

CP124029-L

Create Compelling 2D Sections, Details, and Auxiliary Views from AutoCAD® 3D Models

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Learning Objectives

Learn how to:

- Create drawing views of AutoCAD 3D models for drawing 2D sections and details
- Edit 3D models and their 2D associated drawing views
- Create undocumented auxiliary views from AutoCAD 3D models
- Export drawing views to create 2D, sectional, multiview drawings in model space

Description

This intermediate-to-advanced hands-on lab offers AutoCAD 3D software veterans a chance to explore the 2D model documentation of 3D models from AutoCAD 2018 software. After creating base views and projected views from 3D solid or surface models, we'll explore sectional views. Using 3D models, we'll create full, half, offset, and aligned sectional drawing views. We'll also create circular and rectangular detail views. We will apply editing techniques of the 3D models to then update the derived drawing views. We will also explore Undocumented Auxiliary View features from 3D models. Exporting the 2D sectional drawing views into model space using the EXPORTLAYOUT command will give us true size and shape geometry to edit and annotate as needed. If you've used AutoCAD 3D software in the past to create 2D multiview drawings from 3D models, attend this class and get ready to be surprised.

Your AU Expert

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Create drawing views of AutoCAD 3D models - Drawing 2D sections

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Drawing Views

You can create 2D drawing views from AutoCAD 3D solid or surface models using the **VIEWBASE** command. This command allows you to quickly create a multiview drawing layout. 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 update 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.

Working with Drawing Views

Creating a base view creates an AutoCAD **drawing view** object. A drawing view has a view border, a base grip, and a parameter grip for changing the scale. The scale and certain other properties of the view can be edited. However, the content of the drawing view *cannot* be edited.

When a base view is created, new layers are created by AutoCAD for the drawing view geometry. The layers are created based on support for the type of geometry represented. Object lines (visible lines) in the view are placed on a newly created layer named MD_Visible. Hidden lines in the view are placed on a newly created layer named MD_Hidden, and so on. The drawing view object is placed on the current layer or on the 0 layer. 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.

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 are updated automatically by default. This behavior is controlled by the **VIEWUPDATEAUTO** system variable. When this system variable is set to 1 (on), drawing views automatically update when you open a drawing file or activate a layout with drawing views. The **VIEWUPDATEAUTO** system variable setting is indicated by the **Auto Update** button located on the **Update** panel of the **Layout** tab on the ribbon.

Edit 3D models and their 2D associated drawing views

Editing Drawing Views

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, edge visibility, 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.

As previously discussed, projected views inherit the properties of the base view when created. If you select a projected view with the **VIEWEDIT** command, you can change the display style, edge visibility, and scale.

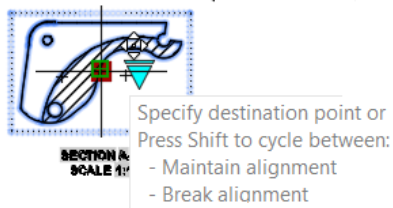
A quick way to change the scale of a view is to use the scale parameter grip. This grip

appears when you single-click on a view. Pick on the grip to access a different scale. To move a drawing view, single-click on the view and then pick on the base grip to move the view directly. When moving a parent view, any child views will move accordingly to maintain alignment. When moving a child view, you can move the view, but it cannot be moved out of alignment with the parent view. This applies to orthogonal views. If you move an isometric view, it is not aligned to other views and can be moved freely around the layout.

Picking on a drawing view grip and right-clicking displays a shortcut menu. The **Stretch** option can be used to move the drawing view. The **Rotate** option can be used to rotate the drawing view. You can rotate the view dynamically using the cursor or you can specify a rotation angle. If a drawing view is rotated, any parent-child relationships that exist between the view and other drawing views are broken.

You can break the alignment between parent and child views when moving a child view. To do this, select the drawing view grip and press [Shift] (tap the Shift key) once. The view is free to move to a different location. To reestablish the alignment, press [Shift] again. The alignment is restored and the child view cannot move out of alignment with the parent view.

If a view is already placed, you can break the alignment by selecting the view pressing (tapping) the SHIFT Key.



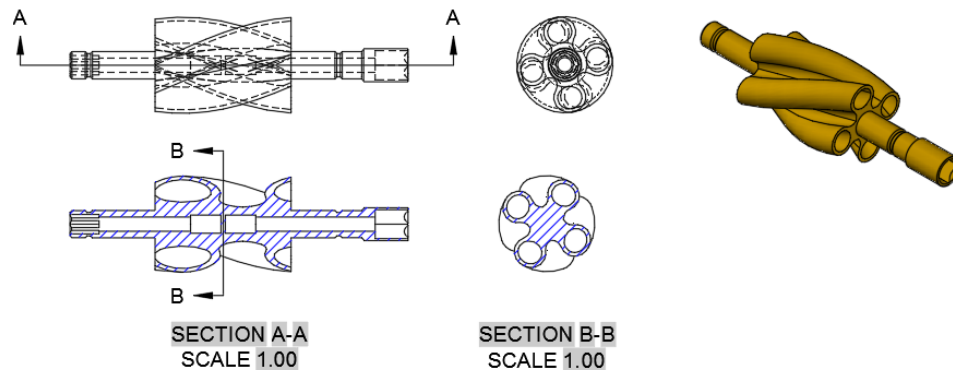
Section Views

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 model.

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.

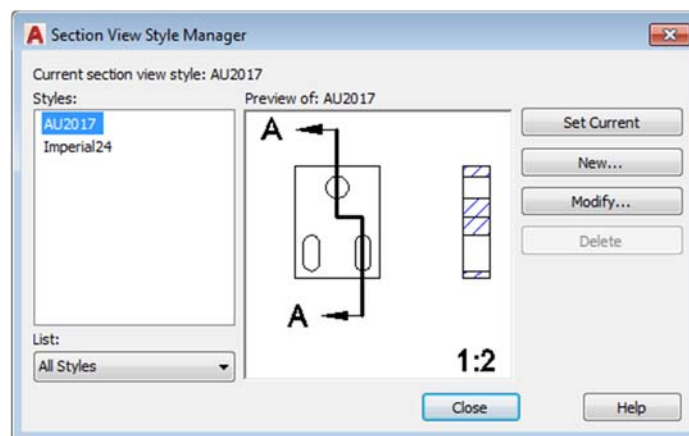
By default, a section identifier is placed with the section line and a section view label is placed with the section view when you create the view.



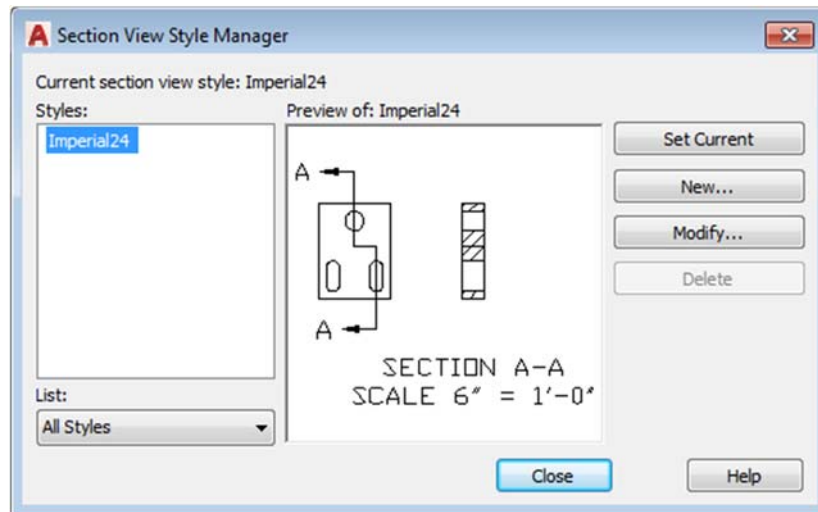
The section identifier is automatically incremented when you place additional section views. The text objects used for the section view label contain fields that update according to changes made to the section view. The appearance of elements in the section identifier and section view label is controlled 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 and modify 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.

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

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 button 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 (Tap the [Shift] key). 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.

