Drawing Views with AutoCAD® 2013: 2D Drawings from AutoCAD® 3D Models - AC2290-L and AC7046-L (repeat)

Drawing Views with AutoCAD® 2013: 2D Drawings from AutoCAD and Autodesk® Inventor® 3D Models - AC6423-V

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This intermediate-to-advanced hands-on lab offers AutoCAD® software 3D veterans and 2D users a chance to explore 2D model documentation of 3D models from AutoCAD 2013 and Autodesk® Inventor® 2013 software. Create base views and projected views from 3D solid or surface models. Use the new Viewbase command to generate a base 2D view and the Viewproj command to create projected orthographic views. We will edit the 3D models and update the derived drawing views. We will add dimensions to the drawing views and modify the 3D model to update the dimensions. Are commands such as Flatshot, Solview, Soldraw, and Solproj superseded? We will find out! We will explore the Exportlayout command for exporting a layout containing drawing views to model space. If you used AutoCAD 3D in the past or are new to 3D but have a good 2D background, attend this lab and get ready to be surprised!

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

- Use Inventor 3D models for drawing view creation in AutoCAD 2013
- Create drawing views from 3D AutoCAD and Inventor models
- Edit 3D models and associative drawing views
- Edit drawing views
- Edit AutoCAD 3D models in Autodesk Inventor Fusion

About the Speaker
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Model Documentation

There are multiple techniques available to create drawing views from 3D models in AutoCAD. Using these techniques, orthographic views and other standard drawing views can be created in order to produce multiview drawings. The techniques available are discussed in this paper.

Drawing Views

You can create 2D drawing views from 3D solid or surface models using the VIEWBASE command. This command allows you to quickly create a multiview drawing layout made up of orthographic views. Drawing views created with the VIEWBASE command are created in paper space (layout space). After creating a layout of views, you can dimension them using AutoCAD dimensioning commands.

Drawing views created with the VIEWBASE command are associative. This means that they are linked to the model from which they are created and can be updated to reflect changes to the model geometry. This capability allows you to keep drawing views up-to-date when design changes are required.

When using the VIEWBASE command, you create a base view of the model and then have the option to create additional views that are projected from the base view without exiting the command. You can create both orthographic and isometric views. A base view created with the VIEWBASE command is defined as a parent view. Views projected from the base view inherit the properties of the base view, such as the drawing scale and display properties, and are placed in orthographic alignment with the base view. If the base view is moved, any projected views are moved with it to maintain the parent-child relationship.

Drawing View and Layout Setup

In order to create views with the VIEWBASE command, you must first activate paper space (layout space). Activate paper space by picking a layout tab. Then, before creating views, set up the page layout as required. If a default viewport appears in the layout, delete the viewport. Then, right-click on the active layout tab and select Page Setup Manager to access the layout settings. Select the desired paper size, drawing orientation, and other settings. If you have a
template drawing already created, you can create a new layout based on the template by right-clicking on a layout tab and selecting **From template**....

By default, drawing views created with the **VIEWBASE** command are generated using third-angle projection. This is the ANSI standard. The default projection used for drawing views can be set using the **VIEWSTD** command. This command and other drawing view commands are accessed from the **Drawing Views** panel on the **Annotate** tab of the ribbon.

Pick the dialog box launcher button at the lower-right corner of the **Drawing Views** panel to initiate the **VIEWSTD** command. This opens the **Drafting Standard** dialog box.

The options in the **Projection type** area determine the type of projection used for new drawing views. By default, the **Third angle** button is selected. If the **First angle** button is selected, views are created using first-angle projection. Select this option to create views in accordance with ISO standards. The options in the **Thread style** area determine the type of thread representation used for views showing threaded features. The thread representation options are used when creating views from Autodesk Inventor models or imported models containing threaded features. A full circle thread representation conforms to the ANSI standard. This is the default option. A partial circle thread representation conforms to the ISO standard.

The options in the **Shading/Preview** area of the **Drafting Standard** dialog box determine
the type of preview that appears when placing a drawing view. When Shaded is selected in the Preview type drop-down list, a shaded view appears. This is the default option. This option provides a preview image of the view orientation before the view is placed. When the Bounding box option is selected, a preview box representing the extents of the view appears. However, no preview image is displayed. The Shaded view quality option is set to 100 dpi by default. A higher setting will provide better resolution quality when a shaded preview is displayed. However, for all practical purposes, a setting of 150 dpi or higher does not produce a dramatic difference in resolution quality.

