



Create Compelling Sections, Details, and Auxiliary Views Using Drawing Views in AutoCAD® 2014

J.C. Malitzke - Digital JC CAD

AC1634-L This intermediate-to-advanced hands-on lab offers AutoCAD 3D veterans a chance to explore the 2D model documentation of 3D models from AutoCAD 2014 and Autodesk® Inventor® software. After creating base views and projected views from 3D solid or surface models, we explore sectional views. Using 3D models, we create Full, Half, Offset and Aligned sectional drawing views. We create circular and rectangular detail views. We edit the 3D models and update the derived drawing views. We add dimensions to the drawing views and modify the 3D model to update the dimensions. We explore the creation of undocumented Auxiliary views. If you used AutoCAD 3D in the past to create 2D multi-view drawings from 3D models, attend this class and get ready to be surprised.

Learning Objectives

At the end of this class, you will be able to:

- Create drawing views of AutoCAD and Inventor 3D models for drawing 2D sections and details
- Add dimensions to 2D drawing views
- Edit 3D models and their 2D associated drawing views
- Create undocumented Auxiliary views from AutoCAD 3D models

About the Speaker

J.C. Malitzke is President of Digital JC CAD Services Inc. and is the former department chair of Computer Integrated Technologies and a faculty member at Moraine Valley Community College in the greater Chicago area. He managed and taught for the college's Autodesk Authorized Training Center. He has been using and teaching Autodesk® products for 28 years. J.C. is co-author to Good-Heart Wilcox Publisher for AutoCAD® and Its Applications Advanced. He is the recipient of several educator awards, including, Professor of the Year, and the Illinois Trustee Association's Faculty Member of the Year, and a top presenter award winner at Autodesk University. J.C. is a Certified Autodesk Instructor for AutoCAD® and Autodesk® Inventor and is an Autodesk Certification Evaluator. This is his 19th year presenting at Autodesk University. He holds a BS degree in education and a MS in industrial technology from Illinois State University. Contact J.C. at: digitaljc@digitaljccad.com

Portions of this document are copyright by Goodheart-Willcox Company, Inc. and reproduced with permission from the textbook AutoCAD and its Applications--Advanced.

For AutoCAD 2013, 2014

Section Views

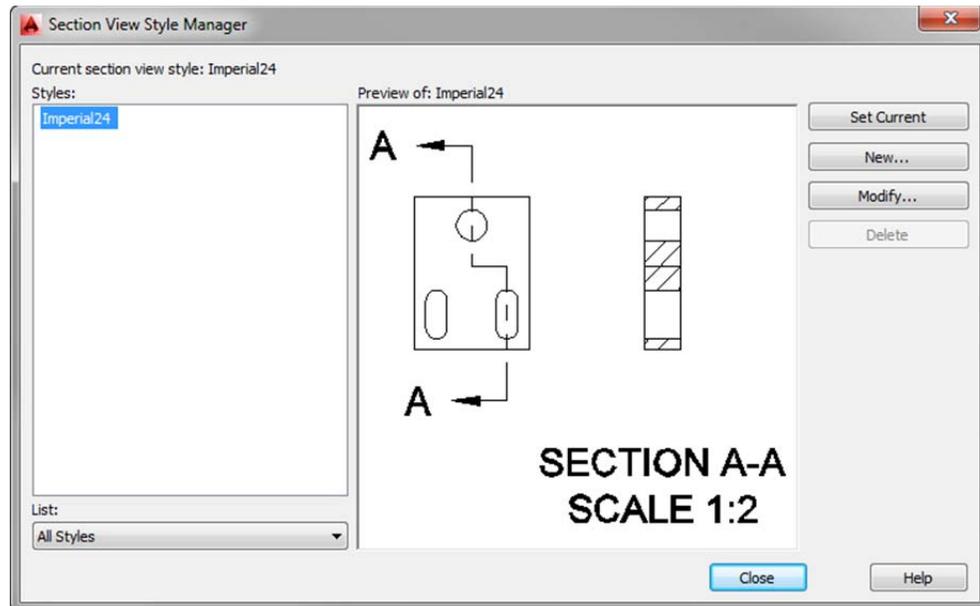
A **section view** shows the internal features of an object along a section line (cutting plane). A section view is projected from an existing view, such as an orthographic top view. The existing view serves as a parent view. To create a section view, you pick points on the parent view to define the section line (cutting-plane line). You can also select an object, such as a line or polyline, to define the section line. Section views are created using the **VIEWSECTION** command. This command can be used to create full, half, offset, or aligned sections from an AutoCAD 3D model or an Autodesk Inventor file.

Section views created with the **VIEWSECTION** command are created in the same paper space layout as other drawing views. Section views are *associative*. A section view is linked to the parent view that creates the section view. As with other types of drawing views, section views are updated automatically when model changes are made if the **VIEWUPDATEAUTO** system variable is set to 1.