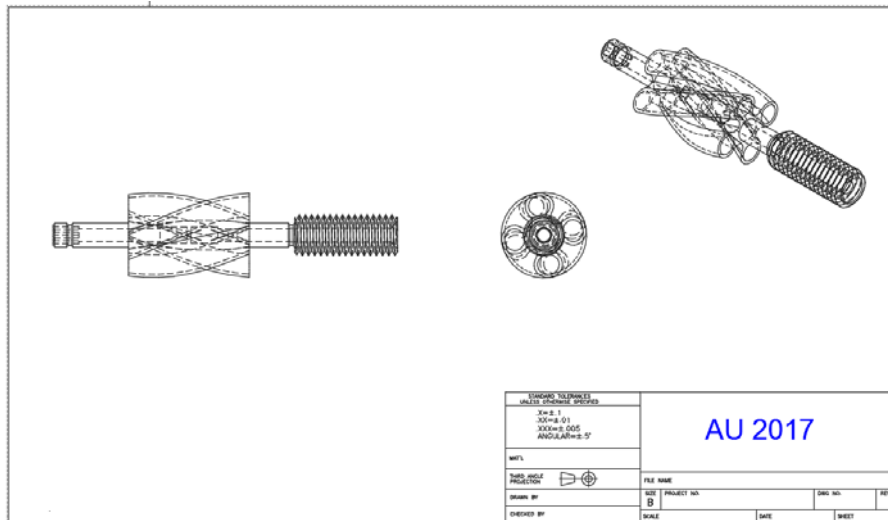
NOTE

After placing the section view, you can use grips to edit the section line and the section identifier. Editing the position of the section line alters the section view. To access grip editing options for a section line, hover over a vertex grip. This displays a shortcut menu with options similar to those for editing a polyline. You can stretch the vertex, add a vertex, add a segment, and flip the viewing direction. Hovering over one of the section identifier grips allows you to move the identifier with or without the section line and reset the identifier to the initial position.

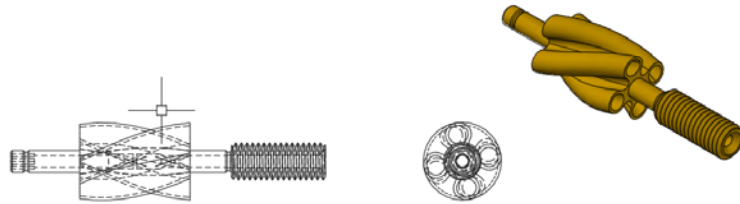
NOTE: In the status bar, turn on **Polar Tracking**, **Object Snap Tracking**, **Object Snaps**, (**Endpoint**, **Midpoint**, **Center**, **Intersection**, **Extension**) and **Lineweight display**.

Exercise 1A Full Section

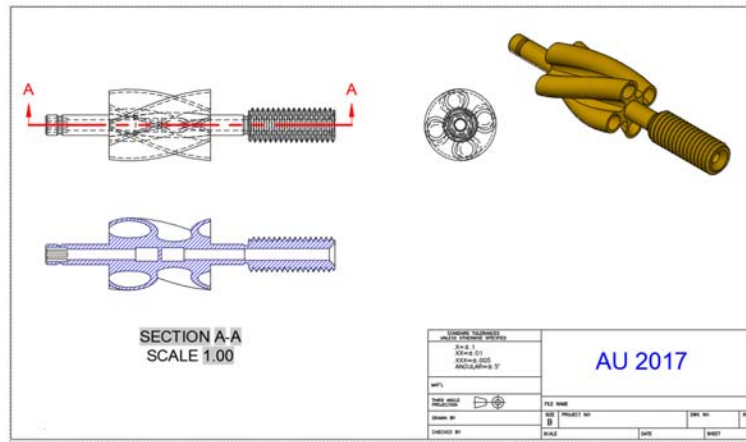
1. **OPEN** the drawing **AU_2017_Impeller.dwg**
2. Select the **ANSI-B Layout** to make the layout current.
3. On the ribbon, select the **Layout tab**.
4. From the **Create View** panel, create a **Base view From Model Space** and place the view as the Front view. Set the **Scale** to 1:1. Project to create a Right Side view. Project the isometric view as shown.



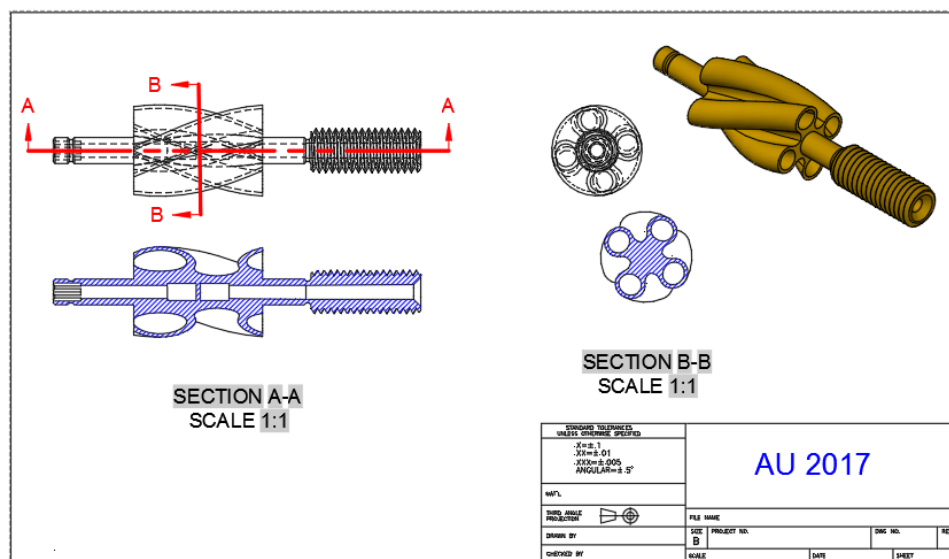
5. Edit the Isometric view by *double* clicking on the **Isometric view** and from the **Appearance panel**, change **From Hidden Lines** to **Shaded with Visible Lines**. Select **Exit** to complete the edit. The isometric view is shaded with visible lines shown.



6. From the **Create View** panel, select the section icon to create a **Full** section by passing the cutting plane line through the front view, horizontally creating section A-A. Place the Full section A-A under the front view as shown.

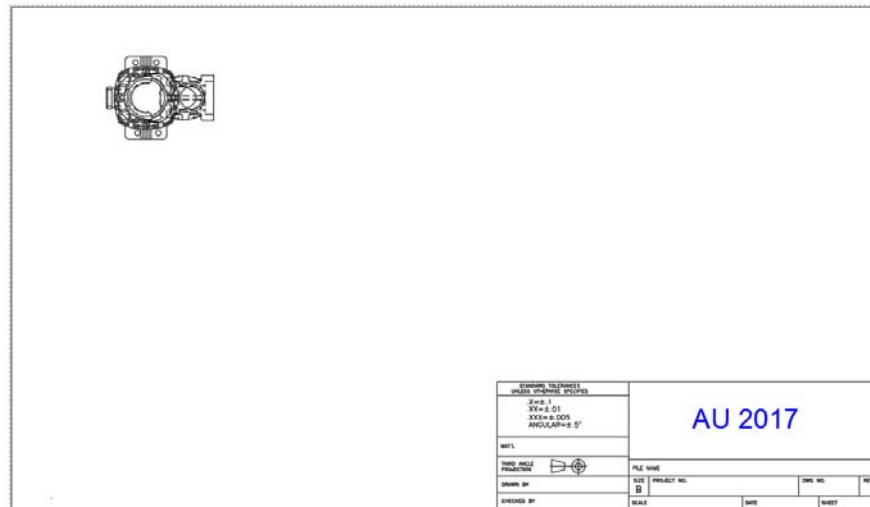


7. From the **Create View** panel, select the section icon to create a **Full** section by passing the cutting plane line through the front view vertically, creating section B-B. Place the Full section B-B under the right side view as shown. Hint: Press (Tap) the [SHIFT] key to break the orthographic alignment when placing section B-B.