Creating Drawing Views

After setting up a layout and making the appropriate drawing view settings, you are ready to place the base view. As previously discussed, drawing views can be created from a 3D solid or surface model. Drawing views created in the layout are generated from the model you have created in model space. If no model exists in the drawing and you initiate the VIEWBASE command. You can select a model created in AutoCAD or a model created in Autodesk Inventor. You can select Base from the Layout tab, Create View panel.
When you select the \texttt{VIEWBASE} command, a preview representing a scaled orthographic view of the model appears. This is the \textbf{base view} of the model. You can then specify the location of the base view or select an option. To select an option, use dynamic input or make a selection from the \textbf{Drawing View Creation} contextual ribbon tab. The \textbf{Orientation} option can be used to select a different view from the default base view. The \textbf{Orientation} options correspond to AutoCAD's six orthographic and four isometric preset views. \textbf{These views are based on the WCS.}

By default, the front view is used as the base view by AutoCAD. Depending on the construction of your model, you may want to use a different view, such as the top view, to serve as the base view. This view will establish the front view on the orthographic drawing. In most cases, the front orthographic view on the drawing describes the most critical contour of the model. You will typically use the front or top view of the 3D model for the front orthographic view on the drawing.

Creating a base view creates an AutoCAD \textbf{drawing view} object. A drawing view has a view border, a base grip, and properties that can be edited, such as the view scale. However, the content of the drawing view \textit{cannot} be edited. When a base view is created, new layers are created by AutoCAD for the drawing view geometry. Object lines (visible lines) in the view are placed on a newly created layer named Visible. Hidden lines in the view are placed on a newly created layer named Hidden. The drawing view object is placed on a newly created layer named Base. Additional layers may be created by AutoCAD, depending on the type of model, display style used, and edges displayed in the view. However, the layers are only created to organize the drawing view geometry. The layer properties can be modified to change the appearance of the drawing view geometry, but the geometry cannot be otherwise modified.

The \textbf{Type} option of the \texttt{VIEWBASE} command is available before picking the initial location of the base view. This option is used to specify whether projected views are placed after placing the base view. By default, this option is set to \textbf{Base and Projected}. With this option, you can place additional, projected views while the \texttt{VIEWBASE} command is still active. Projected views that you place are \textit{projected} from the base view and oriented in the proper orthographic alignment. As previously discussed, projected views have a parent-child relationship with the base view. The base view is the \textit{parent} view and any projected views are \textit{child} views. If the base view is moved, for example, the projected view is moved with it. Projected views are not created with the \texttt{VIEWBASE} command when the \textbf{Type} option is set to \textbf{Base only}. If you place a
base view in this manner and later decide to create projected views from the base view, you can use the \texttt{VIEWPROJ} command. The \texttt{VIEWPROJ} command allows you to place projected views from any selected view.

After you pick the location for the base view, you can press [Enter] to place projected views (if the \texttt{Type} option is set to \texttt{Base and Projected}). Drag the cursor away from the base view and then pick to locate the projected view. The direction in which you move the cursor determines the orientation of the projected view. You can continue placing projected views, or you can press [Enter] to end the command. To end the command without placing projected views, press [Enter] twice.

When you place the base view, you can select an option to change the default style, scale, or visibility of the base view before placing projected views. Projected views inherit the properties of the base view. The \texttt{Style} option is used to set the display style of the base view. The four options available are \texttt{Wireframe}, \texttt{Wireframe with Hidden Edges}, \texttt{Shaded}, and \texttt{Shaded with Hidden Edges}. These options can be selected using dynamic input or the \texttt{Drawing View Creation} contextual ribbon tab.

The \texttt{Scale} option is used to set the scale of the base view. You can select a scale from the drop-down list in the \texttt{Appearance} panel in the \texttt{Drawing View Creation} contextual ribbon tab. You can also specify the scale by typing a value.