By default, a section identifier is placed with the section line and a section view label is placed with the section view when you create the view. The section identifier is automatically incremented when you place additional section views. The text objects used for the section identifier and section view label contain fields that update according to changes made to the section view. The appearance of elements in the section identifier and section view label is controlled by the section view style. A **section view style** defines settings such as the text style and height, direction arrow size and length, and hatch pattern used for sectioning. A section view style is similar to a dimension style and includes similar controls.

VIEWSECTIONSTYLE

The **VIEWSECTIONSTYLE** command is used to create and modify section view styles. This command accesses the **Section View Style Manager** dialog box.



Picking the **New...** button allows you to create a new section view style. Picking the **Modify** button opens the **Modify Section View Style** dialog box for the selected style. The tabs in the **New Section View Style** dialog box or the **Modify Section View Style** dialog box are used to make settings for the section identifier, cutting-plane line, section view label, and section line hatching. You can apply, modify, and delete section view styles during the design process or after a section view is created. Develop standards for section views similar to the standards you develop for text and dimensions. Section view styles should follow company or industry standards.

Once created, section views can be edited by editing the section line and editing properties of the section view, such as the hatch pattern used for the section. The following sections discuss techniques for creating and editing section views.

NOTE

New layers are created by AutoCAD for the drawing view geometry when a section view is created. The section line and section view label are placed on a layer named **MD_Annotation**. The section pattern is placed on a layer named **MD_Hatching**.

Full Section

VIEWSECTION

A full section “cuts” the parent view of the object in half. It is created by making a cut completely through the object.

To create a full section, select the **VIEWSECTION** command and then select the parent view. The parent view can be a base view or a projected view. Next, select the **Type** option and then select the **Full** option. This option can be accessed directly from the ribbon by selecting the **Full** option from the **Section** drop-down list in the **Create View** panel of the **Layout** tab. Once you select the parent view, you are prompted to specify the start point of the section line. Use object snaps and object snap tracking to assist in specifying the start point. Then, drag the cursor and pick the end point of the section line. The section line is created and a preview of the section view is aligned perpendicular to the section line. The dragging direction from the section line determines the viewing direction. Move the section view to the desired location and pick. To break the alignment between views, press the [Shift] key once. To restore the alignment, press the [Shift] key again. When you pick a location for the view, you can select an option or press [Enter] to exit the command. You can adjust options using the **Section View Creation** contextual ribbon tab. You can also use dynamic input or the command line.

The **Hidden lines**, **Scale**, and **Visibility** options allow you to adjust the display style, scale, and edge visibility. These are the same options available with other types of drawing views. The **Projection** option is used to set the type of projection when creating a section line with multiple segments. The **Orthogonal** option projects the view orthogonally and creates a true projection. This is typically preferred, depending on the orientation of the section line. The **Normal** option projects the view normal to the cutting plane and is preferred for certain section line orientations, such as an angled line used to create an aligned section in accordance with conventional drafting practices. The **Depth** option is used to control the visibility of objects “behind” the section line. When you select this option, a depth line appears at the section line. Hovering over this line and dragging allows you to set the depth of the section view. Objects that are behind the depth line will not be visible in the section view. Selecting the default **Full** option includes all objects within the section view. Selecting the **Slice** option removes all objects behind the section line, creating a thin representation section view. The **Slice** option may be practical for special section view documentations.

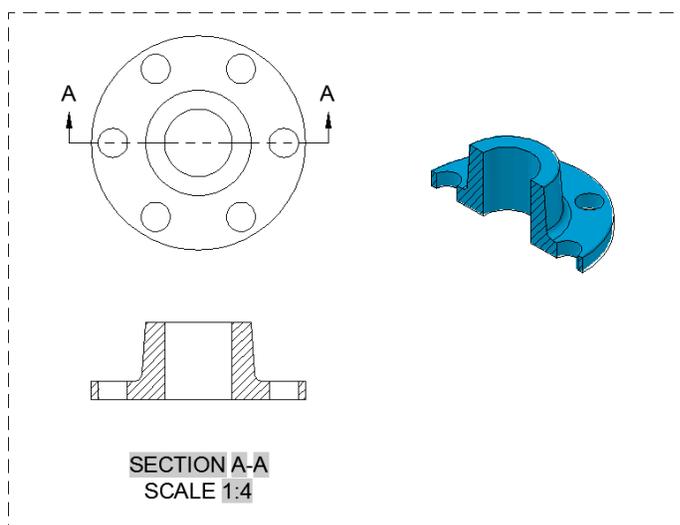
The **Annotation** option allows you to enter the text used for the section identifier and

specify whether a view label is shown. As previously discussed, the section identifier is automatically incremented when creating additional section views. The **Hatch** option is used to specify whether a hatch pattern is used for the section view. The **Move** option allows you to adjust the location of the view after selecting the initial position. When using this option, you can press the [Shift] key to break the alignment between views.

Section views can be used to create projected views. An isometric view has been created from a full section view to show an isometric representation of the “cut.” By default, an isometric view projected from a section view shows the section and inherits the display properties of the parent view.