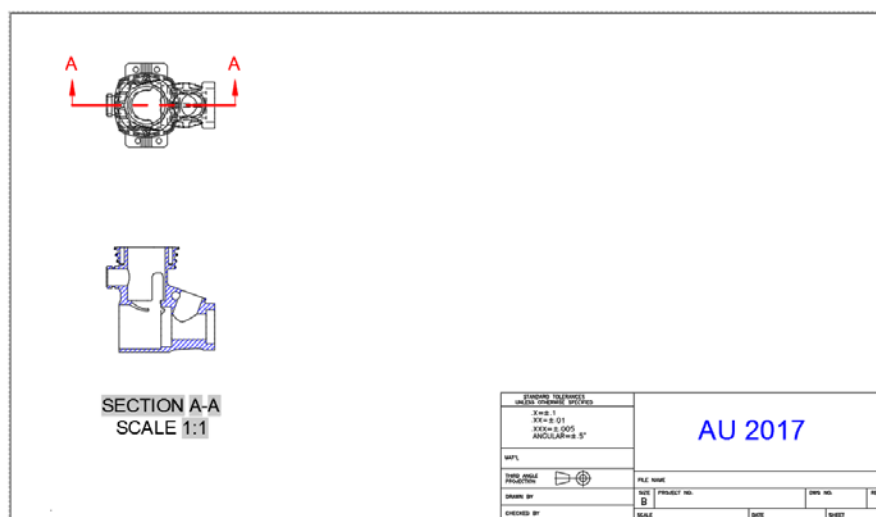


Exercise 1B Full Section

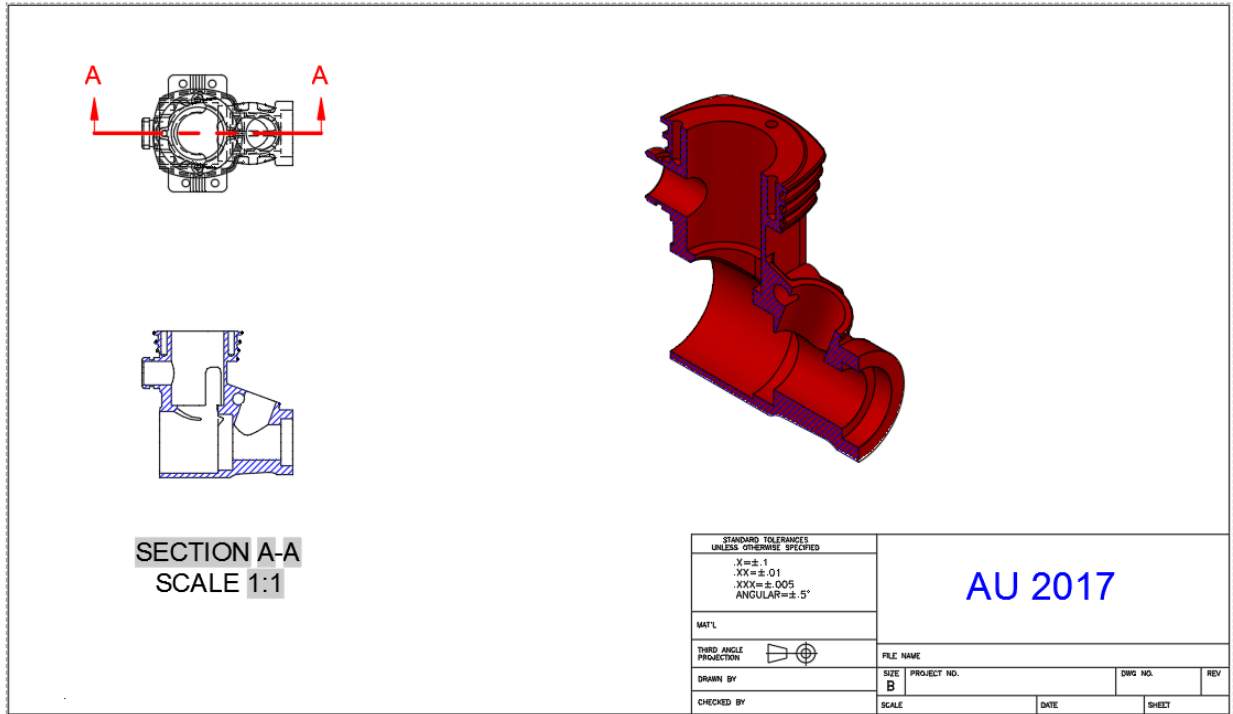
1. **OPEN** the drawing **Engine.dwg**. (Note: this 3D model was imported from the Autodesk Inventor sample files).
2. Select the **ANSI-B Layout** to make the layout current.
3. On the ribbon, select the **Layout tab**.
4. From the **Create View** panel, create a **Base view From Model Space** and change the **Orientation** to **Top**. Place the top view as shown and **Exit**.



5. From the **Create View** panel, select the section icon to create a **Full** section by passing the cutting plane line through the top view, horizontally creating section A-A. Place the Full section A-A under the top view as shown. **Exit** to place the sectional view.



6. Project the isometric view as shown.
7. Edit the Isometric view by *double* clicking on the **Isometric view** and from the **Appearance panel**, change **From Hidden Lines to Shaded with Visible Lines**. **Scale** the isometric view to **2:1**. Select **Exit** to complete the edit. The isometric view is shaded with visible lines shown.

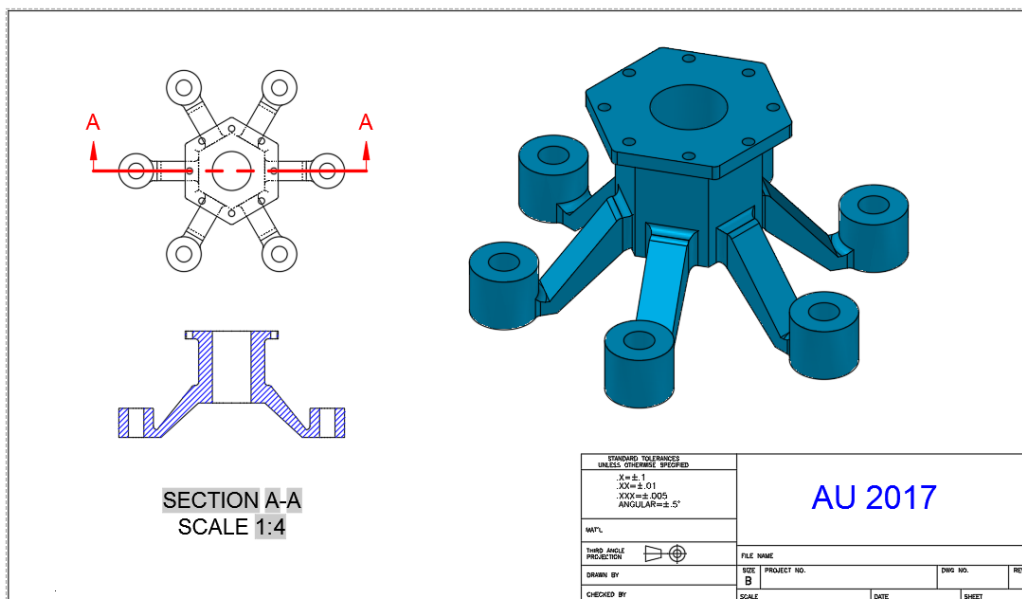


8. Save the drawing as **AU_2017_Engine.dwg**

Exercise 1C 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 **ANSI-B** layout.
3. From the **Layout** tab, **Create View** panel, **Project** 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.
4. Set the **Depth** option so the depth line distance turns off the visibility of objects behind the depth line. Drag the **Depth** Line just slightly behind the large center circle.
5. Select a location for the section view and the O.K.
6. **Project** a project isometric view from the Top view. Edit the view to set **Shaded with visible lines** and turn **off** the **Cut Inheritance**.
7. Scale the isometric view **1:2**.
8. Save the drawing as **SupportBrace_Full.dwg**



Half Section

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 and select the parent view. Next, select the **Type** option and then select the **Half** option. The option is accessed directly from the ribbon by selecting the button 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.

An example of using the **Depth** option to adjust a half section is shown below. In **Figure A**, the half section is created with the section line drawn through the middle of the object. In **Figure B**, 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.

