some other parent view. For example, if you choose the top view, which is actually a view that was projected from the front view, the new view is projected from that top view.



Editing Drawing Views

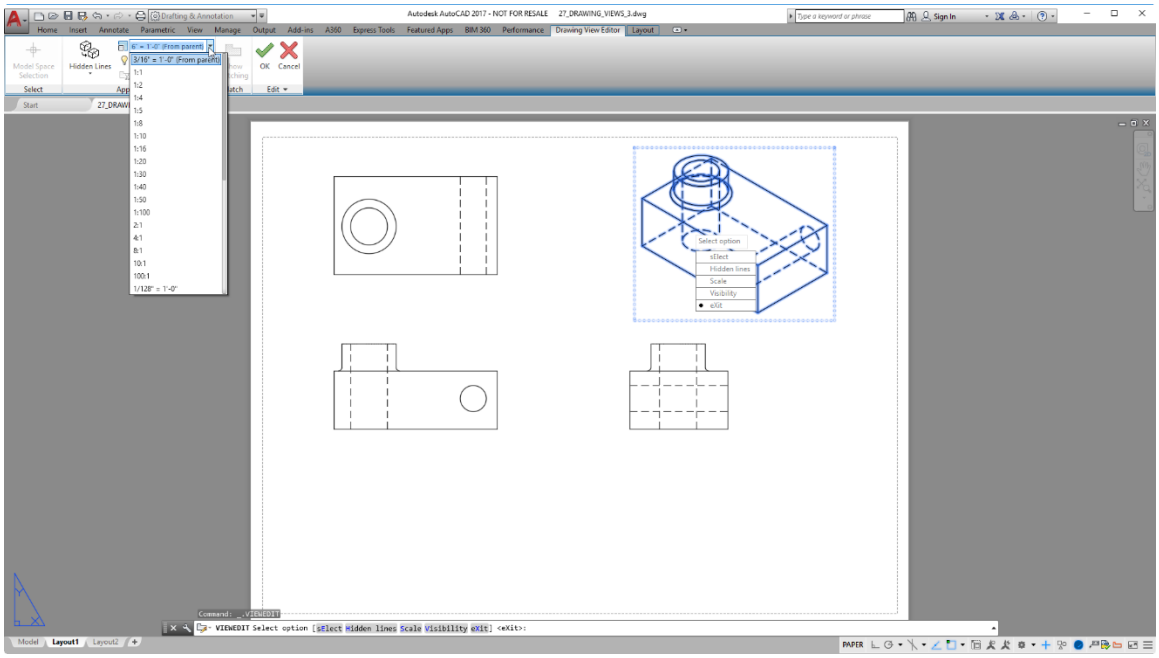
After you have created and placed drawing views, you can edit those views to change their hidden line style, scale, edge visibility, and view options, or to move the view. Remember, however, that you cannot change any of the geometry within the view. The view geometry can only be changed by modifying the 3D objects on which the view is based.

To edit a drawing view, you can click the **Edit View** tool in the **Modify View** panel of the **Layout** ribbon. AutoCAD prompts you to select a view. Select the view you want to modify. Alternately, you can select the view, right-click, and then click **Edit View**, or simply double-click on the drawing view you want to edit. Edit the isometric view.

The ribbon changes to the **Drawing View Editor** contextual ribbon and AutoCAD displays a number of options. You can then use these tools to edit the drawing view. For example, expand the **Hidden Lines** drop-down. The hidden line view style for the isometric view is currently based on its parent view, but you can change it to any of the other available hidden line styles.

The scale of the view is also currently based on the parent view, but if you wish, you can change it to something else.

When you edit a view, any changes you make also affect any views that were projected from that view.



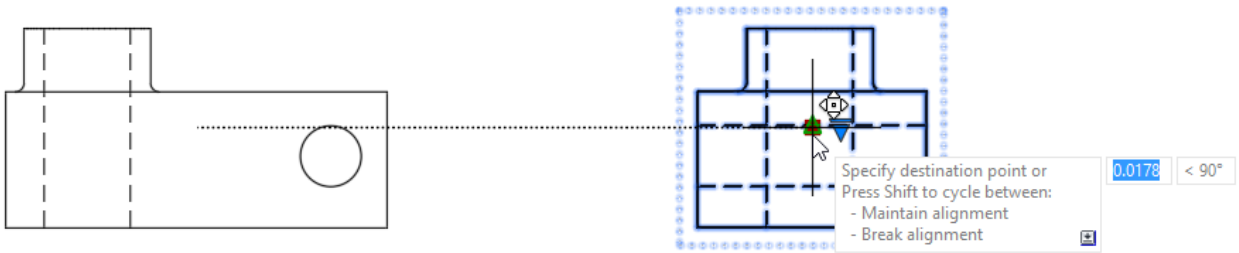
In the **Edge Visibility** drop-down, you can select whether you want to see Interference Edges, Tangent Edges, and so on.

If you click the dialog box launcher (the small arrow in the lower-right corner of the **Appearance** panel), AutoCAD displays the **View Options** dialog. You can then use the tools in this dialog to control the view justification, and if the view was based on an Autodesk Inventor model, you could also control the Line Style and Hidden Line Calculation.

Also note that if you expand the **Edit** panel, there is a **Defer Updates** toggle. This is typically turned off, which means that AutoCAD immediately updates drawing views as soon as you change a view property. When working on a large model, you may want to toggle this off to improve the program's performance, and then toggle it back on again once you have made all of the changes.

When you select a view, you see two grips. You can use the triangular grip to quickly change the scale of the drawing view or click the square grip at the center of the view to move the view.

By default, views remain aligned. If you move the base view, its dependent views also move. If you click on the projected side view, however, you can only move it left and right. You cannot move it up or down because the vertical position of the view is based on its parent view. But if you hover the cursor over the grip, the tooltip tells you that you can press the SHIFT key to cycle between Maintain Alignment and Break Alignment. When you tap the **SHIFT** key, you can now change the vertical location of the view so that it no longer aligns to the right of the base view.

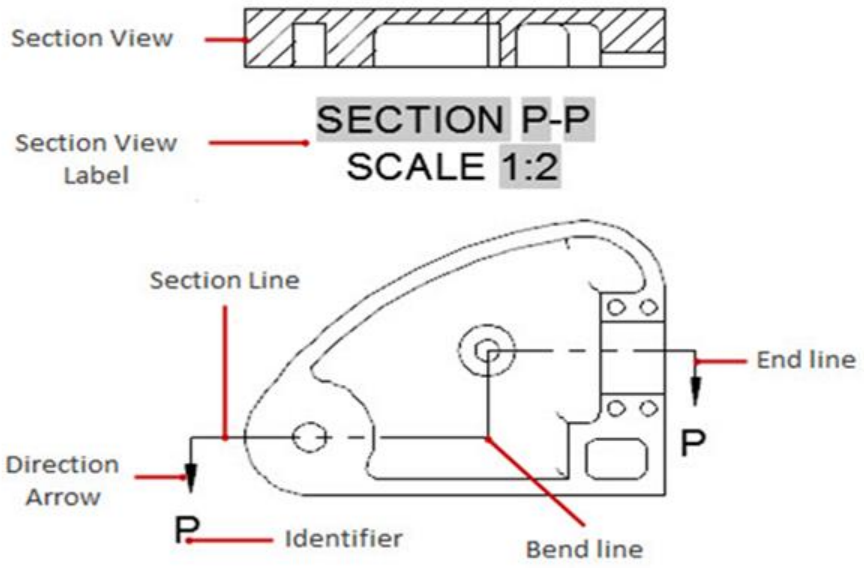


This is generally not considered good drafting practice, however. If you click the square grip again and then tap the **SHIFT** key, the view immediately realigns itself with its parent view.

Creating Section Views

You can use any drawing view as the basis for creating a section view. A *section view* is a special type of projected view created by using a section line to cut through the drawing view in order to reveal what is inside.

When you create a section view, AutoCAD generates a number of entities. In addition to the section view itself, the program typically creates a section view label, a section line, section arrows, a section identifier, and cross-hatching. Depending on the type of section you create, the section line may also include bend lines. The appearance of these elements is controlled by the **Section View Style**.



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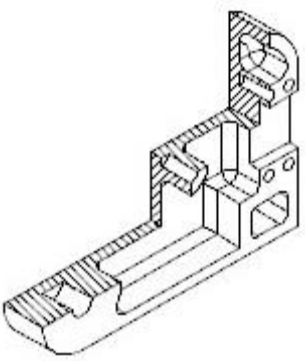
If **Infer Constraints** is on at the time that you create the section view, constraints are inferred between the view geometry and the section line. When view updates take place, the constraints ensure that the section line retains its position in relation to the features it highlights. In some cases, you may require constraints that are more complex than the inferred constraints, such as a section line that must pass through a part exactly 15mm from a particular edge. In such cases, you can manually constrain the section line to the view geometry, using a special edit environment called the *symbol sketch mode*. While working in this special mode, you can add additional geometry to assist you in constraining the section line. Any geometry you create while working in this mode is only visible in symbol sketch mode.

