MFG464143-L

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

J.C. Malitzke
Digital JC CAD Services Inc.

Learning Objectives

- 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

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Description

This intermediate-to-advanced hands-on lab offers AutoCAD 3D software veterans a chance to explore the 2D model documentation of 3D part models from AutoCAD 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.

Speaker

J. C. Malitzke is president of Digital JC CAD Services, Inc., and he is the former chair of the Computer Integrated Technologies department and a faculty member at Moraine Valley Community College in the greater Chicago area. He also managed and taught for the college's Autodesk Authorized Training Center. He has been using and teaching Autodesk, Inc., products for 30+ years. J. C. is co-author of AutoCAD and Its Applications Advanced Goodheart-Willcox. J. C. is an Autodesk University Top Presenter Award winner, an Autodesk Certified Professional and Instructor for AutoCAD software and Inventor software and was Autodesk Certification Evaluator. J.C. has 23+ years presenting at Autodesk University and has was on the Autodesk University Mentoring Steering Committee for 3 years. He holds a BS degree in education and an MS in industrial technology from Illinois State University. Contact J. C. at cadinfo@digitaljccad.com.

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

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Introduction to Model Documentation

There are multiple techniques available to create drawing views from 3D models in AutoCAD. Using these techniques, orthographic, isometric, section, and auxiliary views can be created in order to produce multiview drawings. The techniques available include the following.

• Use the **VIEWBASE**, **VIEWPROJ**, **VIEWSECTION**, and **VIEWDETAIL** commands to create multiview drawings that are derived from and associated to a 3D solid or surface model. The drawing views are placed in a paper space layout. This is the most efficient method to create multiview drawings in AutoCAD.

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.

The **VIEWBASE** command can be used to create drawing views from AutoCAD 3D models or from Autodesk Inventor® part (IPT), assembly (IAM), or presentation (IPN) files. *And from translated 3D models from other CAD softwa*res. You can also create drawing views from models imported with the **IMPORT** command

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.

**Drawing View and Layout Setup**

As previously mentioned, drawing views created with the **VIEWBASE** command are created in paper space. To activate paper space, pick 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…**.

VIEWSTD

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 **Layout** tab of the ribbon.
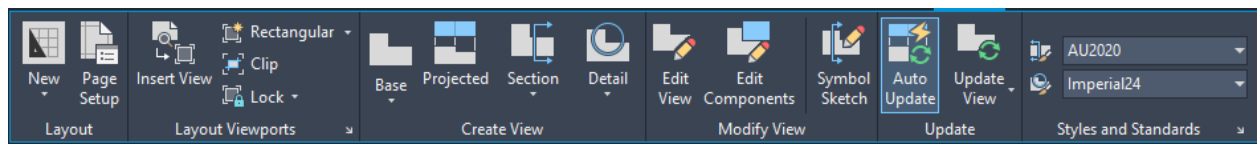
Pick the dialog box launcher button at the lower-right corner of the **Styles and Standards** panel to initiate the **VIEWSTD** command. This opens the **Drafting Standard** dialog box.
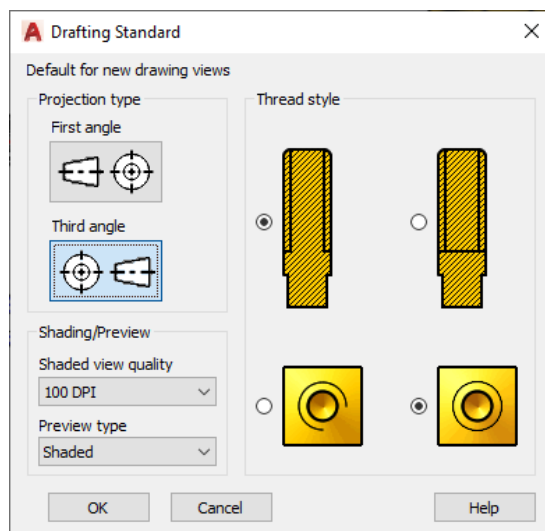

The options in the **Projection type** area determine the type of projection used for new drawing views. By default, the **Third angle** button is selected. If the **First angle** button is selected, views are created using first-angle projection. Select this option to create views in accordance with ISO standards. The options in the **Thread style** area determine the way threaded ends are shown in section views and the type of circular 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.

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, the **Select File** dialog box appears. From this dialog box, you can select a model created in Autodesk Inventor.

When you select the **VIEWBASE** command from the ribbon, a preview representing a scaled orthographic view of the model appears. This is the 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 Drawing View Creation contextual ribbon tab.
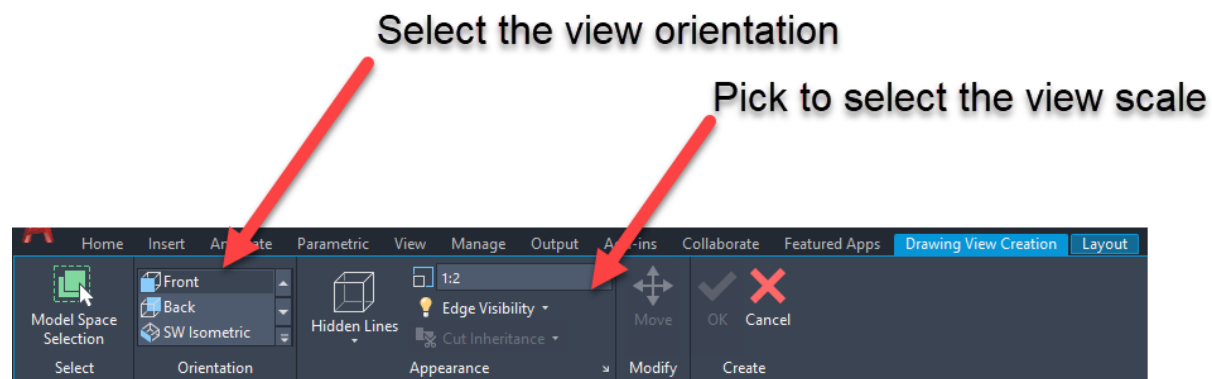
The Select option switches you to model space and allows you to add objects to or remove objects from the base view. If necessary, use the appropriate option to select solids or surfaces to add or remove. Use the Layout option to return to the layout after making changes. The Orientation option can be used to select a different view from the default base view. The Orientation options correspond to AutoCAD's six orthographic and four isometric preset views. 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. In the top view of the model is selected as the base view. The Scale is set to 1:1 with Hidden line visible. 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.
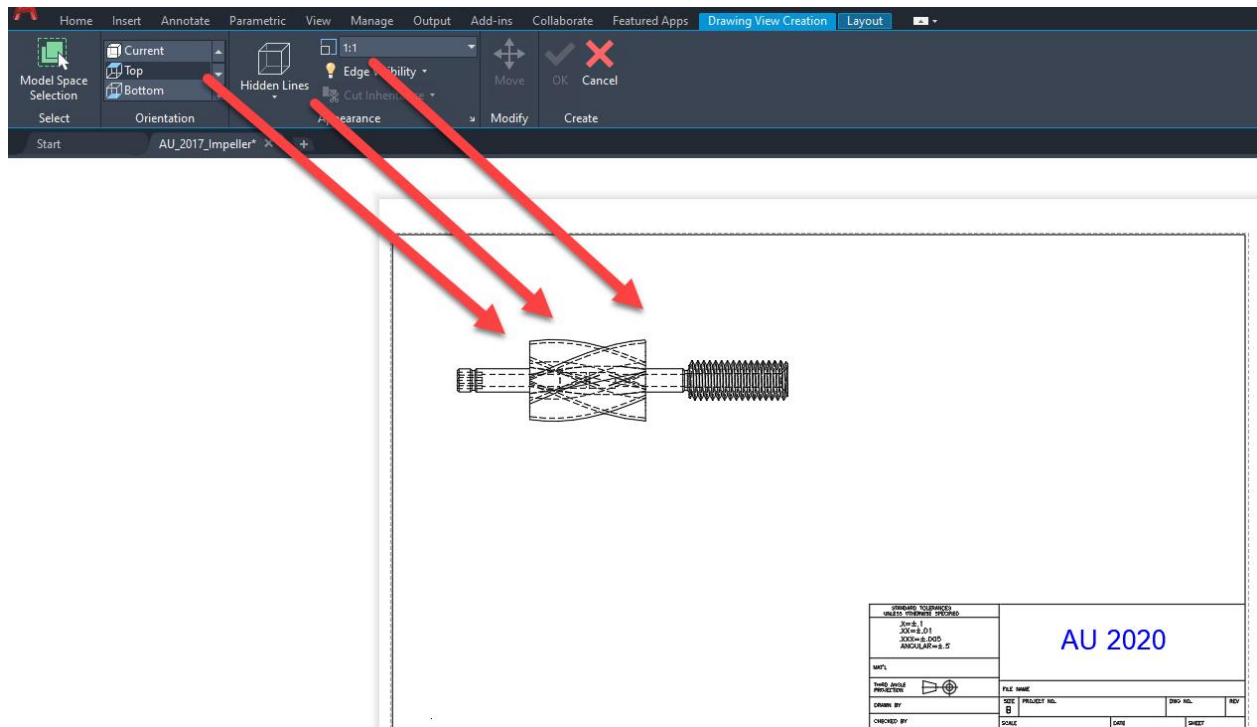
The Type option of the **VIEWBASE** command is available before picking the initial location of the base view. This option is used to specify whether projected views are placed after placing the base view. By default, this option is set to Base and Projected. With this option, you can place additional, projected views while the **VIEWBASE** command is still active. Projected views that you place are 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 parent view and any projected views are child views. If the base view is moved, for example, the projected view is moved with it. Projected views are not created with the **VIEWBASE** command when the Type option is set to 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 **VIEWPROJ** command. The **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 Type option is set to 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.

The display style of the base view can be changed by using the Hidden Lines option. The four display style options available are Visible Lines, Visible and Hidden Lines, Shaded with Visible Lines, and Shaded with Visible and Hidden Lines.
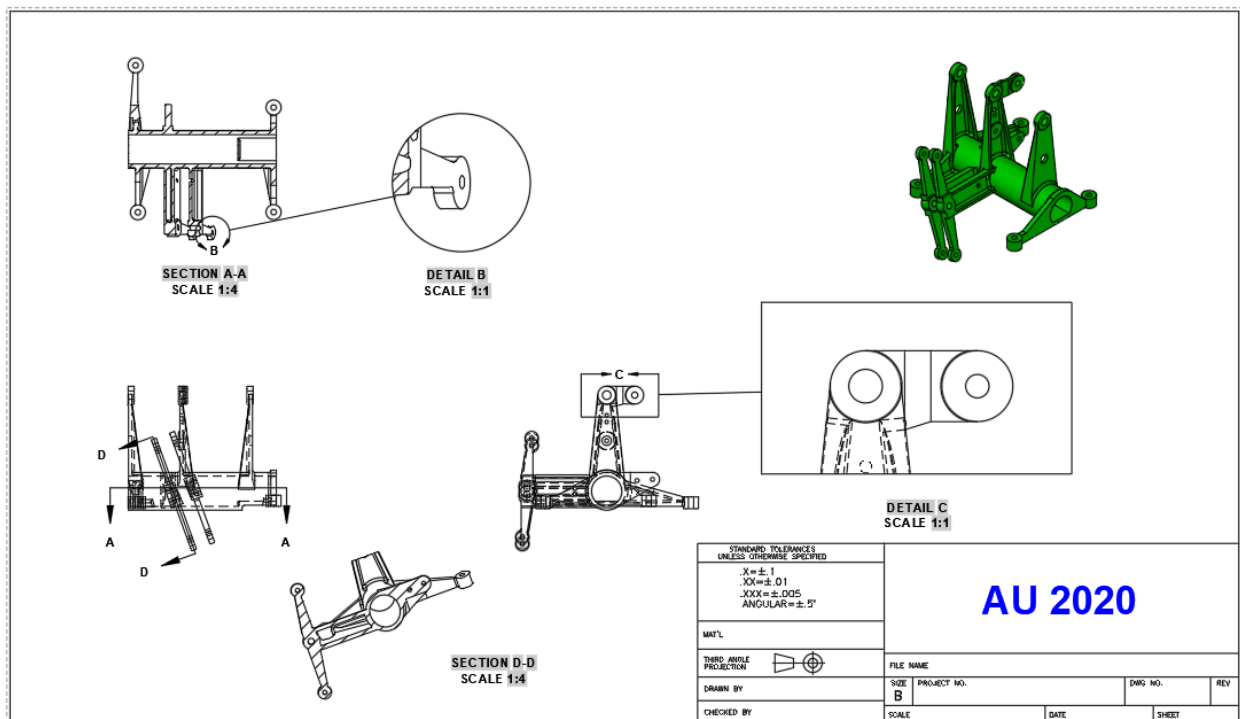
These options can be selected using dynamic input or the drop-down list in the Appearance panel in the Drawing View Creation contextual ribbon tab.