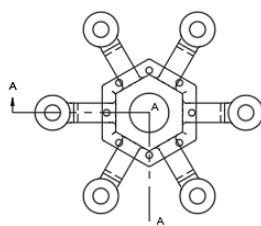


Fig. A
SECTION A-A
SCALE 1:4

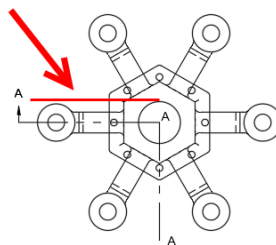
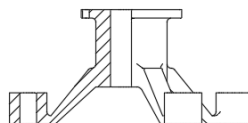
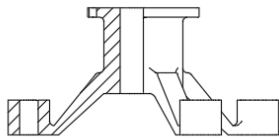
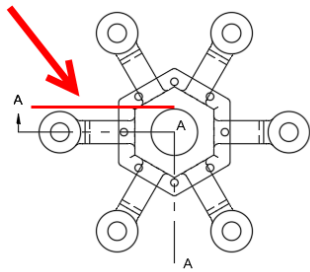


FIG. B
SECTION A-A
SCALE 1:4



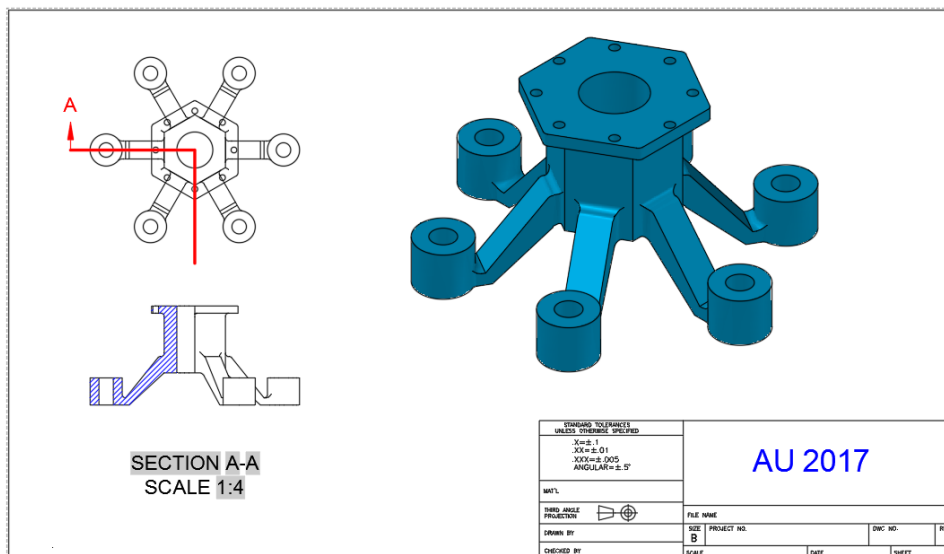
Exercise 2

1. **OPEN** the drawing **Half_Section.dwg**.
2. Select the **ANSI-B** layout.
3. (The Base View has already been created).
4. From the **Layout** tab, **Create View** panel, create a **Half section** as the front view. Use the **Depth** option as shown below. Change the **Scale** to **1:2**.



SECTION A-A
SCALE 1:4

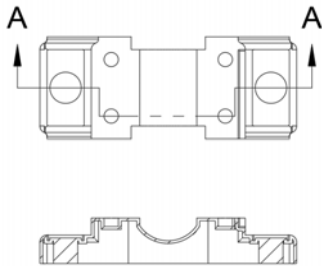
5. **Project** an isometric view from the front view.
6. Edit the isometric view to **Shaded with Visible lines**.
7. Set the scale to **1:2**. Turn off the **Cut Inheritance**. Turn off the **Edge Visibility**, **Tangent edges**.



8. Save the drawing as **Half_Section_Depth.dwg**.

Offset Section

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.

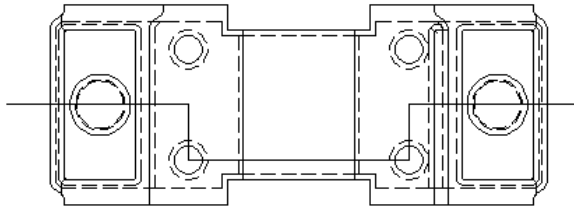


To create an offset section, select the **VIEWSECTION** command and select the parent view. Next, select the **Type** option and then select the **Offset** option. The option is accessed directly from the ribbon by selecting the button from the **Section** drop-down list in the **Create View** panel 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.

Creating a Section View from an Object

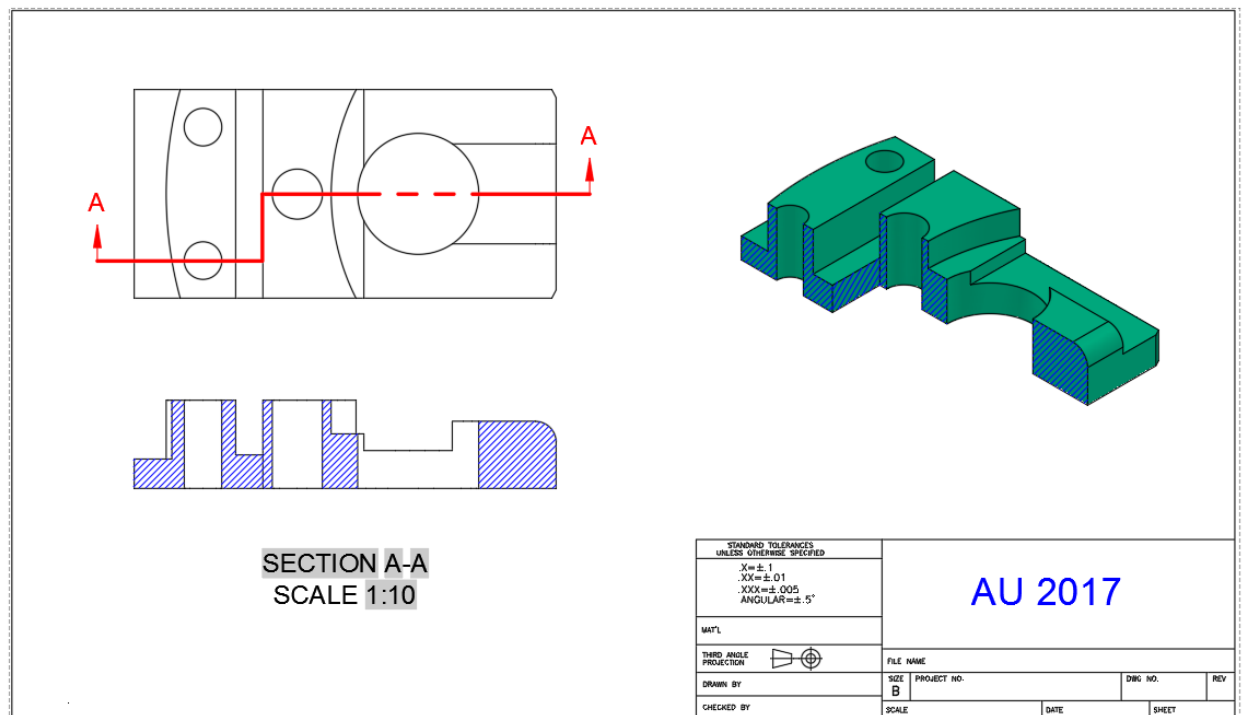
You can select an object in the paper space layout to use as the section line when creating a section view. This is a useful method when it is difficult to locate points using the **VIEWSECTION** command. To use an object as the section line, select the **VIEWSECTION** command, select the parent view, 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.