Exercise 1 Full Section

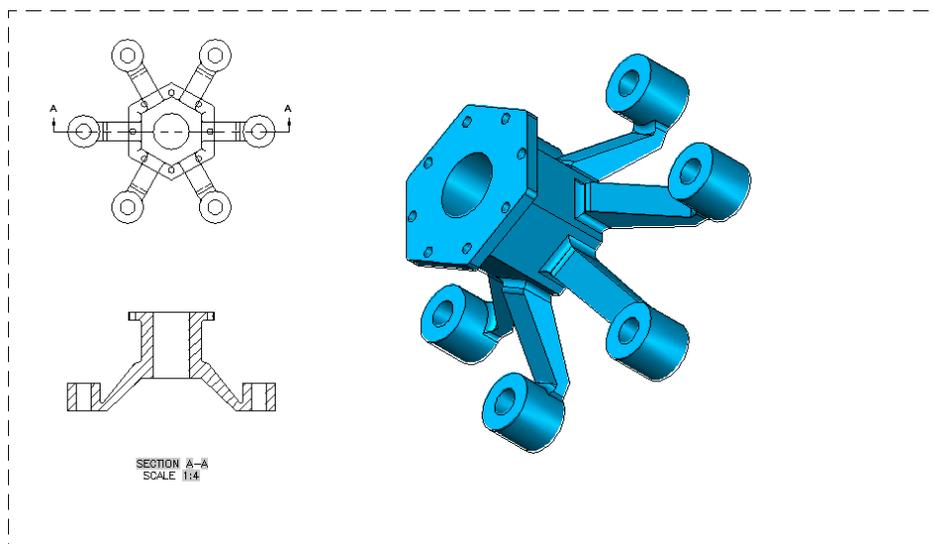
1. Open the drawing **Flanged_Coupler_FullSection.dwg**.
2. Select the ANSI A layout.
3. On the ribbon, select the Layout tab.
4. From the Create View panel, create a base view From Model Space as the Top view. Set orientation to Top with Hidden Lines set to visible and hidden lines.
5. Create a Full section as the front view.
6. Add a projected isometric view projected from the front view.
7. Edit the isometric view to shaded with visible lines. Double click on the isometric view and set Hidden Lines to Shaded with visible lines.
8. Reset the Scale of all views to **1:4**.
9. Double click on the hatch pattern in the front and isometric views Set the hatch scale to **.75**.
10. The final drawing is shown below.
11. Save the drawing as **Full Section.dwg**



Exercise 2 Full Section

An example of using the **Depth** option to adjust a half section is shown below. The full section is created with the section line drawn through the middle of the object. The **Depth** option is selected to move the depth line to the back end of the hole feature. In this case, adjusting the depth of the section line helps clarify the interior detail of the part.

1. Open the drawing **SupportBrace.dwg**.
2. Select the B-Size layout.
3. Create a base view as the top view.
4. Create the front view as a Full section view from the top view. Click on the left circle that represents the foot in the top view to use the start point of the section line (cutting plane line). Hint: turn on tracking and object snap tracking to track for objects in the base or parent view for starting the first section lines endpoint.
5. Click across the part tracking from the left center circle past the right circle. Place the section line as shown.
6. View the preview of the section by moving the section view in-place as shown. The preview will be aligned perpendicular similar to placing drawing views.
7. Set the Depth option so the depth line distance turns off the visibility of objects behind the depth line.
8. Select a location for the section view and the O.K.
9. Create a project isometric view from the Top view.
10. Scale the isometric view **1:2**.
11. Save the drawing as **SupportBrace_Finished.dwg**



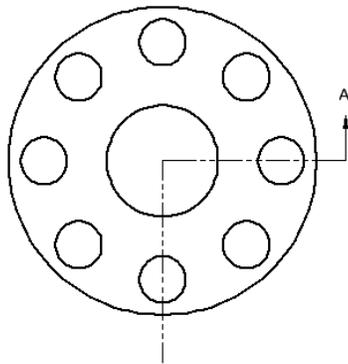
Half Section

VIEWSECTION

A half section is half of a full section. It represents one-quarter of the object cut away. Half sections are most typically used for symmetrical objects. The half of the object that is not sectioned is usually shown as a solid object with no hidden lines.