Components cut by the section line are hatched in the section view. Typically, fasteners and shafts should not be hatched in section views, and when working with views created from Autodesk Inventor models, AutoCAD will not hatch these items. But when working with 3D models created in model space, AutoCAD has no way of recognizing these components, so you will need to manually select such components and exclude them from being hatched.



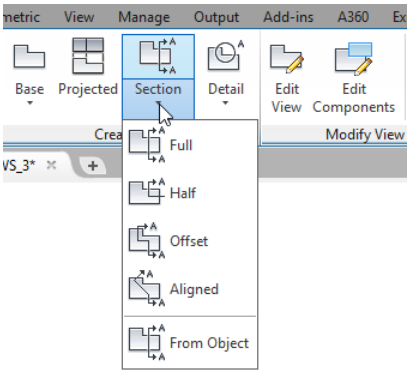
You can also generate projected views from a section view. If you project an isometric view from a section view, the resulting isometric will inherit the cut information, enabling you to create cutaway isometric views. Such views inherit properties such as scale and hatch from the parent section view.

After creating a section view, if you modify the section lines—for example, if you move the section lines—the corresponding section view will update. And if you delete a section line, its corresponding section view will also be deleted.

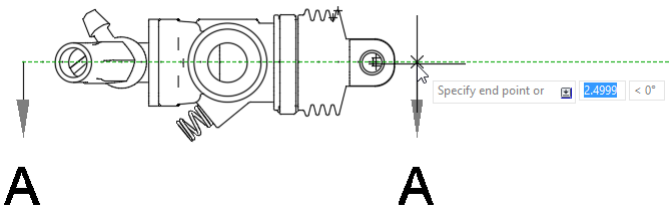
You can create a full section view (by running the cutting plane through the entire length of the object being sectioned), a half section view (by running the cutting plane through a portion of the length of the object being sectioned), an offset section view (by offsetting or bending the cutting plane to show features that are not in a straight line), and an aligned section view (by cutting through the entire object along two non-parallel work planes, such that they pass through specific objects). You can also create a section view using paper space geometry to define the section line. After creating a section view, you can edit the view, modify its hatch patterns, modify the cutting plane, modify the label and identifiers, adjust the depth of the section, and exclude specific components from the section view.

To create a section view, on the **Layout** ribbon, in the **Create View** panel, expand the **Section** tool. Notice that there are five choices, but all five activate the same VIEWSECTION command; the buttons simply correspond to the available view **Type** options.

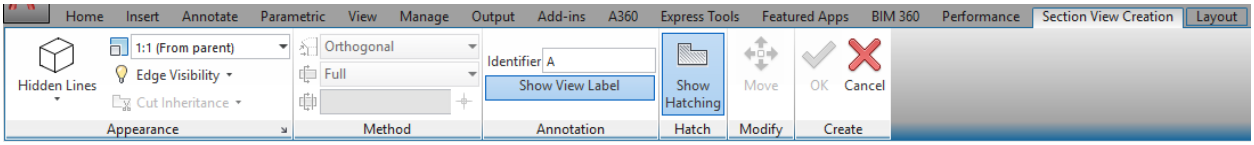
For example, to create a full section view, click the **Full** button. The program prompts you to select a parent view. In this case, select the top view. The program then prompts you to specify the start point for the section line. Since you want the line to always align with the centerline of the carburetor, in the **Status** bar, make sure that the **Infer Constraints** option is toggled on. Also make sure that **Polar Tracking**, **Object Snap Tracking**, and **Object Snap** are active and that **Center** is one of the running object snaps.



Then, track off the center of the left end of the carburetor and click to indicate the start point of the section line. You immediately see the end direction arrows and section identifiers, and the program prompts you to specify the end point of the line. Track along the 0-degree line to the right and select the end point of the section line.



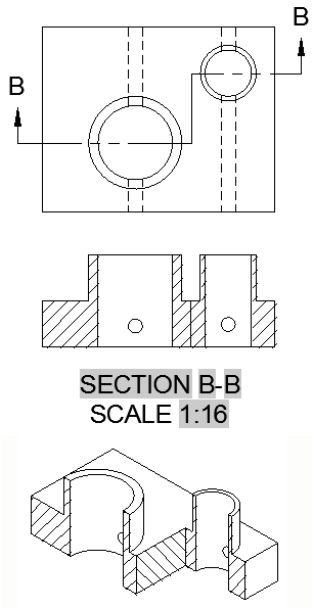
As soon as you do, a preview of the section view appears at the cursor and the ribbon changes to the **Section View Creation** contextual ribbon. You can use the tools in the ribbon to adjust various aspects of the section, such as the hidden line style, scale, and edge visibility. You can also control the section identifier, whether a section view label is created, and whether the section includes hatching.



Also notice that by default, the section preview is constrained to move in a direction perpendicular to the section line, but you can tap the **SHIFT** key to relax this constraint if you wish. Tap again and the constraint is reapplied. Click where you would like to place the section view, and then either click **OK** or press **ENTER** to end the command.

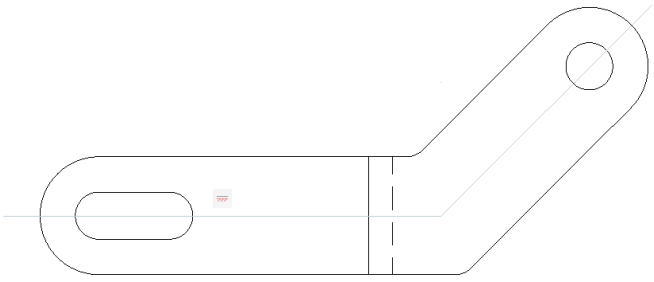
To create an offset section, expand the **Section** tool and click the **Offset** button. AutoCAD prompts you to select the parent view. Again, if you want the section line to always align with specific geometry, be sure that **Infer Constraints** is toggled on and that you use object snap or object snap tracking to lock the cursor to the desired geometry. You can then follow the prompts to create a section line that offsets (or bends) to cut through specific features.

Once you have created a section view, you can use it as the parent view to create other projected views. If you project an isometric view from a section view, the result is a cutaway isometric view.

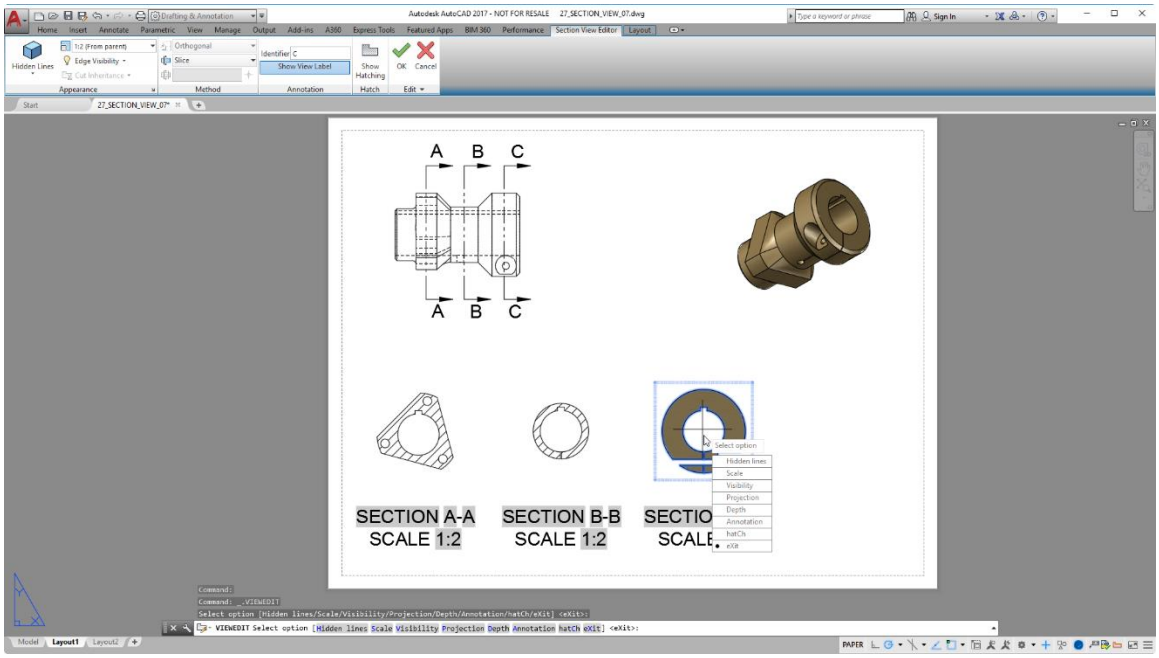


You can also use existing geometry created in paper space to define the cutting line. For example, to create a section view that cuts through an angled mechanical part along the center of both portions of the part, you could first draw some lines in paper space and then use those lines to create the section view. Expand the **Section** tool and select **From Object**. The program prompts you to select the parent view. Once you select the view, AutoCAD prompts you to select objects. You can then select the lines and then press **ENTER**. As soon as you do, the preview of the section appears at the cursor and the ribbon changes to the **Section View Creation** contextual ribbon. You can then complete the creation of the section view.