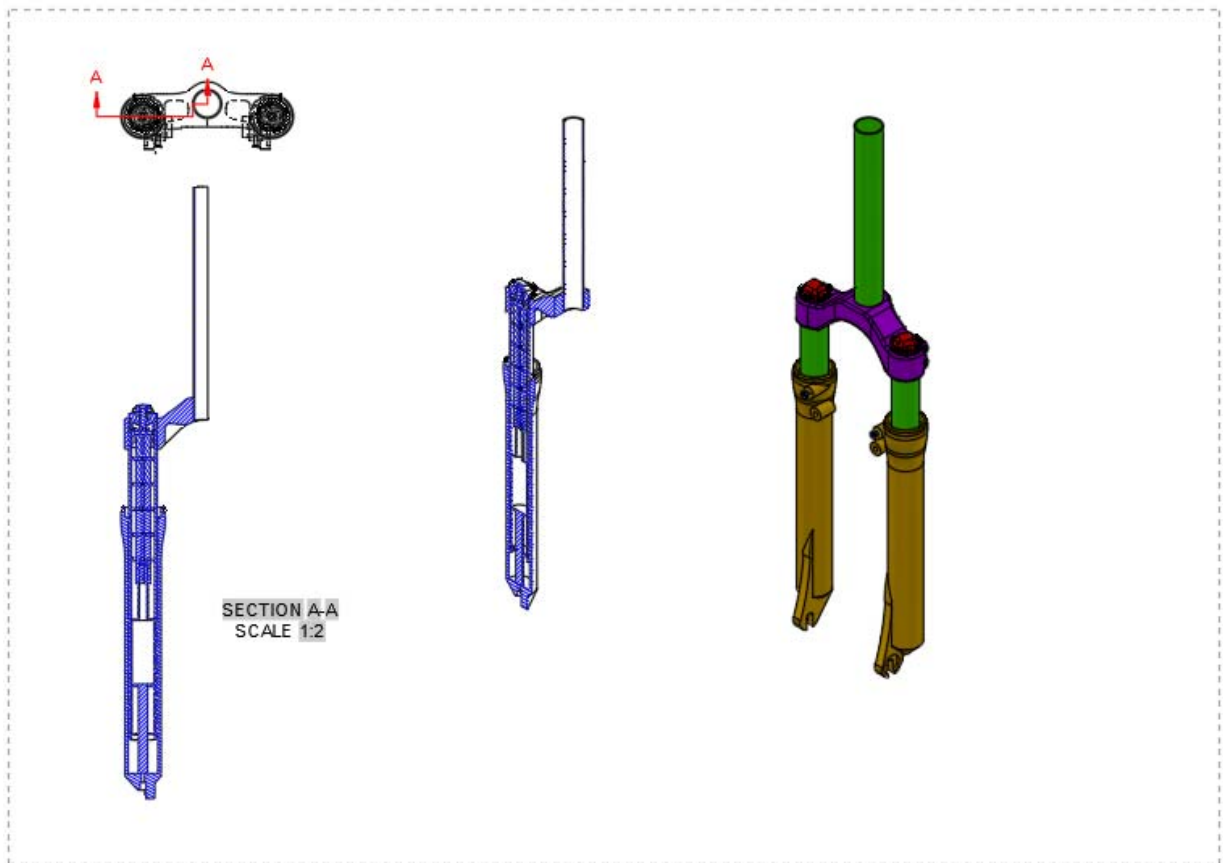
Exercise 3A Offset Section

1. **OPEN** the drawing **Offset_Base_Section.dwg**.
2. Select the **ANSI-B** Layout.
3. The top view has been created for you.
4. **Project** an **Offset section** by selecting the right side first to create of the part first and tracking the cutting plane line to the left to create the front view.
5. **Project the isometric view.**
6. Reset the parent (**top view**) **Scale** to **1:10**.
7. **Project** from the front view as an isometric section view.
8. Edit the isometric view to **Shaded with Visible lines**.
9. Save the drawing as **Offset_Section.dwg**.



Exercise 3B Offset Section

1. **OPEN** the drawing **Front-Fork_Offset.dwg** (Note: this 3D model was imported from the Autodesk Inventor sample files).
2. Select the **Front Fork** layout.
3. The top view has been created for you.
4. **Project** an offset section as the front view. Scale is set to **1:2**. *Start the cutting plane line at the center top view circle as shown.*
5. Project an isometric view from the front view showing the isometric section.
6. Project another isometric view applying **Cut Inheritance**.
7. Edit the one isometric view to **Shaded with Visible lines**.
8. Save the drawing as **Front-Fork_Offset_Final.dwg**.



Aligned Section

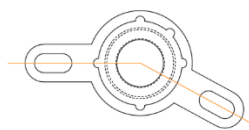
An aligned section is made by passing two nonparallel cutting planes through an object. The resulting section view shows features that are oriented at an angle rotated into the same cutting plane.

The purpose of an aligned section is to show the true size and shape of a feature. For this view, the **Projection** option is set to **Normal**. This is a conventional practice.

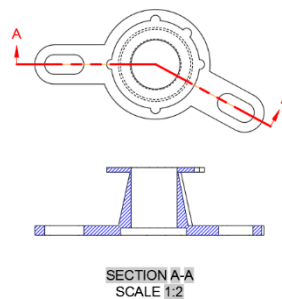
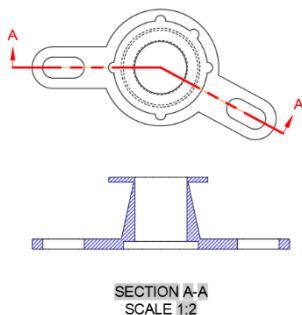
To create an aligned section, select the **VIEWSECTION** command, select the parent view, and access the **Aligned** option. The option is accessed directly from the ribbon by selecting the **Aligned** option from the **Section** drop-down list in the **Create View** panel of the **Layout** tab. Then, 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.

Exercise 4

1. Open the drawing **Aligned_Section.dwg**.
2. Select the **ANSI-B** Layout.
3. The top view has been created for you.
4. **Project an aligned section** by using one of two methods:
 - a. **Project an Aligned section** as the front view. Select the orange polyline as a guide for the aligned section cutting plane line.
 - b. **Project an aligned section** as the front view using the **From Object** method. Select the orange polyline as a guide for the aligned section cutting plane line.



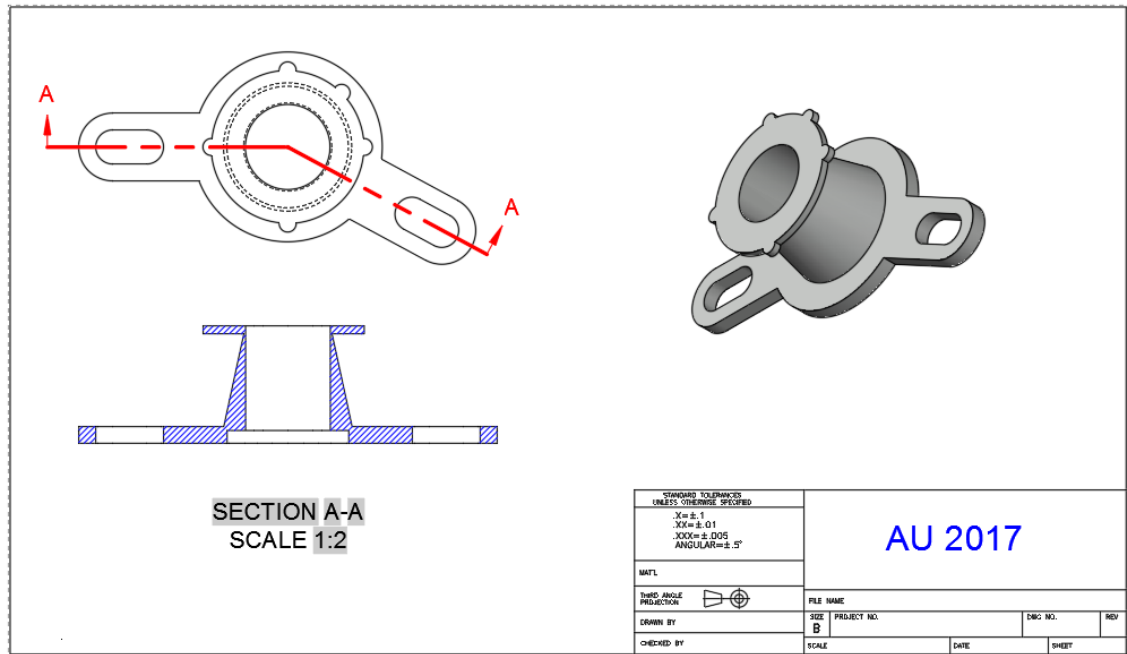
5. Double click on the front view to set the projection from **orthogonal to normal or normal to orthogonal**. Which method of projection is correct? Normal or Orthogonal.