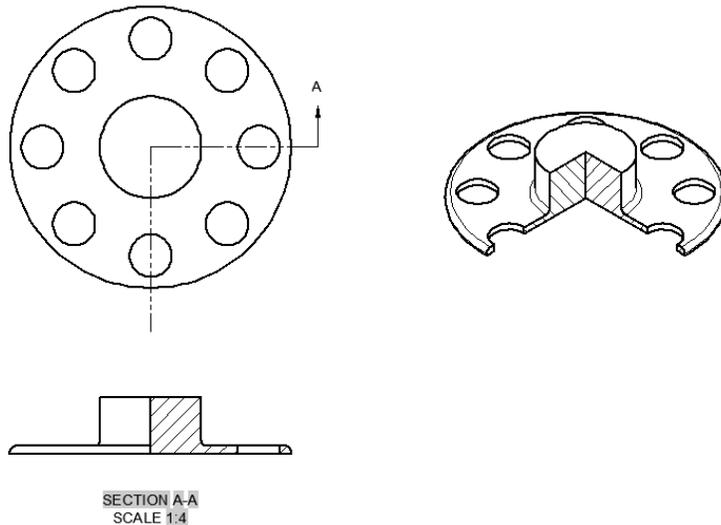
To create a half section, select the **VIEWSECTION** command, select the parent view, and access the **Half** option. The option is accessed directly from the ribbon by selecting the **Half** option from the **Section** drop-down list in the **Create View** panel of the **Layout** tab. Three points are required to define the section line. Use object snaps and object snap tracking to assist in specifying each point. If you pick an incorrect point, use the **Undo** option. After drawing the final segment of the section line, pick to locate the view. You can then select an option or press [Enter] to exit the command. You can adjust options using the **Section View Creation** contextual ribbon tab, dynamic input, or the command line. The options are the same as those available when creating a full section.

1. Open the drawing **Half_Section.dwg**.
2. Select the ANSI A layout.
3. Create a base view as the Top view.
4. Create a half section as the front view. Set scale to 1:4.



SECTION A-A
SCALE 1:4

5. Create the isometric view as shown. Project from the front view.
6. Turn on Show/Hide lineweights. If all the drawing views are not currently on the MD_Visible layer, use Properties to change all drawing views to the MD_Visible layer. (The visible lines of the part should be wider than the section line and hatch lines).



7. Save the drawing as **Half_Section-Finished.dwg**.

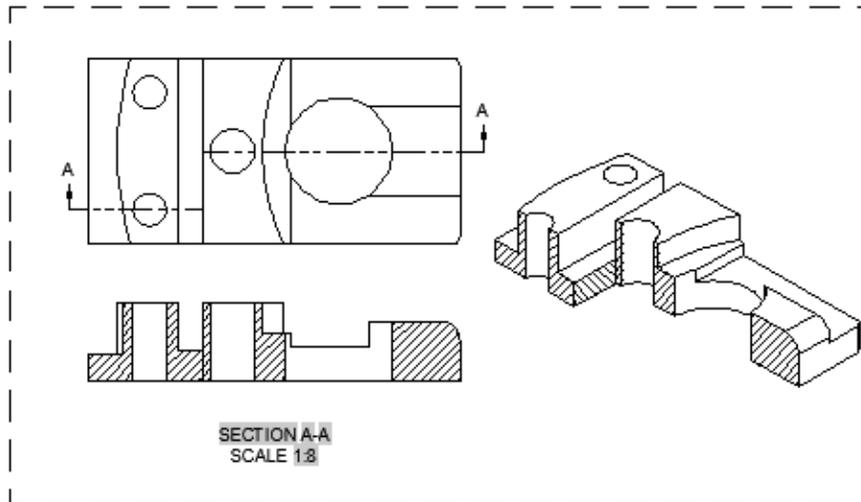
Offset Section

VIEWSECTION

An offset section *shifts* (offsets) the section line to pass through certain features of a part or assembly for better clarification of detail. Typically, the section line consists of several segments drawn through features such as holes and bosses.

To create an offset section, select the **VIEWSECTION** command, select the parent view, and access the **Offset** option. The option is accessed directly from the ribbon by selecting the **Offset** option from the **Section** drop-down list in the **Create View** panel of the **Layout** tab. Then, pick the points to define the section line. Select as many points as needed to define the section. Use object snaps and object snap tracking as needed. If you pick an incorrect point, use the **Undo** option. After drawing the final segment of the section line, select the **Done** option. Then, pick to locate the view. You can then select an option or press [Enter] to exit the command. The options are the same as those available when creating a full section.

1. Open the drawing **Offset_Base_Section.dwg**.
2. Select the ANSI B Layout.
3. The top view has been created for you.
4. Create an offset section as the front view. Scale is set to 1:8.



5. Project from the front view an isometric section view.
6. Change the isometric view to shaded with visible lines.
7. Edit the isometric view by using **Cut Inheritance**. Set **cUt** to **No**. **What happens to the isometric view?**
8. Save the drawing as **Offset Section.dwg**.

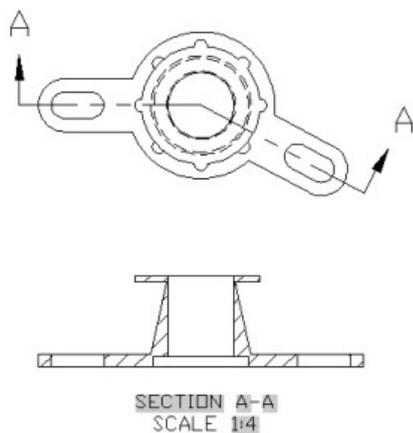
Aligned Section VIEWSECTION

An aligned section is made by passing two nonparallel cutting planes through an object. The resulting section view shows features that are oriented at an angle rotated into the same cutting plane.