You can also create multiple cross section views through the same drawing view and control the depth of each view. This is a great way to illustrate how the shape of an object changes along its length.

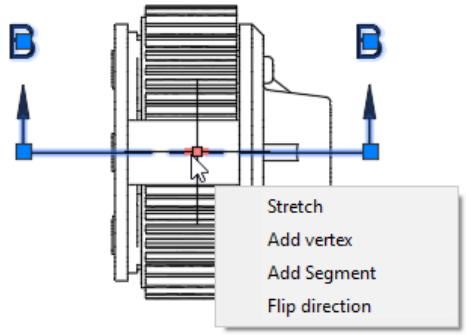


For example, create three cross section views through the side view of a wheel hub. Notice that once you place the section view, in the **Method** panel of the **Section View Creation** contextual ribbon, you can switch from the default **Full** section (which includes all geometry beyond the cutting plane), to either **Slice** or **Distance**. If you choose **Slice**, the cross section only includes the geometry actually cut by the cutting plane. If you want to include some of the geometry beyond the cutting plane, you can select the **Distance** option, and then specify the depth to which you want to include details.

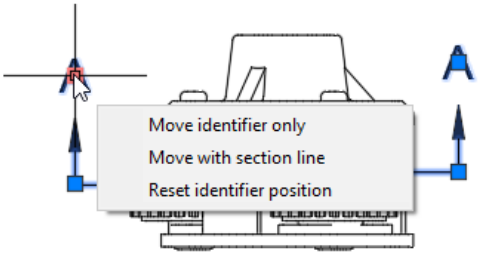


When you create a section view, the components cut by the section line are typically hatched in the section view. Initially, the appearance of the hatch is determined by the section view style. Once the section has been created, however, you can change the hatch pattern of any component in the section view. You can select hatch objects in drawing views and then use the tools in the **Drawing View Hatch Editor** contextual ribbon to modify the hatch the same way you would modify any other hatch object in AutoCAD.

After you have created section views, you can edit the views as well as many of the elements associated with the view. For example, you can use the **Edit View** tool to change the hidden line style, the view scale, or the view identifier. You can also toggle the view identifier or hatching on or off. You can also adjust the section depth. If the section view you are modifying was projected from another section view (such as a cutaway isometric), you can also change the **Cut Inheritance** setting. If you toggle this off, the cutaway isometric would become a standard isometric view.



After you have created a section view, if you change the section plane—for example, if you use grip editing to alter the cutting plane—the section view updates to reflect your changes. When you select the section plane, you can see grips that are very similar to those available when editing a polyline. And if you hover the cursor over a grip, you see an extended tooltip that shows the options available for modifying that particular grip. For example, you can stretch a vertex, add or remove a vertex, add or remove a segment, or flip the direction of the section.

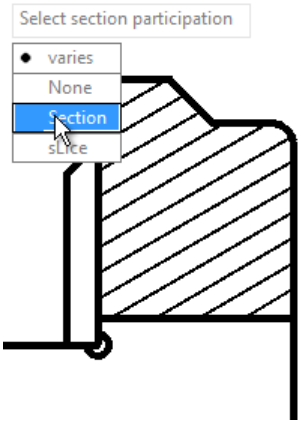


You can also modify the section line identifier. For example, you can change the position of the identifier or change the label. To change the position of the section line identifier, first select the section line. Then, hover the cursor over the identifier you want to move. When the extended

tooltip appears, you can use the tools to move the identifier only, move the identifier and the section line, or reset the identifier back into its default position.

You can also change the identifier label. To do this, first select the section view. You can then change the identifier label by entering a new value in the **Identifier** field in the **Section View Editor** contextual ribbon. Note that you can also make this change using the **Properties** palette.

You can also modify the section view label much the same as you would modify any multiline text object. For example, if you double-click a section view label, the ribbon changes to the **Text Editor** contextual ribbon. Note, however, that the actual section name and scale are fields. You can double-click these elements to edit the field formatting, but you should be careful not to change or delete the fields themselves. And if you wanted to change something about the formatting of the label—for example, if you wanted to include a colon after the word SCALE in the label—it would be much more efficient to modify the section view style than to make this type of change by manually editing each section view label.



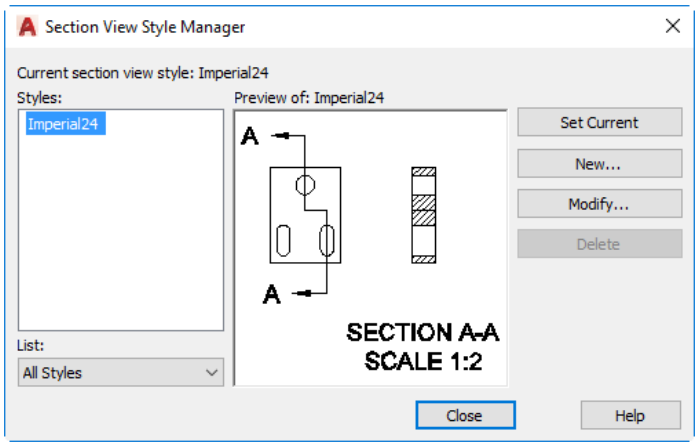
Finally, you can manually exclude objects so that they are not hatched in a particular section view. To exclude components from a section view, on the **Layout** ribbon, in the **Modify View** panel, click the **Edit Components** tool. AutoCAD prompts you to select components. Once you have made your selection, you can change the **Section Participation** setting for those components to **None**, **Section**, or **Slice**.

Controlling the Section View Style

When you create a section view, the appearance of the view—including the section identifier and arrows, the cutting plane, the view label, and the hatch applied to sectioned objects—is determined by the current section view style, much in the same way that text is controlled by the current text style and dimensions by the current dimension style.

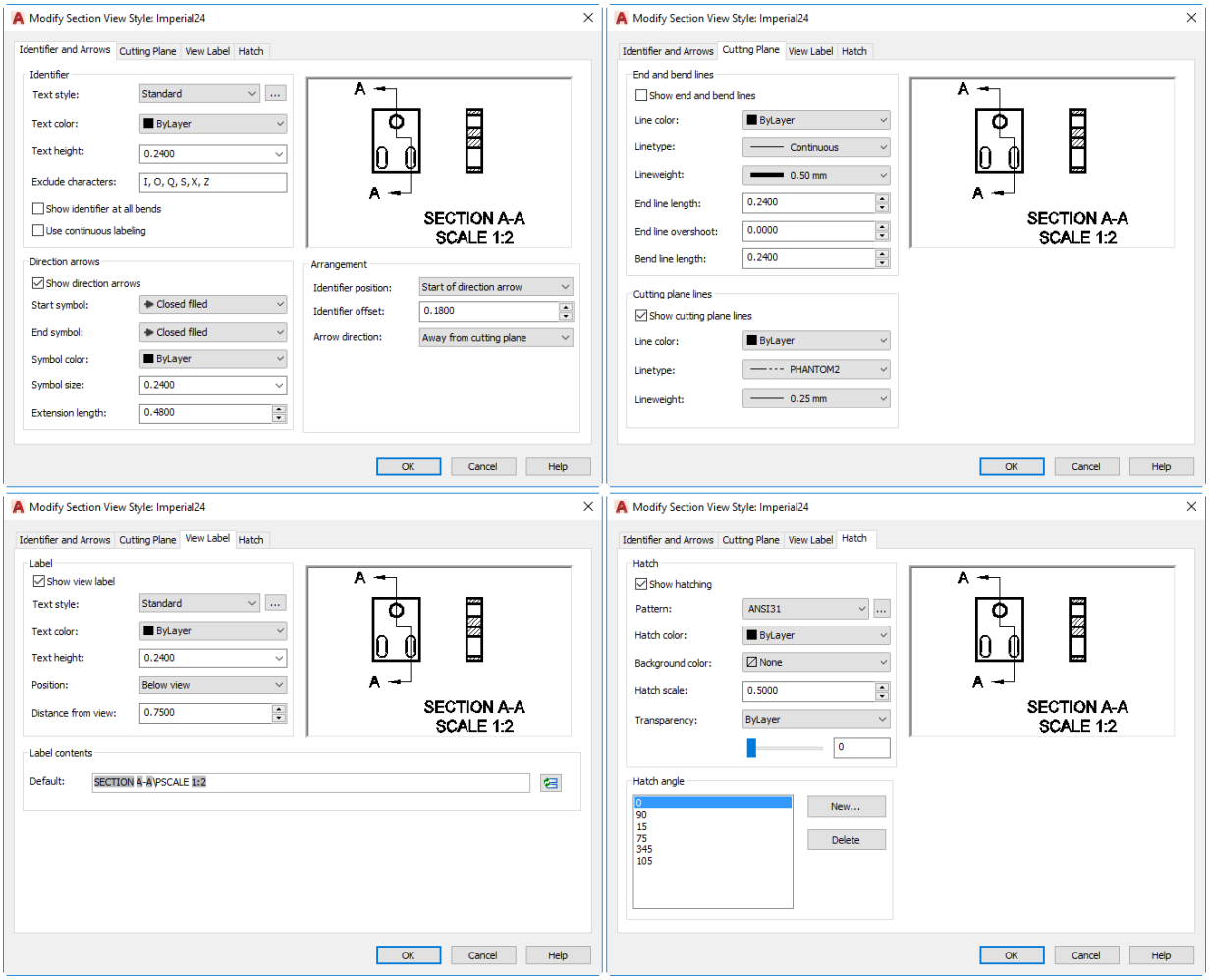
Section view styles provide the means to quickly specify the format of the various section view elements. They also provide a convenient way to ensure that section views conform to drafting standards. AutoCAD comes with two predefined section view styles—Imperial24 and Metric50—that conform to the ANSI and ISO drafting standards, respectively. Of course, like other styles in AutoCAD, you can modify these styles or create and save additional custom styles.