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

The Move option allows you to move the base view after picking the initial location on screen. When you select the 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 VIEWBASE options are again made available.

The Visibility option allows you to control the display of drawing geometry in the view. The Interference edges option controls whether both object and hidden lines are displayed for interference edges. By default, this option is set to No. The Tangent edges option controls whether tangential edges are displayed to show the intersection of surfaces. By default, this option is set to No. If this option is set to Yes, you can specify whether tangent edges are shortened to distinguish them from object lines that overlap. The Bend extents option is only available when working with a model that includes a view with sheet metal bends. The Thread features option controls thread displays on models with screw thread features. The Presentation trails option is used to control the display of trails in views created from presentation files.

Additional drawing view options can be accessed in the Appearance panel of the Drawing View Creation contextual ribbon tab. Selecting the dialog box launcher button at the bottom of the panel 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 Centered, the geometry updates about the center point of the view.

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.

You can create more than one base view in a layout. Additional base views can be used to create assembly or subassembly drawings.

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.
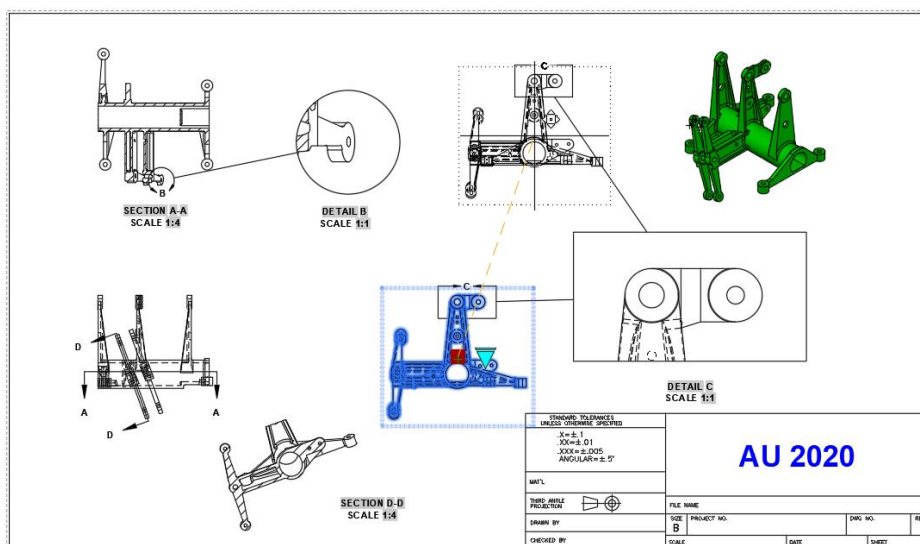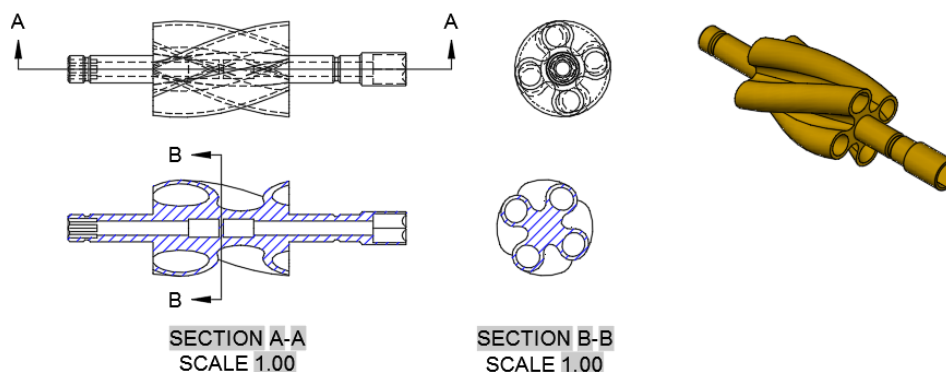
Section Views

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.
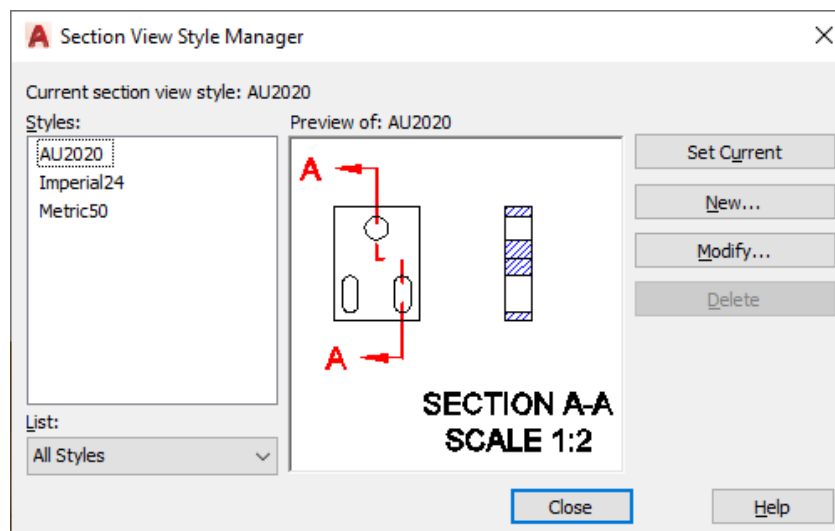
Picking the New… button allows you to create a new section view style. Picking the Modify… button opens the Modify Section View Style dialog box for the selected style. The tabs in the New Section View Style dialog box or the Modify Section View Style dialog box are used to make settings for the section 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.

Professional Tip: Create your own company's section view style. Do not use the AutoCAD default.

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.

Exercises

  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_2020_Impeller.dwg (Note: The original 3D model was created in Autodesk Inventor)**
2. Select the **ANSI-B Layout** to make the layout current.
3. On the ribbon, select the **Layout tab**. Notice that no viewports are available!
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.
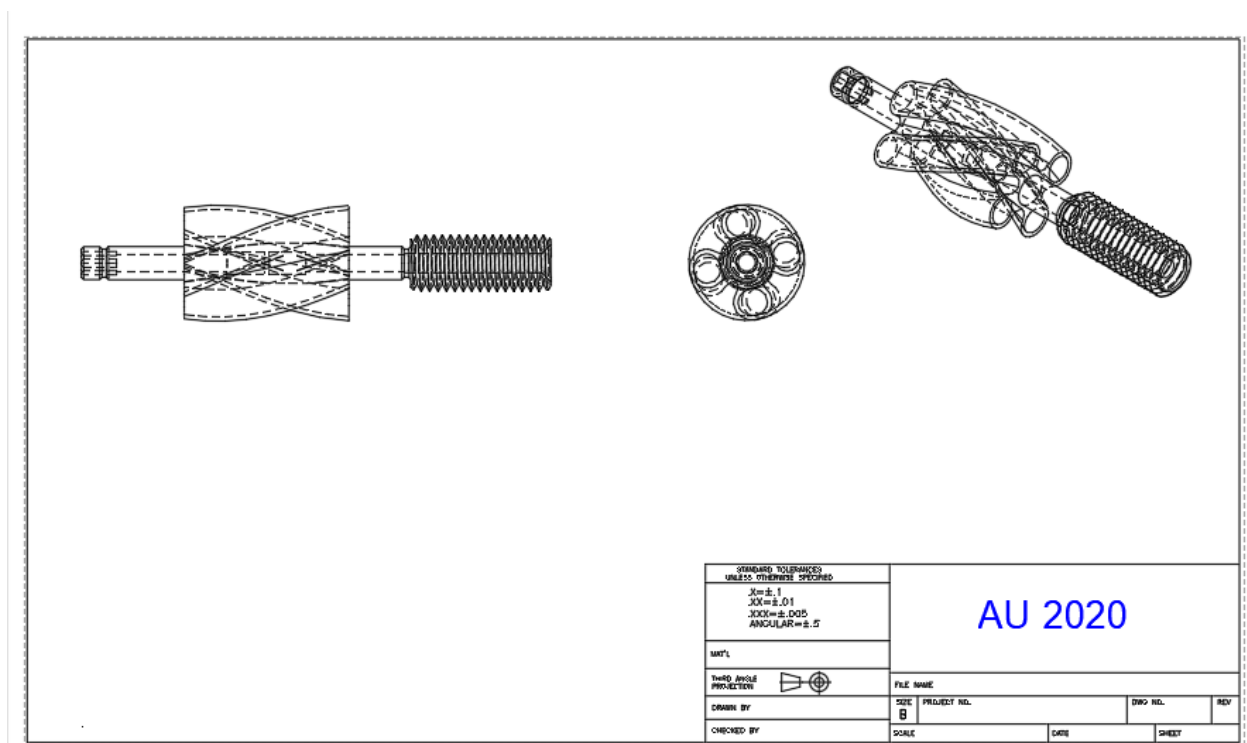
5. Edit the Isometric view by *double* clicking on the **Isometric view** and from the **Appearance panel**, change **From Hidden Lines to Shaded with Visible Lines.** Select **OK** to complete the edit. The isometric view is shaded with visible lines shown.
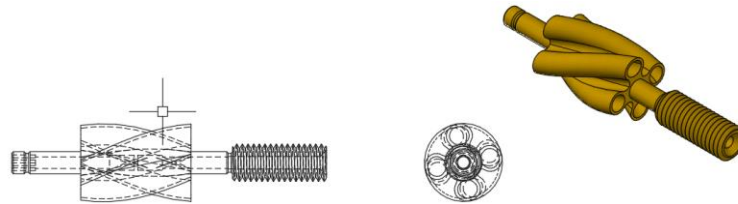
6. From the **Create View** panel, select the section icon to create a **Full** section by passing the cutting plane line through the front view, horizontally creating section A-A. (Note: Make sure the cutting plane line is positioned at the midpoint of the left vertical edge line of the view when selecting to create the cutting plane line). Place the Full section A-A under the front view as shown and **Exit** (Hint: make sure **Show Hatching** is on in the Section View Creation Tab, Hatch panel).
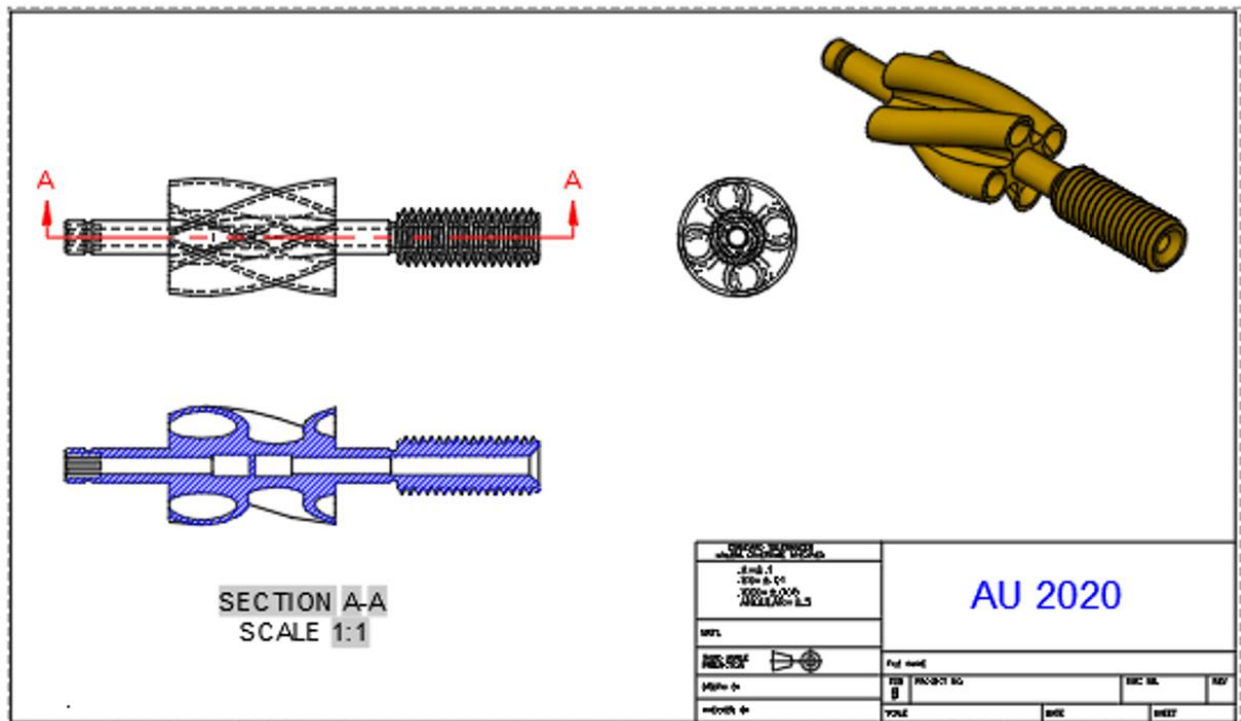
7. From the **Create View** panel, select the section icon to create a **Full** section by passing the cutting plane line through the front view vertically, creating section B-B. Place the Full section B-B under the right-side view as shown. Hint: Select the new section view. Select the grip. Press (Tap) the [SHIFT] key to break the orthographic alignment when placing section B-B.
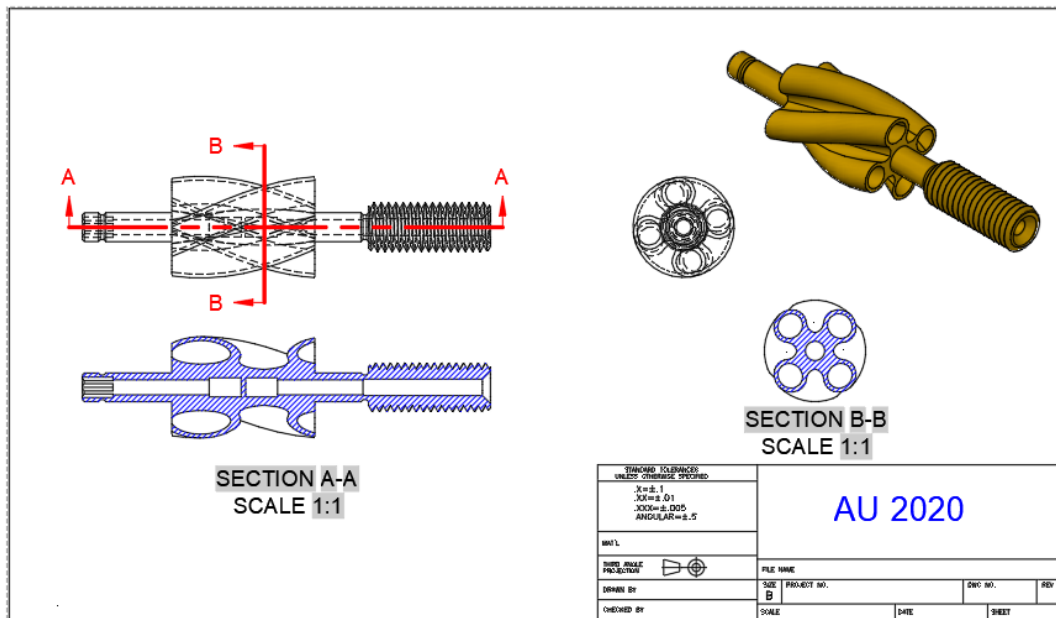
8. **SAVEAS** the drawing **AU_2020_Impeller_Finished.dwg**