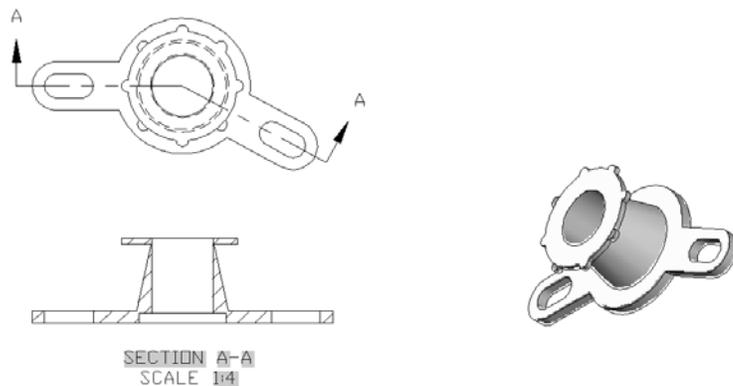
The purpose of an aligned section is to show the true size and shape of a feature. An aligned section is created to show the true size and shape of the right arm. Notice that the right arm is rotated into the center cutting plane to project the view. For this view, the **Projection** option is set to **Normal**. This is conventional practice. To create an aligned section, select the **VIEWSECTION** command, select the parent view, and access the **Aligned** option. The option is accessed directly from the ribbon by selecting the **Aligned** option from the **Section** drop-down list in the **Create View** panel of the **Layout** tab. Then, pick the points to define the section line. Use object snaps and object snap tracking as needed. After drawing the final segment of the section line, select the **Done** option. Then, pick to locate the view. You can then select an option

or press [Enter] to exit the command. The options are the same as those available when creating a full section.

1. Open the drawing **Aligned-Section.dwg**.
2. Select the ANSI A Layout.
3. The top view has been created for you.
4. Create an aligned section as the front view.



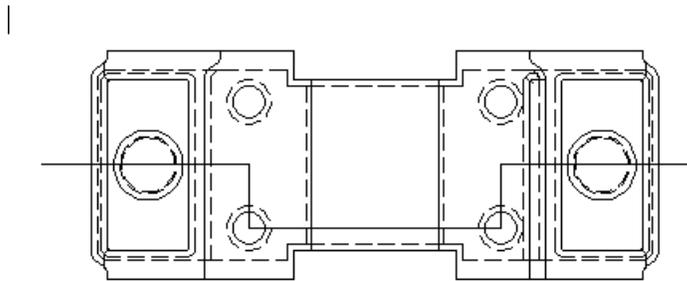
5. Create a new view section style and apply with the text height set to .125.
6. Double click on the front view to set the projection from orthogonal to normal.
7. Project the top view as an isometric view.
8. Edit the isometric view by turn off tangent edges.
9. Edit the isometric view by changes the appearance to shaded with visible lines.



10. Save the drawing as **Aligned.dwg**.

Creating a Section View from an Object

You can select an object in the paper space layout to use as the section line when creating a section view. This is a useful method when it is difficult to locate points using the **VIEWSECTION** command. To use an object as the section line, select the **VIEWSECTION** command, select the parent view, and access the **Object** option. The option is accessed directly from the ribbon by selecting the **From Object** option from the **Section** drop-down list in the **Create View** panel of the **Layout** tab. Then, select the object and press [Enter]. Pick a point to locate the view. The object you select determines the type of section created. A **polyline** is drawn in the desired location prior to accessing the **From Object** option of the **VIEWSECTION** command. This is an alternate way to create the section view and may be easier than picking points. When using the **Object** option, the selected object is automatically deleted after creating the section view.



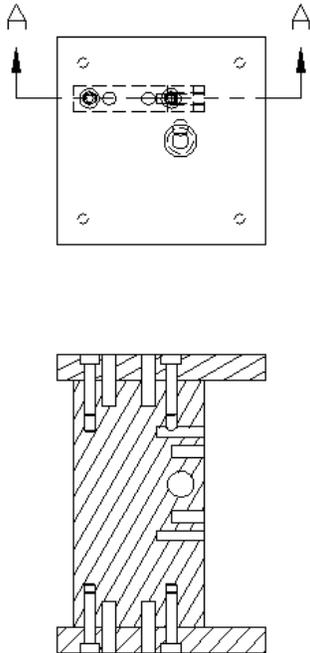
Constraining Section Lines

When picking points to define a section line, you have the option to apply constraints to control the location of the line. If **Infer Constraints** is activated, constraints are automatically applied when you pick points. The **Infer Constraints** button on the status bar controls whether constraints are inferred when drawing new objects. Inferring constraints provides a way to constrain the geometry of the section line to the geometry in the drawing view. This is similar to applying constraints automatically when drawing objects in model space.

Excluding Components from Sectioning

Certain features in section views, such as fasteners, are not shown sectioned. For example, components such as screws, pins, and thin-walled objects in an assembly are shown without section lines. This practice conforms with drafting standards. When creating a section view from

a parent view that includes items such as fasteners and shafts, you can use the **VIEWCOMPONENT** command to control how sectioning is applied.