To create a new section view style or to modify an existing style, on the **Layout** ribbon, click the **Section View Style** button to display the **Section View Style Manager** dialog. The **Styles** list on the left shows the styles currently available in the drawing, and when you select a style in the list, you see a preview of the style in the adjacent window.



To modify an existing style, select it in the list and then click the **Modify** button. To create a new style, click the **New** button. AutoCAD displays a **Create New Section View Style** dialog so that you can name the new style. Then, AutoCAD displays the **New Section View Style** dialog. This dialog contains four tabs. The controls on the **Identifier and Arrows** tab determine the properties for the section identifiers and direction arrows used by the section lines. Notice that you can control the text style, text color, and text height, and exclude certain characters so that they are not used as identifiers. You can also choose to show identifiers at bends.

Controls in the **Direction Arrows** area let you include direction arrows, control the appearance of those arrows at the start and end of the section line, and control their color, size, and extension length. And the controls in the **Arrangement** area determine the placement of section line identifiers in relation to the start and end points of the section line, and whether the arrows point away from or towards the cutting plane.



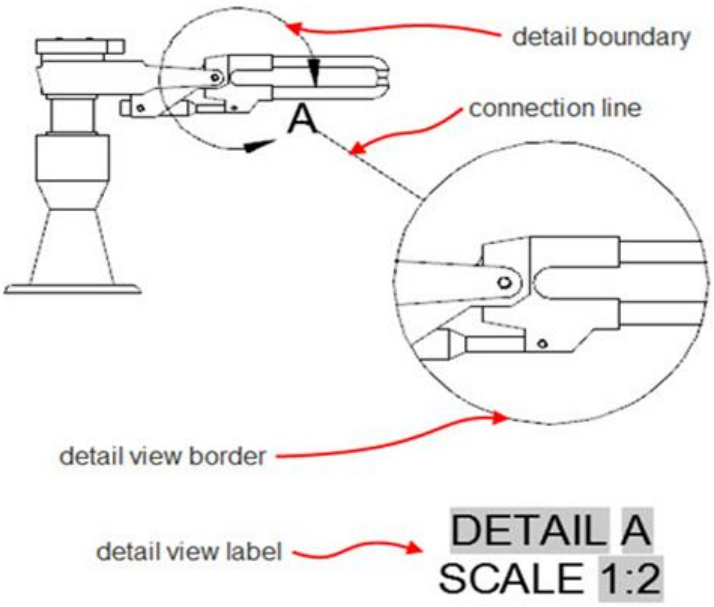
The controls on the **Cutting Plane** tab determine the properties for the ends, bends, and segments of the section line. The controls on the **View Label** tab determine the properties for the text, style, placement, and content of section view labels. And the controls on the **Hatch** tab determine the appearance and behavior of hatch objects within section views.

Once you have created additional section view styles, the new styles appear in the **Section View Style** drop-down on the **Layout** ribbon, and you can easily make a style current or apply a new style to an existing section view.

Creating Detail Views

You can use any drawing view as the basis for creating a detail view. A *detail view* is a projected view from an existing drawing view that shows a specific portion of the view at an enlarged scale.

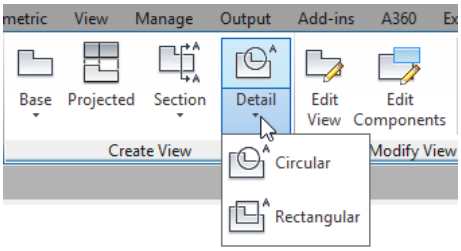
When you create a detail view, AutoCAD generates a number of entities. In addition to the detail view itself, the program typically creates a detail view boundary, a detail view border, a detail view label, and optionally a connection line. The appearance of these elements is controlled by the **Detail View Style**.



If **Infer Constraints** is on at the time that you create the detail view, constraints are inferred between the view geometry and the detail boundary. When view updates take place, the constraints ensure that the detail boundary retains its position in relation to the features it highlights. In some cases, you may require constraints that are more complex than the inferred constraints. In such cases, you can manually constrain the detail boundary to the view geometry using a special environment called the *symbol sketch mode*. While working in this special mode, you can add additional geometry to assist you in constraining the detail boundary. Any geometry you create while working in this mode is only visible in symbol sketch mode.

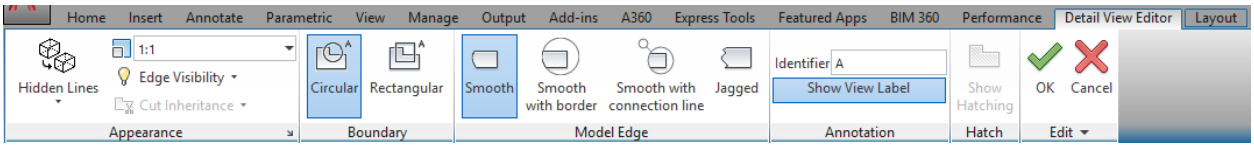
After creating a detail view, if you modify its parent view or the detail boundary, the detail view will update. For example, if you rotate the parent view, the view geometry also rotates but the detail boundary does not change. If you change the scale of the parent view, the detail boundary also rescales. And if you delete the detail boundary, the corresponding detail view is also deleted, and vice versa.

To create a detail view, on the **Layout** ribbon, in the **Create View** panel, expand the **Detail** tool. Notice that there are two choices. Both activate the same VIEWDETAIL command; the buttons simply correspond to the available **Boundary** options.



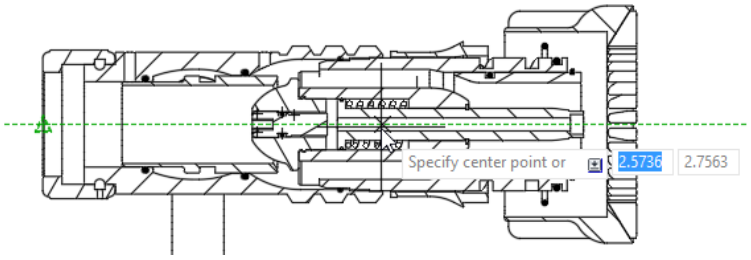
For example, to create a circular detail view, click the **Circular** button. The program prompts you to select the parent view. In this case, select the section view. As soon as you do, the ribbon changes to the **Detail View Creation** contextual ribbon, and the program prompts you to specify the center point. The format of the detail view is controlled

by the current detail view style, but you can override any of these settings using options or the tools in the ribbon.



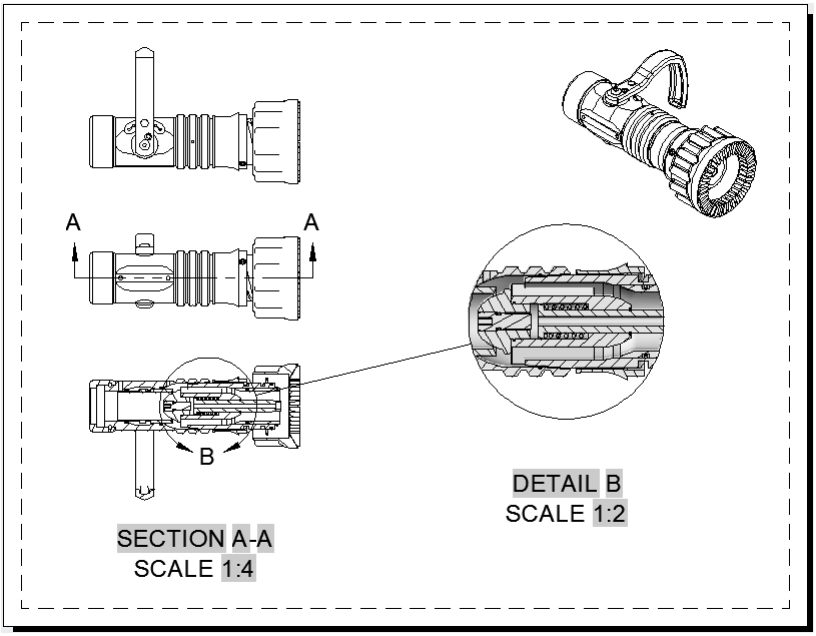
Since you want the detail view to always be aligned with the centerline of the nozzle, in the **Status** bar, make sure that the **Infer Constraints** option is toggled on. Also make sure that **Polar Tracking**, **Object Snap Tracking**, and **Object Snap** are active, and that **Midpoint** is one of the running object snaps.