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.**
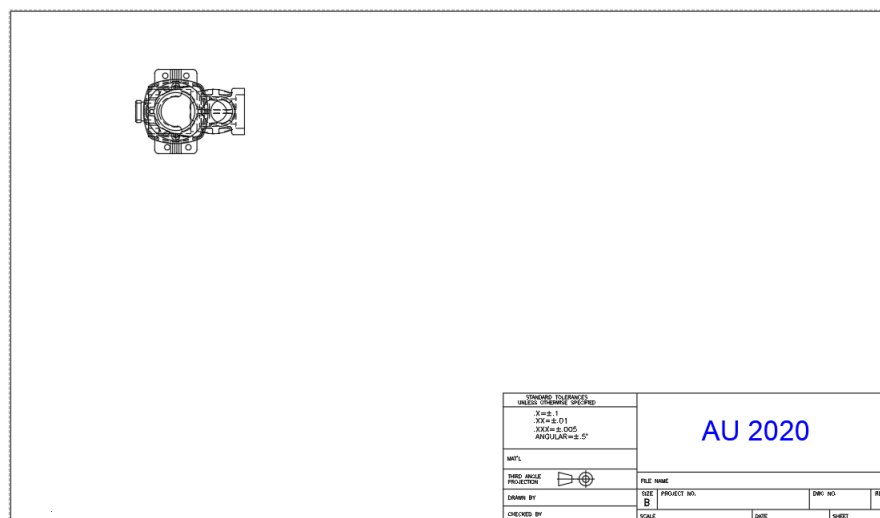
5. From the **Create View** panel, select the section icon to create a **Full** section by passing the cutting plane line through the top view, horizontally creating section A-A. Place the Full section A-A under the top view as shown. Scale is 1:1. **Exit** to place the sectional view.
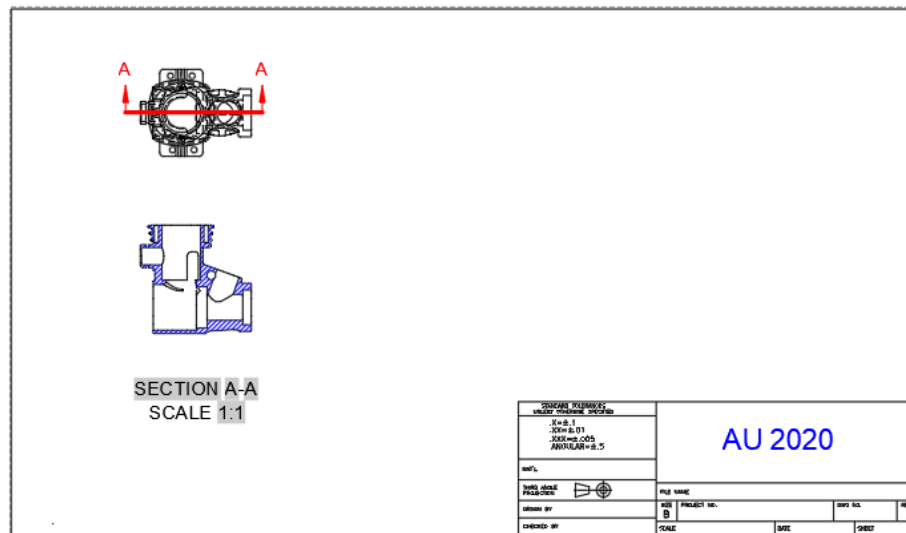
6. From the **Layout tab**, **Create View panel**, select **Projected** and project the isometric view from the front view as shown.
7. Edit the Isometric view by *double* clicking on the **Isometric view** and from the **Appearance panel**, change **From Hidden Lines to Shaded with Visible Lines. Scale** the isometric view to **2:1. Select Exit** to complete the edit. The isometric view is shaded with visible lines shown.
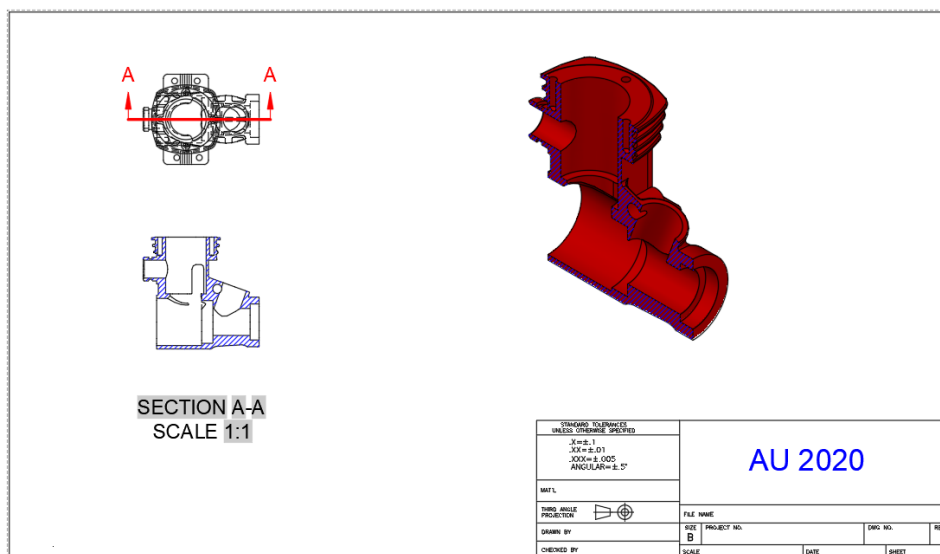
8. Select the front view. Zoom in a little and then select the hatch pattern. Change the hatch pattern to **ANSI 38**.
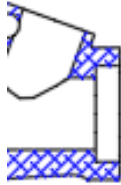
9.   Close the Hach Editor when done.   **SAVE** the drawing as **AU_2020_Engine_Finished.dwg**
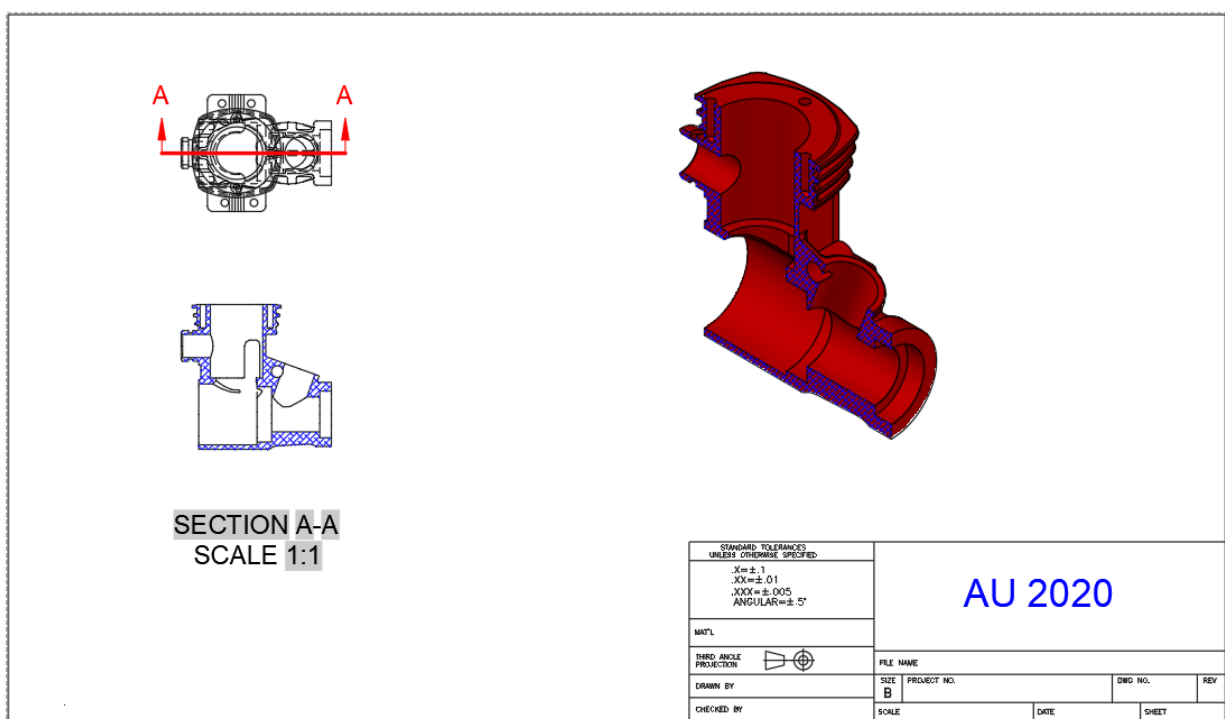
Exercise 1C Full Section

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, **Section** 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 in the top view.