Editing Section Views

The **VIEWEDIT** command, can be used to edit the properties of a section view. You can quickly initiate this command by double-clicking on a section view. This displays the **Section View Editor** contextual ribbon tab. Many of the same options used when creating a section view are available in this tab. Additional options may be available depending on the type of view selected.

The **Cut Inheritance** option is available when a view created from a section view inherits the section cut. An example of this is an isometric view projected from a section view. By default, the isometric view shows the section cut with the sectioned portion hatched. To remove the section cut from the isometric view, expand the **Cut Inheritance** drop-down list in the **Section View Editor** contextual ribbon tab and uncheck the **Section cut** option. This resets the view to an isometric view without sectioning.

Detail Views

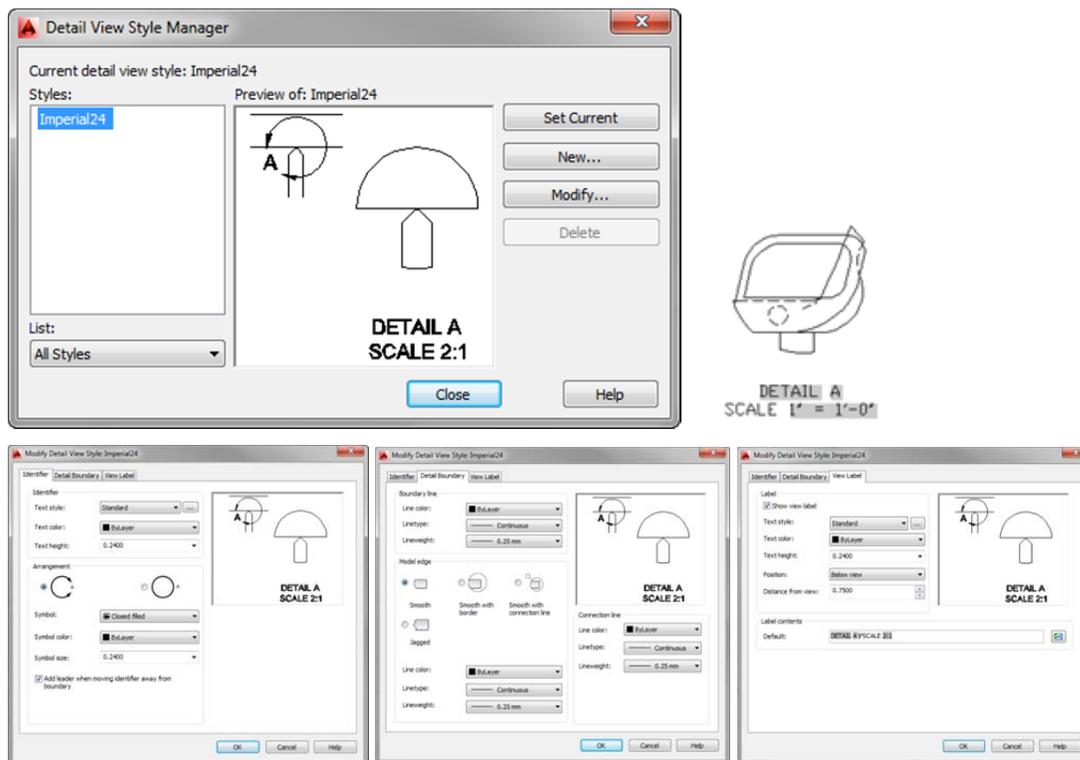
A **detail view** shows a selected portion of a view to clarify model details. A detail view is projected from a parent view and is typically shown at a larger scale. As with other types of projected views, the detail view is linked to the parent view. Detail views are created using the

VIEWDETAIL command. A detail view is created by drawing a circular or rectangular boundary to define the extents of the view. You can create detail views from an AutoCAD 3D model or an Autodesk Inventor file.

Detail views are *associative*. As with other types of drawing views, detail views are updated automatically when model changes are made if the **VIEWUPDATEAUTO** system variable is set to 1.

VIEWDETAILSTYLE

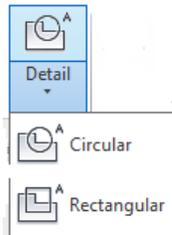
The **VIEWDETAILSTYLE** command is used to create and modify detail view styles. This command accesses the **Detail View Style Manager** dialog box. Picking the **New...** button allows you to create a new detail view style. Picking the **Modify** button opens the **Modify Detail View Style** dialog box for the selected style. The tabs in the **New Detail View Style** dialog box or the **Modify Detail View Style** dialog box are used to make settings for the detail identifier, detail boundary, and detail view label. As with other types of styles, develop standards for detail views in accordance with company or industry standards.



VIEWDETAIL

Creating a detail view is similar to creating a section view. To create a detail view, select the **VIEWDETAIL** command and then select the parent view. The default method for creating

the view is to create a circular detail boundary. This is the preferred display for most detail views. You can change the boundary type to rectangular using the **Boundary** option. With the **Rectangular** option, a rectangular detail boundary is drawn and the detail view has a rectangular outline.