Then, track off the midpoint of the left edge of the nozzle along the 0-degree angle alignment until the cursor is located approximately in the middle of the nozzle, and then click to specify the center point.

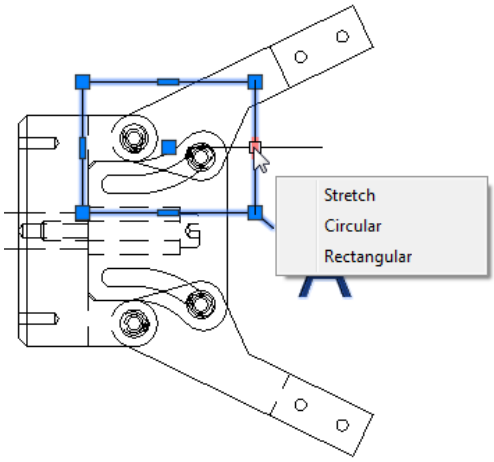


Next, the program prompts you to specify the size of the boundary. Move the cursor down until you are satisfied with the size of the circular boundary, and then click. As soon as you do, a detail view identifier is placed at the point you selected and AutoCAD displays a preview of the detail view at the cursor. The program prompts you to specify the location of the detail view.

Click to place the view. The command remains active, and you can continue to use the options or the tools in the ribbon to adjust the detail view. For example, you can change any of the appearance, boundary, model edge, or annotation settings. Once you are satisfied, click **OK** or press **ENTER** to end the command.

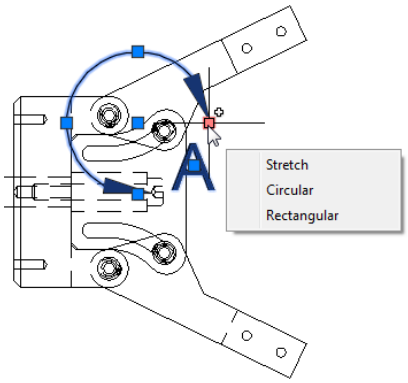


After you have created a detail view, you can edit the view as well as many of the elements associated with the view. For example, if you click **Edit View** and then select the detail view, you can change the hidden line style, the view scale, the boundary type, the model edge, and the view identifier. You can also toggle the view identifier or hatching on or off. All of the changes you make appear in real time, unless you expand the **Edit** panel and toggle on **Defer Updates**.



If you change the shape or size of the view boundary, any changes you make are reflected in the associated detail view. Note that you can select the detail view boundary and then use grips to edit the boundary. Use the square grip at the center to move the center point of the detail view. If you move the cursor over any of the other grips, AutoCAD displays an extended tooltip. You can use tools in the tooltip to change the size or shape of the detail view boundary.

You can also modify the detail view identifier. For example, you can change the position of the identifier or change the label. To change the position of the detail view identifier, first select the detail view border. Then, move the cursor over the identifier. When the extended tooltip appears, you can use tools to move the identifier or reset it back to its default position.



You can also change the identifier label. To do this, first select the detail view and edit it. You can then change the identifier label by entering a new value in the **Identifier** field in the **Detail View Editor** contextual ribbon. Note that you can also select the detail border, and then make this change using the **Properties** palette.

You can also modify the detail view label much the same as you would modify any multiline text object. For example, when you double-click a detail view label, the ribbon changes to the **Text Editor** contextual ribbon. Note that the actual detail name and scale are fields. You can double-click these elements to edit the field formatting, but you should be careful not to change or delete the fields themselves. And if you wanted to change something about the formatting of the label—for example, if you wanted to include a colon after the word SCALE in the label—it would be much more efficient to modify the detail view style than to make this type of change by manually editing each detail view label.

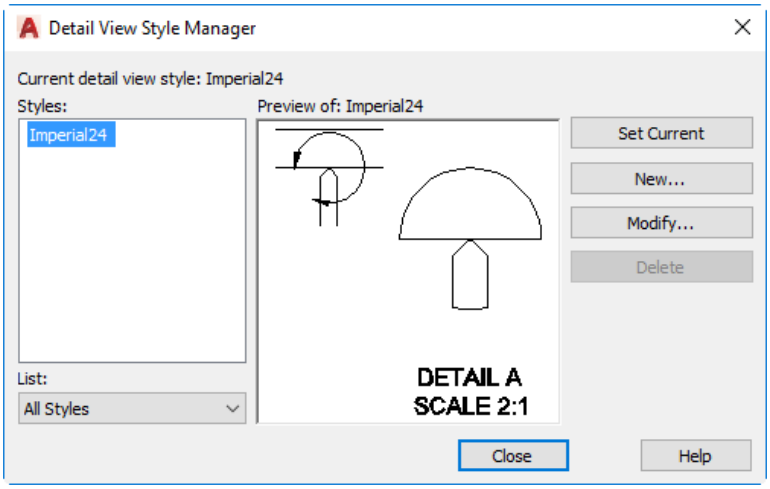
Controlling the Detail View Style

When you create a detail view, the appearance of the view—including the view identifier, the detail boundary, and the view label—are determined by the current detail view style.

Detail view styles provide the means to quickly specify the format of the various detail view elements. They also provide a convenient way to ensure that detail views conform to drafting standards. AutoCAD comes with two predefined detail view styles—Imperial24 and Metric50—that conform to the ANSI and ISO drafting standards, respectively. Of course, like other styles in AutoCAD, you can modify these styles or create and save additional custom styles.

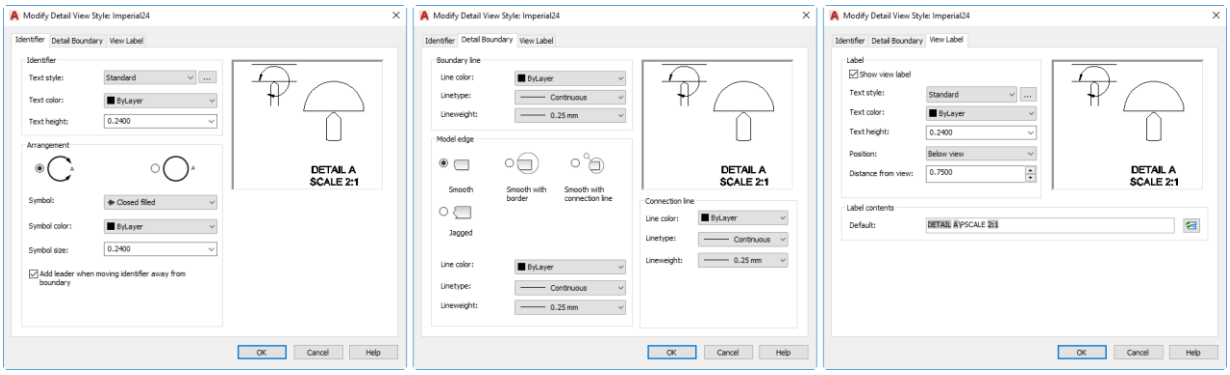
To create a new detail view style or to modify an existing style, on the **Layout** ribbon, click the **Detail View Style** button to display the **Detail View Style Manager** dialog. The **Styles** list on the

left shows the styles currently available in the drawing, and when you select a style in the list, you see a preview of the style in the adjacent window.



To modify an existing style, select it in the list and click the **Modify** button. To create a new style, click the **New** button. AutoCAD displays a **Create New View Style** dialog so that you can name the new style. Then, AutoCAD displays the **New Detail View Style** dialog. This dialog contains three tabs. The controls on the **Identifier** tab determine the properties for the identifiers used by the detail boundary. In the **Identifier** area, you can control the text style, text color, and text height. In the **Arrangement** area, you can control whether the identifier is placed in a gap in the detail view boundary or not, and you can choose the type, size, and color of the arrowhead used. You can also control whether a leader is added if you move the identifier away from its default position.

The controls on the **Detail Boundary** tab determine the properties for the detail boundary line, model edge, and connection line. And the controls on the **View Label** tab determine the properties for the text, style, placement, and content of detail view labels.

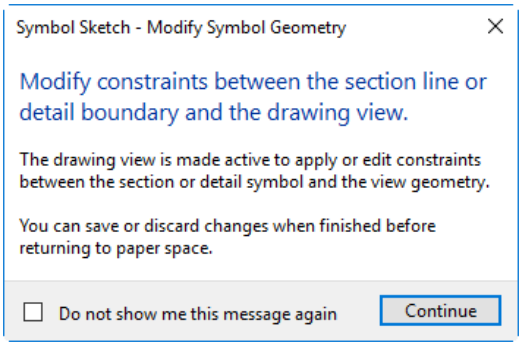


Once you have created additional detail view styles, the new styles appear in the **Detail View Style** drop-down on the **Layout** ribbon and you can easily make a style current or apply a new style to an existing detail view.