SECTION A-A
SCALE 1:4

AU 2020
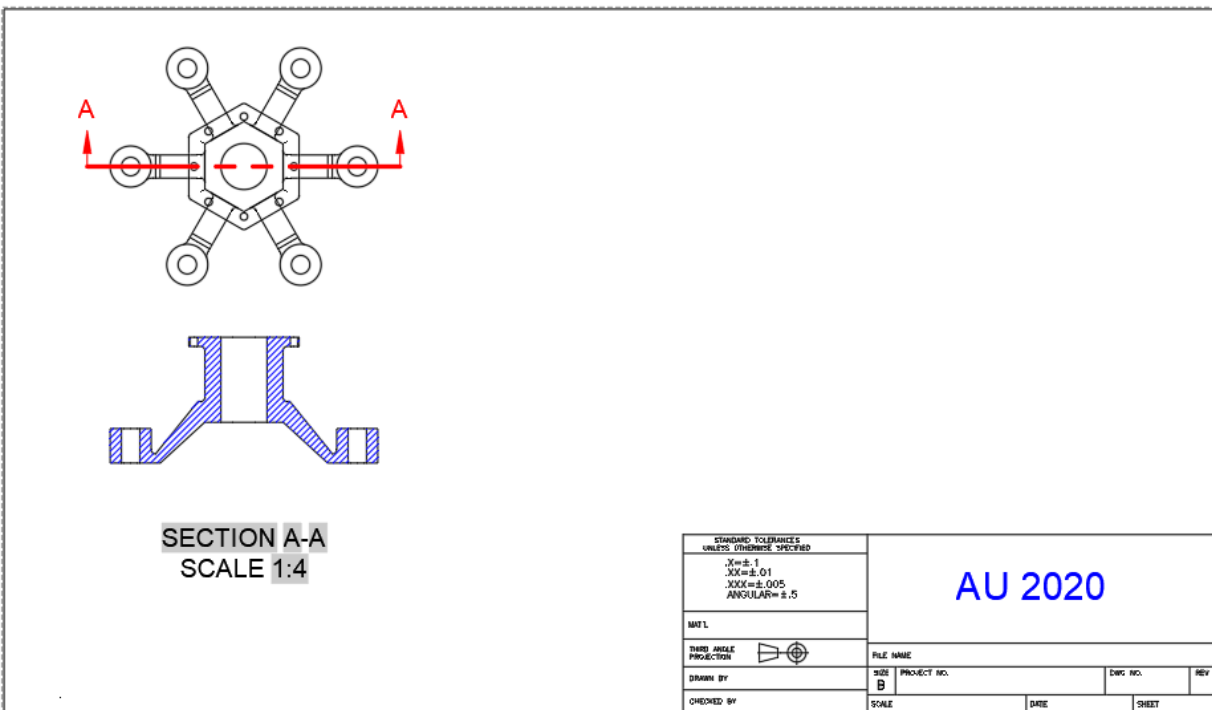
5. Select a location for the section view and the **OK**.
6. Project an isometric view from the **Front sectioned view**. Edit the view to set Shaded with visible lines.
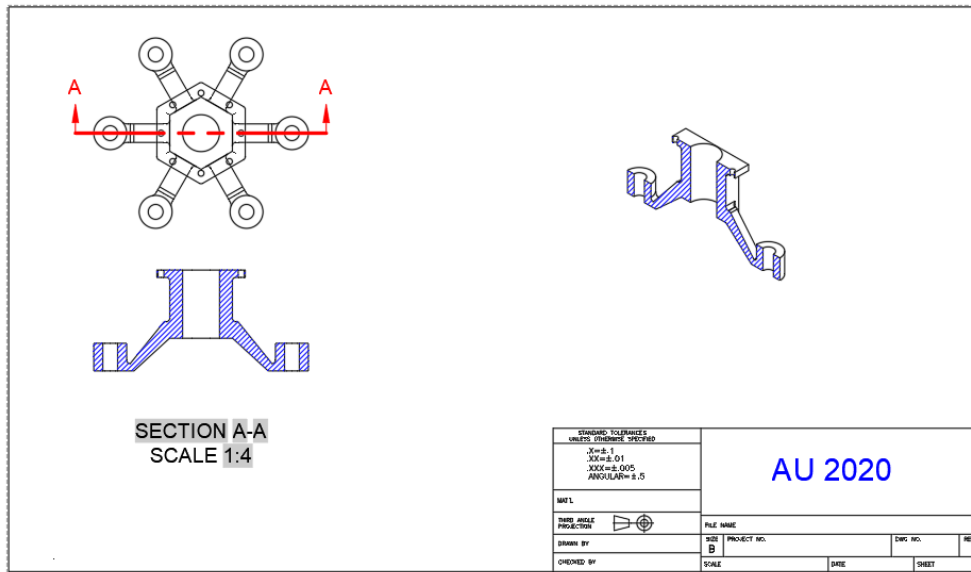
7. Edit the view to set **Shaded with visible lines** and turn off the **Cut Inheritance** by selecting **Section cut**. Scale the isometric view **1:2.**
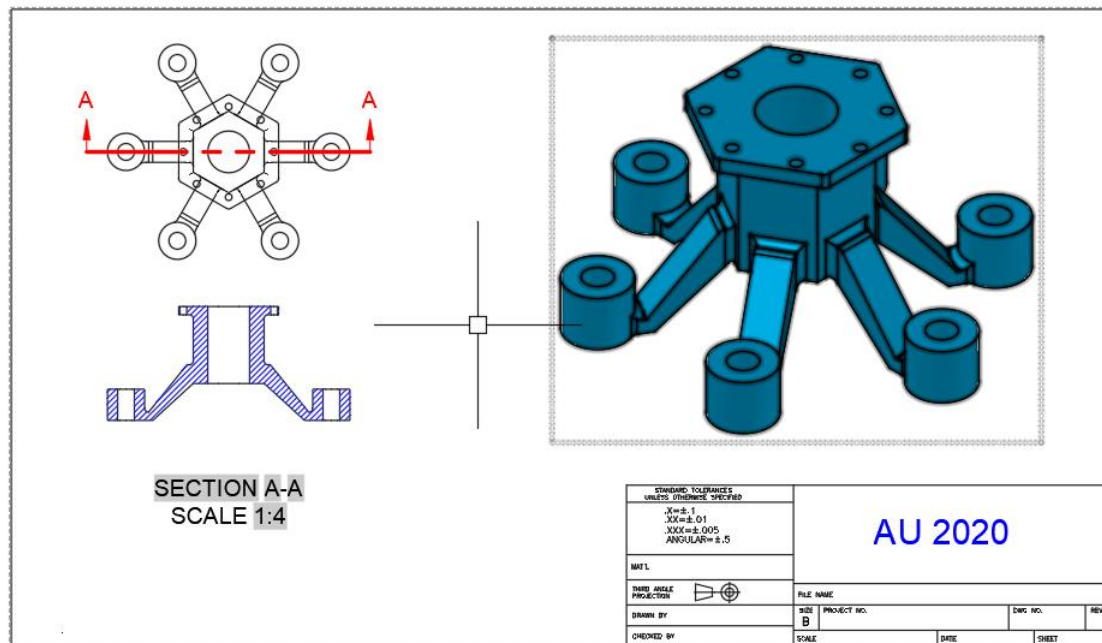
8. **SAVE** the drawing as **SupportBrace_Finished.dwg**

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.
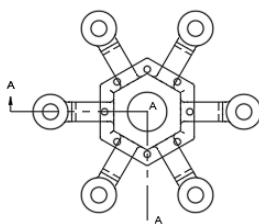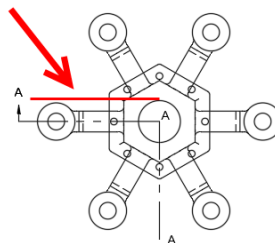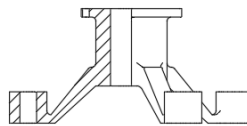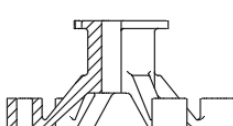

Fig. A — SECTION A-A SCALE 1:4
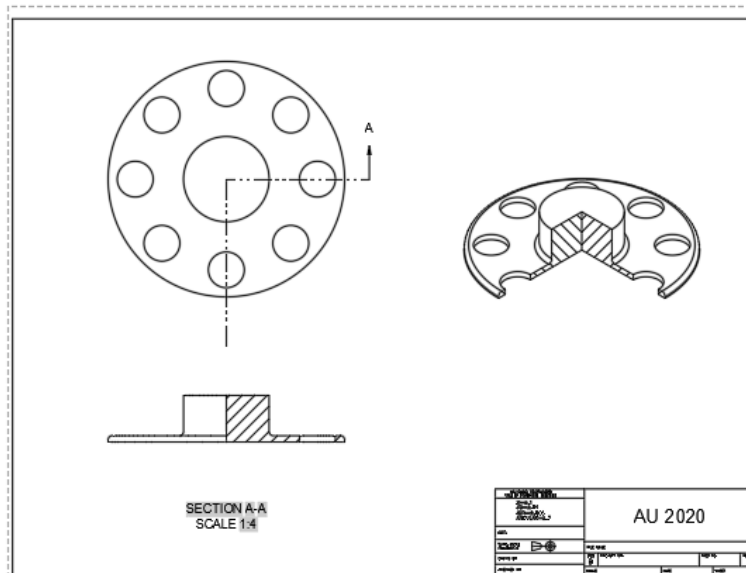
FIG. B — SECTION A-A SCALE 1:4

Exercise 2A Half Section

1. **OPEN** the drawing **Half_Section1.dwg**.
2. Select the **ANSI A** layout.
3. From the **Layout Tab**, **Create View** panel, create a base view with orientation as **Top** view.   Set scale to **1:4**.
4. Create a **Half** section as the front view. Select locations 1 (off right edge, 2 center of circle and 3 location of front sectioned view.
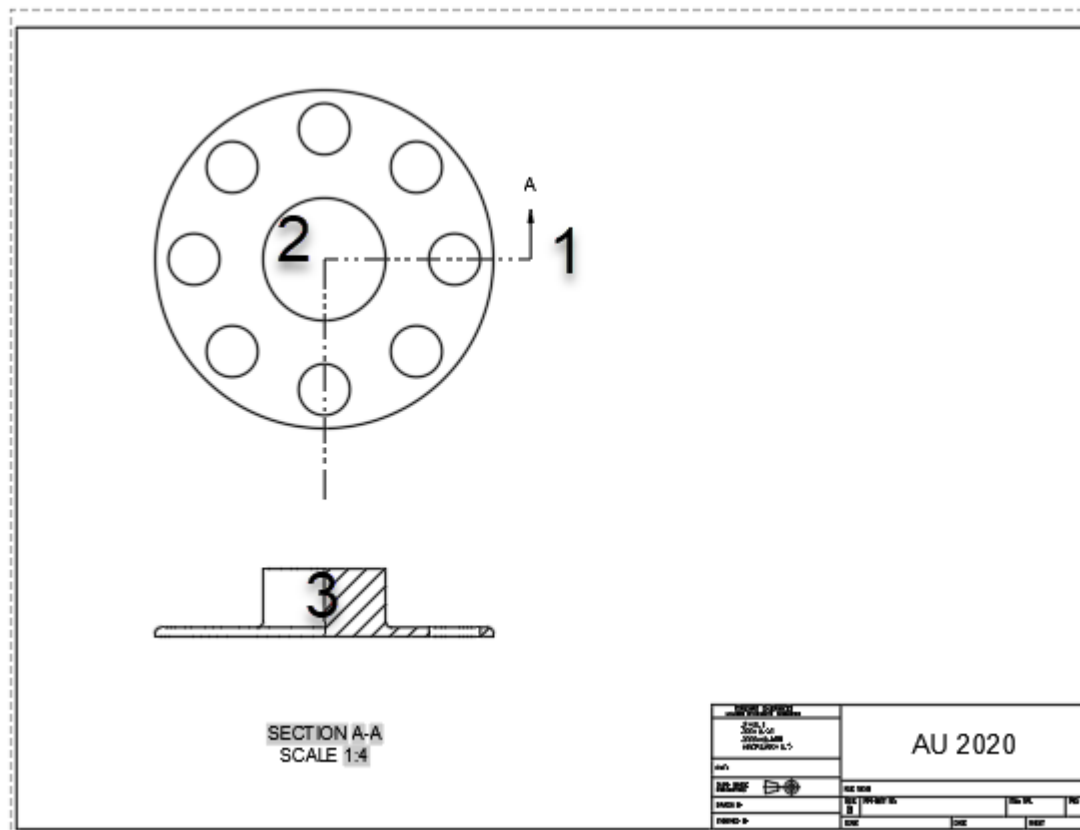

5. Create the isometric view as shown. Project from the front view.
6. Turn on Show/Hide lineweights icon in the Status Bar. If all the drawing views are not currently on the MD_Visible layer, use Properties to change all drawing views to the MD_Visible layer. (The visible lines of the part should be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