NORMAL PROJECTION VS ORTHOGONAL PROJECTION

6. **Project the top view** as an isometric view.
7. Edit the isometric view by turning off tangent edges.

8. Edit the isometric view to **Shaded with Visible lines**.
9. Erase the orange polyline. (You may need to turn on Selection Cycling)
10. Save the drawing as **Aligned.dwg**.



Excluding Components from Sectioning

Certain features in section views, such as fasteners, are not shown. For example, components such as screws, pins, and thin-walled objects in an assembly are shown without section lines. This practice conforms to 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.

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.

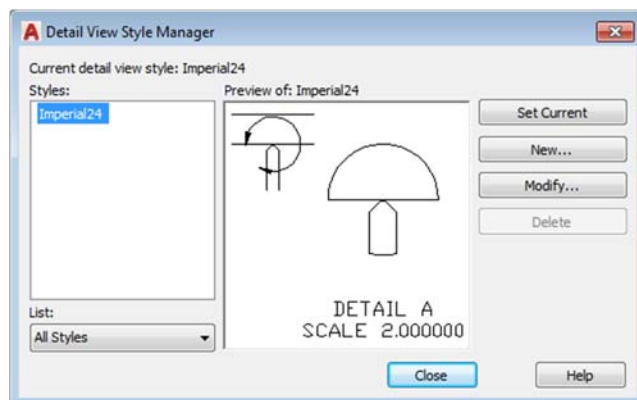
Detail views are created using the **VIEWDETAIL** command. A detail view is created by drawing a circular or rectangular boundary to define the extents of the view. You can create detail views from an AutoCAD 3D model or an Autodesk Inventor model.

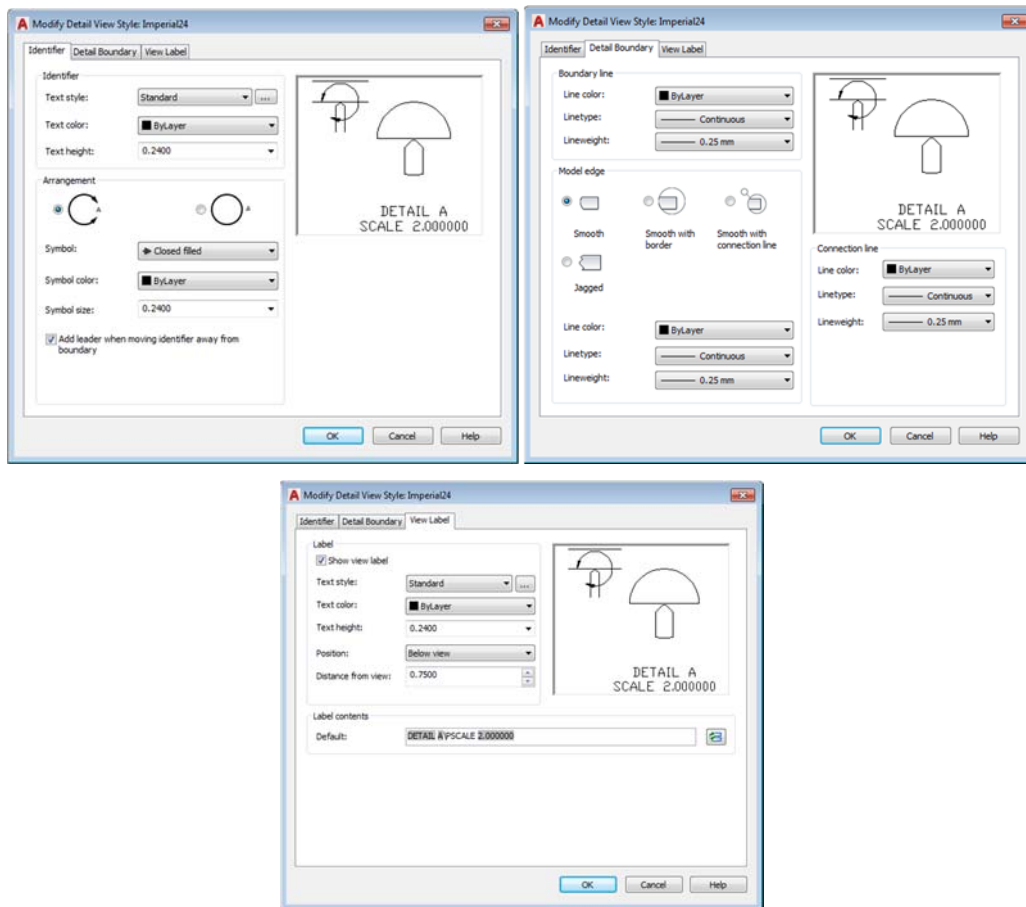
Detail views are associative. As with other types of drawing views, detail views are updated automatically when model changes are made if the **VIEWUPDATEAUTO** system variable is set to 1.

Creating a detail view places a **detail boundary** representing the “cutout” area in the parent view. The detail boundary includes a detail identifier. The detail view is placed in another location on the drawing and includes a view label. The detail identification is automatically incremented when you place additional detail views. The appearance of elements in the detail identifier and detail view label is controlled by the detail view style. A **detail view style** defines settings such as the text style and height, symbol size, and detail boundary appearance.

VIEWDETAILSTYLE

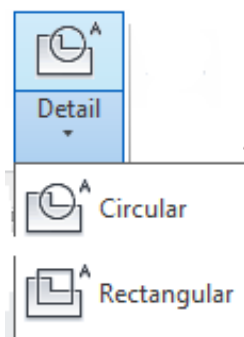
The **VIEWDETAILSTYLE** command is used to create and modify detail view styles. This command accesses the **Detail View Style Manager** dialog box below. Picking the **New...** button allows you to create a new detail view style. Picking the **Modify...** button opens the **Modify Detail View Style** 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.

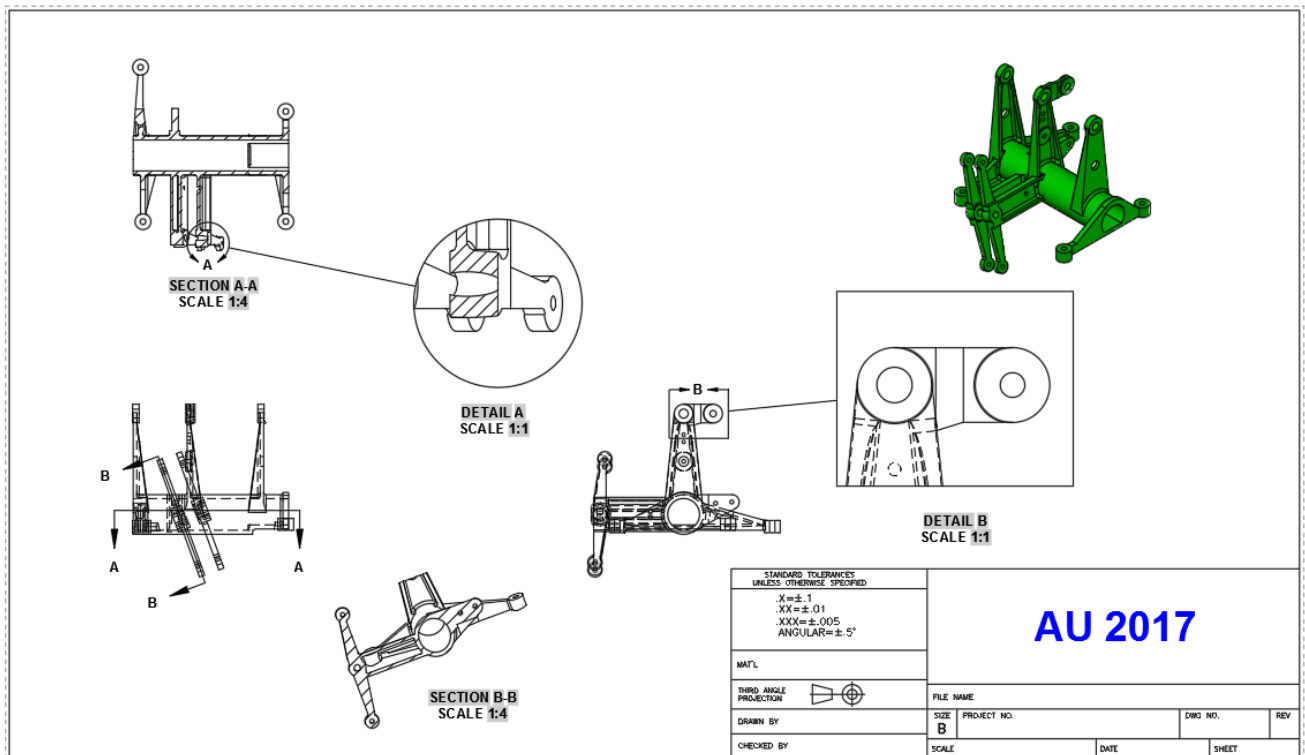
The **Hidden Lines**, **Scale**, **Visibility**, and **Move** options are the same as those previously discussed for section views. The **Model Edge** option is used to adjust the edges of the detail view and set border display and leader options. The **Smooth** option creates a smooth edge for the view. 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 grips. A detail view has a base grip providing access to the standard grip editing options 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.