Constraining Drawing Views to Model Geometry

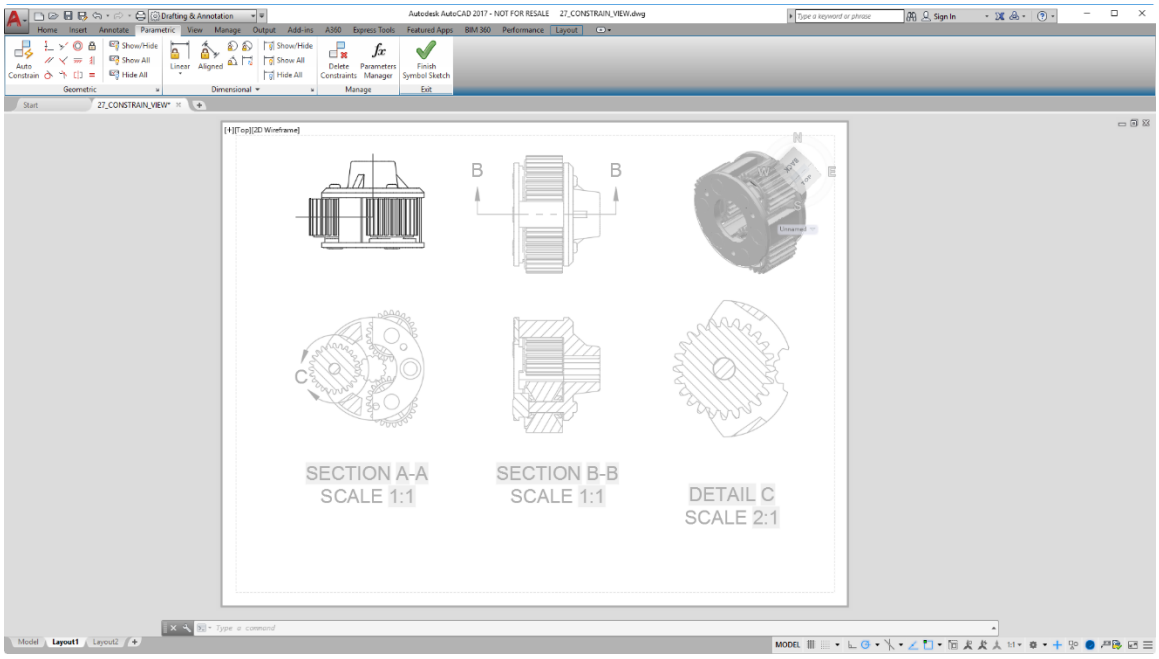
Typically, constraints are inferred between the view geometry and the section line or detail boundary. In most cases, this is sufficient. However, there may be times when you need to add additional constraints. For example, you may need to ensure that a section line passes through a part exactly 15 units from a particular edge. In such cases, you can manually constrain the section

line or detail view boundary to the view geometry using a special edit environment called the *symbol sketch mode*.



To manually add constraints to a section line, on the **Layout** ribbon, in the **Modify View** panel, click the **Symbol Sketch** tool. AutoCAD prompts you to select a section or detail symbol. Click the section line you want to constrain. As soon as you do, AutoCAD displays a dialog to inform you that you are about to modify the constraints between the section line or detail boundary and the drawing view. The drawing view will be made active so you can apply or edit constraints between the section or detail symbol and the view geometry. Once you are done editing the constraints, you will be able to save or discard changes before returning to paper space. Also note that you can hide this message so that you do not see it again. Click **Continue**.

You have just entered the symbol sketch mode. With the exception of the geometry associated with the section view you are editing, all of the geometry has become dimmed, including the section line. The line you see is actually a polyline that coincides with the section line.



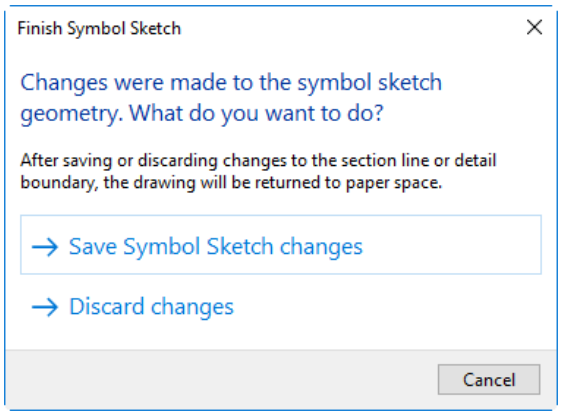
Also notice that the **Parametric** ribbon has become active. You can now use the tools on this ribbon to constrain the cutting plane. For example, you could add a horizontal constraint to the horizontal segment and a vertical constraint to the vertical segment. You could also add dimensional constraints to ensure that the horizontal line remains a specific distance from a specified point on the geometry.

If necessary, you can also add construction geometry such as lines and arcs to help you implement constraints. Any geometry you add while working in symbol sketch mode is considered construction geometry and will not be visible when you exit symbol sketch mode.

Once you have added the necessary constraints, on the ribbon, click **Finish Symbol Sketch**.

AutoCAD displays a **Finish Symbol Sketch** dialog to ask what you want to do with the changes you just made. You can either save or discard the symbol sketch changes. When you click **Save Symbol Sketch Changes**, the drawing immediately updates, and you are back in paper space.

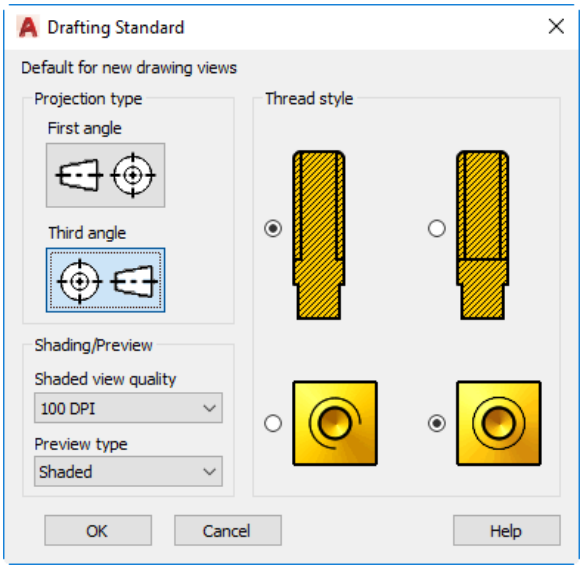
You can use these same methods to constrain detail boundaries.



Setting the Drafting Standards for Drawing Views

When you create drawing views, those views are based on the current drafting standards. Those settings impact all new drawing views you create, but any changes you make to the drafting standards have no effect on any existing drawing views.

Before creating drawing views, you can view and modify the drafting standards on which those views will be based. To do so, on the **Layout** ribbon, in the **Styles and Standards** panel, click the dialog box launcher to display the **Drafting Standards** dialog.



In the **Projection Type** area, you can set the projection angle for drawing views. The projection angle defines where projected views are placed. For example, if the active projection type is set to First angle, top views are placed below the parent view. When set to Third angle, top views are placed above the parent view.

In the **Thread Style** area, you can set the appearance of thread edges in the drawing. Note that you can control the thread edge appearance separately for section views and when thread edges appear in circular projections. Also note that the thread style settings only apply to drawing views based on Autodesk Inventor models.

In the **Shading/Preview** area, you can choose the shaded preview quality and preview type. For example, in the **Shaded View Quality** drop-down, you can specify the resolution of the shaded preview. While higher resolutions provide better quality previews, resolutions above 150 dpi may not be achievable for larger models. When the specified resolution is not achievable, the program automatically drops the quality to an achievable level.

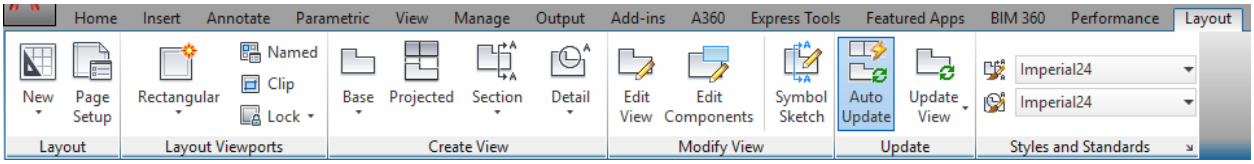
In the **Preview Type** drop-down, you can choose whether AutoCAD displays a shaded view when you are creating a projected view or just shows a bounding box. Note that the settings in the **Shading/Preview** area only affect the preview. Once the view has been placed, its appearance will be based on the actual drawing view settings.

Once you have finished adjusting these settings, click **OK** to close the **Drafting Standard** dialog.