SECTION A-A
SCALE 1:4

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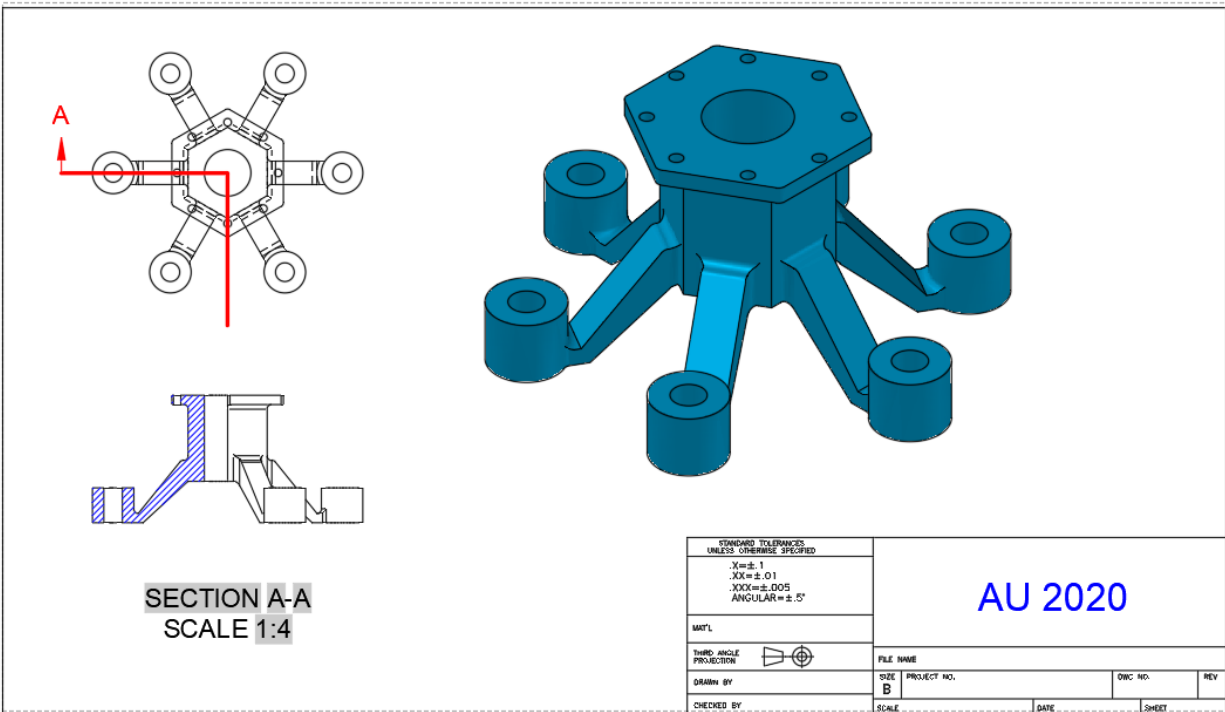7. **SAVE** the drawing as **Half_Section1_Finished.dwg.**

Exercise 2B Half Section

1. **OPEN** the drawing **Half_Section2.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.**
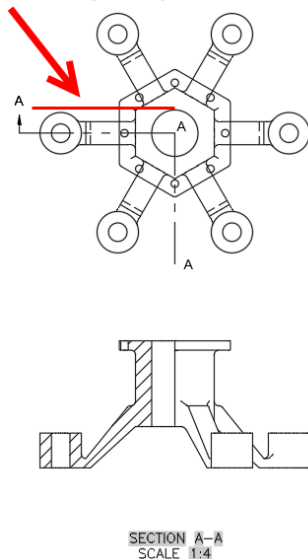
SECTION A-A
SCALE 1:4

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** by selecting **Section cut**. **Turn off the Edge Visibility, Tangent edges.** Select **OK.**

SECTION A-A
SCALE 1:4

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8. **SAVE** the drawing as **Half_Section2_Finished.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.
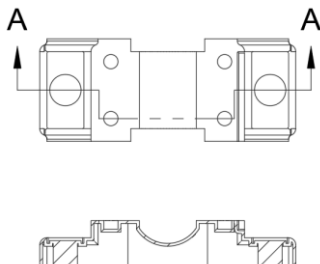
To create an offset section, select the **VIEWSECTION** command and select the parent view. Next, select the **Type** option and then select the **Offset** option. The option is accessed

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_Base_Section_Finished.dwg.**

SECTION A-A
SCALE 1:8

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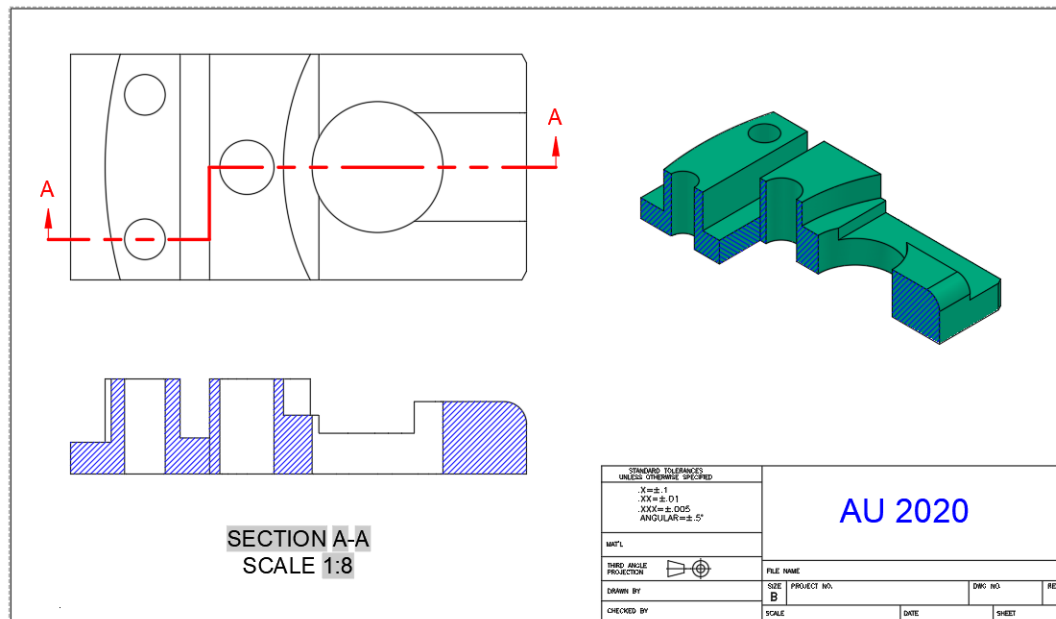
Technical Question:

When you look and the offset section view and the isometric view, do you see any drafting standards errors created by the projections? If so, where?

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. Select the top view and **Edit View** from the **Edit** panel. Change the appearance from **Hidden Lines** to **Visible Lines**.
5. **Projec**t 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.*
6. Project an isometric view from the front view showing the isometric section.
7. Project another isometric view applying **Cut Inheritance**.
8. Edit the one isometric view to **Shaded with Visible lines**.
9. **SAVE** the drawing as **Front_Fork_Offset_Finished.dwg.**

SECTION A-A
SCALE 1:2

AU 2020
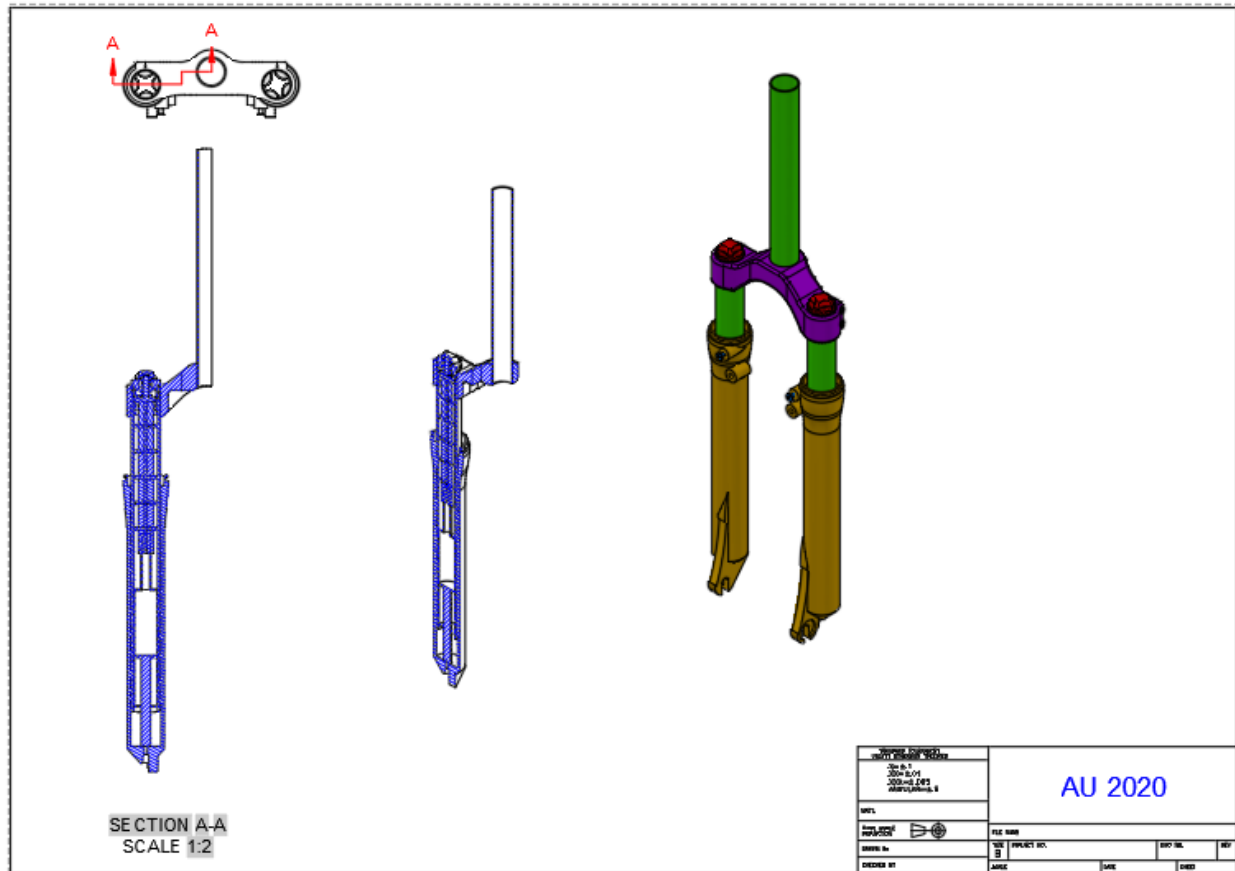
Aligned Section

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 Aligned Section

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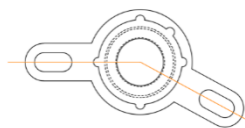
5.   Double click on the front view to set the projection from **orthogonal to normal or normal to orthogonal**.   Which method of projection is correct?   Normal or Orthogonal.
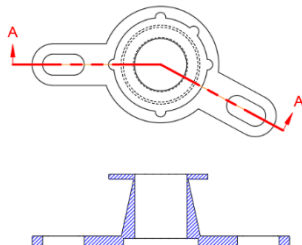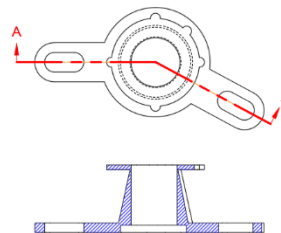

NORMAL PROJECTION   VS   ORTHOGONAL PROJECTION

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_Section_Finished.dwg.**

STANDARD TOLERANCES
UNLESS OTHERWISE SPECIFIED
.X=±.1
.XX=±.01
.XXX=±.005
ANGULAR=±.5°

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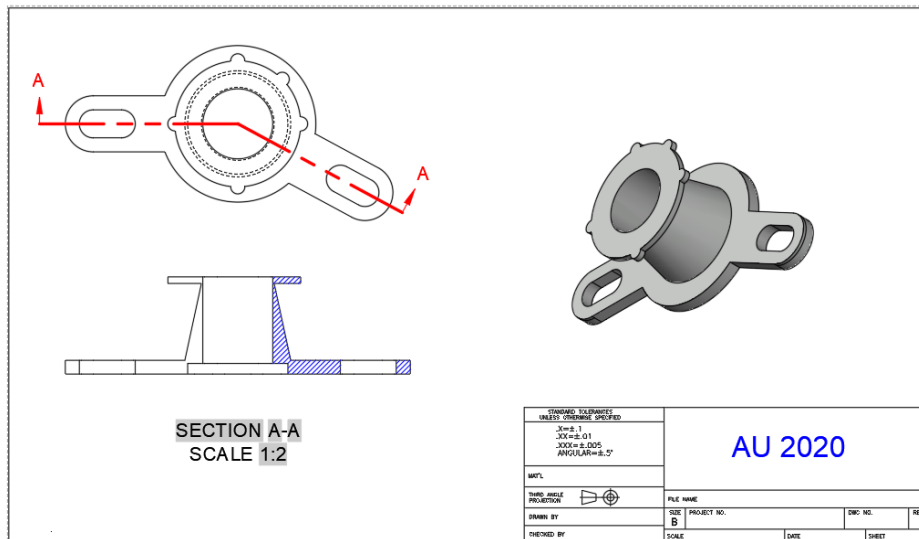
SECTION A-A
SCALE 1:2

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.