Exercise 6 Details

1. **OPEN** the drawing **Airplane_Bracket_Detail.dwg**.
2. Create the **circular** detail view using the **Smooth with connection line** model edge.
3. Double click on the detail view and from the **Appearance** panel, **turn off** the **Edge Visibility** for **Tangent Edges** and **Interference edges**.
4. Create a **rectangular** detail view using the **Smooth with connection line** model edge. Change the scales of both details if needed? **1:1** or **2:1**?
5. (For extra credit-----Create the **Full Section D-D**)
6. Change the section or detail **Identifiers** as needed.



Create undocumented Auxiliary Views

Auxiliary Views

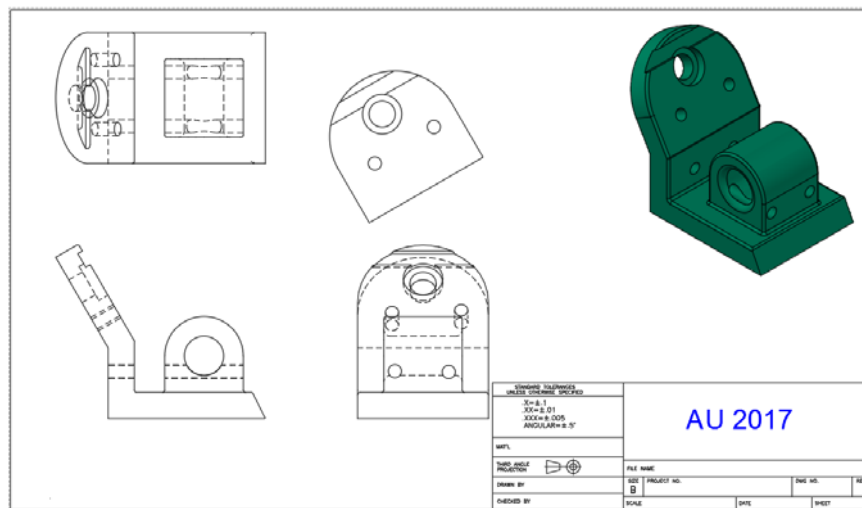
An undocumented feature in AutoCAD is the ability to create an auxiliary view by using a section view. Often, a multiview drawing contains inclined surfaces that do not describe the true size or shape of features in a regular orthographic view. To establish an auxiliary view, you can

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. Notice that since the section line does not intersect the object, no hatch pattern is created.

Exercise 7

1. **OPEN** the drawing **Auxiliary_View.dwg**
2. Select the **ANSI-B** Layout.
3. The four views have been created for you. Use grips to space the views apart as needed.
4. On the ribbon, select the **Layout** tab.
5. **Project** an auxiliary view by creating a full section using the **From Object** method.
You can also use a polyline drawn across the inclined edge of the front view and use the **From Object** option to create the auxiliary view. The orange polyline has been drawn for you. **Thaw** the Construction Layer to use the polyline if needed.

Why is the Cutting plane line and the section lines invisible?



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

Export drawing views – EXPORTLAYOUT command

Have you had the need to create a multiview drawing from an AutoCAD 3D model? Create the associated drawing view and then export the drawing view layout. A new 2d model space

drawing is created!

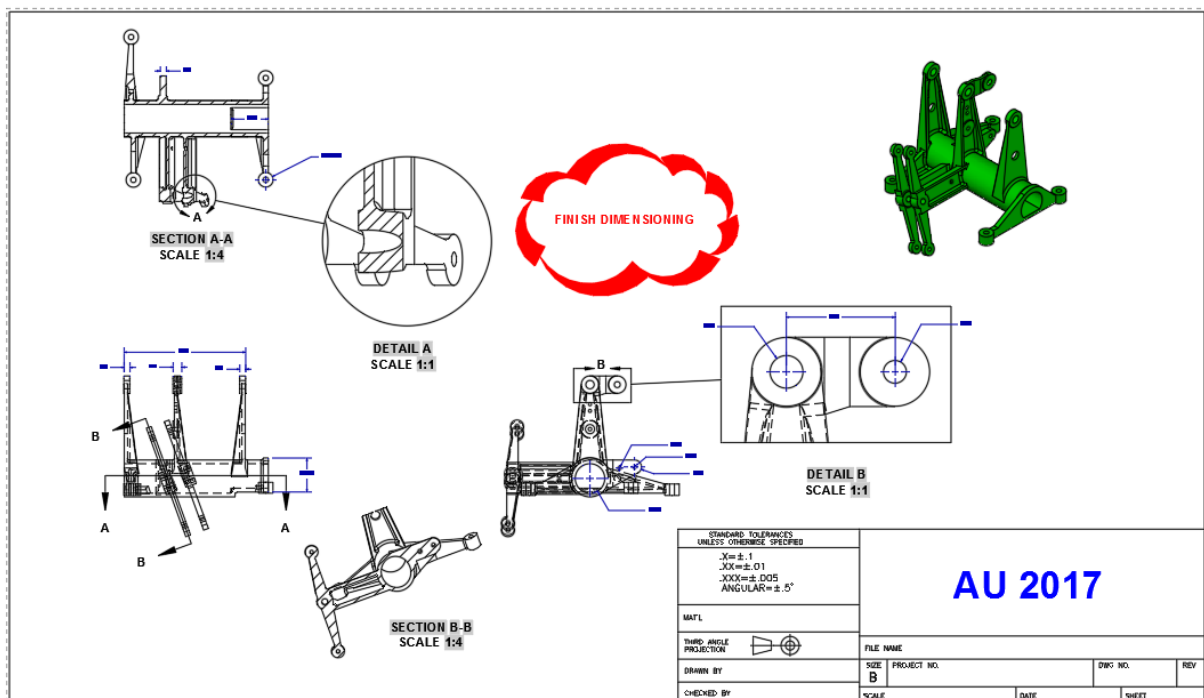
You can use the **EXPORTLAYOUT** command to export a layout containing drawing views into a new drawing (DWG) file. However, this technique will break the associativity between the 3D model and the drawing views. The advantage to using this technique is that the drawing geometry can be edited in the 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.

Exercise 8A

1. **OPEN** the drawing, **Airplane_Bracket_Export.dwg**.
2. Select the **ANSI-B** layout. Notice the two full sections and the two details. Also, some dimensions have been placed. export

We need to export the layout into model space to create a new drawing of only model space objects!