Updating Drawing Views

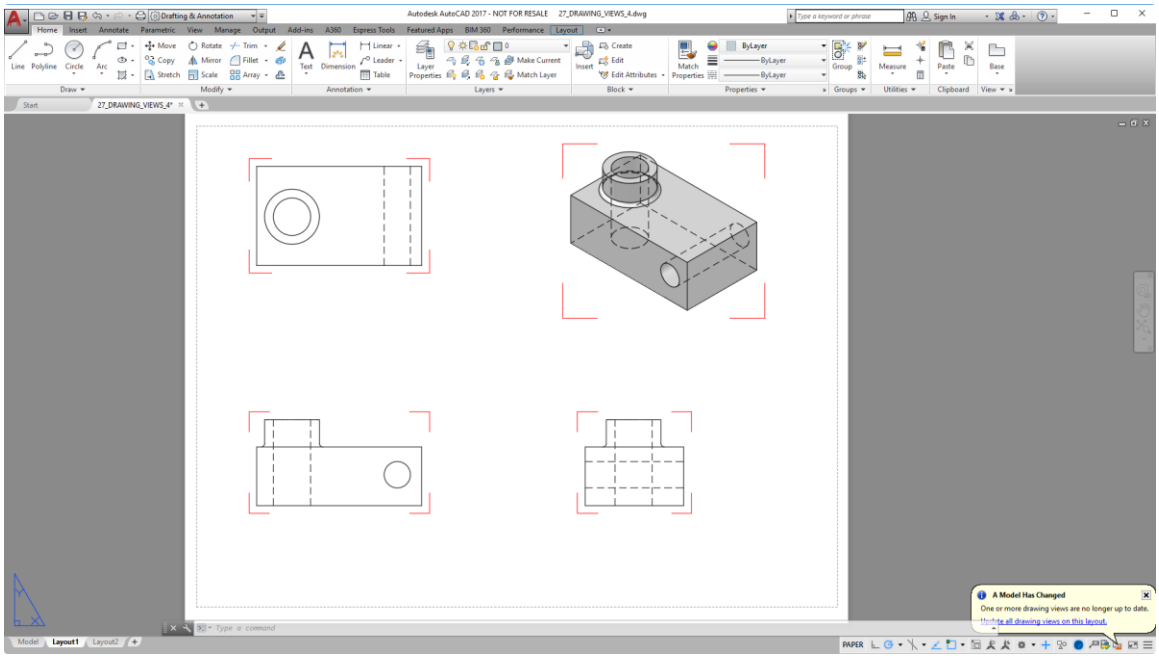
While you cannot modify the geometry within drawing views, drawing views remain associative to the model from which they were created. If you make any changes to the 3D model, the drawing view geometry can be updated to reflect those changes.

On the **Layout** ribbon, in the **Update** panel, the **Auto Update** toggle is typically selected, as indicated by its blue background. When **Auto Update** is on, if the 3D model changes, all of the drawing views update automatically. When **Auto Update** is off, however, the drawing views do not update automatically.



To better see how this works, toggle off **Auto Update**, and then make a change to the 3D model. Then, switch back to the paper space layout.

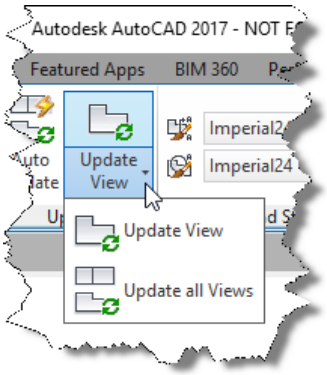
When **Auto Update** is off, if the 3D source model changes, a balloon message displays next to the drawing view icon in the Status bar in the lower-right corner of the application window. Red markers also appear at the corners of any drawing views that are no longer up to date.



Notice that the balloon includes a link. If you click this link, all of the out-of-date drawing views are immediately updated to reflect the changes made to the 3D model.

If instead you close the balloon without updating the views, the red markers remain in the corners of the out-of-date views, and you can see an alert symbol on the drawing view icon on the Status bar. In that case, you can update the views manually, by either right-clicking the drawing view icon on the Status bar, or by using the **Update View** tools in the **Update** panel on the **Layout** ribbon.

You can update either an individual view or all views. For example, if you choose **Update View**, the program prompts you to select the view you want to update. Select a view. Although you can only select views by clicking on them, you can select more than one view. Once the views you want to update are selected, press **ENTER**. The views you selected are immediately updated. You will still see red markers in the corners of the other views that are still out of date.



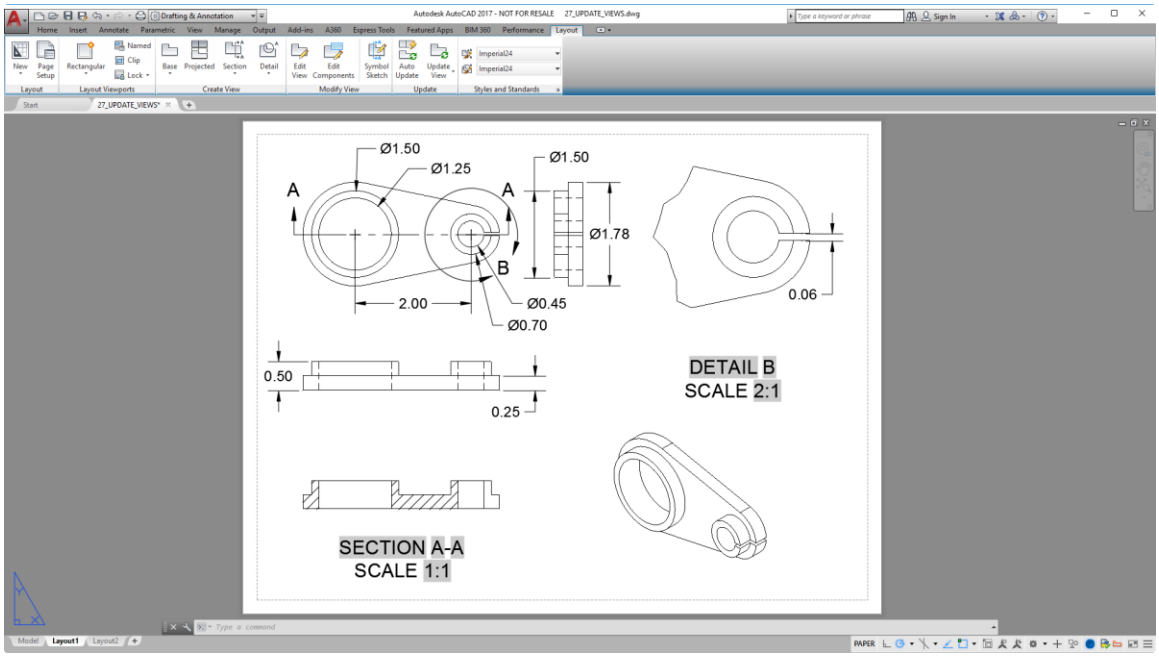
If you use the view update tool and choose **Update All Views**, any views that are still out of date are immediately updated to reflect the changes made to the 3D model.

Monitoring Drawing View Annotations

Once you have created drawing views, you typically add annotations to those views—such as dimensions and notes—in paper space. Those annotations are associated to the drawing view based on the vertices selected or inferred by the selected edge. As a result, if you move, scale, or update the drawing view, the associated annotations remain associated.

Since annotations are associated to the drawing view and the drawing view is associated to the 3D model, it is possible for changes made to a drawing view or to the 3D model to invalidate or *disassociate* annotations.

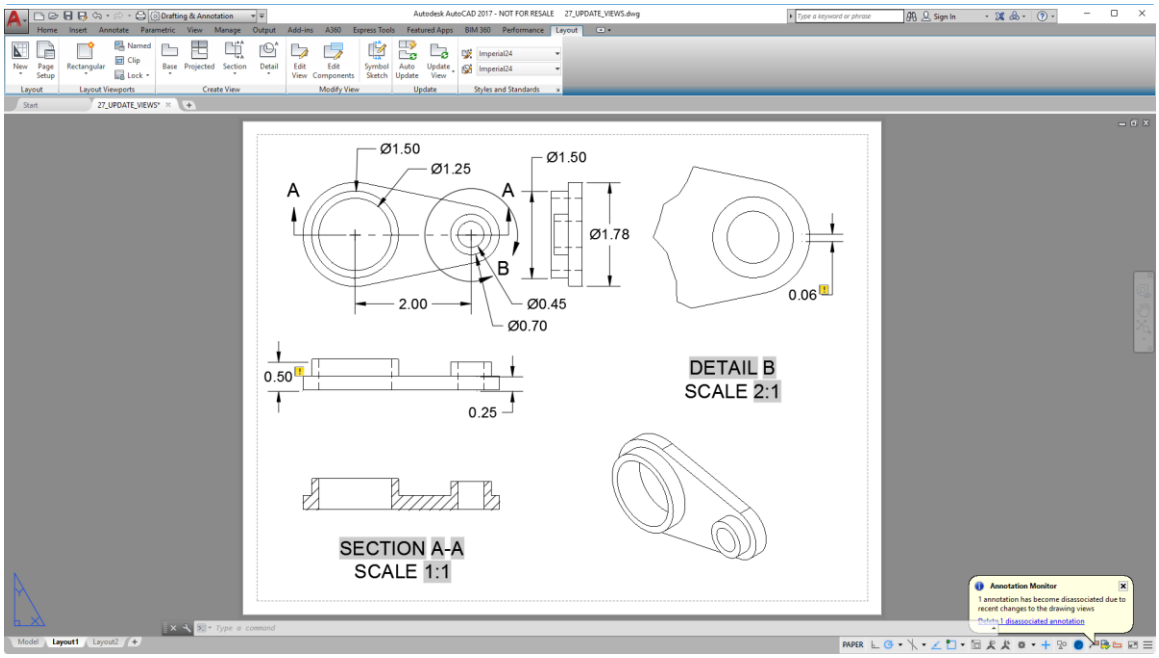
AutoCAD provides a service called Annotation Monitor that monitors the status of annotations and immediately alerts you if any change causes annotations to lose their associativity. You can toggle **Annotation Monitor** on and off using the tool on the **Status** bar. Typically, you should always leave this toggled on.