Exercise 5 Editing Section Views

1. **OPEN** the drawing **Fixture_Assembly.dwg**.
2. Select the **ANSI-B** Layout.
3. On the **Layout** tab, **Modify View panel**, select **Edit Components. From the front view,** select the six socket head cap screws and the six pins to remove the section lines by setting to **None.**
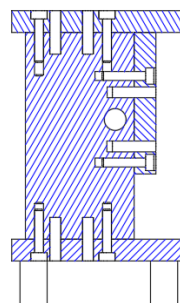
SECTION A-A
SCALE 1:2

4. In the front view, double click the hatch patterns and change the hatch patterns to **ANSI 38**, Aluminum.
5. Notice the changes in the isometric view!
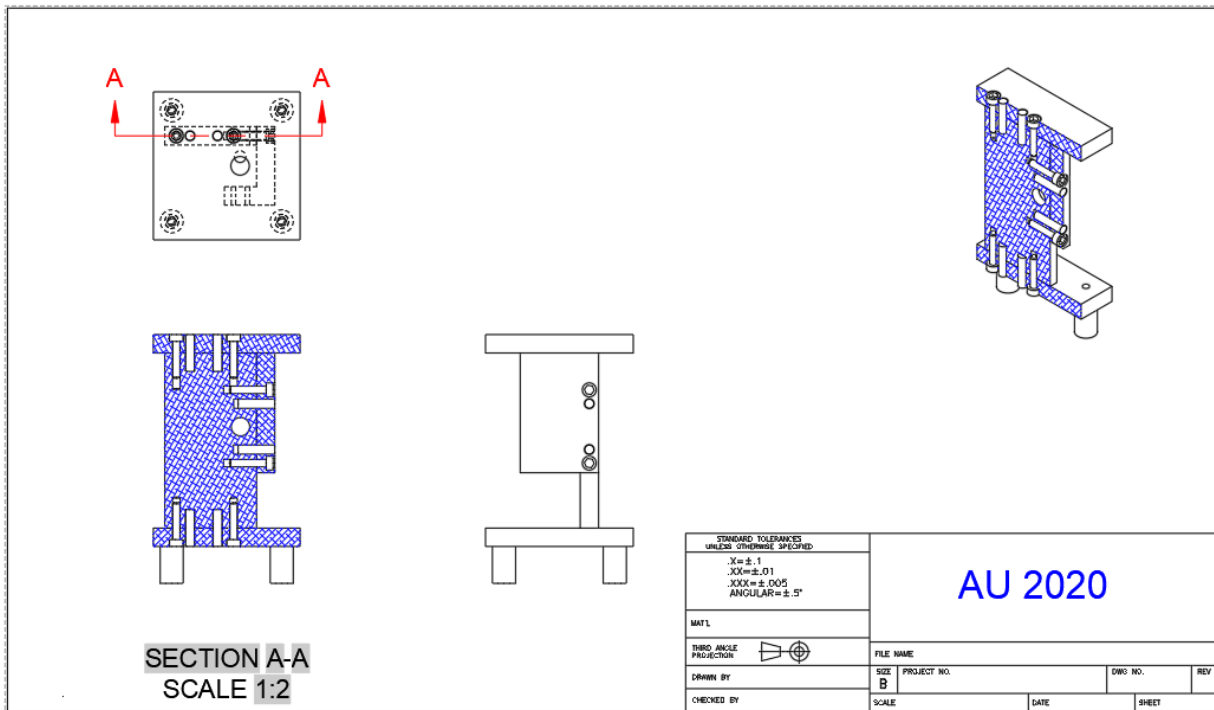
6. **SAVE** the drawing as **Fixture_Assembly_Finished.dwg**

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.
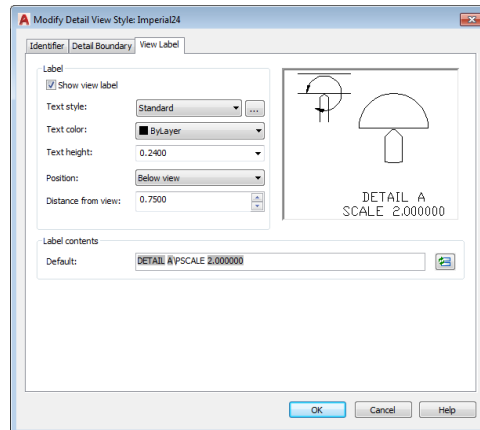
Creating a detail view places a ***detail boundary*** representing the "cutout" area in the parent view. The detail boundary includes a detail identifier. The detail view is placed in another

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.
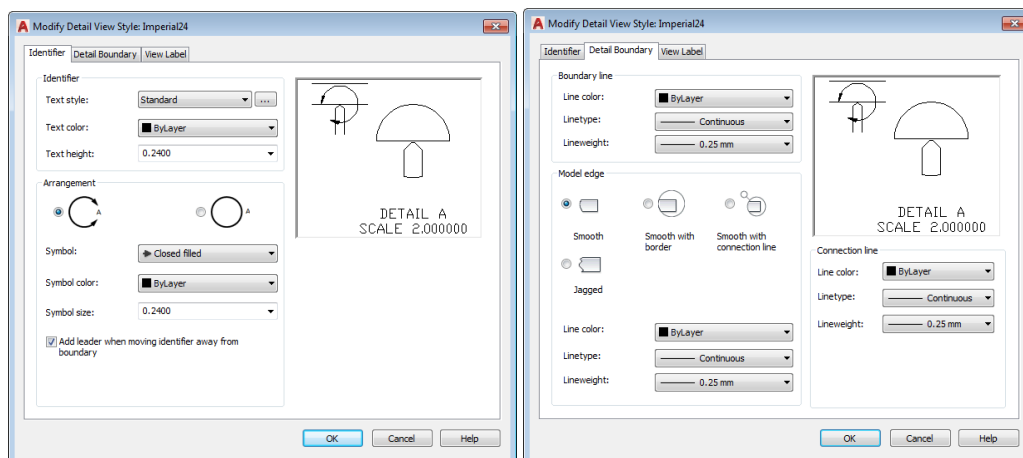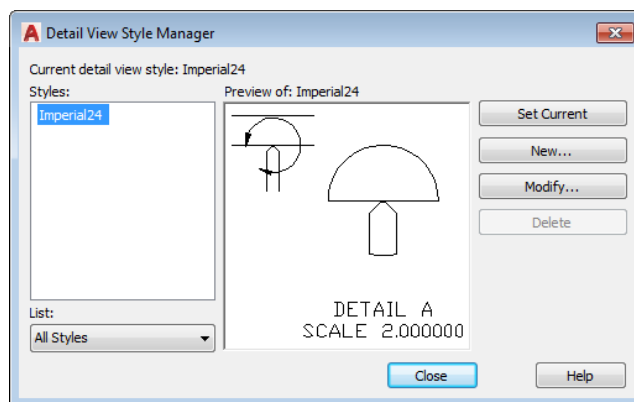
VIEWDETAIL

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.
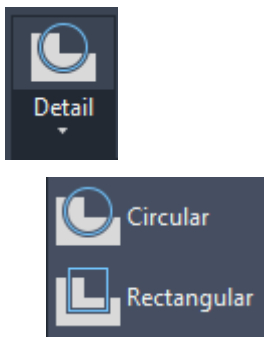
Selecting one of the options from the **Detail** drop-down list in the **Create View** panel of the **Layout** tab on the ribbon begins the command and sets the appropriate boundary type. If you are creating a circular detail boundary, select the parent view and then pick a point to specify the center 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 6A Details

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**.

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_Finished.dwg.**

Exercise 6B Details

1. **OPEN** the drawing **Airplane_Bracket_Detail.dwg. (Note: This 3D model was originally created in Autodesk Inventor).**

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. Change the section or detail **Identifiers** as needed.
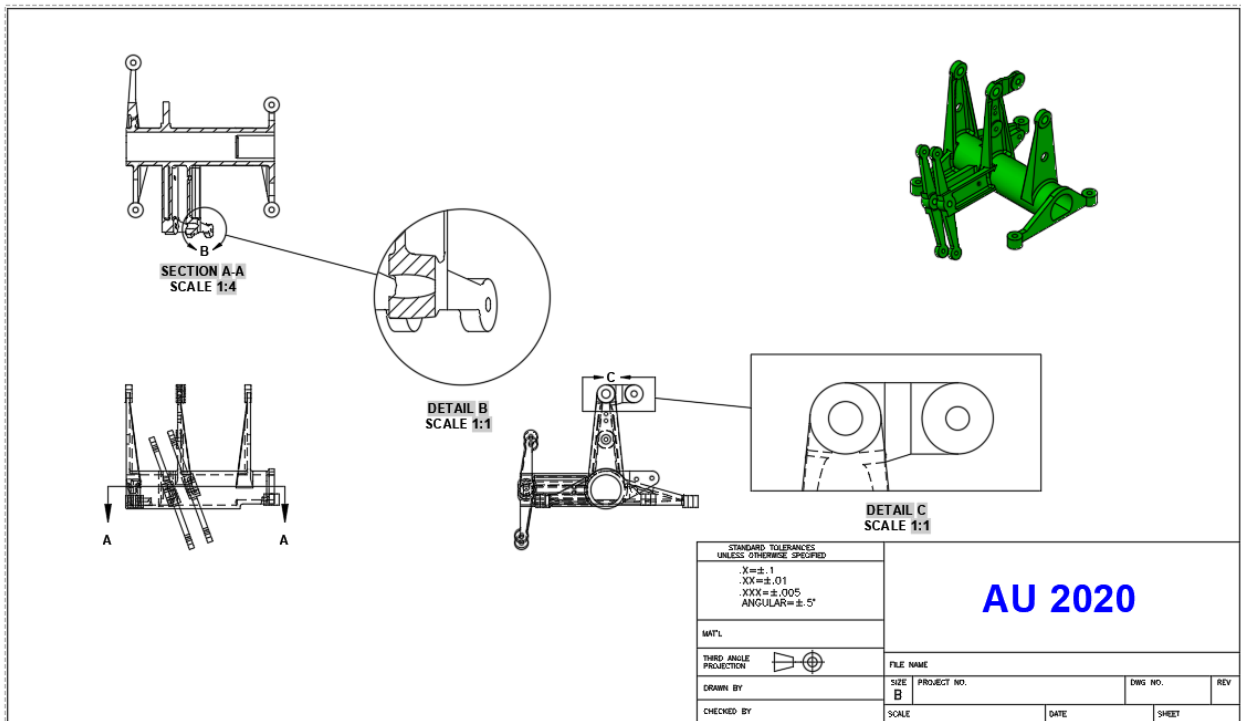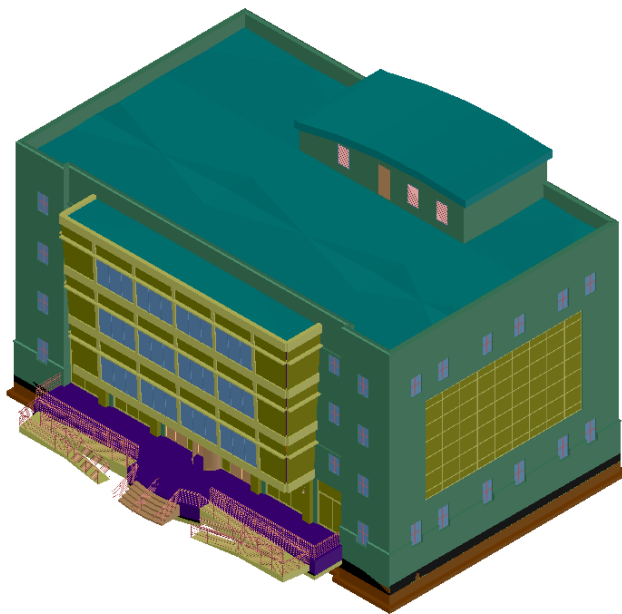

6. **SAVE** the drawing **Airplane_Bracket_Detail_Finished.dwg**

**Exercise 6C Details Extra Credit Challenge Exercise**

Source File: File from **Revit Essentials for Architecture: 2021 and Beyond**. Courtesy of Paul
F. Aubin. www.paulaubin.com.