Selecting one of the options from the **Detail** drop-down list in the **Create View** panel of the **Layout** tab on the ribbon begins the command and sets the appropriate boundary type. If you are creating a circular detail boundary, select the parent view and then pick a point to specify the center of the view. At the next prompt, drag the cursor or enter a value to set the size of the boundary. Then, pick a point to locate the view. A rectangular detail boundary is created in the same manner. After locating the view, you can select an option or press [Enter] to exit the command. You can adjust options using the **Detail View Creation** contextual ribbon tab. You can also use dynamic input or the command line.

The **Hidden Lines**, **Scale**, **Visibility**, and **Move** options are the same as those previously discussed for section views. The **Model Edge** option is used to adjust the edges of the detail view and set border display and leader options. The **Smooth** option creates a smooth edge for the view. This is the default option. The **Smooth with Border** option creates a smooth edge for the view and draws a circular or rectangular border, depending on the type of boundary specified. The **Smooth with Connection Line** option creates a smooth edge, draws a circular or rectangular border, and attaches a leader from the detail symbol in the parent view to the detail view. The **Jagged** option creates the view with a jagged edge. With this option, no border is displayed and the view does not have a leader attached. The **Annotation** option allows you to adjust the view identifier and specify whether a view label is shown.

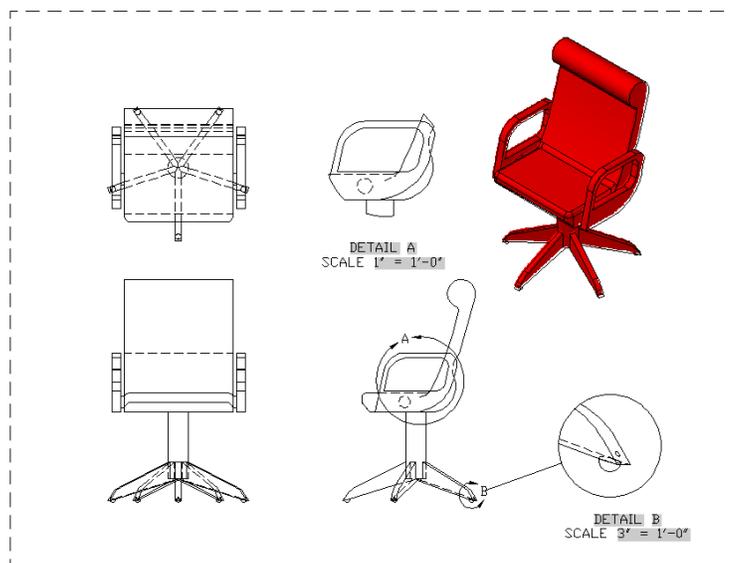
Once created, detail views can be edited by editing the detail boundary or detail identifier. To edit the detail boundary, select the boundary and hover over one of the four boundary grips to display a shortcut menu. The options allow you to stretch the boundary and change the boundary type to circular or rectangular. Hovering over the detail identifier grip allows you to move the identifier or reset the identifier to the initial position.

You can also edit a detail view by using the **VIEWEDIT** command, or by selecting the view to display a base grip and a parameter grip for changing the scale. The detail view label is an mtext object. It can be moved by selecting the label and then selecting the base grip to access the standard grip editing options.

1. Open the drawing **3D-Chair.dwg**.
2. Select the ANSI A layout.
3. Create a base view as the front view. (Note: you will need to set the orientation to right side to place as the front view).
4. Create a four view drawing with the isometric view show as shaded with visible lines. Figure Detail1.jpg



5. Create two circular detail views of the side of the chair and the roller on the leg.
6. Double click on the detail views to change the scales as shown.

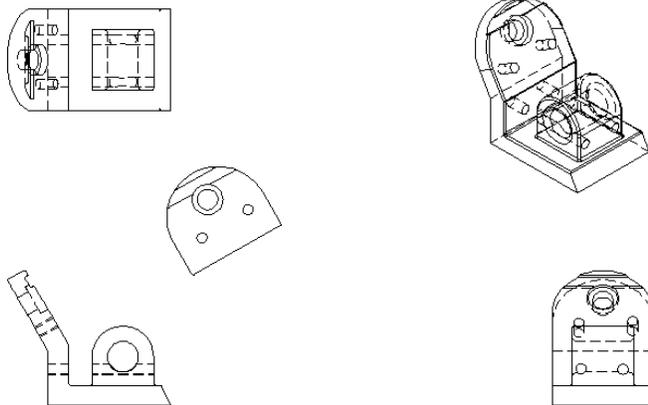


7. Save the drawing as **3D_Chair.dwg**.

Auxiliary Views

An undocumented feature in AutoCAD is the ability to create an auxiliary view by using a section view. Often, a multiview drawing contains inclined surfaces that do not describe the true size or shape of features in a regular orthographic view. To establish an auxiliary view, you can draw a full section line using the **Full** option of the **VIEWSECTION** command. When specifying the section line, pick two points on the inclined surface. If needed, draw a parallel construction line across the inclined surface and use it to create the section line. The auxiliary view plane is oriented parallel to the inclined edge of the surface and the view is created perpendicular to the surface. To remove the display of the section line and view label from the drawing, freeze the **MD_Annotation** layer.