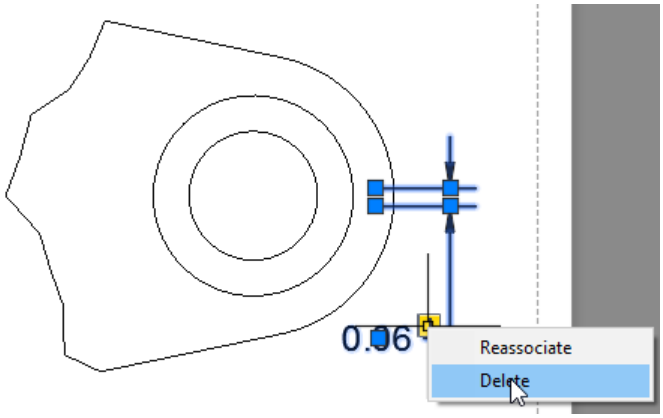
For example, here we see drawing views of a small part. Also notice that on the **Layout** ribbon, in the **Update** panel, **Auto Update** is toggled on by default. You would typically leave this toggled on, but toggle it off for now to better see how this works. Then, switch to model space and make a change to the 3D model.

When you switch back to the paper space layout, since the 3D source model has changed, AutoCAD displays an alert balloon in the Status bar and also adds red markers to indicate any views that are out of date. You must now update those views to reflect the changes. Had **Auto Update** been toggled on, you would not see this alert or these markers. Instead, AutoCAD would have automatically updated the views.

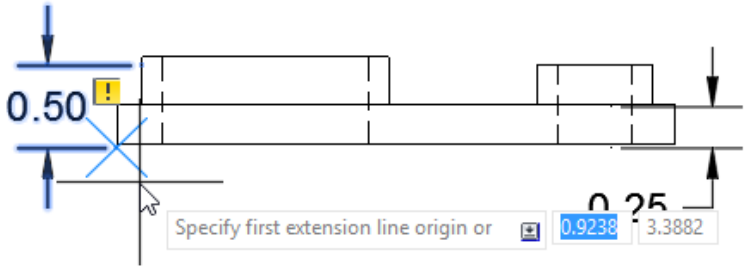
Since you do want to update all of the views, click the link in the alert balloon to update all of the views at once. As soon as you do, the program may display another alert balloon to inform you that a number of annotations have become disassociated due to the changes to the drawing views. In the **Status** bar, you can also see that the **Annotation Monitor** tray icon has turned red to indicate that annotations have become disassociated. In addition, the annotation monitor displays a special badge near each of the affected annotations.



Notice that the balloon includes a link. If you were to click this link, all of the disassociated annotations would be deleted. While the dimension in Detail B is no longer valid and can be deleted, you do not want to delete the 0.5 unit dimension in the front view, so simply close the alert balloon.



Instead, click the badge adjacent to the dimension in the detail view. As soon as you do, AutoCAD displays a menu. You can either reassociate the dimension or delete it. Since this dimension is no longer valid, click **Delete**.



Then, click the badge in the front view and choose **Reassociate**. AutoCAD prompts you to specify the first extension line origin or you could use the **Select Object** option. The program also displays a special icon to indicate the disassociated extension line origin. Snap to the endpoint of the top of the part. Then, the program prompts you to specify the second extension line origin, and again it displays a blue marker. Select the endpoint of the bottom of the part. As soon as you do, the dimension moves and the command ends. The annotation is once again properly associated with the part.

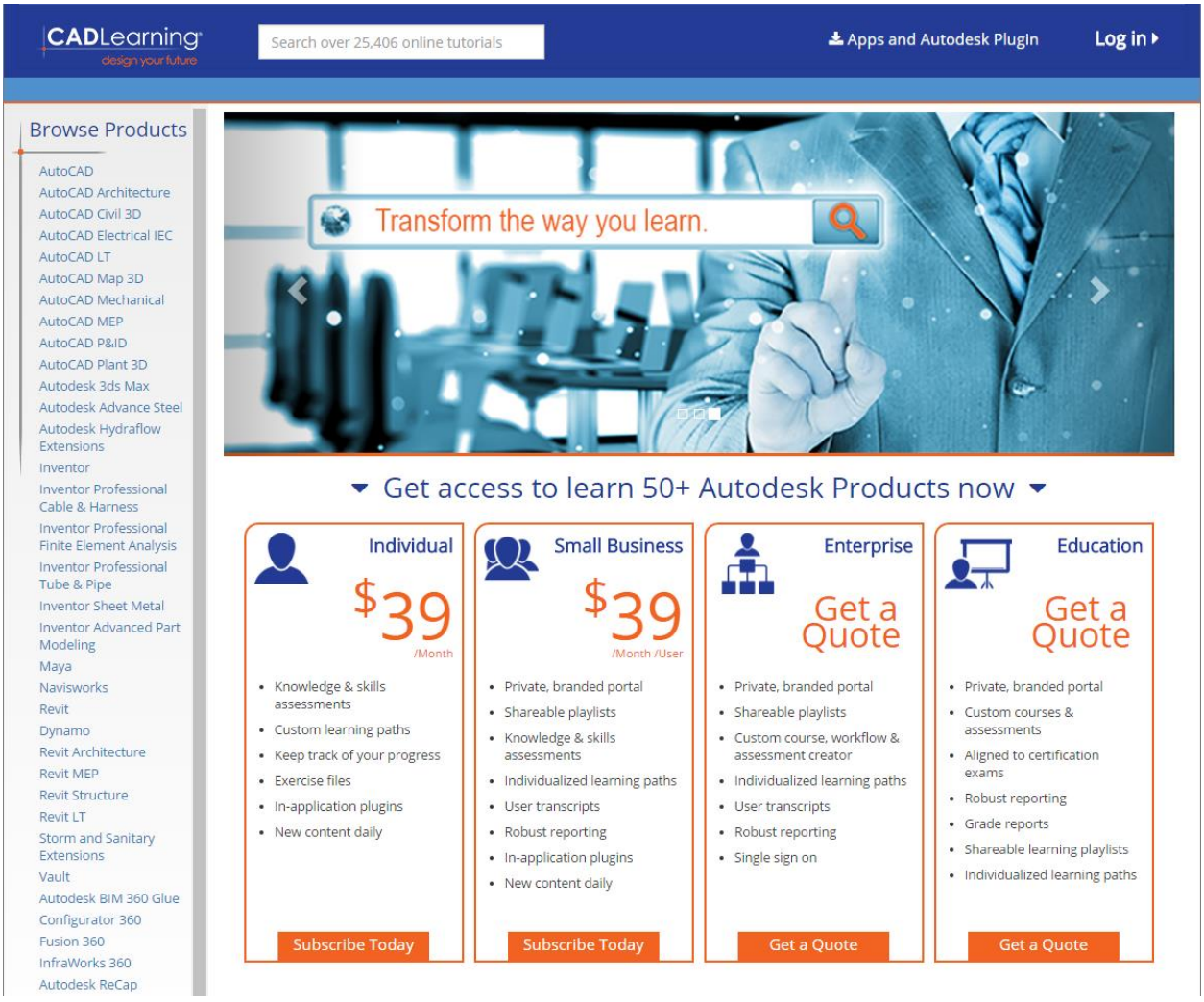
Improvements in the latest release of AutoCAD greatly reduce the chance of dimensions becoming disassociated when drawing views are updated, so you may not encounter this problem. But if you do, you now know how to quickly remedy the situation.

Conclusion

That concludes this class on AutoCAD's Drawing View tools. As you have seen, rather than manually producing two-dimensional drawings to document your three-dimensional designs, you can use these Drawing View tools to create standard orthographic, isometric, section, and detail views directly from your 3D models, and then easily update those views if you subsequently make any changes to those models.

Want to learn more?

To learn even more about model documentation as well as other features and functions of AutoCAD and other Autodesk software, visit 4D Technologies at https://www.cadearning.com.



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