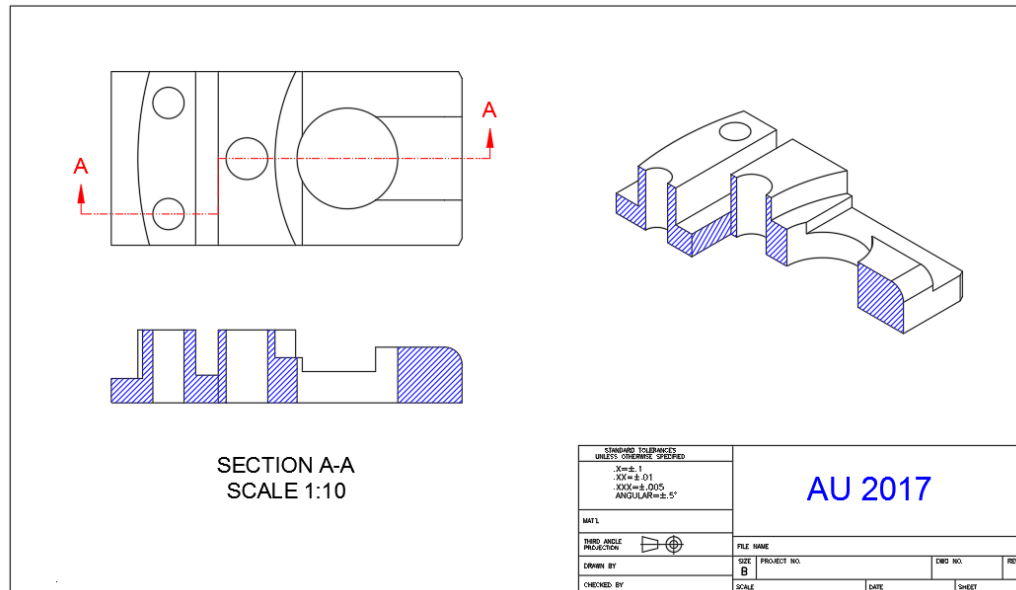
3. Type, **EXPORTLAYOUT** to export the layout. Save the drawing to the lab drawing folder as, **Airplane_Bracket_Export_ANSI_B.dwg**.
4. When the **Exports Layout to Model Space Drawing** dialog appears, select **OPEN**.
5. Select some of the newly created blocks in the drawing. Notice their block names. You can rename the blocks or edit the blocks as needed.



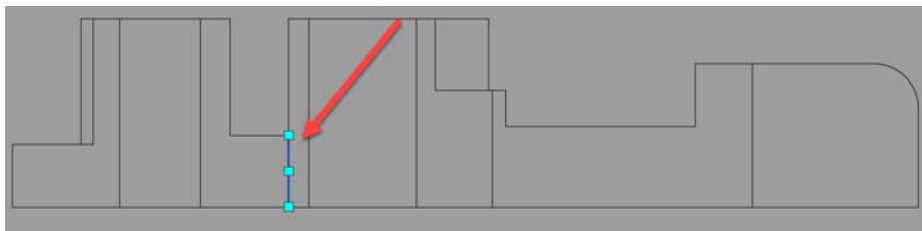
6. For extra credit.....**Explode** the drawing! Type **OVERKILL** and select **ALL** objects. Click **OK** in the **Delete Duplicate Objects** dialog box.
7. How many overlapping objects do you have? How many duplicate objects do you have? Will the file size be smaller when you Save the drawing after running the **OVERKILL** command?
8. Save the drawing as **Airplane_Bracket_in_Model_Space.dwg**

Exercise 8B Removing object lines from a sectional view.

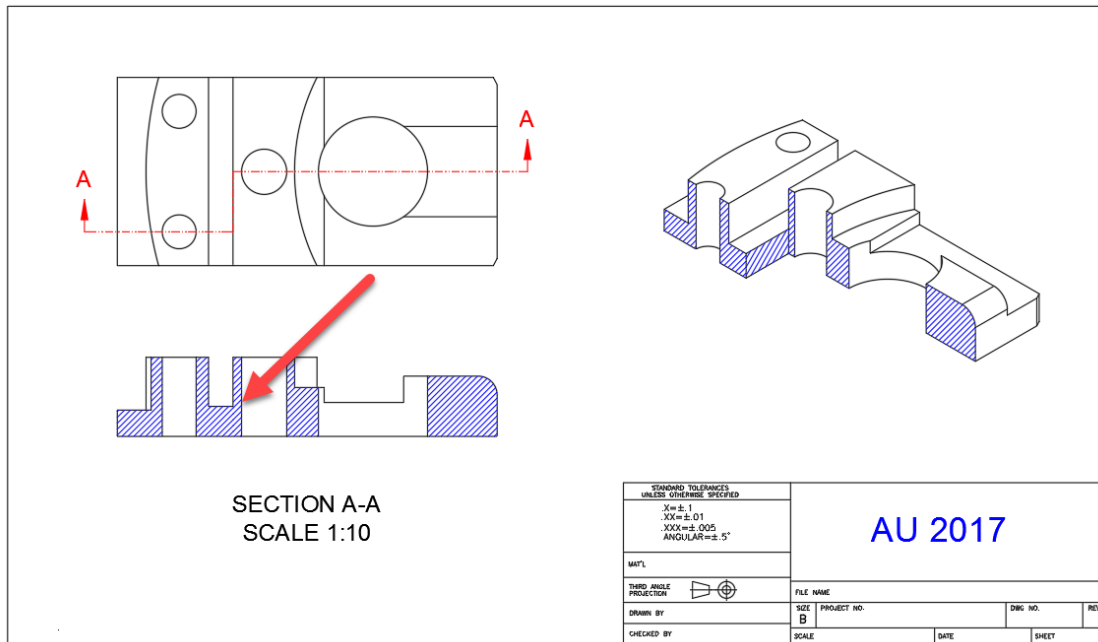
1. **OPEN** the drawing, **Offset_Base_Section_Finished.dwg**.



2. Type, **EXPORTLAYOUT** to export the layout. Save the drawing to the lab drawing folder as, **Offset_Base_Section_Finished_ANSI B.dwg**. (**Overwrite the file if needed**).
3. When the **Exports Layout to Model Space Drawing** dialog appears, select **OPEN**.
4. Use the **BEDIT** command to edit the sectioned view. Erase the object line that should not be included in the sectional view.



5. **Close** the Block Editor and **Save** the changes.
6. The sectioned view has been updated. The sectioned view reflects ANSI standards.



7. Save the drawing as **Offset_Base_Section_in_Model_Space.dwg**.

NOTES:

The **FLATSHOT** command can also be used to create a multiview orthographic drawing from a 3D model. However, the resulting drawing views do not have associative properties. The FLATSHOT command creates a flat projection of the 3D objects in the drawing from the current viewpoint. The view is created in model space. The view that is created is composed of 2D geometry and is projected onto the XY plane of the current UCS.


Constraints

You can turn on constraints to assist in the creation of Multiview geometry.

(From the AutoCAD Help menu...)

To Refine Constraints Between a Section Line and View Geometry

Typically, constraints are inferred between the view geometry and the section line. In most cases this is sufficient. However, under exceptional circumstances you may need to add additional constraints.


Click Layout tab ► Modify View panel ► Symbol Sketch.  Find

Click the section line to constrain.

The symbol sketch mode starts. With the exception of the view geometry, all geometry (including the section line) is dimmed. A polyline that coincides with the section line indicates the cutting plane.

Add the required constraints to constrain the cutting plane.


Tip: If you require construction geometry to implement the constraints, simply add the geometry. All geometry added in symbol sketch mode is construction geometry. They are not visible when you exit symbol sketch mode.

On the ribbon, click Finish Symbol Sketch.  Find

The cutting plane converts to the section line. The construction geometry becomes invisible.

To Refine Constraints Between a Detail Boundary and View Geometry

Note: Model documentation is available only on 64-bit systems.

Click Layout tab ► Modify View panel ► Symbol Sketch.  Find

Click the detail boundary to constrain.

The symbol sketch mode starts. With the exception of the view geometry, all geometry is dimmed.

Add the required constraints to constrain the detail boundary.

Tip: If you require construction geometry to implement the constraints, simply add the geometry. All geometry added in symbol sketch mode is construction geometry. They are not visible when you exit symbol sketch mode.

On the ribbon, click Finish Symbol Sketch.  Find

The construction geometry becomes invisible.