1. Open the drawing **Auxiliary_View.dwg**
2. Select the ANSI A Layout.
3. The four views have been created for you. Use grips to space the views apart.
4. Create an auxiliary view by one of the following two methods:
5. Use a full section by placing a line parallel and touching to the inclined edge of the front view. (Note: you may need to draw a construction line across the inclined edge to assist in placing the full section line) Note: There are rounds at the edges of the face of the part Notice that no hatch pattern is created.
6. You can also draw a polyline across the inclined edge of the front view and use the Object option to create the auxiliary view). Note: There are rounds at the edges of the face of the part). Notice that no hatch pattern is created.
7. Where did the section line go? Freeze the MD_Annotation layer.
8. Erase any construction lines used in placing the section line.



9. Save the drawing as **Aux1.dwg**.

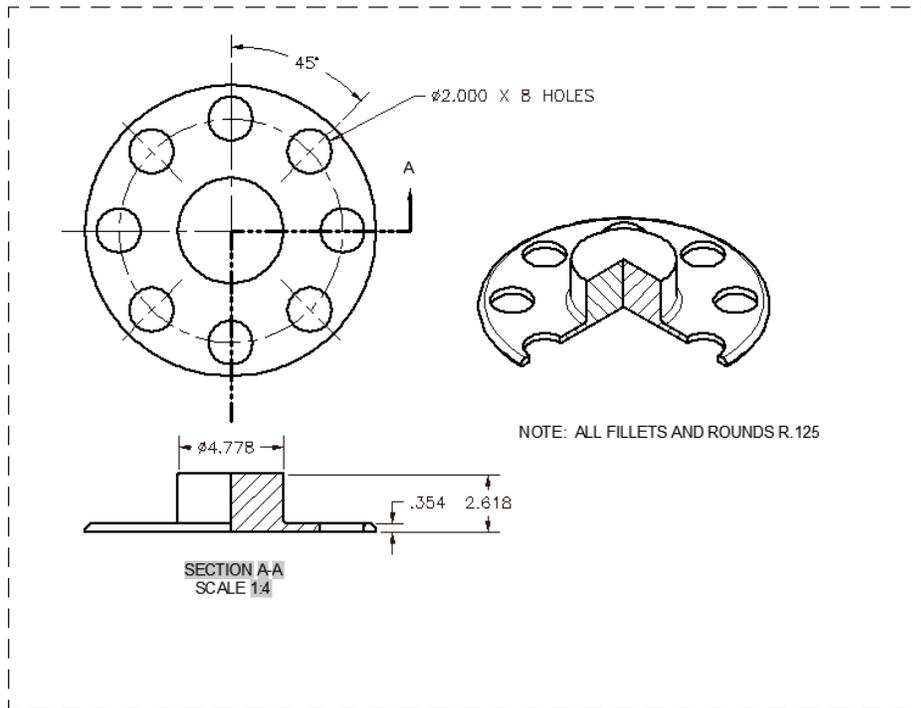
Dimensioning Drawing Views (AutoCAD 2013 and 2014)

After placing drawing views on a paper space layout, you can dimension each view as needed in the layout. When dimensioning drawing views, make sure that the **DIMASSOC** system variable is set to 2 so that associative dimensions are created. Associative dimensions are associated to the dimensions of the model and update when the physical model changes. However, depending on edits to the model geometry, some dimensions can become disassociated.

Associative dimensions added to a drawing view are attached to the underlying object geometry. As the size or location of an object changes, the associative dimensions automatically adjust to their new size or location. Associative dimensions can become disassociated when a 3D model is modified or updated and the dimensions describing the underlying geometry lose their attached location. The **Annotation Monitor** is used to monitor and notify you of any changes in the associativity of dimensions placed in drawing views. This feature can be turned on by picking the **Annotation Monitor** button on the status bar. The **Annotation Monitor** is turned off by default, but it is automatically activated when a model is edited and dimensions are updated. The status of the **Annotation Monitor** is controlled by the **ANNOMONITOR** system variable.

The **Annotation Monitor** icon in the notification tray of the status bar provides feedback regarding the state of associative dimensions. If any associative dimensions become disassociated, the icon turns red and a balloon notification appears. In addition, yellow alert icons appear in the layout next to the disassociated dimensions. You can click the balloon notification link to delete the dimensions all at once, or you can pick on individual alert icons in the layout to update the dimensions.

1. Open the drawing **Half_Section-Dimension.dwg**
2. Set the system variable DIMASSO to 2.
3. Dimension the drawing as shown.



4. Save the drawing as **Half_Section Dimensioned.dwg**
5. Use the **EXPORTLAYOUT** command and export the drawing view as a new drawing. Save the new drawing using the default name.
 - a. Open the new exported layout drawing. You now have the same view as above as a new model space drawing view.