The \texttt{Move} option allows you to move the base view after picking the initial location on screen. When you select the \texttt{Move} option, the view is reattached to the cursor so that you can move it to another location. After you pick a new location, the \texttt{VIEWBASE} options are again made available.

The \texttt{Visibility} option allows you to control the display of drawing geometry in the view. The \texttt{Interference edges} option controls whether both object and hidden lines are displayed for interference edges. By default, this option is set to \texttt{No}. The \texttt{Tangent edges} option controls whether tangential edges are displayed to show the intersection of surfaces. By default, this option is set to \texttt{No}. If this option is set to \texttt{Yes}, you can specify whether tangent edges are shortened to distinguish them from object lines that overlap. The \texttt{Bend extents} option is only available when working with a model that includes a view with sheet metal bends. The \texttt{Thread features} option controls thread displays on models with screw thread features. The \texttt{Presentation trails} option is used to control the display of trails in views created from presentation files.
Additional drawing view options can be accessed by selecting View Options in the Appearance panel of the Drawing View Creation contextual ribbon tab. This option is not available on the command line. Selecting View Options opens the View Options dialog box. The options in the View Justification drop-down list determine the justification of the view. The justification refers to how the view is “anchored.” When changes are made to the model, such as a change in size, the view updates based on the justification setting. If the justification is set to Fixed, geometry in the view unaffected by the edit does not change from the original location. If the justification is set to Center, the geometry updates about the center point of the view.

A multiview drawing layout is created by placing three projected views after placing the base view. The projected views include two orthographic views and an isometric view. Notice that wireframe styles are used for the orthographic views and a shaded style is used for the isometric view. Drawing views can be edited after placing them to change properties as needed. In addition, as previously discussed, drawing views can be updated when changes are made to the model.

NOTE:

The Representation option of the VIEWBASE command is used to create a view based on a model representation and is only available when working with models created in Autodesk Inventor.

Updating Drawing Views

Drawing views maintain an associative relationship with the model from which they are created. However, it is important to note that this associative relationship is controlled by the model, not the drawing view. When making design changes, you make changes to the model geometry, not the drawing view.

During the design process, it is often necessary to make modifications. If you have created drawing views from a model, and then make modifications to the model, the derived base view and any views projected from the base view will require updating. When this occurs, drawing views are considered out-of-date. AutoCAD does not automatically update the views when the model is modified. However, you can quickly update drawing views when needed.
VIEWUPDATE

When you make changes to a model in model space and then switch to a layout with drawing views, red markers appear to indicate that the views are out-of-date. In addition, a balloon notification appears on the status bar. You can pick the link in the balloon notification to update the views, or you can use one of the view update commands in the Drawing Views panel on the Annotate tab of the ribbon. In the example, all of the views are updated at once based on a change to the original part.

Example    Change the cylinders diameters to a larger diameter and then use VIEWUPDATE to update the drawing views. What happens?

This is the preferred method for updating views. However, you can also update views individually by selecting Update View from the drop-down menu in the Drawing Views panel on the Annotate tab of the ribbon. This requires you to select each view to update individually. Selecting Update View is the same as using the VIEWUPDATE command. When using this command, you are prompted to select each view to update. You and also set the views to Auto Update. Select Auto Update from the Layout tab, Update Panel.
NOTE:

Drawing views are different from viewports. You cannot pick inside of the view and edit or modify the drawing geometry by any standard AutoCAD editing technique. Drawing views can only be updated by changing the 3D model geometry.

PROFESSIONAL TIP:

In order to modify solid primitive subobjects in a composite solid model, the solid history must be recorded.

1. Open the drawing Flange_Base.dwg
2. Select the ANSI B layout.
3. Create a 4 view drawing as shown below.
4. Turn Auto Update off.

5. Set the orientation to Top and place the top view.
6. Project the front, right side and isometric views.
7. Double click on the isometric view and change the visibility to Shaded with visible lines.
8. In the 3D model, use sub-object editing to change the diameter of the center bore to 2.00 (radius
1.00). Make changes in the properties palette.

9. Change the diameter of the filleted boss hole to **.600 (radius 3.00)**.

10. Update your drawing.

11. Save the drawing as **Flange_Base_Done.dwg**

Drawing views can be edited after being created. Like other AutoCAD objects, drawing views can be moved or rotated. In addition, certain properties of drawing views, such as the display style and scale, can be modified.