Let's take a look as a Revit file saved as a DWG with views we will create in AutoCAD.

OPEN the drawing **18 Commercial Building_PA.dwg.**

1. Compete the views below as shown below using the **ANSI-C** layout.
2. Create a Top, Front, Right Side, and Isometric views first.
3. Create a rectangular Detail A from the front view and scale at ¼"=1'-0".
4. Create a circular Detail B from the isometric view showing the revolving door and
    scale at ½"=1'-0"
5. **SAVE** the drawing as **18 Commercial Building_PA_Finished.dwg.**
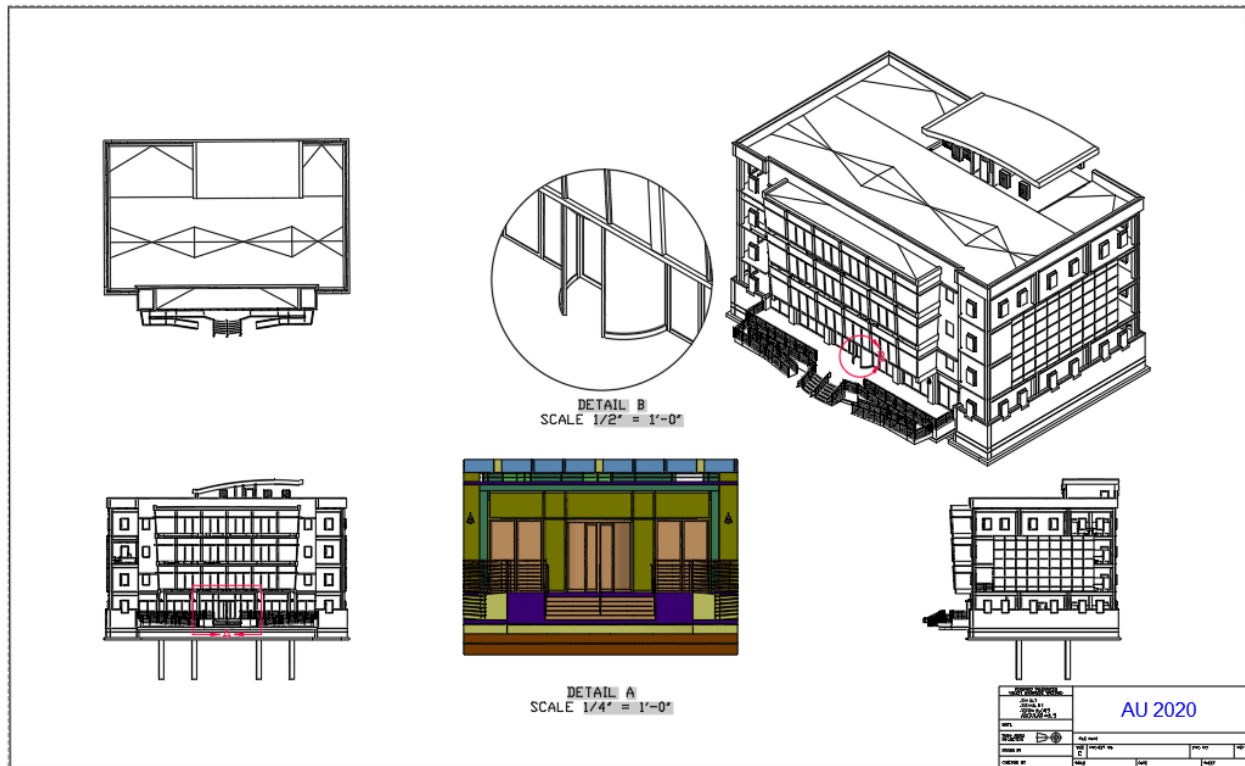
Create undocumented Auxiliary Views

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 Auxiliary View

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.

   Question: What happened to the cutting plane line and section lines?
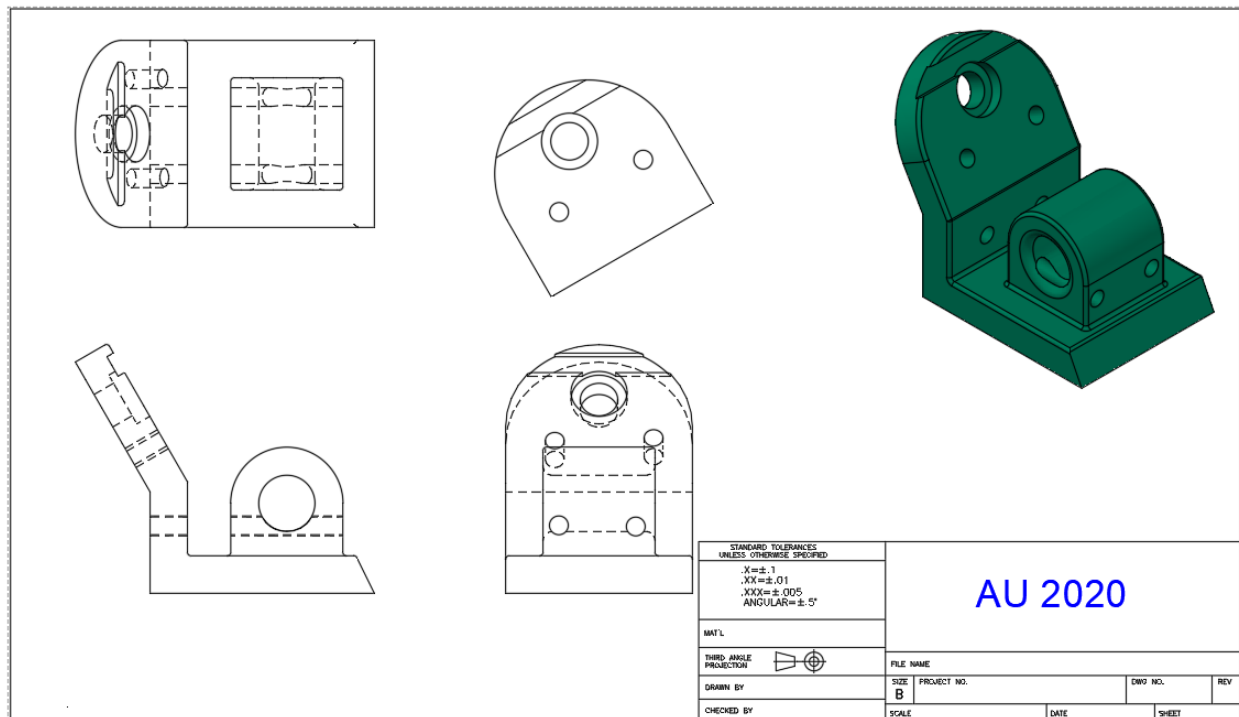
Answer: The layers MD_Annotation and MD_Hatching are frozen.

6. **SAVE** the drawing as **Auxiliary_View_Finished.dwg.**

Dimensioning Drawing Views

You can dimension your drawing views in a layout by following the same procedures as you would dimension in paper space.

Select one of the finished drawing you have already created and dimension the drawing as needed. The example below is:
A**irplane_Bracket_Export_Finished.dwg**
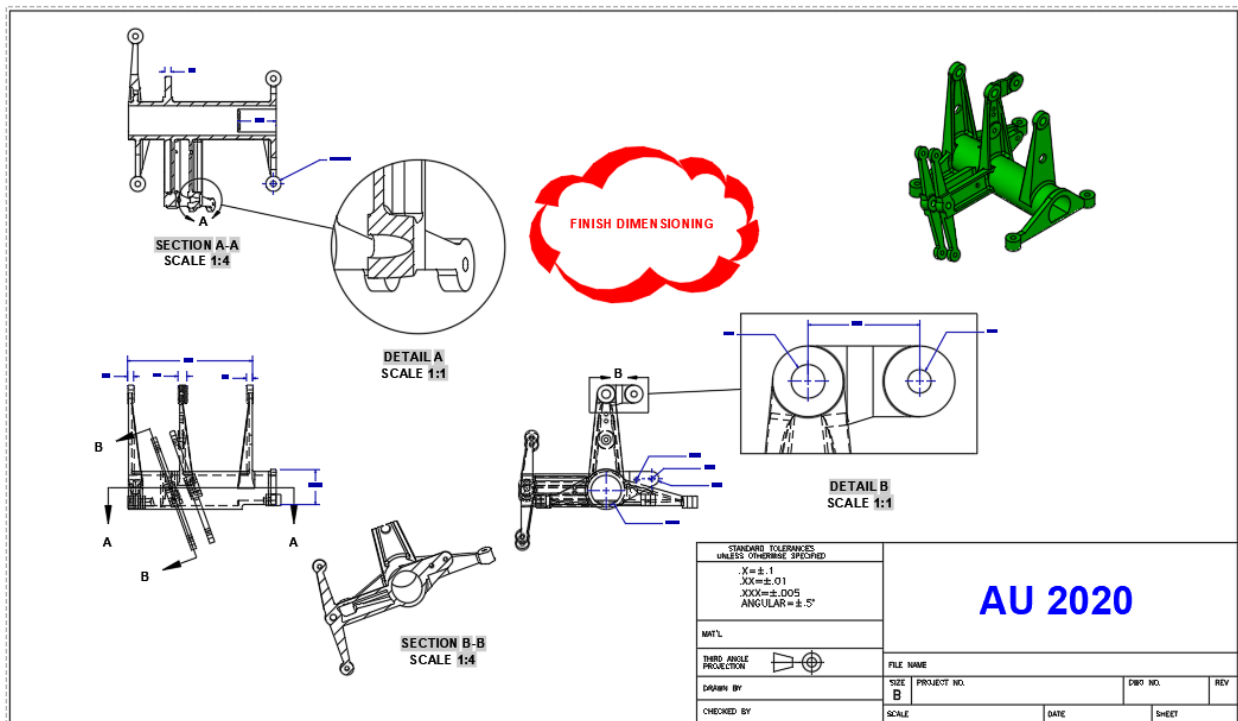
## Export drawing views – EXPORTLAYOUT command

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 EXPORT LAYOUT

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.
   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 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.
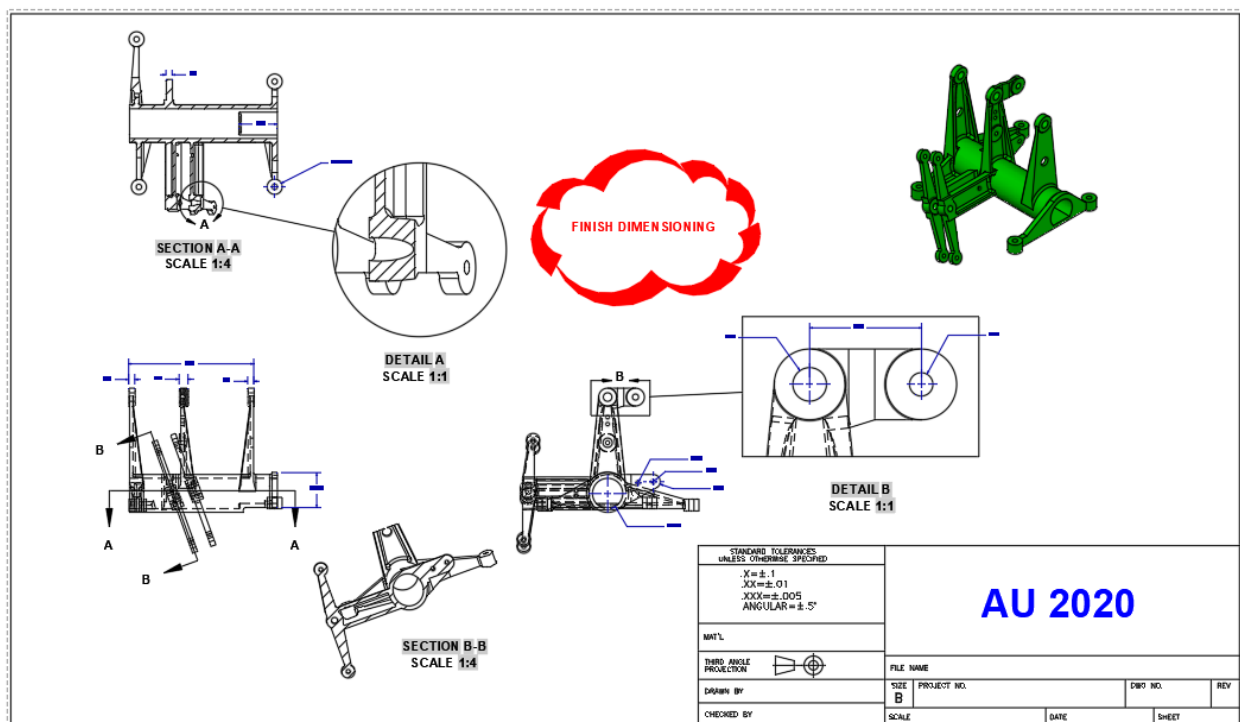