**VIEWEDIT**

The **VIEWEDIT** command can be used to edit the properties of a drawing view. You can quickly initiate this command by double-clicking on a view. The **Drawing View Editor** contextual ribbon tab appears. The editing options are similar to the options available when you create a base view.

**Creating an Assembly drawing and adding parts to the assembly**

1. Open the drawing **Fixture Assembly.dwg**

2. Create the base view (front) and the top, right side and isometric views.

3. Scale the isometric view to a larger scale.

4. Use the **INSERT** command to insert the plug at X=20, Y=8 and Z=7, centered on the top hole as shown. Update the drawing.
5. Move other parts as needed. Remember to update the drawing!

Creating Drawing Views from Inventor part files (IPT’s)

Drawing views can be created from sources other than AutoCAD 3D models. You can create views from Autodesk Inventor® part (IPT), assembly (IAM), or presentation (IPN) files. You can also create drawing views from models imported with the IMPORT command. Drawing views can also be created from an assembly or subassembly to create an assembly drawing.

CLOSE all AutoCAD drawings.

OPEN: Blank Drawing.dwg.

OPEN: Coupler3.ipt in Autodesk Inventor.

1. In AutoCAD, turn off Auto Update.
2. Create the 4 view drawing of the Coupler3.ipt file starting with a Base view From Inventor inside AutoCAD.
3. Change the LTSCALE as needed.
4. Save the drawing as Coupler3.dwg.
5. Close the drawing.
6. In Autodesk Inventor, **edit Holes 6 and 7** (the two larger holes and the one smaller hole) in Autodesk Inventor to a **.03** diameter.

7. Save the **Coupler3.ipt** file.

8. Open the AutoCAD drawing **Coupler3.dwg**. Update the views.

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**Dimensioning Drawing Views**

1. Open the drawing **Flange_Base_Done.dwg or Flange_Base_Finished.dwg**.

2. Type **DIMASSOC** and set the variable to **2**.

3. Add the three dimensions as shown on the ANSI B layout.

4. Edit the **.4000** diameter hole to **3.000**.

5. Edit the **.600** diameter hole to **.500**.

6. Edit the **1.00** height to **.750**.

7. Update the drawing. What happens? Use **DIMREASSOSSOCIATE**.
8. Do not save the drawing.

After placing drawing views on a paper space layout, you can dimension each view as needed. 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.

If you dimension drawing views with associative dimensions, make changes to the model, and then update the views, the dimensions may become disassociated. In this case, AutoCAD will ask if you want to reassociate the dimensions. See Figure 15-14. If you select Try to reassociate annotations, AutoCAD will attempt to reassociate the dimensions automatically. If AutoCAD is unable to reassociate all dimensions, you will be prompted to use the `DIMREASSOCIATE` command. Selecting Run DIMREASSOCIATE will allow you to pick points in the view to reassociate the dimensions. For more information on the DIMREASSOCIATE command and associative dimensioning, refer to AutoCAD and Its Applications—Basics.

Using the EXPORTLAYOUT Command

You can use the EXPORTLAYOUT command to export a layout containing drawing views into a new drawing (DWG) file. However, this technique will break the associativity between the 3D model and the drawing views. The advantage to using this technique is that the drawing geometry can be edited in the same way as any other AutoCAD 2D geometry in model space. The disadvantage is that the 2D geometry has lost any associativity back to the 3D model.
When using the `EXPORTLAYOUT` command, you save the exported layout as a DWG file. The drawing views become blocks in the new drawing file.

1. Open the drawing `flange.dwg`.
2. Type `EXPORTLAYOUT`.
3. Save the default name to create the new DWG. Open the new DWG when prompted.
4. What are the 4 new AutoCAD objects? Make modifications to the drawing as needed. Dimension in Model Space as needed.
5. Do not save the drawing.
New for AutoCAD 2013

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.
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. If you change the scale of the parent view, the detail view scales accordingly.

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)**

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

Autodesk Fusion.......  
Try editing your 3D models in Autodesk Fusion and see what happens to your drawing views! You will be pleasantly suprised.