6. For extra credit……**Explode** the drawing!   Type **OVERKILL** and select **ALL**

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**
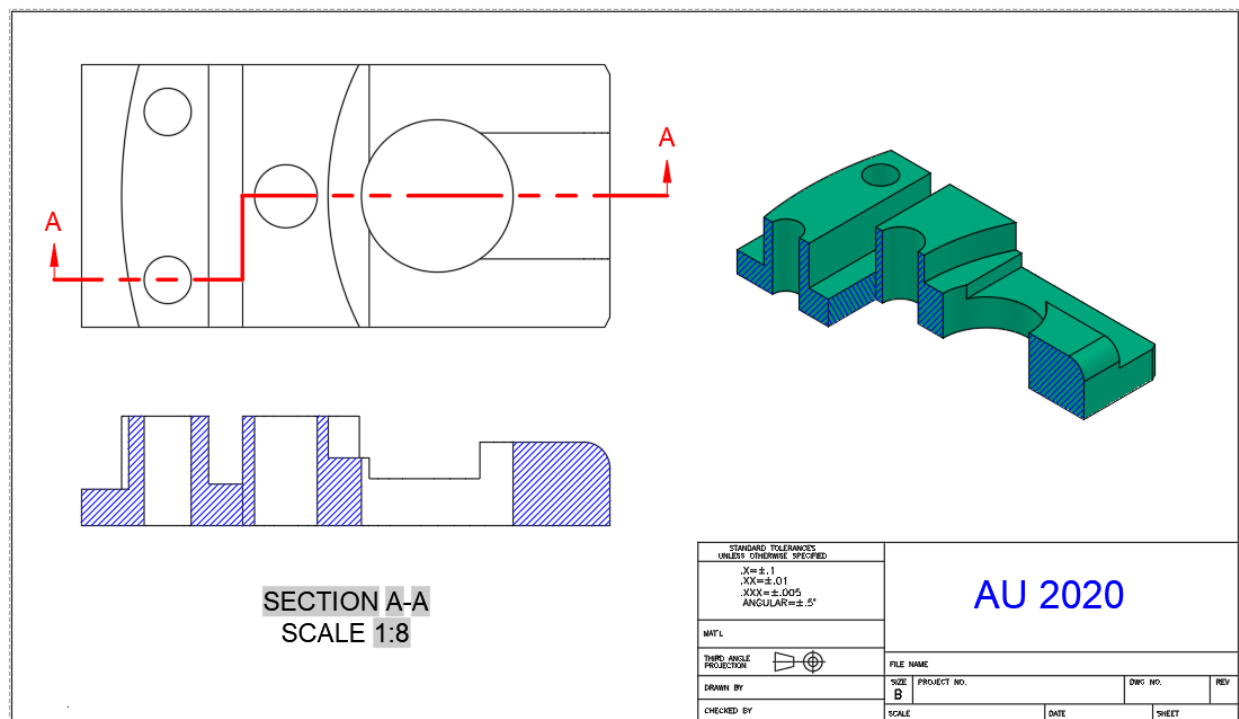
2. Type, **EXPORTLAYOUT** to export the layout. Save the drawing to the lab drawing folder as, **Offset_Base_Section_Finished_ANSI B.dwg. (Overwrite the file if needed).**

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.
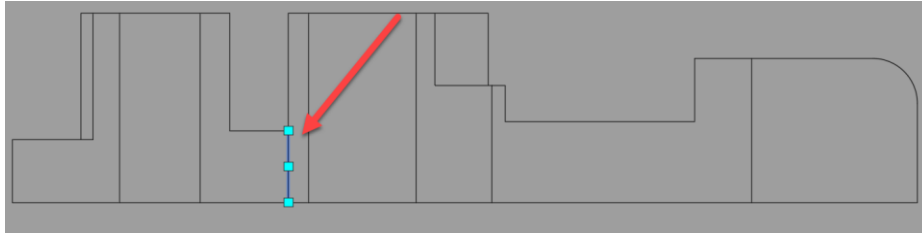
5. **Close** the Block Editor and **Save** the changes.

6. The sectioned view has been updated. The sectioned view reflects ANSI standards.
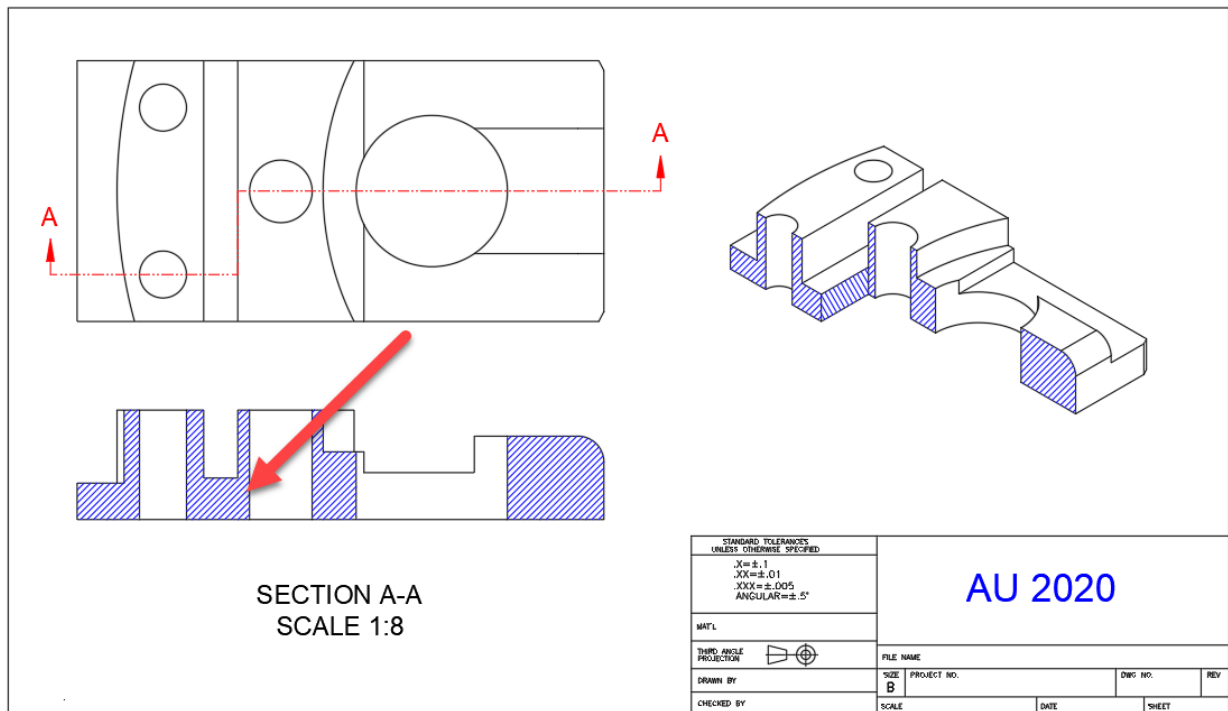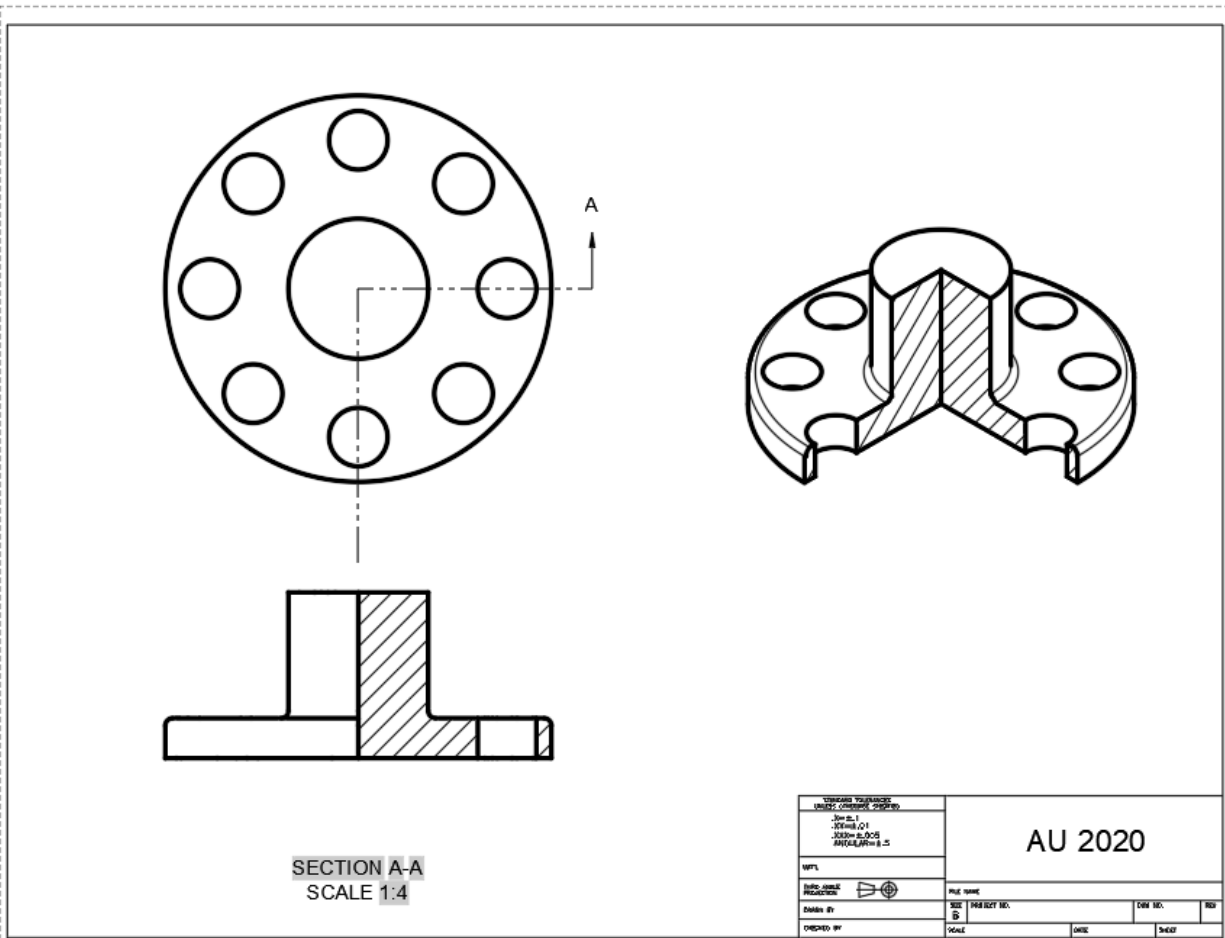

7. **SAVE** the drawing as **Offset_Base_Section_in_Model_Space.dwg.**

Edit the Model - Drawing Views Change

Exercise 9A Editing the 3D model

1. **OPEN** the drawing as **Half_Section1_Finished.dwg.**
2. Press and hold the **CTRL** key and select the top of the part. Use the gizmo to extrude the top part by a value of 3.
3. Do the same for the flanged base with the hole to a value of **1**.
4. Select the **ANSI A** layout. Notice the changes in the drawing views. The drawing views are linked to the 3D model.

5. **SAVE** the drawing as **Half_Section1_Edited.dwg.**
6. **OPEN** other drawings what you have worked on and try re-designing the 3D model. What happens to the drawing views?

SECTION A-A
SCALE 1:4

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